

ALPHACAM 2020.1 STANDARD 5 AXIS





Copyright

Copyright © 2019 Hexagon AB and/or its subsidiaries and affiliates. All rights reserved.

Any copyright or other intellectual property right of whatever nature which subsists or may subsist in the presentation and/or content of the programs (including without limitation its look, feel, visual or other non-literal elements) remains the property of Hexagon AB and/or its subsidiaries and affiliates or its licensor(s) absolutely.

No part of this publication may be reproduced, transmitted, transcribed, stored in a retrieval system or translated into any language, in any form or by any means, electronic, mechanical, optical, chemical, manual or otherwise, without the express written permission of Hexagon AB and/or its subsidiaries and affiliates.

Unauthorised reproduction or distribution of these programs or any part thereof is unlawful and may result in civil or criminal penalties.

Windows is a trademark of Microsoft Corporation.

All product names mentioned in this publication, and not listed above, are acknowledged as the trademarks of the respective manufacturers and producers of such products.

Hexagon AB and/or its subsidiaries and affiliates makes no representations or warranties with respect to the contents hereof and specifically disclaim any implied warranties of satisfactory quality or fitness for any particular purpose. Further, Hexagon AB and/or its subsidiaries and affiliates reserves the right to revise this publication and to make changes in the contents hereof without obligation to notify any person of such changes or revisions.

The information contained within this document is subject to change without notice and does not represent a commitment on the part of the vendor. The software described in this document is furnished under a licence agreement and may be used or copied only in accordance with the terms of the agreement.



Contents

Copyright	i
Conventions used in this manual	vii
Recommended Operating Systems and Hardware for ALPHACAM	viii
Supported Operating Systems	viii
ALPHACAM Minimum Specifications	viii
Hexagon Customer Portal	ix
ALPHACAM esupport	x
Introduction	1
Objective	1
Tool Axis Conversion	2
Convert 3D work plane tool path to Vertical	3
Convert 2 Axis tool path in work plane to Vertical	4
4 Axis XZ or YZ Rotation	6
Through Axis	6
Parallel tool paths	6
Helical Z tool paths in work plane	7
5 Axis	8
XZ Rotation Confined with Boundary	8
Normal to Curve or Curves	9
XY Rotation Tool at Fixed Tilt Angle	10
Project through Point	11
Project through Axis	12
Confine with Boundary	14
Project through Guide Curves	15
Cut With side of tool	20
Practical example Chair Leg	21
Roughing	22
General	23
Levels and Cuts	24



ALPHACAM 2020.1 Standard 5 Axis

Machining Data	25
Tool Data	26
Machining the Small Rectangular Angled Face	27
Setting up the Work Plane	27
Machining the Small Rectangle	27
General	28
Lead-In/Out Links	29
Machining Data	
Rest Finish	31
Tool Data	32
Machining the Two Large Rectangular areas	
Setting up the Work Plane	33
Machining the Large Rectangles	33
Machining the Small triangle area	34
Setting up the Work Plane	34
Machining the Small Triangle	34
Finish large flat side wall	35
Drill Holes	
Profiling the Leg	37
Cut Between 2 Geometries theory.	40
Ghost Tools	40
Line Length	42
Front Curve	44
General	44
Machining Data	45
3D Lead-In/Out	46
5-Axis Toolpath Optimiser	47
Tool Data	48
Back Curve	50
General	50
Machining Data	51
5-Axis Toolpath Optimiser	53



ALPHACAM 2020.1 Standard 5 Axis

Tool Data	
Ends	
Small 3D triangular area	57
Create the initial driving toolpath	
Applying the Final Machining	
General	60
Practical example Twisted Handrail	
Roughing	65
Primary Stage	65
General	
Levels and Cuts	67
Machining Data	
Tool Data	
Secondary Stage	70
Rough the Spindle rebate	74
Machining Data	77
3D Lead-in/Out	
Rough the Spindle rebate tops	
End Trimming	
Alternative End Trimming 1	
Alternative End Trimming 2	
Alternative End Trimming 3	
Finishing	
Surface Extraction	
Creating the new floor driving surface	
Rebate floor finishing	
Rebate Inside Wall Finishing	
End Finish Trim	
Finish handrail profile	
Finish blending curved profile	



Table of Images

FIGURE 1 - CUSTOMER SUPPORT PORTAL	IX
FIGURE 2 - ESUPPORT PAGE	Х
FIGURE 3 - EXPLANATION OF TOOL AXIS CONVERSION DIALOGUE	2
FIGURE 4 - SPIRAL CYCLE CREATED IN A WORK PLANE	3
FIGURE 5 - SPIRAL CYCLE CONVERTED TO A 3 AXIS TOOLPATH NOT ON A WORK PLANE	3
FIGURE 6 - 2D POCKET CYCLE APPLIED ON A WORK PLANE	4
FIGURE 7 - 2D POCKET CYCLE CONVERTED FROM A WORK PLANE TO VERTICAL	5
FIGURE 8 - PARALLEL STRATEGY AS APPLIED	6
FIGURE 9 - TOOL AXIS CONVERSION AT THE CENTRE OF THE CIRCLE	6
FIGURE 10 - Z ENHANCED UNDERCUTS CONVERTED TO VERTICAL THROUGH AN AXIS	7
FIGURE 11 - INITIAL HORIZONTAL Z TOOLPATH	8
FIGURE 12 - TOOLPATH ALTERED USING A BOUNDARY TO PREVENT COLLISIONS	8
FIGURE 13 - TOOLPATH CONVERSION USING A GUIDE CURVES	9
FIGURE 14 - CONVERTED TOOLPATH ALONG A SPECIFIED ANGLE FROM VERTICAL	10
FIGURE 15 - TOOL PATH CONVERSION THROUGH A POINT ON THE TOOL SIDE	11
FIGURE 16 - THROUGH A POINT ON THE OPPOSITE SIDE FROM THE TOOL	11
FIGURE 17 - INITIAL PARALLEL CYCLE	12
FIGURE 18 - TOOL PATH CONVERTED USING A LINE ON THE OPPOSITE SIDE FROM THE TOOL	13
FIGURE 19 - INITIAL HORIZONTAL Z CONTOURS TOOL PATH COLLIDING WITH THE PART	14
FIGURE 20 - CONVERTED TOOL PATH USING A BOUNDARY TO PREVENT COLLISIONS	14
FIGURE 21 - BEFORE CONVERSION.	14
FIGURE 22 - BEFORE CONVERSION. FIGURE 22 - BEFORE CONVERSION SIDE AND END SHOTS	15
FIGURE 22 - BEFORE CONVERSION SIDE AND END SHOTS FIGURE 23 - CONVERTED TOOL PATH USING TWO GUIDE CURVES	
FIGURE 23 - CONVERTED TOOL PATH USING TWO GUIDE CURVES	17
FIGURE 24 - CUT SPLINE OR POLYLINE OPTIONS USING ULTIMATE ALPHACAM	18
FIGURE 23 - CUT SPLINE OR FOLTLINE OF HON AFPLIED TO A FORM TOOL	19
FIGURE 26 - EXAMPLE FILE FOR CUT SPLINE OR POLYLINE	20
FIGURE 27 - SAMPLE CHAIR LEG SOLID MODEL	21
FIGURE 28 - SUPPORT BLOCK SETTINGS	21
FIGURE 29 - 3D MACHINING CYCLE SELECTION DIALOGUE	22
FIGURE 30 - Z CONTOUR ROUGHING GENERAL TAB.	23
FIGURE 31 - Z CONTOUR ROUGHING LEVELS AND CUTS TAB	24
FIGURE 32 - Z CONTOUR ROUGHING MACHINING DATA TAB	25
FIGURE 33 - Z CONTOUR ROUGHING TOOL DATA TAB	26
FIGURE 34 - 3D MACHINING CYCLE SELECTION DIALOGUE	27
FIGURE 35 - FLAT AREA OFFSET GENERAL TAB	28
FIGURE 36 - FLAT AREA OFFSET LEAD-IN/OUT LINKS TAB	29
FIGURE 37 - FLAT AREA OFFSET MACHINING DATA TAB	30
FIGURE 38 - FLAT AREA OFFSET REST FINISH TAB	31
FIGURE 39 - FLAT AREA OFFSET TOOL DATA TAB	32
FIGURE 40 - TOP EDGE FOR EXTRACTION	34
FIGURE 41 - EDGE TO EXTRACT	35
FIGURE 42 - HOLES TO DRILL	36
FIGURE 43 - EDGES FOR MACHINING EXTRACTED	37
FIGURE 44 - EDIT 3D POLYLINE DIALOGUE OPTIONS	38
FIGURE 45 - SETTING THE EXTEND OPTIONS TO ALL FOUR POLYLINES SIMULTANEOUSLY	38
FIGURE 46 - 3D POLYLINES EXTENDED TO SUIT	39
FIGURE 47 - CUT BETWEEN 2 GEOMETRIES THEORY	40
FIGURE 48 - HOW POINT MATCHING WORKS WHEN LINES ARE SET CORRECTLY	40
FIGURE 49 - WHEN GHOST TOOLS ARE OPPOSITE TO EACH OTHER	41
FIGURE 50 - POINT MATCHING ON LINES OF EQUAL LENGTH	42
FIGURE 51 - POINT MATCHING ON UNEQUAL LINES	42



FIGURE 52 - ENSURING THAT THE POLYLINES ALL ARE WORKING IN THE SAME DIRECTION	43
FIGURE 53 - CUT BETWEEN 2 GEOMETRIES GENERAL TAB	44
FIGURE 54 - CUT BETWEEN 2 GEOMETRIES MACHINING DATA TAB	45
FIGURE 55 - CUT BETWEEN 2 GEOMETRIES 3D LEAD IN/OUT TAB	46
FIGURE 56 - 5 AXIS OPTIMISER TAB	47
FIGURE 57 - CUT BETWEEN 2 GEOMETRIES TOOL DATA TAB	
	48
FIGURE 58 - APPLIED TOOLPATH	49
FIGURE 59 - CUT BETWEEN 2 GEOMETRIES GENERAL TAB	50
FIGURE 60 - CUT BETWEEN 2 GEOMETRIES MACHINING DATA TAB	51
FIGURE 61 - CUT BETWEEN 2 GEOMETRIES 3D LEAD IN/OUT TAB	52
FIGURE 62 - 5 AXIS OPTIMISER TAB	53
FIGURE 63 - CUT BETWEEN 2 GEOMETRIES TOOL DATA TAB	54
FIGURE 64 - SECOND SIDE APPLIED TOOL PATH	55
FIGURE 65 - END FACES EXTRACTED	56
FIGURE 66 - SMALL TRIANGLE SECTION	
	57
FIGURE 67 - EXTRACTED BOUNDARY FOR MACHINING THE SMALL AREA	57
FIGURE 68 - PRIMARY GUIDE 2D POCKET CYCLE	58
FIGURE 69 - 3D MACHINING CYCLE SELECTION DIALOGUE	59
FIGURE 70 - DRIVE CURVES GENERAL TAB	60
FIGURE 71 - DRIVE CURVES USING TOOL PATHS	60
FIGURE 72 - COMPLETED PART SEEN IN SIMULATION	61
FIGURE 73 - SAMPLE HANDRAIL FILE	62
FIGURE 74 - USING THE ASSOCIATE FOR AUTO-UPDATE OPTION	62
FIGURE 75 - SETTING UP THE SUPPORT BLOCK INFORMATION	63
FIGURE 76 - HANDRAIL WITH MATERIAL AND SUPPORT BLOCK	64
FIGURE 77 - 3D MACHINING CYCLE SELECTION DIALOGUE	65
FIGURE 78 - Z CONTOUR ROUGHING GENERAL TAB	66
FIGURE 79 - Z CONTOUR ROUGHING LEVELS AND CUTS TAB	67
FIGURE 80 - Z CONTOUR ROUGHING MACHINING DATA TAB	68
FIGURE 81 - Z CONTOUR ROUGHING TOOL DATA TAB	69
FIGURE 82 - THIRD BOUNDARY SHAPE ON FRONT FACE WORK PLANE	70
FIGURE 83 - USE THE AUTO UPDATE OPTION	71
FIGURE 84 – LEVELS AND CUTS USING SPECIFIC Z DEPTHS	72
FIGURE 85 - CREATED GUIDE LINE	73
FIGURE 86 - EDIT 3D POLYLINE OPTIONS	
	74
FIGURE 87 - EXTENDING ALL EXTRACTED LINES	74
FIGURE 88 - POLYLINES SET UP CORRECTLY FOR MACHINING	75
FIGURE 89 - RENAMING GEOMETRY ELEMENTS FOR EASE OF IDENTIFICATION	75
FIGURE 90 - CUT BETWEEN 2 GEOMETRIES GENERAL TAB	76
FIGURE 91 - CUT BETWEEN 2 GEOMETRIES MACHINING DATA TAB	77
FIGURE 92 - CUT BETWEEN 2 GEOMETRIES 3D LEAD-IN/OUT TAB	78
FIGURE 93 - CUT BETWEEN 2 GEOMETRIES OPTIMISER	79
FIGURE 94 - CUT BETWEEN 2 GEOMETRIES TOOL DATA TAB	80
FIGURE 95 - APPLIED TOOL PATHS	81
FIGURE 96 - CUT SPLINE OR POLYLINE MACHINING DATA	82
FIGURE 97 - AXIS TO ROTATE ON	83
FIGURE 98 - TOOL ANGLE DROP DOWN OPTIONS	83
FIGURE 99 - TOOL ANGLE GUIDE OPTIONS	84
FIGURE 100 - CUT SPLINE OR POLYLINE GENERAL TAB	85
FIGURE 101 - CUT SPLINE OR POLYLINE TOOL DATA TAB	86
FIGURE 102 - SELECT THE UPPER POLYLINE AS THE GUIDE	86
FIGURE 103 - CUT SPLINE OR POLYLINE MACHINING DATA TAB	87
FIGURE 104 - CUT SPLINE OR POLYLINE 3D LEAD-IN/OUT TAB	88





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
- \square

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.

Current User							Support Status: St
	Home						Site Map Log C
Search	Licensing	Support	Community	Reports	Downloads	Account Details	Help
Recent Items							
	Software Licenses	111 Technical Support Cases	Customer Community Forums	Technical Support Metrics	K Software	🔏 Login Details	How to guides
	S Manual Revocation	Support Charter	🗳 FAQ	Professional Services	Training Materials	Company Details	Customer Notification
		Remote Support	Knowledge Base	O ⁱ Maintenance Schedule			💐 Portal Help Videos
				👻 License History			License Documentation
				💱 Support Status			



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.

esupport.verosoftware.com/alphacam/			
	HEXAGON = Q Search		
	ALPHACAM Alphacam - Blog Forums Knowledge Base Videos Training Sub-Groups v More		
	Forum Rules		
	The Alphacam Customer Community Forum membership is formed of knowledgeable Alphacam resellers and users from around the world with diverse backgrounds and experiences. Alphacam's Customer Community Forums is a place to join conversations, collaborate with others, and share valuable information you won't find anywhere else. We ask that you please follow these simple posting guidelines.		
	Rules of the eCommunity		
	The #1 rule is to discuss Alphacam technology in a constructive way.		
	Alphacam's technical support, bugs, development tasks, or reseller support should be taken directly to your Alphacam reseller. Alternately log a support case here		
	While debating and discussion is fine, we will not tolerate rudeness, profanity, insulting posts or personal attacks.		
	You agree that the administrative staff of the Customer Community Forums reserve the right to remove, edit, move or close any thread, private message, forum, social group, or any other aspect of the site for any reason we see fit. You agree that the administrative staff has the right disable, ban, delete, or modify user accounts for any reason.		
Figure 2 - esupport page	9		

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.

Tool Axis Conversion 💽		
Apply Tool Axis Conversion		
Туре		
O 3-Axis	Image: S-Axis	
🔘 4-Axis (XZ rot)	4-Axis (YZ rot)	
4-Axis (XY rot), Tool at Fixed Angle from Vertical		
Tool Angle fr	rom Vertical 0	
Action		
Convert to Vertical or Current Work Plane		
O Through Point		
Through Axis		
Confine with Boundary		
Normal to Curve/Curves		
Normal to Surfaces		
Normal to Solid Model Bodies		
Normal to Solid Model Faces		
	Side Tilt Angle 0	
	Direction Angle	
ОК	Cancel	

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.

 $\mathbf{\nabla}$



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.

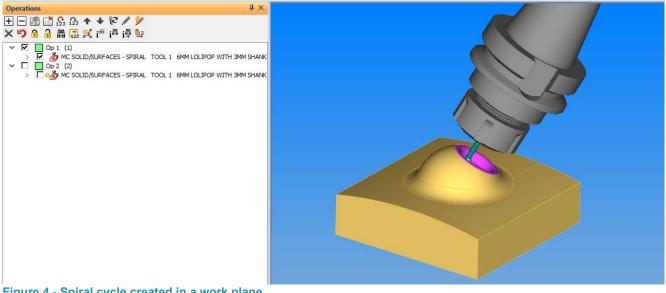


Figure 4 - Spiral cycle created in a work plane

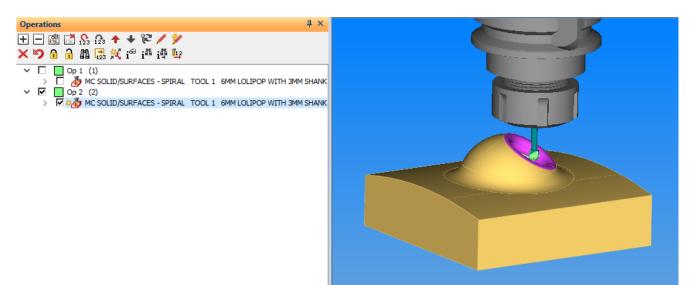


Figure 5 - Spiral cycle converted to a 3 axis toolpath not on a work plane

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.

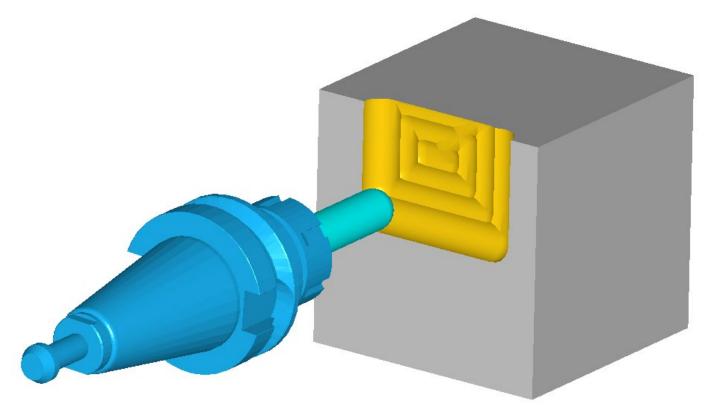


Figure 6 - 2D Pocket Cycle applied on a Work Plane



ALPHACAM 2020.1 Standard 5 Axis

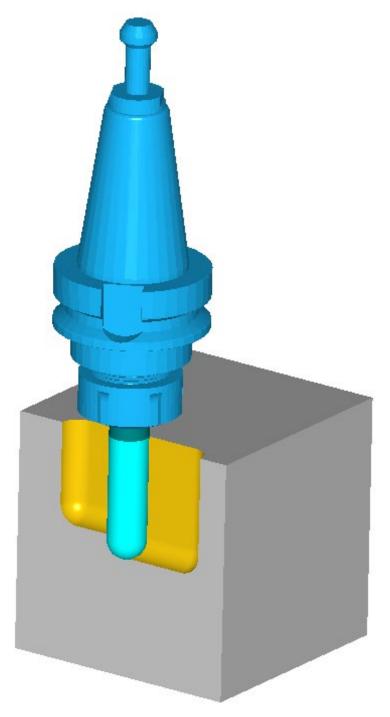


Figure 7 - 2D pocket cycle converted from a work plane to Vertical



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

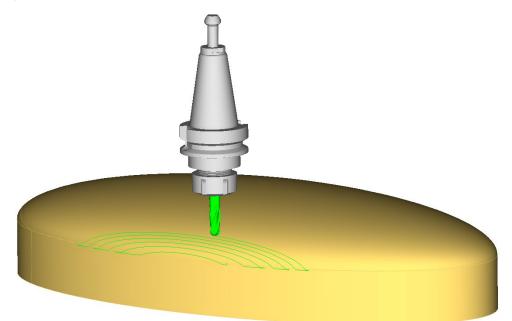


Figure 8 - Parallel Strategy as applied

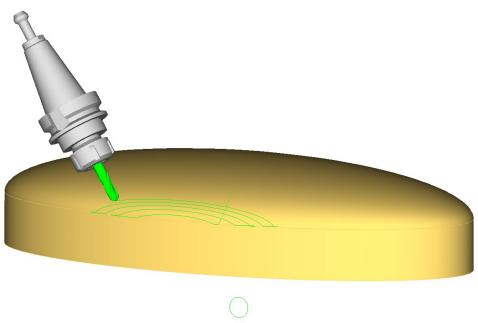
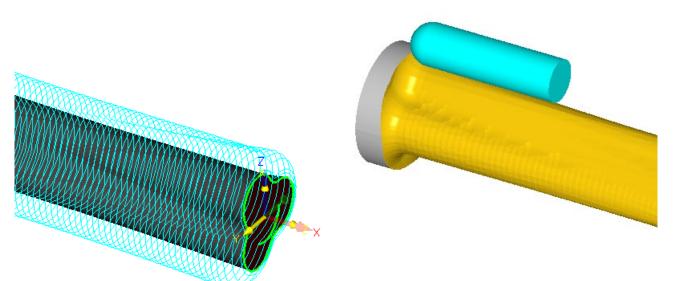


Figure 9 - Tool Axis Conversion at the centre of the circle



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.

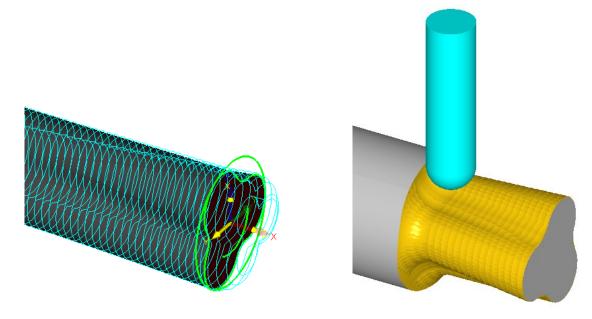


Figure 10 - Z Enhanced Undercuts converted to Vertical through an axis



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.

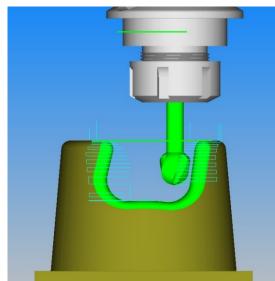
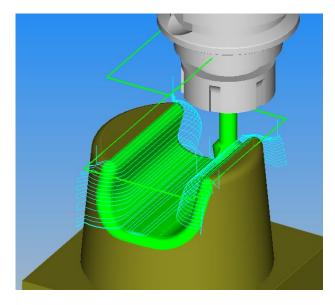


Figure 11 - Initial Horizontal Z toolpath



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.

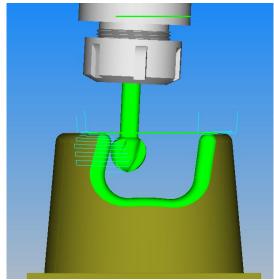
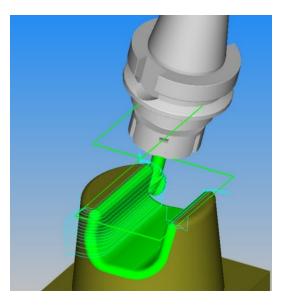


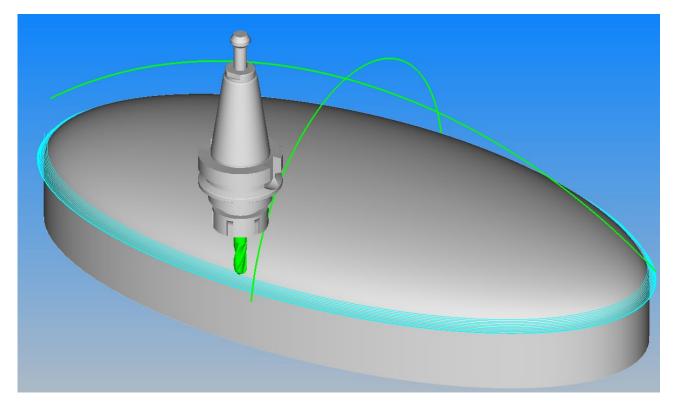
Figure 12 - Toolpath altered using a boundary to prevent collisions





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.

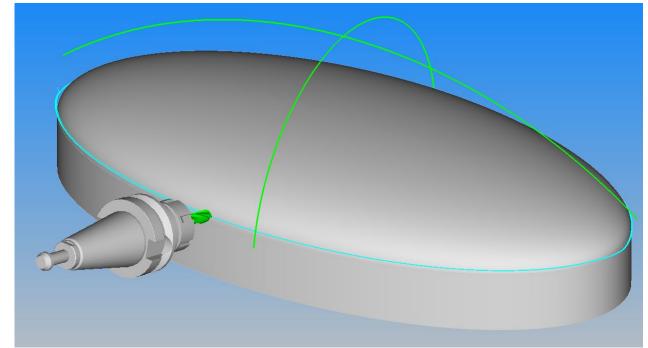


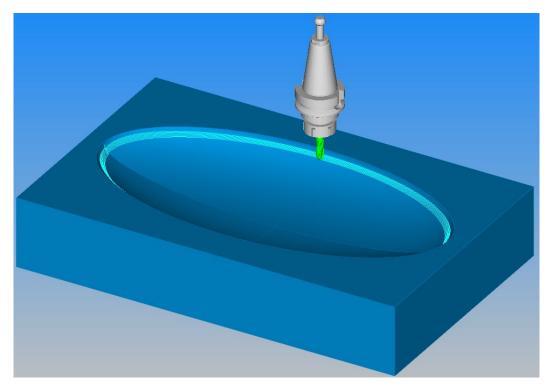
Figure 13 - Toolpath conversion using a guide curves



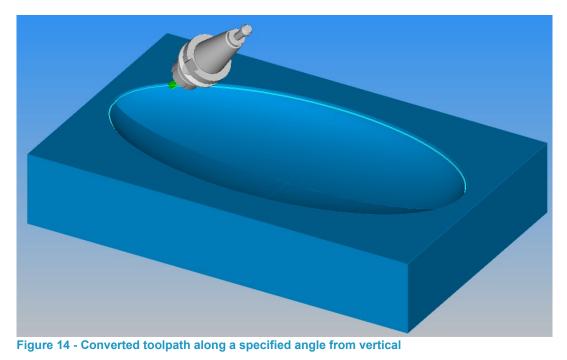
XY Rotation Tool at Fixed Tilt Angle

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.





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 axis 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.

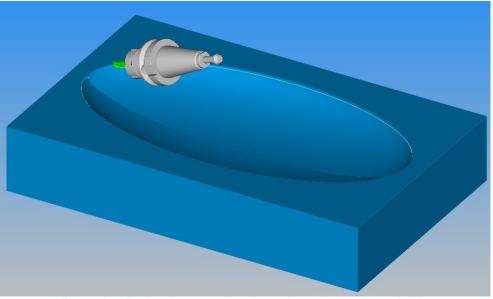
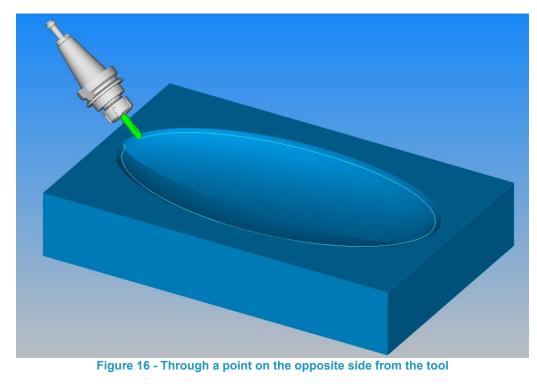


Figure 15 - Tool path conversion through a point on the tool side

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





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.

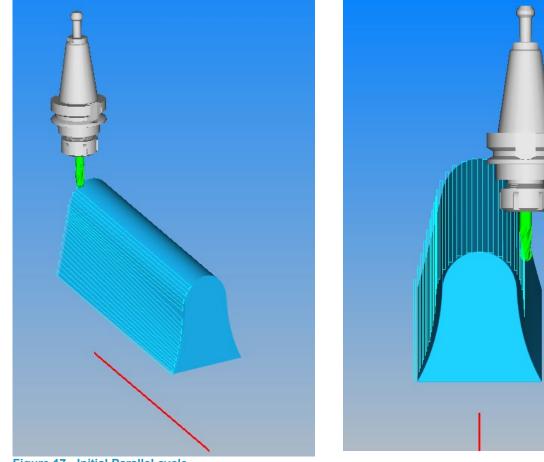


Figure 17 - Initial Parallel cycle

After Conversion tool adjusted to point at polyline axis.



ALPHACAM 2020.1 Standard 5 Axis

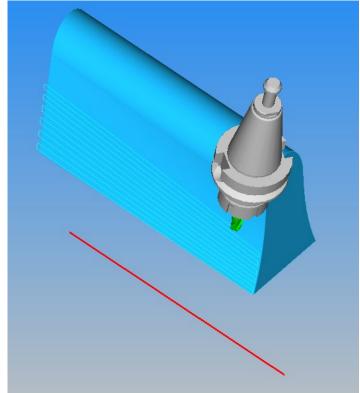
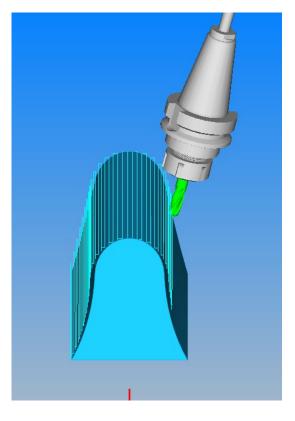


Figure 18 - Tool path converted using a line on the opposite side from the tool





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.

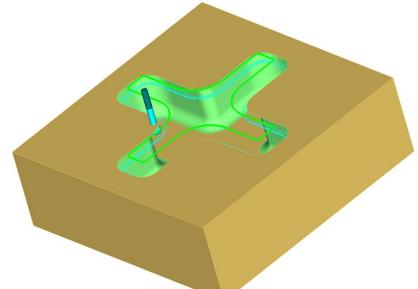


Figure 19 - Initial Horizontal Z Contours tool path colliding with the part

After 5 axis conversion with boundary.

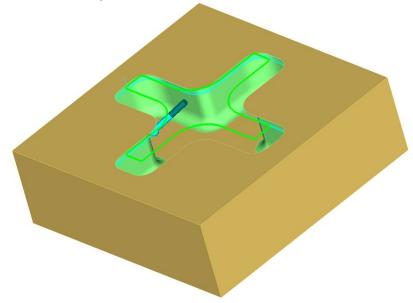


Figure 20 - Converted tool path using a boundary to prevent collisions



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.

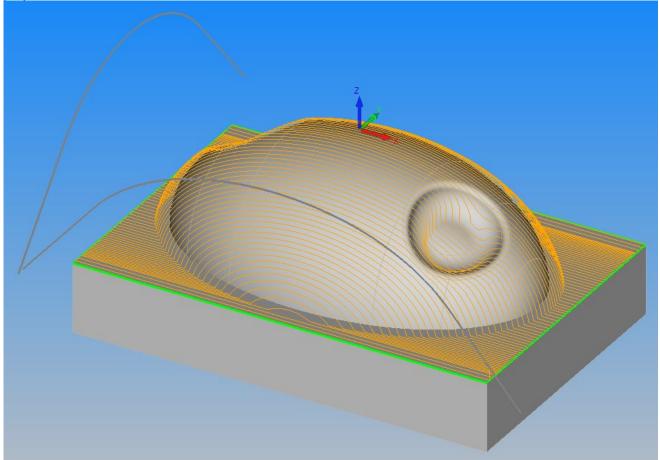


Figure 21 - Before Conversion.



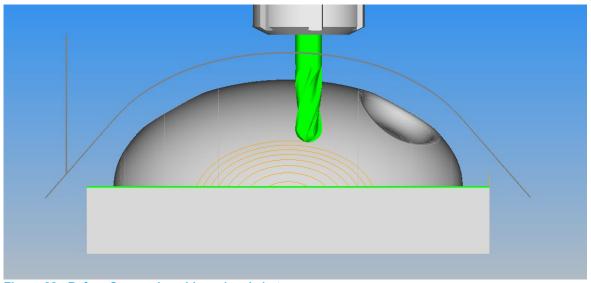
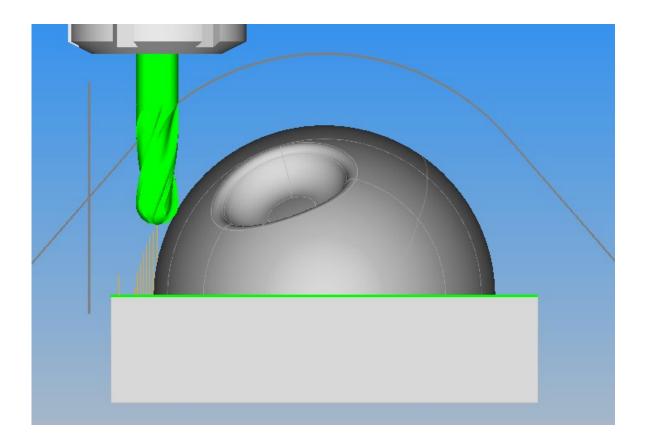


Figure 22 - Before Conversion side and end shots





After Conversion.

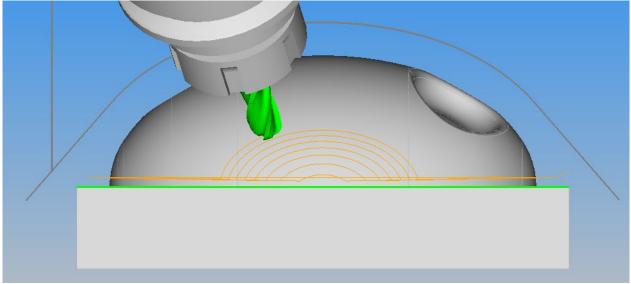
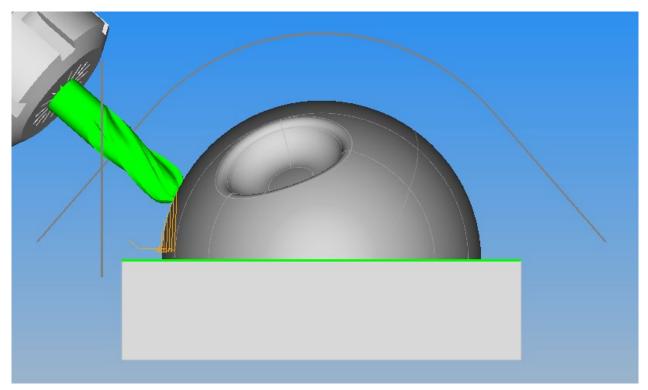


Figure 23 - Converted tool path using two guide curves





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.

Machine Along Spline/Polyline	×
Machining Data 3D Lead-In/Out	
Chord Tolerance for Spline 0	
Tangency Tolerance ° 5	
Type 5-Axis V	
Show Ghost Tools	
Machine Comp (G41/42)	
Tool Angle	
Parallel to	
Model	
◯ Surfaces	
O Solid Model Bodies	
Solid Model Faces	
Tool Side	
Centre Right	
OK Cancel Help	
Un Cancer Hep	

Figure 24 - Cut Spline or Polyline options using Ultimate ALPHACAM



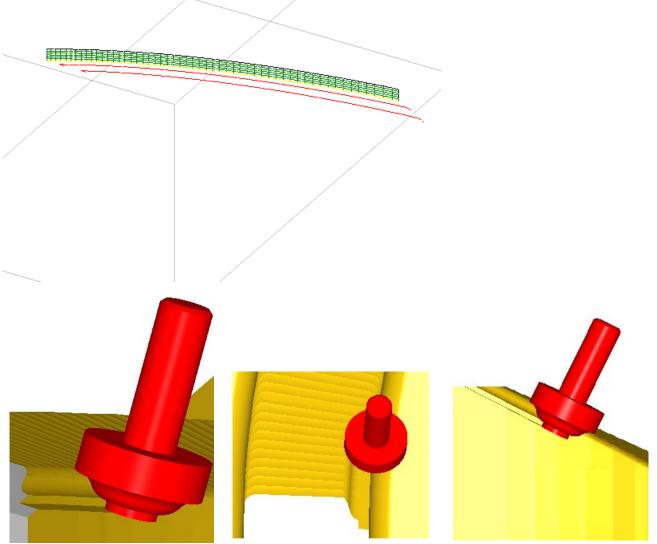


Figure 25 - Cut Spline or Polyline option applied to a form tool

- 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 **D** 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

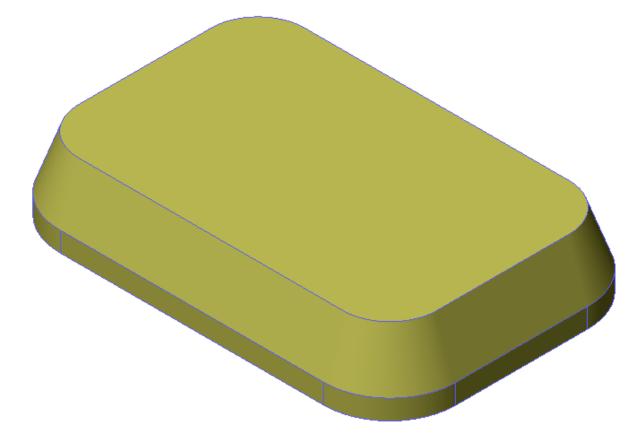


Figure 26 - Example file for Cut Spline or Polyline

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





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,

Define Clamp X	
Name Support Block	
Number 1	
Solid Type	
O Revolved Extruded	
Top Z -75	
Bottom Z -150	
Save with Machine	
Moveable	
Can Move in	
X Y Z Has Parent	
\sim	
Confine Within Parent	
Set Positioning Check Geometry	
Set Containment Check Geometry	
Can Pop Up	
Pop Up Distance 0	
Feed Rate 0	
Clamp Notes	
Select Texture	
No Texture 🗸	
Set as default	
OK Cancel	
Figure 28 - Support block set	tings

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.

3D Machining	×
Туре	
3-Axis \checkmark	
Fixed Tool Angle 0	50
Strategy	
Z Contour Roughing \sim	
Along Intersection Constant Cusp Drive Curves Flat Area Offset Helical Z Horizontal Z Contours Machine Surface with Tool Side Parallel Parameter Lines Projected Contours Radial Rest Machining Spiral Z (Enhanced Undercuts) Z Contour Roughing Help	Checks Avoid Fouling Non-Machined Surfaces/Solids Offset O Check for Gouging on Current Surface Check Adjacent Machined Surfaces OK Cancel

Figure 29 - 3D Machining cycle selection dialogue



General

Set the options as shown below.

neral	Levels and Cuts	Machining Data	Tool Data			
Tool		IP No.				
Ċ	nange Tool					
Туре						
	Contour			Cut [Direction	0
_	linear Spiral			Smoothing	g Radius	0
	Waveform			Back Pass	Retract	0
	Close Open Pockets Jse Max Feed on B					
-Mill T	'ype Climb	⊖ Cor	ventional	Opt	timised	
	ligh Speed Corneri			Minimun	n Radius	0
	erial Selection					
	Current Material			Use <u>B</u> oundaries		
0	Select Material(s) Auto-update mater operations	ial from previous		Detect Undercut		
O	Jse Geometry as M	laterial				



Levels and Cuts

Set the options as shown below.

1 Lowella and Oct.							
eral Levels and Cuts	Machini	ng Data	Tool Dat	a			
Z Levels (or Distance fi	rom Plane	:)			Ø		
Safe Rapid	Level	20					
Feed Down Dis	stance	8					
Materi	ial Top	0					
Material B	Bottom	-66.7307					
Max Depth p	er Cut	10					
Upper	Z Limit	0		Intermediat	e Slices (Dept	th = 0 for r	none) —
Lower	Z Limit	-66.7307		M	ax Depth per	Cut 2	
Auto				I	Percentage F	eed 100)
		nal Z leve	els		0.45 - 7		
Entry		nal Z leve	els				
		nal Z leve	~		e Cutting Too		
Entry		nal Z leve	~	⊡ Centro Igle for Lead-Ir			
Entry		nal Z leve	~	gle for Lead-Ir		ntal) 5	
Entry Approach Type		nal Z leve	~	gle for Lead-Ir	n (0 = Horizor	ntal) 5	
Entry Approach Type		onal Z leve	∽ Slope An	gle for Lead-Ir	n (0 = Horizor	ntal) 5	
Entry Approach Type Cutting Order	Ramp		∽ Slope An	gle for Lead-Ir	n (0 = Horizor	ntal) 5	
Entry Approach Type Cutting Order	Ramp		∽ Slope An	gle for Lead-Ir	n (0 = Horizor	ntal) 5	
Entry Approach Type Cutting Order	Ramp		∽ Slope An	gle for Lead-Ir	n (0 = Horizor	ntal) 5	
Entry Approach Type Cutting Order	Ramp		∽ Slope An	gle for Lead-Ir	n (0 = Horizor	ntal) 5	
Entry Approach Type Cutting Order	Ramp		∽ Slope An	gle for Lead-Ir	n (0 = Horizor	ntal) 5	



Machining Data

Make the options as shown below.

🍜 Z Contour Roughing	×
General Levels and Cuts Machining Data Tool Data	
Machining	
XY Stock to be Left 3	
Z Stock to be Left 3	
Chord Tolerance 0.8	
Adaptive Feeds Width of Cut	- 1
Adaptive Feed Rate Main Cuts 40]
Maximum (%) 100 Clean-Up Cuts 40]
Minimum (%) 50 Intermediate Cuts 40]
Increment (%) 20	
Trochoidal for full-width moves	
Link Method Always Ramp V Optimised	
XY Standoff 1 O Clearance	
Feed When Plunging Avoid Rollover	
OK Cancel Hel	p

Figure 32 - Z Contour Roughing Machining Data tab



Tool Data

💑 Z Contour Roughing		×
General Levels and Cuts Machi	nining Data Tool Data	
Tooling		
Tool Number	1	
Offset Number	1	
Diameter	80	
Spindle Speed	24000	
Down Feed	5000	
Cut Feed	5000	
Coolant		
 None 		
◯ Mist		
◯ Flood		
O Through Tool		
	OK Cancel Help	

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 **ID** (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** J 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.

Machining		
Туре		
3-Axis	\sim	
Fixed Tool Angle 0		
Strategy		
Flat Area Offset	~	
Flat Area Offset Machine	~	Checks
<u> </u>	~	Checks Avoid Fouling <u>N</u> on-Machined Surfaces/Solids
Machine	~	Avoid Fouling Non-Machined
Machine	~	Avoid Fouling Non-Machined Surfaces/Solids



<LClick>[OK] to continue.

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



General

🥭 ЗD М	lachining - Flat Area Of	fset			×
General	Lead-In/Out and Links	Machining Data	Rest Finish	Tool Data	
	Op No. I: FLAT - 12 MM + HOLDE nange Tool	2R			3
Mach	nining XY Stock to be Left Z Stock to be Left Safe Rapid Level	0		ap on Open Elements; Tool Rad x ess Above Surface to Rapid Down to	0
				OK	Cancel

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

🍎 3D M	lachining - Flat Area Of	ifset				×
General	Lead-In/Out and Links	Machining Data	Rest Finish	Tool Data		
Lead	l-In/Out and Links					
	Specify Moves Between C	luts				
	Extension Length	0		3		
	Ramp Length	0	é			
	Arc Radius	0	C			
	Ramp Angle	0				
I	Join Paths with Arcs					
					ок с	ancel
						ancer

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

eneral Lead-In/Out and Links	Machining Data	Rest Finish	Tool Data		
Cut Spacing uses					
Width of Cut					
🔿 Cusp Height					
Width	3				
Cusp	0.5				
Mill Type		Tolerar	ices		
◯ Climb		Chord Tolerance along Cut 0.02			
○ Conventional		Facet Tolerance = Chord 0.25			
Optimised			Tolerance x	,	
Strategy		Cut Be	tween Z Levels		
◯ Contour		🗹 Aut	to		
Linear			Upper Z		
◯ Finish Pass			Lower Z		
Close Open Pockets					
Cut Direction 30		Option	S		
		(Connect Aircut Length 0		
			ОК	Cancel	

Figure 37 - Flat Area Offset Machining data tab

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



Rest Finish

3D M	lachining - Flat Area Of	ffset				>
General	Lead-In/Out and Links	Machining Data	Rest Finish	Tool Data		
Prev	ious Tool					
	Take Account of Previous	Machining				
V 1	Auto					
	Find Tool				Ø	
	110100					
					ОК	Cancel

Figure 38 - Flat Area Offset Rest Finish tab

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



Tool Data

🍎 3D M	lachining - Flat Area Of	fset				×
General	Lead-In/Out and Links	Machining Data	Rest Finish	Tool Data		
Tooli	ing					
	Tool Number	2				
	Offset Number	2				
	Diameter	12				
	Spindle Speed	18000		- 4		
	Down Feed	2000				
	Cut Feed	2000				
Cool	ant					
	None					
0	Mist					
Of	Flood					
01	Through Tool					
					OK	Cancel

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 P** 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.



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, \odot Optimised, \odot Linear, Cut Direction = 90 and \checkmark 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

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

WORK PLANES > Set Work Plane Origin

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

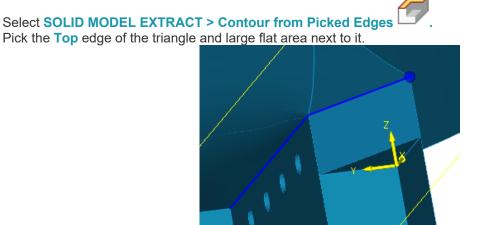


Figure 40 - Top edge for extraction

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 how 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.

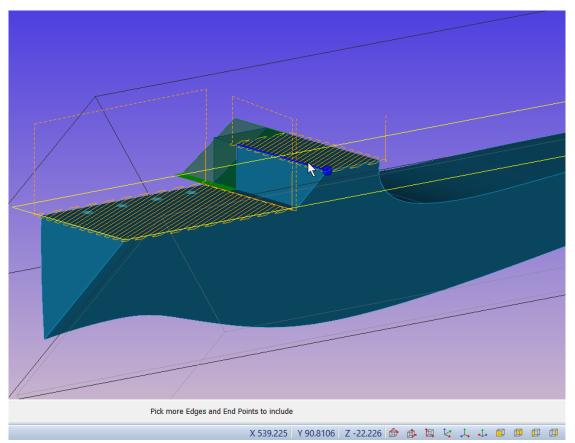


Figure 41 - Edge to Extract

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 O 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.

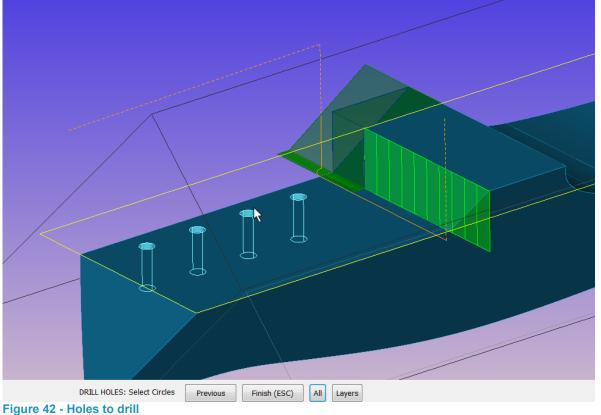
Use Apply Auto Lead In/Out to drive onto and from the part.



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 **O** Drillable holes, **O** Use Current and **O** All bodies. I 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 Select the Attracted hole details on the flat face as the required geometries. Select the extracted hole details on the flat face as the required geometries.





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 > X** 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.

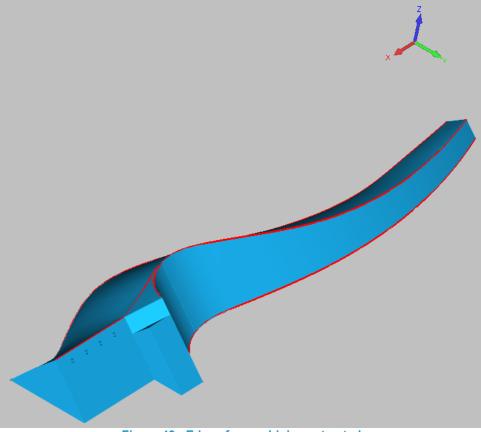


Figure 43 - Edges for machining extracted



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** 4, in the dialogue box

Edit 3D Polyline		(x				
Option							
Change	Insert	O Delete					
© List	Append	Reverse					
Convert to 2	2D Geometry						
◎ Fillet	Extend	C Loop					
OK Cancel							

Figure 44 - Edit 3D Polyline dialogue options

Choose • Extend. <LClick> [OK].

In the next dialogue.

Extend Polyline
Extend
By Distance 20
To 2D Boundaries
To Active WorkPlane
◎ To Surfaces
© To Solids
Polyline Selection
Single Multiple
Extend at
○ Start ○ End
OK Cancel

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

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



ALPHACAM 2020.1 Standard 5 Axis

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.

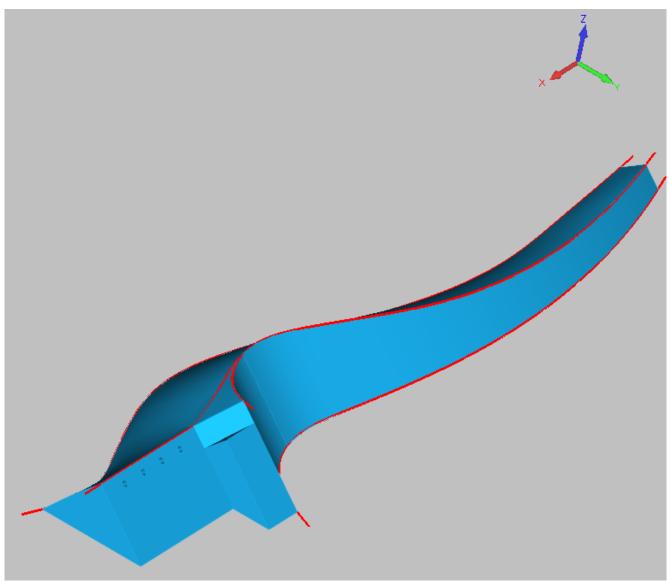


Figure 46 - 3D Polylines extended to suit

<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.

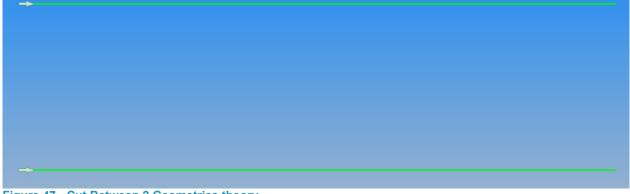


Figure 47 - Cut Between 2 Geometries theory

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.

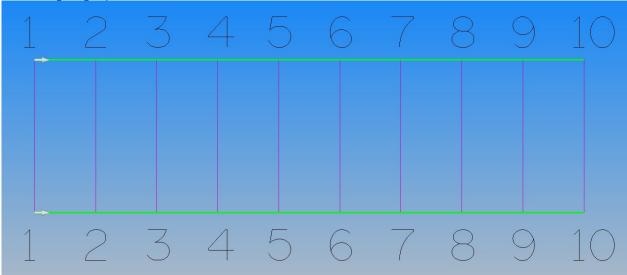


Figure 48 - How point matching works when lines are set correctly

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.

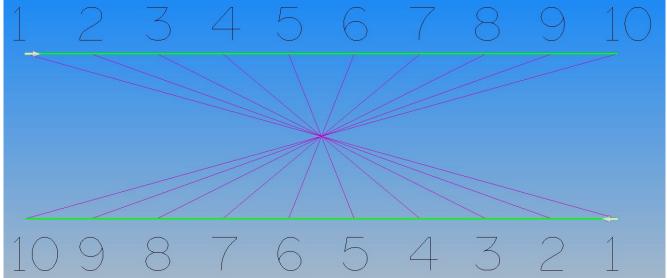


Figure 49 - When ghost tools are opposite to each other

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.

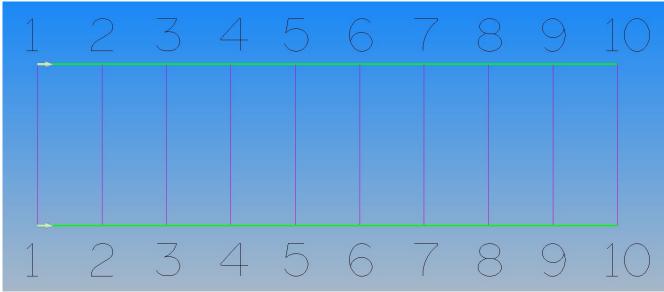


Figure 50 - Point matching on lines of equal length

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.

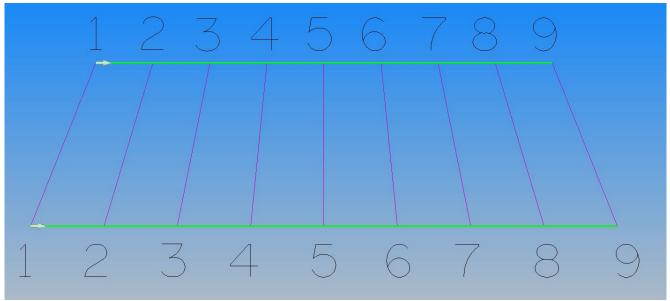


Figure 51 - Point matching on unequal lines

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.



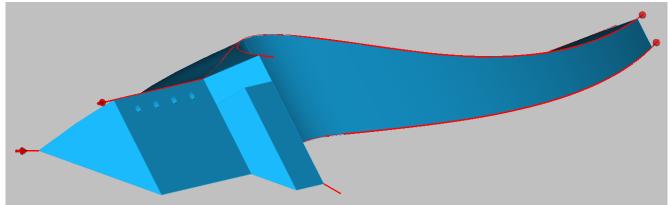


Figure 52 - Ensuring that the polylines all are working in the same direction

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

and select the Flat - 25mm with holder.

Select MACHINE > Cut Between 2 Geometries

General

eral	Machining Data	3D Lead-In/Out	5-Axis Toolpath Optimise	er Tool Data	
Tool					
	O	p No. 3		e	1
Tool	: FLAT - 25MM WIT	'H HOLDER			
					70
Ch	nange Tool				
Mach	nining				
	Safe Rapid	Level 50	Feed	Down Distance	0
	Rapid Down Dist	tance 10		Initial Z Stock	0
	Initial XY :	Stock 0		Final Z Stock	0
	Final XY :	Stock 0			
	Number of	Cuts 1			
XY C	orners				
	Round				
•	Straight				
OL	.oop				
	Loop R	adius 0			

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

差 Cut B	etween 2 Geom	etries			\times
General	Machining Data	3D Lead-In/Out	5-Axis Toolpath Optimiser	Tool Data	
	nining				
	Step L	ength 2.5		2	
	Chord	Error 0.25			
Tool					
	<u>entre</u>				
	Right				
				Consul	u la la
			OK	Cancel	Help

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

差 Cut Between 2 Geometries				\times						
General Machining Data 3D Lead-In/Out 5-	Axis Toolp	oath Optimiser	Tool Data							
Apply Auto Lead In/Out	Apply Auto Lead In/Out									
– Lead-in		Lead-out								
Both 🗸		Both		~						
Sloping		Sloping								
Line Length In 20	>>		Line Length Out	20						
Arc Radius In 10	<<		Arc Radius Out	10						
Line/Arc Angle In 90			Line/Arc Angle Out	90						
Z Slope In Amount 0		2	Slope Out Amount	0						
Feedrate Modifier (%) 100		Fee	edrate Modifier (%)	100						
Lead-In Side		-Lead-Out S	lide							
● <u>L</u> eft		€Left								
○ <u>C</u> entre		○ <u>C</u> entre								
O <u>Rig</u> ht		○ <u>R</u> ight								
Overlap 0		F	eed Down Distance	0						
Chord Tolerance for Arcs 0.1										
		OK	Cancel	Help						

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

Using the \square 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

	Between 2					
neral	Machinin	ig Data	3D Lead-In/Out	5-Axis Toolpath O	ptimiser Tool Data	
	cis Toolpat		er			
	Jse optimi					0
	Copy lead	in/out				
	Copy feed	down dis	tance			
Di	istance to	retract b untwist r				
		antinoti			ø	
Rota	ation					
		Machine	Configuration			
		Туре	Min °	Max °	Free Rotation	
	Х Тоо		~ 0	0		
	Y Nor		v 0	0		
	Z Too		v 0	0		
	Clockwise					
	Cardanic A			0		
	° (Cardanic	Angle	0		

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

eneral	Machining Data	3D Lead-I	n/Out	5-Axis Toolpath Optimiser	Tool Data	
Tool	ing					
	Tool N	lumber 3				
	Offset N	lumber 1			- 11 A	
	Dia	ameter 2	5			
	Spindle	Speed 1	5000			
	Dow	n Feed 5	000			
	Q	t Feed 5	000			,
Cool	ant					
	None					
0	Mist					
0	Flood					
0	Through Tool					
				ОК	Cance	el Help

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.

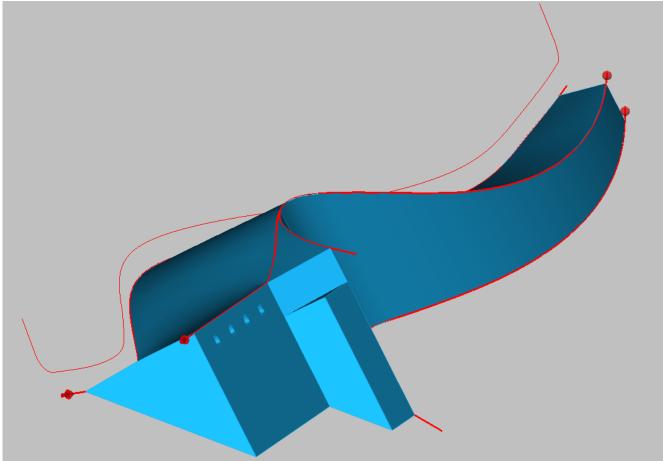


Figure 58 - Applied toolpath



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

差 Cut Between 2 Geometries				×
General Machining Data 3D Le	ad-In/Out 5-Axis	Toolpath Optimiser	Tool Data	
Tool Op No. Tool: FLAT - 25MM WITH HC Qhange Tool	3			
Machining				
Safe Rapid Level	50	Feed D	own Distance	0
Rapid Down Distance	10		Initial Z Stock	0
Initial XY Stock	0		Final Z Stock	0
Final XY Stock	0			
Number of Cuts	1			
XY Corners Roll Round Straight Loop Loop Radius	0			
		OK	Cance	l Help

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

差 Cut B	Between 2 Geom	etries			\times
General	Machining Data	3D Lead-In/Out	5-Axis Toolpath Optimiser	Tool Data	
Mach	nining				
	Step L	ength 2.5		Π	
	Chord	d Error 0.25			
Teel	Side				L I
	_eft		4		
	<u>C</u> entre				
	<u>R</u> ight				
			ОК	Cancel	Help
	0.15.1			Cancer	heip

Figure 60 - Cut Between 2 Geometries Machining Data tab

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



差 Cut Between 2 Geometries	×
General Machining Data 3D Lead-In/Out 5-Axis Too	olpath Optimiser Tool Data
Apply Auto Lead In/Out	
I→ +E C ² I	
Lead-in	Lead-out
Both ~	Both ~
Line Length In 20	Line Length Out 20
Arc Radius In 10 <<	Arc Radius Out 10
Line/Arc Angle In 90	Line/Arc Angle Out 90
Z Slope In Amount	Z Slope Out Amount
Feedrate Modifier (%) 100	Feedrate Modifier (%) 100
Lead-In Side	Lead-Out Side
● Left	●Left
○ <u>C</u> entre	◯ <u>C</u> entre
◯ <u>R</u> ight	◯ <u>R</u> ight
Overlap 0	Feed Down Distance 0
Chord Tolerance for Arcs 0.1	
	OK Cancel Help

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

Using the \square 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

	Between 2					
neral	Machinin	ig Data	3D Lead-In/Out	5-Axis Toolpath O	ptimiser Tool Data	
	cis Toolpat		er			
	Jse optimi					0
	Copy lead	in/out				
	Copy feed	down dis	tance			
Di	istance to	retract b untwist r				
		antinoti			ø	
Rota	ation					
		Machine	Configuration			
		Туре	Min °	Max °	Free Rotation	
	Х Тоо		~ 0	0		
	Y Nor		v 0	0		
	Z Too		v 0	0		
	Clockwise					
	Cardanic A			0		
	° (Cardanic	Angle	0		

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

eneral	Machining Data	3D Lead-In/O	ut 5-Axis Toolpath	Optimiser	Tool Data	
Tool	ing					
	Tool N	lumber 3				
	Offset N	lumber 1				
				1		
	Dia	ameter 25				
	Spindle	Speed 1500)			
	Dow	n Feed 5000				
	Qu	t Feed 5000				
Cool	ant					
	None					
0	Mist					
0	Flood					
0	Through Tool					
				OK	Cancel	Help

Set the tooling information to suit.
</pr



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.

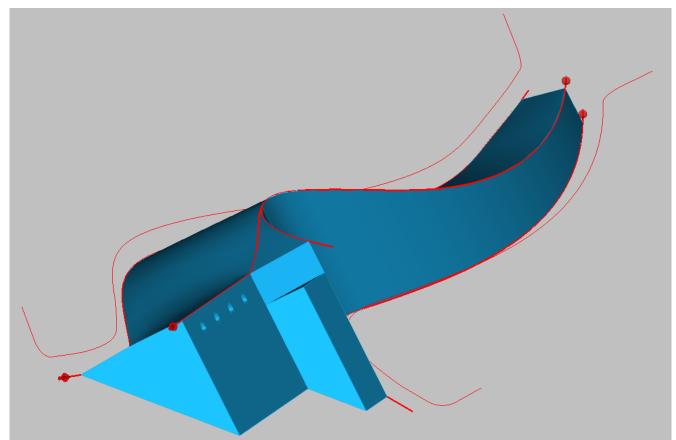


Figure 64 - Second side applied tool path



ALPHACAM 2020.1 Standard 5 Axis

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.

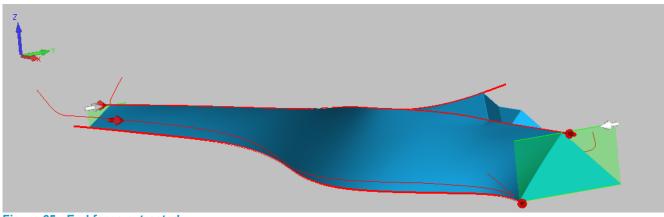


Figure 65 - End faces extracted



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.



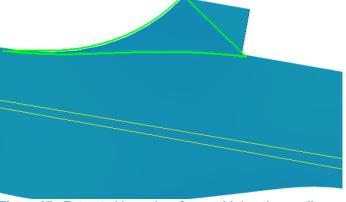


Figure 67 - Extracted boundary for machining the small area



Create the initial driving toolpath.

Use MACHINE > Select Tool ³ 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.

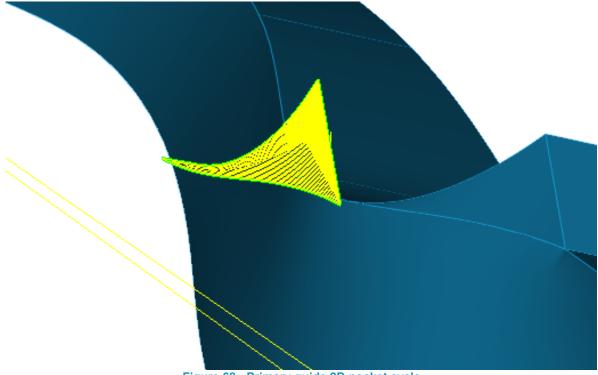


Figure 68 - Primary guide 2D pocket cycle



Applying the Final Machining

Use MACHINE > 3D Machining

D Machining	×
Туре	
3-Axis 🗸	
Fixed Tool Angle 0	
Strategy	
Drive Curves V	
Machine	Checks
Select Faces	□ Avoid Fouling <u>N</u> on-Machined Surfaces/Solids
Use Guide Curves	Offset 0
Boundaries	Check for Gouging on Current Surface
\checkmark	Check Adjacent Machined Surfaces
Help	OK Cancel

Figure 69 - 3D Machining Cycle Selection Dialogue

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



General

💑 3D Machining - Drive Curves	×
General Lead-In/Out and Links Tool Data	
Tool Op No. 2 Tool: BALL END - 6MM + HOLDER Change Tool	
Drive Curves - Use	Tolerances Chord Tolerance along Cut
Chord Tolerance 0.005	Facet Tolerance = Chord Tolerance x 0.22
Pick Drive Curves	Angle Between Tool and Surface Normal 0
Machining	
XY Stock to be Left 0	Safe Rapid Level 50
Z Stock to be Left 0	Thickness Above Surface to Rapid Down to
	OK Cancel Help

Figure 70 - Drive Curves General tab

Select O 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.

Drive Curves - Use	
◯ Geometries	
Tool Paths	
Chord Tolerance	0.005
Pick Drive Curves	✓

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.

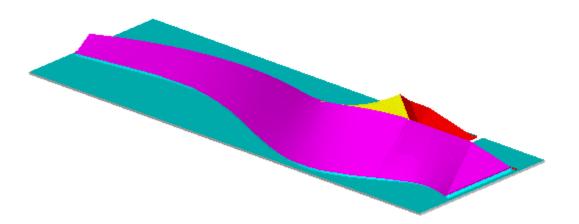


Figure 72 - Completed part seen in simulation

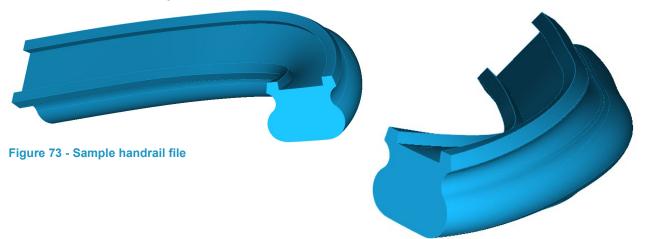
This is the one remaining cycle that after creation Cannot be edited; any errors will require the cycle to be completely re-created.

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



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 navigate 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 ticked.

Auto Set Material					
Material Top Z 10					
Material Bottom Z -150					
Material XY Stock 10					
Ignore Tool Paths					
Delete Existing					
Manually Select					
✓ Associate for auto-update					
Select Texture					
No Texture					
Set as default					
OK Cancel					

Figure 74 - Using the Associate for auto-update option



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

	(Ci	٠	
Using MACHINE > Clamps/Fixtures > Define Clamps/Fixtures <	7		-
Only of the second seco			

Select the rectangle as the shape and set the options as follows,

Define Clamp				
Name Support Block Number 1 Solid Type				
© Revolved				
Top Z -150 Bottom Z -300				
Save with Machine Moveable Can Move in X V Z Has Parent				
Confine Within Parent Set Positioning Check Geometry Set Containment Check Geometry Can Pop Up				
Pop Up Distance 0 Clamp Notes				
Select Texture No Texture				
Set as default				
OK Cancel				

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.



ALPHACAM 2020.1 Standard 5 Axis

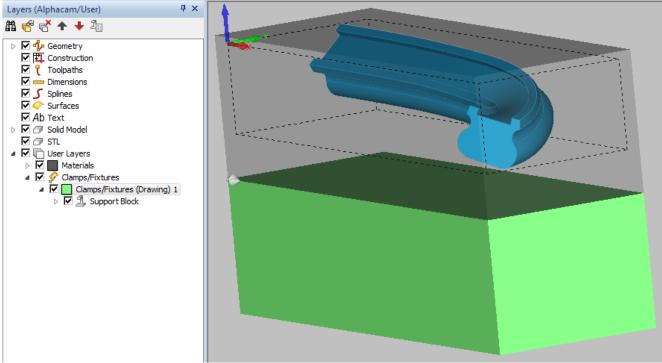


Figure 76 - Handrail with material and support block

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

Туре	
3-Axis	✓
Fixed Tool Angle 0	
Strategy	
Z Contour Roughing	
Machine	Checks
Select Faces	Avoid Fouling <u>N</u> on-Machined Surfaces/Solids
Use Guide Curves	Offset 0
Boundaries	Check for Gouging on Current Surface

Figure 77 - 3D Machining Cycle selection dialogue

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



General

ata	
ĥ	
Cut Direction	0
Smoothing Radius	0
_	
back Pass Retract	0
nal Optimised	
Minimum Radius	0
Use <u>B</u> oundaries	
Detect Undercut Stock	
Pick Material(s)	
	Smoothing Radius Back Pass Retract

Figure 78 - Z Contour Roughing General tab



Levels and Cuts

Make the options as shown below.

🍎 Z Co	ntour Roughing						\times
General	Levels and Cuts	Machi	ning Data	Tool D	ata		
ZLe	vels (or Distance fro	om Plar	ne)			0	
	Safe Rapid	Level	20				
	Feed Down Dis	tance	5				
	Materia	al Top	10				
	Material Bo	ottom	-150				
	Max Depth pe	r Cut	10				
	Upper Z	Limit	10		Intermediate Slices		one)
	Lower Z	Limit	-120			oth per Cut 1	
	Auto				Percen	ntage Feed 100	
Entr	y Approach Type	Helix		∽ Slope	Centre Cutti Angle for Lead-In (0 = 1		
					Maximum Plu	inge Depth 5	
Cutt	ing Order						
	by Zone		⊖ Ьу Ц	.evel			
Ē	Prismatic Geometry						
					ОК	Cancel	Help

Figure 79 - Z Contour Roughing Levels and Cuts tab



Machining Data

Make the options as shown below.

🍎 Z Coi	ntour Roughing			×
General	Levels and Cuts	Machining Data Tool Data		
Mach	nining XY Stoc	k to be Left 3	Î	
		k to be Left 3 d Tolerance 0.25		I
	otive Feeds Adaptive Feed Rate		Width of Cut Main Cuts	12.5
	Maximun		Clean-Up Cuts	12.5
	Minimun	n (%) 50	Intermediate Cuts	12.5
Links	Frochoidal for full-w		Link Type	
	LINK Method		 Optimised Clearance 	
F	eed When Plunging		Avoid Rollover	
			OK Cancel	Help

Figure 80 - Z Contour Roughing Machining Data tab



Tool Data

Set the tooling options as required.

eneral	Levels and Cuts	Machining Data	a Tool Data			
Tooli	ing					
	Tool Nun	mber 2				
	Offset Nun	mber 2			b - 1	
	Diam	eter 25				
	Spindle Sp	beed 15000				
	Down F	Feed 5000				
	Cut F	Feed 5000				
Cool	ant					
	None					
	Mist					
O	Flood					
0	Through Tool				•	

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 geometries so that the final boundary looks like this. , edit the resulting two

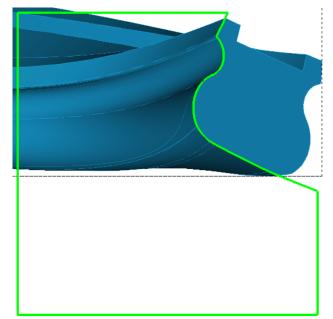


Figure 82 - Third boundary shape on front face work plane

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.





Left Boundary area

🍎 Z Co	ntour Roughing			×
General	Levels and Cuts	Machining Data Tool	Data	
	Op I: FLAT - 25MM WIT hange Tool	D No. 1		
	Contour Linear Spiral Waveform Close Open Pockets Jse Max Feed on Ba		Cut Direction Smoothing Radius Back Pass Retract	0
Mill T	ype Climb	◯ Conventio	onal Optimised	
ים	iigh Speed Cornerin Take Account of <u>P</u> re		Minimum Radius	0
	erial Selection		Use Boundaries	
	Select Material(s)		Detect Undercut Stock	
	Auto-update materia operations Jse Geometry as Ma		Pick Material(s)	
			OK Cano	cel Help

Figure 83 - Use the Auto Update option





ALPHACAM 2020.1 Standard 5 Axis

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.

🍎 Z Contour Roughing		×
General Levels and Cuts Machi	ning Data Tool Da	ata
Z Levels (or Distance from Plar	ne)	
Safe Rapid Level	20	
Feed Down Distance	5	
Material Top	10	
Material Bottom	-150	
Max Depth per Cut	10	
Upper Z Limit	10	Intermediate Slices (Depth = 0 for none)
Lower Z Limit	-120	Max Depth per Cut 1
Auto		Percentage Feed 100
Include Flat Areas as addit Entry Approach Type Helix	tional Z levels	Centre Cutting Tool
	Slope A	ngle for Lead-In (0 = Horizontal) 5
		Maximum Plunge Depth 5
Cutting Order		
by <u>Z</u> one	⊖ by <u>L</u> evel	
Prismatic Geometry		
		OK Cancel Help

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

ALPHACAM 2020.1 Standard 5 Axis

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.

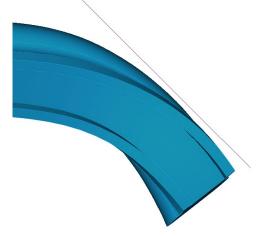


Figure 85 - Created guide line

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.

Edit 3D Polyline		— ×
Option		
Change	Insert	O Delete
⊚ List	Append	Reverse
Convert to 2	2D Geometry	
Fillet	Extend	Loop
	ок	Cancel

Figure 86 - Edit 3D Polyline options

Choose **O Extend.** <LClick> [OK].

In the next dialogue.

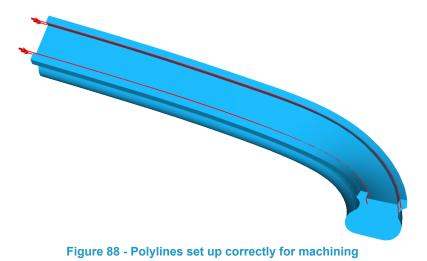
Extend Polyline	— ×
Extend	
OBy Distance	20
To 2D Boundaries	
To Active WorkPlane	
To Surfaces	
To Solids	
Polyline Selection	
 Single Multip 	le
Extend at	
🔘 Start 🛛 🔘 End	OB Both
OK Can	cel

Figure 87 - Extending all extracted lines

Set the option to **O** By Distance and make the value 20mm. Make the option for line selection to **O** Multiple and the ends option to **O** 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.



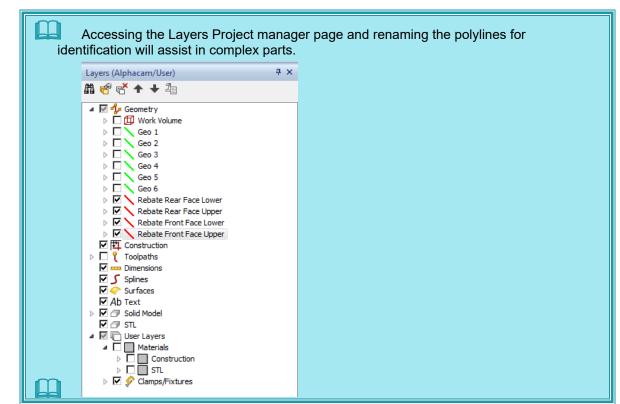


Figure 89 - Renaming geometry elements for ease of identification



Select MACHINE > Cut Between 2 Geometries

General

eral Machining Data 3D Le	ad-In/Out	5-Axis T	oolpath Op	timiser	Tool Data		
Tool							
Op No.	1						
Tool: FLAT - 25MM WITH HO	LDER						
			୍ଦି				
			1	4		ALL I	
Change Tool							
Machining							
Safe Rapid Level	50			Feed Do	own Distance	0	
Rapid Down Distance	10			1	initial Z Stock	0	
Initial XY Stock	0				Final Z Stock	3	
Final XY Stock	3						
Number of Cuts	1						
XY Corners							
O Roll Round							
Straight							
CLoop							
Loop Radius	0						
				OK	Can	cel	Help

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

g Data Step Le Chord	ength 2.5	5-Axis Toolpath Optimiser	r Tool Data	Ş
				રે
				ହ
Chord	Error 0.25			ହ
				8
		4		

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.

eral	Machining Data	3D Lead-In/Out	5-Axis Toolp	ath Optimiser	Tool Data	
	pply Auto Lead In	/Out				
E+	×E 🚰					
Lead	-in			Lead-out		
Bot	h		\sim	Both		\sim
s	loping			Sloping		
	Line	Length In 5	>>		Line Length Out	5
	Arc	Radius In 1	<<		Arc Radius Out	5
	Line/Arc	: Angle In 90			Line/Arc Angle Out	90
	Z Slope I	n Amount 0		:	Z Slope Out Amount	0
	Feedrate Mo	difier (%) 100		Fe	edrate Modifier (%)	100
Lea	ad-In Side			-Lead-Out 9	Side	
۲	<u>L</u> eft					
0	<u>C</u> entre			◯ <u>C</u> entre	t.	
0	<u>Rig</u> ht			O <u>Rig</u> ht		
		Overlap 0		F	eed Down Distance	0
	Chord Tolerance					
	Zhora rolerand	STOLMICS 011				

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.

Cut Between 2 Geom	etries			
eneral Machining Data	3D Lead-In/Out	5-Axis Toolpath Optimiser	Tool Data	
- 5-Axis Toolpath Optimi	ser			
Use optimiser				
Copy lead in/out			`	
Copy feed down di	stance			
Distance to retract l untwist				
Rotation				
Use Global Machine	Configuration			
Axis Type	Min °	Max °	Free Rotation	
X Tool	~ 0	0	\checkmark	
Y None	~ 0	0	\checkmark	
Z Tool	~ 0	0	\checkmark	
Clockwise <u>T</u> ilt				
Cardanic Axis				
° Cardanic	Angle	0		
		ОК	Cancel	Help
ure 93 - Cut Between 2				



Tool Data

neral	Machining Data	3D Lea	d-In/Out	5-Axis	Toolpath Opt	imiser	Tool Data	
Tooli							_	
	Tool N	umber	1					
	Offset N		1				di ka	
	Dia	meter	25					
	Spindle	Speed	15000					
	Dowr	n Feed	5000					
	Cu	t Feed	5000			·		
Cool]			
	None							
	Mist							
	Flood							
0	Through Tool							
						OK	Cance	Help

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

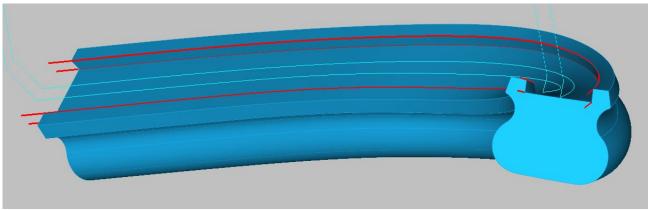


You are prompted to select;

- 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.







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.

Machine Al	long Spline/Polyline)
lachining Data	3D Lead-In/Out 5-Axis Toolpath Optimiser	
Chord	Tolerance for Spline 0	
Та	angency Tolerance ° 5	
	Type 5-Axis 🗸	
Show G	host Tools	
Machine	e Comp (G41/42)	
Tool Angle		
Parallel to	~	
Model		
◯ <u>S</u> urfac		
	Model <u>B</u> odies	
Solid N	Model Faces	
Tool Side		
€Left	○ <u>C</u> entre ○ <u>R</u> ight	
Corners		
	und Straight	
Toleranc	e for Round Corners 0.01	
	OK Cancel	Help
	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 4-Axis (YZ rot)	4-Axis (XZrot) 4 Axis (YZrot)	Rotational toolpath about the Y axis. Rotational toolpath about the X axis.
4-Axis (XZ rot) st Tools 4-Axis (YZ rot) 5-Axis Figure 97 - Axis to rotate on	5-Axis	Full 5 axis motion toolpath.
Tool Angle	Normal To	Cuts the path using the bottom of the cutter.
	Parallel To	Cuts the path using the flutes of the cutter.
Tool Angle	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.
Normal to Normal to Parallel to Angle to Normal of	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 Path Fixed Angle - Perpendicular to X or Y Axis Fixed Tool Angle Set Tool Vector Figure 98 - Tool Angle drop down options	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

Model Surfaces	Surfaces	Uses a chosen surface to create the Tool Angle from.		
 Solid Model Bodies Solid Model Faces 	Solid Model Bodies	Uses a chosen solid model body as the reference for the Tool Angle.		
Figure 99 - Tool Angle guide options	Solid Model Faces	Uses a selected face from a solid model as the Tool Angle reference.		
To ol 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		to turn on the Ghost Tools without exiting the he choice of side easier.		
□ Machine Comp (G41/42)	, , ,	e to post processors for machine controls that 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

🦻 Machine Along Spline/Polyline		×
General Tool Data		
Tool Op No. 3 Tool: FLAT - 25MM WITH HOLDER Change Tool	2	
Machining		
Safe Rapid Level 50	Number of Cuts 1	
Feed Down Distance 10 In	nitial Depth of Cut 0	
Depth of Cut -1	Initial Stock 0	
Stock to be Left 0 Bi-Direct	ctional	
	K Cancel	Help

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

🦻 Machine Along Spline/Polyli	ne				×
General Tool Data					
Tooling					
Tool Number	1				
Offset Number	1		- 6		
Diameter	25		1		
Spindle Speed	15000				
Down Feed	5000				
Cut Feed	5000				
Coolant					
 None 					
() Mist					
◯ Flood					
O Through Tool					
		_			
			ОК	Cancel	Help

Figure 101 - Cut Spline or Polyline Tool Data tab

<LClick> [OK].

When prompted for a polylin	ne to select,		
MC SPLINE/POLYLINE: Select	Previous	Finish (ESC)	All

<LClick> one of the upper polylines extracted earlier.

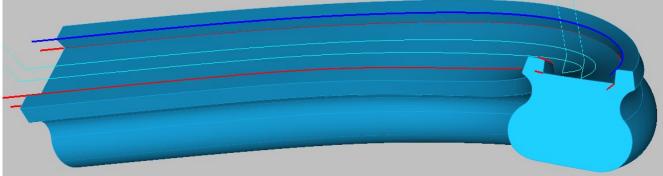


Figure 102 - Select the upper polyline as the guide

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



Machining Data

Mach	nine Along Splin	e/Polyline			>
eneral	Machining Data	3D Lead-In/Out	5-Axis Toolpath Optimiser	Tool Data	
	Chord <u>T</u> olerance	for Spline 0			
	Tangency To	lerance ° 2		n	
	Type	5-Axis	\sim		
□s	how <u>G</u> host Tools			8	\backslash
M	1achine Comp <mark>(</mark> G4	1/42)			\setminus
- Tool	Angle				
	mal to		~		
Mo	del				
0) <u>S</u> urfaces				
0) Solid Model <u>B</u> odie	S			
۲) Solid <u>M</u> odel Face	5			
Tool	Side				
OL	eft	● <u>C</u> entre	○ <u>R</u> ight		
Corn	ers				
	Roll Round		Straight		
То	olerance for Roun	d Corners 0			
			OK	Cancel	Help

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

hining Data 3D Lead-In/O	ut 5-Axis Toolp	ath Optimis	er	
Apply Auto Lead In/Out				
₽ +8 🛃 🔒				
Lead-in			Lead-out	
Both	\sim		Both	~
Sloping			Sloping	
Line Leng	th In 5	>>	Line Length Out	5
Arc Radi	us In 1	<<	Arc Radius Out	5
Line/Arc Ang	ile In <mark>90</mark>		Line/Arc Angle Out	90
Z Slope In Am	ount 0		Z Slope Out Amount	0
Feedrate Modifier	(%) 100		Feedrate Modifier (%)	100
Lead-In Side			Lead-Out Side	
◯ <u>L</u> eft			⊖ <u>L</u> eft	
Oentre			Ontre	
○ <u>Rig</u> ht			◯ <u>Rig</u> ht	
Ovi	erlap <mark>0</mark>		Feed Down Distance	0
Chord Tolerance for	August 0, 1			

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

hining Data	3D Lead-In/Out	5-Axis Toolpath	Optimiser		
5-Axis Toolp	oath Optimiser —				
Use opti	miser				
Copy lea	ad in/out				
Copy fe	ed down distance	:			
Distance	to retract before untwist rapids		0		
Rotation					
🗹 Use Glob	oal Machine Confi	guration			
Ab	kis Type	Min °	Max °	Free Rotation	
ХТ	iool 🗸 🗸	0	0	\checkmark	
Y N	lone 🗸	0	0	\checkmark	
ΖT	iool 🗸	0	0	\checkmark	
Clockwis	e Tilt				
Cardani					
	° Cardanic Angle	0			

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.

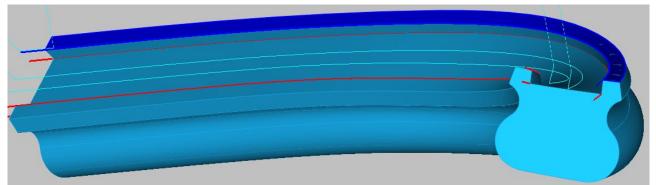


Figure 106 - Selecting the face to machine

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

Repeat the process on the second top face.

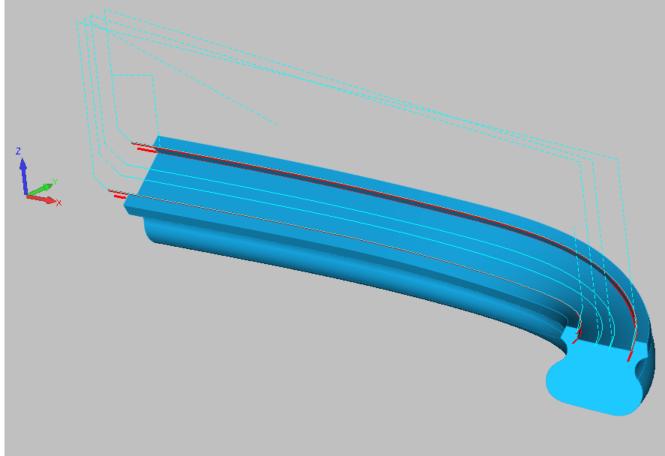


Figure 107 - Applied tool paths



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.

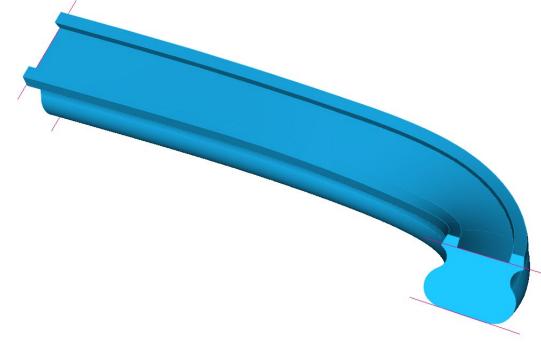


Figure 108 - Created work planes and geometry for the end faces

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.

and making the Tool Angle option to O Parallel to O Solid

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

7, then using

MACHINE > Cut Spline or Polyline

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

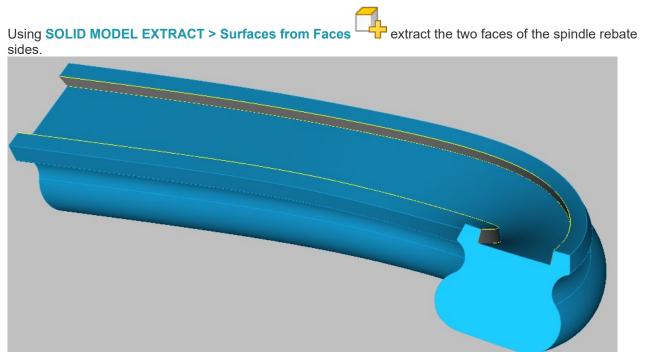


Figure 109 - Extracting the side wall surfaces

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

Offset X					
Amount					
Distance 6					
Offset to Point					
Offset on both sides					
What					
O Line / Arc O Geometry					
Surface					
Offset as Geometry					
Delete Original					
OK Cancel					

Figure 110 - Using the Surface offset option



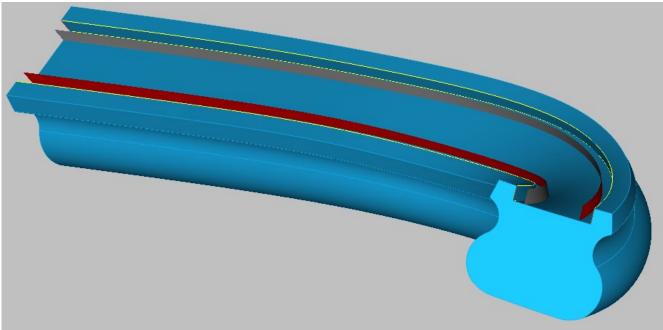
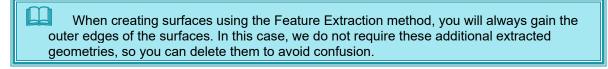


Figure 111 - Extra surfaces after using the Offset command

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.



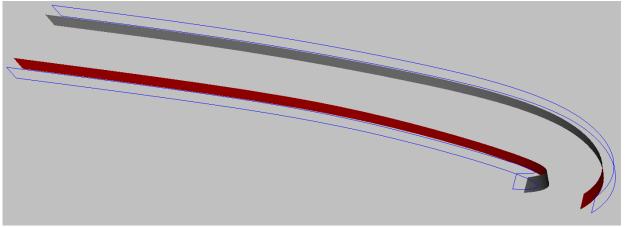


Figure 112 - Delete the original surfaces and the acquired outline profiles



Creating the new floor driving surface

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

9	₽ † >	<u>></u>	5
Edit Surfaces *	Reverse Tool Side	Create Sections	Project 3D to 2D
Edit Un- Exte	ak Intersecti t Surface Co Trim Surfac end Surface overt to STL	es	
Extr	ract Edge fr	om Surface	5

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.

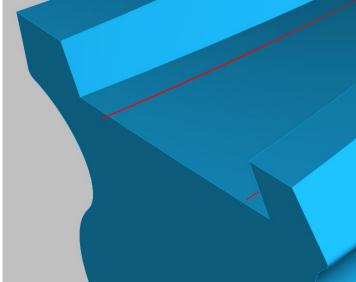


Figure 116 - The polylines will need manipulation in this example



Edit 3D Polyline			×				
Option							
◯ Change	◯ Insert	◯ Delete					
🔾 List	O Append	◯ Reverse					
O Convert to 2D Geometry							
○ Fillet	Extend						
OK Cancel							
OK Cancel							

Figure 117 - Edit 3D Polyline dialogue

<LClick> [OK].



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

Extend Polyline	×
Extend	
By Distance	10
O To 2D Boundaries	
O To Active WorkPlane	
O To Surfaces	
◯ To Solids	
Polyline Selection	
◯ Single	 Multiple
Extend at	
⊖ Start ⊖ End	Both
ОК	Cancel

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.

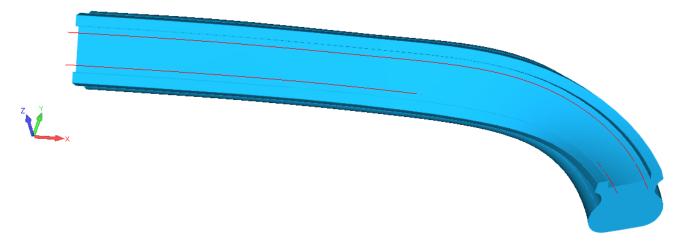


Figure 119 - Extended Polylines beyond the end of the part



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

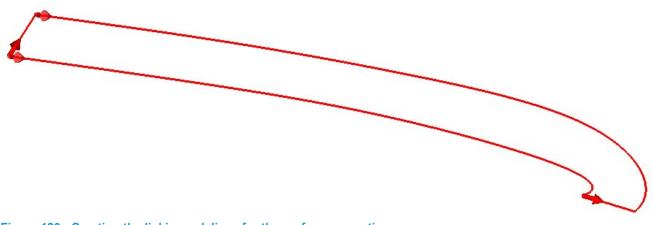


Figure 120 - Creating the linking polylines for the surface generation process

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.

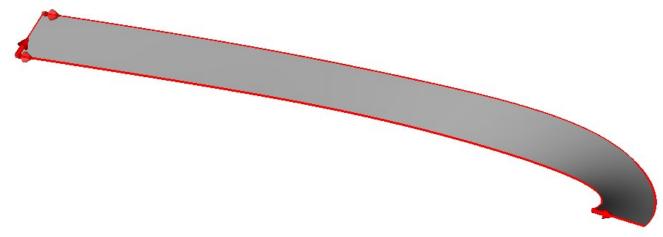


Figure 121 - Completed surface for machining



Rebate floor finishing

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

Machining	
Туре	
5-Axis	
Fixed Tool Angle 0	
Strategy	
Parameter Lines	~
Machine	Checks
Machine	Checks Avoid Fouling Non-Machined Surfaces
_	Avoid Fouling Non-Machined Surfaces



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

3D Machining - Parameter Lines General Machining Data Tool Data	×
Tool Op No. 1 Tool: BALL END - 10MM + HOLDER	
Machining Stock to be Left 0 Safe Rapid Level 50	Overlap on Open Elements: Tool Rad × Thickness Above Surface to Rapid Down to
	OK Cancel

Figure 123 – Parameter Lines General tab

Set the options as shown.

Machining Data

3D Machining - Parameter Lines	;
General Machining Data Tool Data	
Cut Spacing uses	
Width of Cut	1100-
◯ Cusp Height	
Width 0.2	
Cusp 0.5	
·	-uul
Options	
Bi-Directional	Tolerances
Angle Between Tool and Surface Normal	Chord Tolerance along Cut 0.02
	OK Cancel

Figure 124 – Parameter Lines Machining Data tab

Set the options as shown.



Tool Data

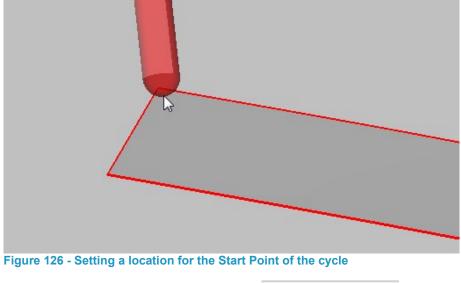
eneral Machining Data Tool D	ata		
Tooling Tool Number Offset Number Diameter Spindle Speed Down Feed Cut Feed	1 10 18000 2000 2000		
Coolant None Mist Flood Through Tool			

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.

Pick Near Start Point





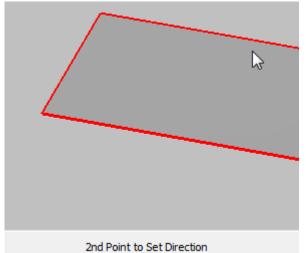


Figure 127 - Choosing the second point for direction of cut

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.

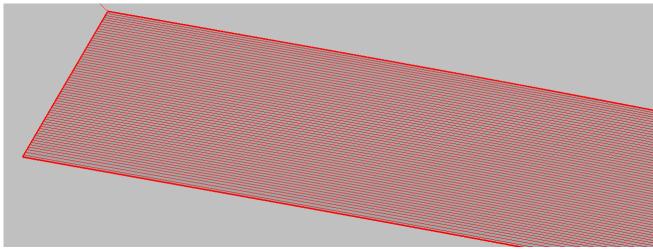


Figure 128 - Applied tool path



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.

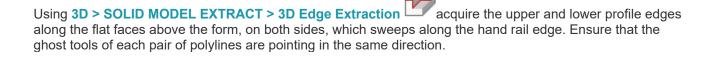


Finish handrail profile

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.



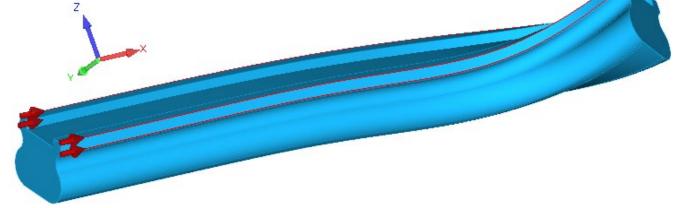


Figure 129 - Extracting the outside polyline options

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.



差 Cut Between 2 Geometries			×
General Machining Data 3D Lead-In/Out Too	l Data		
Apply Auto Lead In/Out			
Lead-in		Lead-out	
Line ~		Line	~
Line Length In 15	>>	Line Length Out	15
Arc Radius In 10	<<	Arc Radius Out	10
Line/Arc Angle In 0		Line/Arc Angle Out	0
Z Slope In Amount 0		Z Slope Out Amount	0
Feedrate Modifier (%) 100		Feedrate Modifier (%)	100
Lead-In Side		Lead-Out Side	
) Centre		() Centre	
Right		● Right	
Overlap 0 Chord Tolerance for Arcs 0.1		Feed Down Distance	0
		OK Cancel	Help

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.

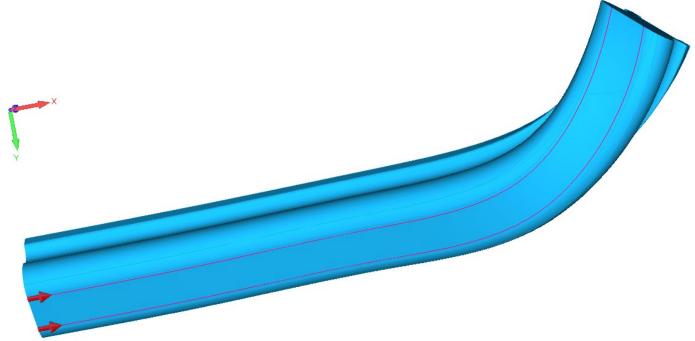


Figure 131 - Underside polylines as the guides to drive the handrail form tool

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.

Machine Ald	ong Spline/Polyline			
achining Data	3D Lead-In/Out 5-Axis	Toolpath Optimiser		
Chord	Tolerance for Spline 0			
Та	ngency Tolerance ° 5			
	Type 5-Axis	~ (
Show <u>G</u> h	ost Tools			
Machine	Comp (G41/42)			
Tool Angle				
Angle to No	rmal of	~		
Model			Side Tilt Angle 0	
○ <u>S</u> urfac	es			
◯ Solid M	odel <u>B</u> odies		Direction Angle 180	
Solid M	odel Faces			
Tool Side				
() <u>L</u> eft	○ <u>C</u> entre	(● <u>Rig</u> ht		
Corners				
	nd	Straight		
Tolerance	for Round Corners 0.0	1		
			< Cancel	Help

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



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

🦻 Machine Along Spline/Polyline			
Machining Data 3D Lead-In/Out 5-Axis Toolpa	ath Optimi	ser	
Apply Auto Lead In/Out			
Lead-in Both ~		Lead-out Both	~
Sloping Line Length In 75	>>	Sloping	
Arc Radius In 1	<<	Arc Radius Out 5	
Line/Arc Angle In 0		Line/Arc Angle Out 90	
Z Slope In Amount 0		Z Slope Out Amount 0	
Feedrate Modifier (%) 100		Feedrate Modifier (%) 10	0
Lead-In Side		Lead-Out Side	
○ <u>L</u> eft ○ <u>C</u> entre		○ <u>L</u> eft ○ <u>C</u> entre	
O <u>c</u> entre			
Overlap 0 Chord Tolerance for Arcs 0.1	4	Feed Down Distance 0	
Figure 400		OK Cancel	Help

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.



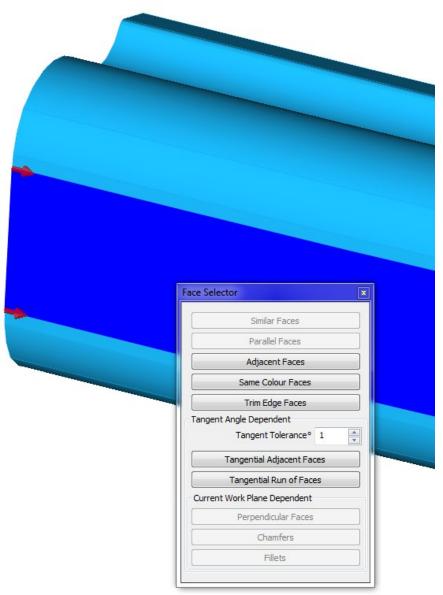


Figure 134 - Setting the underside face as the guide for the cut



<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.

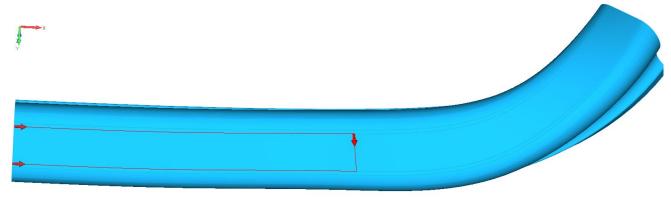


Figure 135 - Approximate location to trim the polylines to



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.

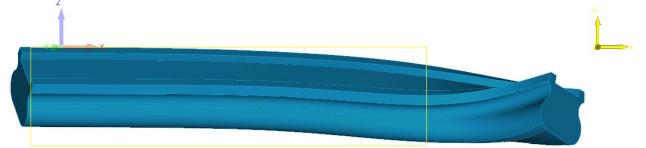
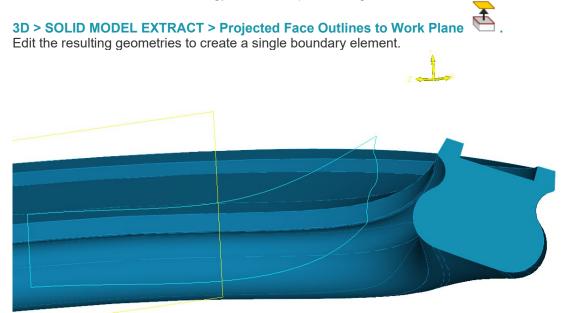


Figure 136 - Work Plane generated on the side using Normal to View

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





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

D Machining	×
Туре	~
3-Axis 🗸	
Fixed Tool Angle 0	
Strategy	
Projected Contours V	
Machine	Checks
Use Boundaries	Avoid Fouling Non-Machined Surfaces/Solids
Select Faces	Check for Gouging on Current Surface
Use Guide Curves	Check Adjacent Machined Surfaces
Help	OK Cancel
Help	OK Cancel



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.



ALPHACAM 2020.1 Standard 5 Axis

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.

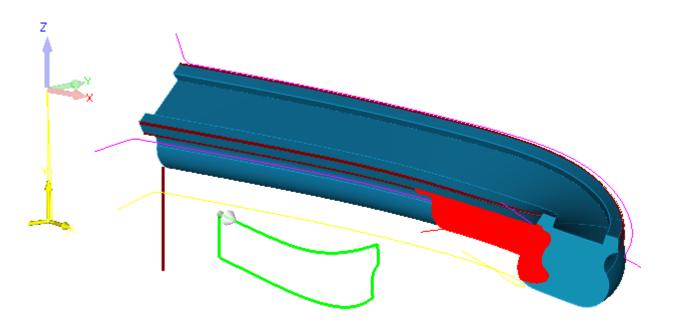


Figure 139 - Geometry boundary initial applied tool path and extra polyline



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.

Tool Axis Conversion			
Apply Tool Axis Conversion			
Туре			
3-Axis S 5-Axis			
4-Axis (XZ rot)			
O 4-Axis (XY rot), Tool at Fixed Angle from Vertical			
Tool Angle from Vertical 0			
Action			
Convert to Vertical or Current Work Plane			
Through Point			
O Through Axis			
Confine with Boundary			
Normal to Curve/Curves			
Normal to Surfaces			
Normal to Solid Model Bodies			
Normal to Solid Model Faces			
Side Tilt Angle 0			
Direction Angle 0			
OK Cancel			

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.



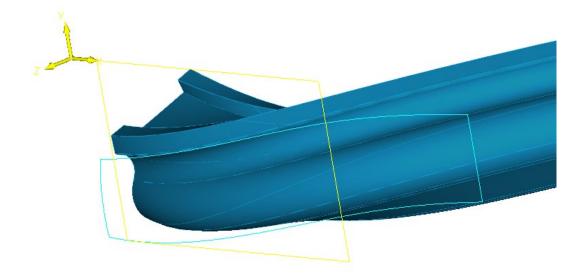


Figure 141 – Work Plane and suitable boundary on the reverse side

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 **O** Through Point option and when prompted select the bottom of the previously created polyline. When requested, select the [No] option for 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.



ALPHACAM 2020.1 Standard 5 Axis

Version amendments

N/	Amendment Description	^	Software Version	Amondod Data
V	Amendment Description	A	Soltware 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





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

