HEXAGON
MANUFACTURING INTELLIGENCE

ALPHACAM 2020.1
STANDARD 5 AXIS

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.

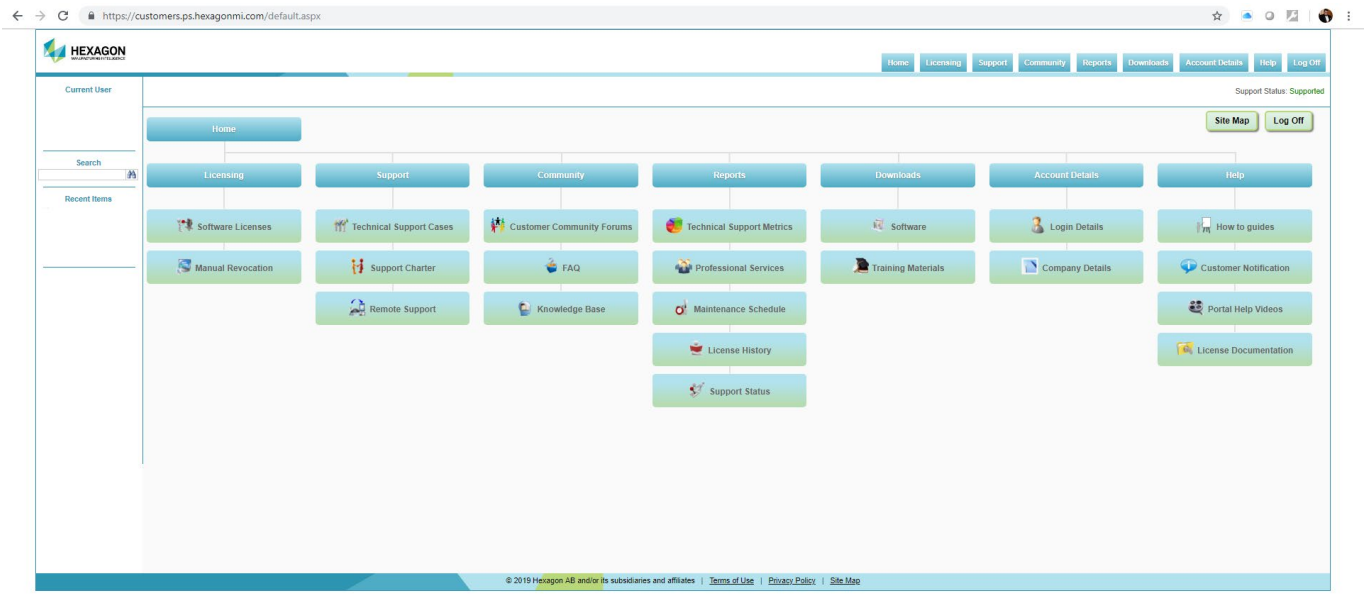


Figure 1 - Customer Support Portal

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.

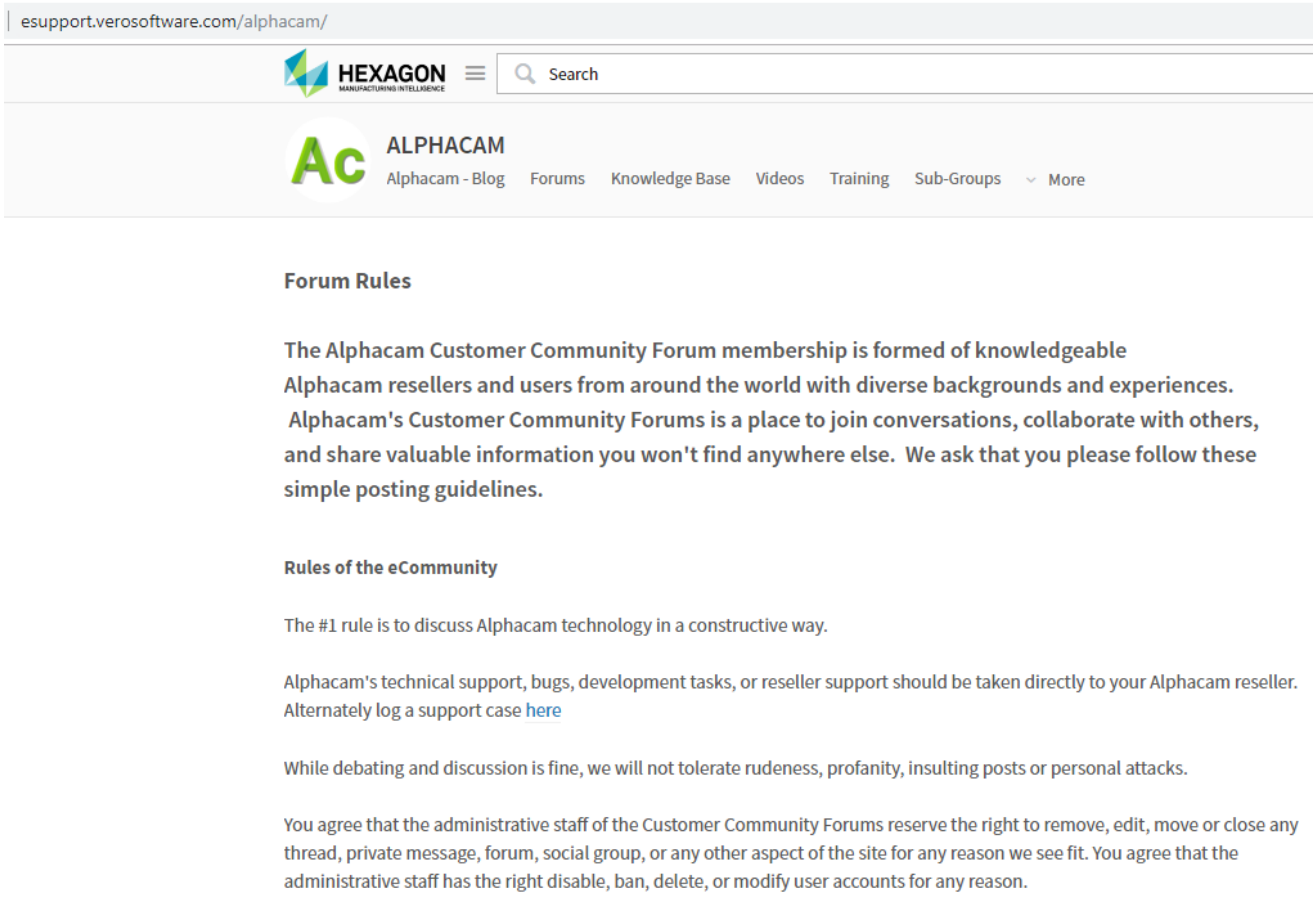


Figure 2 - esupport page

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.

Introduction

The use of 5 Axis machining is a growing area within the manufacturing world but one thing to remember is that not all 5 Axis machining is moving all 5 axes at the same time.

This is the difference between Positional 5 Axis (sometimes referred to as 3+2 machining) and Simultaneous 5 Axis. The first thing an ALPHACAM user needs to be able to identify is which of these two types of machining is required. Just because you can simultaneous 5 axis tool path, does not mean that you should.

In some circumstances the machine tool will perform erratically or give a poor surface finish when trying to apply a simultaneous 5 axis tool path to a part, where a positional 5 axis or 4 axis tool path would have given better results.

Objective

The purpose of this supplement is to add an understanding of the practical requirements applied in ALPHACAM when dealing with 5 Axis toolpaths. To make best us of this supplement the user must have a working knowledge of the three advanced areas of ALPHACAM, Feature Extraction, Solid Manipulation & Machining practices and Work planes. These three modules of ALPHACAM will be used to assist in the creation of suitable solutions to 5 Axis tool path needs.

Tool Axis Conversion

Further control of 2D and 3D tool paths can be achieved with tool axis conversion.

Tool Axis Conversion is only available using the Ultimate level of ALPHACAM

Tool axis conversion is only possible on tool paths using Ball cutters.
Tool Paths created by ALPHACAM are already either 2 or 3 Axis so you may wonder why we want 2 and 3 Axis Conversion. Sometimes it is easier to get the type of cutting strategy required by creating the Paths in an angled work Plane.

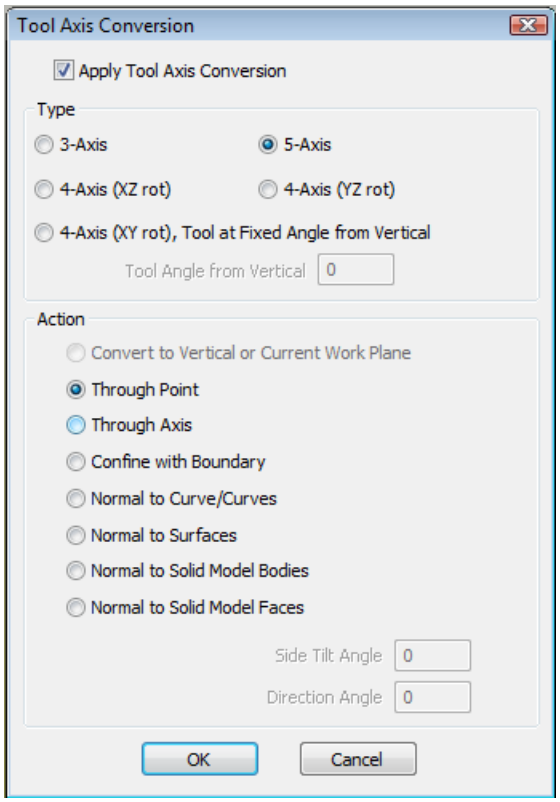


Figure 3 - Explanation of Tool Axis Conversion dialogue

The actions available will depend upon the type to which you are converting.

☑ It is useful to mention at this point the checkbox “Tool can Under-cut” on the Dialogue in the Z Level type strategies, if a lollipop tool is selected then this will be on by default, however If the tool is normal it will be off by default, but the user can activate it, and this will let the tool undercut in preparation for the conversion.

Tool Axis conversion is better applied to specific tool path operations via the project manager local popup menu command.

Convert 3D work plane tool path to Vertical

In this example the Spiral Path on an Angled Plane is converted to 3axis vertical which would yield a very good surface coverage.

When the 3 Axis Type is selected, all the Action Selections are greyed out. The selected tool path should be converted into Flat Land or if a work plane is active, the user is prompted to confirm the transformation to the current plane.

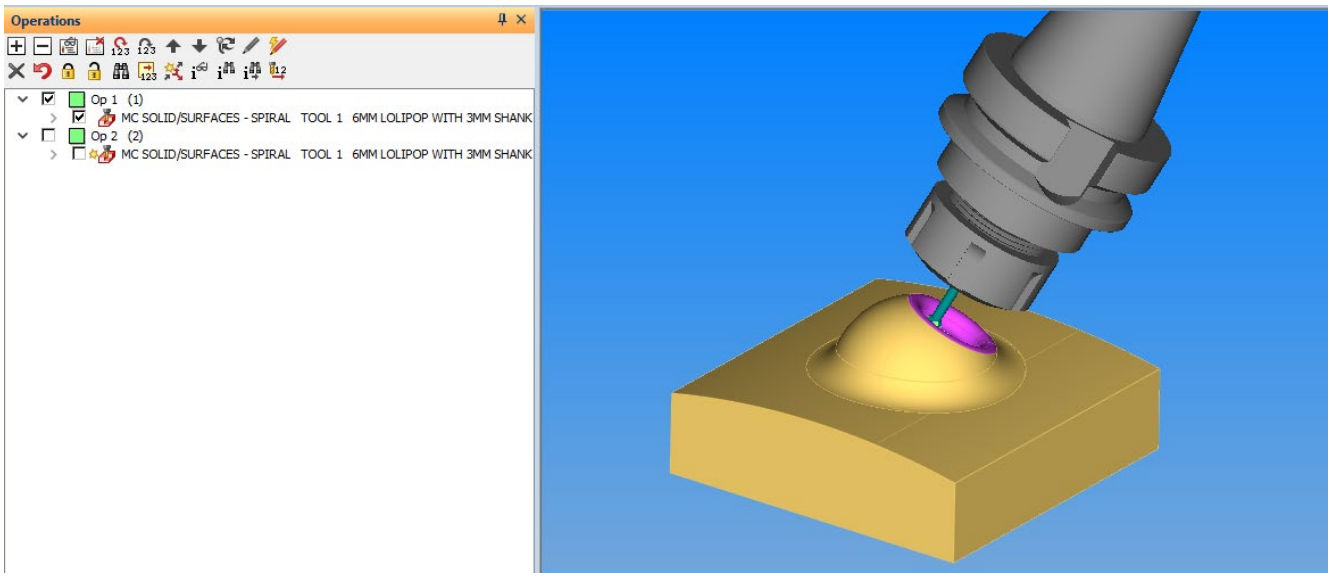


Figure 4 - Spiral cycle created in a work plane

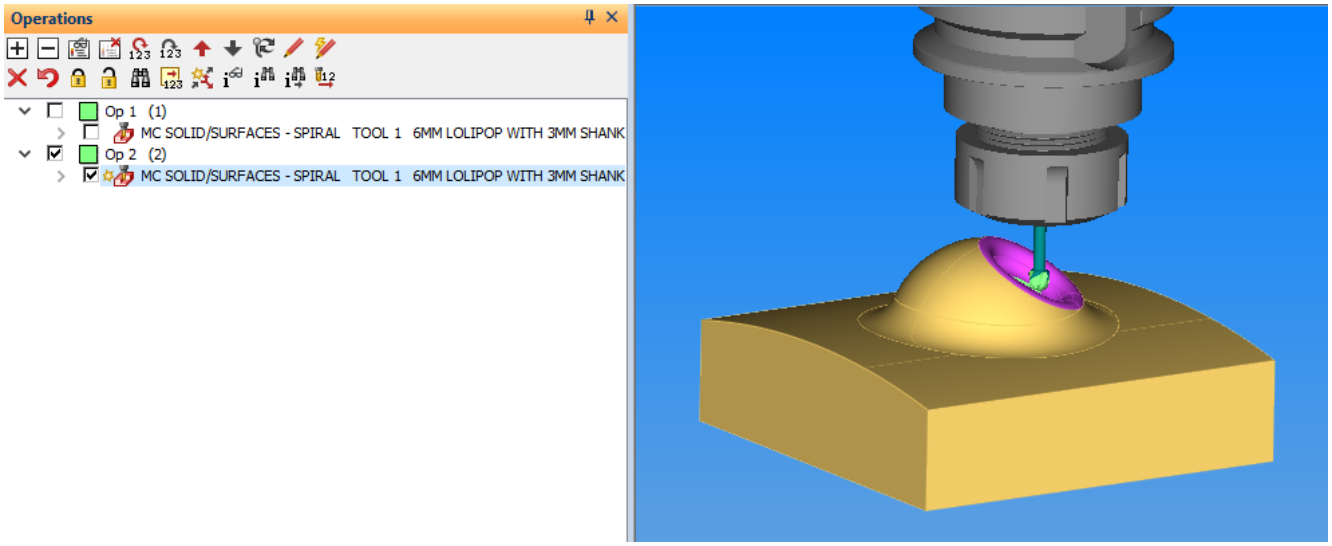


Figure 5 - Spiral cycle converted to a 3 axis toolpath not on a work plane

Note that any toolpath that has had **Tool Axis Conversion** applied to it will be marked with a Gold Star.

Convert 2 Axis tool path in work plane to Vertical

In this example, there are 2 operations each multi cut at a single depth, the depth being increased for the second cut.
The first image shows the tool orientation as created.
The second shows the tool path after conversion.
The simulation shows the ball ended cutter however this could be changed to a flat ended end cutter on the machine.

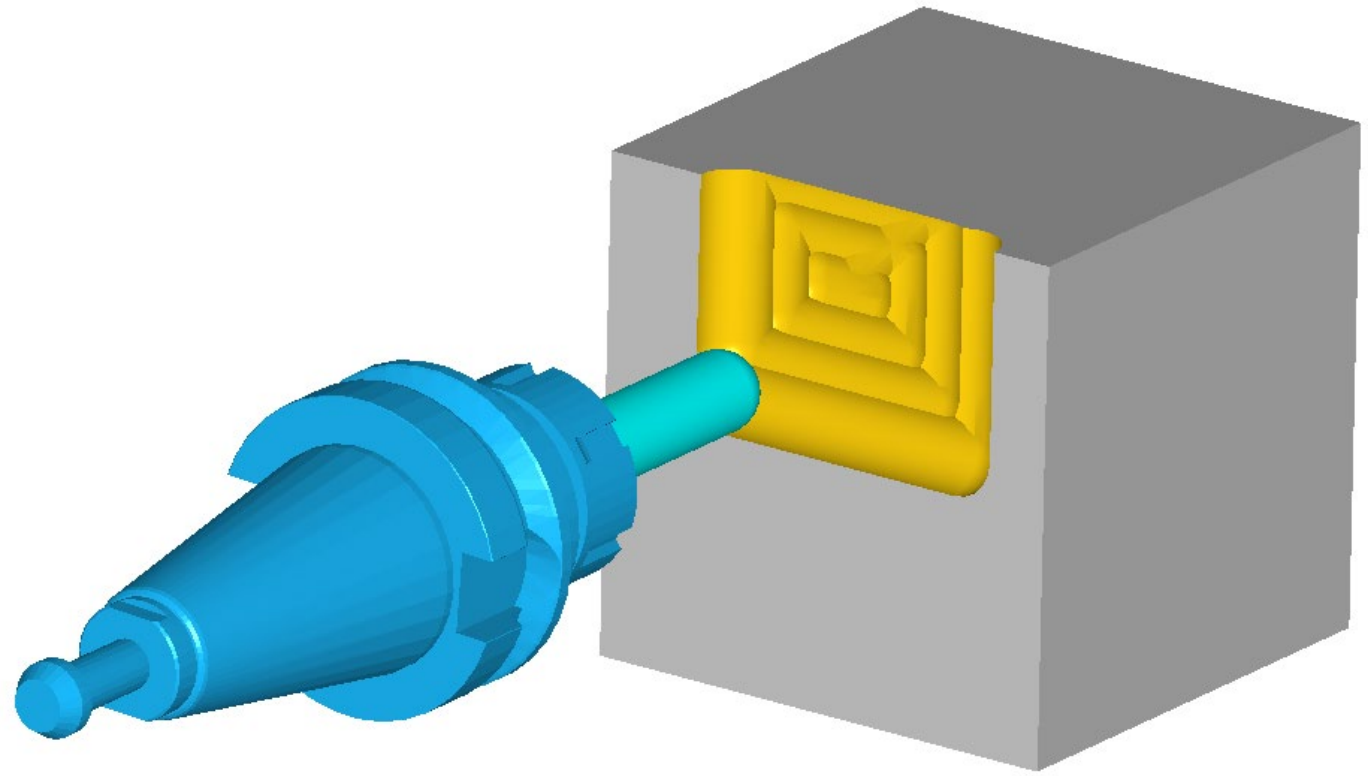


Figure 6 - 2D Pocket Cycle applied on a Work Plane

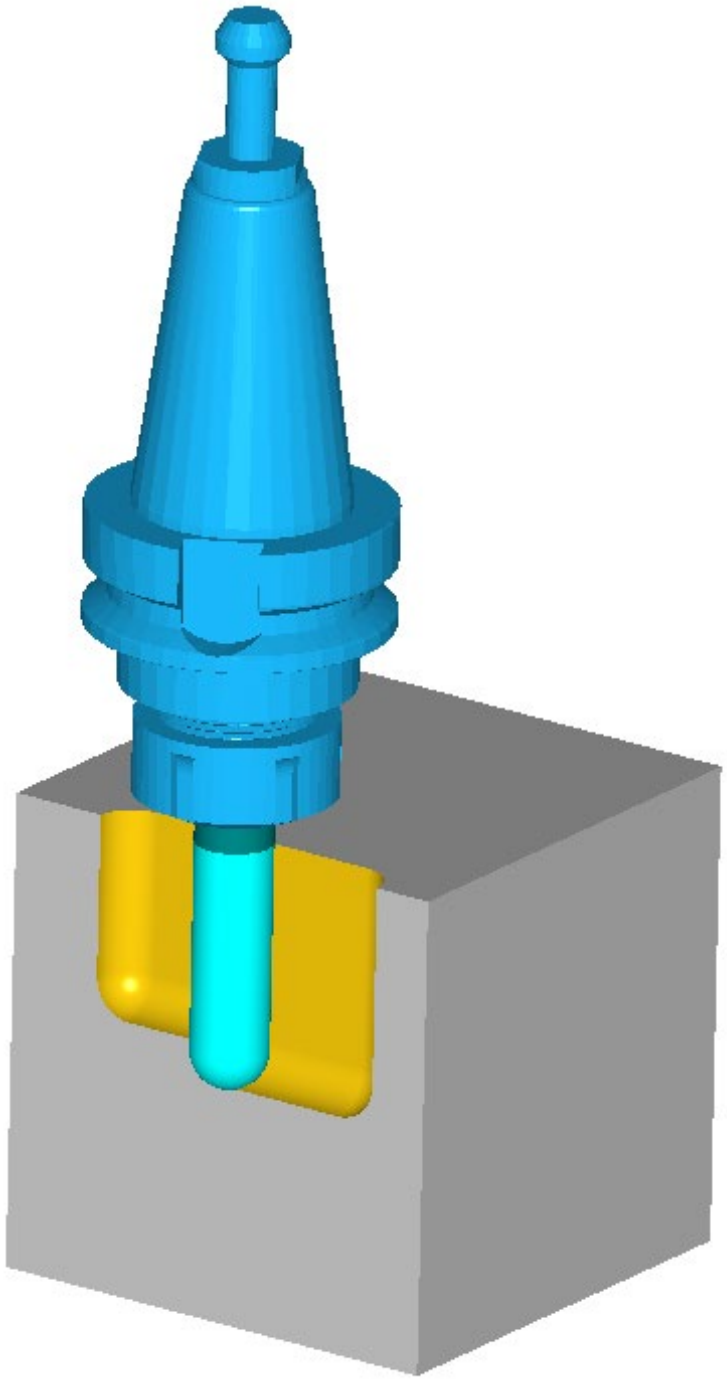


Figure 7 - 2D pocket cycle converted from a work plane to Vertical

4 Axis XZ or YZ Rotation

Through Axis

With this option the user will be asked to select a point on the Axis. The conversion will then make the tool paths Rotate about the selected axis.

Parallel tool paths

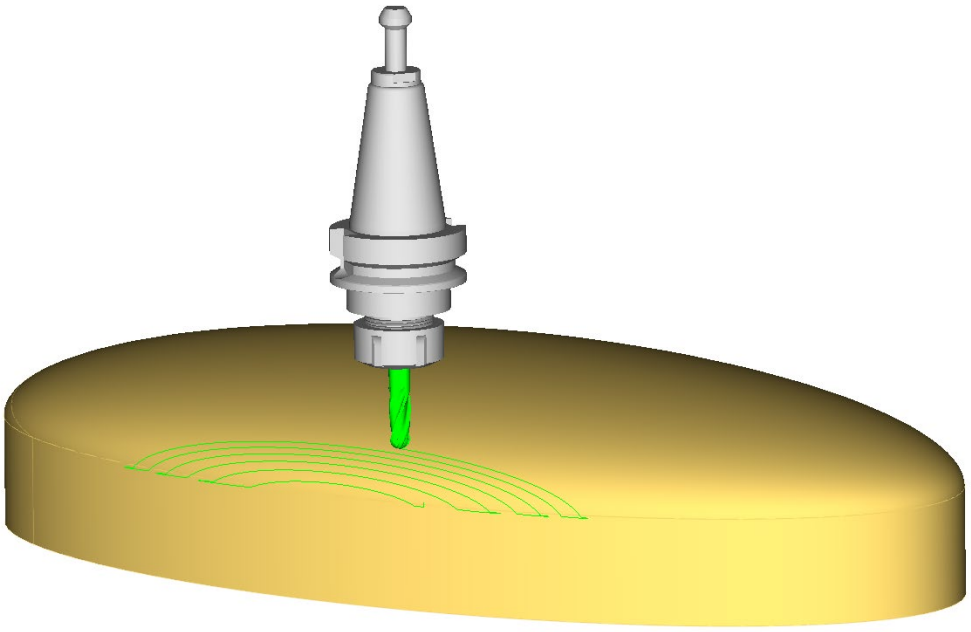


Figure 8 - Parallel Strategy as applied

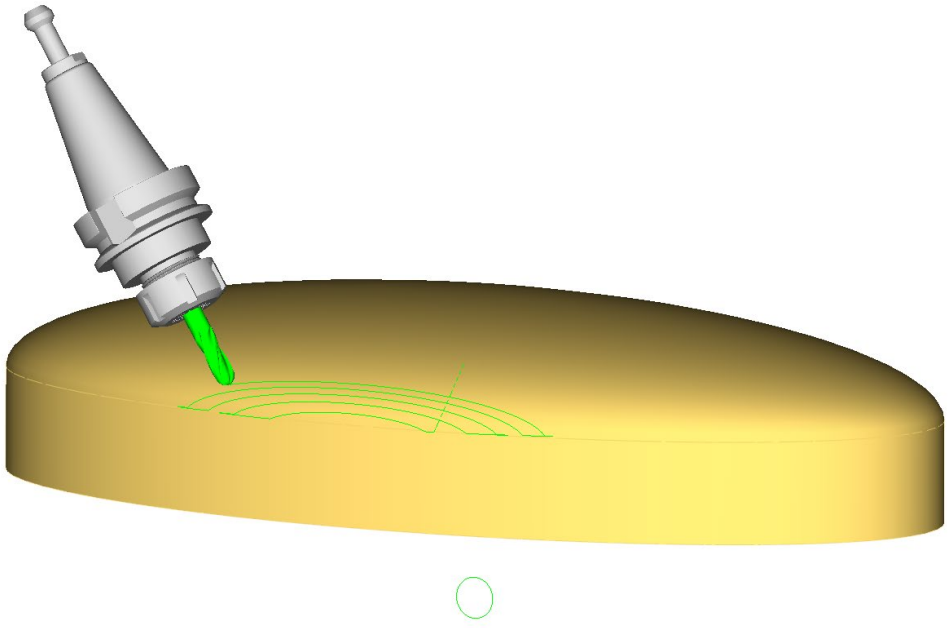
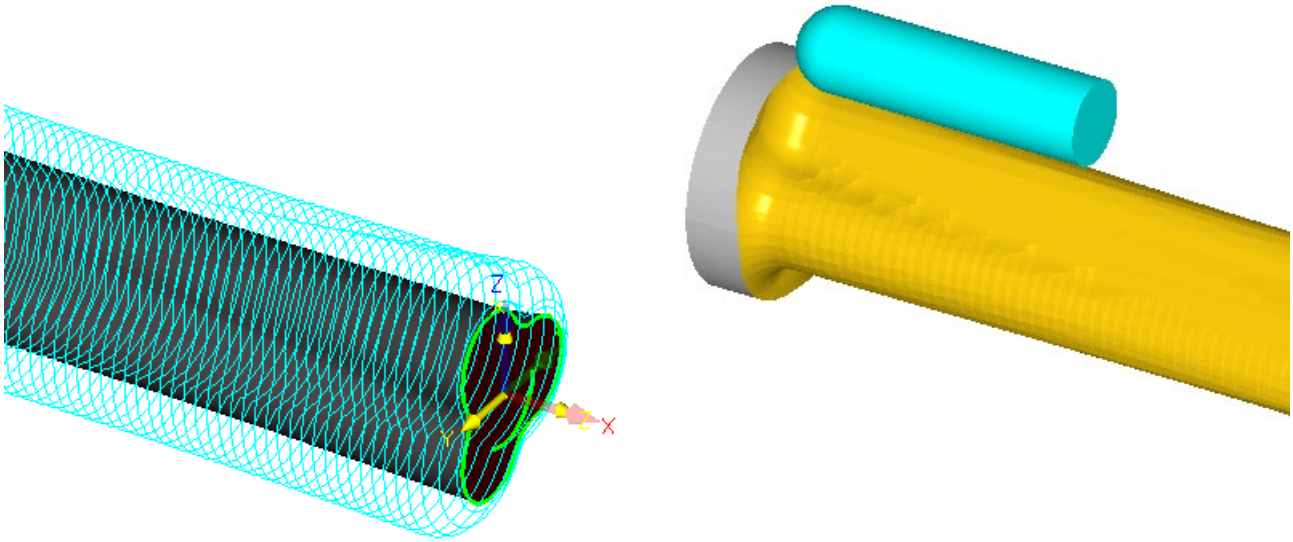


Figure 9 - Tool Axis Conversion at the centre of the circle

Helical Z tool paths in work plane

Used with Z (enhanced undercuts)
Before Conversion.



After Conversion through axis at centre of circle on end, along the global X.

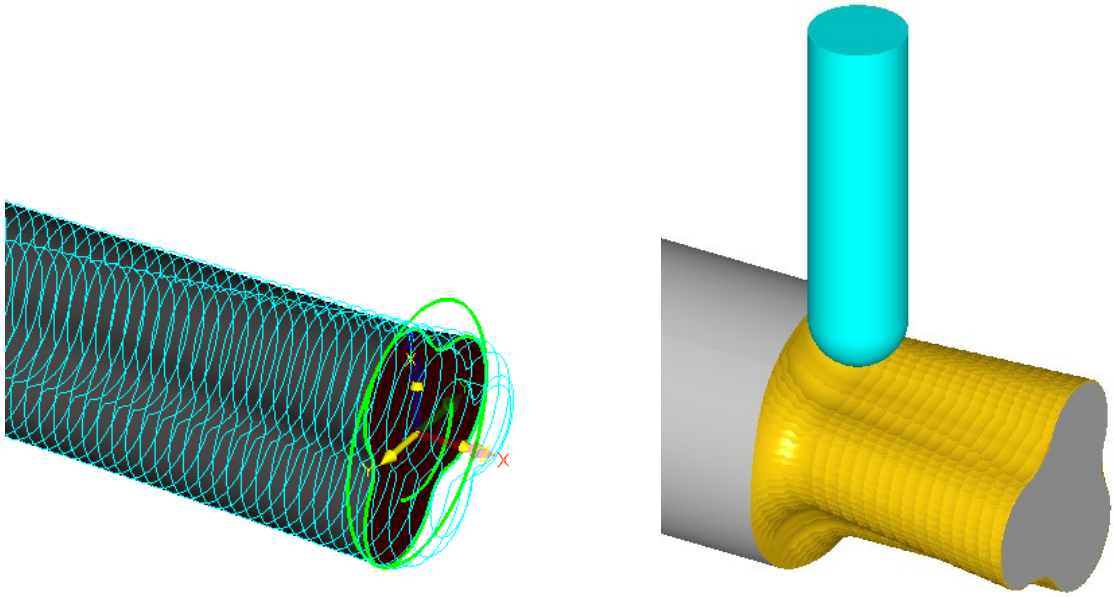


Figure 10 - Z Enhanced Undercuts converted to Vertical through an axis

5 Axis

XZ Rotation Confined with Boundary

This will work with the same principles as the 5 axis option with the limitation of the Axis control dependent upon the Axis configuration selected. YZ or XZ
Before Conversion machining with lollipop tool Horizontal Z using lower boundary and selected faces, setting the option for tool can undercut.

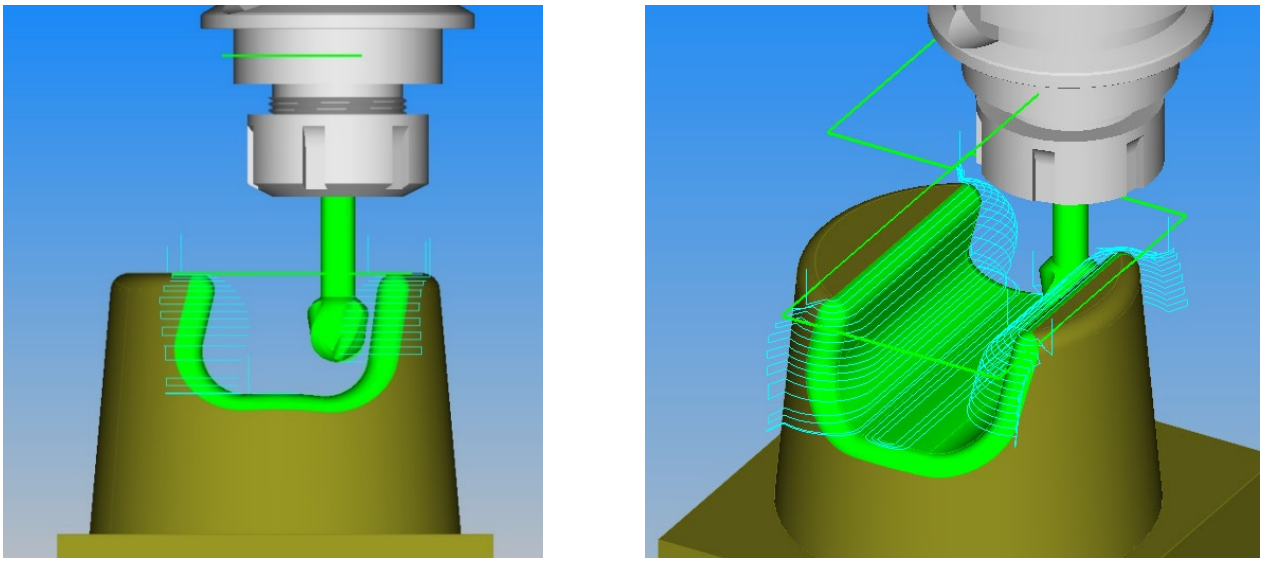


Figure 11 - Initial Horizontal Z toolpath

After Conversion tool confined to upper boundary, this will allow tool paths to be created with a lollipop tool but a straight ball tool can be used on the machine.

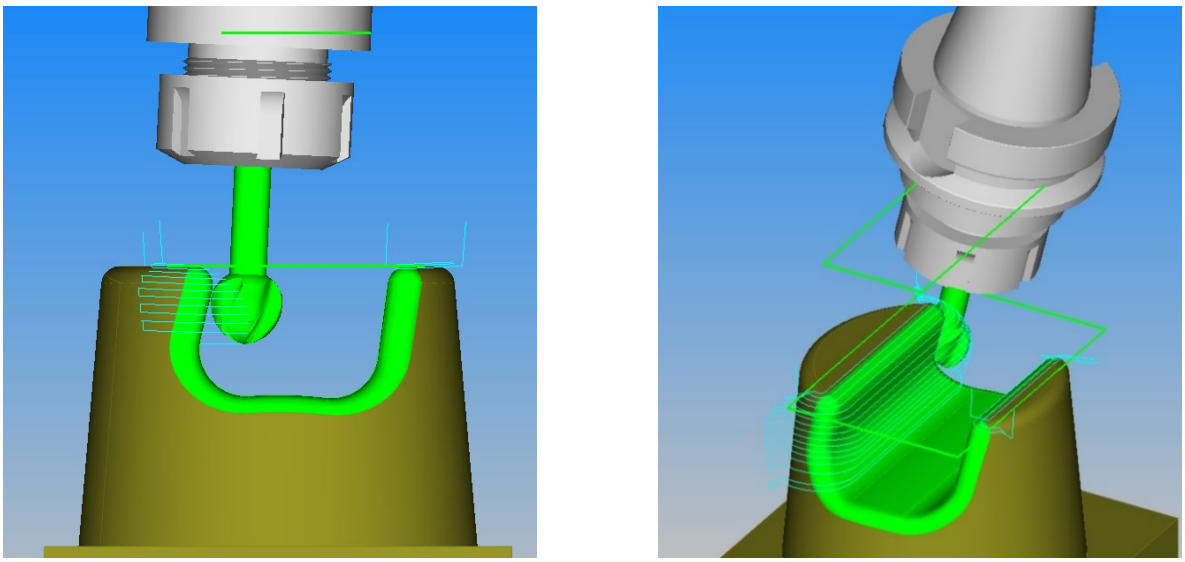
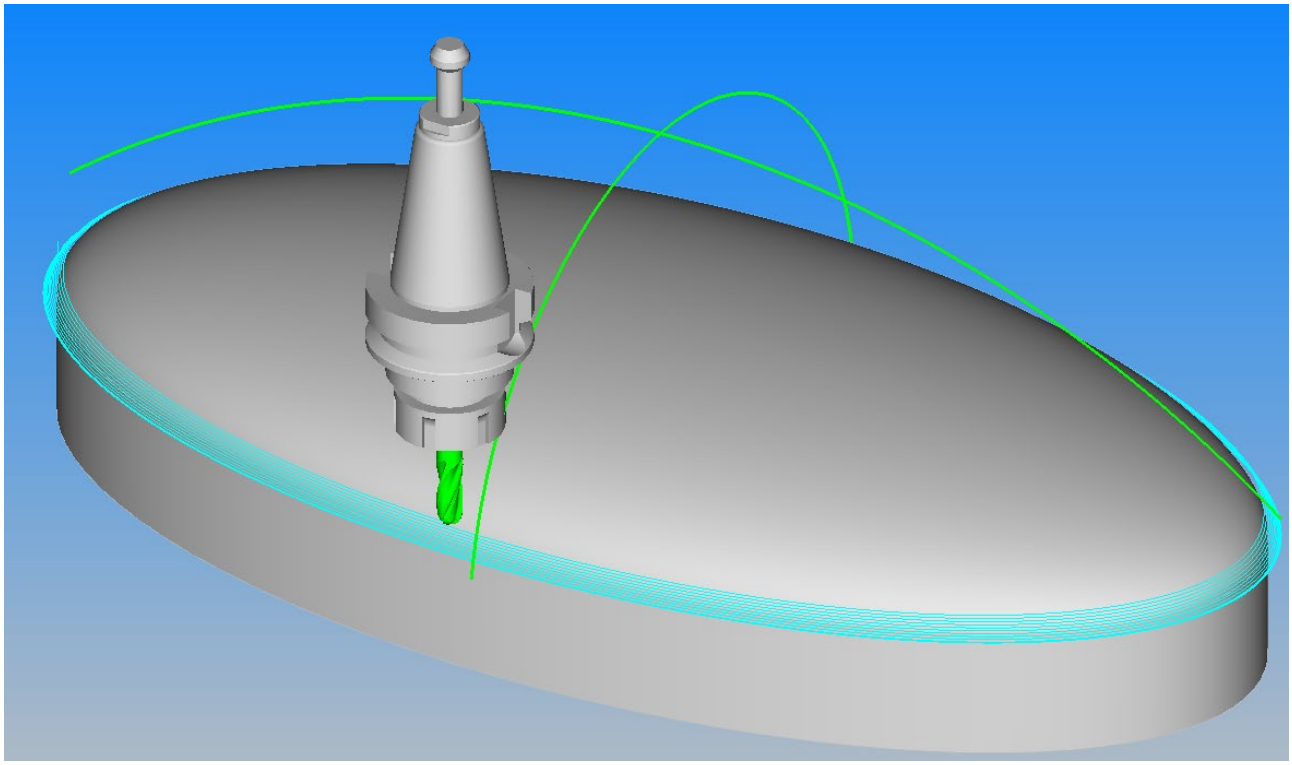


Figure 12 - Toolpath altered using a boundary to prevent collisions

Normal to Curve or Curves

Before Conversion machining tool paths created using Constant Cusp.



After Conversion using guide curve on plane causes tool to cut on the side of the ball for better machining.

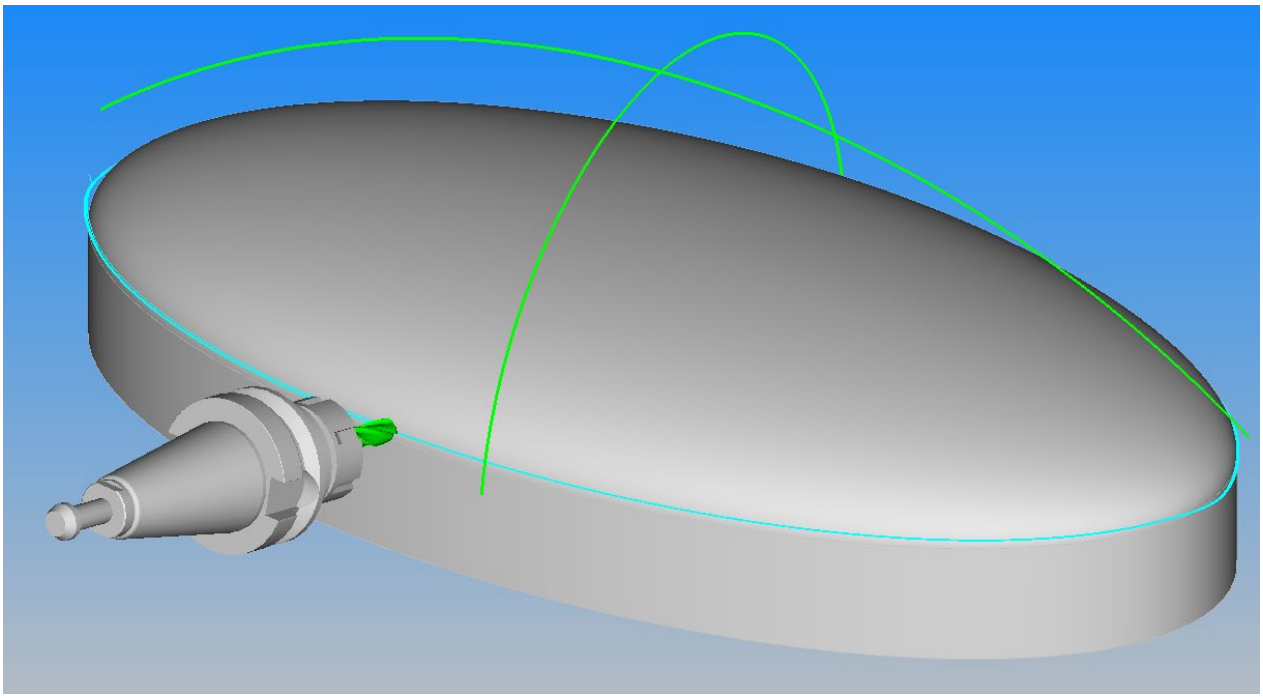
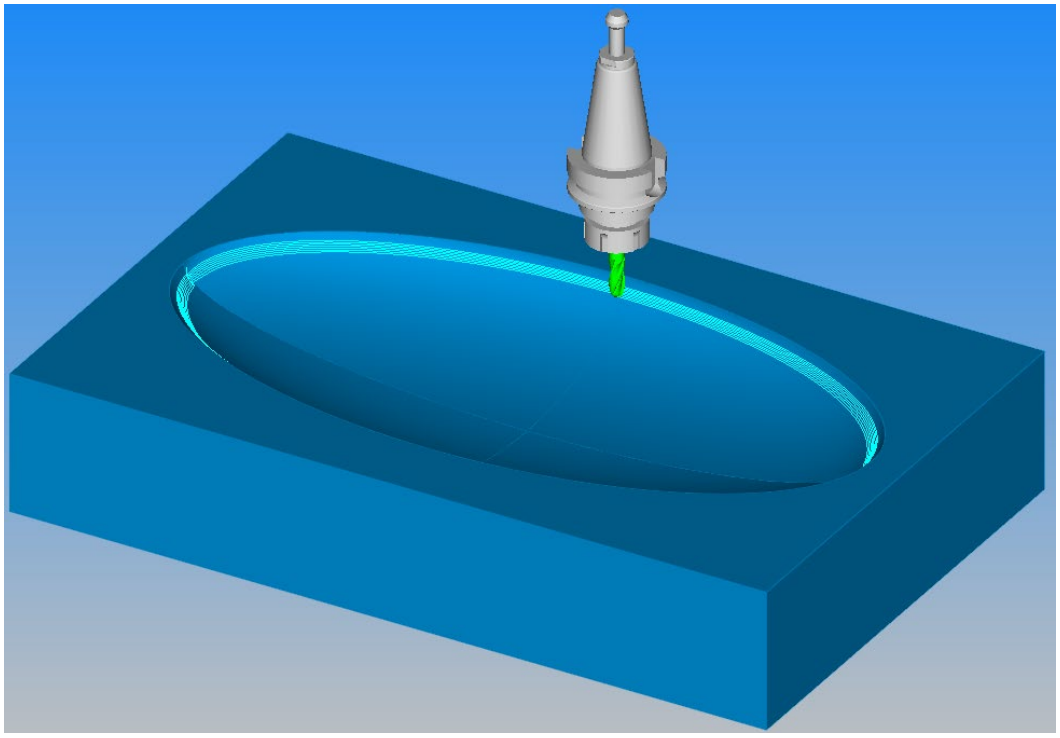


Figure 13 - Toolpath conversion using a guide curves

XY Rotation Tool at Fixed Tilt Angle

With this option the user will be asked to select an XY point in the flat Plane. The conversion will then be made similar to the 5 Axis option, but the tool will stay at the fixed tilt angle and the XY Vector around the Z axis will point towards the selected point.
Before Conversion Projected Contours.



After Conversion to a tilt angle of 45° and the point selected at centre of the recess.

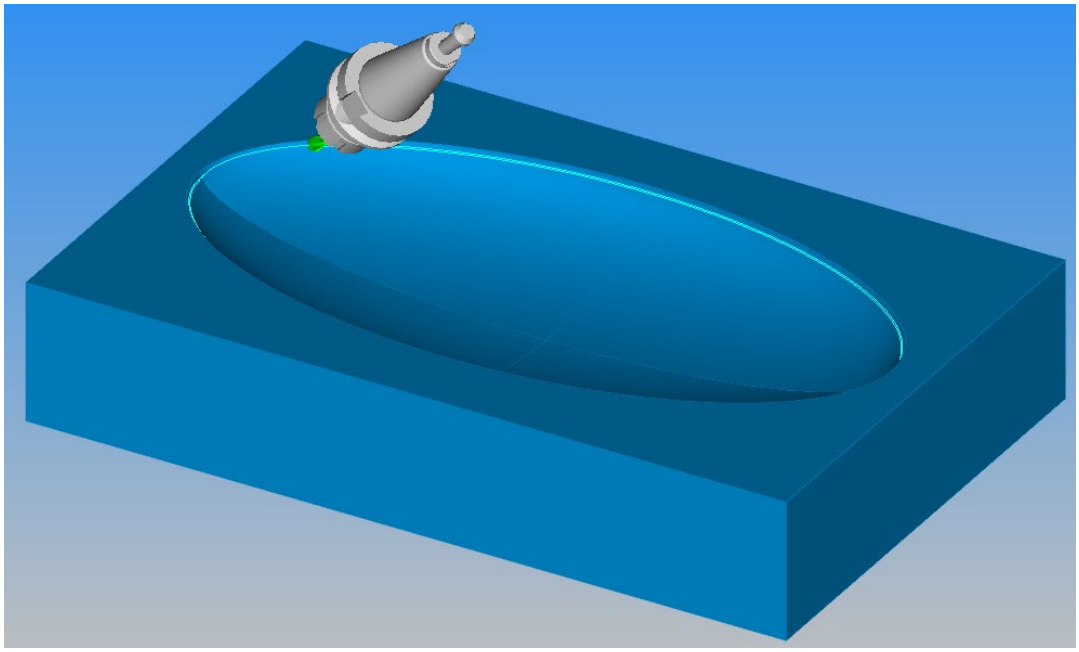


Figure 14 - Converted toolpath along a specified angle from vertical

Project through Point

The tool axis vector will be transformed so that the tool axis will always pass through or point towards a user defined point. There is a prompt asking the user if the picked point is on the tool side of the part. If the point is on the tool side of the part, then the tool axis will pass through that point. If the point is not on the tool side of the part, then the tool will point towards the user defined point, but not pass through it.
Before Conversion Projected contours as #26 XY Rotation Tool at Fixed Tilt Angle previous.

After Conversion with point at centre of dish and at Z 100 on tool side.

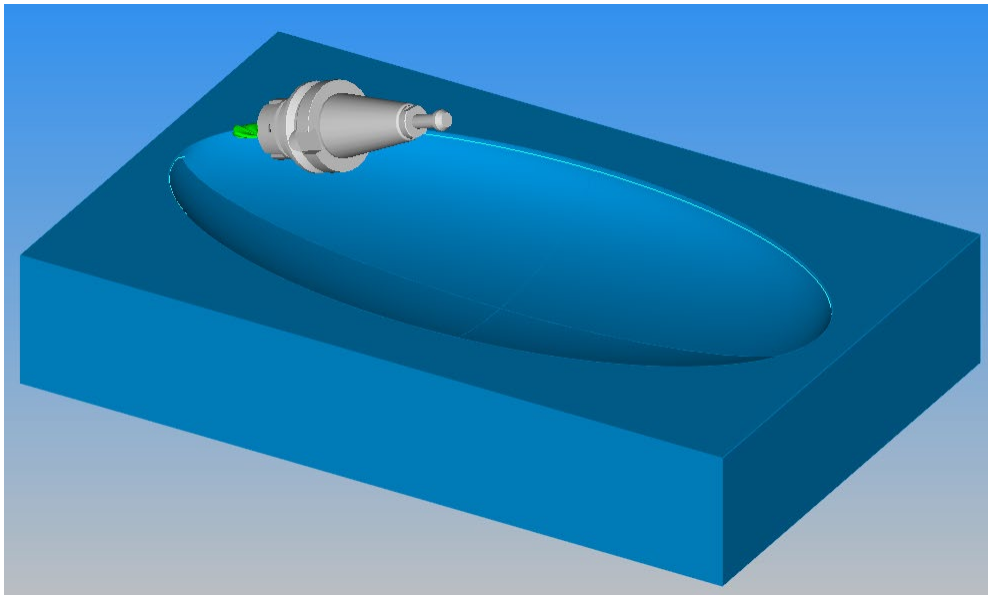


Figure 15 - Tool path conversion through a point on the tool side

After Conversion with point at centre of dish and at Z -200 not on tool side.

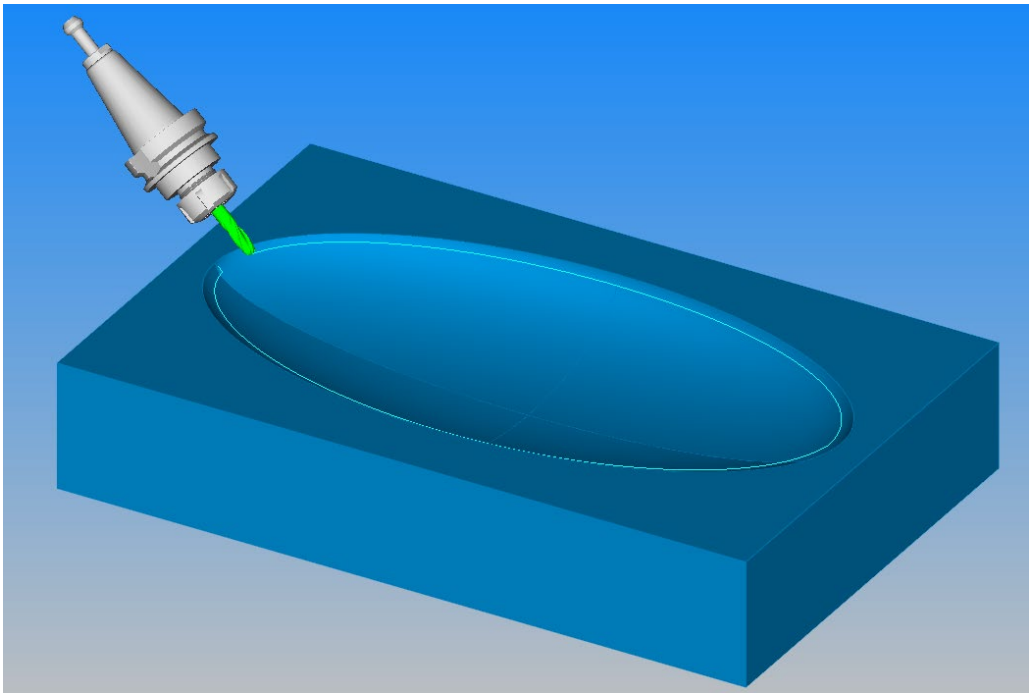


Figure 16 - Through a point on the opposite side from the tool

Project through Axis

With this transformation, the tool axis will always be normal to the selected Line Axis. The user will be prompted to select Line of Axis of Revolution.

Using the points that describe this axis and the point at the centre of the ball tool we will calculate the vector perpendicular to the axis in the plane created by the three points.

Before Conversion Parallel cuts.

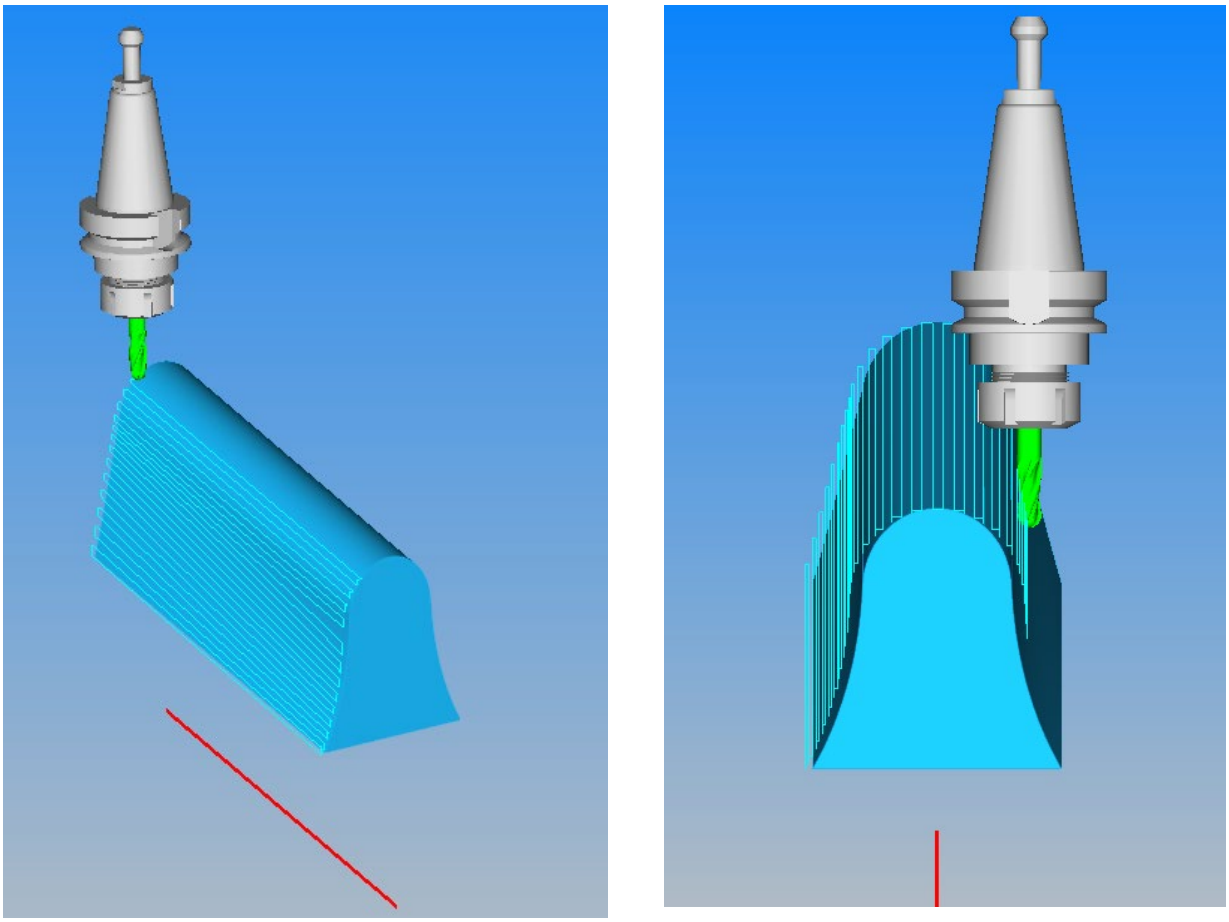


Figure 17 - Initial Parallel cycle

After Conversion tool adjusted to point at polyline axis.

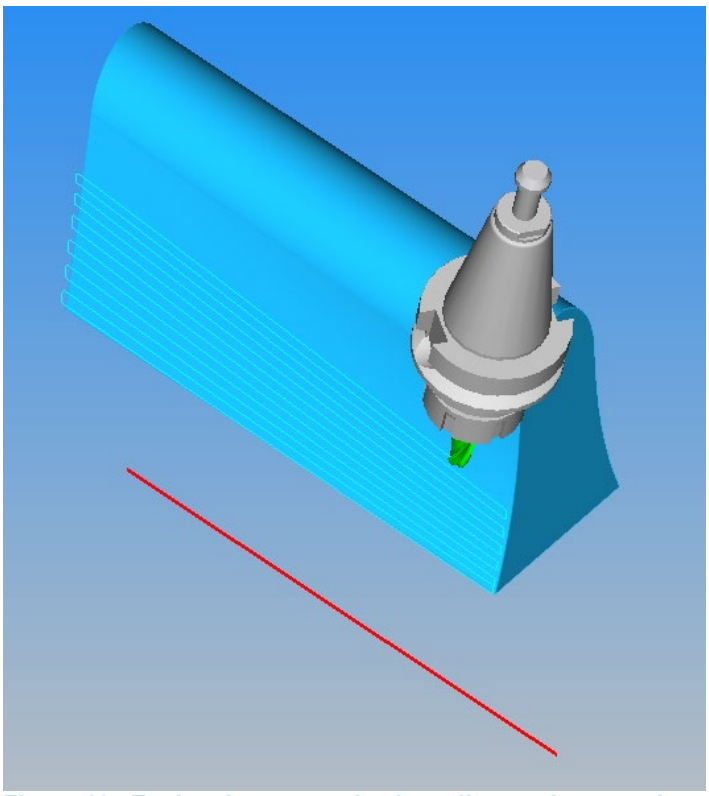
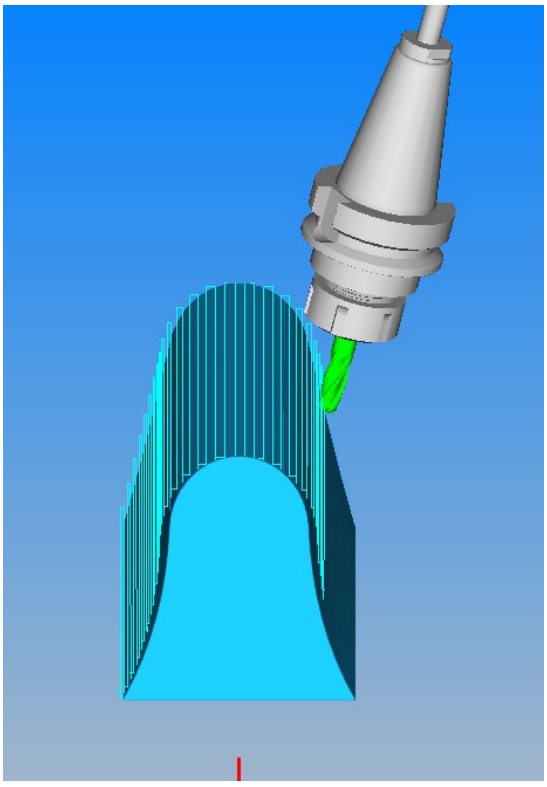


Figure 18 - Tool path converted using a line on the opposite side from the tool



Confine with Boundary

This method requires the user to select a boundary. The boundary must be a closed geometry and the tool direction Inside\Outside will determine as to whether the tool axis remains inside or outside the boundary. When the tool exceeds the boundary, it can no longer remain vertical, so the tool axis needs to be tilted keeping the tool normal to the boundary.
Before Conversion tool path cut using Horizontal Z Contours allowing the tool to undercut and cut from bottom to top.

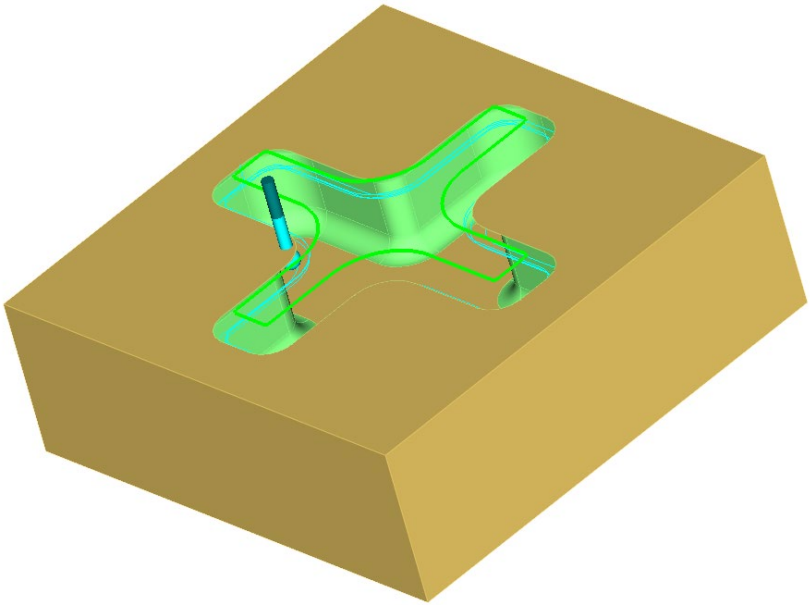


Figure 19 - Initial Horizontal Z Contours tool path colliding with the part

After 5 axis conversion with boundary.

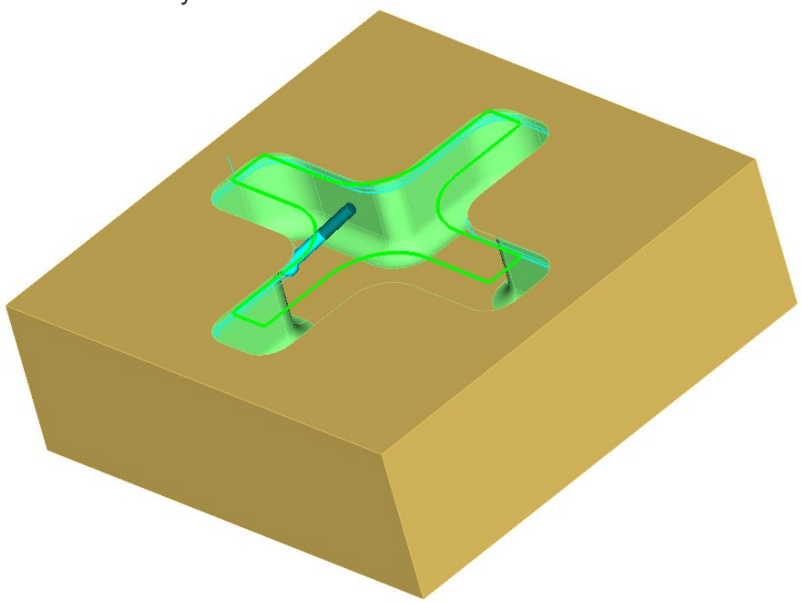


Figure 20 - Converted tool path using a boundary to prevent collisions

Project through Guide Curves

The tool axis control can be set with either one or two curves. If there is only one curve selected, then the plane that the geometry is within controls the angle of the tool axis, which is then rotated within that plane to be normal to the profile at the nearest point. If there are two guide curves, then they must be in planes that are perpendicular to each other.

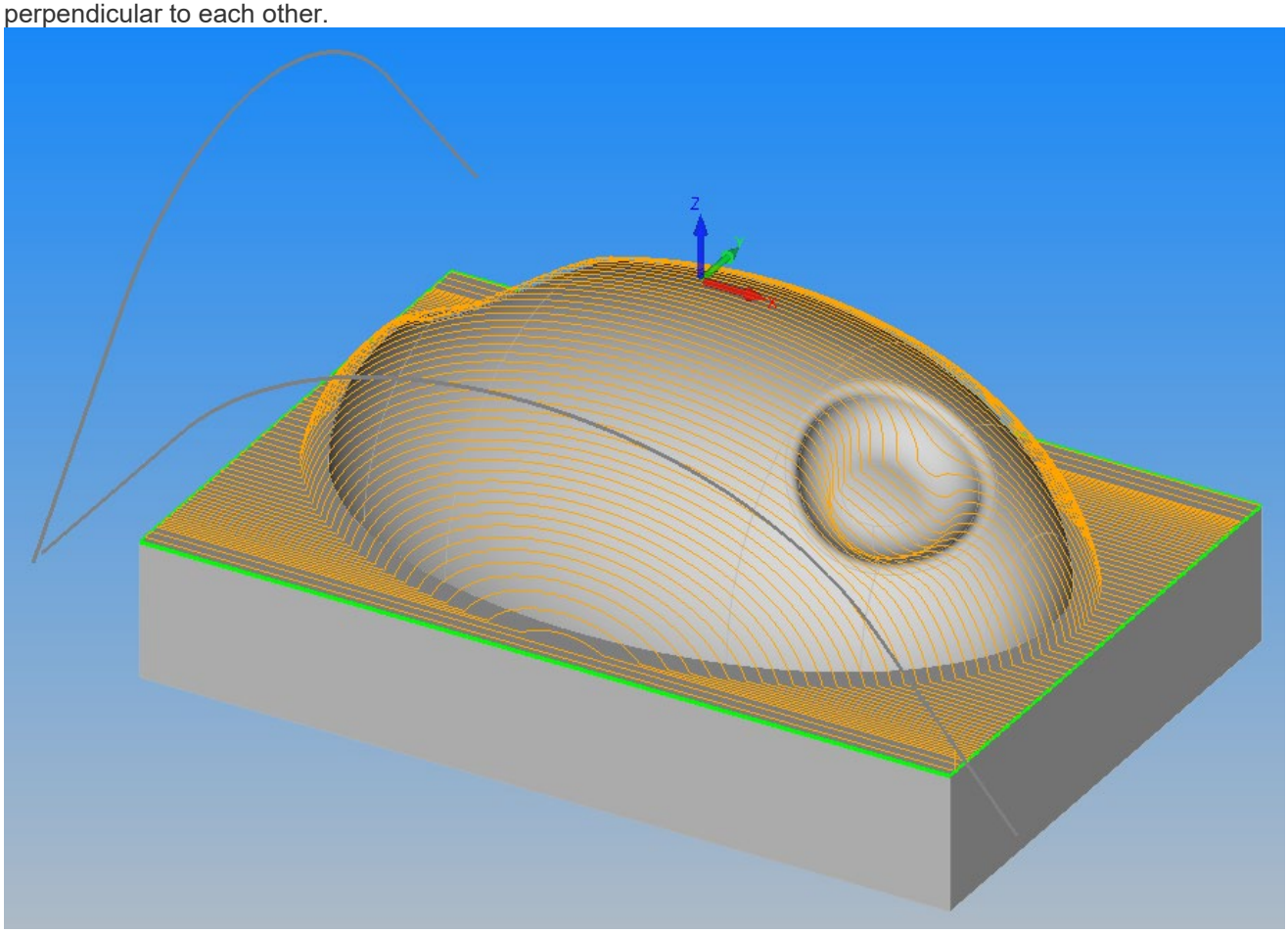


Figure 21 - Before Conversion.

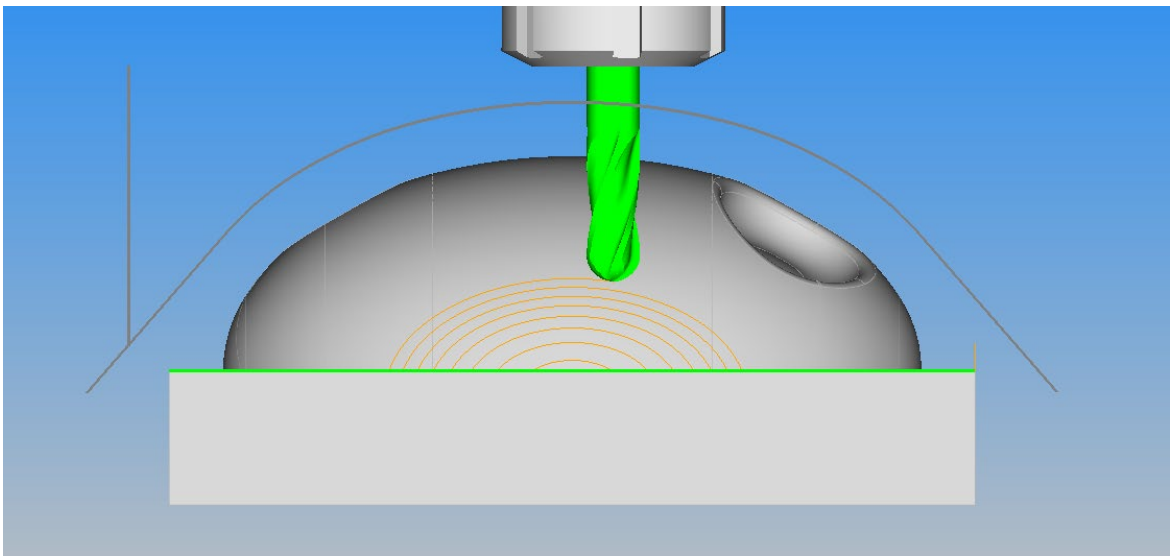
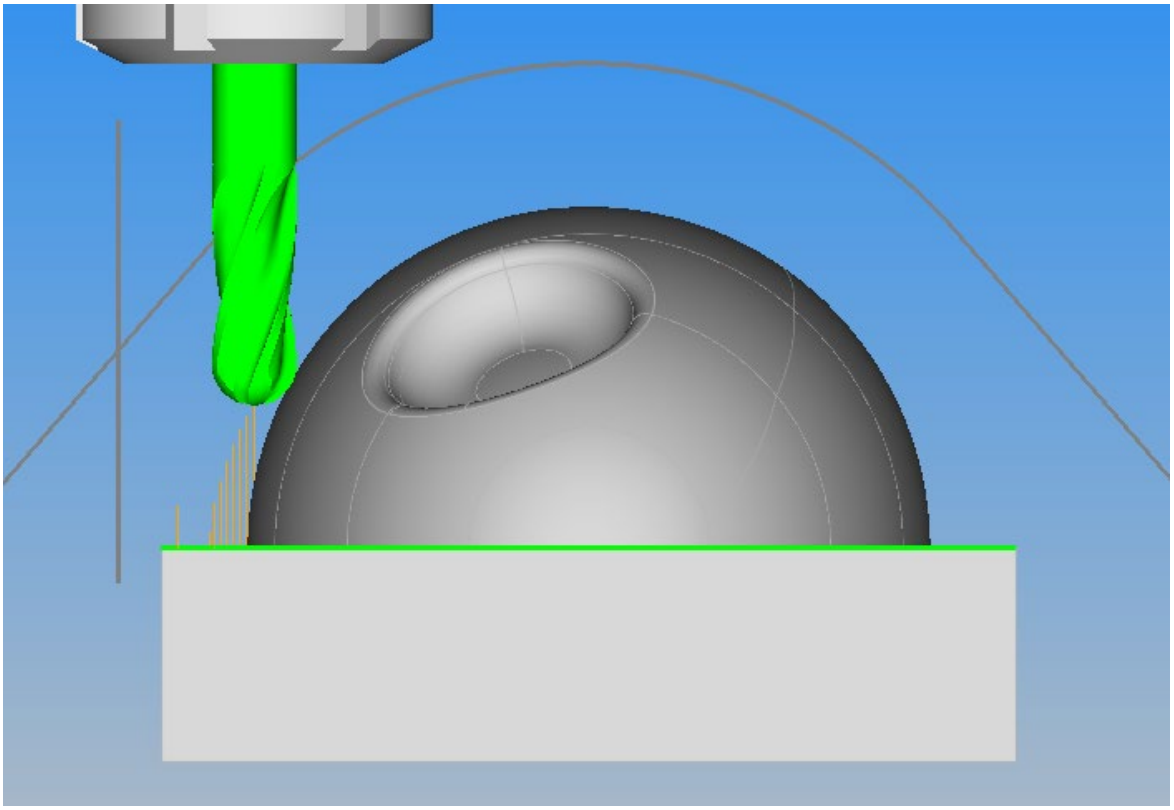


Figure 22 - Before Conversion side and end shots



After Conversion.

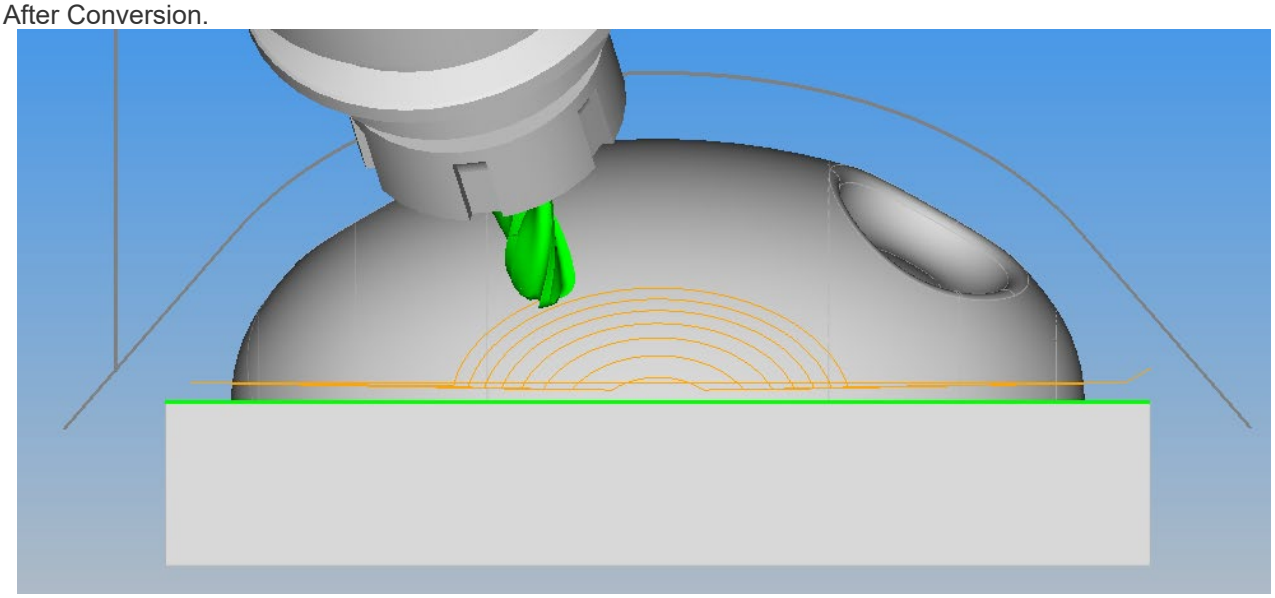
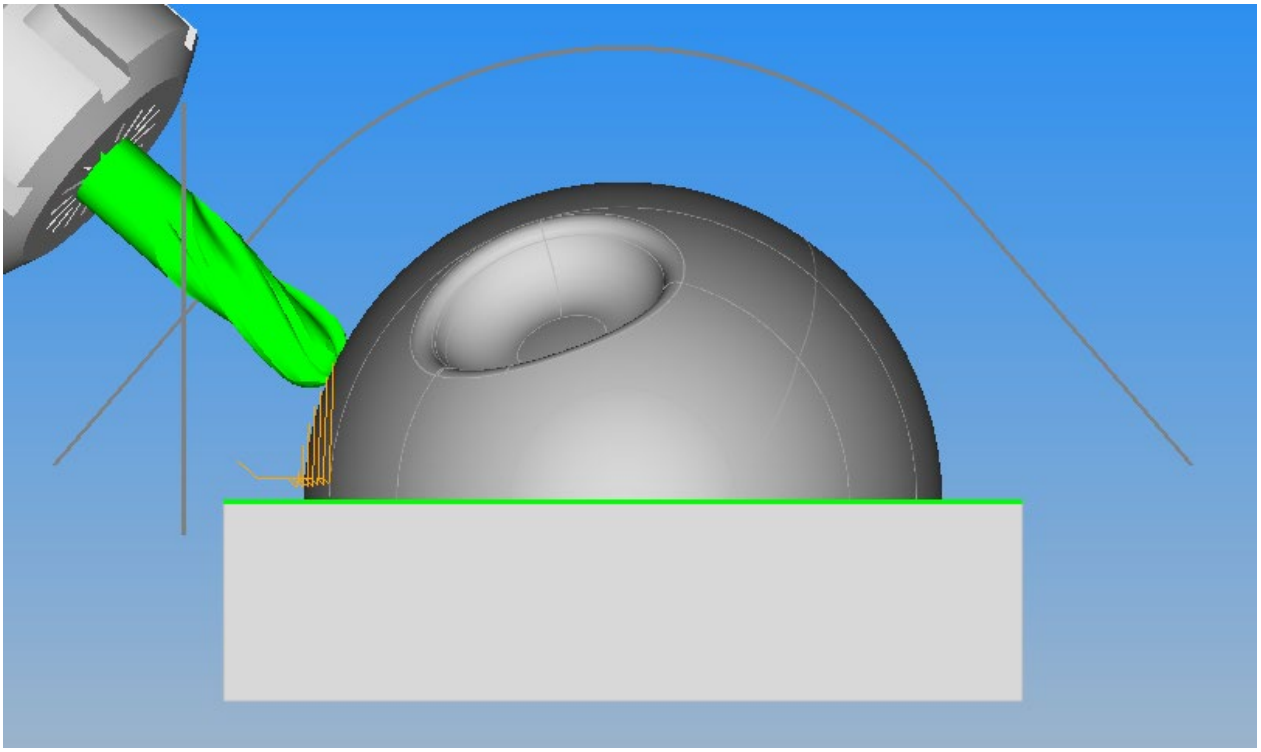


Figure 23 - Converted tool path using two guide curves



Cut Spline or Polyline

This command will drive any tool along any spline or polyline. If a spline is being machined, the Chord Tolerance is asked for, which determines the accuracy with which the straight line XYZ tool moves match curves in the spline. For a polyline, the tool is simply moved along the straight line segments.
In **Standard,** and **Advanced** modules, the tool is assumed to be vertical, with the tool tip centrally on the spline or polyline, and you are not given any options about tool angle.

If the module is **Ultimate** and you have selected a 4 or 5 axis post processor, you are able to set the tool to be **⊙ Left, ⊙ Centre** or **⊙ Right** of the spline or polyline.

The check box **☐ Show Ghost Tools will** immediately show the direction of the spline or polyline so that you can determine which side is left and which is right.

You can set the Tool Angle to be Normal to nearest Surface/Solid, Parallel to nearest Surface/Solid or at angles relative to the line or as absolute angles. For angles relative to the line, the angle is measured from the vertical, and the sign is set by looking along the direction of the spline or polyline. That is, looking at the 'back of the tool as it moves away from you along the line. If the tool is leaning to the left, the angle is counter-clockwise and is therefore positive. If the tool is leaning to the right, the angle is clockwise and the sign is negative.

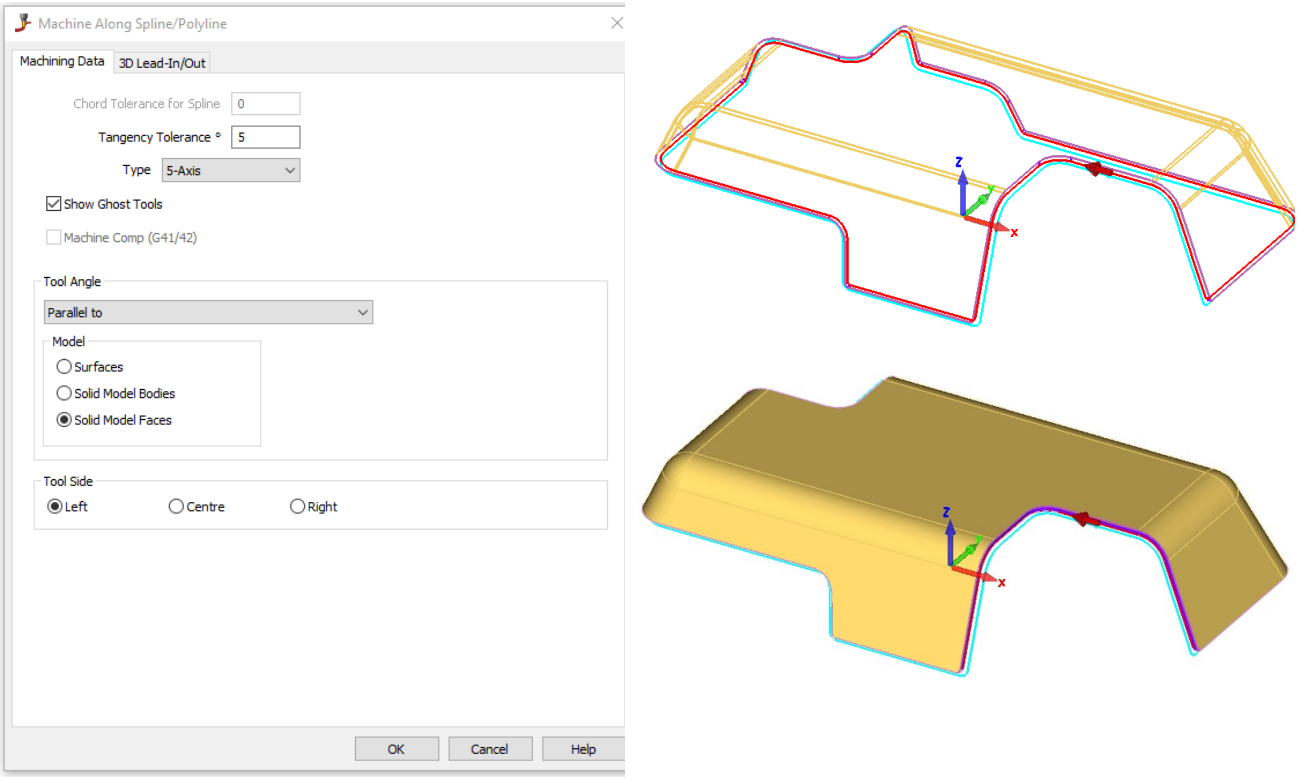


Figure 24 - Cut Spline or Polyline options using Ultimate ALPHACAM

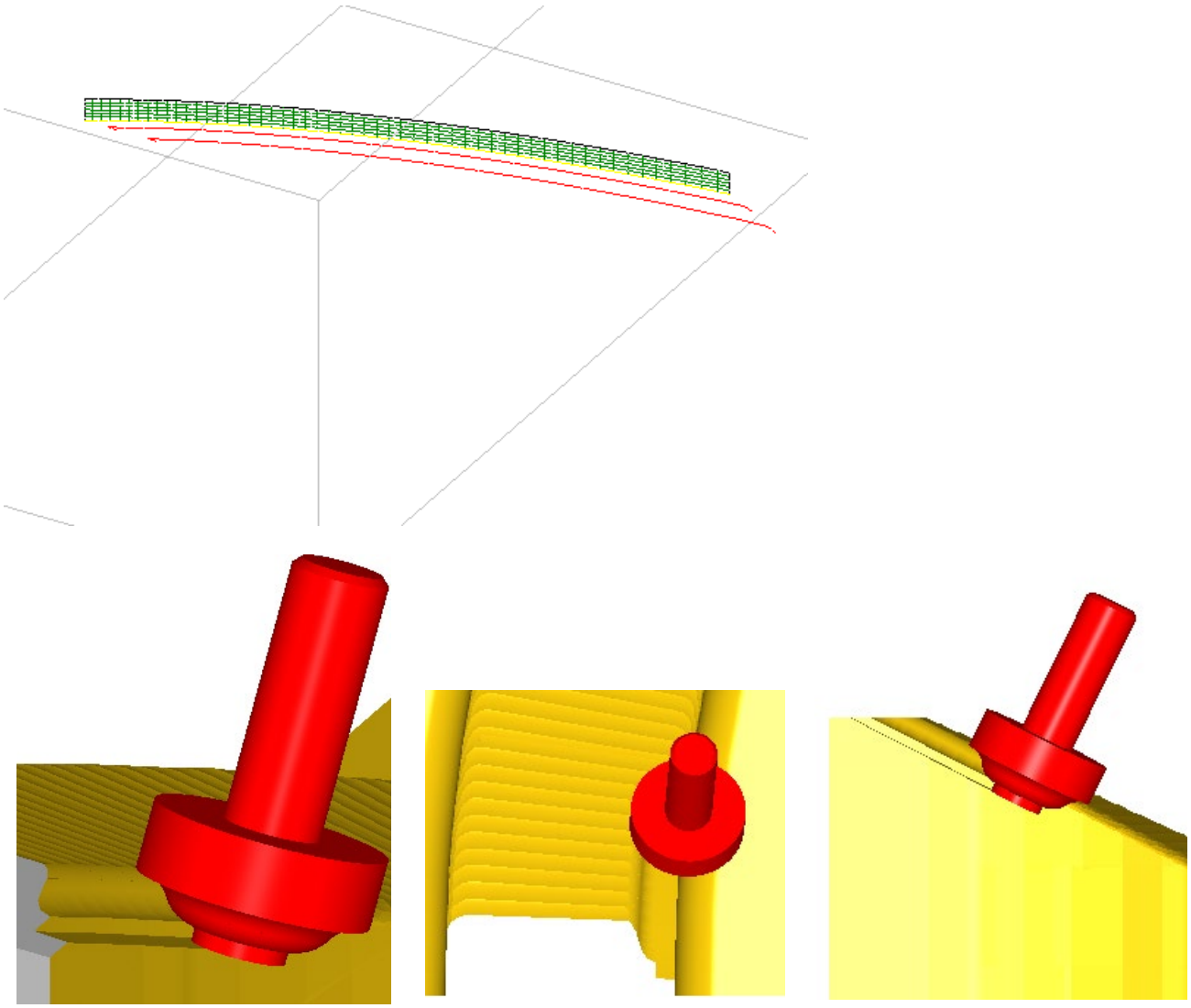


Figure 25 - Cut Spline or Polyline option applied to a form tool

If the selected post processor has been configured to take account of the ability of the controller to apply Tool Radius Compensation (G41/42) (this is indicated in the post by setting $148 to 1 and the tool has been set to be right or left of the spline or polyline, then a check box – ☐ **Machine Compensation –** is enabled. Select this to produce the necessary code.

Please note that the post has to have the correct code in $40. Post variable TCF = 1 if G41/42 is selected when ALPHACAM is used, TCX, TCY, TCZ gives the tool displacement unit vector, that is the direction from the contact point to the tool tip. CPX, CPY, CPZ gives the contact point on the polyline.

Cut With side of tool

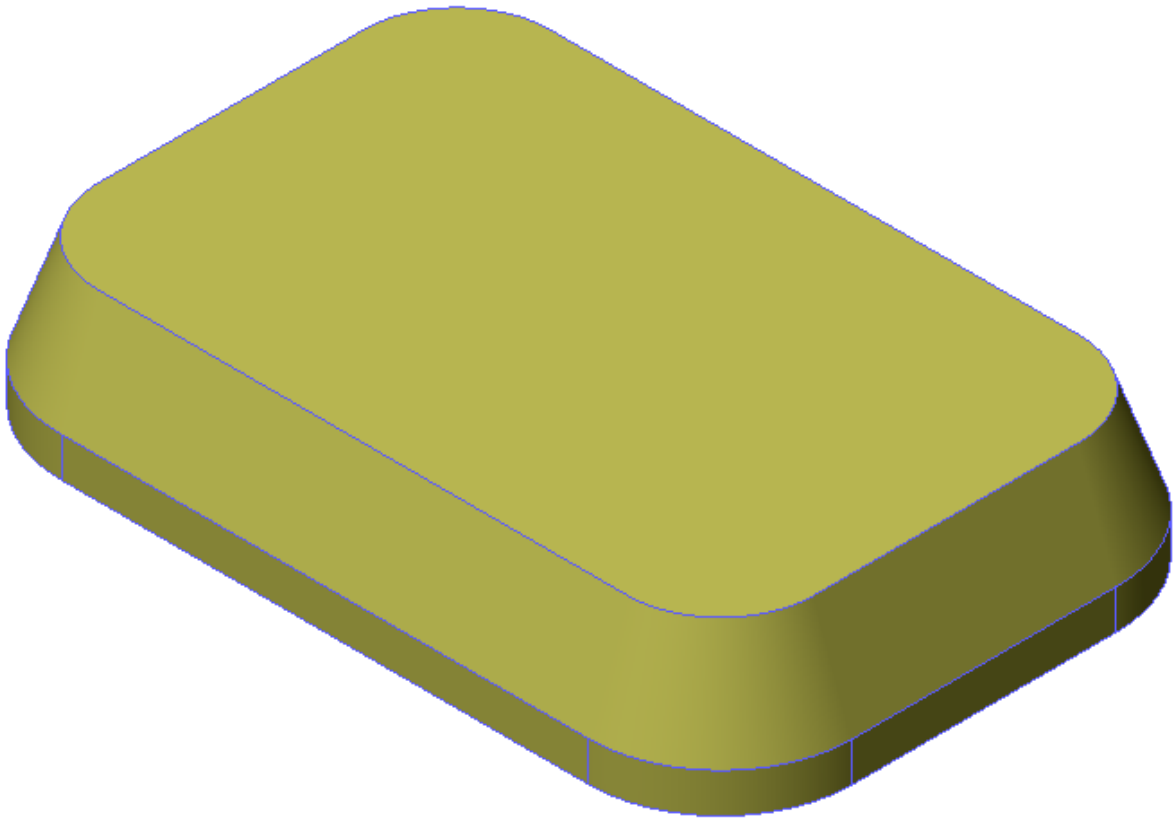


Figure 26 - Example file for Cut Spline or Polyline

The angle can be cut using either:
Cut spline or Polyline parallel to model faces
or
Cut between 2 Geometries

Practical example Chair Leg

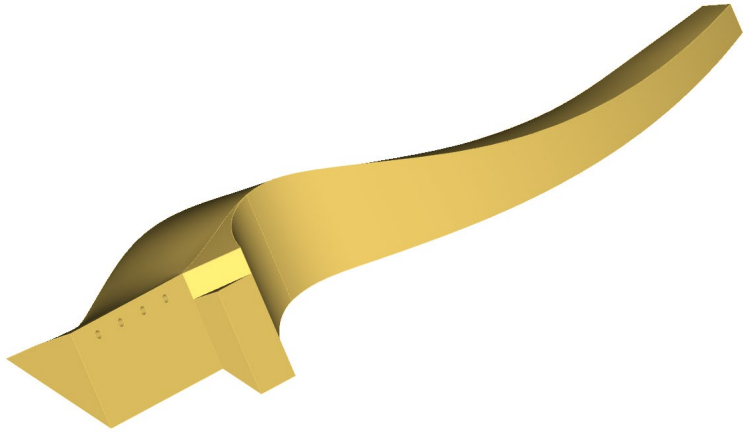


Figure 27 - Sample Chair Leg solid model

In this tutor led example we will look at the practical application of Positional 5 Axis tool paths and Simultaneous 5 Axis toolpaths so that you can understand the differing requirements.

Select **HOME > Open** and navigate to **"….\ALP TRG 210 Standard 5 Axis 2020\Examples\Drawings\"** folder and open **"Chair Leg".**

Create a material around the part using **3D > Auto Set Material.**
Make the values in the dialogue as follows**, Material Top = 0, Material Bottom = -75, Material XY Stock = 5**
Create a Rectangle from one corner of the new material to the diagonal opposite corner.

Using **MACHINE > Clamps/Fixtures > Define Clamps/Fixtures**
Select the rectangle as the shape and set the options as follows,

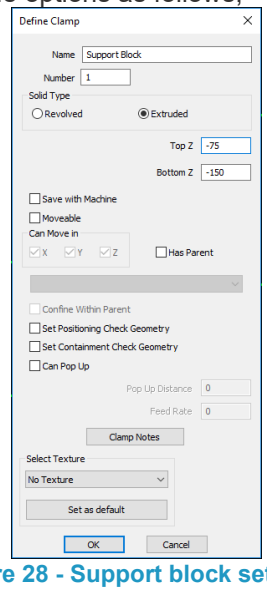


Figure 28 - Support block settings

This creates a correct material for the part to be machined from and a support block to lift the part up from the machine bed or pods so that the 5 Axis system can rotate without any collisions. This is an important consideration when working with 5 Axis toolpaths.

Roughing

Use **MACHINE > Select Tool** and from the Standard 5 Axis Folder select the **Flat 80mm + Holder.**

Using **MACHINE > 3D Machining**
Make the options as below for **Z Contour Roughing.**

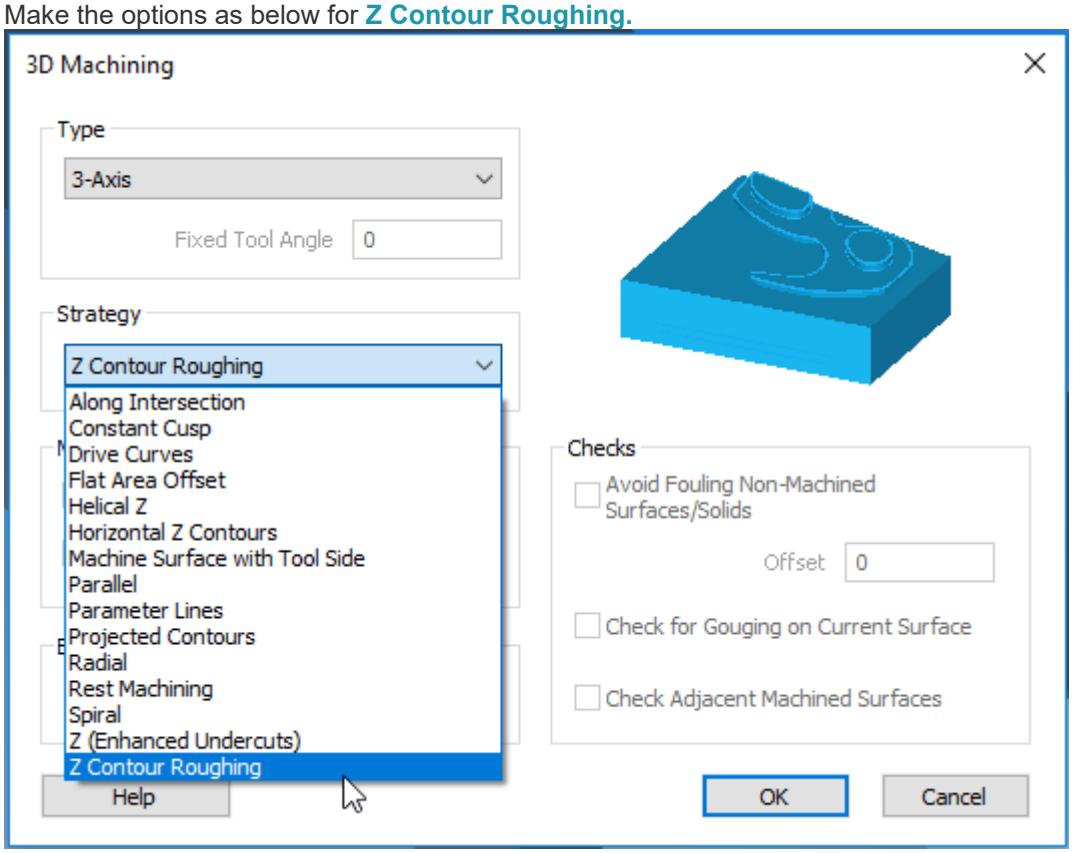


Figure 29 - 3D Machining cycle selection dialogue

General

Set the options as shown below.

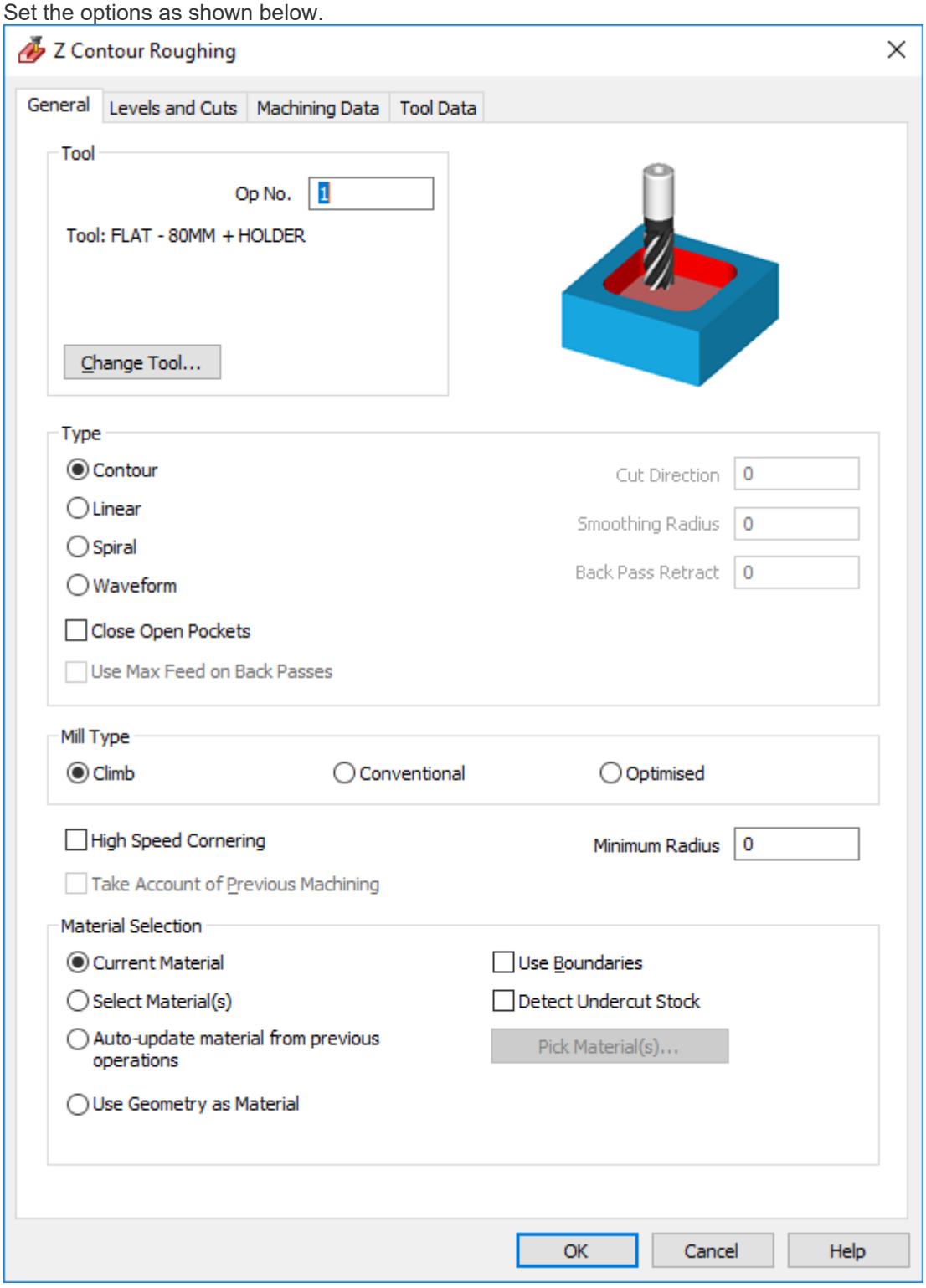


Figure 30 - Z Contour Roughing General tab.

Levels and Cuts

Set the options as shown below.

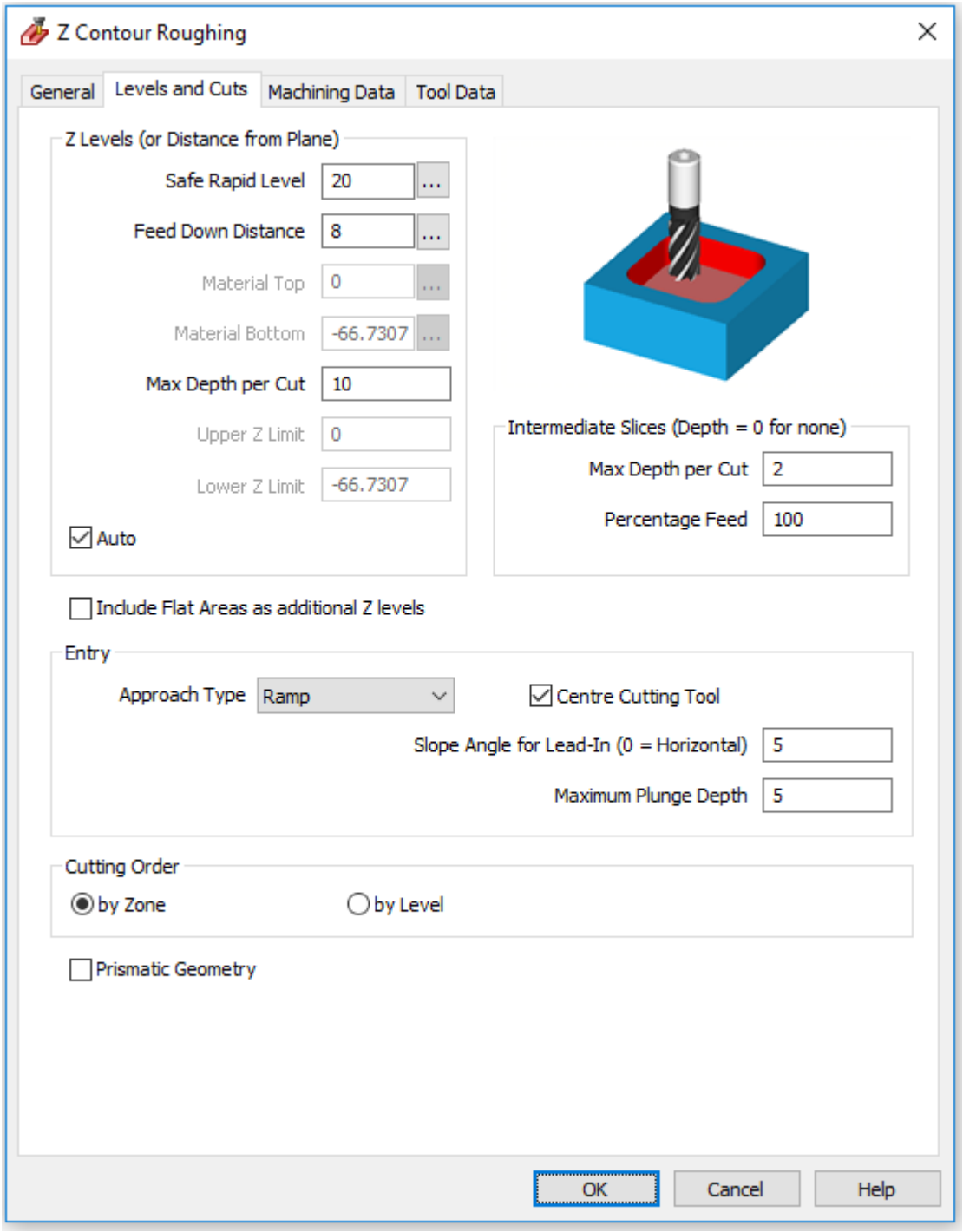


Figure 31 - Z Contour Roughing Levels and Cuts tab

Machining Data

Make the options as shown below.

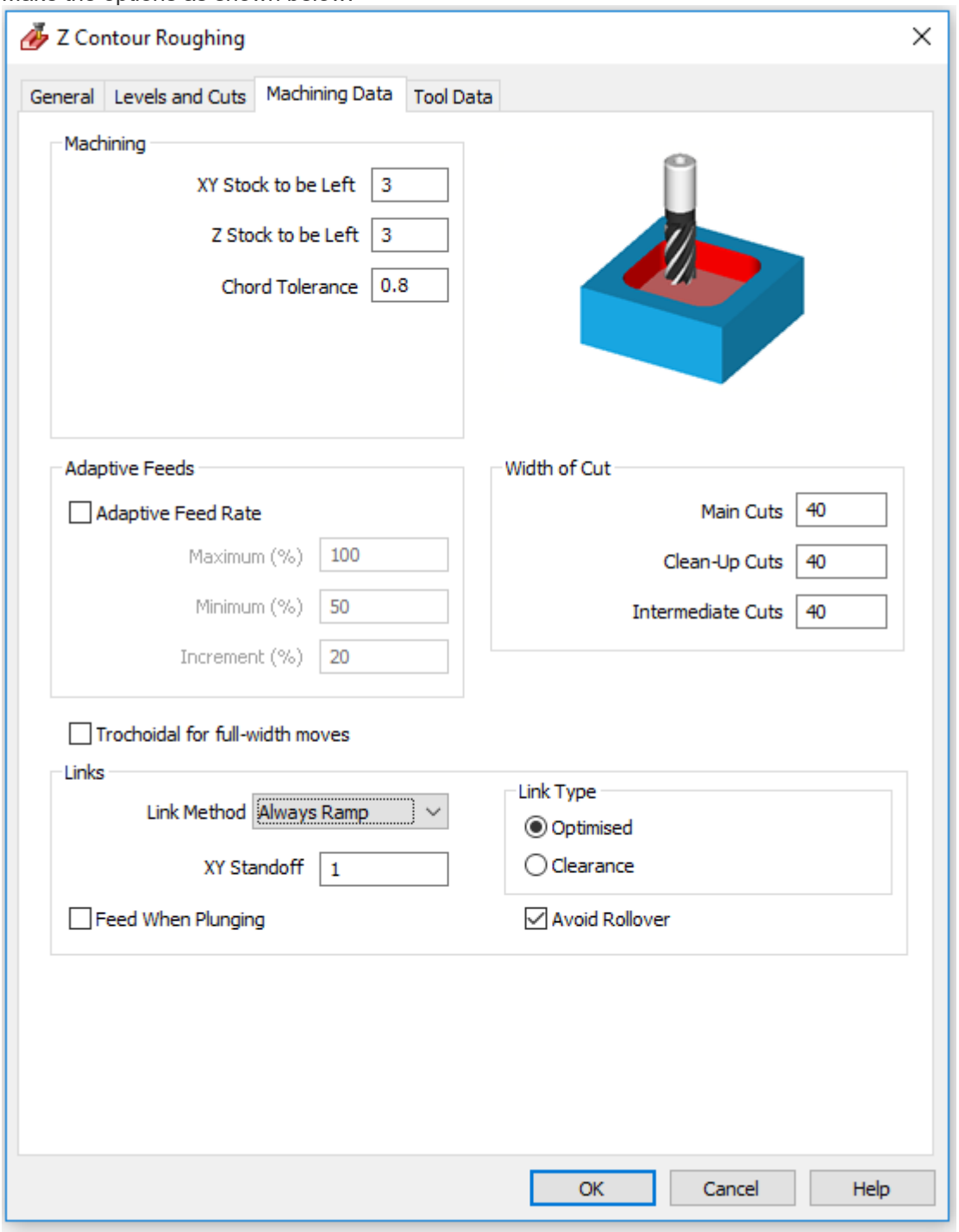


Figure 32 - Z Contour Roughing Machining Data tab

Tool Data

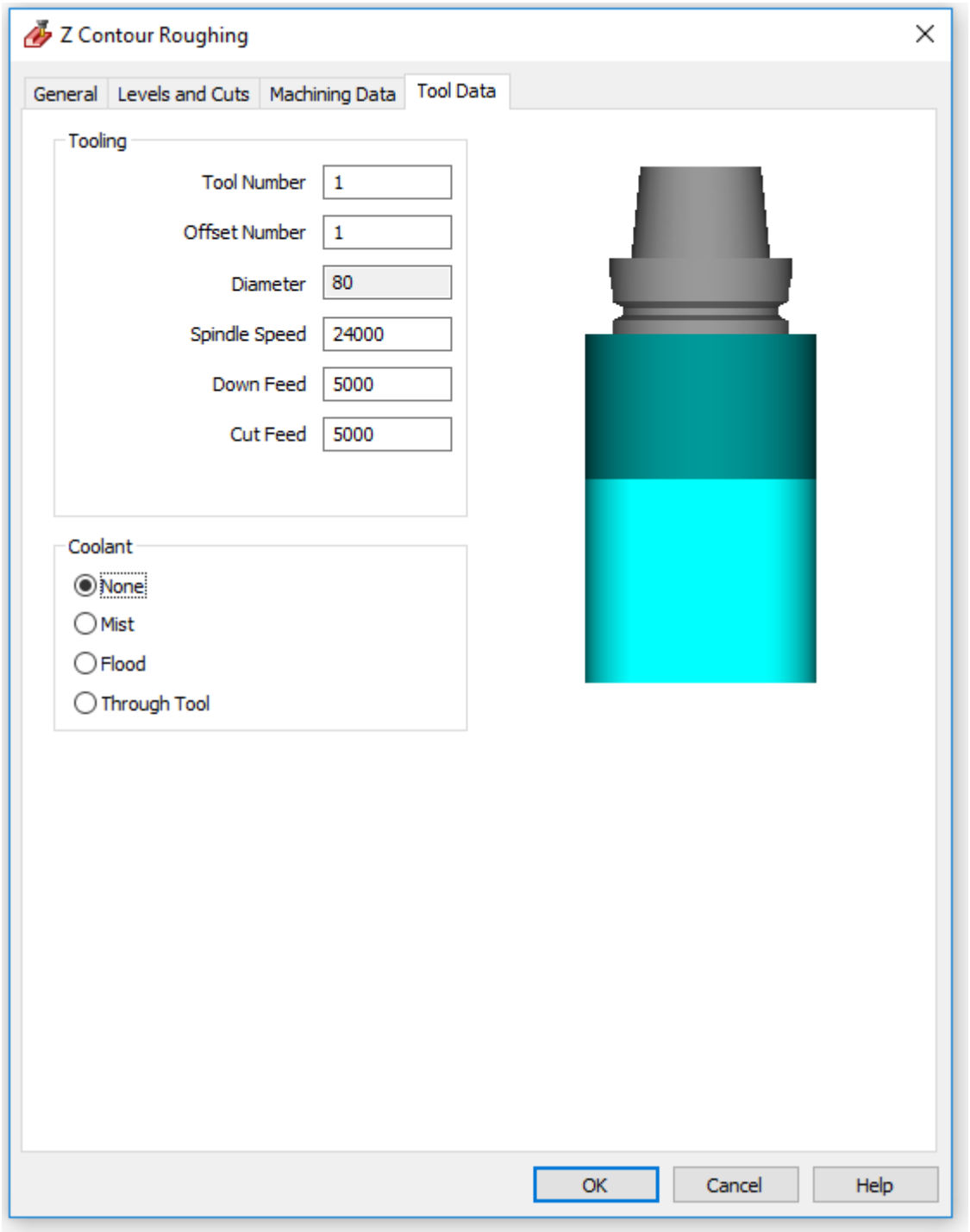


Figure 33 - Z Contour Roughing Tool Data tab

<LClick> [OK] to continue.
When prompted, **<LClick>** the chair leg model to apply the tool path.

Machining the Small Rectangular Angled Face

Setting up the Work Plane

To allow the machining to be created correctly on the fixing faces of the chair leg, first you need to create **Work Planes** that will control the **Tilt** and **Twist** angles of the 5-Axis movements.
We need to create three new work planes, one for the smallest rectangle, one to suit the two large flat areas and a final one to suit the small triangle area.

Using **WORK PLANES > From Solid Model Face** , pick the small rectangular area.

Manipulate the work plane using **WORK PLANES > Reverse Current Plane** (if required) so that the X axis points along the longer edge of the face, relocate the work plane datum with

WORK PLANES > Set Work Plane Origin to a suitable corner of the face in question.

Alter the **Properties** of this new work plane to give it a suitable name.

Machining the Small Rectangle

From the Work Planes Tab on the Project Manager, ensure that the work plane for the **Small Rectangle** is active.

Use **MACHINE > Select Tool** and choose the **Flat – 12 mm + Holder.**

Using MACHINE > 3D Machining
Make the options as below for **Flat Area Offset.**

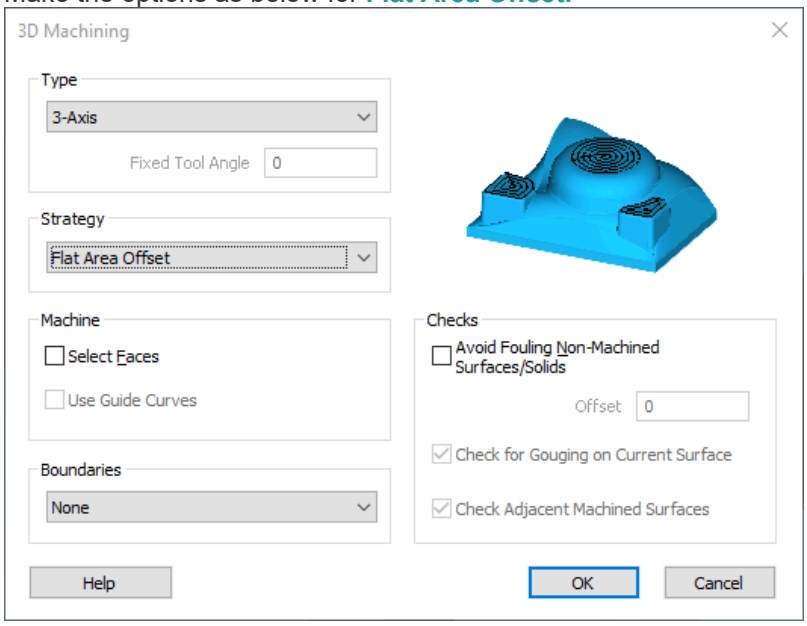


Figure 34 - 3D Machining cycle selection dialogue

<LClick>[OK] to continue.

Note the information dialogue because we are using a work plane.

General

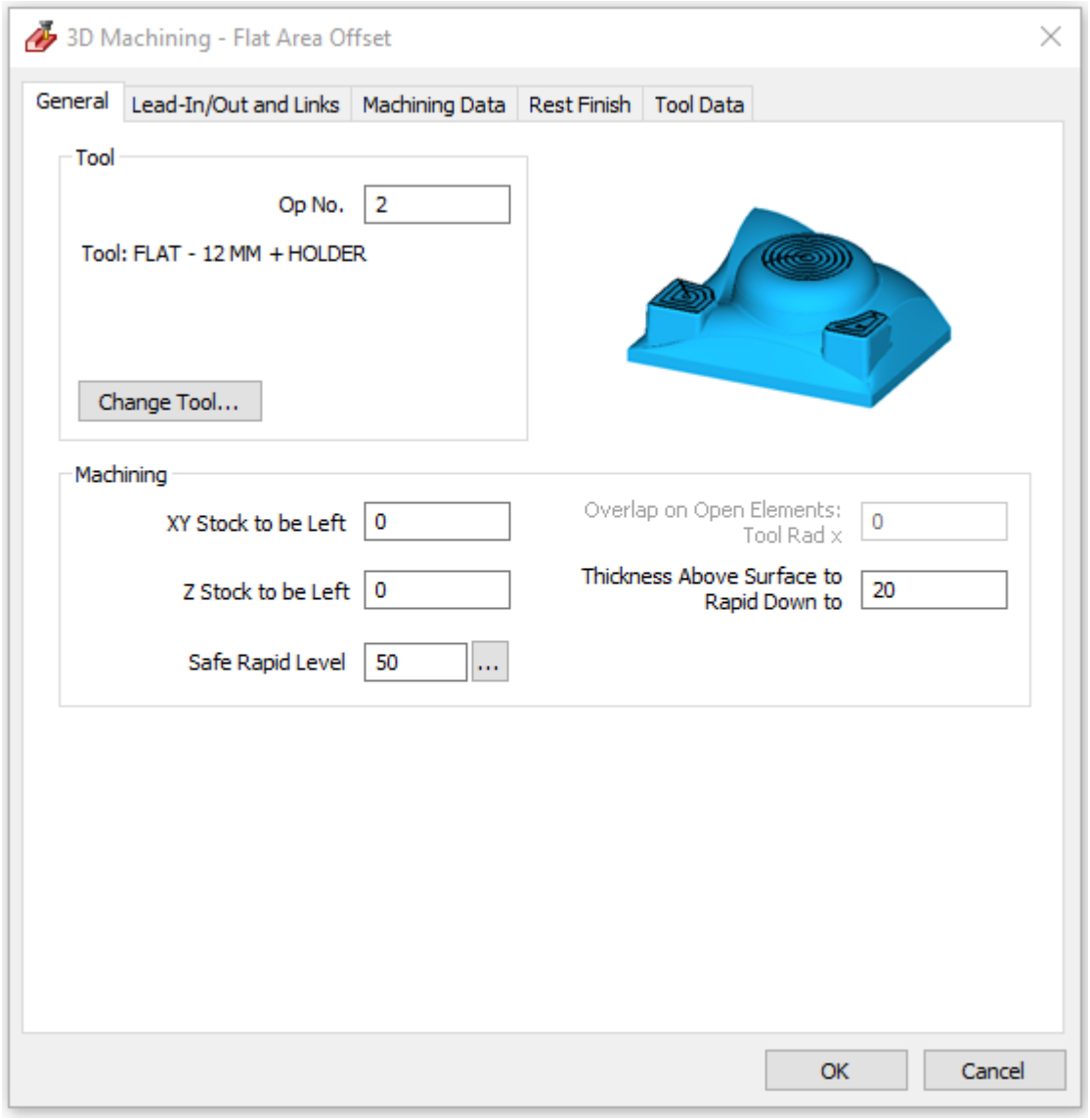


Figure 35 - Flat Area Offset General tab

Set the **Safe Rapid Level = 50** and **Thickness Above Surface to Rapid Down to = 20** these will alter to suit your machining practices when used in a real machining situation.
Both **Stock to be left** options are set to 0.

Lead-In/Out Links

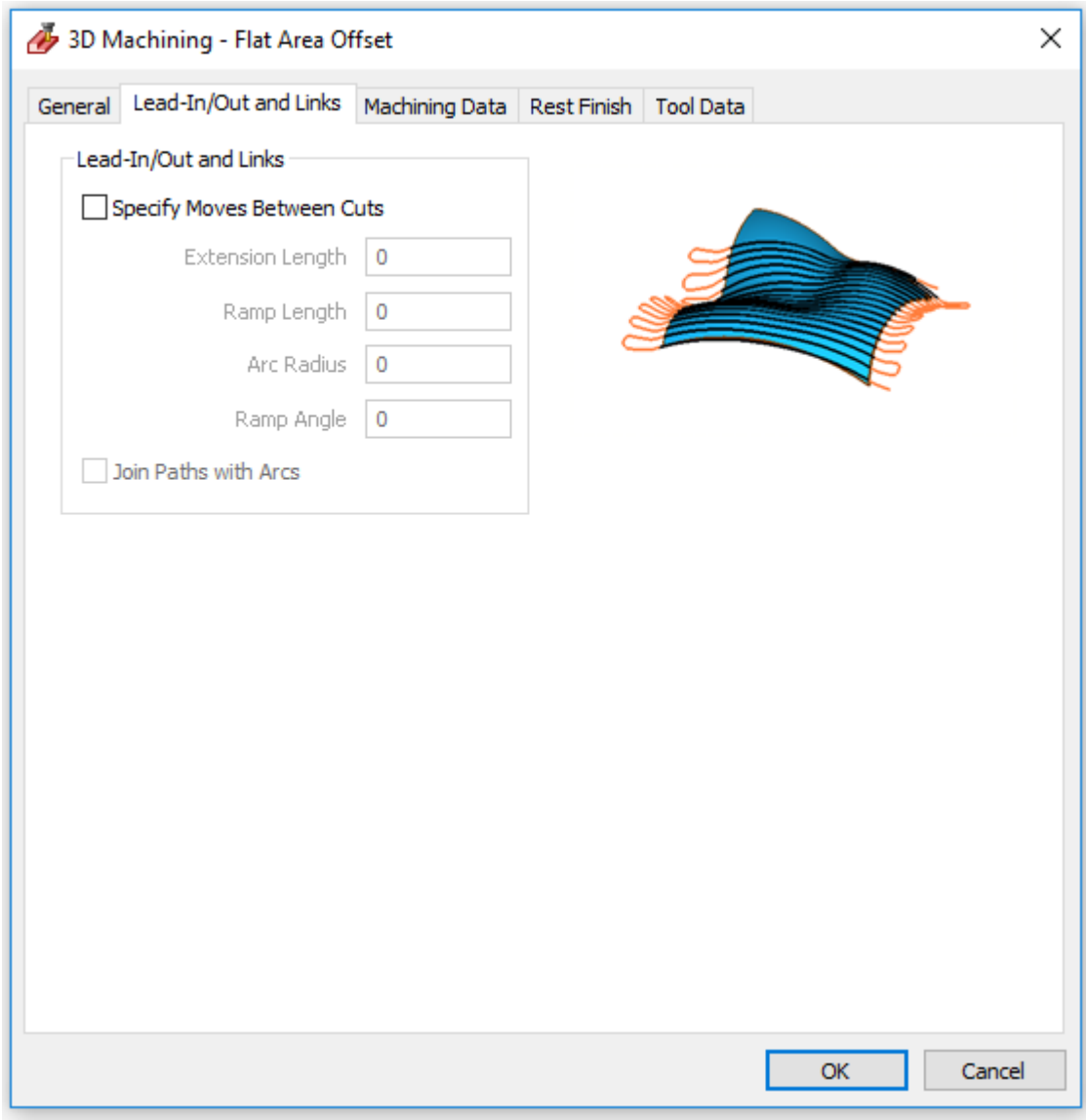


Figure 36 - Flat Area Offset Lead-In/Out Links tab

The **Lead-In/Out Links** tab allows the user to add any lead in or link moves to the toolpath as required to achieve the best results.

Machining Data

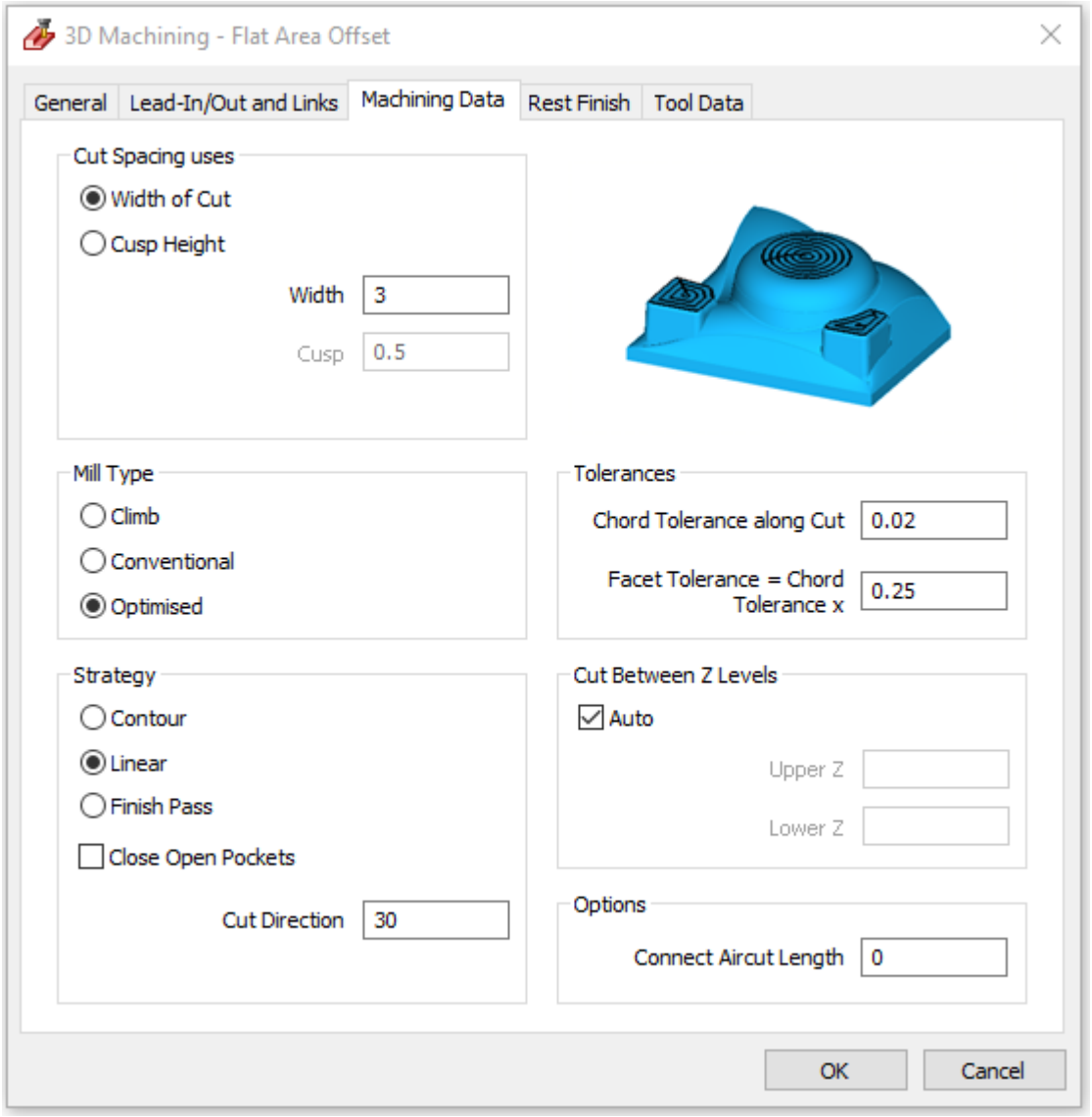


Figure 37 - Flat Area Offset Machining data tab

Set **Width = 3,**
⊙ Optimised,
⊙ Linear,
Cut Direction = 30
and **☑ Auto.**

Rest Finish

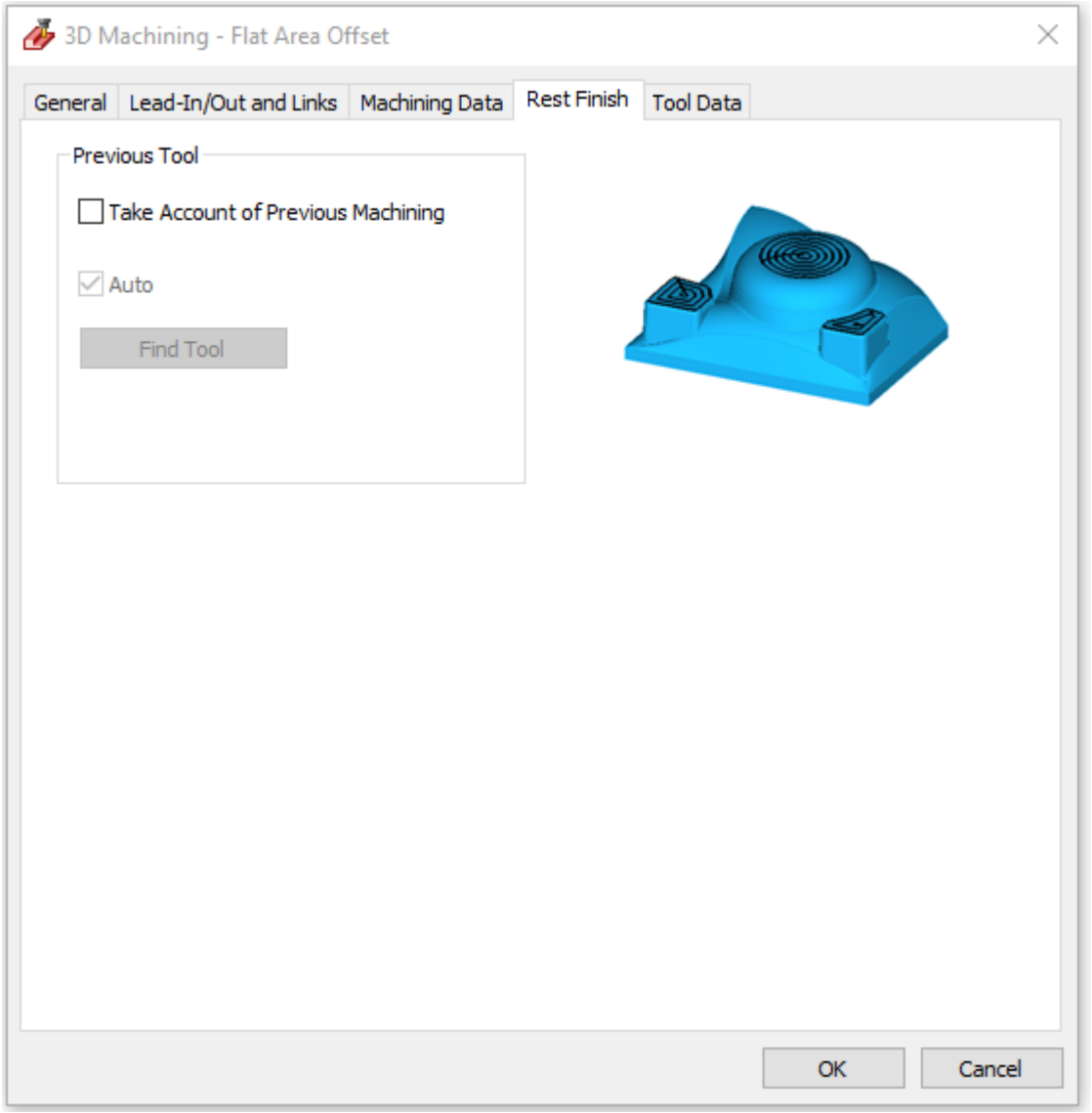


Figure 38 - Flat Area Offset Rest Finish tab

No options are required as this is the first tool to be used.

Tool Data

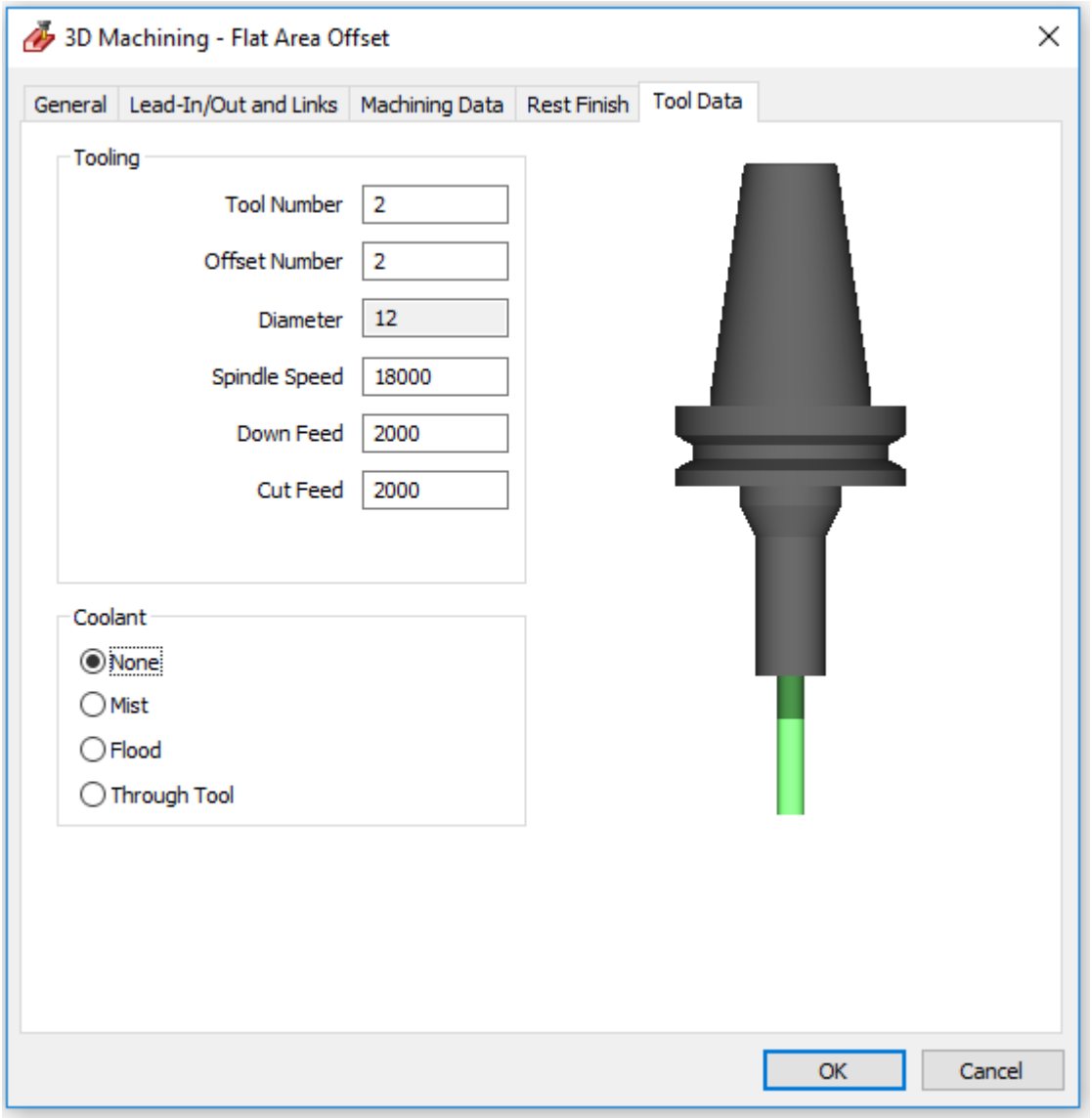


Figure 39 - Flat Area Offset Tool Data tab

<LClick> [OK] to continue and select the **Chair Leg** model as the solid to machine.

Machining the Two Large Rectangular areas

Setting up the Work Plane

Using **WORK PLANES > From Solid Model Face**  , pick the upper large rectangular area.

Manipulate the work plane using **WORK PLANES > Reverse Current Plane**  (if required) so that the X axis points along the longer edge of the face, relocate the work plane datum with

WORK PLANES > Set Work Plane Origin  to a suitable corner of the face in question.

Alter the **Properties**  of this new work plane to give it a suitable name.

Machining the Large Rectangles

From the Work Planes Tab on the Project Manager, ensure that the work plane for **Two Large Rectangles** is active.

Select **MACHINE > 3D Machining**  .
Choose the machining strategy **Flat Area Offset.**

Make all the settings as the previous machining (if you have not had to close ALPHACAM, they will be remembered). Alter the Cut Direction to 0 or 90 to give a better tool path.

Set the **Safe Rapid Level = 50** and **Thickness Above Surface to Rapid Down to = 20** these will alter to suit your machining practices when used in a real machining situation.

Both **Stock to be left** options are set to 0.

On the final dialogue page set **Width of cut = 3, ⊙ Optimised, ⊙ Linear, Cut Direction = 90** and **☑ Auto.**

Click [OK] to continue and select the **Chair Leg model** as the solid to machine.

Machining the Small triangle area

Setting up the Work Plane

Using **WORK PLANES > From Solid Model Face**  , pick the small triangular area.

Manipulate the work plane using **WORK PLANES > Reverse Current Plane**  (if required) so that the X axis points along the longer edge of the face, relocate the work plane datum with

WORK PLANES > Set Work Plane Origin  to a suitable corner of the face in question.

Alter the **Properties**  of this new work plane to give it a suitable name.

These three work planes will allow us to place 3D machining strategies and conventional 2D machining onto these areas easily.

Machining the Small Triangle

From the Work Planes Tab on the Project Manager, ensure that the work plane for the **Small Triangle** is active.

Select **SOLID MODEL EXTRACT > Contour from Picked Edges**  .
Pick the **Top** edge of the triangle and large flat area next to it.

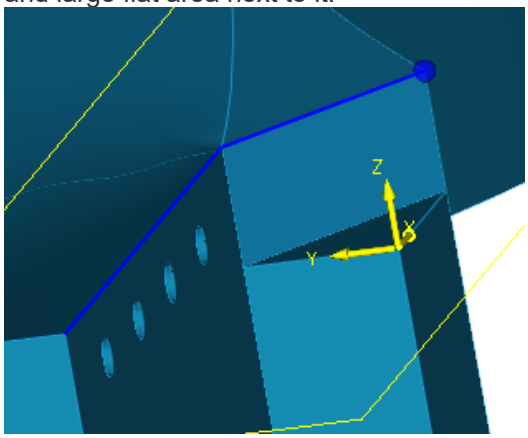


Figure 40 - Top edge for extraction

Use an edge of the **Triangle** face as the **Bottom** edge and one of the top edges as the **Top.**

We do not need to machine the entire length of the geometry, so we need to use

EDIT > Break, Join etc. > Break  on the long edge above the hole closest to the triangle flat. Then Delete the unwanted piece of geometry.

Use **MACHINE > Rough/Finish**  **⊙ Auto Z** and **Selected.** Complete the dialogue boxes to suit, making the **Z Stock Amount = 0** to finish the profile directly onto the flat triangle face.
Use **Apply Auto Lead In/Out** to drive onto and from the part.

Finish large flat side wall

From the Work Planes Tab on the Project Manager, ensure that the work plane for **Two Large Rectangles** is active.

Select **3D > SOLID MODEL EXTRACT > Contour from Picked Edges**  .
Pick the **Top** edge of the wall that meets the smaller of the two flat areas.

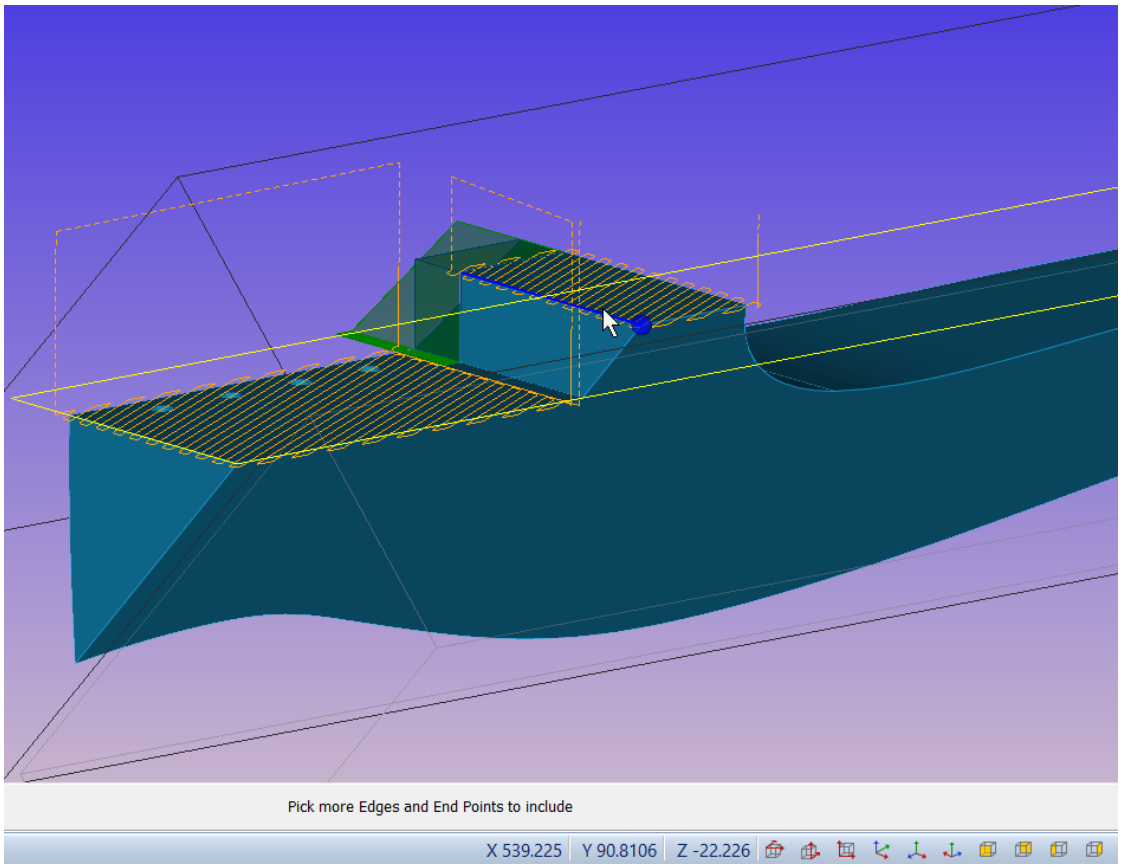


Figure 41 - Edge to Extract

The bottom reference is any edge of the lower flat; the top reference is any edge on the upper flat.

Extend this contour by each end using **EDIT > Break, Join etc. > Extend By Distance** 
Use 45mm as the value to extend above the holes then 30mm to extend the lower section.

> This prevents excessive Lead In/Out manipulation to achieve the desired machining of the edge.

Use **MACHINE > Rough/Finish**  **⊙ Auto Z** and **Selected.**
Complete the dialogue boxes to suit, making the **Z Stock Amount = 0** to finish the profile directly onto the flat face.
Use **Apply Auto Lead In/Out** to drive onto and from the part.

Drill Holes

From the Work Planes Tab on the Project Manager, ensure that the work plane for **Two Large Rectangles** is active.

Use **SOLID MODEL EXTRACT > Automatic Extraction** make the options to **⊙ Drillable holes, ⊙ Use Current** and **⊙ All bodies.**

> Note that the Z levels are based on the value from the work plane so you do not need to create a secondary work plane on the face that the holes actually occupy.

Use **MACHINE > Select Tool** and select the **Drill - 6mm + holder.**

Select **MACHINE > Drill/Machine Holes Auto Z**
Set options as **Holes to Match Drill Diameter,** accept the default settings on the remaining dialogue pages or alter to suit depending on your needs.
Select the extracted hole details on the flat face as the required geometries.

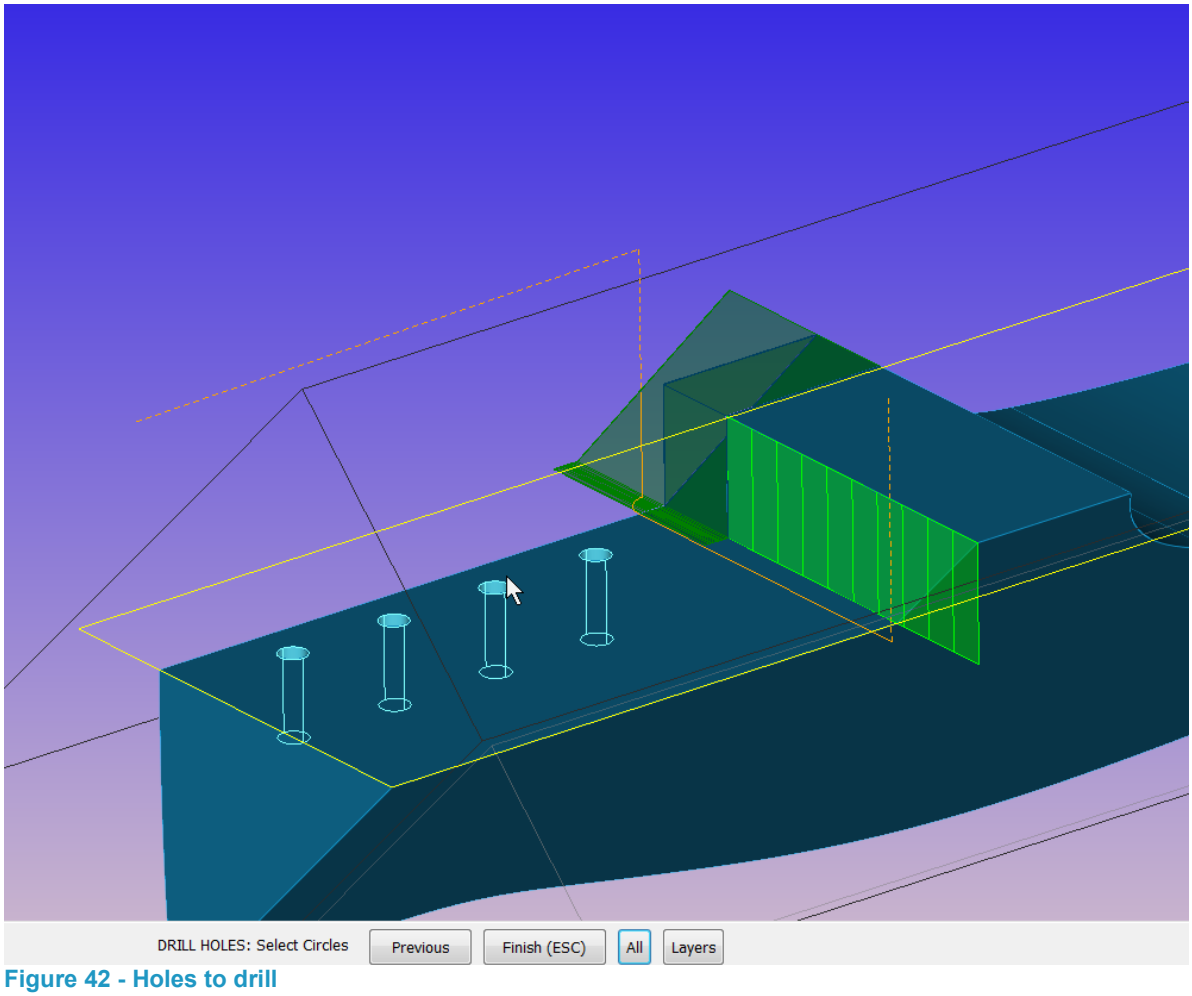


Figure 42 - Holes to drill

Profiling the Leg

The type of machining strategies used to create the actual finished faces of the Chair Leg require the use of Primary and Auxiliary geometries to drive the tool in a simultaneous 5 axis movement.

Firstly, if any work planes are active, select **Work Plane > Cancel Work Plane** 
To acquire the correct geometries to drive the machining we need to extract edges from four sections of the solid model.

Using **SOLID MODEL EXTRACT > 3D Edge Extraction**  acquire the profile along the bottom edge of the model on the front side. Use the same command to acquire the 3D edge running along the spine of the leg. These two polylines will form the basis of the machine driving elements for the front face.

Using the same techniques extract the edges from the bottom edge at the rear of the model and along the spine at the rear of the model. The edges required on the rear detail need to follow from the small triangle flat end of the leg to the end of the swept curve and no further.

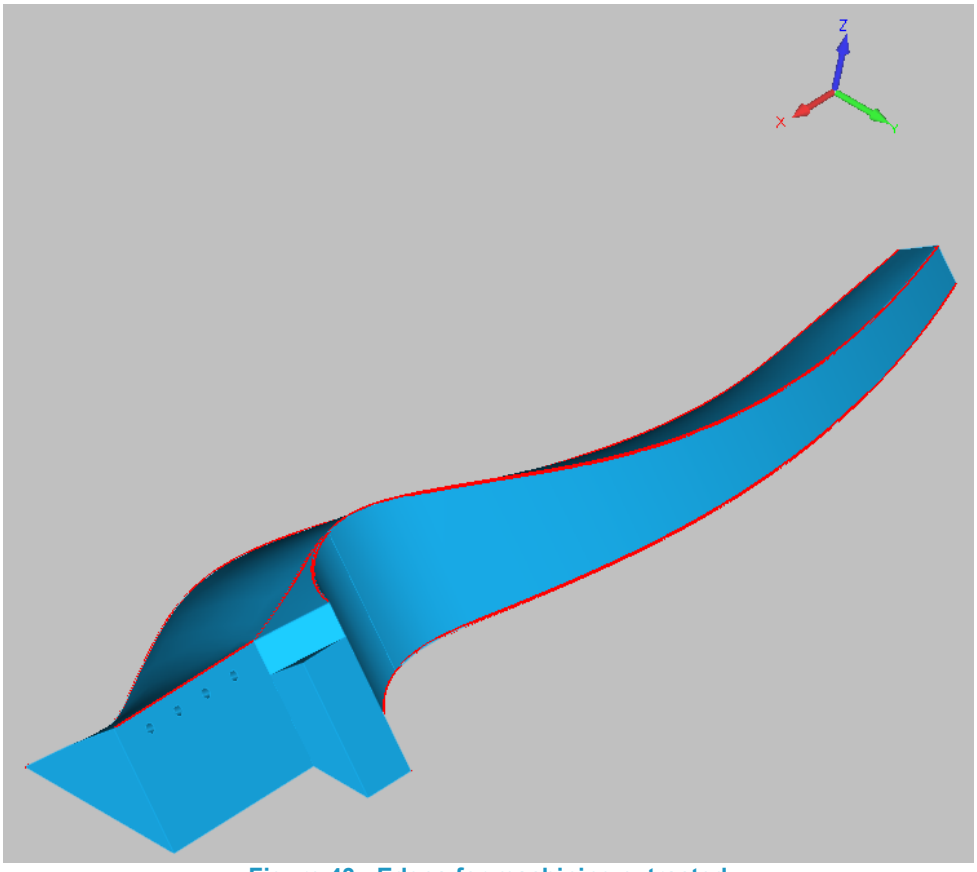


Figure 43 - Edges for machining extracted

As an aid to the manufacturing process and to ensure that the tool does not collide with the part we extend these four Polylines away from the ends of the model.

Use **GEOMETRY > Edit 3D Polyline** ,in the dialogue box

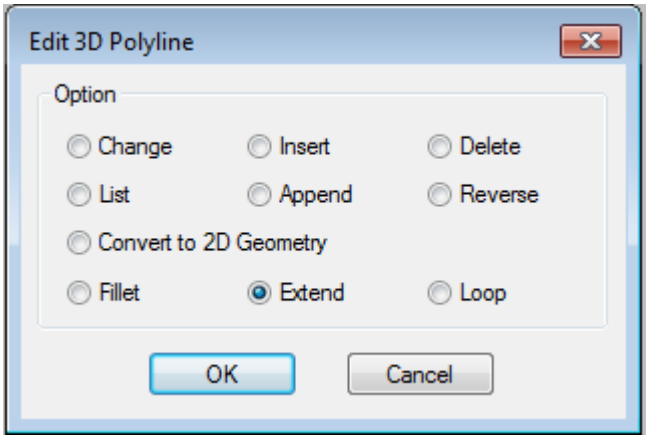


Figure 44 - Edit 3D Polyline dialogue options

Choose **⊙ Extend. <LClick> [OK].**

In the next dialogue.

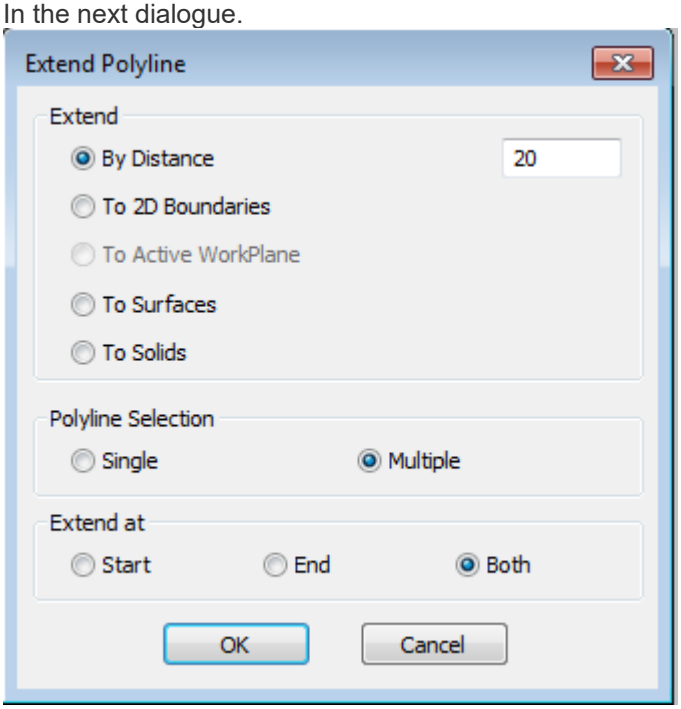


Figure 45 - Setting the Extend options to all four polylines simultaneously

Set the option to **⊙ By Distance** and make the value 20mm.
Make the option for line selection to **⊙ Multiple** and the ends option to **⊙ Both**
<LClick> [OK] to continue, the **<LClick> [All]** at the bottom to select the four polylines.
<LClick> [Finish] to complete.

In some cases, you may need to Zoom in close to the Polylines to choose the correct part of them to allow the Extend command to function correctly should the above settings not work.

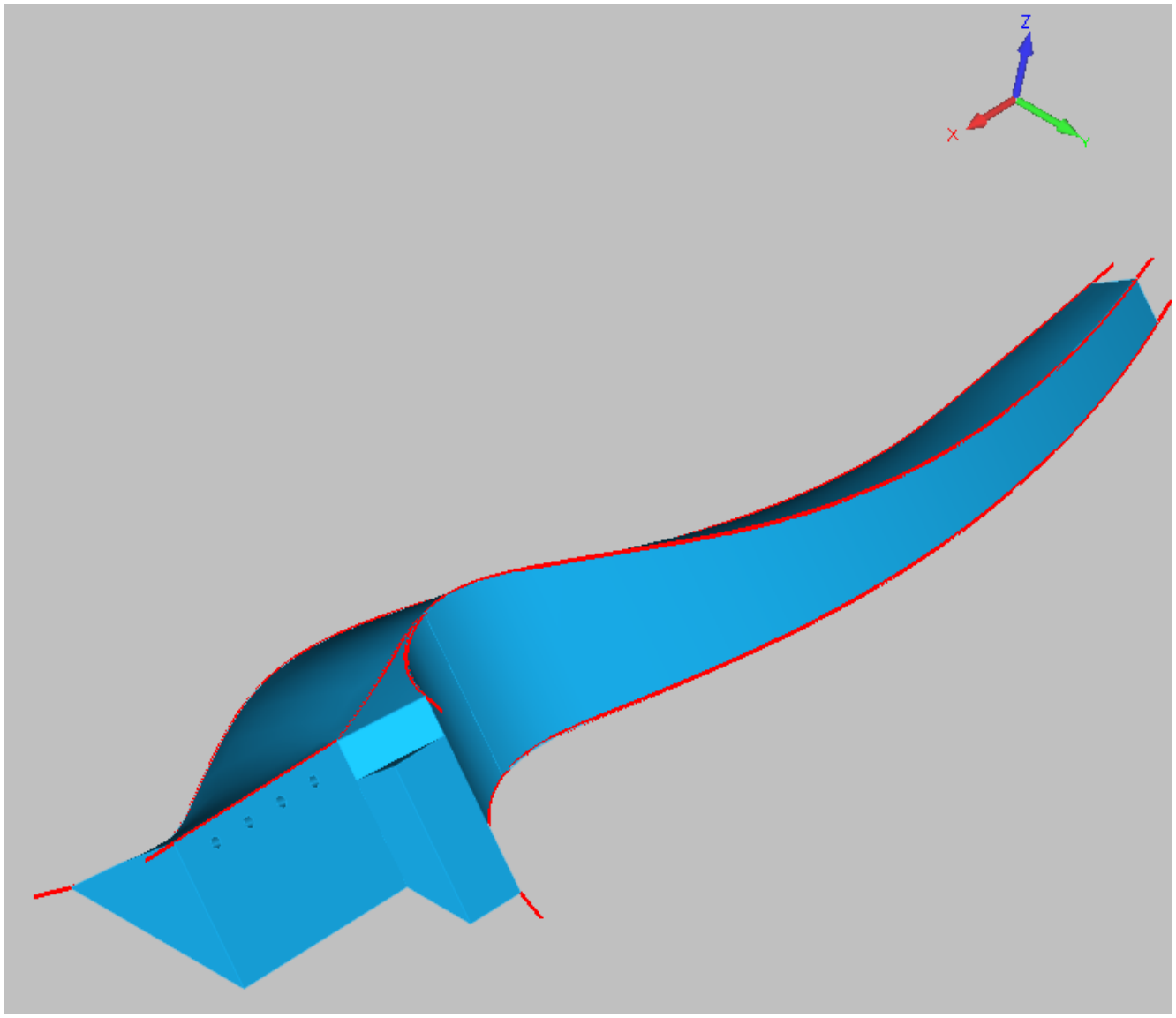


Figure 46 - 3D Polylines extended to suit

<LClick> [Finish] to complete the actual command.

Cut Between 2 Geometries theory.

When using the command **Cut Between 2 Geometries**, there are two main areas to take into consideration.

- Ghost Tool direction.
- Length of lines.

Ghost Tools

In any use of this type of machining cycle, the direction of the ghost tools is important as in all machining, however, as we will be working with one line guiding the bottom of the cutter and another guiding the upper level of the cutter, it is even more important that that the pair are both travelling in the same direction. Consider a simple 2D straight cut; this can be illustrated as this image.

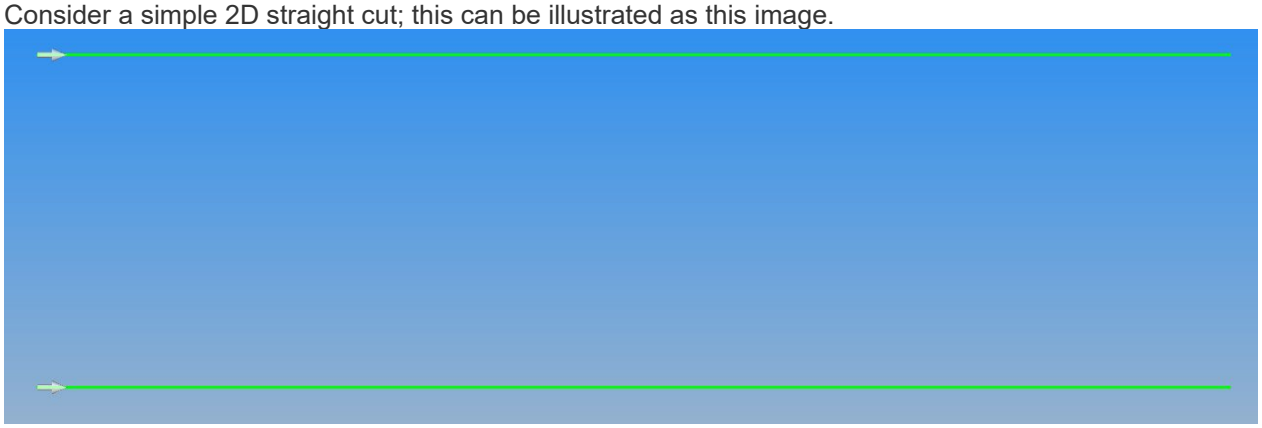


Figure 47 - Cut Between 2 Geometries theory

With one line representing the top of the part and another for the bottom, both line travelling in the same direction giving a perfect cut.

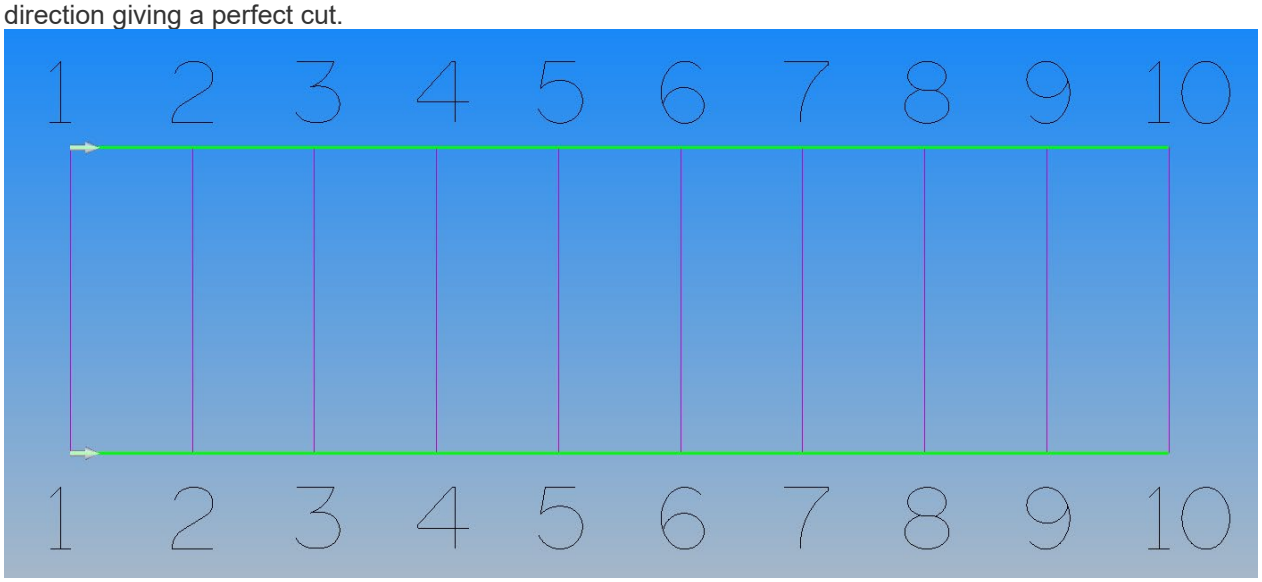


Figure 48 - How point matching works when lines are set correctly

The tool path is matching up the lines in a system similar to the construction lines and numbers shown.

Now consider what happens if the **Ghost Tools** are in opposite directions.

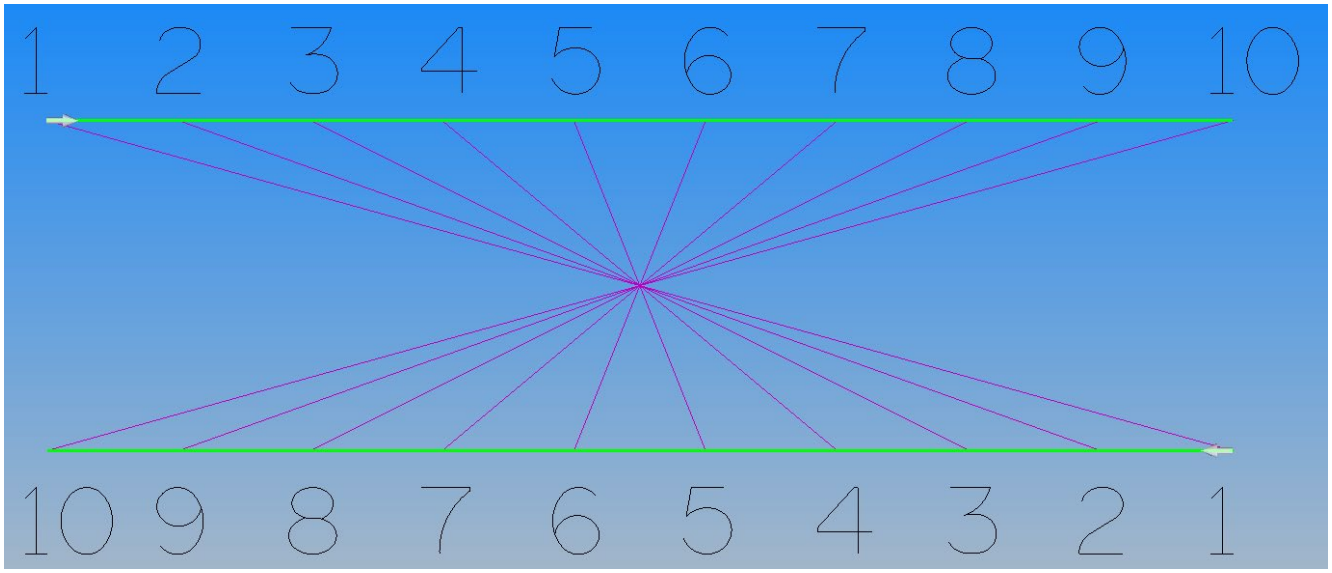


Figure 49 - When ghost tools are opposite to each other

The process will still link up the numbers, but now the tool will be leaning across the part to start with, standing straight in the centre, then leaning across to finish.

This is an unacceptable action for the tool.

Line Length

In a similar manner as the Ghost Tools, but not as catastrophic, are the lengths of the lines.
Again, looking at the first example, the lines are both as long as each other and will give a vertical tool path action when applied.

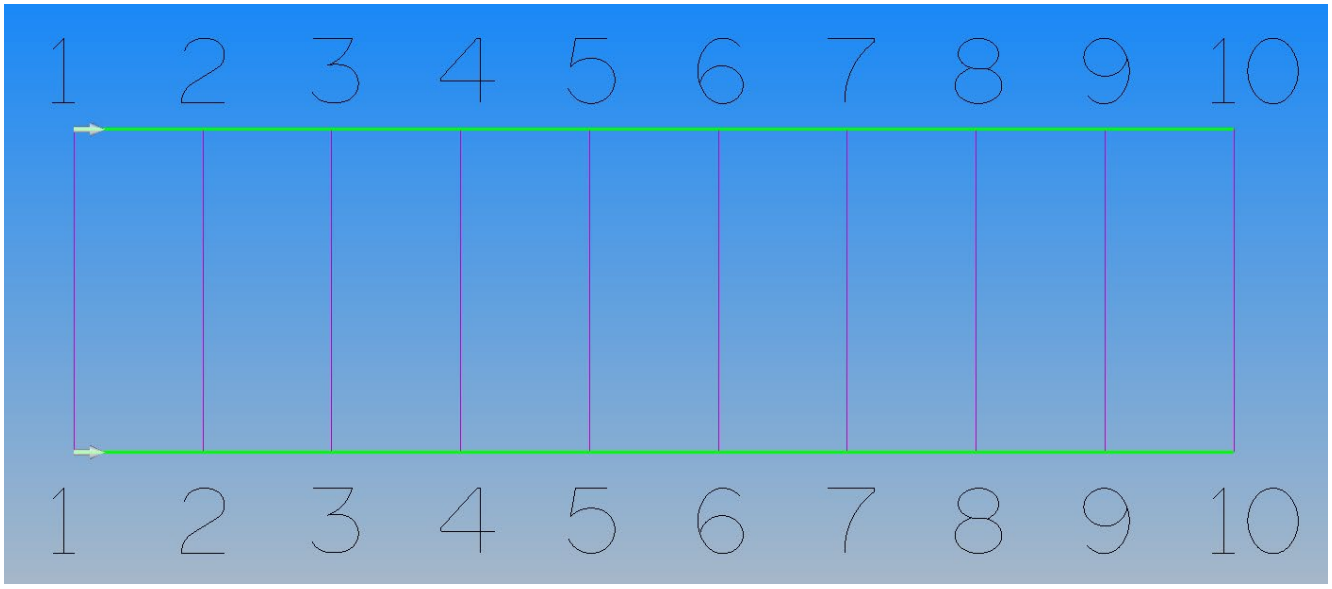


Figure 50 - Point matching on lines of equal length

Having lines of unequal length will cause the tool to lean and may give an unacceptable motion but not to the same excess as if the Ghost Tools are opposite each other.

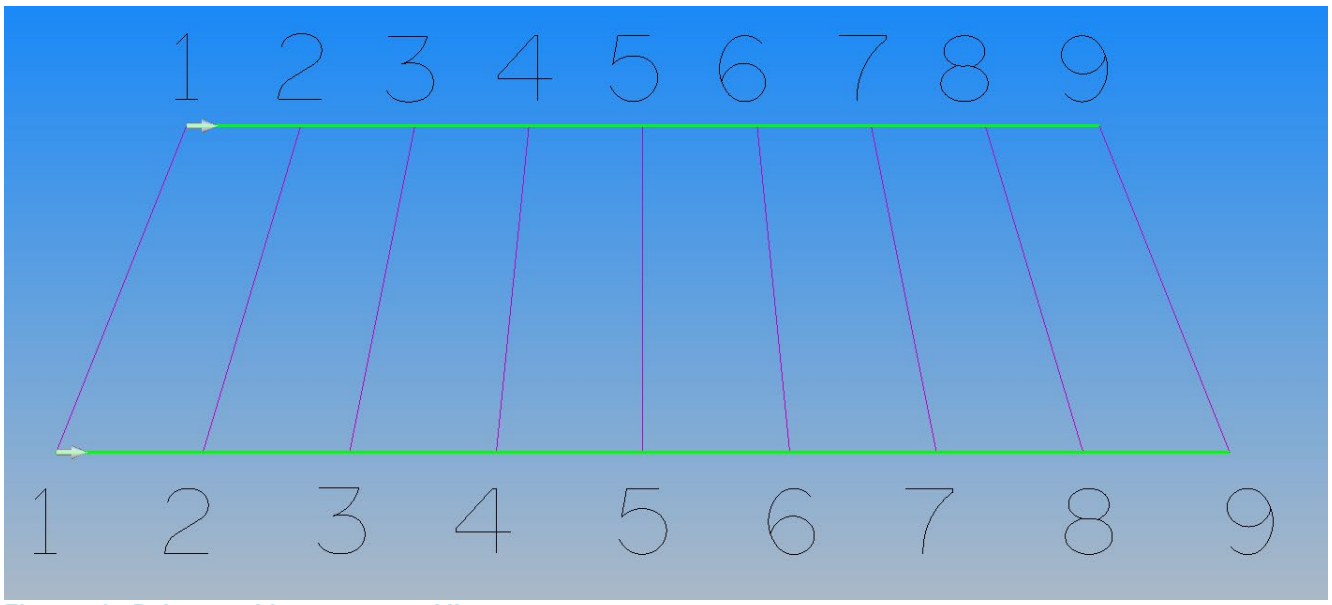


Figure 51 - Point matching on unequal lines

Altering the lengths of the lines so that they are of a similar length can prove beneficial in the long run, but care must be taken to follow the original form and not to impact on any surrounding areas.

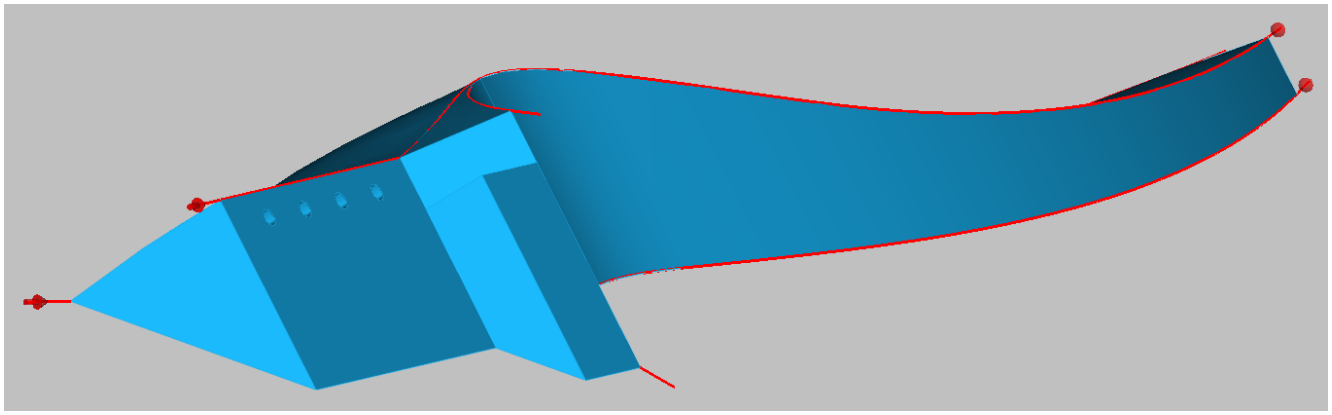


Figure 52 - Ensuring that the polylines all are working in the same direction

Turn on the **Ghost Tools** and check the extracted polylines to ensure that they are correct, adjust any that would impair your machining process.

Front Curve

> Depending on how you have extracted the edges, it may prove necessary to check the Ghost tools on Polylines to ensure that their direction matches.
> Also, you need to be aware that the lengths of the polylines need to be the same or very close so that the drive of the tool is correct and does not create uneven or incorrect tool paths.

Use **MACHINE > Select Tool** and select the **Flat - 25mm with holder.**

Select **MACHINE > Cut Between 2 Geometries**

General

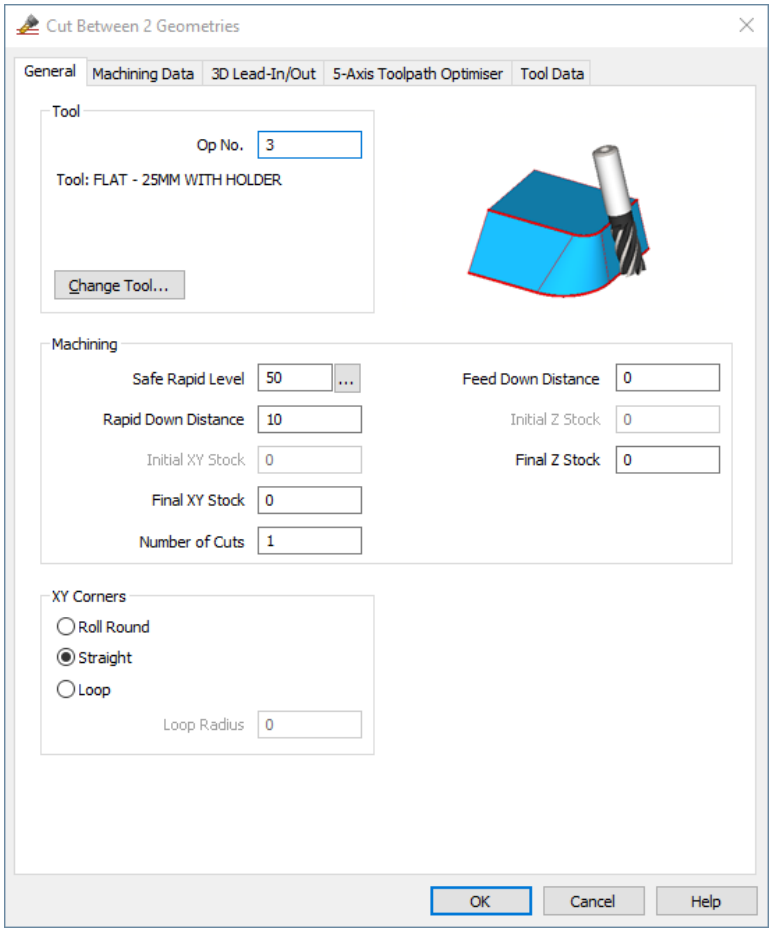


Figure 53 - Cut Between 2 Geometries General tab

Set the options as follows;
Set **Safe Rapid Level = 50, Rapid Down Distance = 20, Feed Down Distance = 10,** all other settings can be left at the default options.

> Setting the Final Z Stock to a negative figure will drive the cutter deeper into the stock if so needed, cutting past the finished bottom edge.

Machining Data

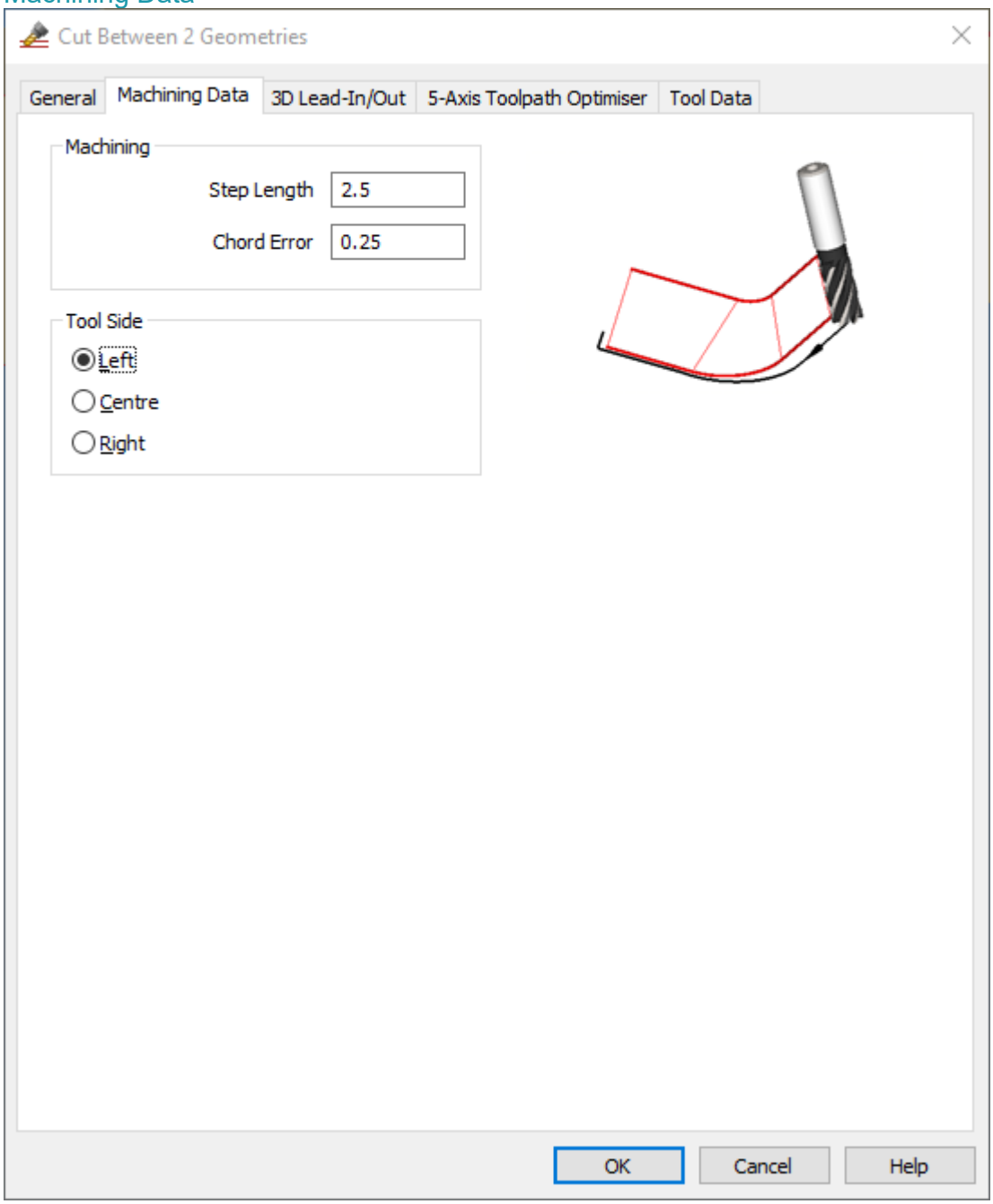


Figure 54 - Cut Between 2 Geometries Machining Data tab

Set the Tool Side to ⊙ **Left** (if your ghost tools start at the flats/holes end of the leg).

3D Lead-In/Out

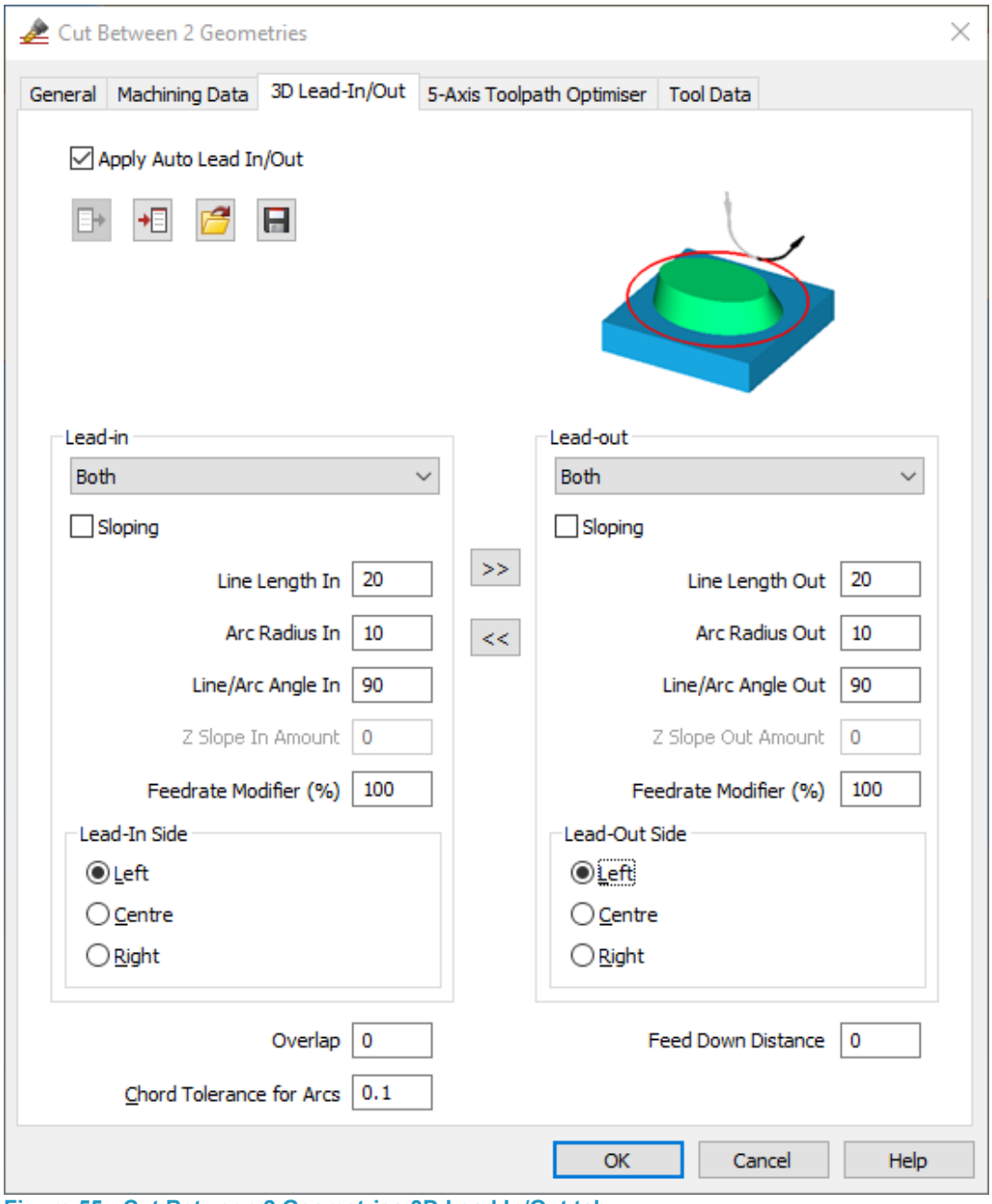


Figure 55 - Cut Between 2 Geometries 3D Lead In/Out tab

Using the **☑ Auto Lead In/out Edit** button, make the settings to **Both** and the options as **Arc = 10, Line = 20, Angle = 90°.**

> Note that on a 3D Lead-In/Out the Radius and Line values are actual distances and not multipliers of the Tool Radius

5-Axis Toolpath Optimiser

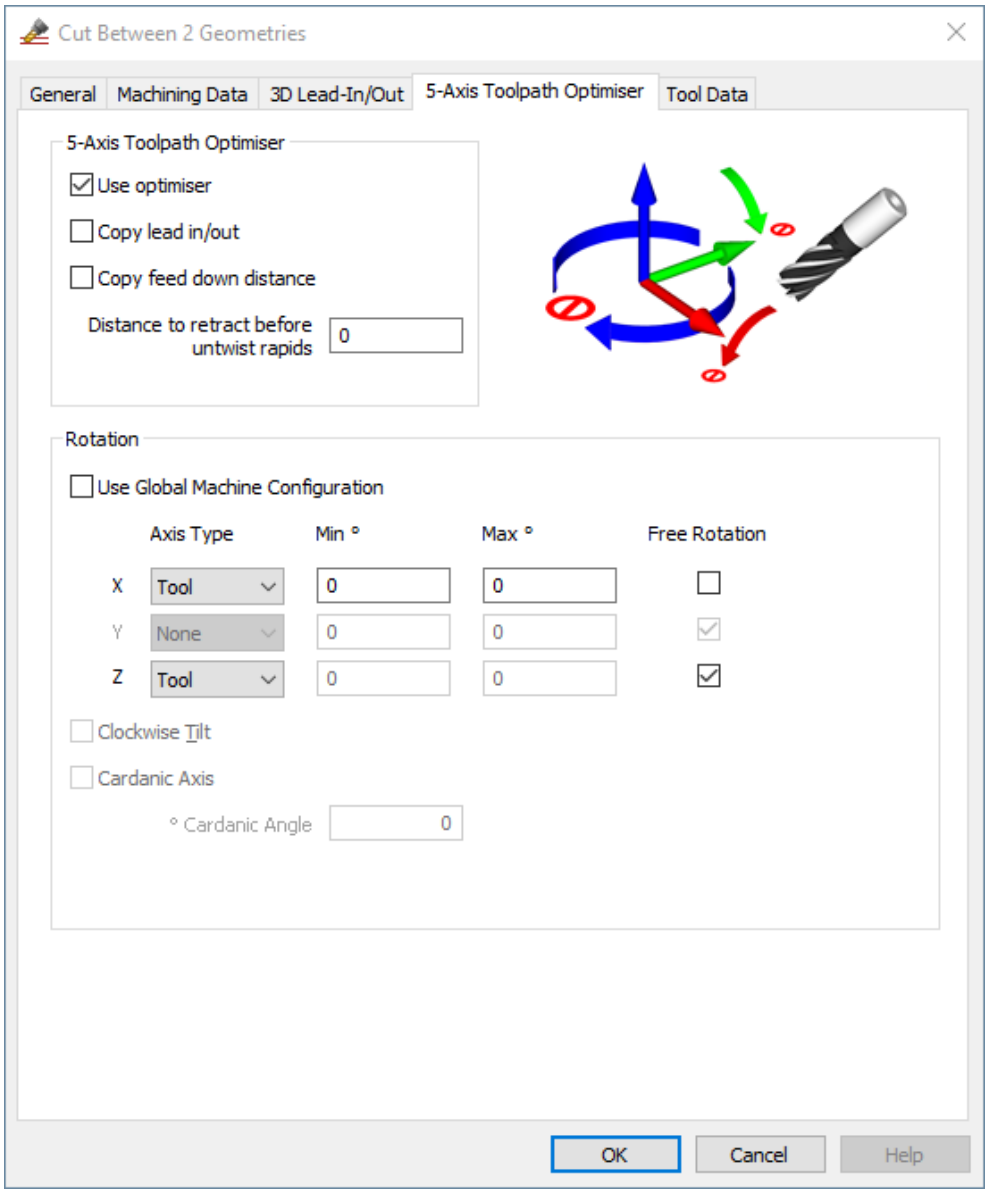


Figure 56 - 5 Axis Optimiser tab

The 5-Axis Toolpath Optimiser allows you to adjust the settings of the currently set up simulation machine to correctly allow for any breaks in the toolpath that maybe required to allow certain head type machines to unwind prior to continuing the toolpath motion.

The applied 3D Lead-In/Out can be applied to the split toolpaths using the **copy** option.

Should an axis be required to be locked to prevent any rotation from it using the Aixs Type drop down and selecting the Non option.

You may also wish to restrict the angles of rotation from the prebuilt machine using the **Min/Max** options when the **Free Rotation** option is not active.

Tool Data

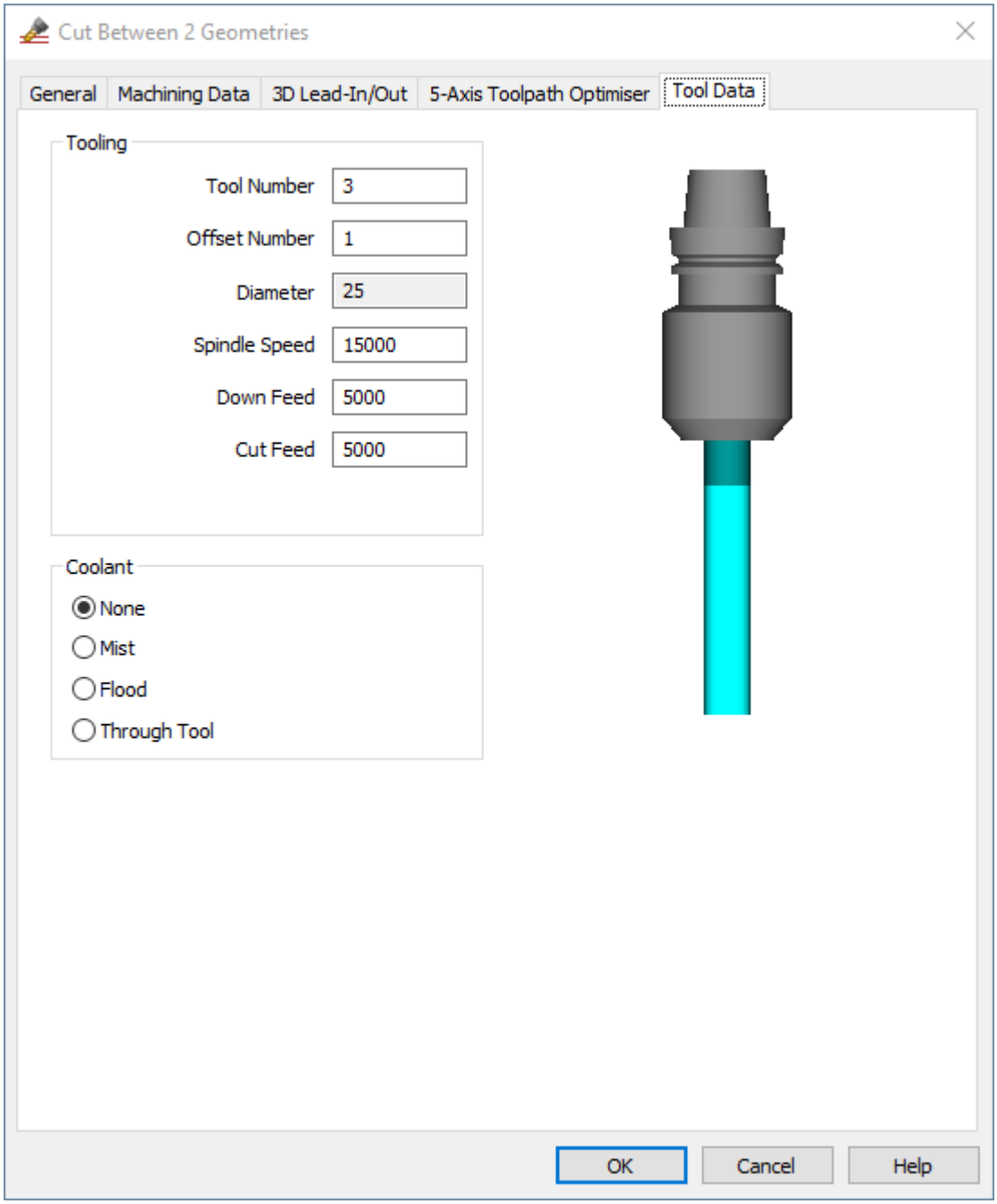


Figure 57 - Cut Between 2 Geometries Tool Data tab

Set the tooling information to suit.
<LClick> [OK].

You are prompted to select;

1. **Programming Geometry.** This guides the bottom of the tool.
2. **Auxiliary Geometry.** This guides the upper section of the tool for lean.

<LClick> the **Bottom Polyline** as the **Programming Geometry** and the Polyline on the **Spine** of the leg as **Auxiliary Geometry.**

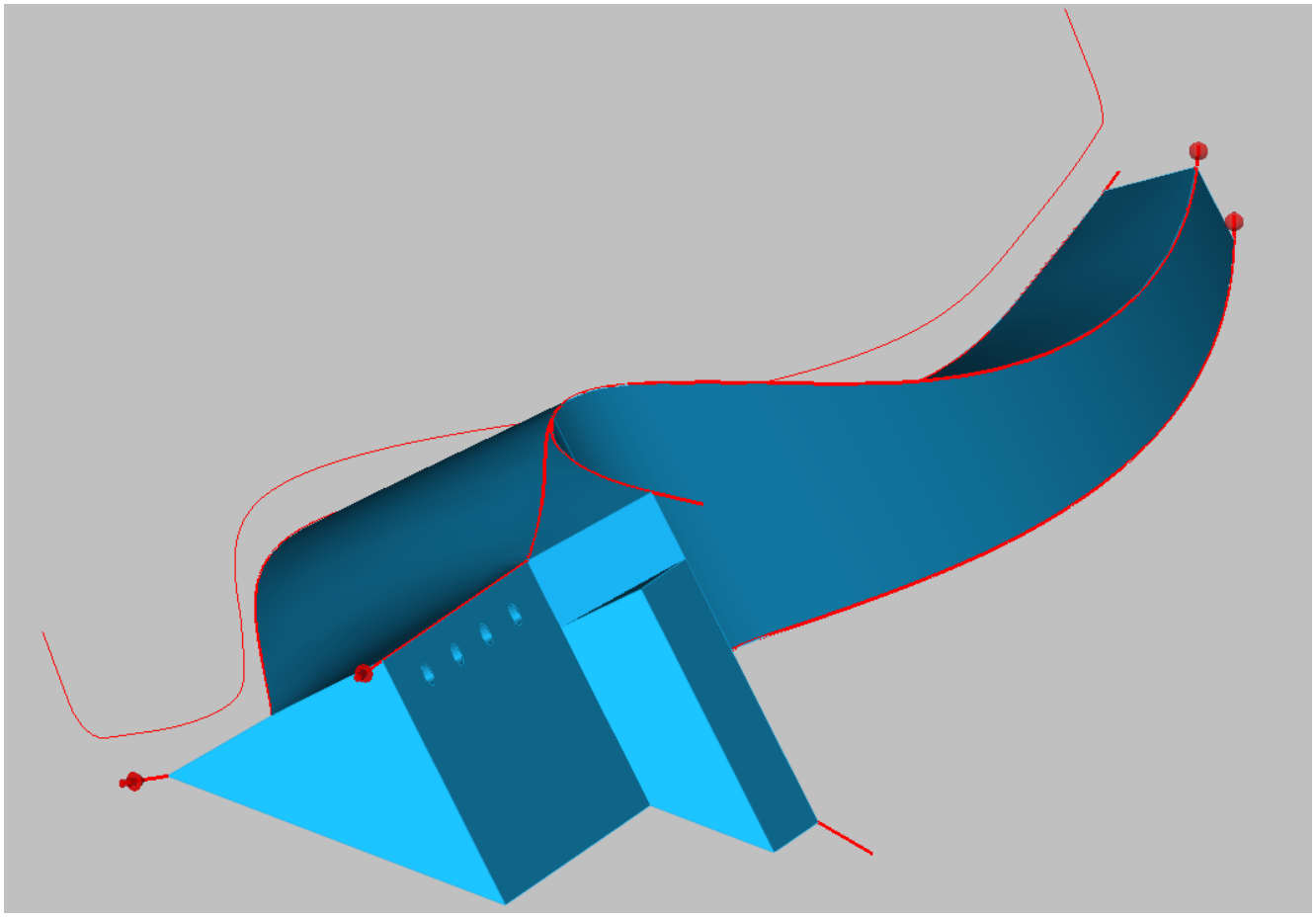


Figure 58 - Applied toolpath

Back Curve

Depending on how you have extracted the edges, it may prove necessary to check the Ghost tools on Polylines to ensure that their direction matches.
Also, you need to be aware that the lengths of the polylines need to be the same of very close so that the drive of the tool is correct and does not create uneven or incorrect tool paths.

Select **MACHINE > Cut Between 2 Geometries**

General

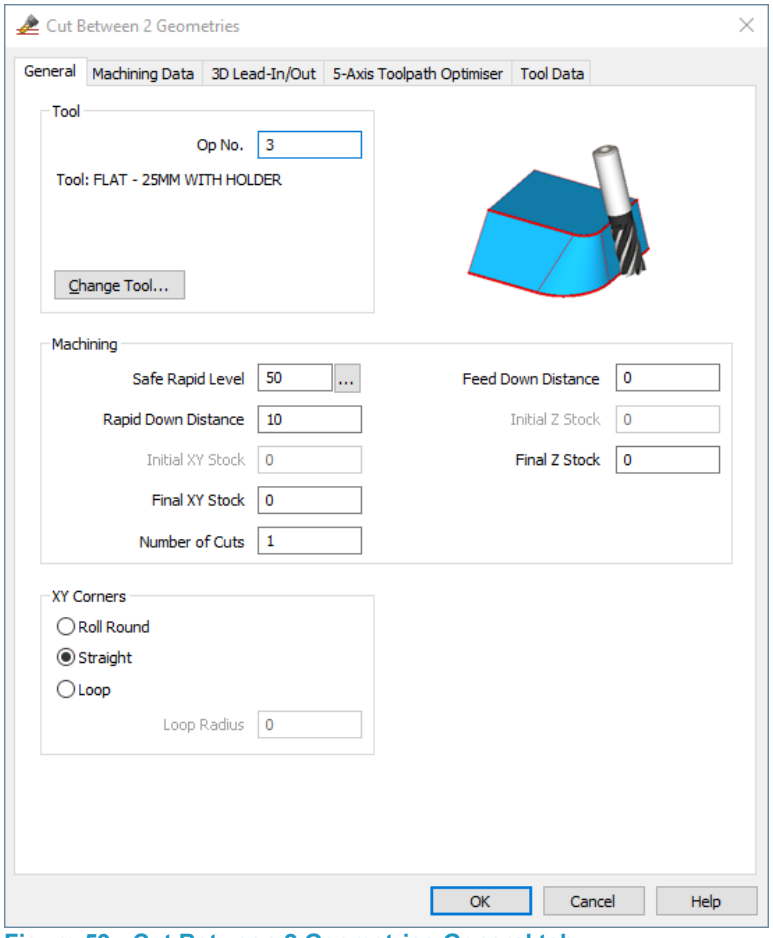


Figure 59 - Cut Between 2 Geometries General tab

Set the options as follows;
Set **Safe Rapid Level = 50, Rapid Down Distance = 20, Feed Down Distance = 10,** all other settings can be left at the default options.

Setting the Final Z Stock to a negative figure will drive the cutter deeper into the stock if so needed, cutting past the finished bottom edge.

Machining Data

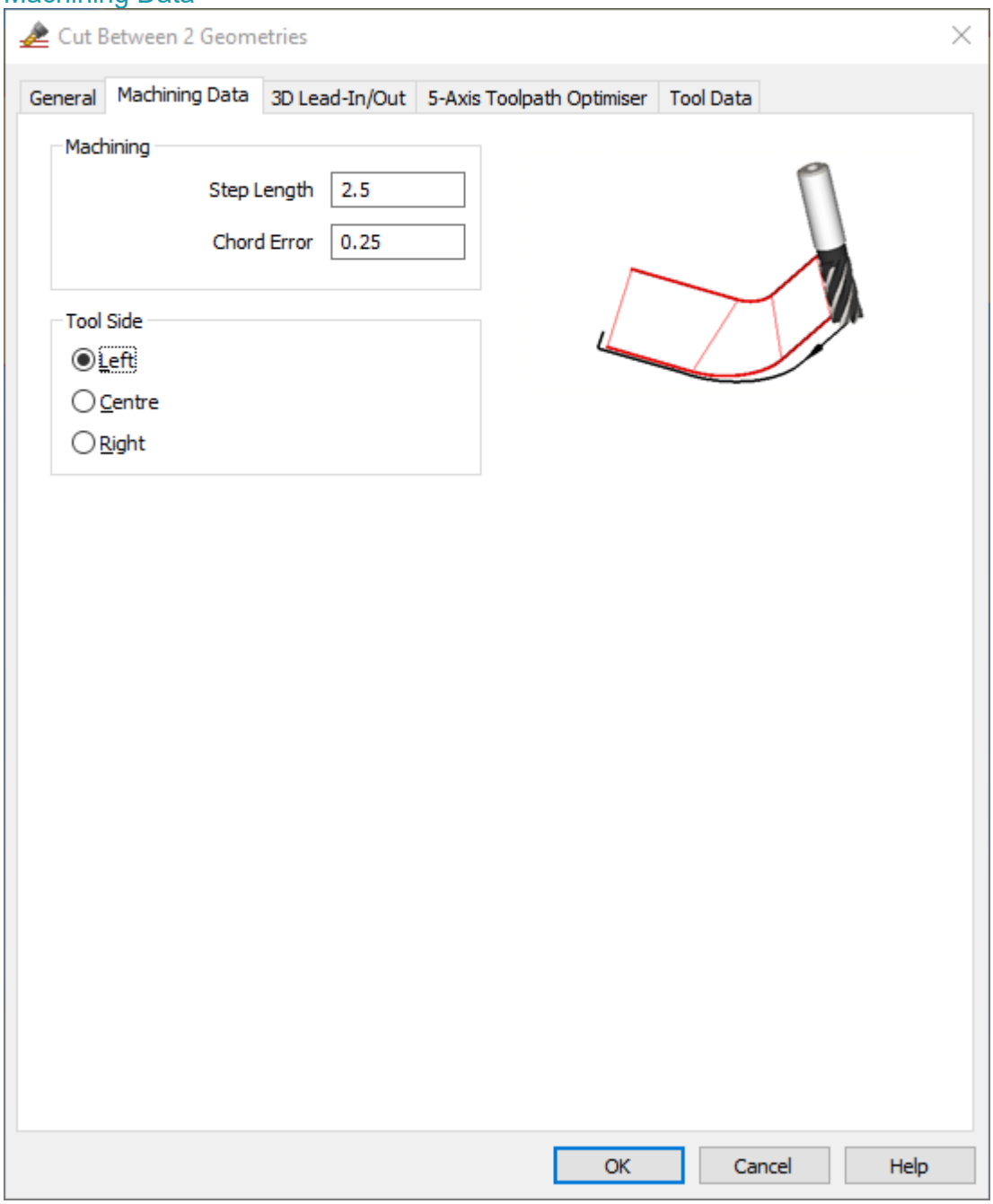


Figure 60 - Cut Between 2 Geometries Machining Data tab

On the second dialogue, set the Tool Side to ⊙ **Left** (if your ghost tools start at the flats/holes end of the leg).

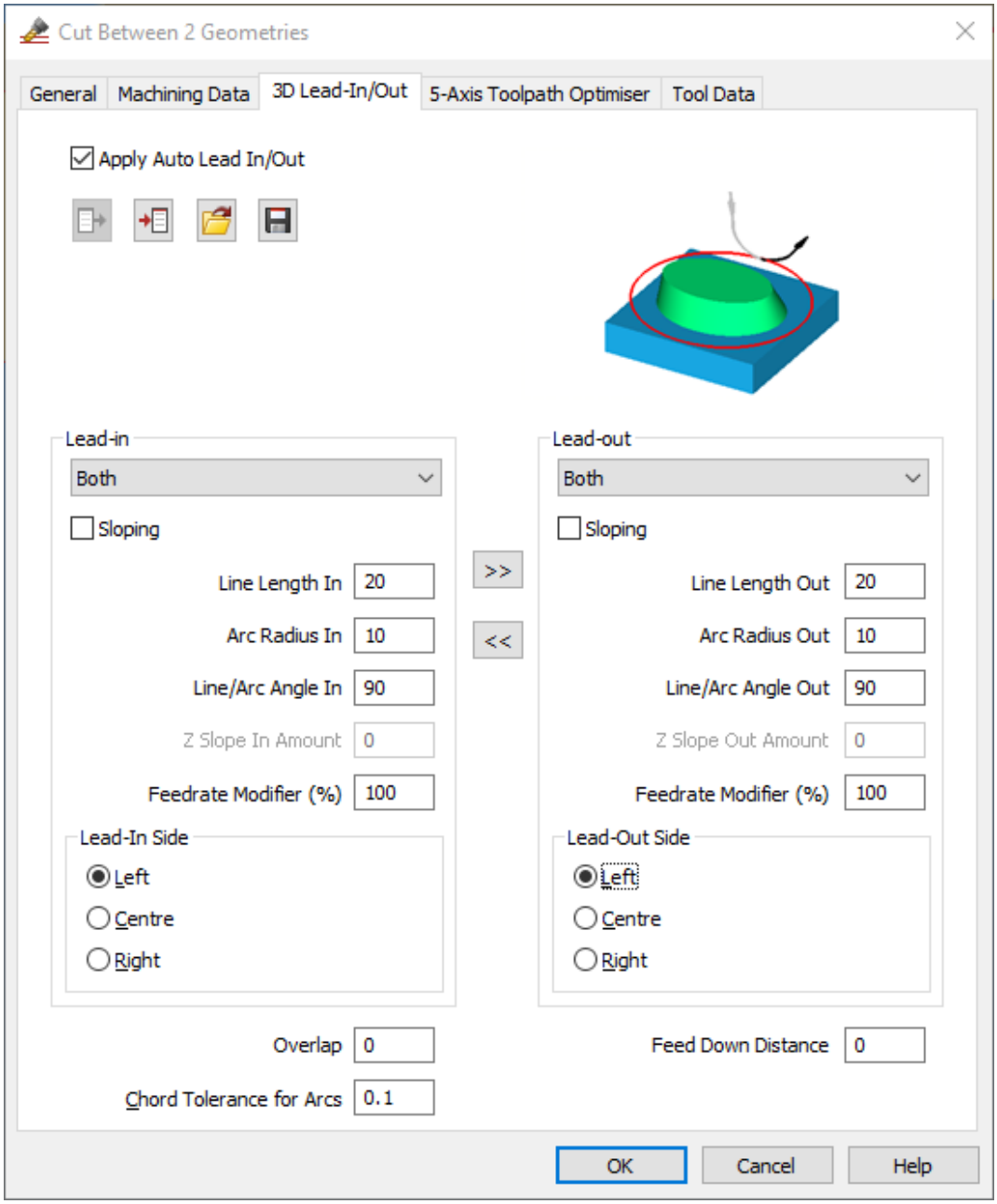


Figure 61 - Cut Between 2 Geometries 3D Lead In/Out tab

Using the **☑ Auto Lead In/out Edit** button, make the settings to **Both** and the options as **Arc = 10, Line = 20, Angle = 90°.**

> Note that on a 3D Lead-In/Out the Radius and Line values are actual distances and not multipliers of the Tool Radius.

5-Axis Toolpath Optimiser

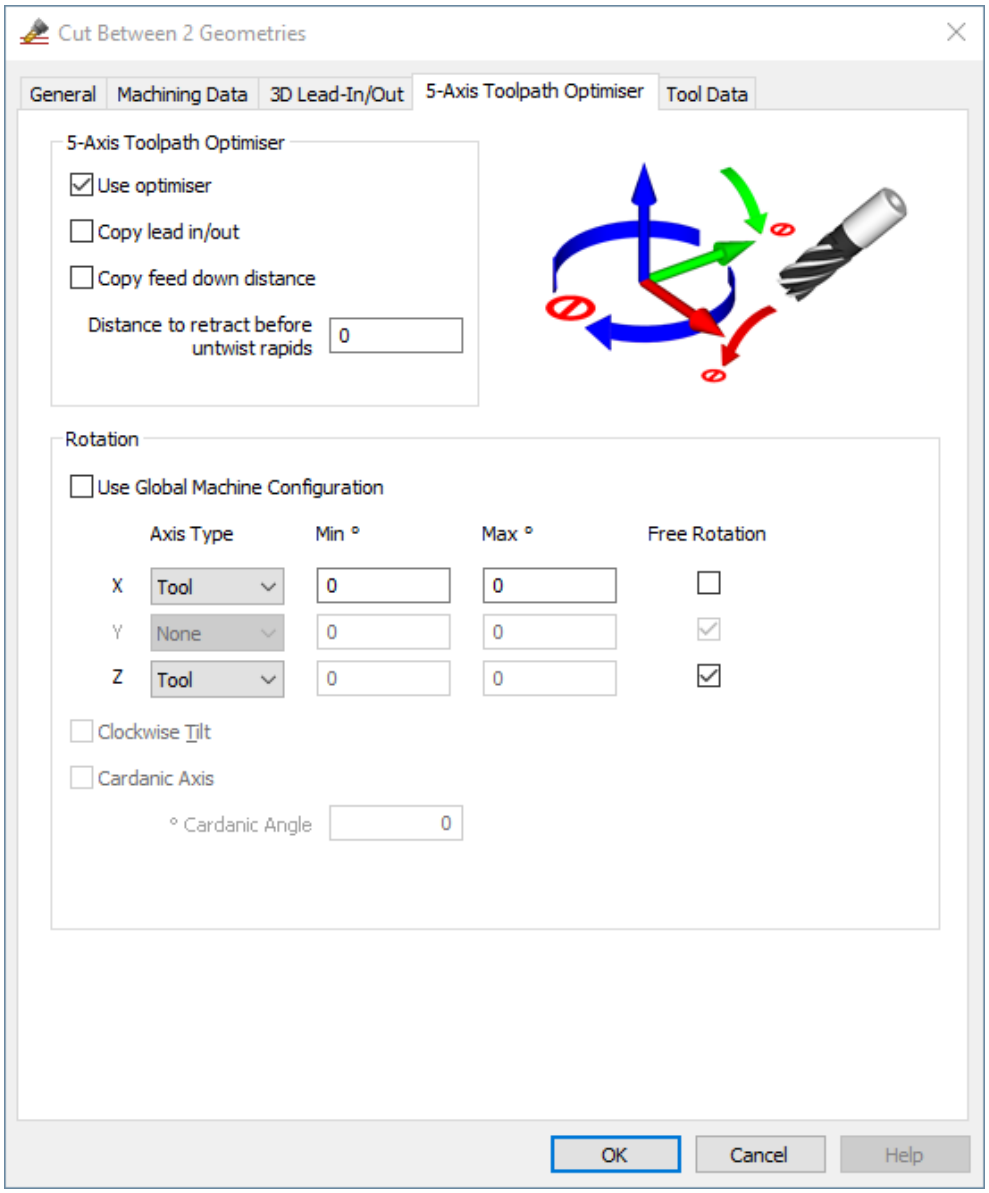


Figure 62 - 5 Axis Optimiser tab

The 5-Axis Toolpath Optimiser allows you to adjust the settings of the currently set up simulation machine to correctly allow for any breaks in the toolpath that maybe required to allow certain head type machines to unwind prior to continuing the toolpath motion.

The applied 3D Lead-In/Out can be applied to the split toolpaths using the copy options.

Should an axis be required to be locked to prevent any rotation from it using the **Axis Type** drop down and selecting the Non option.

You may also wish to restrict the angles of rotation from the prebuilt machine using the **Min/Max** options when the **Free Rotation** option is not active.

Tool Data

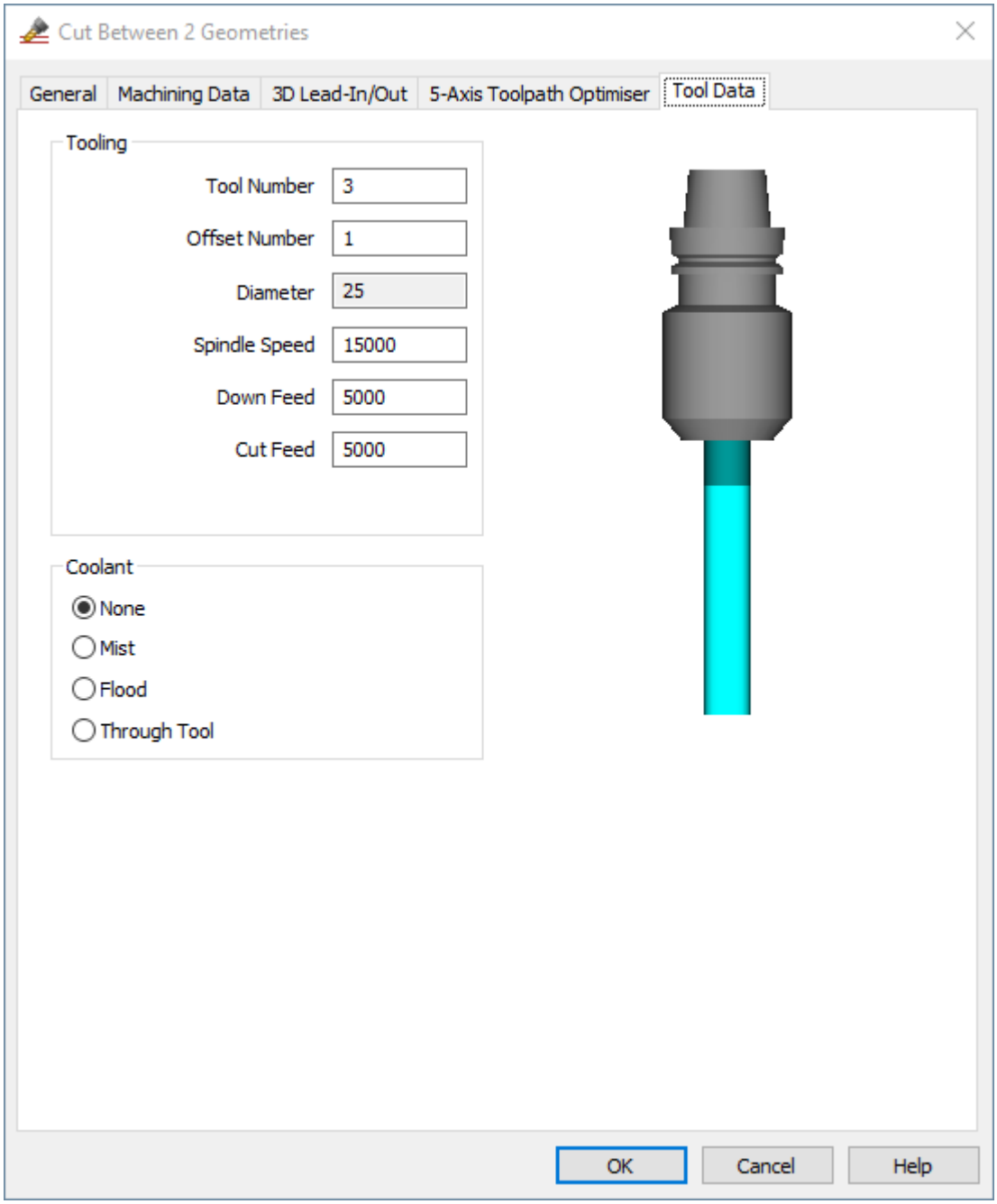


Figure 63 - Cut Between 2 Geometries Tool Data tab

Set the tooling information to suit.
<LClick> [OK].

You are prompted to select;

1. **Programming Geometry.** This guides the bottom of the tool.
2. **Auxiliary Geometry.** This guides the upper section of the tool for lean.

<LClick> the **Bottom Polyline** as the **Programming Geometry** and the Polyline on the **Spine** of the leg as **Auxiliary Geometry.**

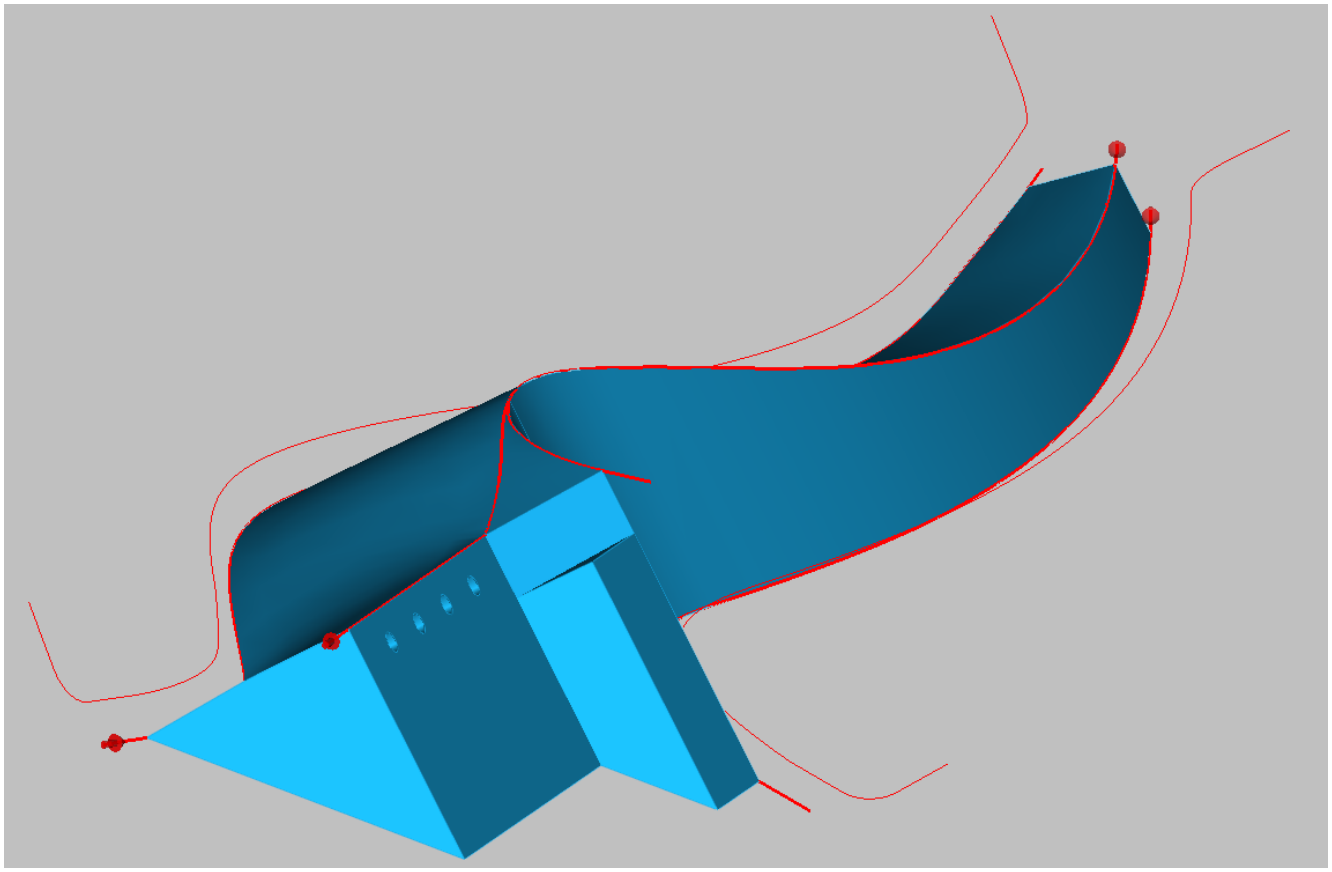


Figure 64 - Second side applied tool path

Ends

Select **SOLID MODEL EXTRACT > Contour from Picked Edge** .
Select the bottom of the Chair Leg to the right of the holes as the edge to extract.

The **Bottom** of the part for bottom edge and the **top** of the end triangle face as top edge.
Repeat on smaller end of leg.

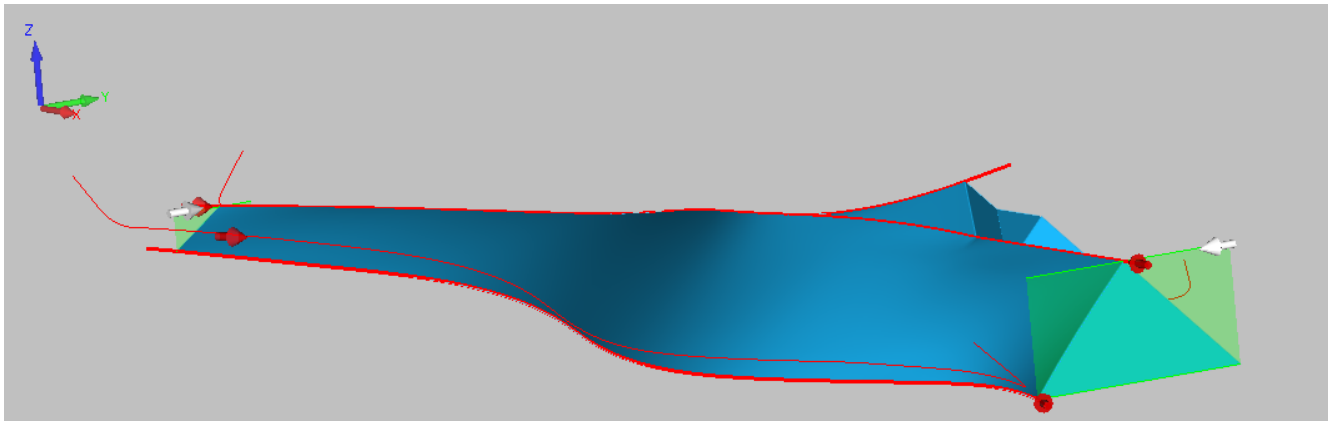


Figure 65 - End faces extracted

Use **MACHINE > Rough/Finish** ⊙ **Auto Z** and **Selected.**

Choose the two extracted edges as the geometries. Complete the dialogue boxes to suit, making the **Z Stock Amount = -1** to finish the profile below the thickness of the finished part.

Use **Apply Auto Lead In/Out** to drive onto and from the part.

Small 3D triangular area

This area needs to have a 3D finishing strategy applied to it but also requires a new work plane to control the tilt and twist of the 5 axis head. Manipulate the view of the chair leg so that you are looking directly at the small 3D triangular area above the main curved sweep.

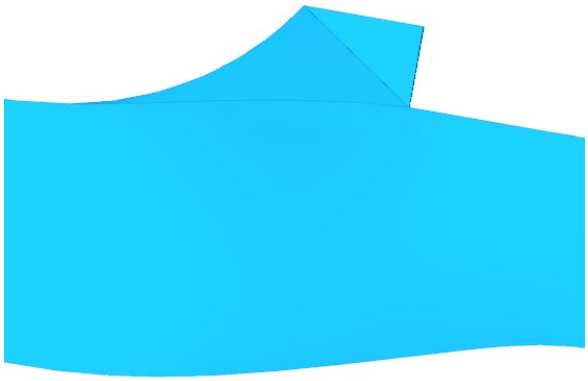


Figure 66 - Small triangle section

Select **WORK PLANES > Normal to View** 
This will create a work plane based on exactly how you are viewing the model at the moment you select the command.

Alter the **Properties**  of this new work plane to give it a suitable name.

Use **SOLID MODEL EXTRACT > Projected Face Outlines to Work Plane .**  **.**

Offset  by 1mm and set the option to **Delete the original.**

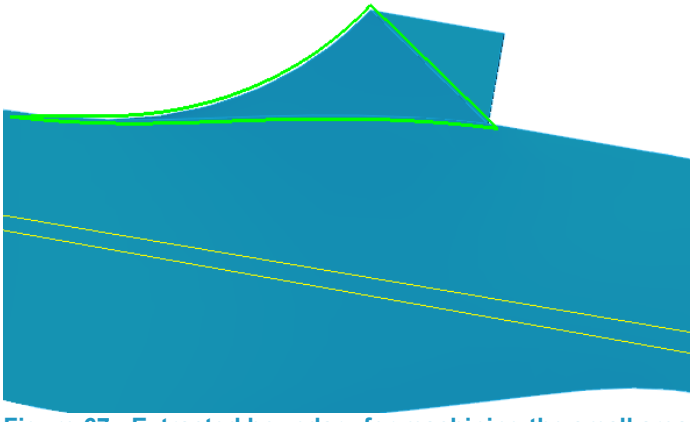


Figure 67 - Extracted boundary for machining the small area

Create the initial driving toolpath.

Use **MACHINE > Select Tool**  and select the **Ball End - 6mm + holder.**

Select the tool directions option and reset the **Ghost Tool** for the extracted geometry to **Centre.**

The type of 3D Finishing strategy we will us on this area, **Drive Curves**, requires us to first create the items which will be used as the driving elements. This will be achieved using a standard 2D pocketing cycle; the only option on the Pocketing Cycle that will bear any relevance to the Drive Curves is the amount of step over needed which is influenced by the **Width of Cut** option in the Pocketing dialogue box. This is usually a percentage figure of the chosen cutter but for this process we need to alter it to a value of 0.4mm to finish machine the area in question.

After clicking **[OK]** to complete the pocketing dialogue boxes, click **[OK]** on the Soft Boundary warning. The tool will cut to the actual geometry.

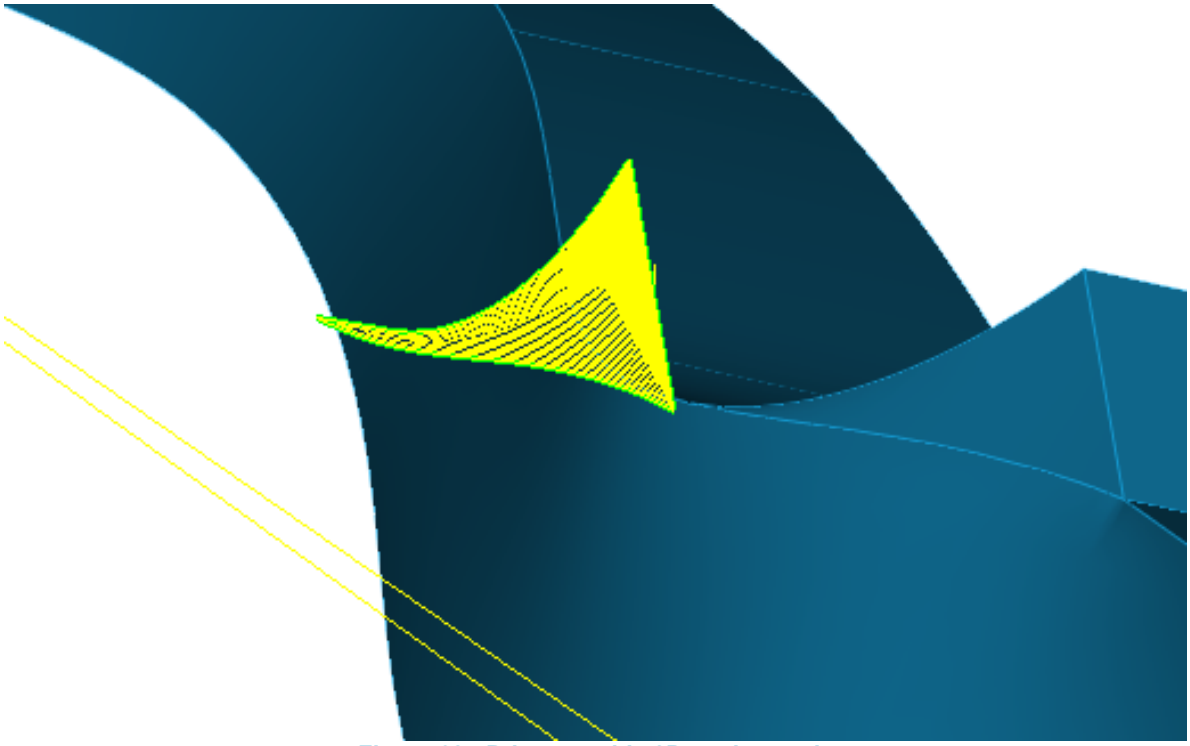


Figure 68 - Primary guide 2D pocket cycle

Applying the Final Machining

Use **MACHINE > 3D Machining** 

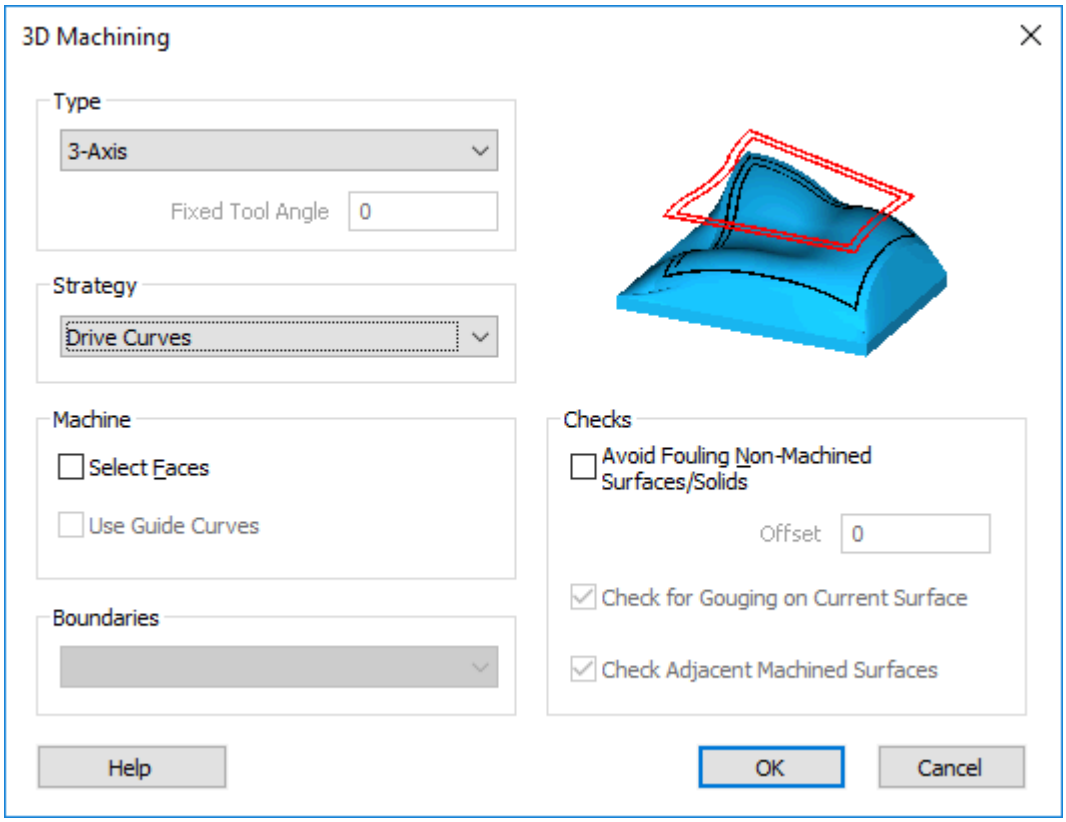


Figure 69 - 3D Machining Cycle Selection Dialogue

In the strategy selection dialogue, pick **Drive Curves** then **<LClick> [OK]** to continue.

General

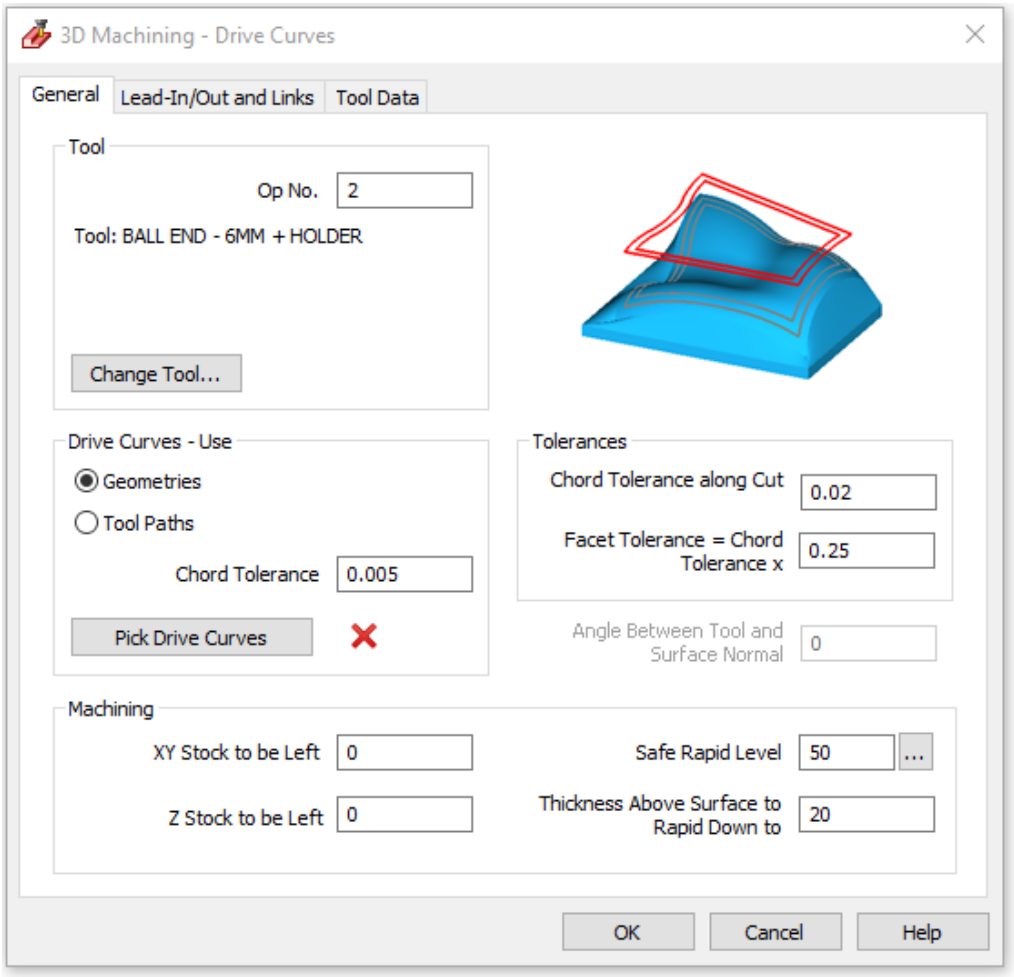


Figure 70 - Drive Curves General tab

Select ⊙ **Toolpaths.**
<LClick> [Pick Drive Curves].
The cycle dialogue will disappear and you will need to **<LClick>** on the previously created 2D pocket cycle. Once you have chosen the driving toolpath, the dialogue will reappear with a green tick to indicate that you can continue.

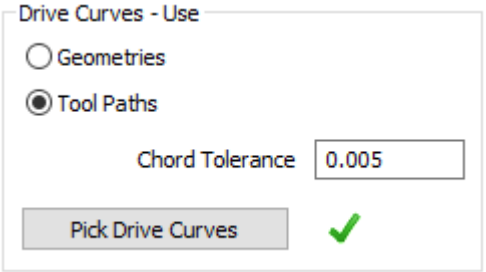


Figure 71 - Drive Curves using Tool Paths

<LClick> [OK] to proceed and when prompted, choose the previously created 2D pocket as the Toolpaths for the Drive Curve.

The only options that need setting are the **Safe Rapid Level=50** and the **Thickness Above Surface to Rapid down To=10.**

<LClick> [OK] to continue and to complete the dialogue action.

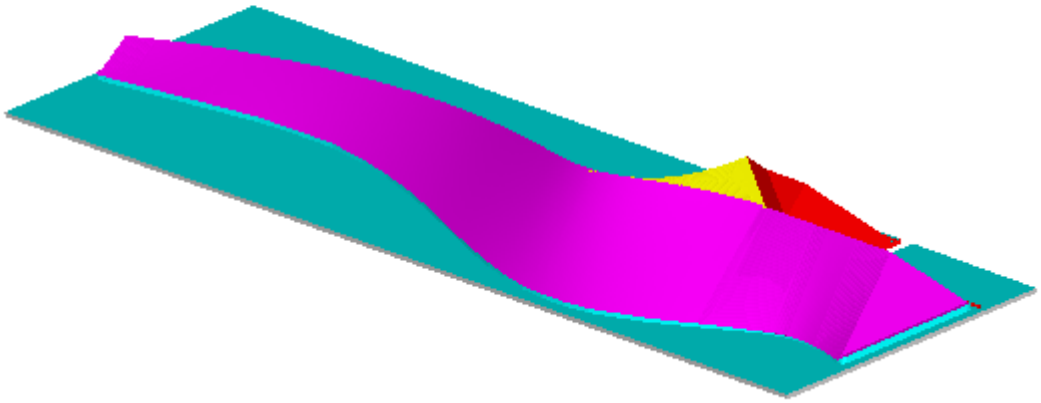


Figure 72 - Completed part seen in simulation

This is the one remaining cycle that after creation Cannot be edited; any errors will require the cycle to be completely re-created.

Copying the 2D cycle and projecting the copy will leave an easy method of editing the original rather than re-creating.

Save your job with a suitable name.

Practical example Twisted Handrail

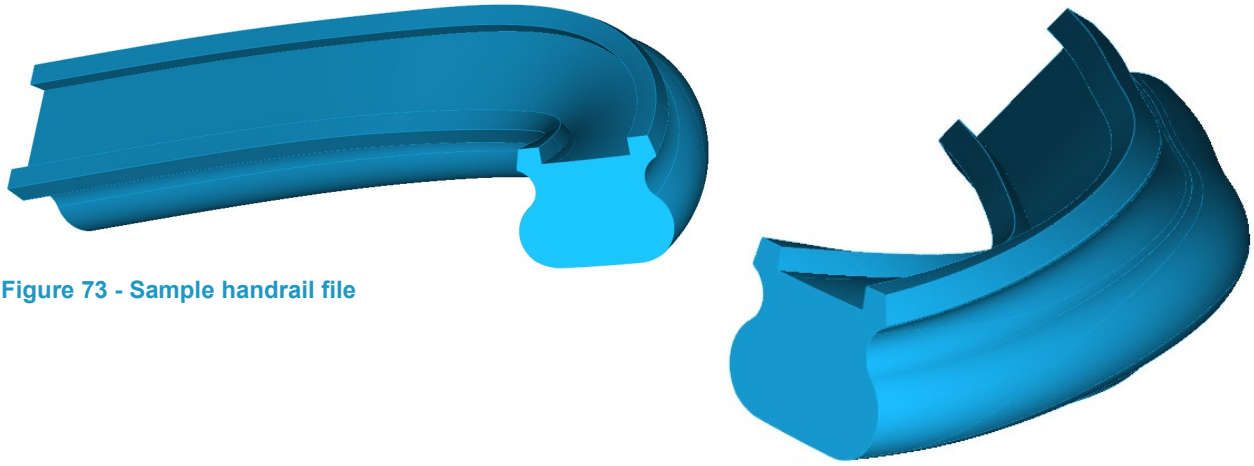


Figure 73 - Sample handrail file

In this tutor lead example, we will look at the practical application of Positional 5 Axis tool paths, Simultaneous 5 Axis tool paths and Tool Axis Conversions so that you can understand the differing requirements.

Select **HOME > Open**  **and n**avigate to **"….\ALP TRG 210 Standard 5 Axis 2020\Examples\Drawings\" folder** and open **"Twisted Handrail".**

Create a material around the part using **3D > Auto Set Material.** 
Make the values in the dialogue as follows**, Material Top = 10, Material Bottom = -150, Material XY Stock = 10,** ensure that **☑ Associate for auto-update is tick**ed.

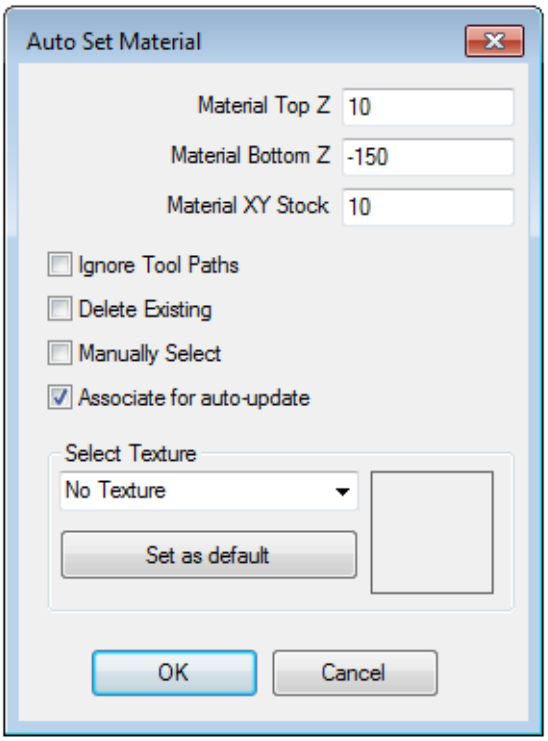


Figure 74 - Using the Associate for auto-update option

Create a Rectangle from one corner of the new material to the diagonal opposite corner.

Using **MACHINE > Clamps/Fixtures > Define Clamps/Fixtures** 
Select the rectangle as the shape and set the options as follows,

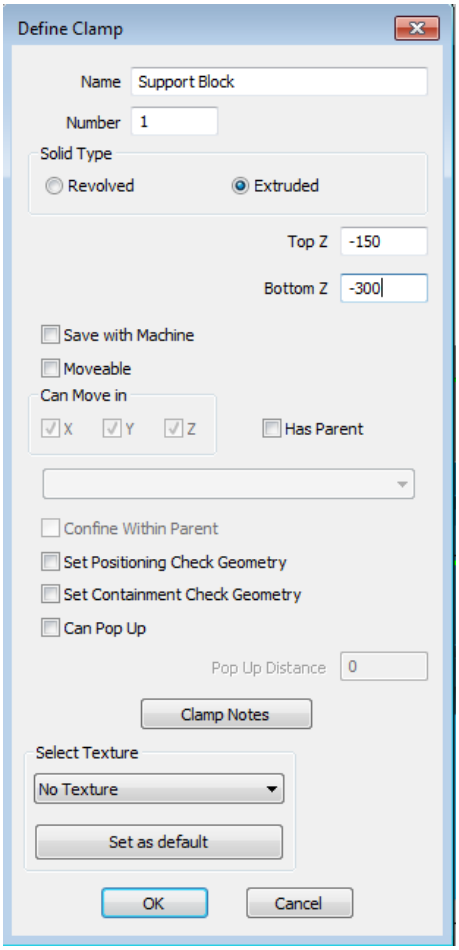


Figure 75 - Setting up the support block information

This creates a correct material for the part to be machined from and a support block to lift the part up from the machine bed or pods so that the 5 Axis system can rotate without any collisions. This is an important consideration when working with 5 Axis toolpaths.

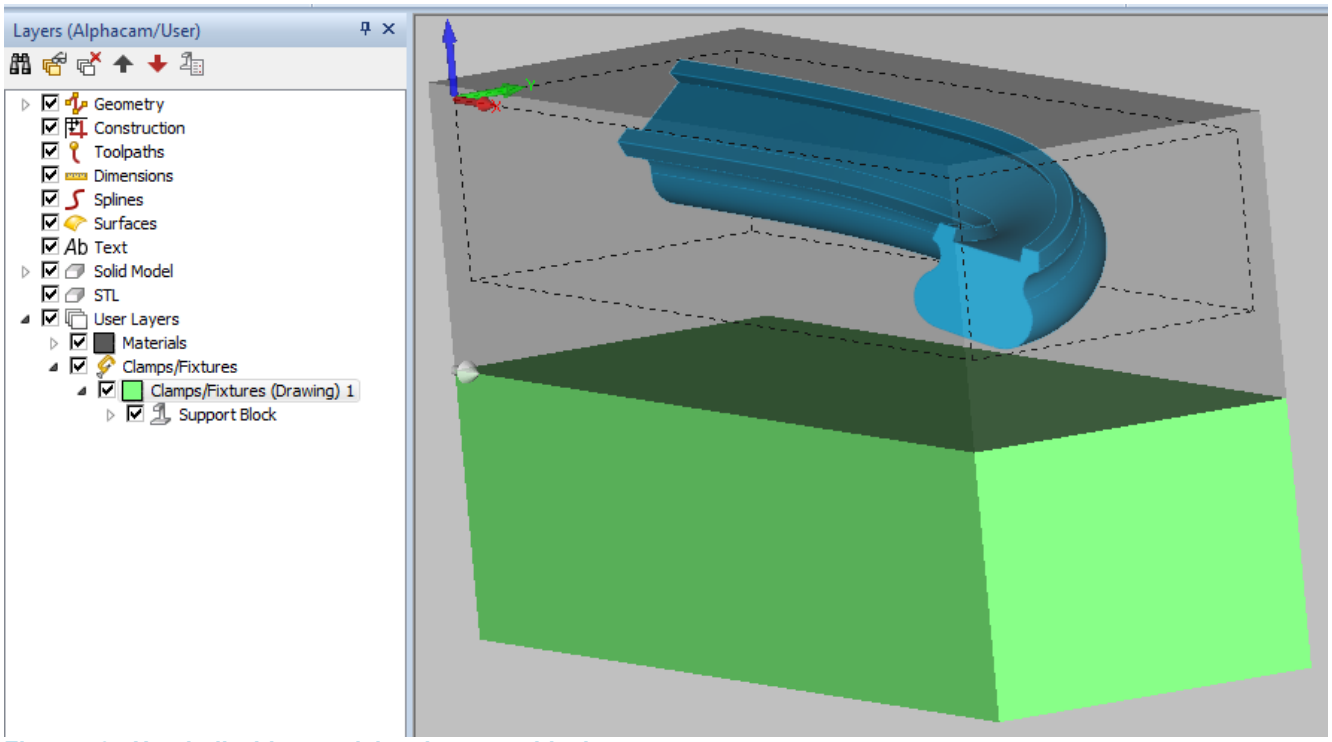


Figure 76 - Handrail with material and support block

All clamps are created on their own **User Layer** and are coloured a dark red.

This is very close to ALPHACAM's default collision colour so it may prove advisable to change the layer colour to make any collisions more evident.

Roughing

Primary Stage

From the Training Folder select the **Flat 25mm with Holder.**

Using **MACHINE > 3D Machining**  ,

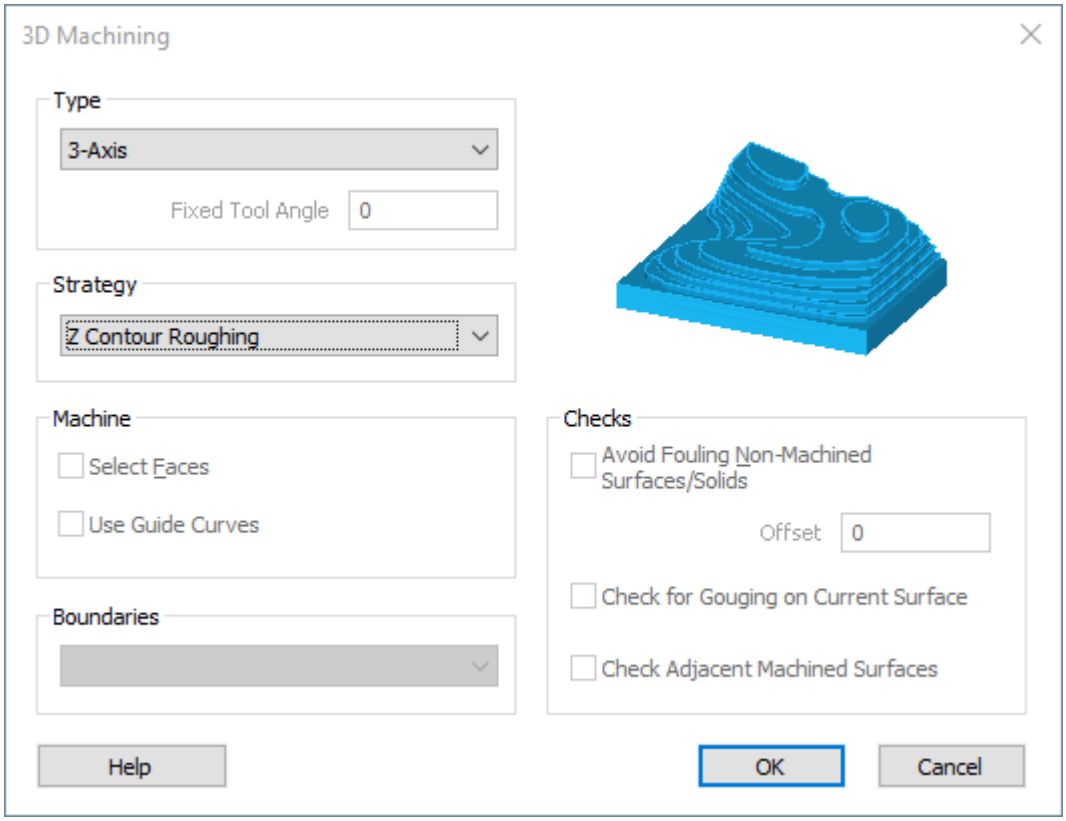


Figure 77 - 3D Machining Cycle selection dialogue

Set the options to **Z Contour Roughing.**

General

Make the options as shown below.

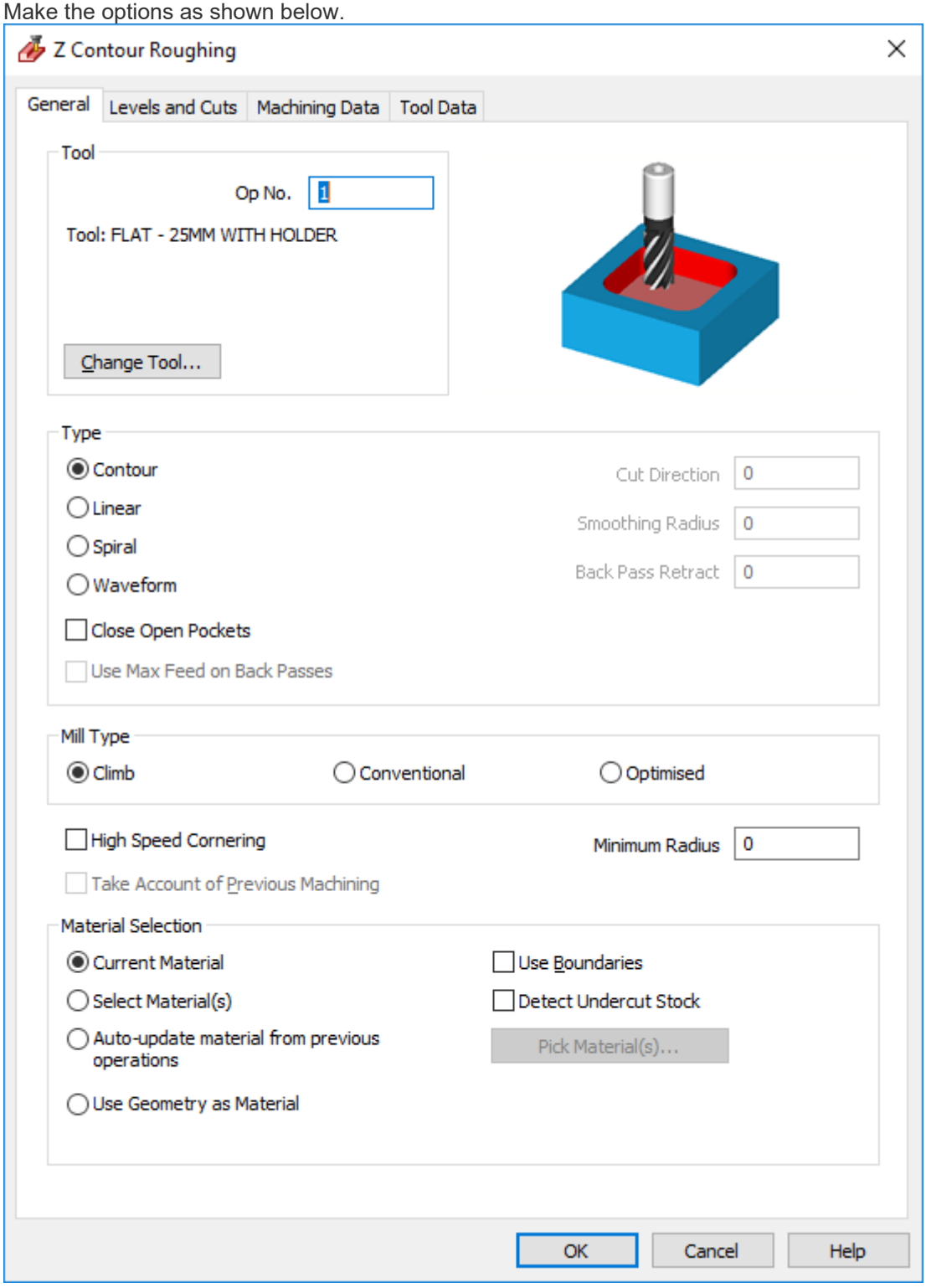


Figure 78 - Z Contour Roughing General tab

Levels and Cuts

Make the options as shown below.

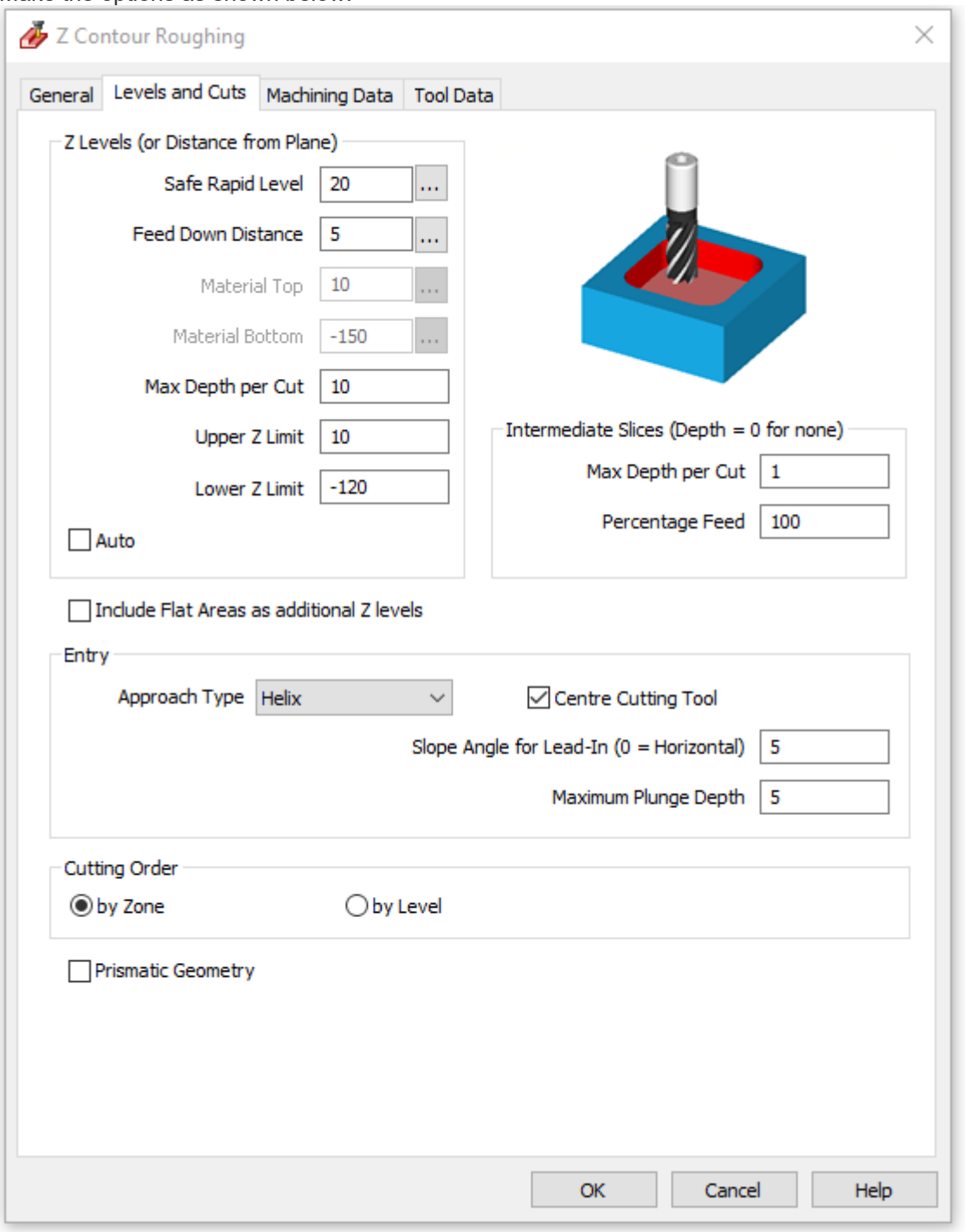


Figure 79 - Z Contour Roughing Levels and Cuts tab

Machining Data

Make the options as shown below.

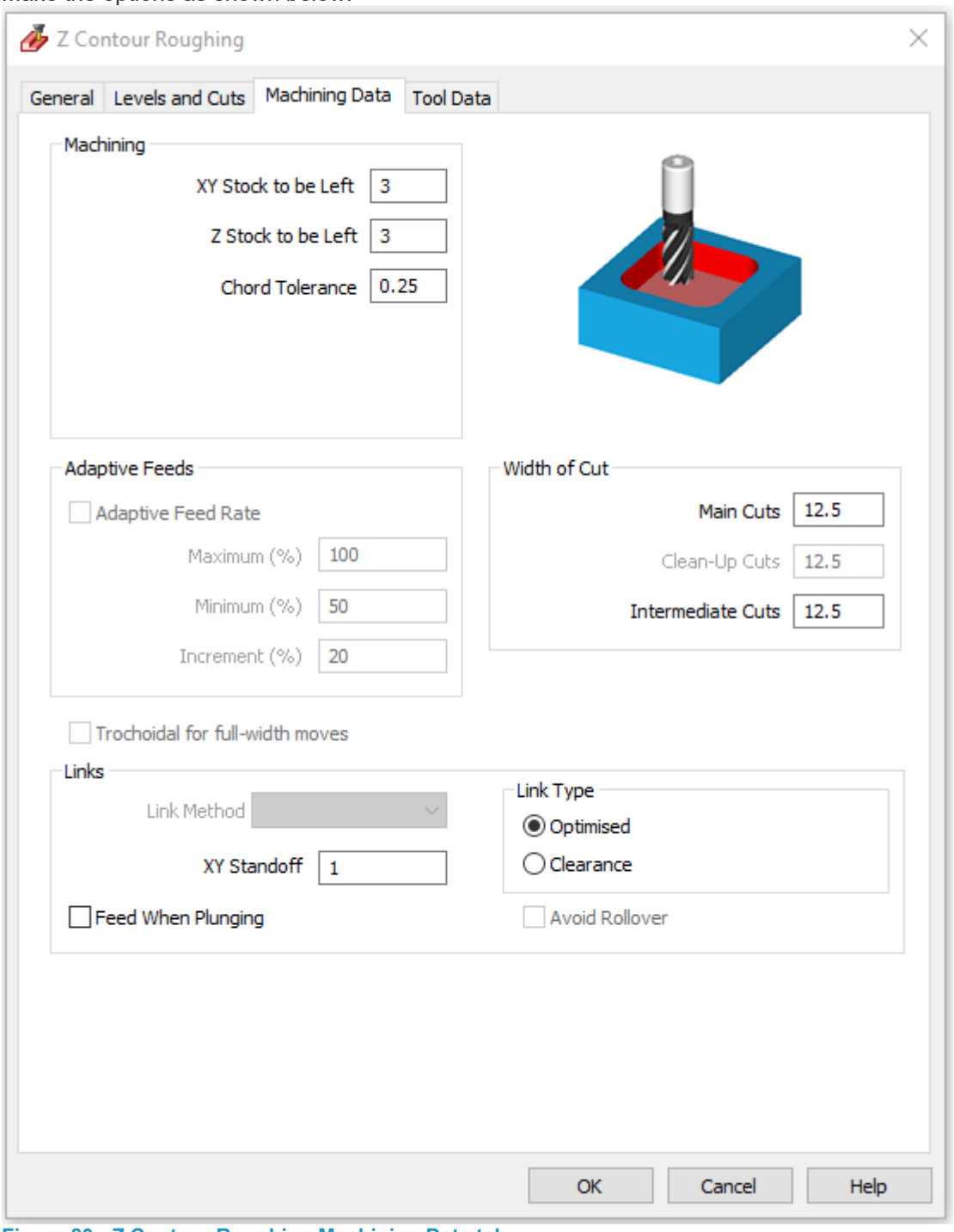


Figure 80 - Z Contour Roughing Machining Data tab

Tool Data

Set the tooling options as required.

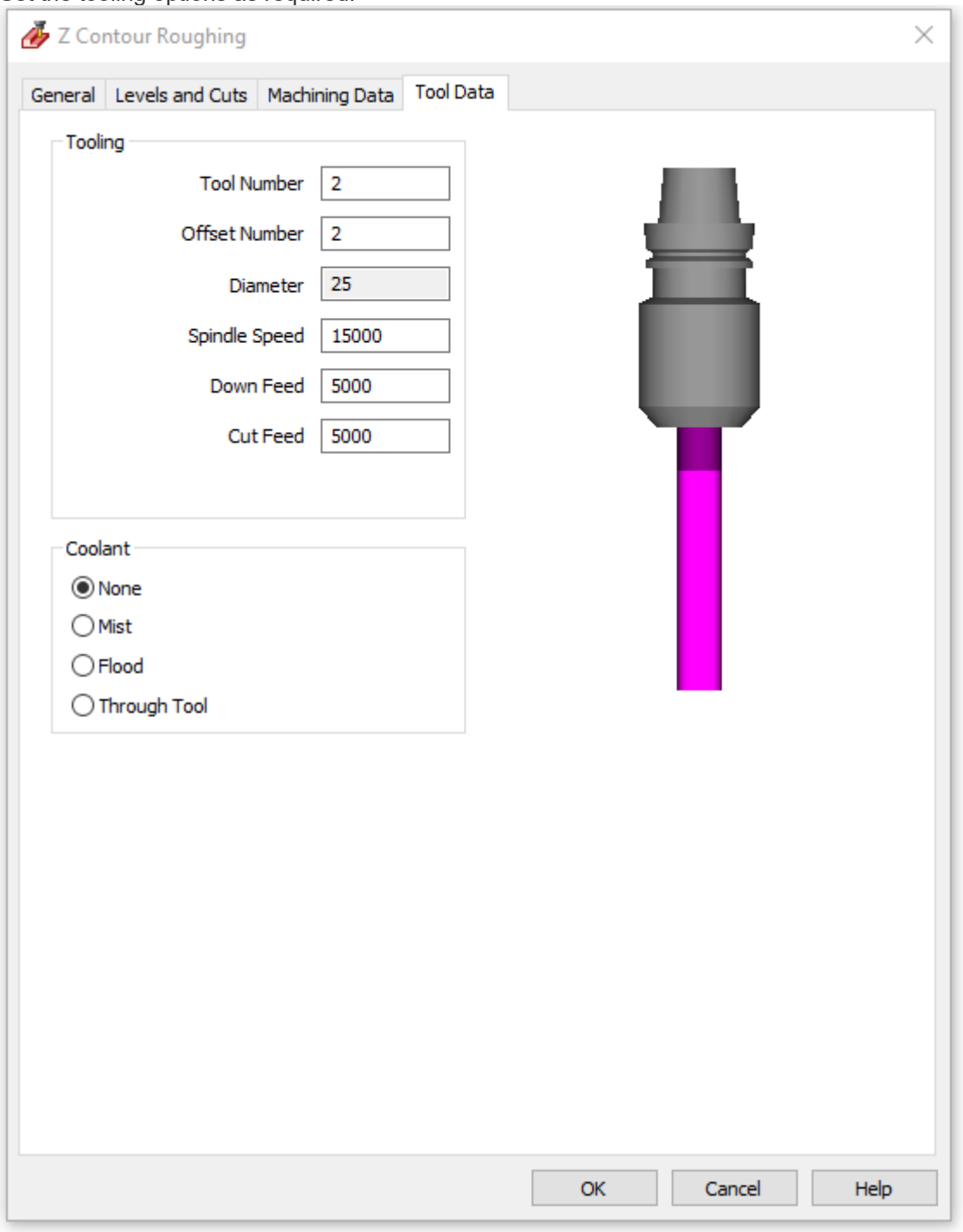


Figure 81 - Z Contour Roughing Tool Data tab

<LClick> [OK], when prompted select the handrail solid as the item to machine and allow ALPHACAM to apply the toolpath as per your settings.

Secondary Stage.

Front Face

For the secondary roughing stages, we need to create three work planes to control the position of the 5 axis system. These new work planes will be on the front face of the work volume, on the rear face of the work volume and a work plane based on an angled slice through the work volume.

Create a work plane on the front face using **WORK PLANES > Slice Through Work Volume**  .
If the work plane datum does not line up with the global datum,

use **WORK PLANES > Set Work Plane Origin** 
Make two rectangular boundaries to suit the areas to be machined on this work plane. The positions of the boundaries are as follows;
X-5, Y-135 to **X280, Y-22.**
and
X270, Y-135 to X430, Y-22.

A third boundary needs to be created using a rectangle with these values;
X412, Y-135 to **X550, Y-22.**
Then extracting the end face of the hand rail to the work plane using

SOLID MODEL EXTRACT > Projected Face Outlines to Work Plane  , edit the resulting two geometries so that the final boundary looks like this.

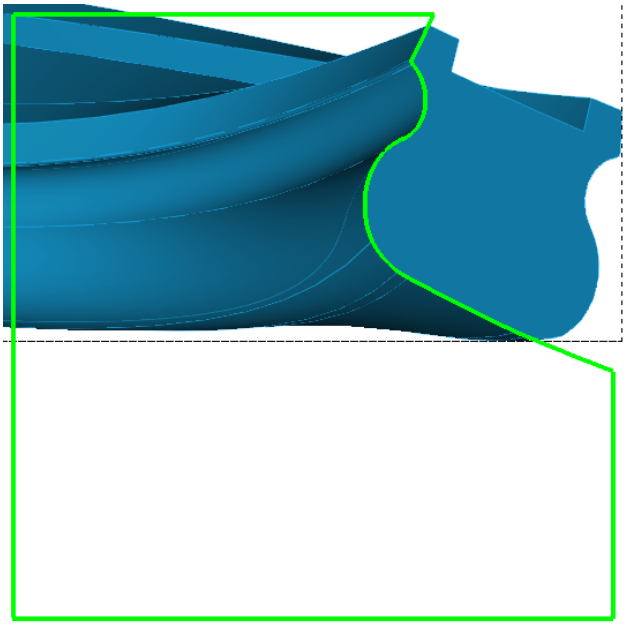


Figure 82 - Third boundary shape on front face work plane

As the end detail used a Feature Extracted profile, the Tool directions will have been set using the current ALPHACAM defaults, for this process we require the **Tool Side** to be on **Centre**, so the boundary will need adjusting to suit using the Tool Directions option.

Using **MACHINE > 3D Machining**  ,
Set the strategy to **Z Contour Roughing.**

Left Boundary area

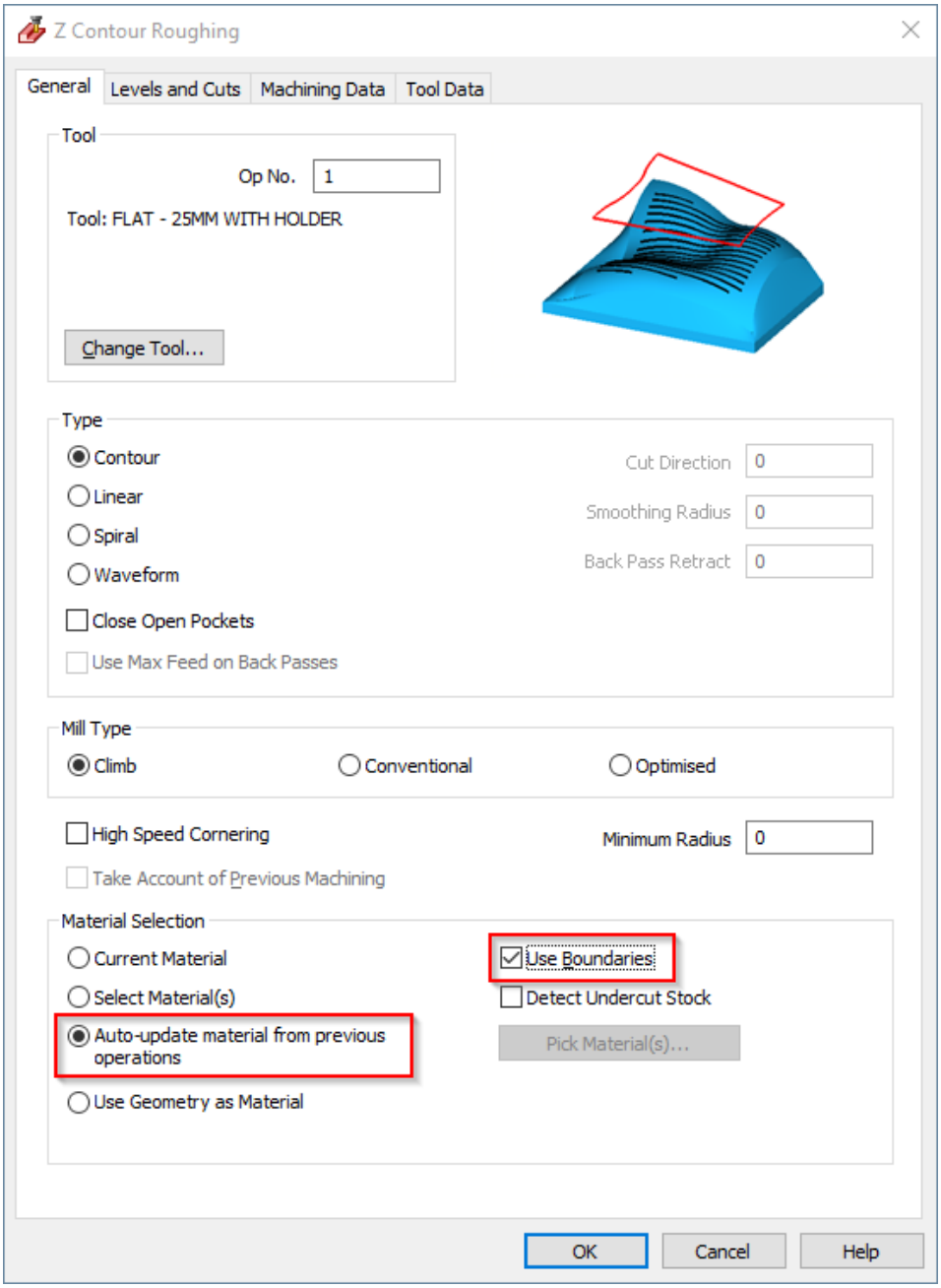


Figure 83 - Use the Auto Update option

Note the changes to ⊙ Auto-update material from previous operations and ☑ Use Boundaries.

On the Levels and Cuts tab, ensure that the ☐ **Auto option** is **NOT** ticked to allow you to enter alternate depth options. Note that the upper and lower Z limits cannot be exceeded from their original values.
For the first boundary section make the options as shown below.

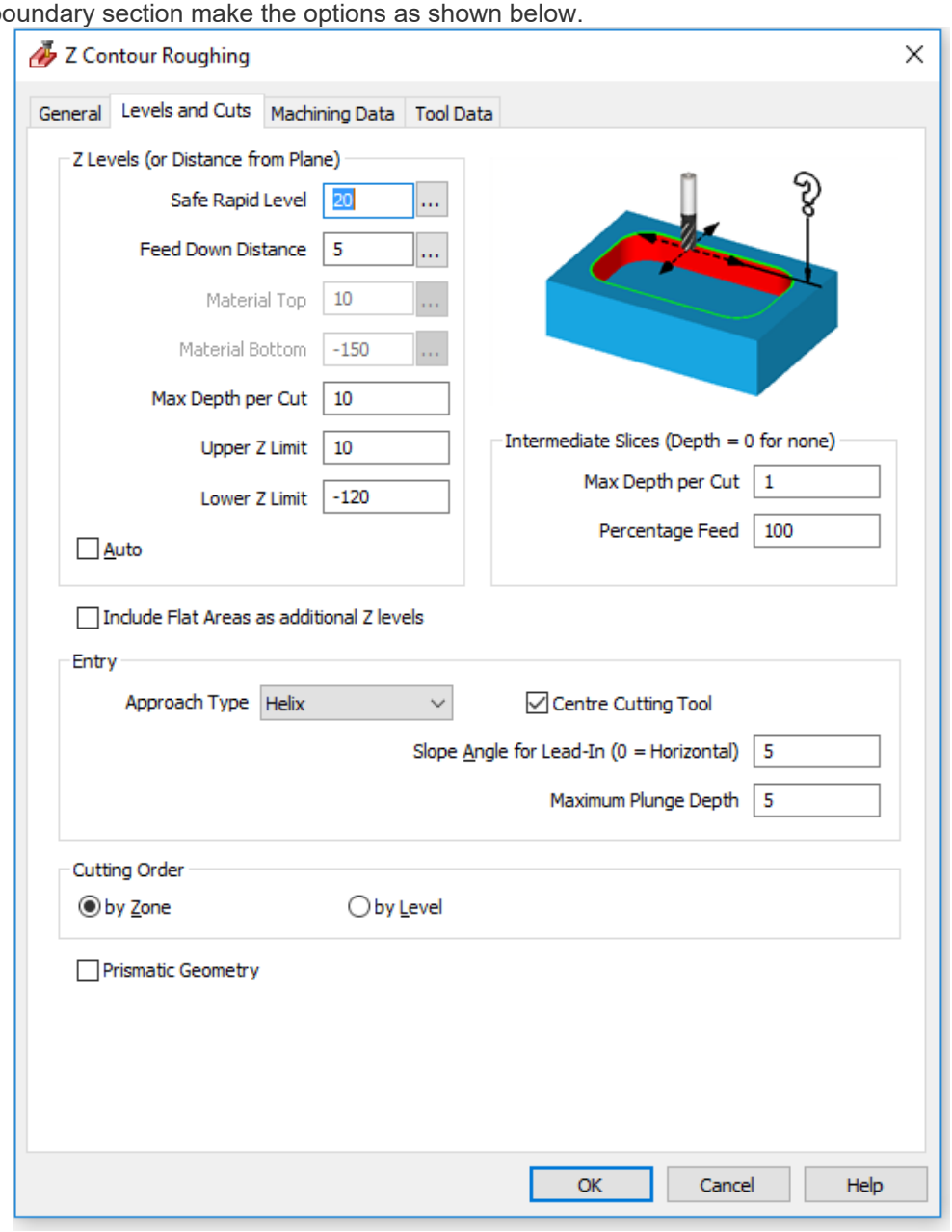


Figure 84 – Levels and Cuts using specific Z depths

The remaining tabs are unaltered as we are using the same tool. Apply the tool path when prompted.

Centre Boundary area

The only difference on the Centre Boundary options is to alter the second dialogue box **Material Bottom value** to **-100.**

Right Boundary area

The only difference on the Right Boundary options is to alter the second dialogue box **Material Bottom value** to **-50.**

Rear Face

Create a work plane on the rear face using **WORK PLANES > Slice Through Work Volume**  . Use the Reverse Current Plane option to ensure that the work plane is set up correctly.

Use **WORK PLANES > Set Work Plane Origin**  to set the work plane datum to the top left corner of the work volume when looking directly at that face,
Create a rectangular boundary as follows; **X-5, Y-101** to **X555, Y-22.**

Using **MACHINE > 3D Machining**  ,
Set the strategy to **Z Contour Roughing.**

Set the dialogue boxes as per the previous options but on the Levels and Cuts tab, alter the **Material Bottom** value to **-20.**

Angled reference Face

Cancel any work planes in use and alter the view of the part to the XY top down option. In this view, create a line across the curved section of the hand rail to allow us to rough machine the material on the side of the turn.

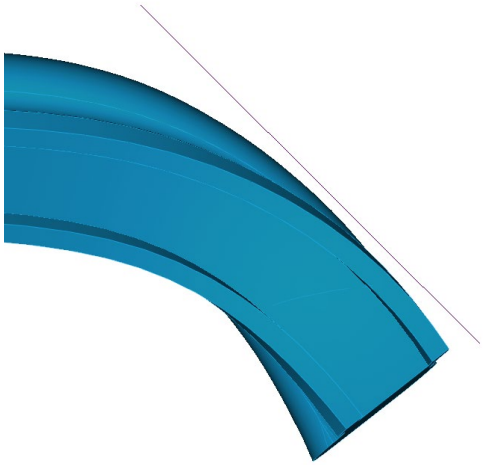


Figure 85 - Created guide line

Use this line to create a work plane as previously done using

WORK PLANES > Slice Through Work Volume  , then click the angled line as the reference element.
On this new work plane, create a suitable rectangular boundary that encloses all the curved section of the handrail and overlaps the previous machining.

Using **MACHINE > 3D Machining**  ,
Set the strategy to **Z Contour Roughing.**
Set the dialogue boxes as per the previous options but on the Levels and Cuts tab alter the following;
Alter the **Material Bottom** value to **-40.**

Rough the Spindle rebate

Using **SOLID MODEL EXTRACT > 3D Edge Extraction**  acquire the profile along the top and bottom edges of the spindle rebate walls.

Use **GEOMETRY > Edit 3D Polyline**  ,in the dialogue box.

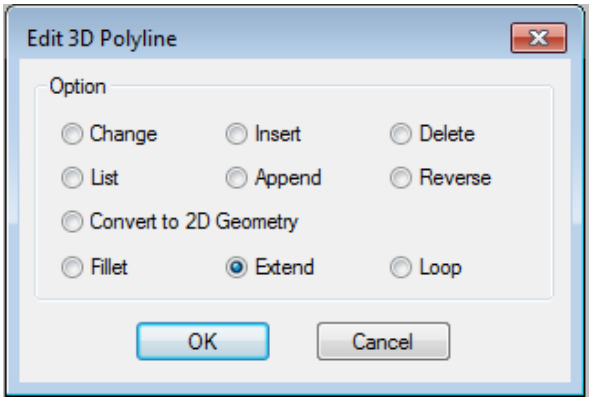


Figure 86 - Edit 3D Polyline options

Choose ⊙ **Extend. <LClick> [OK].**

In the next dialogue.

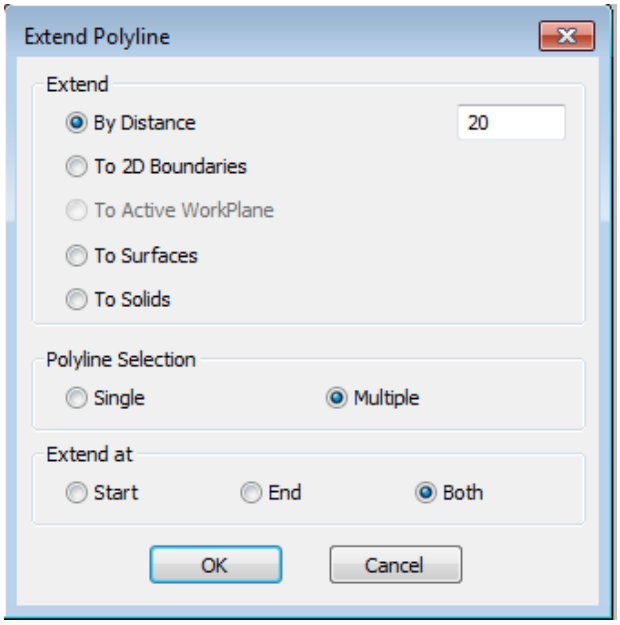


Figure 87 - Extending all extracted lines

Set the option to ⊙ **By Distance** and make the value 20mm.
Make the option for line selection to ⊙ **Multiple** and the ends option to ⊙ **Both**
<LClick> [OK] to continue, the **<LClick> [All]** at the bottom to select the four polylines.
<LClick> [Finish] to complete.

Also check that the ghost tools of the polylines are all pointing in the same direction.

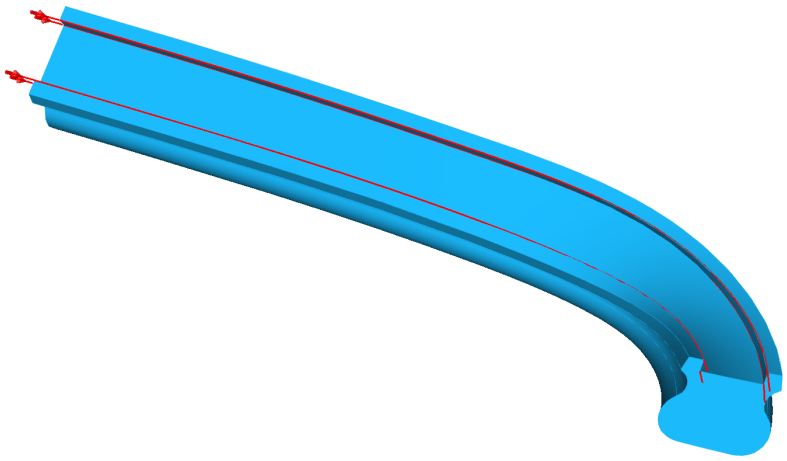


Figure 88 - Polylines set up correctly for machining

Accessing the Layers Project manager page and renaming the polylines for identification will assist in complex parts.

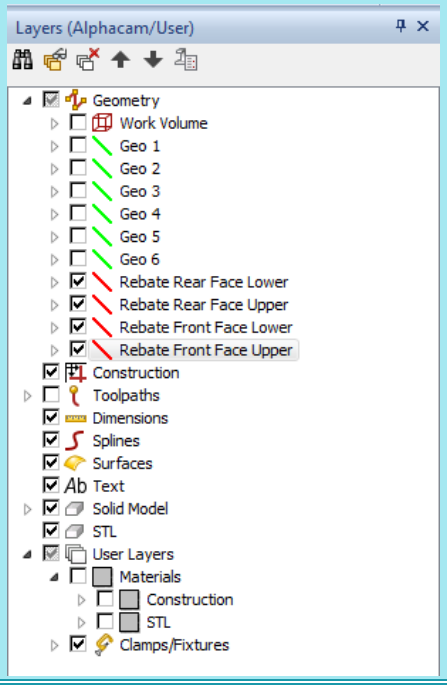


Figure 89 - Renaming geometry elements for ease of identification

Select **MACHINE > Cut Between 2 Geometries**

General

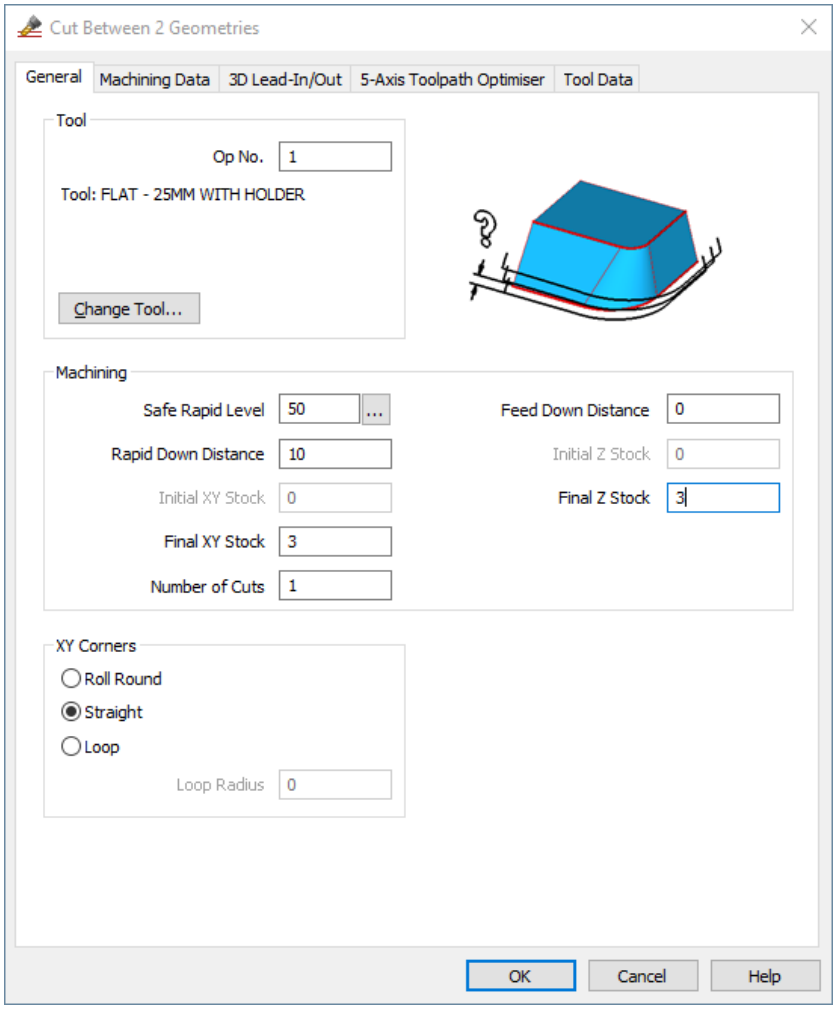


Figure 90 - Cut Between 2 Geometries General tab

Set **Safe Rapid Level = 50,**
Rapid Down Distance = 20,
Feed Down Distance = 10.
Make the **Final XY Stock = 3** and the **Final Z Stock = 3.**

> Setting the Final Z Stock to a negative figure will drive the cutter deeper into the stock if so needed, cutting past the finished bottom edge.

Machining Data

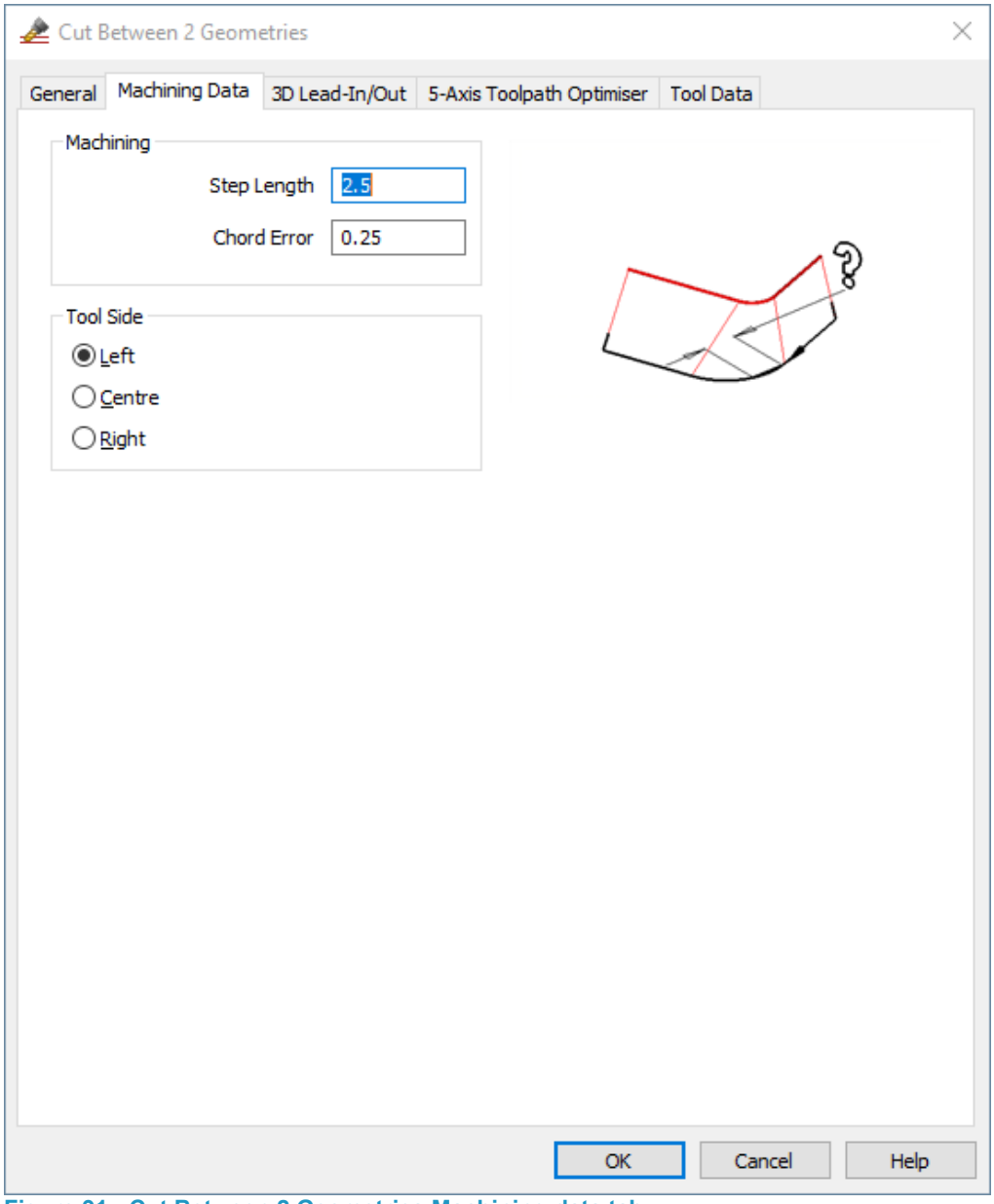


Figure 91 - Cut Between 2 Geometries Machining data tab

On the second dialogue, set the Tool Side to;

- ⊙ **Left** if you wish to machine the front edge first,
- ⊙ **Right** if you intend to machine the rear face first.

> Remember that both options are based on the direction of the Ghost Tools, you will need to have these visible when using this machining cycle.

3D Lead-in/Out

As this machining is to and from areas outside of the model, there is no real need for lead in/out settings.

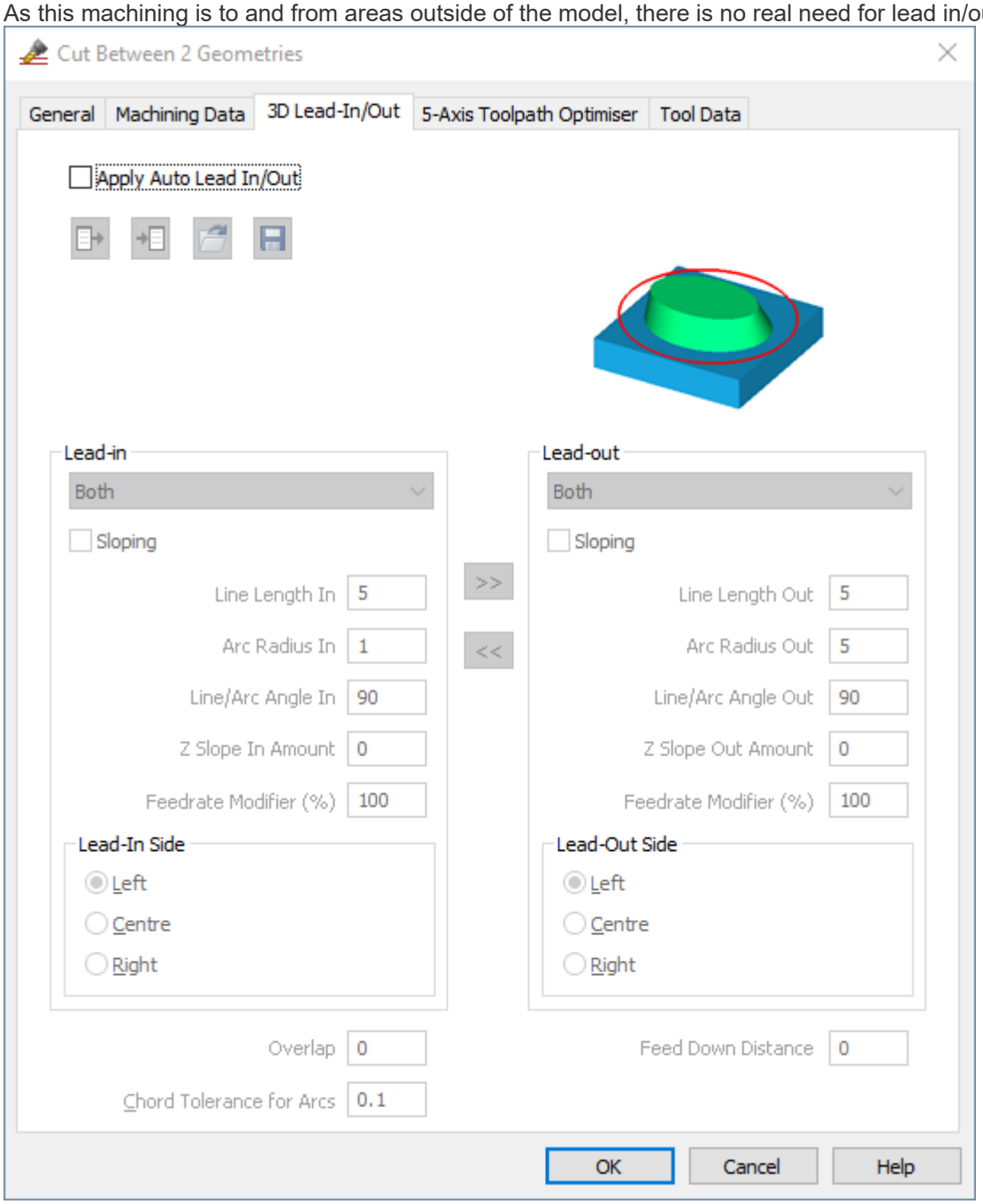


Figure 92 - Cut Between 2 Geometries 3D Lead-In/Out tab

5-Axis Toolpath Optimiser

As this machining a simple pass along the side of the part, there is no requirement for the Optimiser to be active.

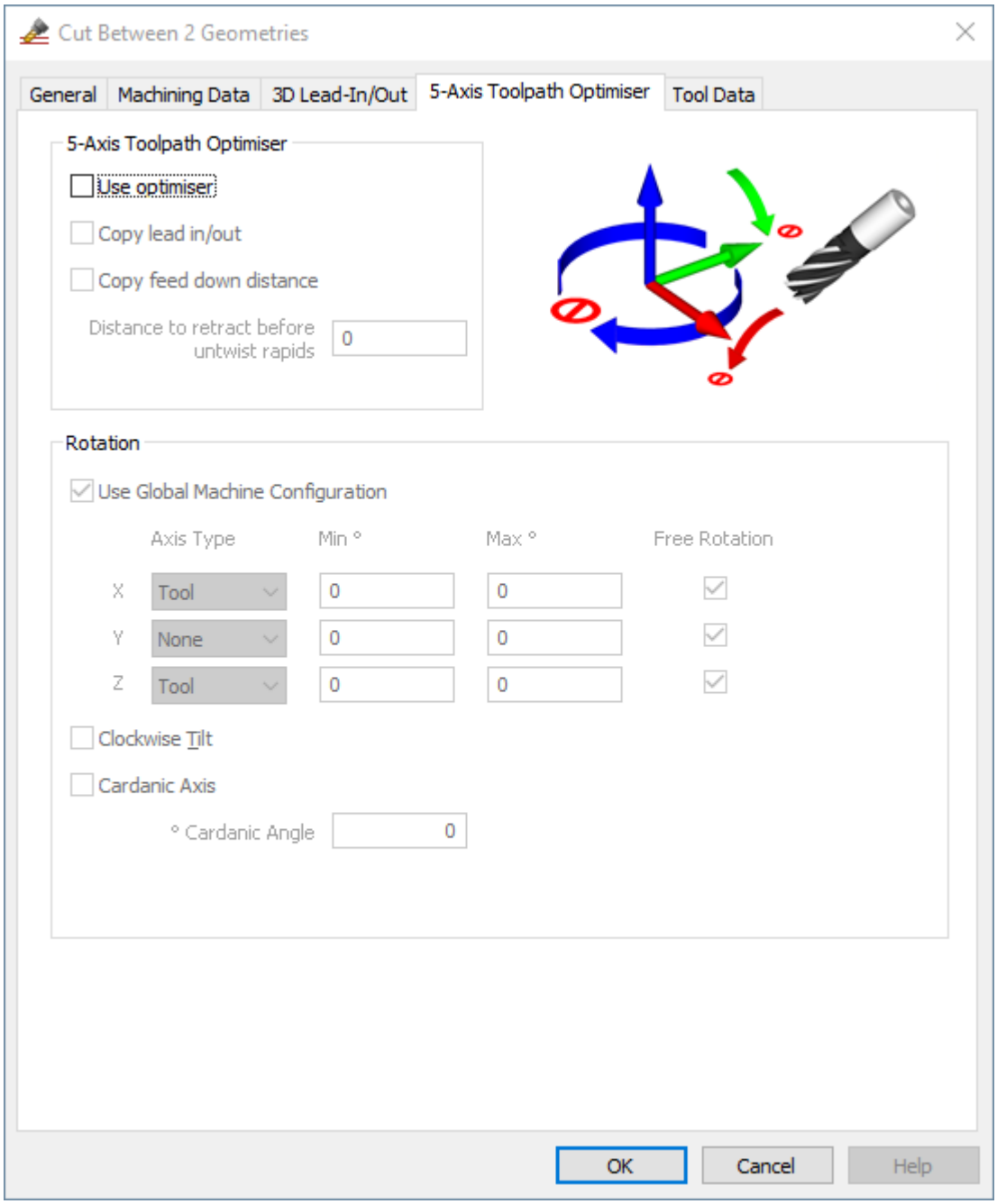


Figure 93 - Cut Between 2 Geometries Optimiser

Tool Data

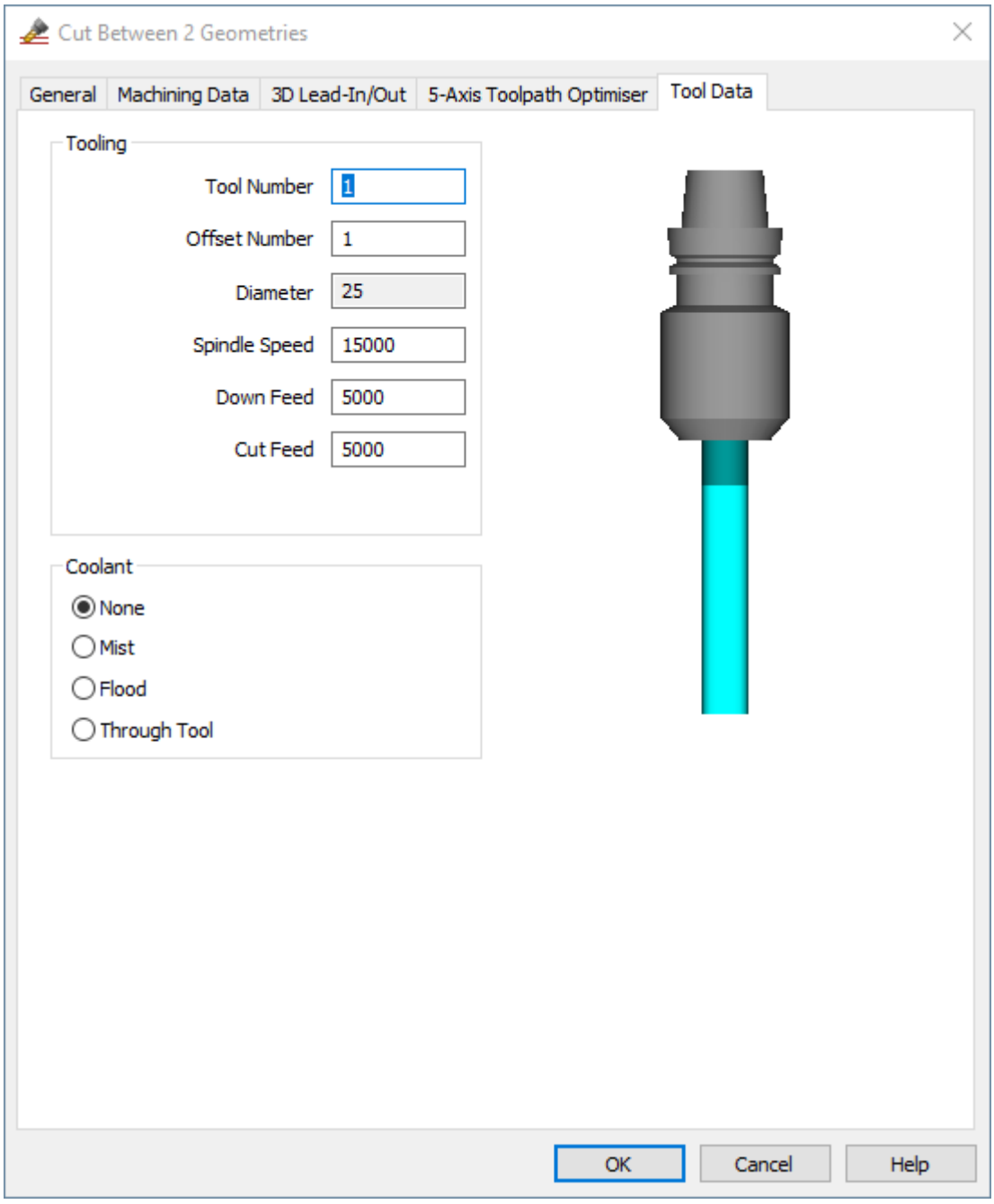


Figure 94 - Cut Between 2 Geometries Tool Data tab

Set the tooling information to suit.
<LClick> [OK].

You are prompted to select;

1. Programming Geometry. This guides the bottom of the tool.
2. Auxiliary Geometry. This guides the upper section of the tool for lean.

<LClick> a **Bottom Polyline** as the **Programming Geometry** and an **Upper Polyline** as **Auxiliary Geometry** of the face you intend to machine.

Perform the process on the second wall to complete the roughing of the rebate.

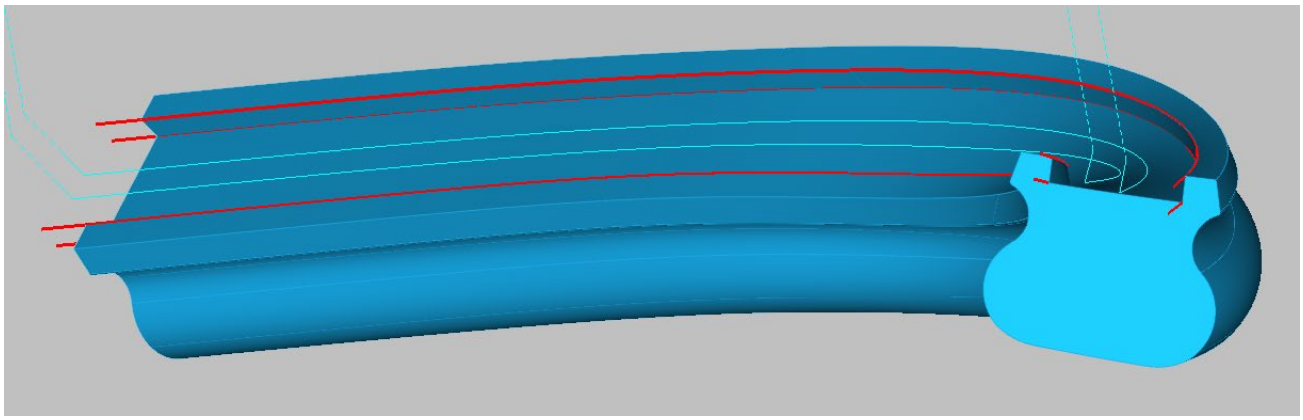


Figure 95 - Applied tool paths

Rough the Spindle rebate tops

Using **Cut Between 2 Geometries** is one method of creating a simultaneous 5 axis tool path, a second method is to use the solid model faces as guides.

This cycle is **Cut Spline or Polyline** 
Cut Spline or Polyline uses a single polyline as a guide for the bottom of the cutter but instead of a second line for the upper guide, the faces of the solid model are used to control the twist and tilt.

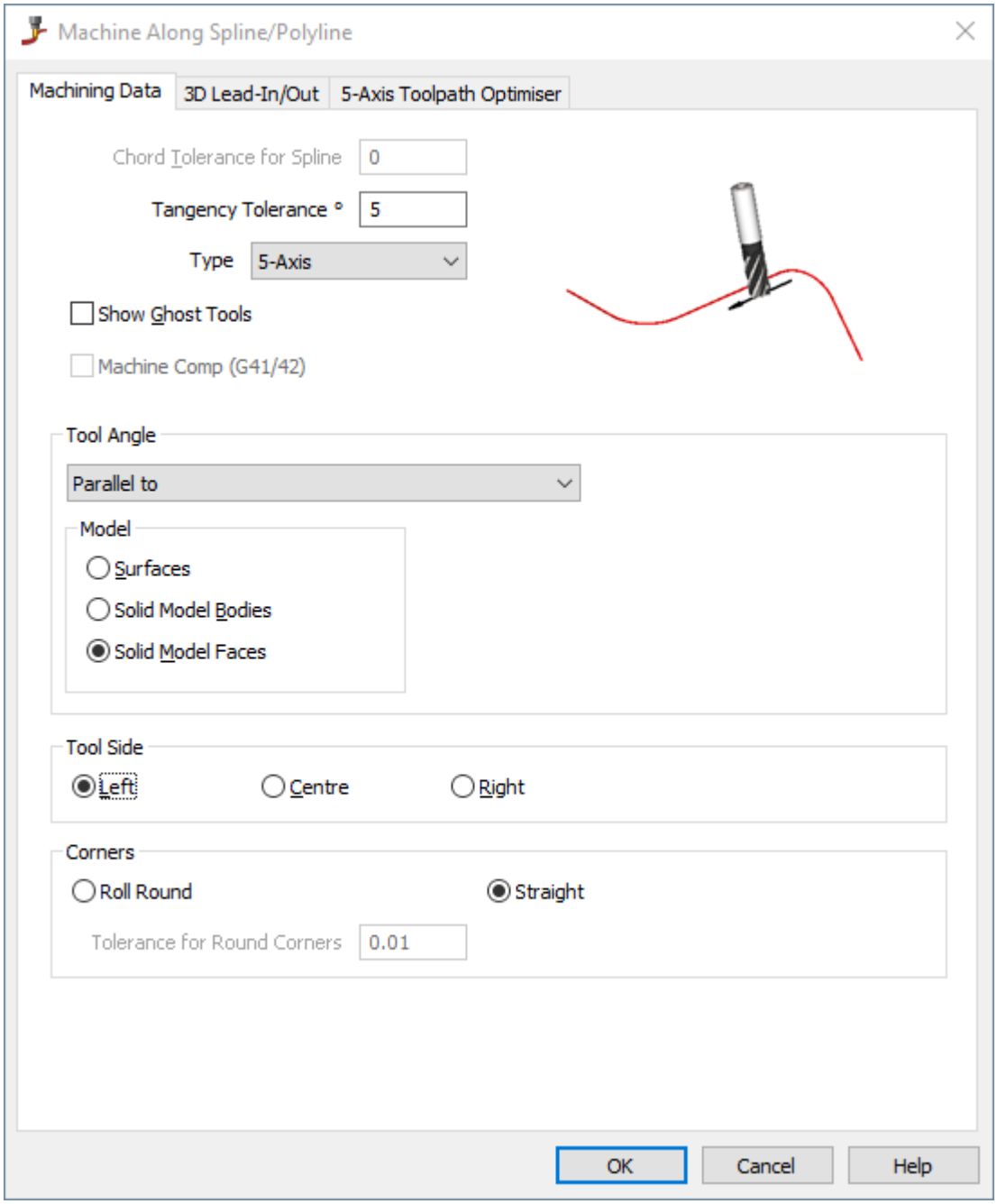


Figure 96 - Cut Spline or Polyline Machining Data

Tool Axis Control options

On the second dialogue page are several specific options that allow fine control of the action of the toolpath.

Type

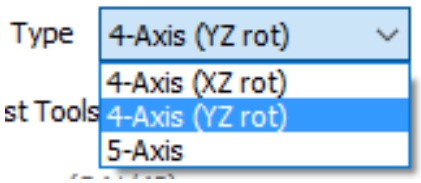


Figure 97 - Axis to rotate on

4-Axis (XZrot)	Rotational toolpath about the Y axis.
4 Axis (YZrot)	Rotational toolpath about the X axis.
5-Axis	Full 5 axis motion toolpath.

Tool Angle

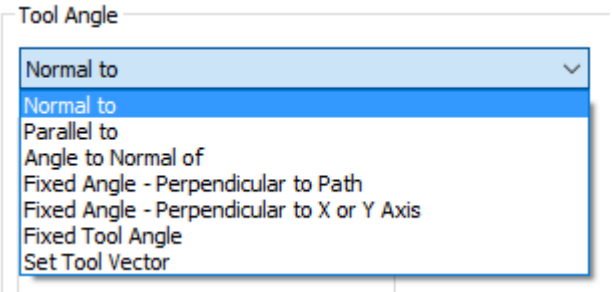


Figure 98 - Tool Angle drop down options

Normal To	Cuts the path using the bottom of the cutter.
Parallel To	Cuts the path using the flutes of the cutter.
Angle to Normal of	Creates a toolpath based on the bottom of the cutter which can then be altered using the Side Tilt and Direction Angle options.
Fixed Angle - Perpendicular to Path	Allows for a manipulation of a 5 Axis toolpath based on a specific vector angle to the chosen polyline.
Fixed Angle – Perpendicular to X or Y Axis	As above but in relation to the X and Y movement.
Fixed Tool Angle	Allows for a user defined angle set to be applied to the toolpath independent of the chosen polyline.
Set Tool Vector	A specific style of angle control using values between 0 and 1 to describe an angle setting with zero being 0° and 1 being 90°.

Guide Options

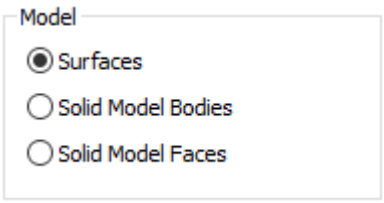


Figure 99 - Tool Angle guide options

Surfaces	Uses a chosen surface to create the Tool Angle from.
Solid Model Bodies	Uses a chosen solid model body as the reference for the Tool Angle.
Solid Model Faces	Uses a selected face from a solid model as the Tool Angle reference.

Tool Side	**Left** **Centre** **Right** Based on the Ghost Tools, these options are set to suit the required side of the area to be cut.
☐ **Show** Ghost Tools	Allows the user to turn on the Ghost Tools without exiting the cycle to make the choice of side easier.
☐ Machine Comp (G41/42)	Only applicable to post processors for machine controls that can apply tool compensation to non 2D toolpaths.
☐ Apply Auto Lead-In/Out	Applies a Lead-In/Out based on the dialogue options set.

Consideration must also be taken to the limits of the actual machine movements when selecting the controlling aspects for this type of cycle.

Use **MACHINE > Cut Spline or Polyline**

General

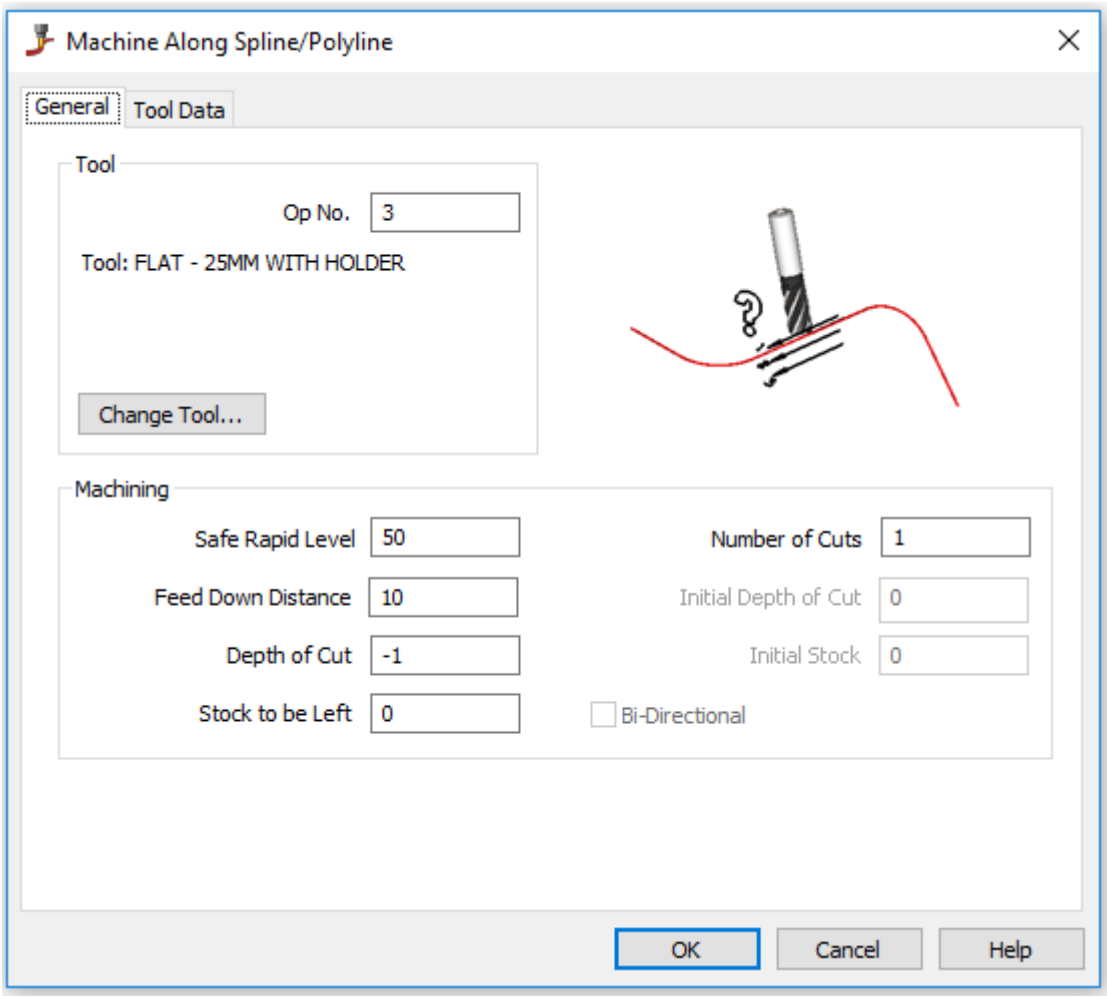


Figure 100 - Cut spline or Polyline General tab

Set the **Safe Rapid Level** to **50,** the **Feed Down Distance** to **20.** Set the **Depth of Cut** to **-1.**

> This is the only Depth dialogue box that requires a **negative** value to leave material **ON** in the **Z** axis direction.

Tool Data

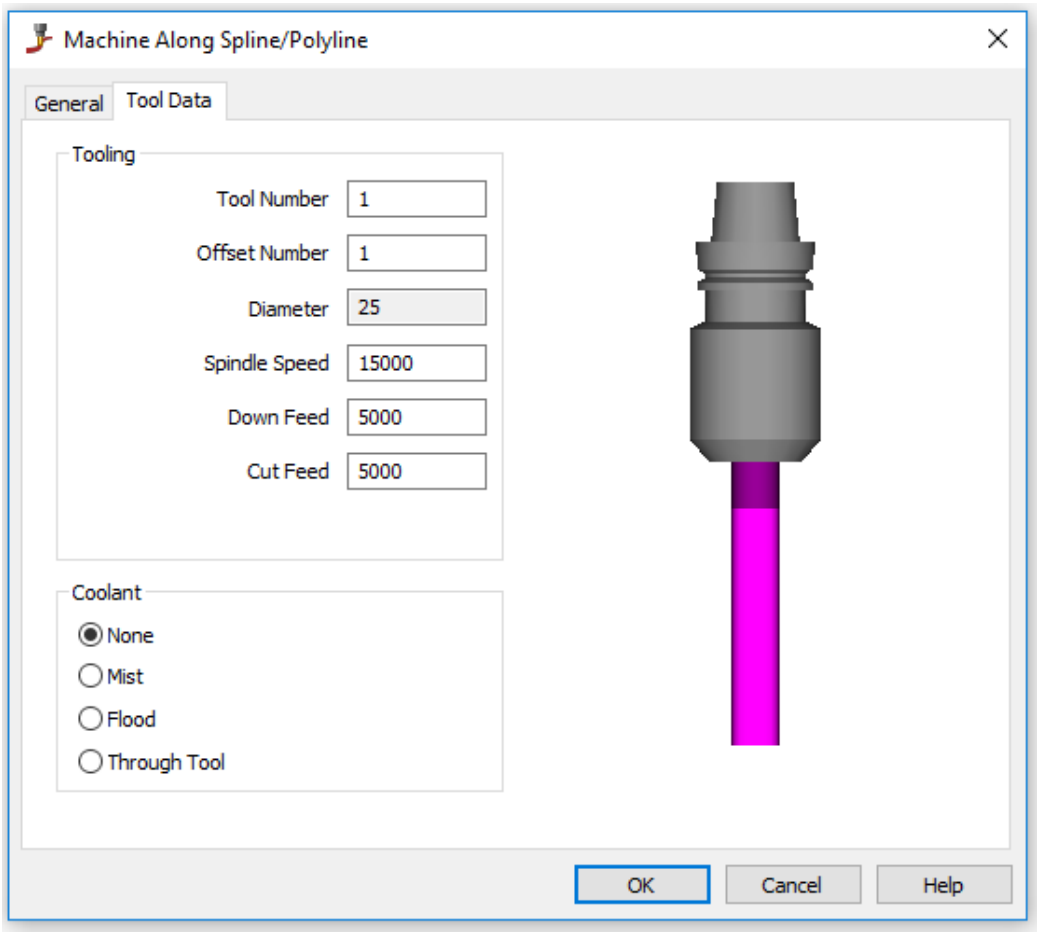
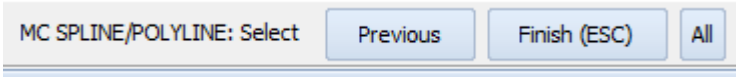


Figure 101 - Cut Spline or Polyline Tool Data tab

<LClick> [OK].

When prompted for a polyline to select,



<LClick> one of the upper polylines extracted earlier.

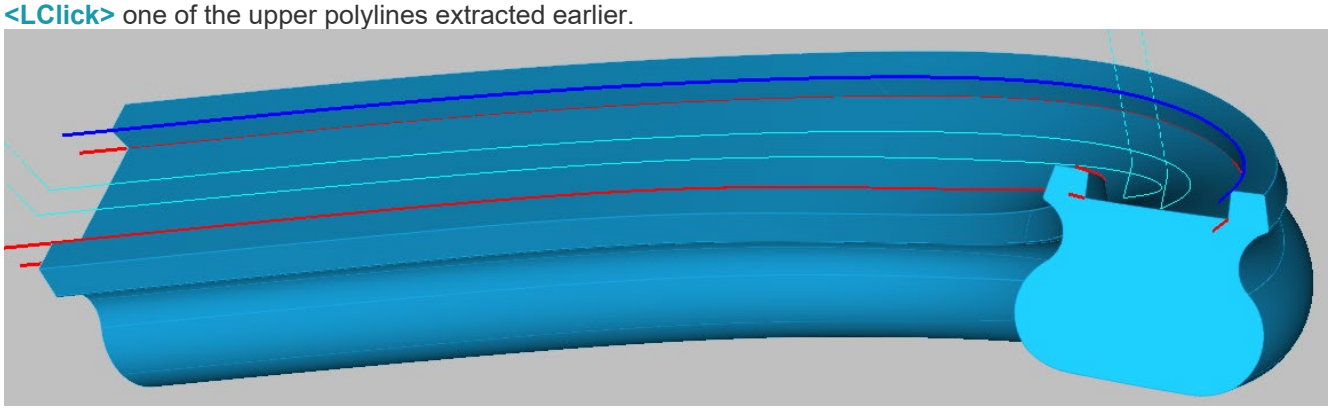


Figure 102 - Select the upper polyline as the guide

<LClick> [Finish (ESC)] to continue.

Machining Data

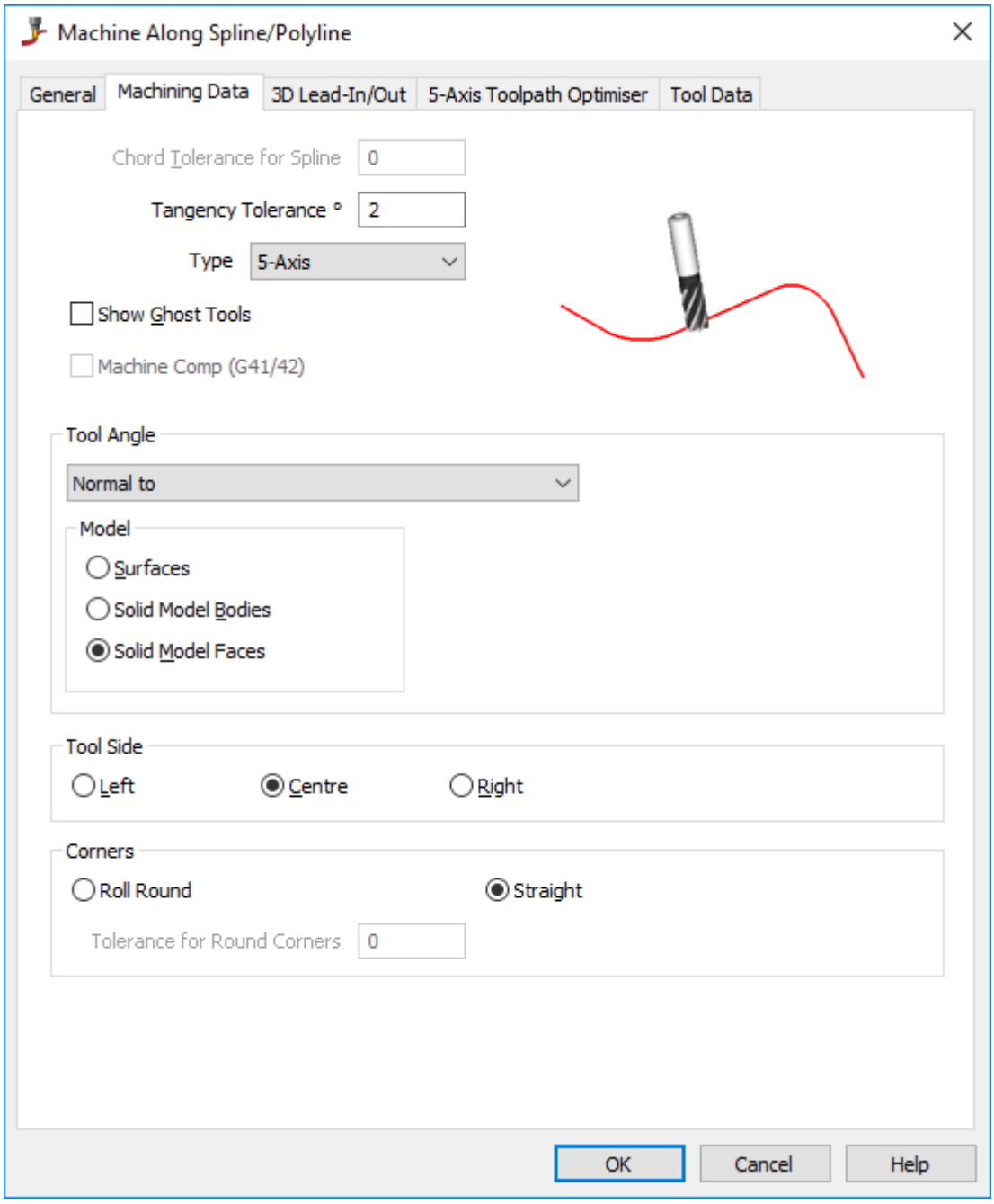


Figure 103 - Cut Spline or Polyline Machining Data tab

Setting the Tool Angle to **Normal to** allows for machining on the bottom of the cutter and the
⊙ **Solid Model Faces** controls the tilt and twist of the head or table depending on the machine type.
⊙ **Centre** places the full diameter of the tool over the face we wish to machine.

3D Lead-In/Out

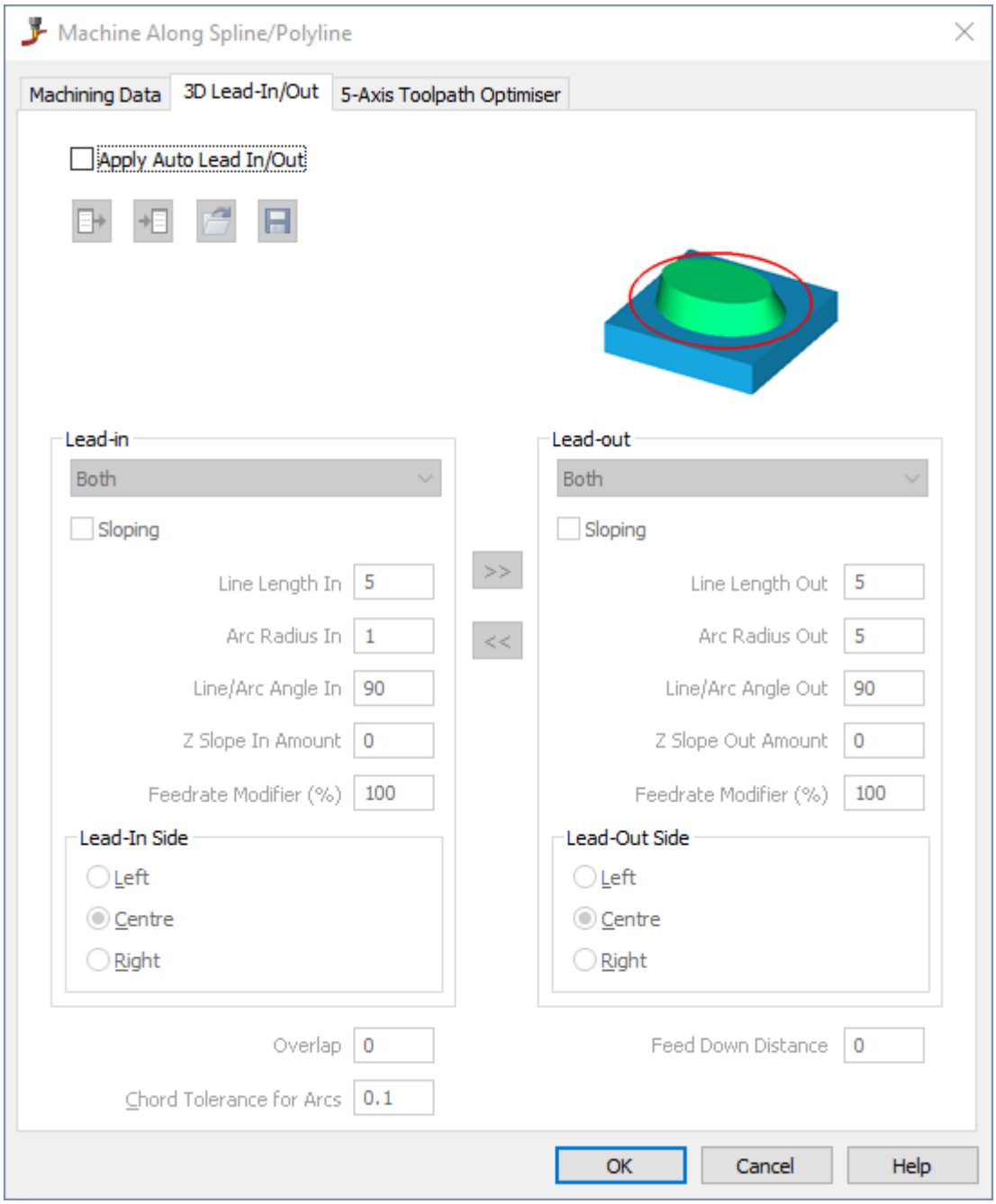


Figure 104 - Cut spline or Polyline 3D Lead-In/Out tab

Due to the extended polyline, no lead in or out is required on this example.

<LClick> [OK].

5-Axis Optimiser

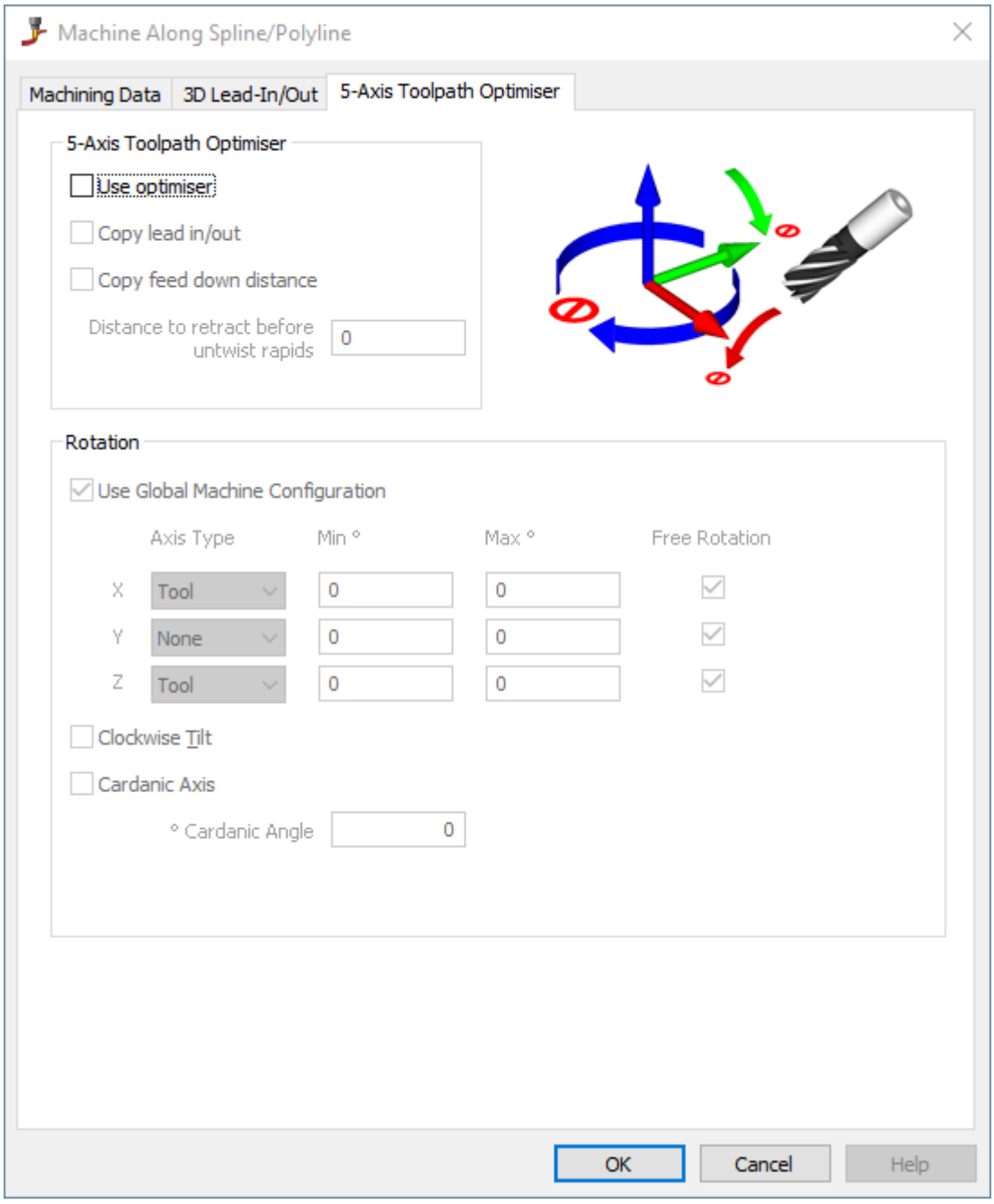


Figure 105 - Machine Polyline 5-Axis Optimiser

Machine Along Spline/ Polyline also has the option to use the 5-Axis Toolpath Optimiser settings if required.

When prompted, select the solid model face that corresponds to the chosen polyline.

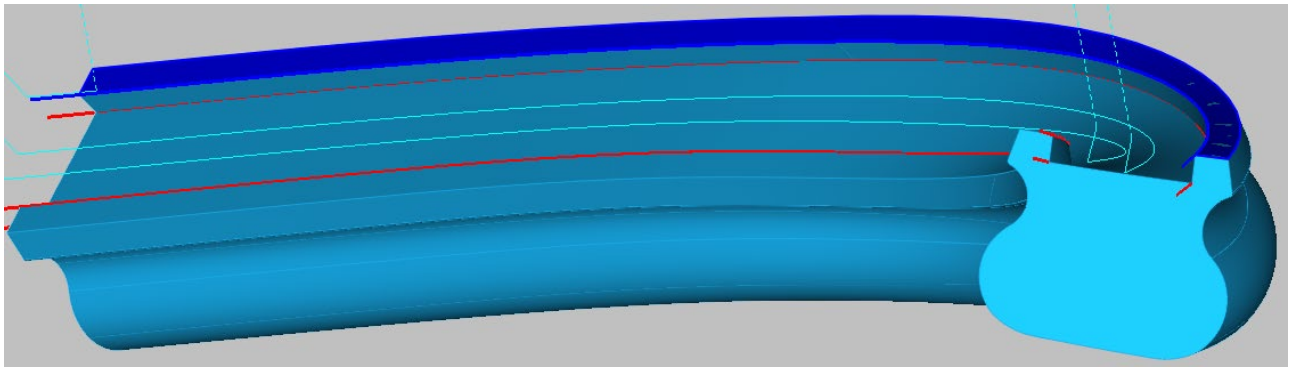


Figure 106 - Selecting the face to machine

<LClick> [Finish (ESC)] to continue.

Repeat the process on the second top face.

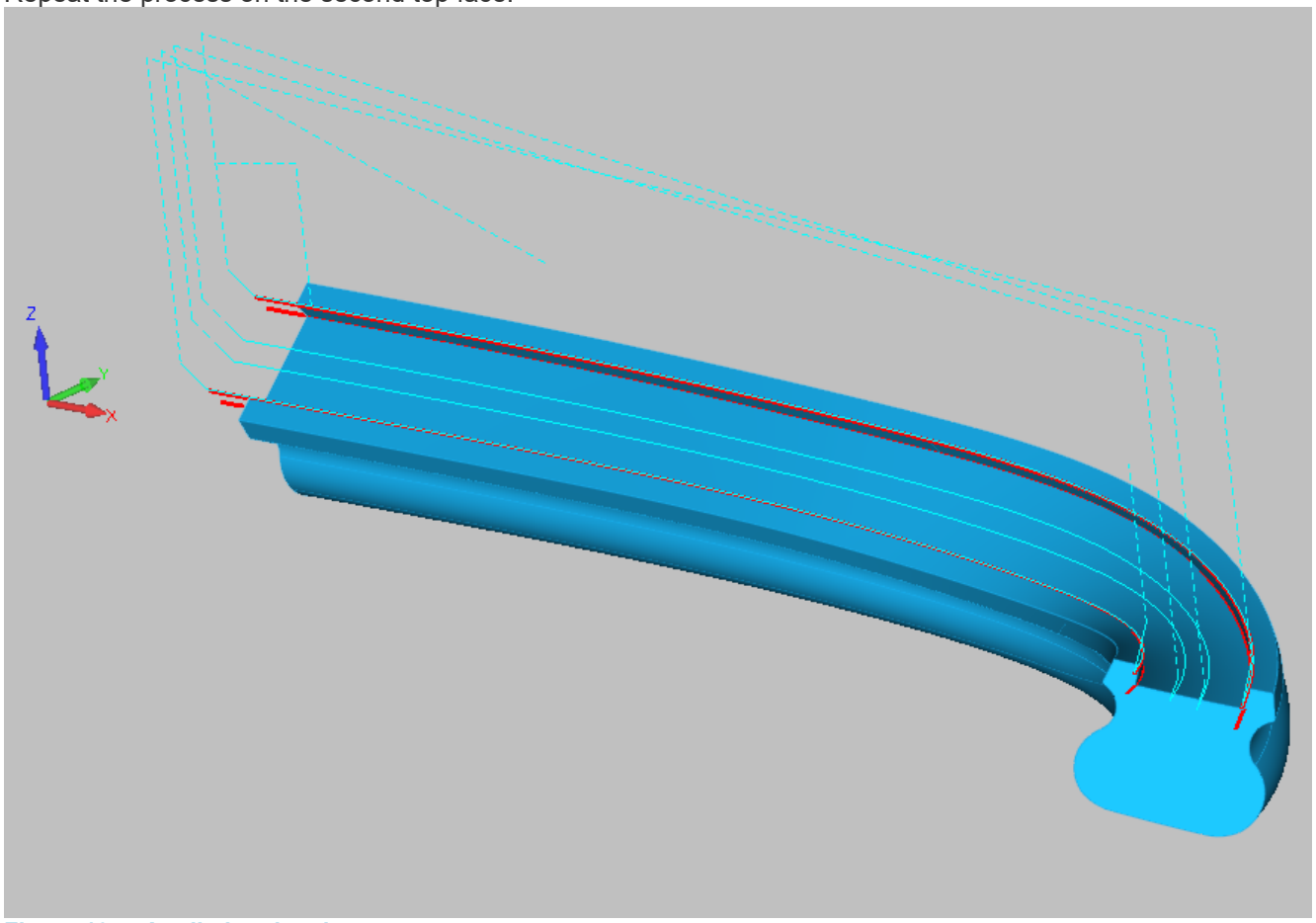


Figure 107 - Applied tool paths

End Trimming

To create the correct driving geometries for the two ends we need to set up two new work planes using

WORK PLANES > From Solid Model Face .
Make a work plane from the left hand end of the model.

Draw two Lines, one along the bottom edge of the rebate feature and a second along the bottom face of the handrail. Extend these lines 25mm on each end so that they are beyond the model features.
Use the Ghost tools and Tool Directions if required to ensure that the two lines are travelling in the same direction.
Repeat the process on the right hand end of the hand rail.

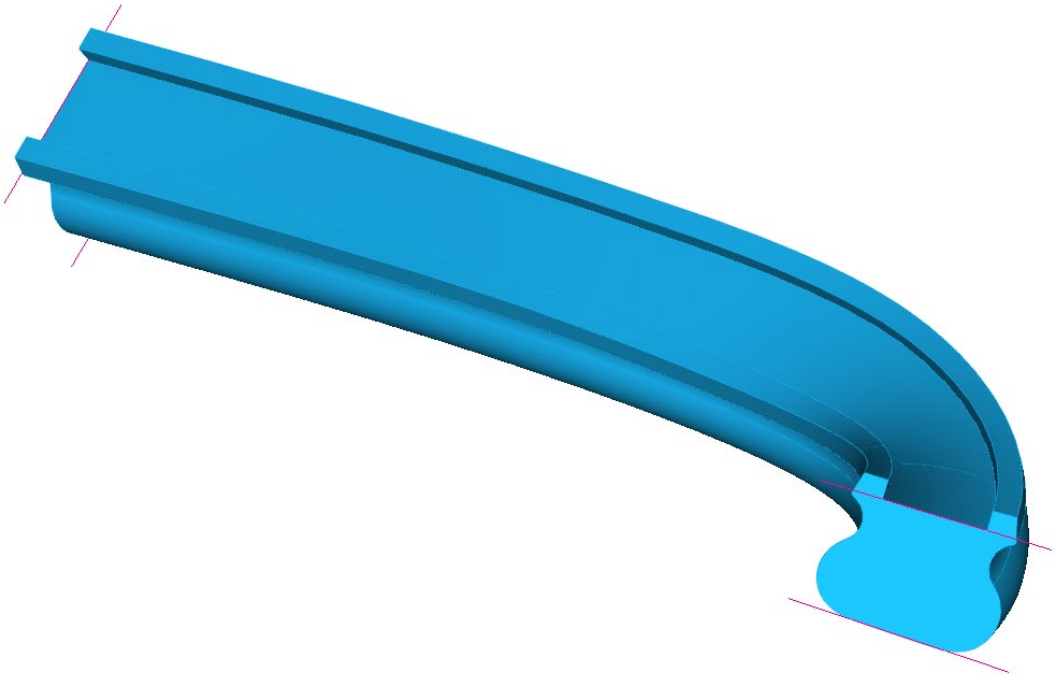


Figure 108 - Created work planes and geometry for the end faces

Select **WORK PLANES > Cancel Work Plane**

Select **MACHINE > Cut Between 2 Geometries** set the options as follows;

Set **Safe Rapid Level = 50,**
Rapid Down Distance = 20,
Feed Down Distance = 10 on the first dialogue page, make the **Final XY Stock = 1** and the **Final Z Stock = -1.**
On the second dialogue, set the **Tool Side** options to suit the tool being on the outside of the model depending on the ghost tool directions.
As this machining to and from areas outside of the model, there is no real need for lead in/out settings required.

Alternative End Trimming 1

If the manufacturing method permits, then the use of the

SOLID MODEL EXTRACT >Edge for Sawing  extraction technique can be used instead.
This method will generate the correct geometry options that will then create the work planes for the toolpaths as the sawing cycle is applied.
This method is only suitable for machines that support 5-axis sawing.

Alternative End Trimming 2

Extracting the lower edge of each face as shown using

SOLID MODEL EXTRACT > 3D Edge Extraction  **,** then using

MACHINE > Cut Spline or Polyline  and making the **Tool Angle option** to **⊙ Parallel to ⊙ Solid Model Faces.**
Then applying the cycle as previously described.

Alternative End Trimming 3

Depending on the limitations of the machine in use, it could also be feasible to use Feature Extraction to generate the complete end face profiles and then use a 2D pocket cycle to clean the faces.

Finishing

To create a good working element to finish the rebate floor, it is necessary to manipulate the model and create guide surfaces for the machining strategy we will use.

Surface Extraction

Using **SOLID MODEL EXTRACT > Surfaces from Faces**  extract the two faces of the spindle rebate sides.

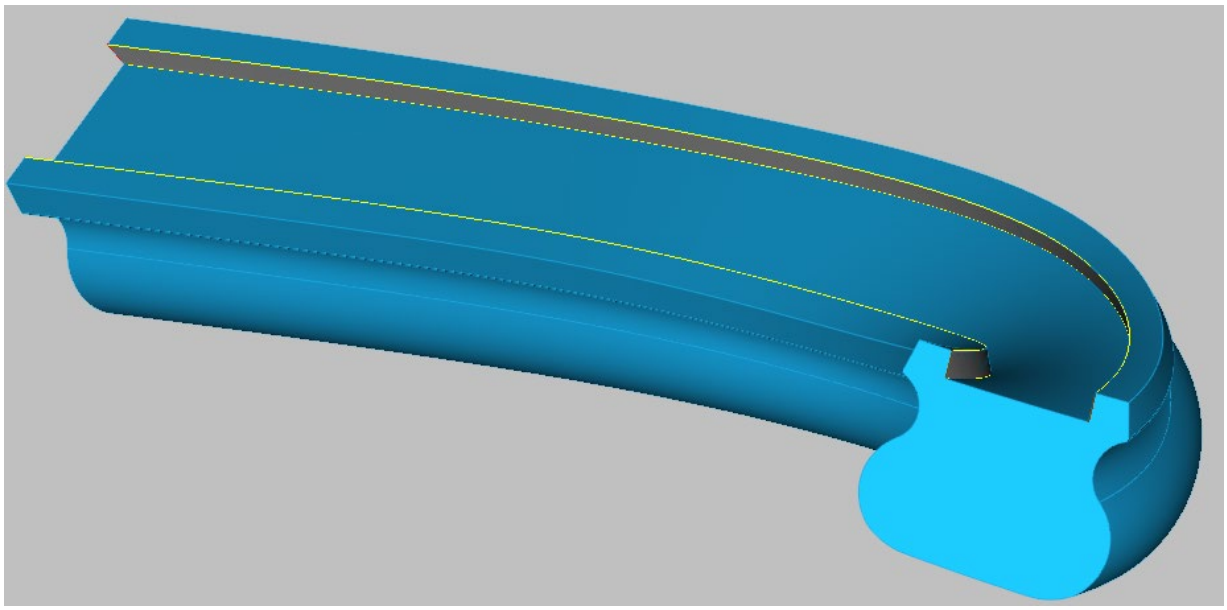


Figure 109 - Extracting the side wall surfaces

Use **EDIT > Break,Join etc > Offset**  to offset the side faces of the spindle rebate inbound by **6mm.** setting the option ☑ **Delete Original** may assist in identifying the required options later.

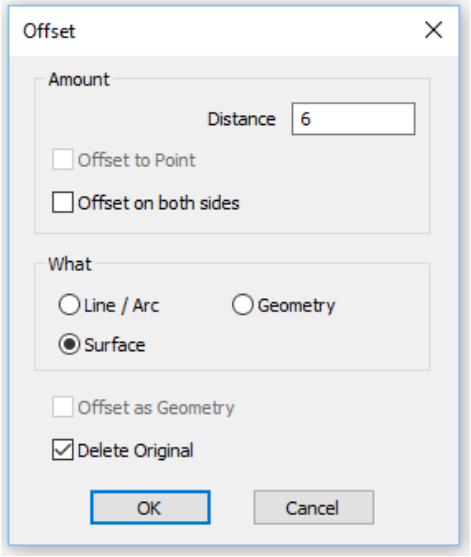


Figure 110 - Using the Surface offset option

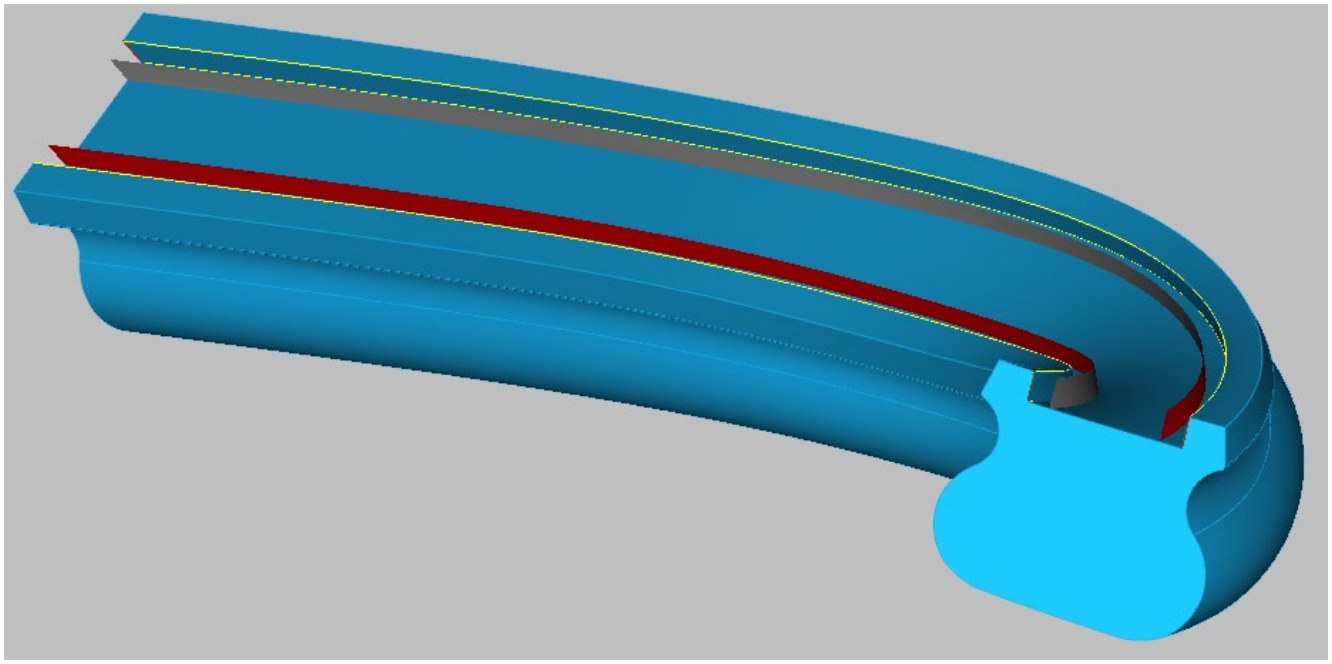


Figure 111 - Extra surfaces after using the Offset command

Note that there is no requirement to choose a side for the offset when working with surfaces, the offset will always be in the direction away from the **Silver (outer) surface** face.

Offsetting in this manner will allow a 10mm Ball End cutter to finish the rebate floor without touching the walls.

When creating surfaces using the Feature Extraction method, you will always gain the outer edges of the surfaces. In this case, we do not require these additional extracted geometries, so you can delete them to avoid confusion.

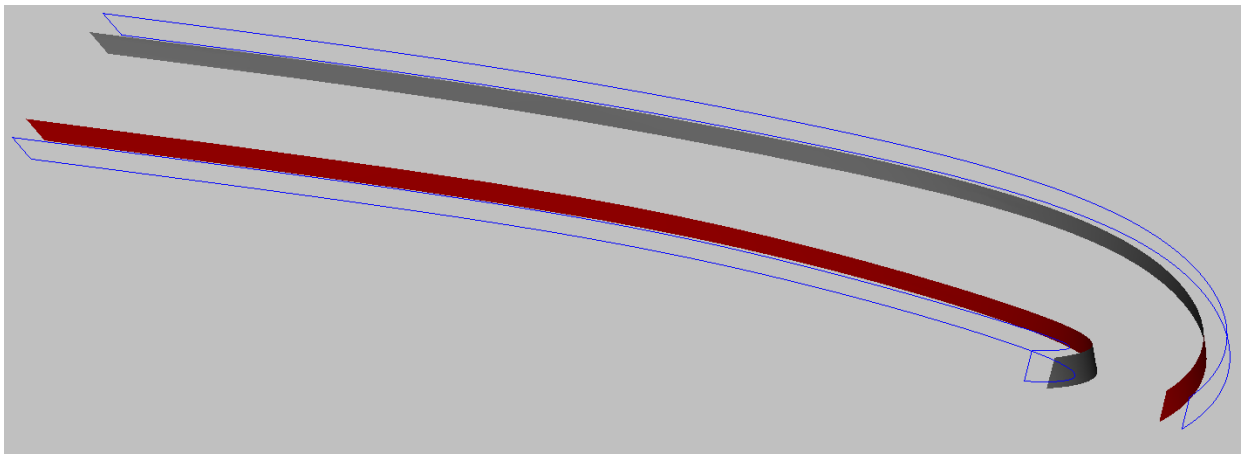


Figure 112 - Delete the original surfaces and the acquired outline profiles

Creating the new floor driving surface

Extract the edges of the two new surfaces with **3D > Edit Surfaces > Extract Edge From surfaces.**

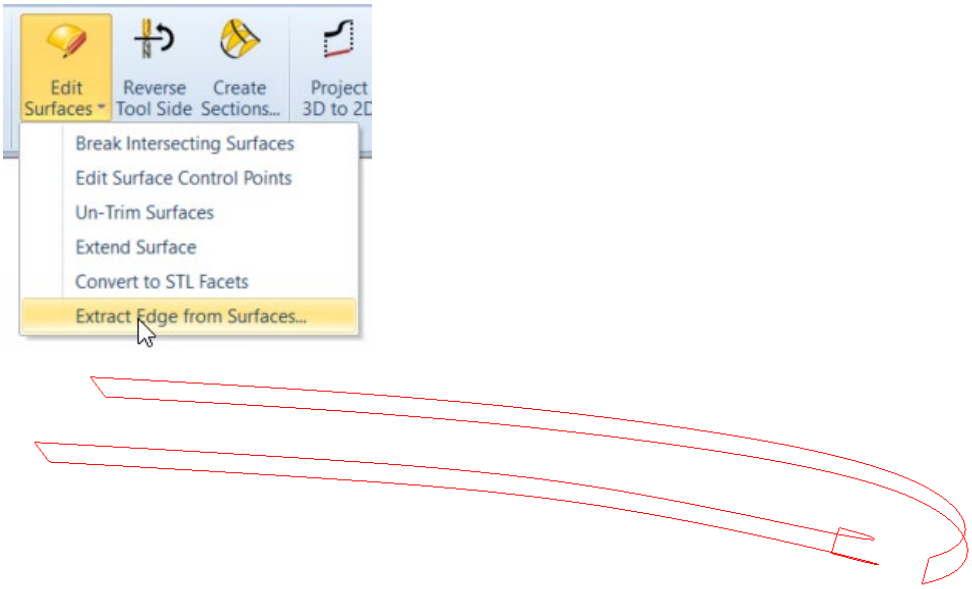


Figure 113 - Full extracted edges of offset surfaces

Break the extracted polylines on the lower edges at each extreme corner.

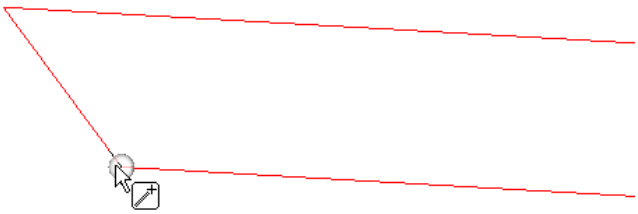


Figure 114 - Location to Break the geometry

Delete the unwanted lines that formed the top edges and the ends of the surfaces.

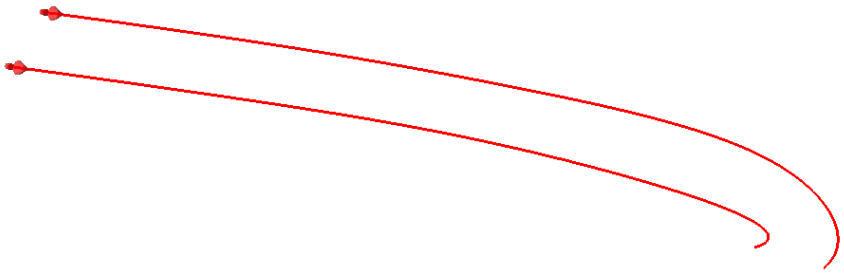


Figure 115 - Remaining required polylines

Depending on how the extraction and offset process worked, you may have to use the **Join command** to create two single polylines.

To allow the toolpath to machine correctly the polylines need to be extended as the offsetting process creates a shorter line on one side.

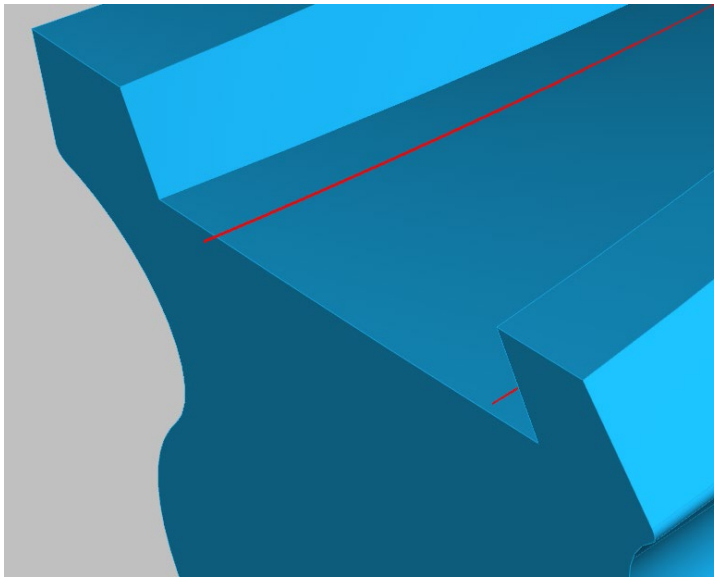


Figure 116 - The polylines will need manipulation in this example

Use **GEOMETRY > Edit 3D Polyline**  ,
in the dialogue box choose ⊙ **Extend.**

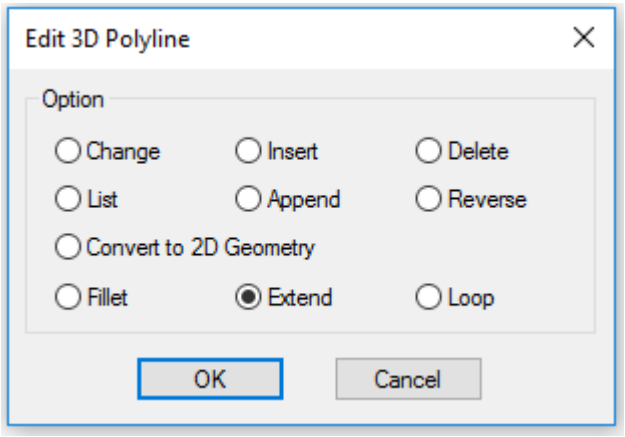


Figure 117 - Edit 3D Polyline dialogue

<LClick> [OK].

Set the option to ⊙ **By Distance** and make the value 10mm.
Make the option for line selection to ⊙ **Multiple and the** ends option to ⊙ **Both.**

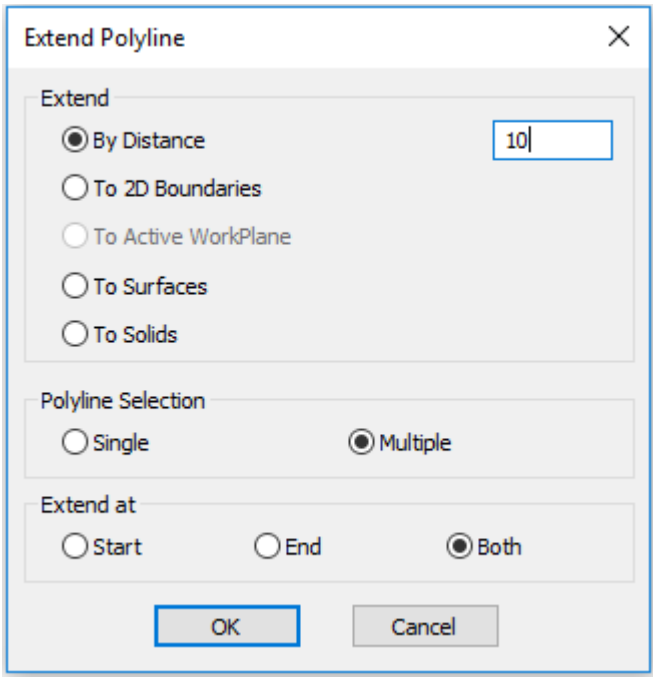


Figure 118 - Extend Polyline dialogue

<LClick> [OK] to continue, then **<LClick> [All]** at the bottom to select the four polylines.
<LClick> [Finish (ESC)] to complete.

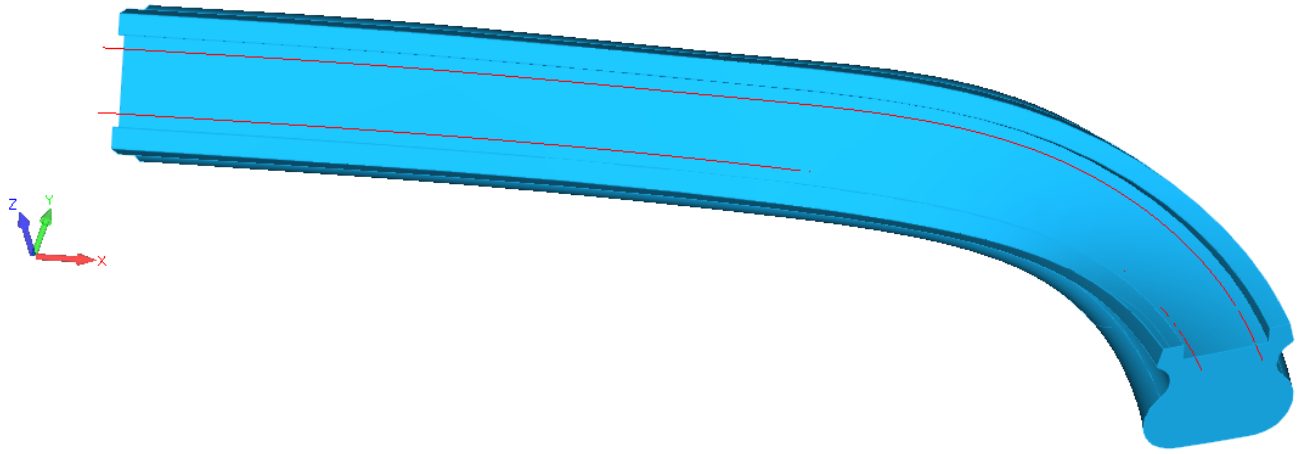


Figure 119 - Extended Polylines beyond the end of the part

Finally create a polyline from one bottom edge line to the other.

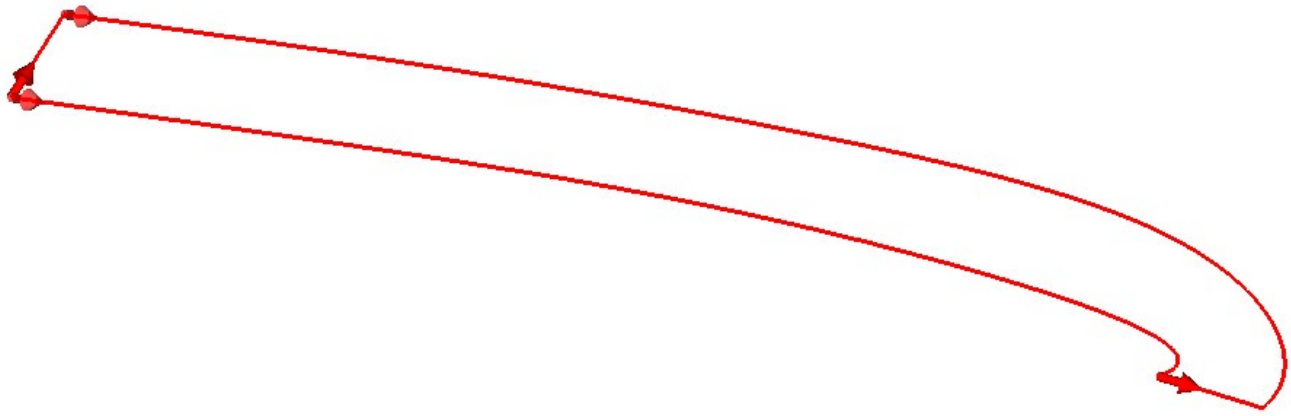


Figure 120 - Creating the linking polylines for the surface generation process

Using **GEOMETRY > 3D Surfaces** 
Make the option in the dialogue box **⊙ Coons Patch (4 curves),** Pick one of the linking polylines as the first line, select one of the bottom surfaces edges as the second, choose the second link line as the third option and finally the second surface edge as the last element.

You will be presented with a dialogue box asking for a number of steps along the major and minor sides, these values go towards the precision of the finished surface. Higher values will give a more precise surface to follow and by that, a better quality of finish.

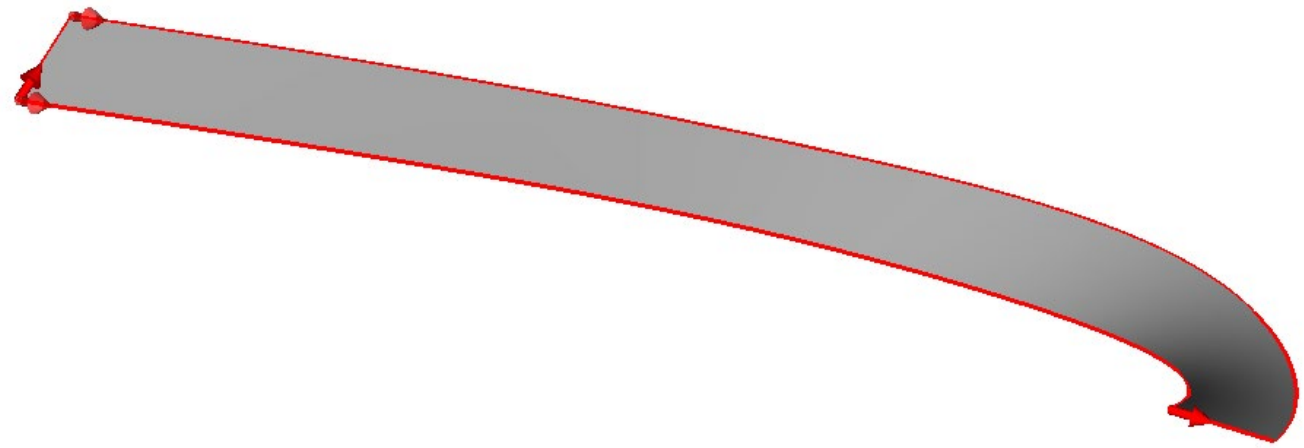


Figure 121 - Completed surface for machining

Rebate floor finishing

Select the **Ball End 10mm + holder** for the next machining operation.

Use **MACHINE > 3D Machining** 

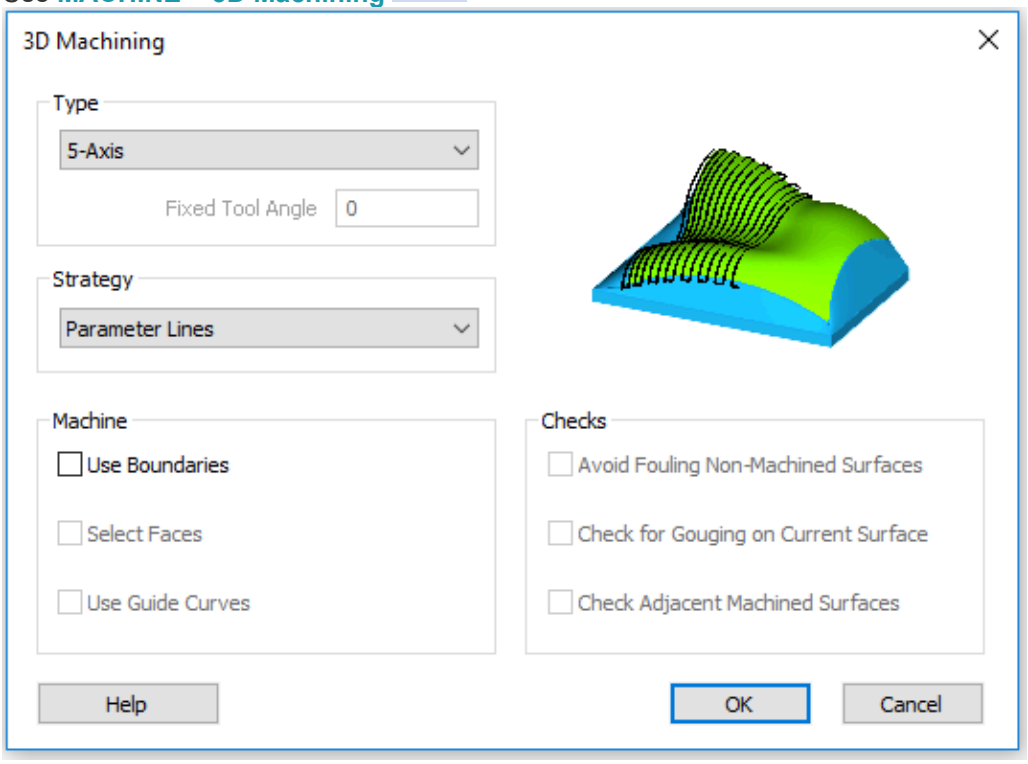


Figure 122 - 3D Machining strategy selector

Set the **Type** drop down to **5-Axis** and the **Strategy** drop down to **Parameter Lines**

As a rule, it is usually easier to program a 3 Axis tool path and then use the conversion processes detailed earlier in this manual to create the required 5 Axis path, however in this example, this is one of those times that you can apply the cycle directly as a 5 Axis option from the start due to the simple nature of the area to be machined will not cause any strange axis moves.

General

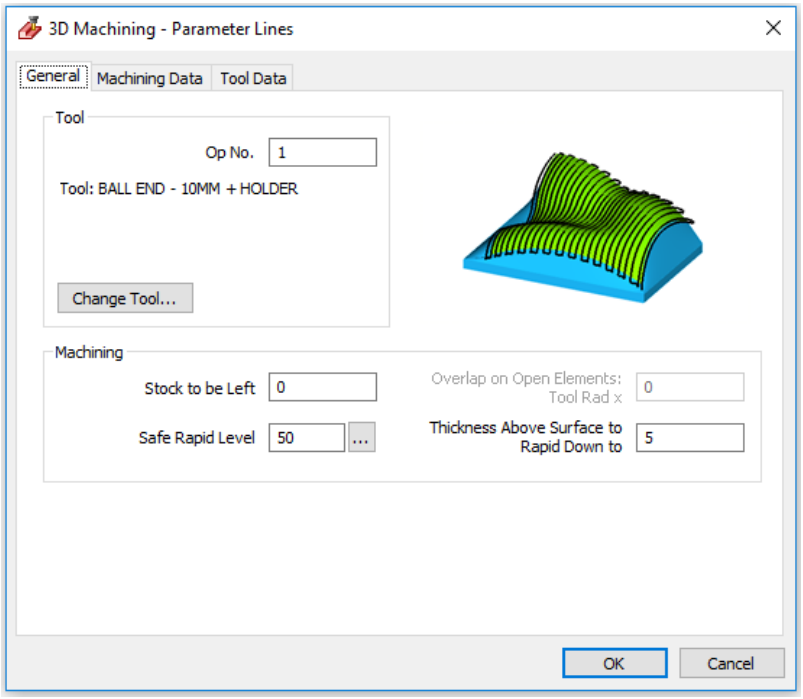


Figure 123 – Parameter Lines General tab

Set the options as shown.

Machining Data

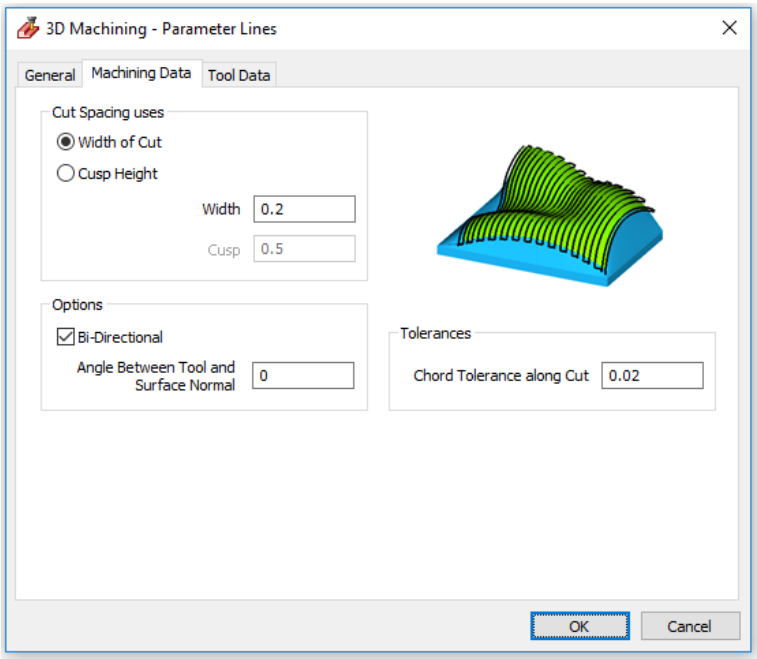


Figure 124 – Parameter Lines Machining Data tab

Set the options as shown.

Tool Data

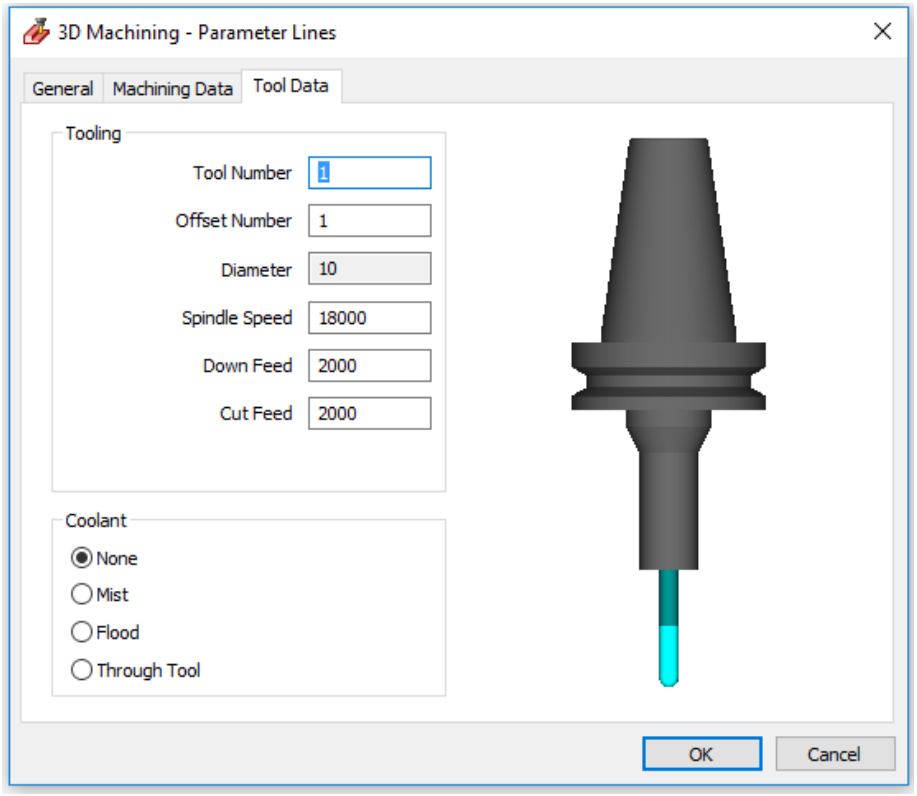


Figure 125 - Parameter Lines Tool Data tab

Set the options to suit, then **<LClick> [OK].**

Select the newly created base surface as the item to machine, for the **Start Point** click close to one corner of the open ends of the rebate.

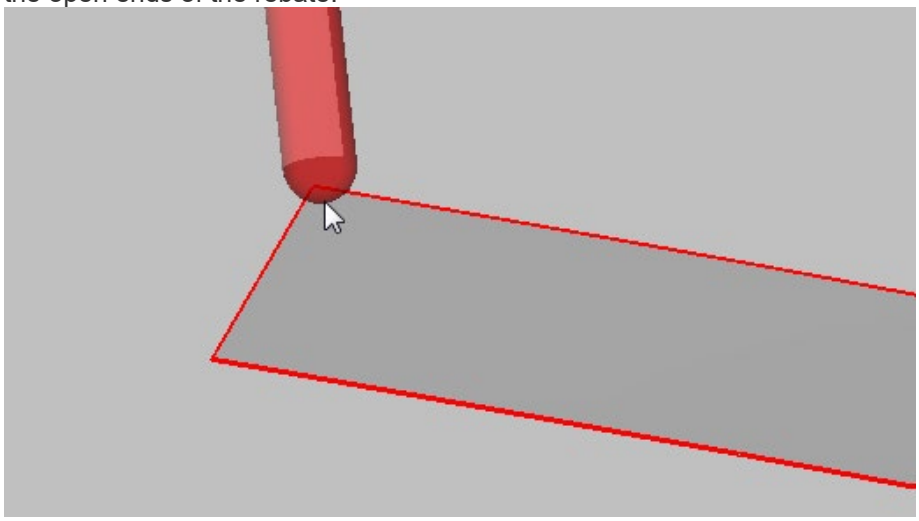
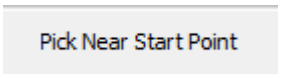


Figure 126 - Setting a location for the Start Point of the cycle



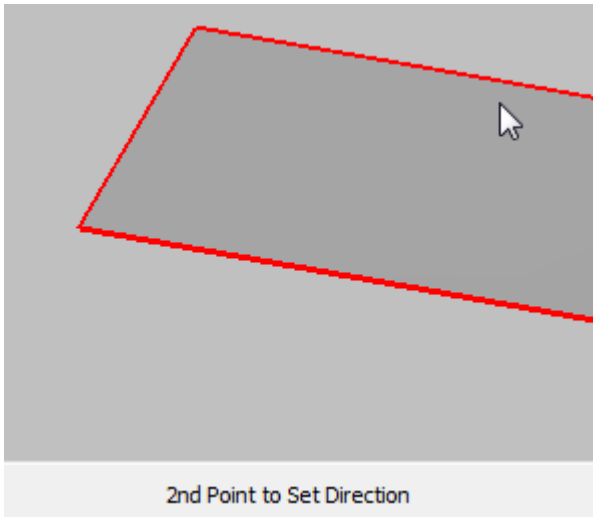


Figure 127 - Choosing the second point for direction of cut

For the **Direction** option, click along the rebate to create the direction vector.

Direction of cut is dictated by the second choice, selecting a point along the sweep of the rebate will drive the tool path around the sweep whereas selecting a point along the open edge away from the chosen start point will force the cut to be across the rebate.

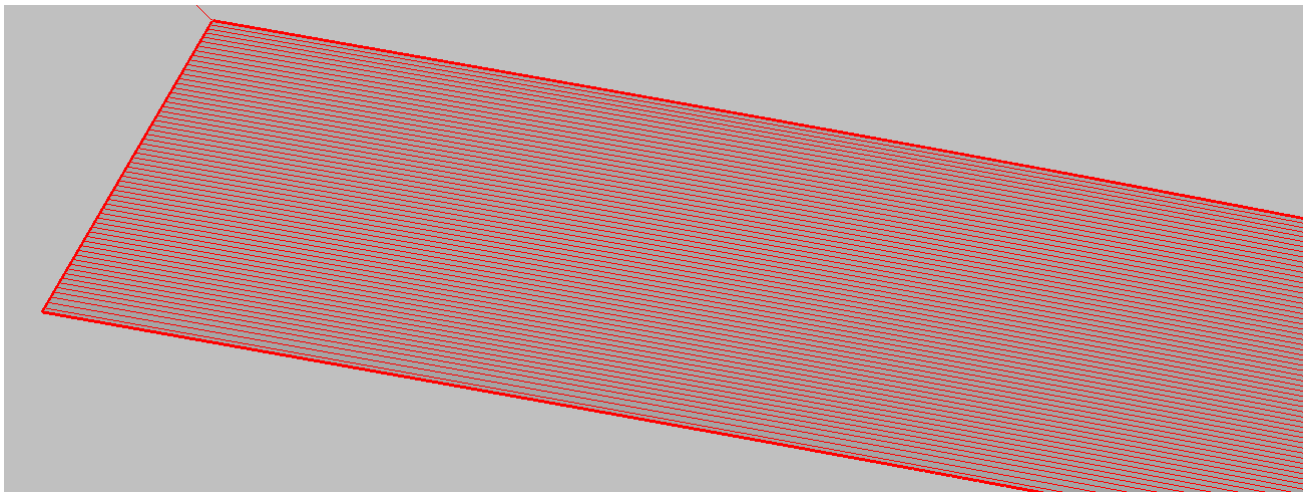


Figure 128 - Applied tool path

Rebate Inside Wall Finishing

To finish the side walls of the rebate, change the tool to the **12mm Flat + Holder.**

Select **MACHINE > Cut Between 2 Geometries**  set the options as follows;
Set **Safe Rapid Level = 50, Rapid Down Distance = 20, Feed Down Distance = 10** on the first dialogue page, make the **Final XY Stock =0** and the **Final Z Stock = 0.**

On the second dialogue, set the Tool Side options to suit the tool being on the inside of the model depending on the ghost tool directions. As this machining is to and from areas outside of the model, there is no real need for lead in/out settings required.
When you have made you choices in all dialogue boxes and clicked the appropriate **[OK]** buttons, pick **Bottom Polyline** as the **Programming Geometry** and the **Top Polyline** of the wall as **Auxiliary Geometry.**
Perform the process on the second wall to complete the finishing of the rebate walls.

End Finish Trim

Select **WORK PLANES > Cancel Work Plane** 

Select **MACHINE > Cut Between 2 Geometries**  set the options as follows;
Set **Safe Rapid Level = 50, Rapid Down Distance = 20, Feed Down Distance = 10** on the first dialogue page.
Set the **Number of Cuts =3, Initial Z Stock = 50, Final XY Stock = 0** and the **Final Z Stock = -1.**

> The only requirement in this example for 3 cuts is that the tool length does not permit a single pass; if a suitable cutter was selected then the Number of Cuts could be set to zero.

On the second dialogue, set the Tool Side options to suit the tool being on the outside of the model depending on the ghost tool directions. As this machining to and from areas outside of the model, there is no real need for lead in/out settings required.
When you have made you choices in all dialogue boxes and clicked the appropriate **[OK]** buttons, pick **Bottom Polyline** of one end as the **Programming Geometry** and the **Top Polyline** of the same end as **Auxiliary Geometry.**

> Alternatively, if your process permits, then using the Saw option to drive a 5 Axis saw cut will work as well, so long as you have used the Edge for Sawing option as described earlier.

Finish handrail profile

Care must be taken when working with Form Cutters, as we will be using on this hand rail form.
In certain circumstance the actual form cutter will not be able to create the entire swept profile due to the twist and turn of the hand rail.
The size of the cutter and the twist of the form created will tend to force the cutter to overcut and dig into the finished part. This is unacceptable and a process plan for the form needs to be created to allow the form cutter to do as much work as it can and save time in the programming process.
This will also allow you to understand which areas require specific 5-axis toolpaths to be generated to blend the form correctly.

Using **3D > SOLID MODEL EXTRACT > 3D Edge Extraction**  acquire the upper and lower profile edges along the flat faces above the form, on both sides, which sweeps along the hand rail edge. Ensure that the ghost tools of each pair of polylines are pointing in the same direction.

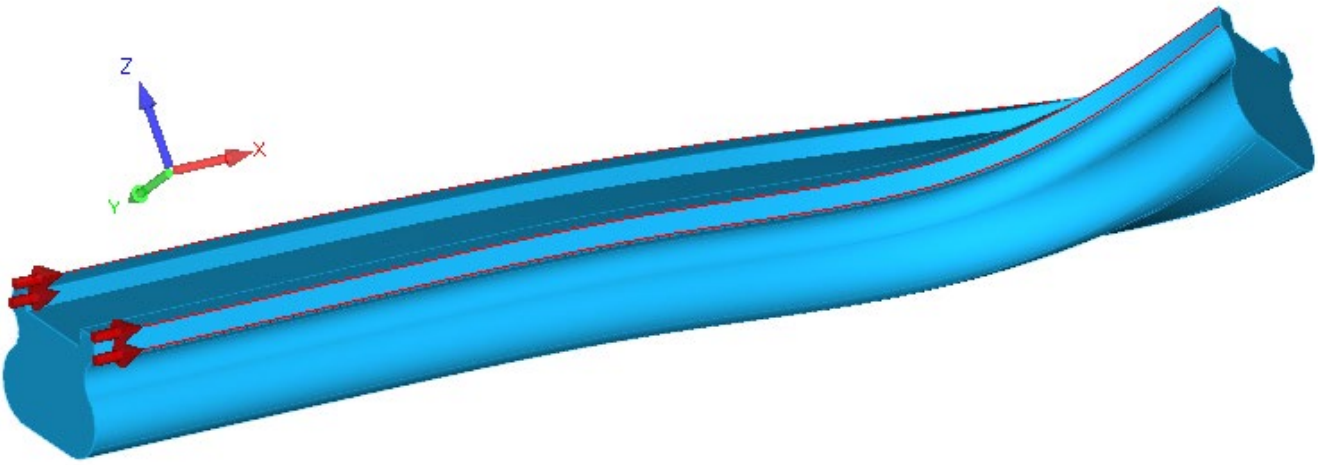


Figure 129 - Extracting the outside polyline options

Select **MACHINE > Cut Between 2 Geometries**  set the options as follows;
Set **Safe Rapid Level = 50, Rapid Down Distance = 20, Feed Down Distance = 10** on the first dialogue page, make the **Final XY Stock =0** and the **Final Z Stock = -2.**
On the second dialogue, set the Tool Side options to suit the tool being on the outside of the model depending on the ghost tool directions.

> Note that whilst **Cut Between 2 Geometries** has been described in the last three machining options, the use of **Cut Spline or Polyline** is also an acceptable option, it all depends on the items required to control the toolpath as to which would be the most suitable option.

Use the **Edit** option for the ☑ **Apply Auto Lead In/Out** and make the values as follows

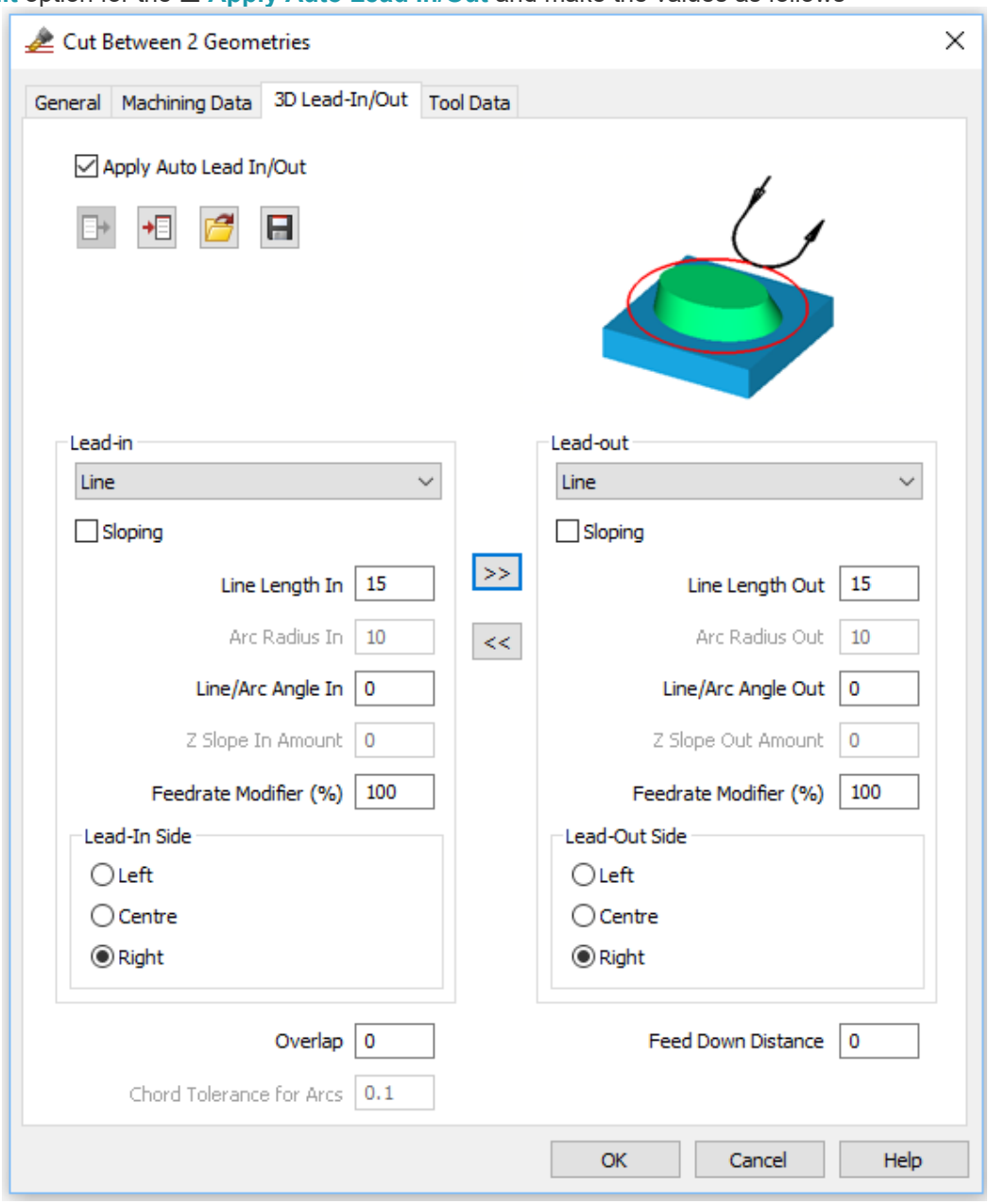


Figure 130 - Lead In/Out modifiers

> The Lead-Out Side will depend on the side the tool is set to work on, if you are on the Right in the Machining set up dialogue then the setting on this dialogue needs to be Right

Use the **lower polyline** of one face as the driving geometry and the **upper polyline** of the same face as the auxiliary item. Repeat the machining cycle on the opposite face.

From the tooling library, select the **Handrail Cutter.**

Using **3D > SOLID MODEL EXTRACT > 3D Edge Extraction**  acquire the two inner face edges of the underside of the hand rail.

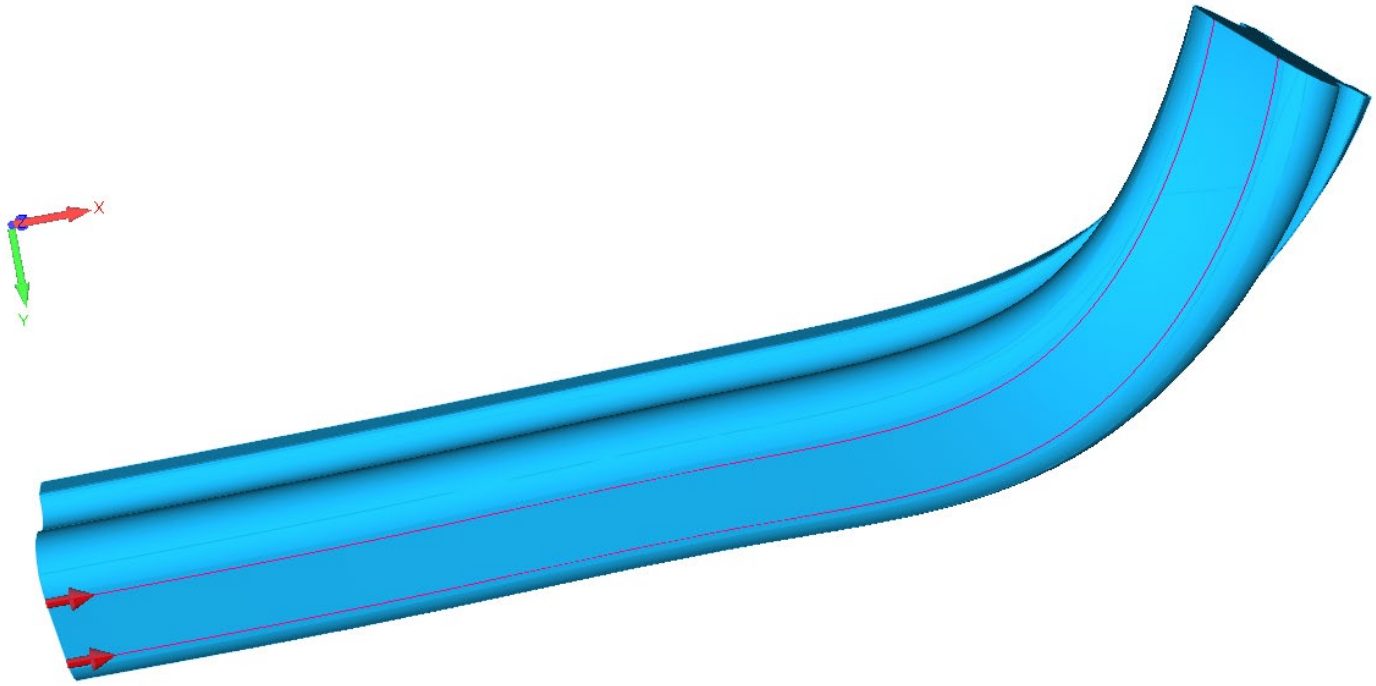


Figure 131 - Underside polylines as the guides to drive the handrail form tool

Use **MACHINE > Cut Spline or Polyline**  Set the **Safe Rapid Level to 50,** the **Rapid Down To** value to **20.** Set the **Depth of Cut** to **0.** Set the **Stock to Be Left** to **15** to use this pass as a Roughing cut on the front side.
Make any alterations to the tool data, then **<LClick> [OK].**

Select the polyline closest to the front side of the hand rail as the item to be cut.

On the **Machining Data tab,** set the options as per this image.

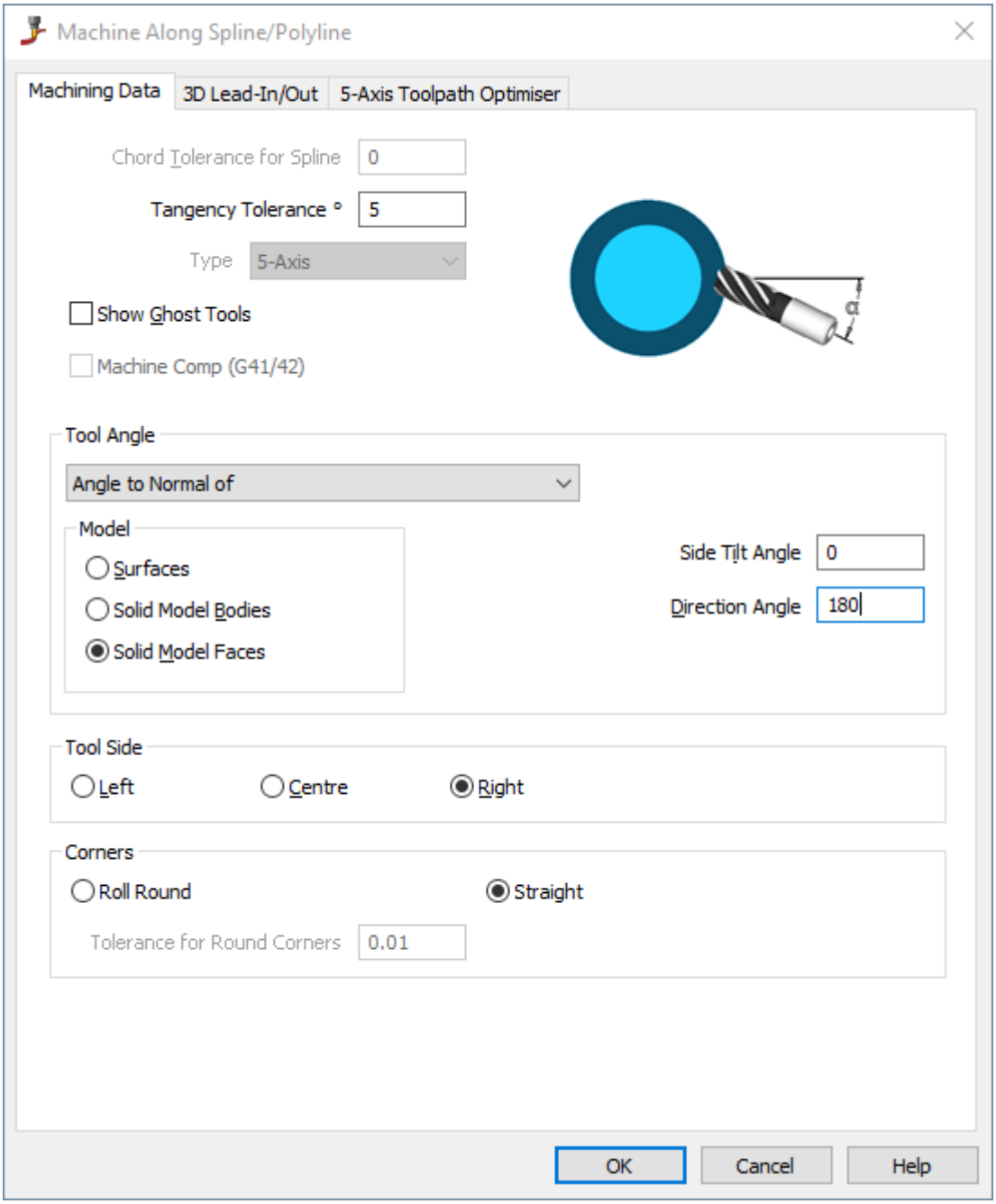


Figure 132 - Using Angle to Normal of allows use of underside options

Set the ☑ **Apply Lead In/Out** options and make the settings as

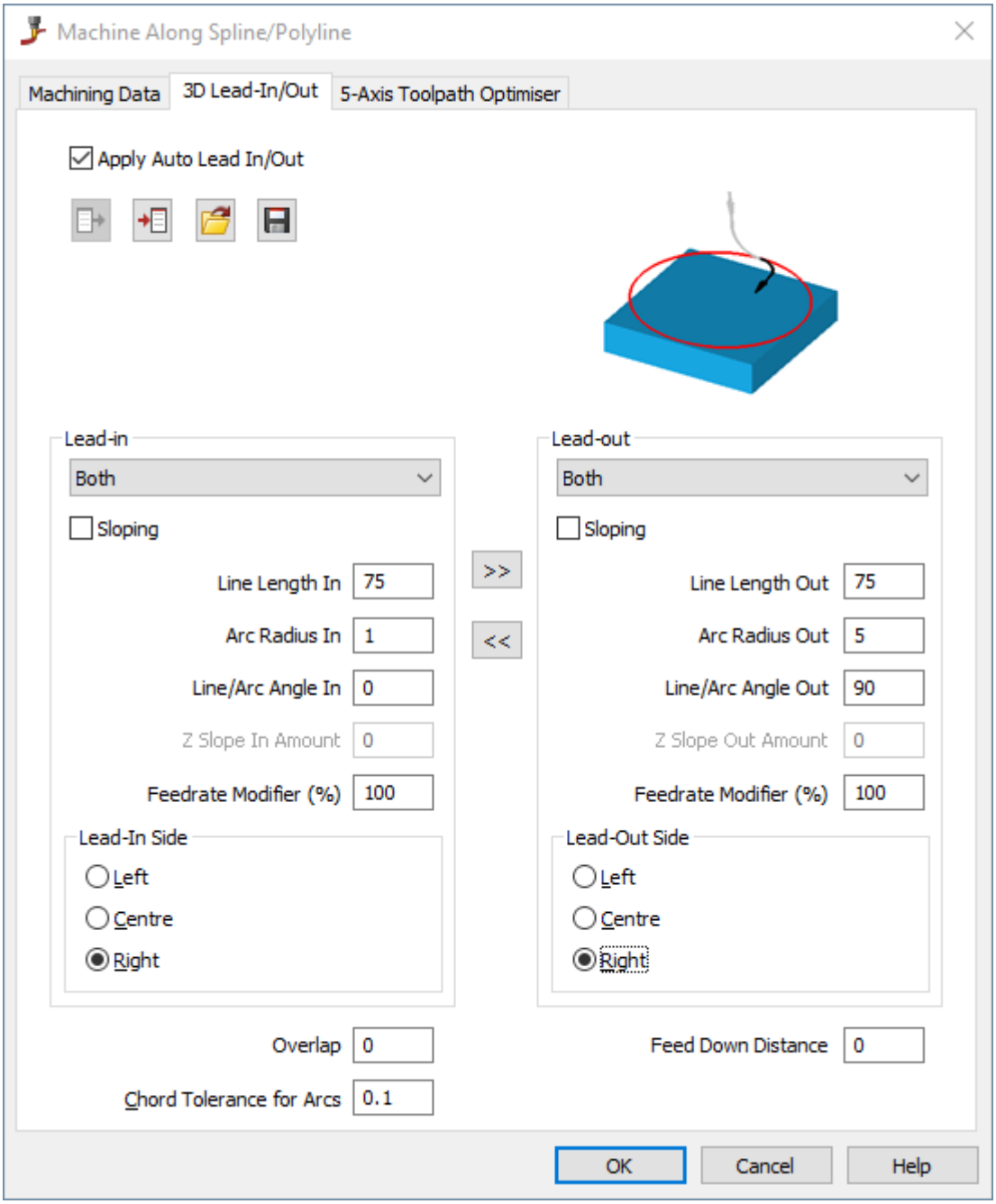


Figure 133 - Lead In/Out options

The 5-Axis Optimiser is not required for this operation

<LClick> [OK] to continue.
For the final face selection, choose the very bottom face of the hand rail.

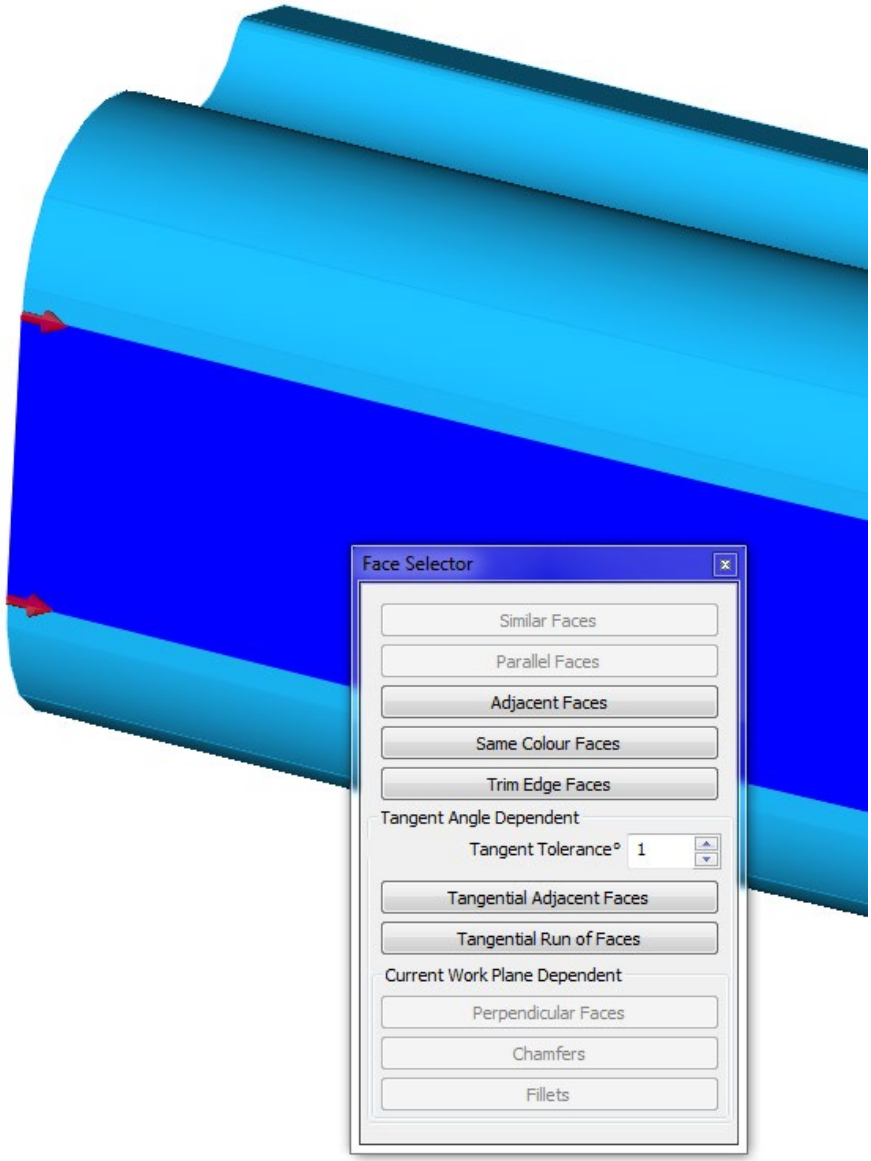


Figure 134 - Setting the underside face as the guide for the cut

<RClick> on this new operation and select the option **Copy Operation**, edit the copy and change the **Stock to Be Left** to **0** to create a **Finish pass.**
Repeat the process for the far side profile, there may be no requirement for a Roughing pass on the far profile so alter the **Stock to Be Left** to **0.**

> ALPHACAM will remember the initial settings of the strategy, not the edited ones.

Choose the far polyline as the driver. Select the bottom face of the hand rail.

> At this point it may prove beneficial to run a simulation for the part to see how the two Finishing cutters interact with each other and if the Handrail Cutter is gouging.
>
> If there are visible issues, you would need to edit the toolpaths or alter the polylines to enhance the machining to produce a more qualified solution.

As you can see from this example there are several errors with trying to run the form cutter along the entire handrail. These are the typical scenarios you will encounter in real world manufacturing.

The type of modification will require one of the following options:

- Break and trim the polylines then update the tool paths.
- Break and delete the tool paths to the required positions and alter the Lead In/out.

Both are viable options, the only concern being that modifying the tool paths and not the underlying geometries could lead to problems if the tool paths are subsequently updated due to new feeds/speeds etc. and they may return to the original length of the polyline.
It may prove slightly more difficult be adjusting the primary geometry can lead to more stable results.

Trim the two underside polylines at approximately this location, then update the Lead In/Out to suit and update the tool paths.

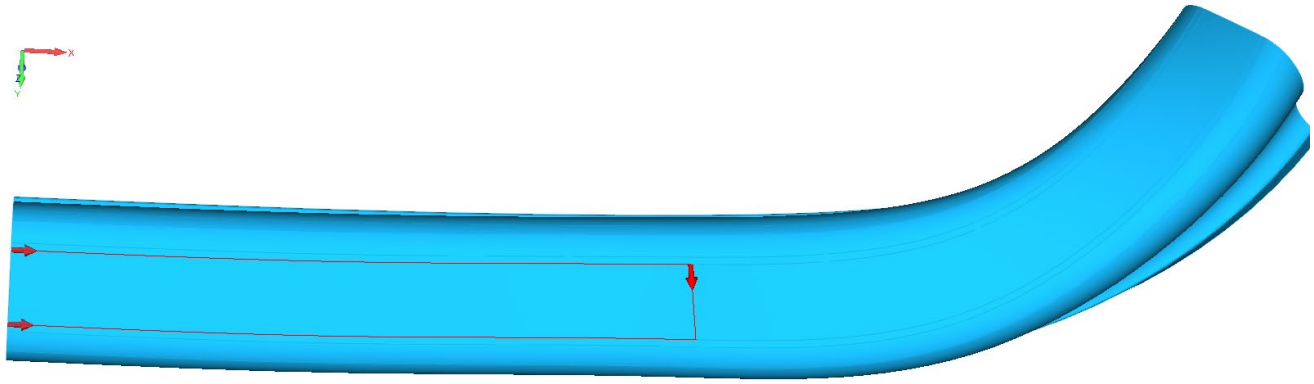


Figure 135 - Approximate location to trim the polylines to

Finish blending curved profile

In a situation like this hand rail where the Form cutter will gouge the part, it is necessary to employ some of the more versatile options available in ALPHACAM via the **Tool Axis Conversion** settings as described earlier in this manual.
In this final section, we will look at practical applications of the Tool Axis Conversion process.
To place the correct style of machining in the remaining areas of the hand rail, we need to create a new work plane to the front side using the **WORK PLANES > Normal to View**  option, this will create a work plane based on exactly how you are viewing the model at the moment you select the command.

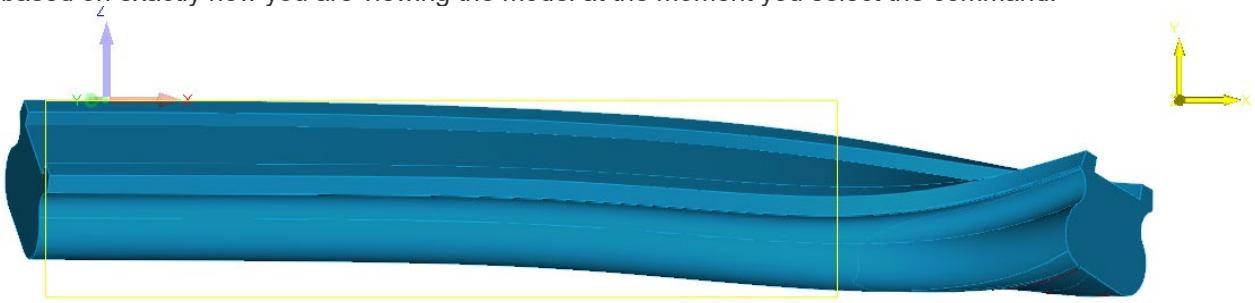


Figure 136 - Work Plane generated on the side using Normal to View

On this new work plane, extract all the side profile faces (remember the top most face has already been finished so does not need selecting) to the work plane using

3D > SOLID MODEL EXTRACT > Projected Face Outlines to Work Plane  .
Edit the resulting geometries to create a single boundary element.

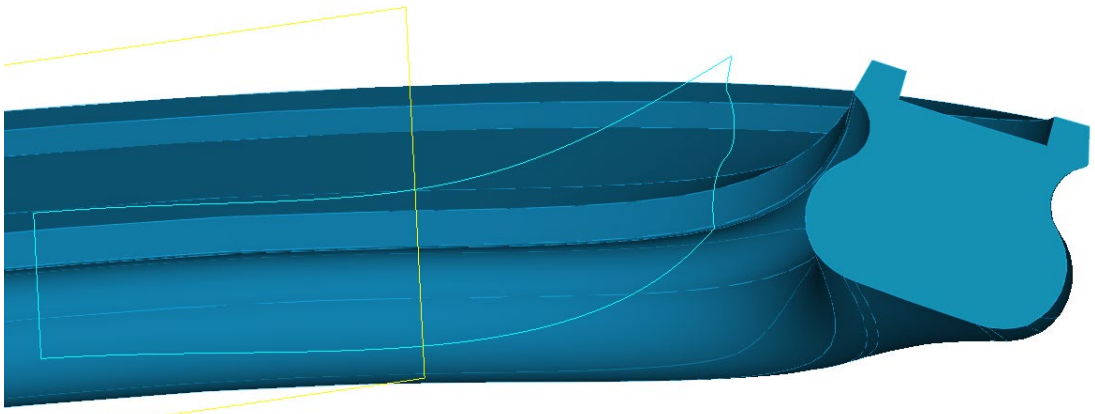


Figure 137 - Extracted and manipulated geometry to create a suitable boundary

You should have a similar result to the image above.
The face extraction process has created a boundary to contain the machining.
Ensure that the Tool Side is on Centre to allow machining to include the boundary.
Feature Extraction will have set the tool side but it will be incorrect for the processing we need.

From the Tool Library, select the **Ball End - 6mm + holder.**

Select **MACHINE > 3D Machining** 

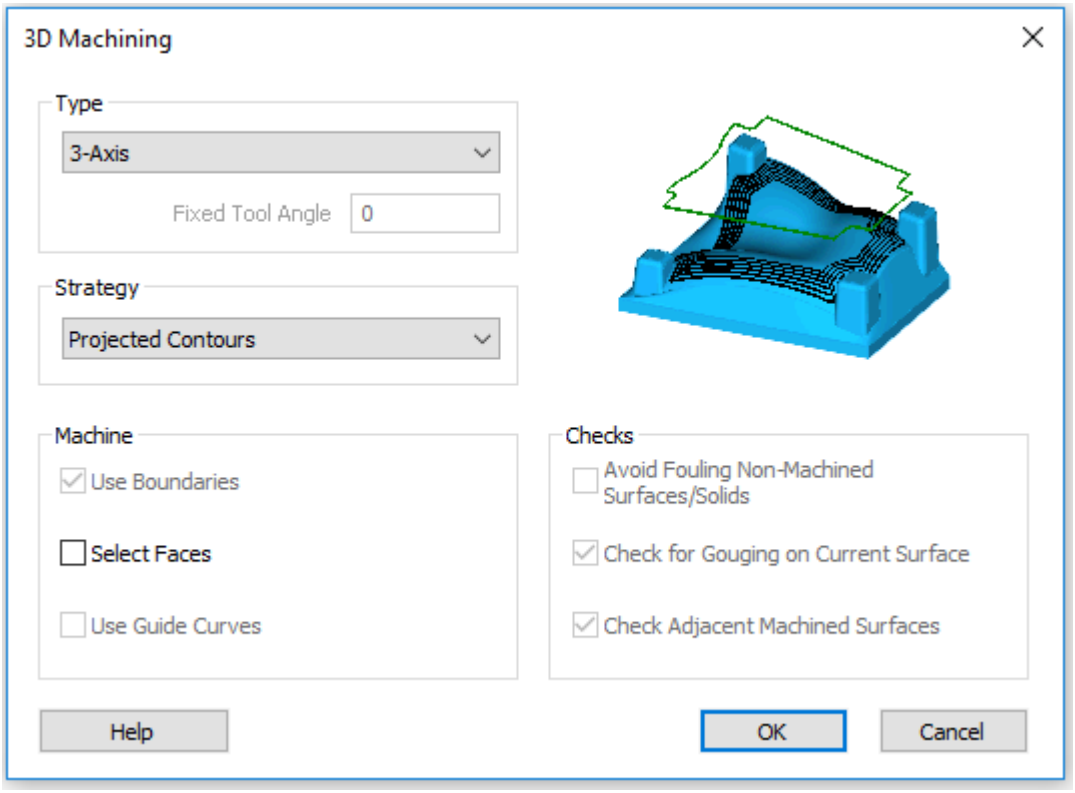


Figure 138 - 3D Machining strategy selection

Make the options **3-Axis and Projected Contours.**

> It is very rare to set a Solid Machining cycle to a direct 5 Axis option due to the control points used for the Tilt and Twist actions. It is better to apply a good working 3 Axis option and then manipulate to suit the 5 Axis motion required.

Select the outer boundary created from the part faces when prompted.
Set both **Stock** options to **5mm** and the **Safe Rapid** and **Rapid Down** to options to **100mm.**
Set a suitable Width of cut to finish the part, this can be altered later.
Select the four main faces of the hand rail used to generate the boundary when prompted to create the tool path.
To alter the 3-Axis tool path to the 5-Axis item we wish to use, we will need to create a reference line that the conversion process can use.

Use **GEOMETRY > 3D Polyline**  from **X280, Y-165, Z0** to **X280, Y-165, Z-80** to create this axis line.

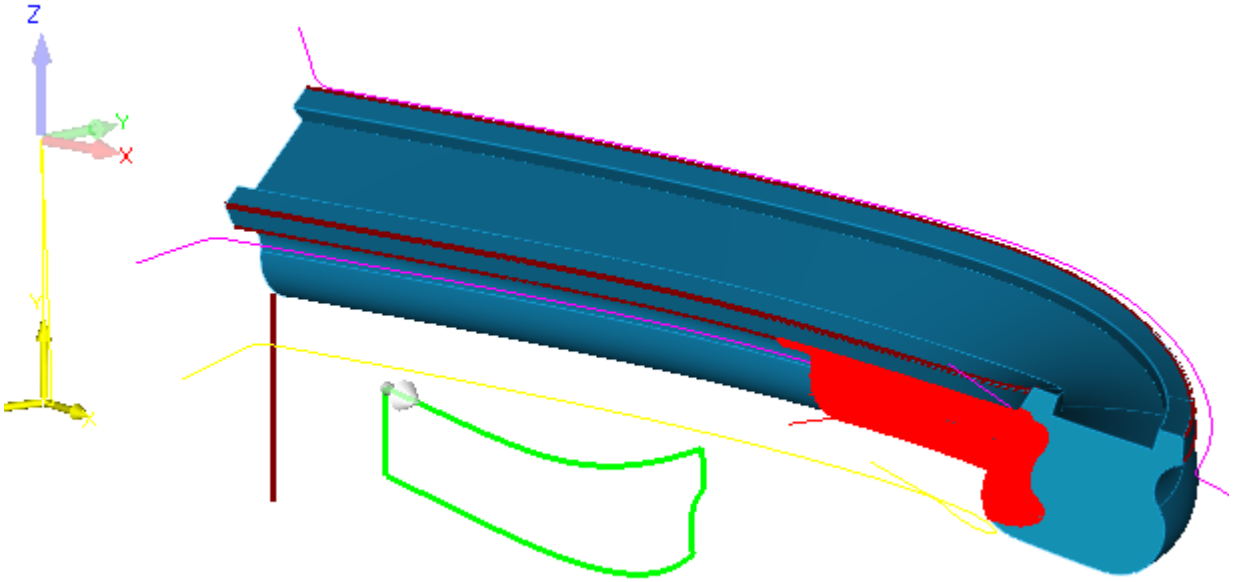


Figure 139 - Geometry boundary initial applied tool path and extra polyline

In the **Operations Project Manager** page, locate the **Projected Contours** tool path previously created and **<RClick>** on the cycle.
Select the **Tool Axis Conversion** option.

Make the settings match this dialogue box.

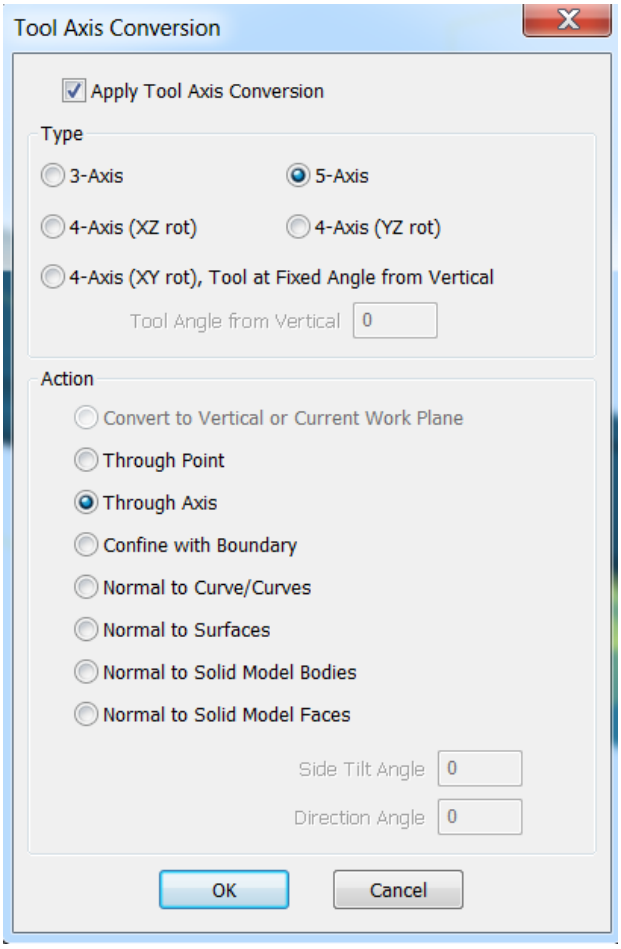


Figure 140 - Tool axis conversion options

When prompted for the **Axis of Revolution,** select the previous Polyline.
To allow the correct direction of conversion you will be asked if the Axis lies on the **Tool side** of the part, it does so click **[Yes].**

To create the Finish tool path for this set of faces, right click on the new tool path and choose the **Copy Operation** option.
Then right click on the copied tool path and select **Edit.**
Alter both **Stock** options to 0 to create the finishing tool path.

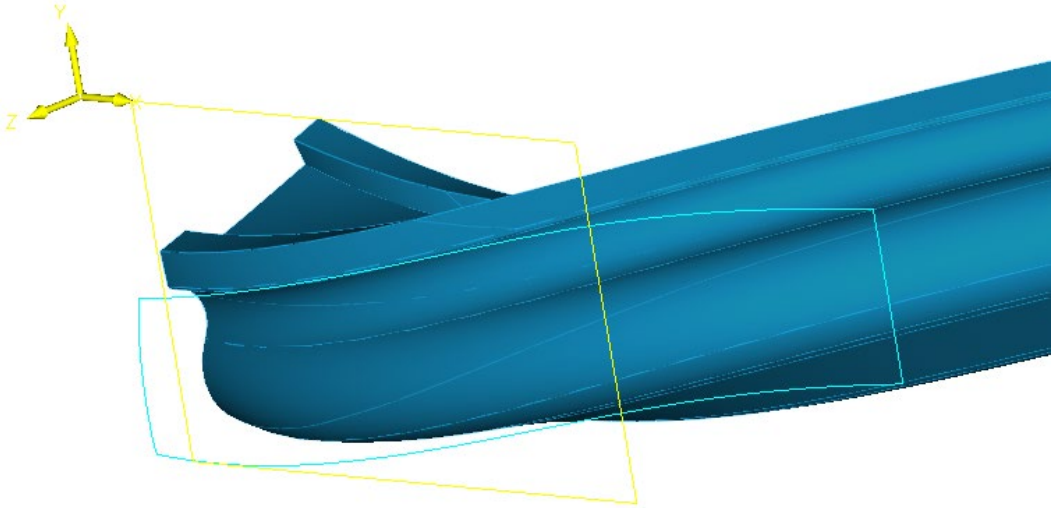


Figure 141 – Work Plane and suitable boundary on the reverse side

Using the Work Plane created for the Rear roughing process, extract the edges of the main faces for the rear of the curve.
If the previously created Rear Angled work plane does not give the best aspect to select the required faces for machining, create a new work plane using the Normal to View option as for the first side.

> **NOTE**
>
> If you must use the Normal to View option, you will need to manipulate the resulting work plane using the **Parallel to Current Plane** option to move to the correct side of the model. Work planes created in the Normal to View manner will always have their Z origin matching the Global origin.

Use the same tool and machining process to create a finish tool path on this view of the part. Depending on the amount of material left from the Form Cutter pass, there may not be a need for a roughing version of the Projected Contour tool path.
Use the Tool Axis Conversion option once more but this time, select the **⊙ Through Point option** and when prompted select the bottom of the previously created polyline.
When requested, select the **[No] option f**or the point side.

> Using the Automatic Rapid Manager will assist in creating correct lift of the head to transition to differing areas of the part during machining.

There may be a requirement to add manual paths to force correct simulation as a safety feature.

Version amendments

V	Amendment Description	A	Software Version	Amended Date
11	Minor text formatting alterations.	1	2020.1	11/10/2019
11	Template altered to Hexagon branding	0	2020.0	15/03/2019

HEXAGON
MANUFACTURING INTELLIGENCE

ALPHACAM 2020.1
Standard 5 Axis


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