



HEXAGON
MANUFACTURING INTELLIGENCE

ALPHACAM 2020.1 STANDARD 2D MACHINING



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.....	xiv
Recommended Operating Systems and Hardware for ALPHACAM.....	xv
Supported Operating Systems	xv
ALPHACAM Minimum Specifications.....	xv
Hexagon Customer Portal.....	xvi
ALPHACAM esupport.....	xvii
Introduction	1
Standard 2D Machining Fundamentals.....	2
Outline Program Production	2
Housing Plate Process Plan	3
Material Creation.....	5
AutoSet Material.....	5
Set Material.....	6
Move to machining position	8
Setting Start Point, First and Order Options.....	16
Start Pt.....	16
First.....	16
Order	17
Order Geometries Auto	18
Order Tool Paths Auto	18
Order Geometries & Tool paths Manual	18
Order Geometries Selected Auto.....	18
Order Geometries By Intersection	18
Order Geometries Selected By Intersection	18
Tool Directions.....	19
Direction of Machining.....	19
Tool Directions Dialogue	20
Hard Boundaries	21

Soft Boundaries.....	21
Default Options.....	22
Get Default.....	22
Store Default.....	22
Load.....	22
Save.....	22
Original Ghost tools.....	23
Set the Tool Directions.....	23
Set the Start Points.....	24
Fundamentals Conclusion.....	25
Red Geometry.....	26
No Ghost Tools visible.....	26
Tools on Centre Error Message.....	26
Area Clearance, Standard Pocketing.....	27
Types.....	27
⊙ Vertical.....	28
⊙ Sloping.....	28
⊙ Profiled.....	28
⊙ Auto Z.....	29
☑ Use Z Contour Roughing.....	29
☐ Use Z Contour Roughing.....	30
☑ Z Contour Roughing.....	30
General.....	31
Contour or Linear.....	32
Spiral.....	32
Take account of Previous Machining.....	32
Include additional Rough/Finish Pass.....	32
Levels and Cuts.....	33
Z Levels (or Distance from Plane).....	34
Fixed Z options.....	36
Auto Z options.....	36

Number of Cuts	36
NC Code for Multiple cuts	36
Cutting Order.....	37
Depths of Cut	37
Slope and Profile Parameters	37
Machining Data.....	38
Set/Unset Open Elements	39
Tool Data	40
Area Clearance, Z Contour Roughing Options	41
Types	41
General	42
Type	43
Smoothing Radius (Waveform option only)	44
Back Pass Retract (Waveform option only)	44
Use Max Feed on Back Pass (Waveform option only)	44
Mill Type.....	45
High Speed Cornering	45
Take Account Previous Machining.....	45
Include additional Rough and Finish Pass.....	45
Levels and Cuts.....	46
Z Levels (or Distance from Plane)	47
Max Depth per Cut.....	47
Intermediate Slices (Sloping Sides Only).....	47
X Y Corners (Vertical and Auto Z Only).....	48
Approach Type.....	48
Slope Angle for Lead-In	49
Maximum Plunge Depth.....	49
Cutting Order.....	49
Helix Diameter.....	49
Sloping Side	50
Corner Type	50
Machining data	51

XY Stock to be Left	51
Z Stock to be Left	51
Chord Tolerance	52
Overlap on Open elements	52
Width of Cut for Final Pass	52
Width of Cut	52
Trochoidal for full-width moves	53
Link Method.....	53
Link Type.....	53
XY StandOff	54
Feed When Plunging	54
Avoid Rollover	54
Adaptive Feed Rates	55
Tool Data	57
Rough or Finish, Vertical Sides.....	58
Types	58
General	59
Tool Compensation Overview.....	60
Compensation ☉Tool Centre.....	61
Compensation ☉G41/42 On Tool Centre	62
Compensation ☉Machine Comp (G41/42)	63
<input checked="" type="checkbox"/> Apply Compensation on Rapid Approach/Retract	64
<input checked="" type="checkbox"/> Take Account of Previous Machining	64
X Y Corners.....	65
<input checked="" type="checkbox"/> Auto Break-out (Open paths only)	66
Levels and Cuts	67
Z Levels (or Distance from Plane)	67
Number of Cuts	67
<input type="checkbox"/> Bi-Directional (Open Paths only)	67
<input type="checkbox"/> Helical Interpolation	68
NC Code for Multiple cuts	68
Depths of Cut	68

Machining Data.....	69
Machining Options	69
Widths of Cut.....	70
Cutting Order.....	70
Spiral stock removal.....	70
Use Local Ordering	70
Reverse Cut Direction	70
Links.....	71
Lead-In/Out.....	72
Lead-In/Out options.....	73
Style of Lead-In/Out.....	73
Tool Entry, Lead In/Out options	74
Line	74
Arc	76
Both	77
Overlap	79
Tool Data	80
Non Vertical Sides (Standard and higher modules)	81
Pocketing using Sloping	81
Profiled	83
Pocketing using Profiled	83
Rough and Finish using Profiled sides	86
Previous Tool Profile	89
Chamfer Profiling.....	90
Drill/Tap Holes Options.	91
General.....	92
Type	92
NC Code	93
Which Holes	93
Levels and Cycle Data.....	94
Drilling Fixed Z	94

Drilling Auto Z.....	94
Traverse At.....	94
Hole Depth	94
Pecking Fixed Z	95
Pecking Auto Z.....	95
Tapping Fixed Z	96
Tapping Auto Z.....	96
Boring Fixed Z.....	96
Boring Auto Z	96
Chamfering Fixed Z.....	97
Chamfering Auto Z.....	97
Common options on all Drill/ Tap Holes Auto Z options.....	98
<input checked="" type="checkbox"/> Rapid Level is Absolute	98
<input checked="" type="checkbox"/> Set Stock to Upper Level	98
Auto Z Hole Depth Selection.....	99
Tool Data	100
Selecting Geometries	100
Chamfering cycle.....	101
Machining Holes	103
Pocket Holes	104
Operation Manipulation.....	105
Add and Remove geometries in machining strategies.....	106
Adding or Re-selecting Geometries in a cycle	106
Removing Geometries from a cycle	107
Move to own operation	107
Setting and Using the Solid Simulation.....	108
Setting up the Material.....	108
Using Set Material.....	108
Using Auto Set Material	110
Simulation.....	111
Close Simulation	111

Shade/Solid Simulation	111
Stop Simulation	111
Run Simulation.....	111
Pause Simulation	111
Show Tools	111
Operation by Operation.....	111
Single Step.....	111
Show Tool colours.....	111
Simulation speed	112
Miscellaneous options	112
Vertical Section	112
Horizontal Section	112
Remove Sectioning	112
Save Stock	112
Split into Separated Parts	112
Tolerance	112
Transparency	113
Material transparency slider	113
Clamping and Fixture transparency slider	113
Tooling transparency slider	113
Tool Holder transparency slider	113
Machine transparency slider	113
Configure	113
Solid Simulator	114
Send To Simulator	114
Send to Rapid Result.....	114
Sawing.....	115
Types.....	115
Z Levels.....	116
Geometries.....	116
General	117
Cut To	118

Cut On.....	118
Cut Past	119
Levels and Cuts.....	120
Scratch Cut	120
Minimum Length of Cut.....	120
Machining Data.....	121
Foul Avoidance	121
Tool Data	122
Set Element Saw Angle.....	123
3D Cut/Engrave	124
General	124
Method	125
Levels and Cuts.....	125
Machining Data.....	126
Chord Error	126
Step Length.....	126
Tool Data	127
Simple 2.5D Engraving.....	128
Quick Edit	129
Machining Styles.....	130
Machining Styles Project Manager.....	130
Creating a style.....	131
Organising Machining Styles.....	131
Manipulation and Duplication of Machining Styles.....	131
Applying Machining Styles.....	131
Apply Style to Selected Geometries	131
Apply Style to Selected Layer	131
Assign Layers for Geometries.....	131
Create Layers for Styles.	131
Assign Layer for Boundaries.....	131
Assign Layer for Points	131

Auto Styles Macro	131
NC Program Production Process	132

Table of Images

FIGURE 1 - CUSTOMER SUPPORT PORTAL	XVI
FIGURE 2 - ESUPPORT PAGE	XVII
FIGURE 3 - HOUSING PLATE GEOMETRY	4
FIGURE 4 - HOUSING PLATE SAMPLE FINISHED SOLID MODEL	4
FIGURE 5 - AUTO SET MATERIAL DIALOGUE	5
FIGURE 6 - MATERIAL OUTLINE IN PLACE PRIOR TO SELECTION	6
FIGURE 7 - SELECT THE REQUIRED PROFILE	6
FIGURE 8 - SET MATERIAL DIALOGUE	7
FIGURE 9 - SHOW ALL OPTION	8
FIGURE 10 - GEOMETRY FOR ENCLOSING RECTANGLE OPTION	9
FIGURE 11 - APPLIED RECTANGLE	10
FIGURE 12 - ESSENTIAL OFFSET OPTIONS	11
FIGURE 13 - SELECT THE RECTANGLE TO OFFSET	11
FIGURE 14 - COMPLETED OFFSET	12
FIGURE 15 - ESSENTIAL METHOD READY TO MOVE	13
FIGURE 16 - SHOW ALL OPTION	14
FIGURE 17 - ALL VISIBLE	14
FIGURE 18 - BASE POINT LOCATION PRIOR TO <LCLICK>	15
FIGURE 19 - HOUSING PLATE CORRECTLY LOCATED FOR MACHINING	15
FIGURE 20 - ORDER DIALOGUE OPTIONS	17
FIGURE 21 - CLIMB AND CONVENTIONAL CUTTING	19
FIGURE 22 - TOOL DIRECTIONS FOR OPEN GEOMETRIES	20
FIGURE 23 - TOOL DIRECTIONS FOR CLOSED GEOMETRIES	20
FIGURE 24 - START POINT OPTIONS	20
FIGURE 25 - AUTO START POINT LOCATIONS	20
FIGURE 26 - SETTING DEFAULT OPTIONS	22
FIGURE 27 - GHOST TOOLS AS CREATED ON THE HOUSING PLATE	23
FIGURE 28 - GHOST TOOLS SET CORRECTLY FOR MACHINING	23
FIGURE 29 - GEOMETRY START POINTS SET TO SUIT GOOD MANUFACTURING	24
FIGURE 30 - ERROR DISPLAYED ON OPEN GEOMETRIES WHEN SETTING INSIDE/OUTSIDE	26
FIGURE 31 - TOOLPATH ON CENTRE ERROR DIALOGUE	26
FIGURE 32 - CONTOUR POCKET TYPES	27
FIGURE 33 - EXAMPLE OF FEATURE EXTRACTED GEOMETRIES	29
FIGURE 34 - GENERAL TAB OF THE POCKETING CYCLE	31
FIGURE 35 - POCKETING USING THE CONTOUR OPTION	32
FIGURE 36 - POCKETING USING THE LINEAR OPTION	32
FIGURE 37 - POCKETING USING SPIRAL OPTION	32
FIGURE 38 - LEVELS AND CUTS TAB OF THE POCKETING CYCLE	33
FIGURE 39 - SAFE RAPID POSITION IN Z	34
FIGURE 40 - RAPID DOWN TO POSITION IN Z	34
FIGURE 41 - MATERIAL TOP IN Z	35
FIGURE 42 - Z FINAL DEPTH	35
FIGURE 43 - MACHINING DATA TAB ON POCKETING	38
FIGURE 44 - TOOLPATH PRIOR TO USING SET/UNSET OPEN ELEMENTS	39
FIGURE 45 - TOOLPATH AFTER TO USING SET/UNSET OPEN ELEMENTS	39
FIGURE 46 - TOOL DATA TAB IN POCKETING	40
FIGURE 47 - TYPE TAB OF THE POCKETING CYCLE USE Z CONTOUR ROUGHING ACTIVATED	41
FIGURE 48 - GENERAL TAB OF THE POCKETING CYCLE USING Z CONTOUR ROUGHING	42
FIGURE 49 - CONTOUR POCKET USING Z CONTOUR ROUGHING	43
FIGURE 50 - LINEAR POCKET USING Z CONTOUR ROUGHING	43
FIGURE 51 - SPIRAL POCKET USING Z CONTOUR ROUGHING	43

FIGURE 52 - WAVEFORM POCKET USING Z CONTOUR ROUGHING	43
FIGURE 53 - HIGH SPEED CORNERING ACTIVE	45
FIGURE 54 - HIGH SPEED CORNERING NOT ACTIVE	45
FIGURE 55 - VERTICAL DIALOGUE OPTIONS	46
FIGURE 56 - SLOPING SIDES DIALOGUE	46
FIGURE 57 - AUTO Z DIALOGUE OPTIONS	46
FIGURE 58 - XY CORNERS SET TO ROLL ROUND	48
FIGURE 59 - XY CORNERS SET TO STRAIGHT	48
FIGURE 60 - APPROACH OPTIONS	48
FIGURE 61 - OPTIONS ONLY AVAILABLE USING SLOPING SIDES	50
FIGURE 62 - MACHINING DATA DIALOGUE OPTIONS	51
FIGURE 63 - CHORD TOLERANCE	52
FIGURE 64 - TROCHOIDAL TOOLPATH EXAMPLE	53
FIGURE 65 - LINKING BETWEEN DIFFERENT AREAS	53
FIGURE 66 - AVOID ROLLOVER ACTIVE	54
FIGURE 67 - AVOID ROLLOVER INACTIVE	54
FIGURE 68 - ADAPTIVE FEED RATES OPTIONS	55
FIGURE 69 - FEED RATE CHANGES WHEN USING ADAPTIVE FEED RATES	56
FIGURE 70 - TOOL DATA	57
FIGURE 71 - TYPES TAB FOR ROUGH OR FINISH CYCLE	58
FIGURE 72 - GENERAL TAB	59
FIGURE 73 - NC CODE ACTUAL LOCATIONS ON SAMPLE PART	61
FIGURE 74 - NC CODE ACTUAL LOCATIONS ON SAMPLE PART	62
FIGURE 75 - NC CODE ACTUAL LOCATIONS ON SAMPLE PART	63
FIGURE 76 - EXAMPLES OF XY CORNERS	65
FIGURE 77 - LOOP VALUE OPTIONS	65
FIGURE 78 - KNIFE LOOPS NOT ACTIVE	65
FIGURE 79 - KNIFE LOOPS ACTIVE	65
FIGURE 80 - AUTO BREAK-OUT OPTIONS	66
FIGURE 81 - AUTO BREAK-OUT TOOLPATHS	66
FIGURE 82 - LEVELS AND CUTS	67
FIGURE 83 - HELICAL INTERPOLATION NOT ACTIVE	68
FIGURE 84 - HELICAL INTERPOLATION ACTIVE	68
FIGURE 85 - MACHINING DATA TAB ON ROUGH OR FINISH	69
FIGURE 86 - LINKS DROP DOWN OPTIONS	71
FIGURE 87 - LINK SET TO NONE	71
FIGURE 88 - LINK SET TO STRAIGHT	71
FIGURE 89 - LINK SET TO S-LINK	71
FIGURE 90 - LINK SET TO LOOPED	71
FIGURE 91 - WARNING IF USING LINKS WITH THE WRONG MACHINE COMPENSATION OPTION	71
FIGURE 92 - LEAD IN/OUT TAB ON ROUGH OR FINISH	72
FIGURE 93 - LEAD-IN/OUT DATA FILE OPTIONS	73
FIGURE 94 - DROP DOWN SELECTOR FOR THE TYPE OF LEAD-IN	73
FIGURE 95 - LEAD-IN/OUT USING LINE	74
FIGURE 96 - LINE LEAD-IN USING A 90° APPROACH ANGLE	74
FIGURE 97 - SLOPING OPTION ACTIVE	75
FIGURE 98 - CALCULATION OF THE SLOPE ANGLE	75
FIGURE 99 - USE RAMP ANGLE ACTIVE	75
FIGURE 100 - LEAD-IN/OUT USING ARC	76
FIGURE 101 - ARC LEAD-IN USING A 90° APPROACH ANGLE	76
FIGURE 102 - ARC LEAD-IN AND OUT	76
FIGURE 103 - LEAD-IN/OUT USING BOTH	77
FIGURE 104 - BOTH LEAD-IN USING A 90° APPROACH ANGLE	77

FIGURE 105 - TANGENTIAL OPTION OFF	77
FIGURE 106 - TANGENTIAL OPTION ON	77
FIGURE 107 - AUTO LINK OFF	78
FIGURE 108 - AUTO LINK ON, TANGENTIAL ON	78
FIGURE 109 - AUTO LINK ON, TANGENTIAL OFF	78
FIGURE 110 - OVERLAP VALUE OPTIONS	79
FIGURE 111 - USING A POSITIVE OVERLAP VALUE	79
FIGURE 112 - USING A NEGATIVE OVERLAP FOR A SUPPORT TAG	79
FIGURE 113 - TOOL DATA TAB	80
FIGURE 114 - SLOPING SIDES LEVELS AND CUTS TAB STANDARD POCKETING	81
FIGURE 115 - SLOPING SIDES TAB Z CONTOUR ROUGHING POCKET	81
FIGURE 116 - LEVELS AND CUTS TAB USING SLOPING SIDES	82
FIGURE 117 - LEVELS AND CUTS TAB USING PROFILED SIDES ON POCKETING	83
FIGURE 118 - PROMPT TO PICK THE CROSS SECTION PROFILE	84
FIGURE 119 - PROMPT FOR THE APPLICATION POINT OF THE CROSS SECTION	84
FIGURE 120 - WIREFRAME TOOLPATH OF PROFILED SIDES POCKET	85
FIGURE 121 - SIMULATED PROFILED SIDES POCKET	85
FIGURE 122 - LEVELS AND CUTS TAB USING PROFILED SIDES ON ROUGH OR FINISH CYCLE	86
FIGURE 123 - ERROR DISPLAYED WHEN A CROSS SECTION PROFILE WOULD OVERLAP ITSELF	88
FIGURE 124 - PREVIOUS TOOL PROFILE APPLICATION	89
FIGURE 125 - ORIGINAL APPLIED ROUGH CYCLE	89
FIGURE 126 - USING THE PREVIOUS TOOL PROFILE OPTION	89
FIGURE 127 - DRILLING OPTIONS	91
FIGURE 128 - GENERAL TAB FOR ALL DRILLING	92
FIGURE 129 - LEVELS AND CYCLE DATA TAB FIXED Z DRILLING	94
FIGURE 130 - LEVELS AND CYCLE DATA AUTO Z DRILLING	94
FIGURE 131 - LEVELS AND CYCLE DATA PECKING FIXED Z	95
FIGURE 132 - LEVELS AND CYCLE DATA PECKING AUTO Z	95
FIGURE 133 - LEVELS AND CYCLE DATA TAPPING FIXED Z	96
FIGURE 134 - LEVELS AND CYCLE DATA TAPPING AUTO Z	96
FIGURE 135 - LEVELS AND CYCLE DATA TAB BORING FIXED Z	96
FIGURE 136 - LEVELS AND CYCLE DATA TAB BORING AUTO Z	96
FIGURE 137 - LEVELS AND CYCLE DATA CHAMFERING FIXED Z	97
FIGURE 138 - LEVELS AND CYCLE DATA CHAMFERING AUTO Z	97
FIGURE 139 - SETTING STOCK OPTIONS IN AUTO Z DRILLING OPERATIONS	98
FIGURE 140 - AUTO Z HOLE DEPTH OPTIONS	99
FIGURE 141 - TOOL DATA DRILL/TAP HOLES	100
FIGURE 142 - CHAMFER DEPTH CALCULATION FIXED Z	101
FIGURE 143 - CHAMFER DEPTH CALCULATION AUTO Z	101
FIGURE 144 - CHAMFER CALCULATION FOR NO CHAMFER OUTER DIAMETER	102
FIGURE 145 - CHAMFER CALCULATIONS WHEN GIVEN THE OUTER CHAMFER DIAMETER	102
FIGURE 146 - GENERAL TAB USING MACHINE HOLES	103
FIGURE 147 - GENERAL TAB USING POCKET HOLES	104
FIGURE 148 - <RCLICK> OPTIONS FOR CYCLES IN MILL/ ROUTER/ STONE	105
FIGURE 149 - ACCESSING THE ABILITY TO ADD OR RE-SELECT GEOMETRIES ON A CYCLE	106
FIGURE 150 - <RCLICK> OPTIONS TO REMOVE GEOMETRIES FROM OPERATIONS	107
FIGURE 151 - MOVING A GEOMETRY FROM A SINGLE CYCLE INTO TWO SEPARATE CYCLES	107
FIGURE 152 - NEW LISTING WITH TWO SEPARATE CYCLES	107
FIGURE 153 - ADD-INS MANAGER	108
FIGURE 154 - SET MATERIALS OPTION DIALOGUE	109
FIGURE 155 - AUTO SET MATERIAL OPTION DIALOGUE	110
FIGURE 156 - SIMULATION CONTROLS FROM THE PROJECT MANAGER	111
FIGURE 157 - SIMULATION SPEED SLIDER CONTROLS	112

FIGURE 158 - SIMULATION TOLERANCE	112
FIGURE 159 - SIMULATION TRANSPARENCY SLIDERS	113
FIGURE 160 - EXTERNAL SIMULATOR WINDOW	114
FIGURE 161 - TYPES TAB ON SAWING CYCLE	115
FIGURE 162 - GENERAL TAB ON SAWING	117
FIGURE 163 - EXPLANATION OF CUT TO	118
FIGURE 164 - EXPLANATION OF CUT ON	118
FIGURE 165 - EXPLANATION OF CUT PAST	119
FIGURE 166 - LEVELS AND CUTS TAB ON SAWING	120
FIGURE 167 - MACHINING DATA TAB ON SAWING	121
FIGURE 168 - TOOL DATA TAB ON SAWING	122
FIGURE 169 - SET ELEMENT SAW ANGLE PROMPT	123
FIGURE 170 - ANGLE ENTRY DIALOGUE	123
FIGURE 171 - GENERAL TAB 3D CUT/ENGRAVE	124
FIGURE 172 - LEVELS AND CUTS TAB	125
FIGURE 173 - MACHINING DATA TAB	126
FIGURE 174 - TOOL DATA TAB	127
FIGURE 175 - ENGRAVING USING 2D EXACT ENGRAVING	128
FIGURE 176 - QUICK EDIT WINDOW	129
FIGURE 177 - QUICK EDIT ALTERATION DIALOGUE	129
FIGURE 178 - MACHINING STYLES PROJECT MANAGER PAGE	130

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 **n**ames are represented as bold text. And the value to be entered will be represented by Bold Text.
For example: Enter the value **50** in the **Offset** field. Or
When prompted for the X & Y values type **100,50 <Enter>**



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



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



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



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



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



This is a highlighted note to emphasise information



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



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

1. This is the first line of a number list item
 2. This is the second item of the numbered instructions, which you must
 3. Follow in sequence.
- This is a list
 - of items, in which
 - The order is not important.

Recommended Operating Systems and Hardware for ALPHACAM

Supported Operating Systems

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



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



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

ALPHACAM Minimum Specifications



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

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

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



If using Autodesk Inventor Files, please check the current Inventor View requirements at [autodesk inventor view](#)

Hexagon Customer Portal

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

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

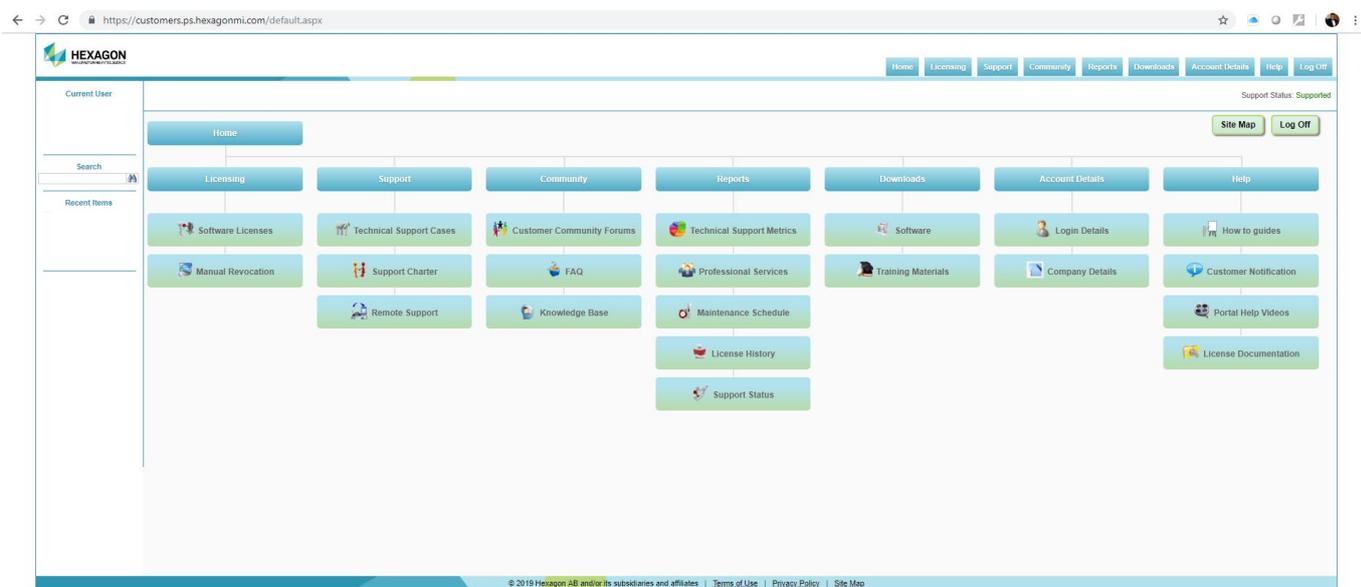
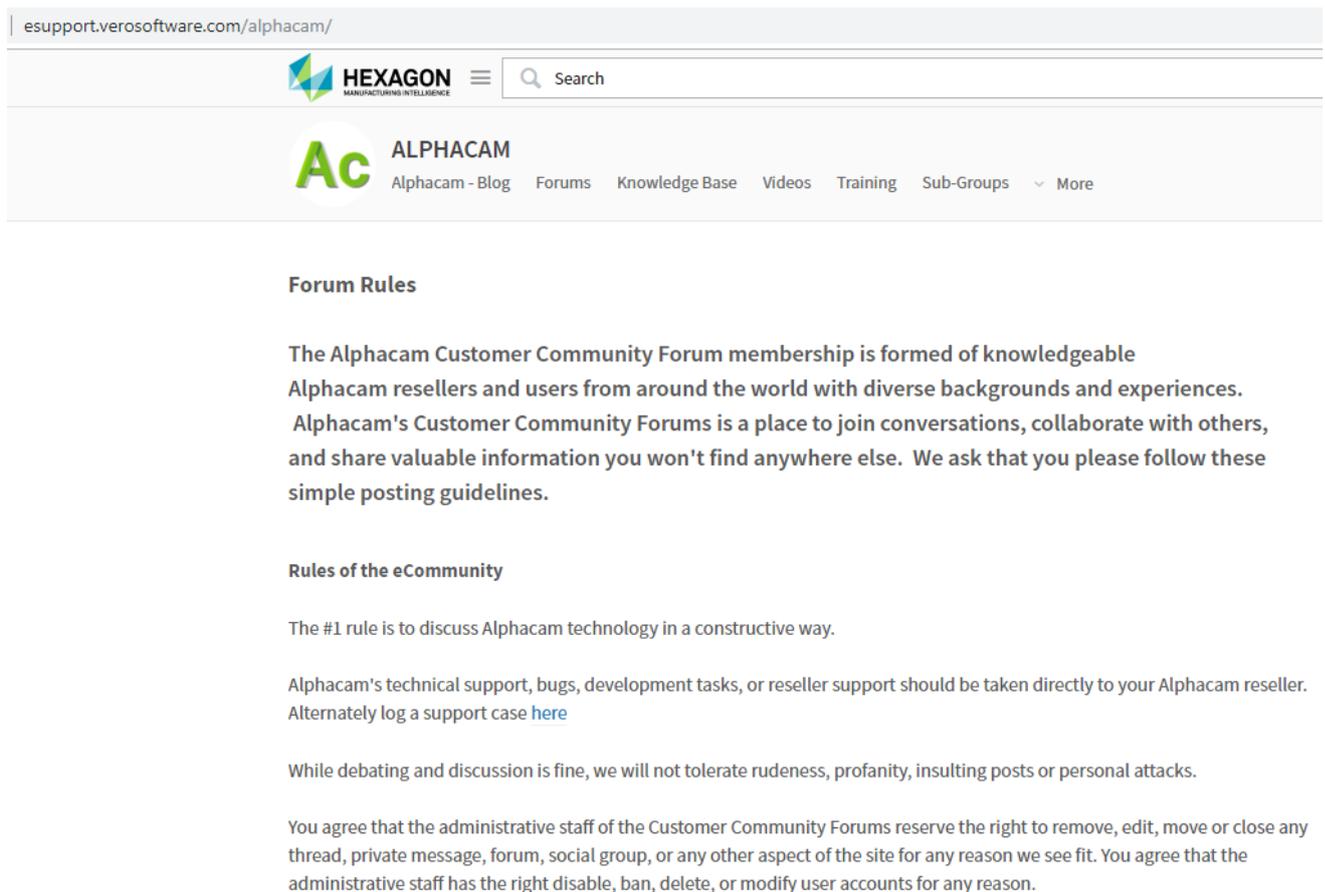


Figure 1 - Customer Support Portal

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

ALPHACAM esupport

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



The screenshot shows the ALPHACAM esupport page. At the top, there is a navigation bar with the Hexagon logo, a search bar, and a menu. Below the navigation bar, there is a section for ALPHACAM with a sub-menu containing links to 'Alphacam - Blog', 'Forums', 'Knowledge Base', 'Videos', 'Training', 'Sub-Groups', and 'More'. The main content area is titled 'Forum Rules' and contains the following text:

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 to disable, ban, delete, or modify user accounts for any reason.

Figure 2 - esupport page

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

Introduction

The purpose of this manual is to introduce the concepts and terminology used in ALPHACAM and to explain the various concepts of machining that will be made available to the user.

In some instances, prior knowledge to manufacturing processes is not required as it is the purpose of the manual to explain these in detail. That is not to say though that working knowledge of your own manufacturing techniques should be applied when required to supplement and compliment the methods used in the ALPHACAM environment.

This manual should be used in conjunction with other supplied documentation to build a complete working reference set for current and future users.

It has been written as a reference manual and not as a definitive manufacturing guide and should be used as such to gain knowledge of the methods to be applied in ALPHACAM.

Where applicable, the pdf to refer to will be indicated in certain sections of the manual to assist in building your knowledge of the processes and methods used in ALPHACAM.

Standard 2D Machining Fundamentals

The first stage of any manufacturing process is to create a method to follow, this may be a complete written instruction set or information gained through years of working practice that is remembered. Using ALPHACAM is no different; there are stages that must be followed so that the part is machined correctly. The following list gives an example of the process flow for creating a finished machined part in ALPHACAM.

Outline Program Production

1. Produce a process plan.
2. Create or Import the required Geometry.
3. Create a material to provide a starting shape for the solid simulation.
4. Reposition the job to a suitable location to fit on your machining requirements.
5. Ensure sound geometry for manufacturing the part.
6. Set Tool Side/Cutting Directions for all profiles, i.e. direction of cut and side of cutting.
7. Set Start Points on all profiles i.e. where the cutter will start on the profile.
8. Select the required post processor if more than one is available.
9. Select the Material cutting conditions (where applicable).
10. Select the tool for the operation; remember to **<LClick>** to accept the tool.
11. Select and action the operation(s):
Remember to **<RClick>** after selecting the profiles to create the tool paths.

Or

12. Select and apply the applicable machining style.
13. Repeat 9, 10 and 11 or 12 for all operations in the Process Plan.
14. Apply any additional lead in and out strategy. (if required)
15. Save the Job.
16. Run the simulation,

Output the NC code, you may use Alphaedit to communicate to the machine tool controller.



It is worth pointing out that some of the command controls detailed in this manual may not be available in the Essential level of ALPHACAM.

Housing Plate Process Plan

OP. NO.	DESCRIPTION	TOOL NO.	TOOL DESCRIPTION
1	Pocket Centre Recess Profile 6 and central flanges Profiles 7 & 8 leaving 0.5mm Stock in X and Y.	1	16mm Flat
	Spiral Pocket Ø20 & Ø30 Holes (Profiles 9,10).	1	16mm Flat
	Rough & Finish Profile 2, 3mm deep 4 XY cuts 34mm Initial Stock.	1	16mm Flat
	Rough Profile 1 Outer Edge 16 deep leave 0.5mm XY Stock	1	16mm Flat
2	Residual Pocket Profiles 6,7,8 leave 0.5mm Stock and Finish Profile to size.	2	8mm Flat
	Residual Rough Profile 1 leave 0.5mm stock.	2	8mm Flat
3	Spot Chamfer Ø8 Holes 1, at Z 0 and produce the 1mm x 30° chamfer.	3	15mm x 120 Spot Drill
	Spot Chamfer Ø6 Holes 3, at Z -3 and produce the 1mm x 30° chamfer.	3	15mm x 120 Spot Drill
	Spot Chamfer Ø10 Circles 5, at Z-3.	3	15mm x 120 Spot Drill
4	Drill Ø8 mm Holes 1, Through.	4	8mm Twist Drill
5	Drill Ø6 mm Holes 4, 10 Deep.	5	6mm Twist Drill
	Drill Ø6 mm Holes 3, 12 Deep.	5	6mm Twist Drill
6	Machine 2mm Radii on top edges. (Profiles 2, 7, 8, 9, 10)	6	2mm Corner Rounding Tool
	Machine 2mm Radii on Profile 6 at -3.	6	2mm Corner Rounding Tool
7	Chamfer 1x45° on Outer Edge Profile 1.	7	90° inclusive Chamfer Tool
8	Engrave Part No.27854	8	30° taper Engraving Tool
9	Finish Profile 1 the outer edge Through using Helical strategy.	2	8 mm Flat tool

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

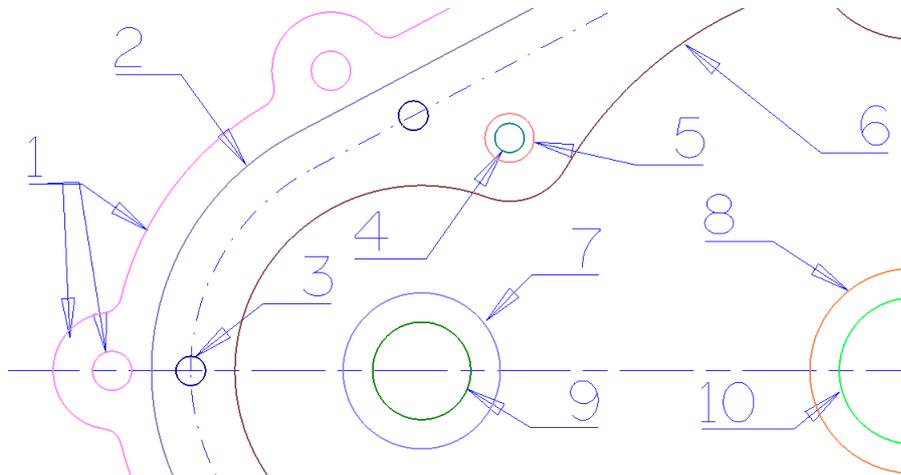


Figure 3 - Housing Plate geometry

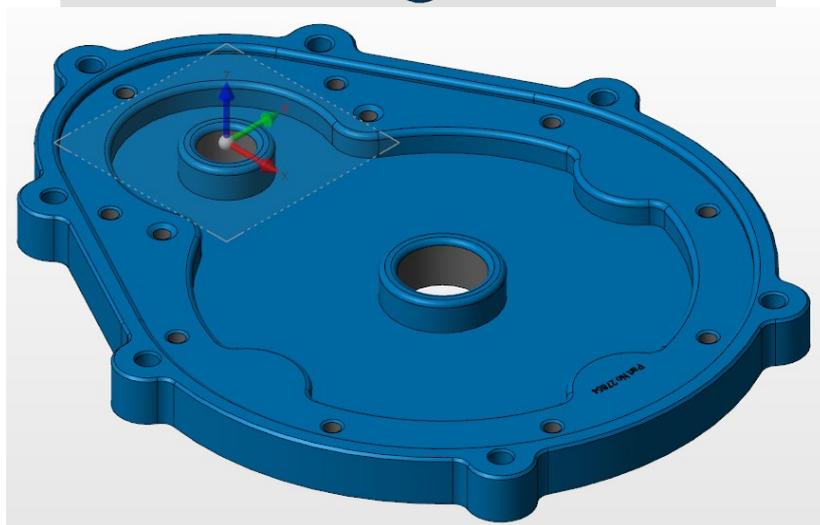
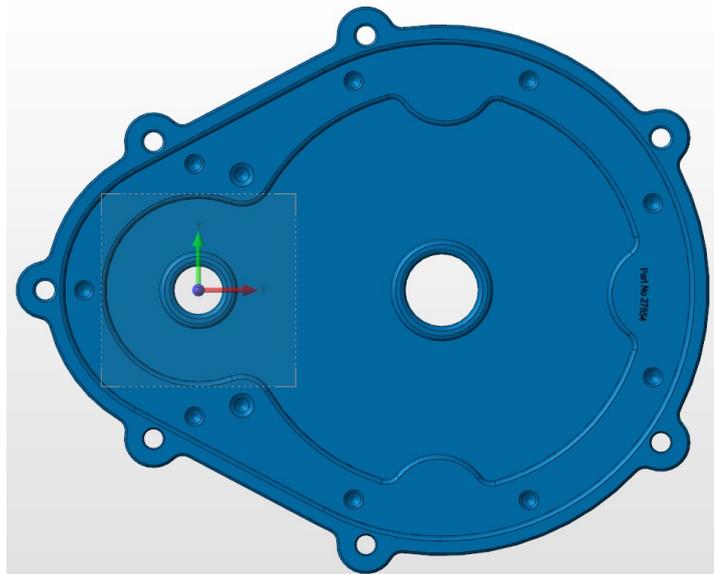


Figure 4 - Housing Plate sample finished solid model

Material Creation

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



Note that neither of the following detailed options for material creation are available in the Essential modules of ALPHACAM.

AutoSet Material

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

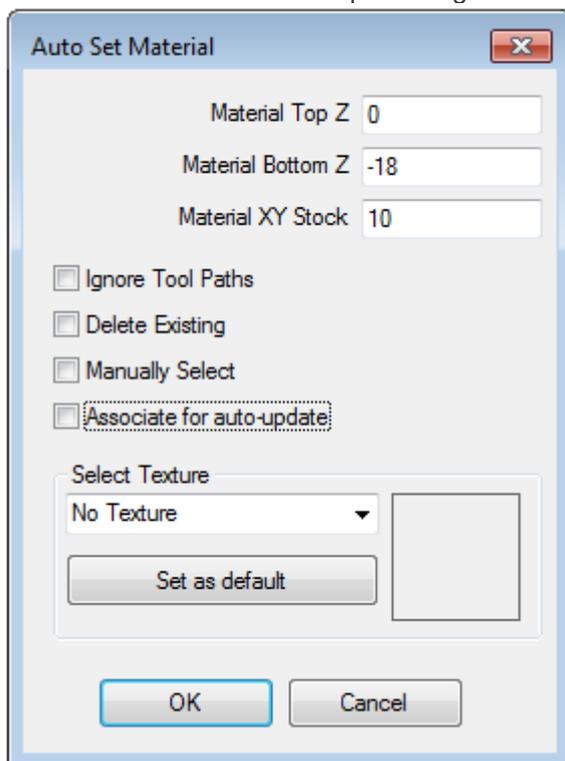


Figure 5 - Auto Set Material dialogue

Make the values in the dialogue as follows,

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

If the option Auto Set Material is not available, use **ADD-INS/Macros > Add ins**



then look for **Create Default Material** on the left hand window, tick this option to add it to the 3D menu.



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

Set Material

Another method is to use **3D > Set Materials** 

With this method, a shape must be drawn around the item to be machined that represents the physical shape of the material to use.

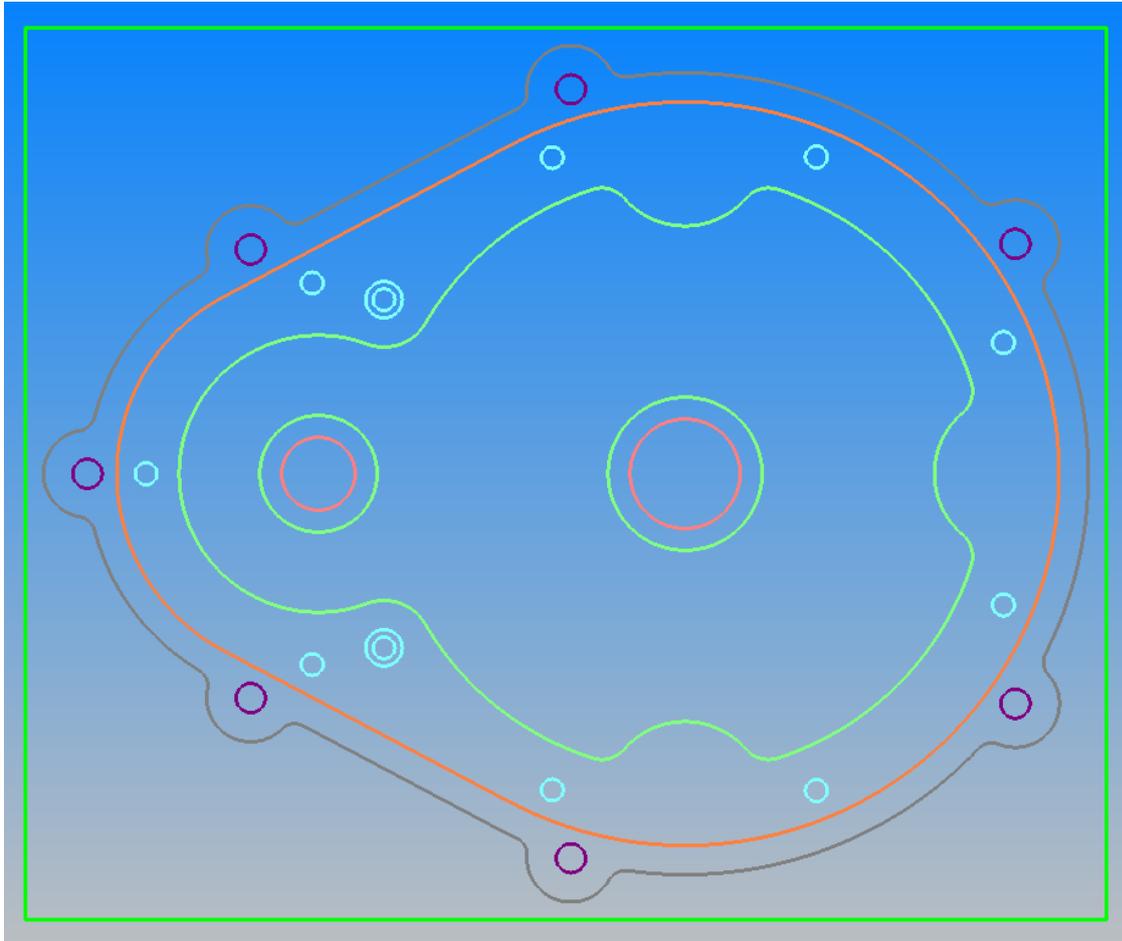


Figure 6 - Material outline in place prior to selection

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

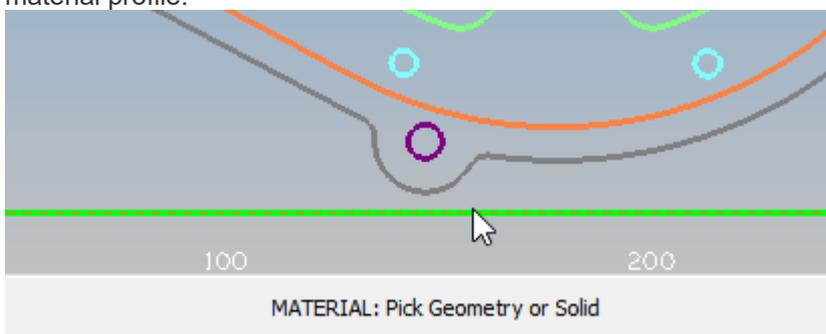


Figure 7 - Select the required profile

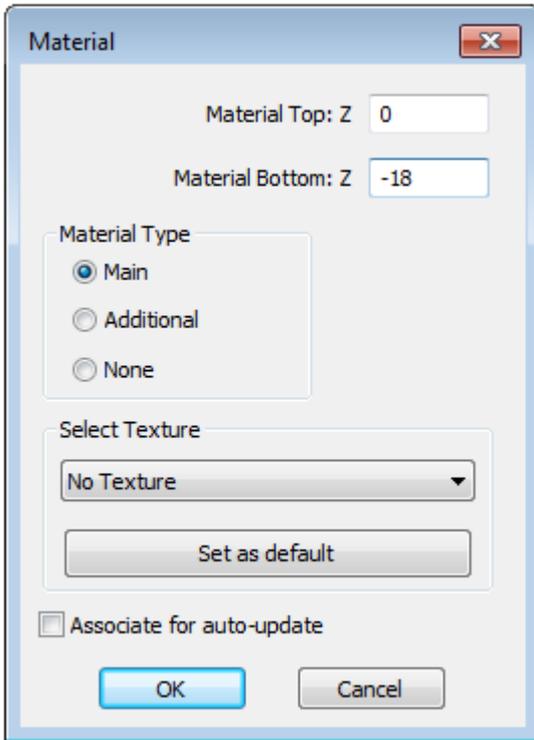


Figure 8 - Set Material dialogue



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



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

Move to machining position

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

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

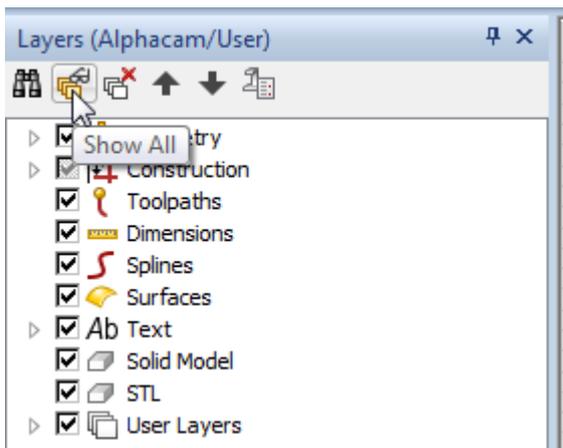


Figure 9 - Show All option

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

Essential Module

As we have seen in the previous Material section, **ALPHACAM Essential** has no ability to create the impression around the part of a material automatically.

There is a simple way to achieve this using the existing geometry options.

Use **GEOMETRY > Special Geometries > Enclosing Rectangle** 

When prompted, **<LClick>** on the outer profile geometry to select.

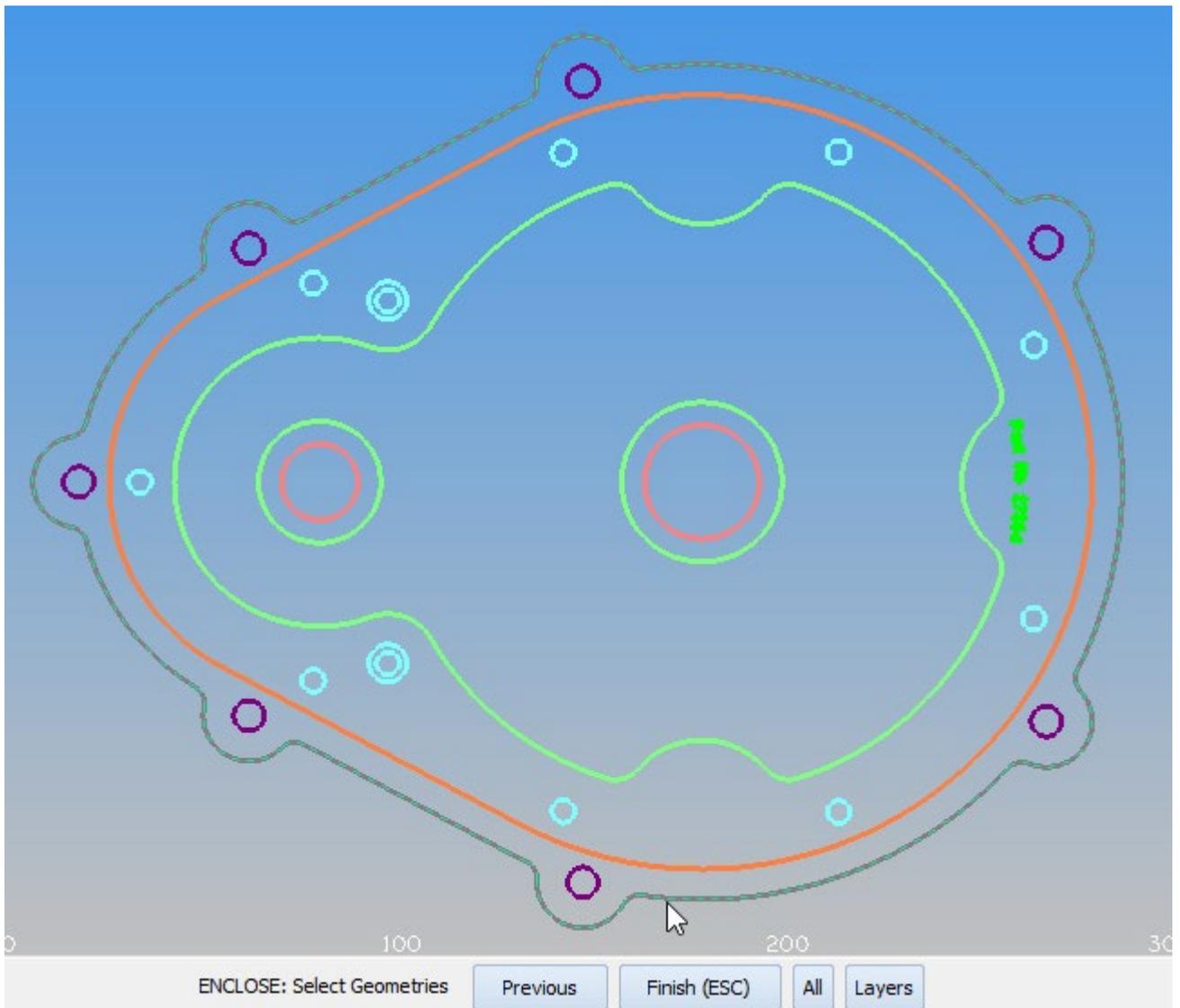


Figure 10 - Geometry for Enclosing Rectangle option

<RClick> to finish.

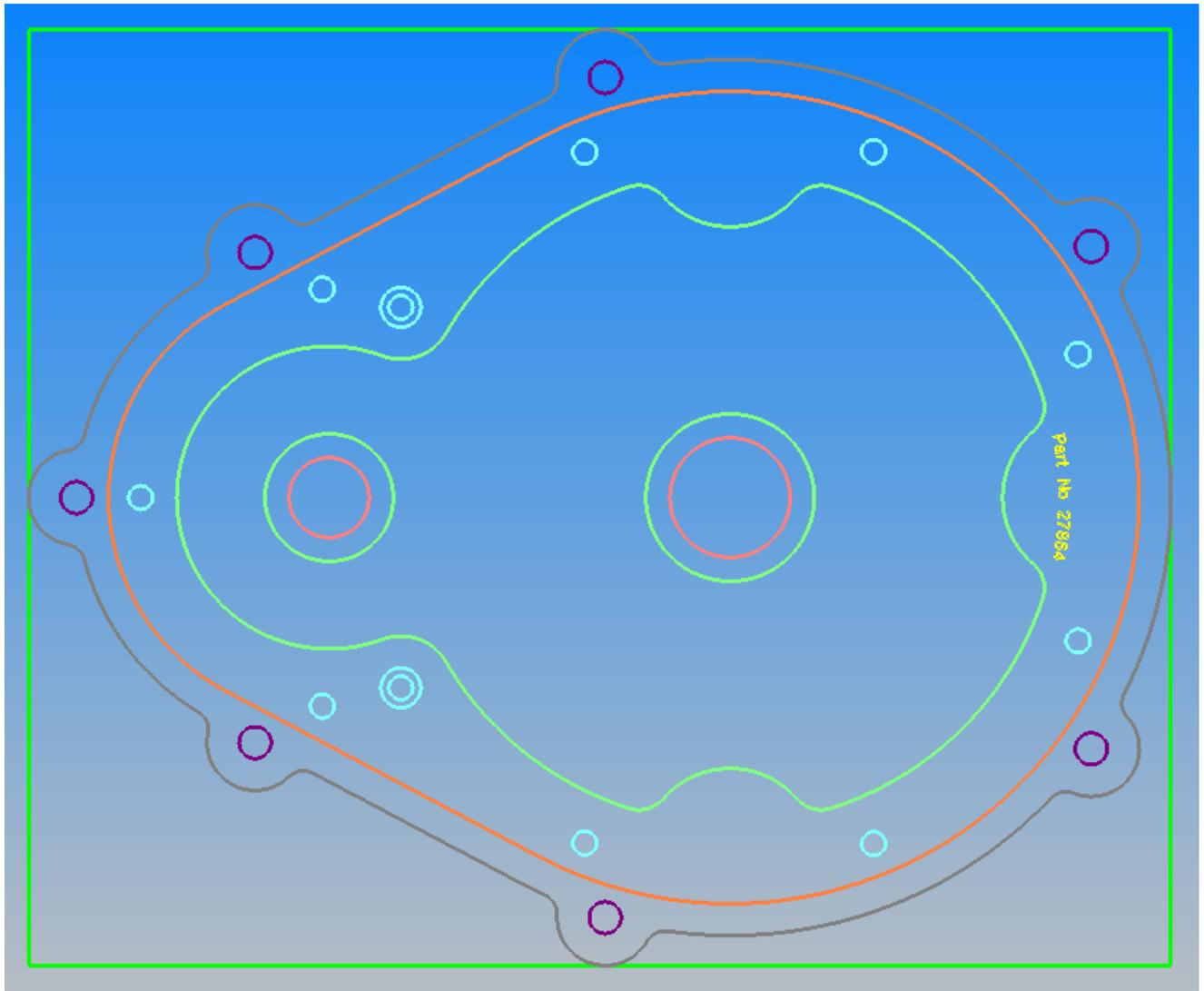


Figure 11 - Applied rectangle

Use **EDIT > Offset** 

Make the options as shown below.

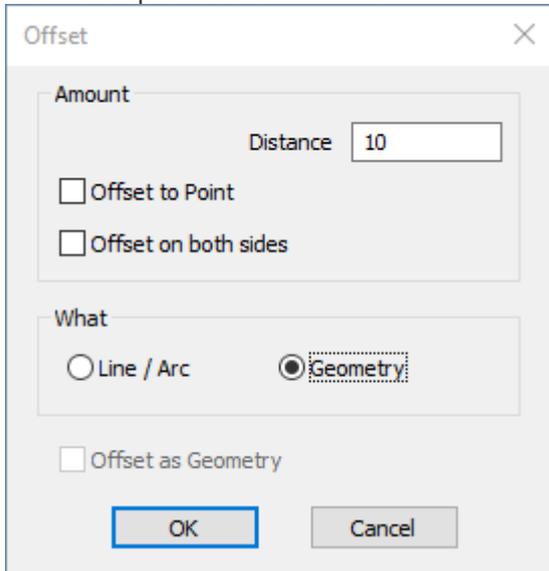


Figure 12 - Essential Offset Options

<LClick> [OK] to continue.

When prompted, **<LClick>** on the enclosing rectangle to select.

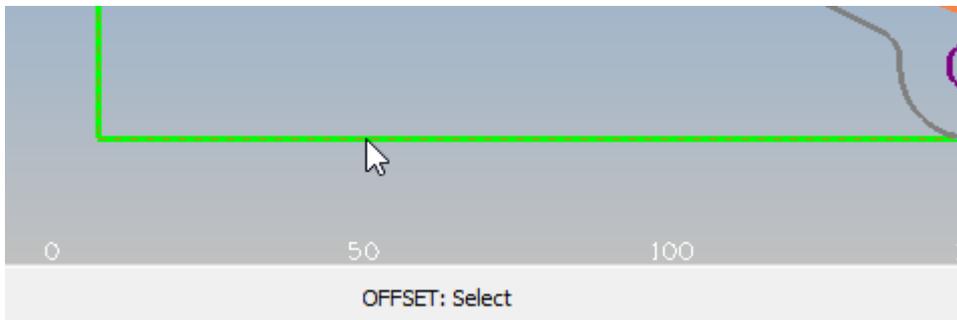


Figure 13 - Select the rectangle to offset

Once chosen, **<LClick>** outside of the rectangle to place the offset item.

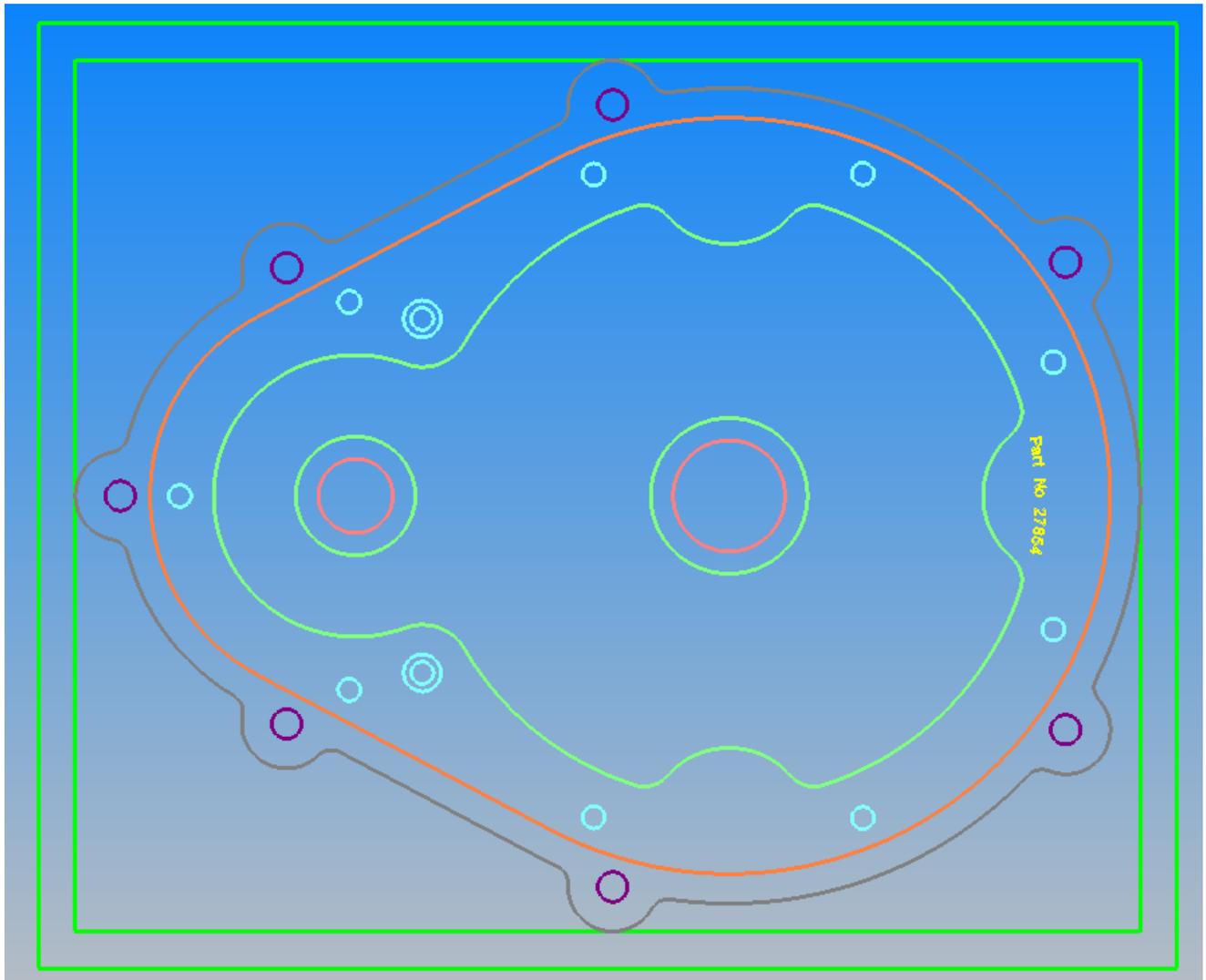


Figure 14 - Completed offset

Use **EDIT > Delete** 

<LClick> on the original inner rectangle as the item to delete.
<RClick> to continue.

When prompted, **<LClick> [OK]** in the deletion confirmation dialogue.
<RClick> to finish.

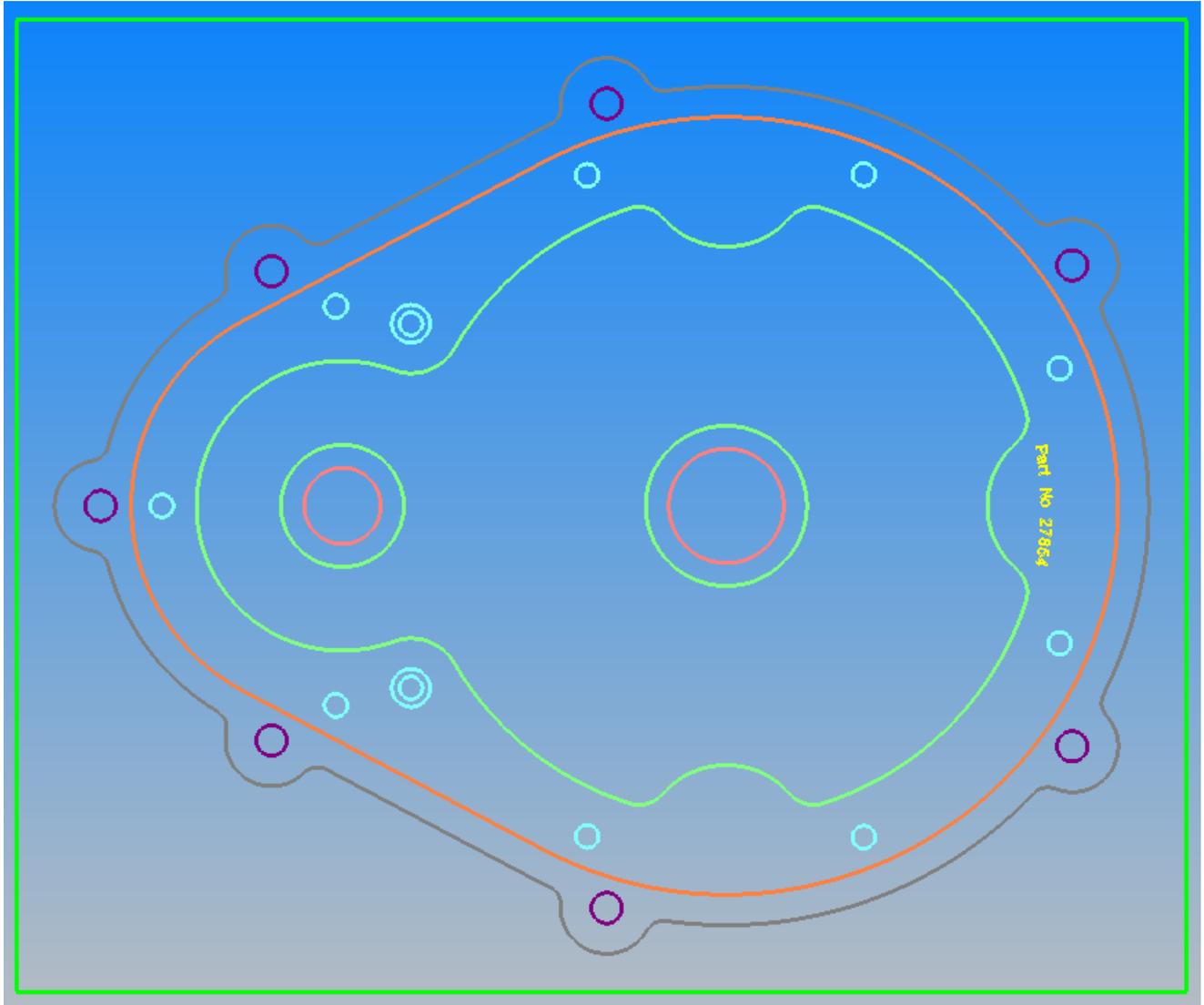


Figure 15 - Essential method ready to move

Use **Show All** to make your entire drawing visible, remember that;

 **That which is not visible will not be acted upon.**

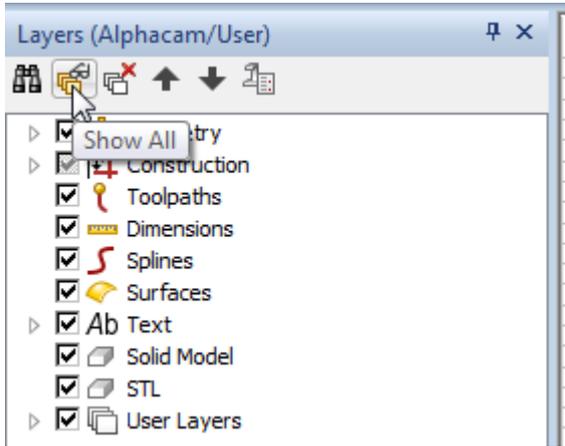


Figure 16 - Show All option

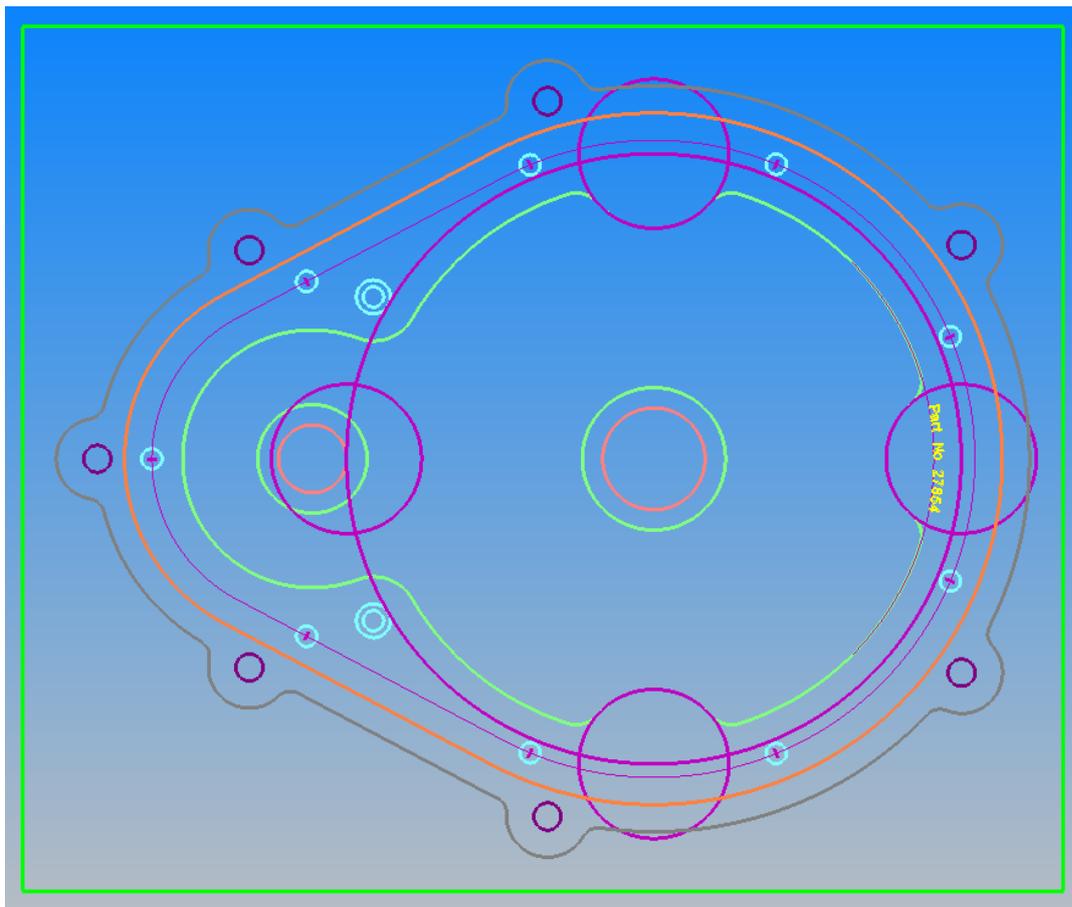


Figure 17 - All visible

Using **EDIT > Move, Copy etc > Move** 

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

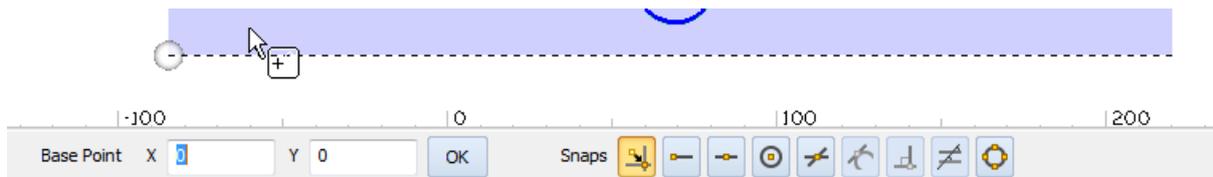


Figure 18 - Base point location prior to <LClick>

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

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

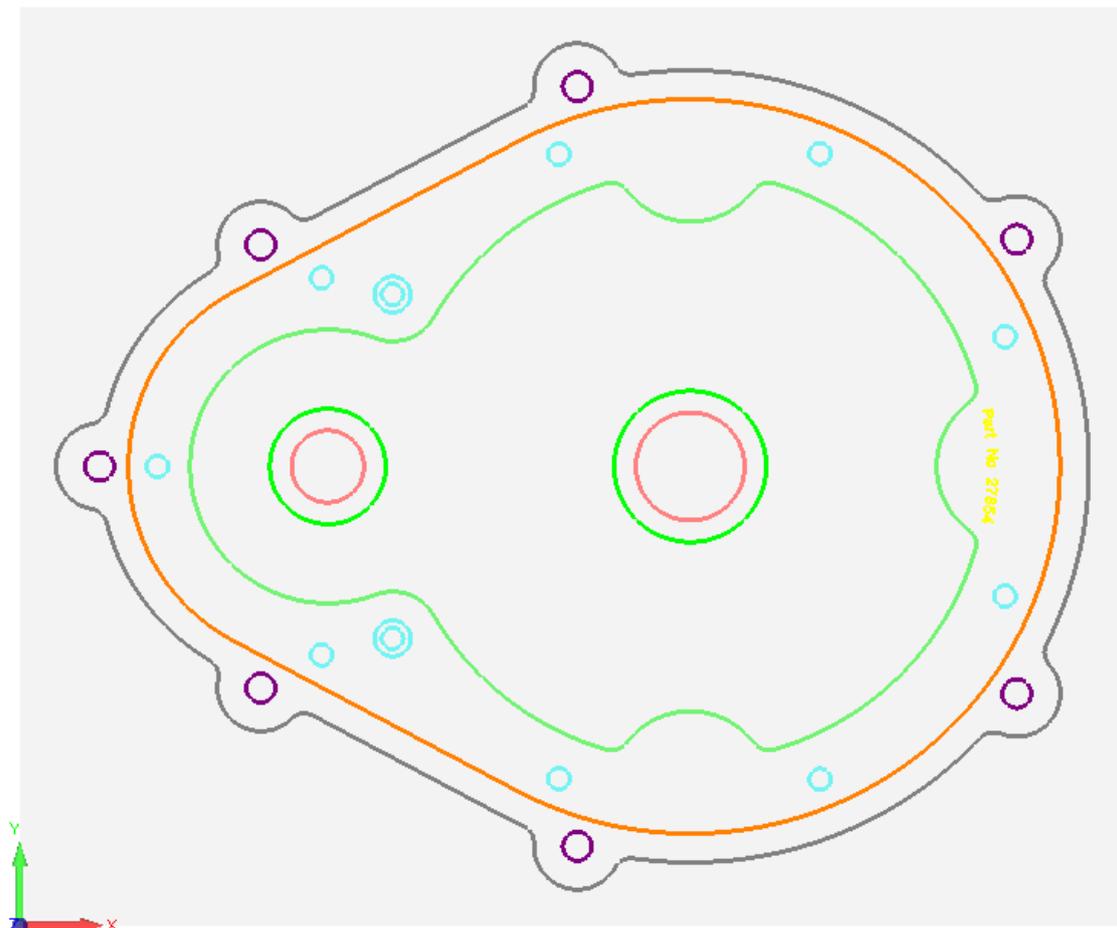


Figure 19 - Housing Plate correctly located for machining

Setting Start Point, First and Order Options

Setting the **Start Point** for Machining is an important step, if this location is set incorrectly it can lead to incorrect machining that could damage or destroy your finished component. Ordering is also very important as ALPHACAM will always machine the selected geometries in the order that they were drawn, which may not be how you wish to manufacture the part.

There are two ways to set the starting point for any geometry:

- Using the **EDIT > Start Order > Start Pt**  .
- Using the **MACHINE > Tool Directions**  dialogue by checking the Set Start Point on closed profile to **Auto**. Or **Manual**.

If you know where you wish to start machining from on a drawing that you have created, then using the first option above is the better method.

You can always reset the start location after placing toolpaths on the job as you may not know where the original start point is on a drawing that someone else has created and you may not know you need to alter it until after you have applied the toolpaths.

Start Pt

The **EDIT > Start, Order > Start Pt**  command allows you to change the start point on a closed geometry or tool path. The start point is indicated by the position of the ghost tool arrow; this is where the tool will start from.

To see the ghost tools,

- Select **VIEW > Display Options > Ghost Tools** 

or

- Press **<Ctrl> + <G>** on the keyboard.

When the command is actioned, you are prompted to select the new position for the start point. The start point can be selected as the picked point or by use of a snapping function on the geometry.



Note that there is no requirement to select the geometry first and then state the Start Point, the action of setting the Start Point automatically selects the geometry.

First ¹²³

The **EDIT > Start, Order > First** ¹²³ command allows the user to set the first geometry in a pattern or set of tool paths.

When the command is actioned, the dialogue allows you to set if the command is being applied to **Geometry** or **Tool Paths**. The command line then prompts you to select the **First Item**.



Geometries are ordered as they are drawn or duplicated.

Order ¹²³ →

The **EDIT > Start, Order > Order** ¹²³ → command allows the user to adjust the order of a set of geometries or toolpaths. When the command is selected, the order dialogue is displayed.

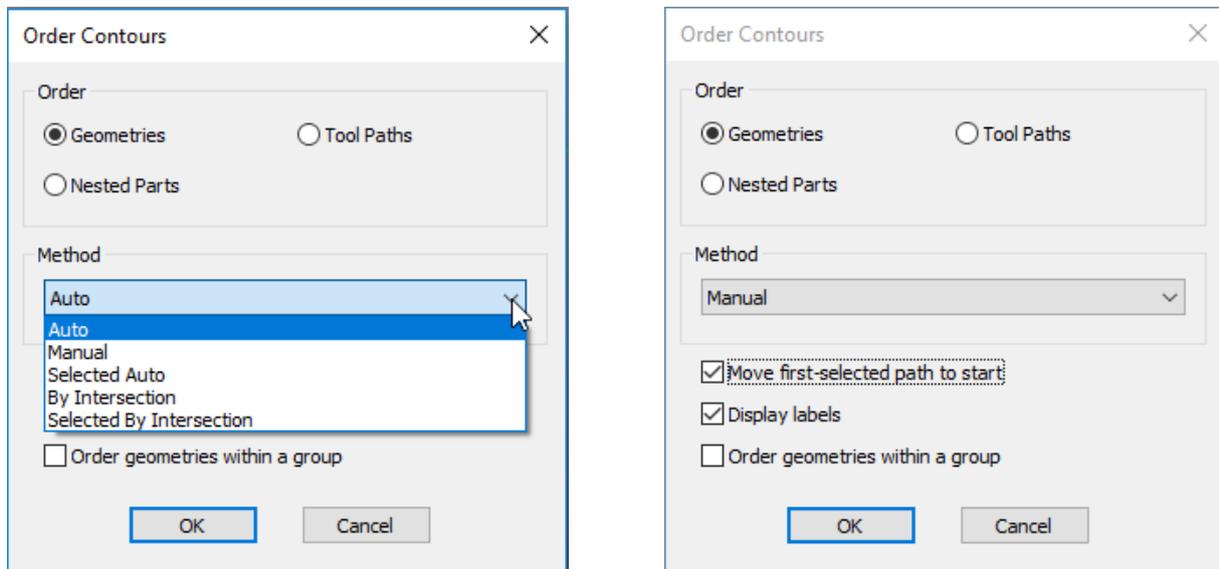


Figure 20 - Order dialogue options

This dialogue has two sections:

The top section is what you want to order **Geometries** or **Tool paths**.

The drop down section is for the method of reordering,

- Auto
- Manual
- Selected Auto
- By Intersection
- Selected By Intersection

Move first-selected path to start allows for the initially selected Toolpath, when using the **Selected Auto** option, to be the first in the list.

Setting this active, the first item you choose becomes the first item in the chosen set, inactive, you will be asked to select the **First in the series** once you have completed your choices.

Display Labels allows for a visual indicator of the order of all visible items to be shown.

Order Geometries within a Group allows for the manipulation of Grouped entities without having to Ungroup them first

Order Geometries Auto

This setting will automatically reorder all the visible geometries. The visible geometry with the lowest identification number will be the starting geometry; the second geometry is set by whichever start point is nearest to the first, the third geometry is set by whichever start point is nearest the second, etc.... The first geometry and tool path can be set using **EDIT > Start, Order > First** 

Order Tool Paths Auto

This setting will automatically reorder all the visible tool paths of the same tool. The first machined visible tool path will be the starting tool path, the second tool path is set by which start point is nearest to the first, the third tool path is set by which start point is nearest to the second, etc.... The reorder will only consider tool paths machined with the same tool.

Order Geometries & Tool paths Manual

This setting allows the user to manually pick the order of the geometries or tool paths. This could be used if the machining is required to be kept further apart to reduce heat or swarf build-up.

Order Geometries Selected Auto

This option does not automatically select all the visible geometries but allows the user to select the geometries that are to be considered in the reorder routine. E.g. where a pattern of holes requires reordering but additional visible geometries are to be ignored.

The user selects the required items to re-order, then on completion of the selection process ALPHACAM will prompt the user to indicate the item to become the new first in the list. After this the process of next closest start point is applied as per the Automatic option details earlier.



If a pattern of holes is to be drilled etc. using subroutines, it may be necessary to set the order of the geometries as subroutine tool paths cannot be reordered.

Order Geometries By Intersection

Using a single geometry passing through a selection of geometries, this command will automatically identify the intersecting geometries and order those in the direction that the guide geometry has passed across them.

Order Geometries Selected By Intersection

Allows the user to identify which intersecting geometries are to be ordered, then these are ordered in the sequence that they intersect.

Tool Directions

Before you apply any machining strategies, it is necessary to set the **Ghost Tools** on your geometries to the correct side for the machining operation.

When geometry is created the default ghost tool setting is to **Centre** in the same manner as drawing with a pencil.

In this instance if you were to machine the part, the centre line of the tool cylinder would sit directly on the line and would cause an outer profile to be cut too small by the size of the tool radius, and a pocket area to be cut too large by the tool radius.

To compensate for this ALPHACAM uses the **MACHINE > Tool Directions ** command to assign specific sides of a geometry to place the tool on as well as the option to alter the direction of travel via clockwise (**CW**) or anti clockwise (**CCW**) options.

It is also applicable on closed profiles to set the ghost tools position, as this relates to where the cutting operation will start from on the profile.

Direction of Machining

There are two specific conventions when talking about the direction a cut will take, these are Climb and Conventional machining.

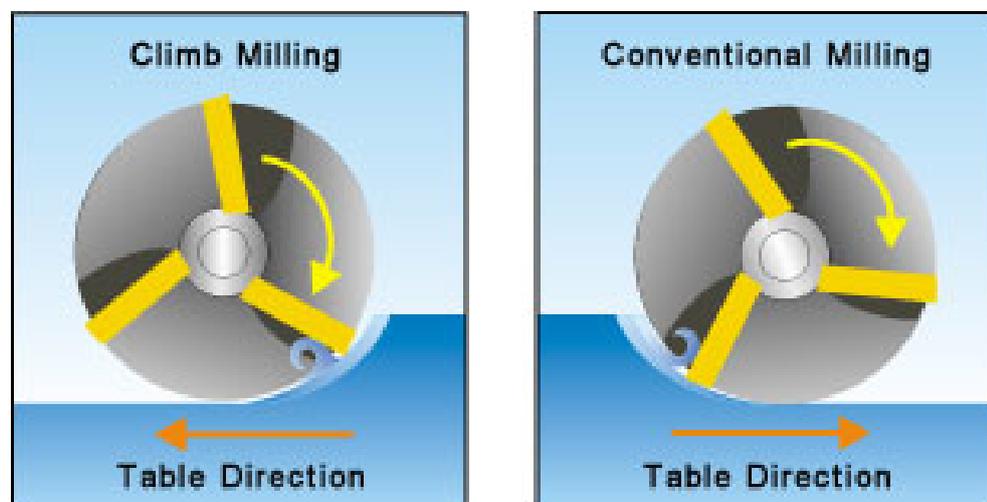


Figure 21 - Climb and Conventional cutting

Climb or Down cutting creates a cut that is at its thickest as the tool breaks into the material and as the table or head moves along the profile, the thickness of the chip decreases. This type of cutting method is predominantly used in the metalworking industries to produce good quality finishes.

Conventional or Up cutting creates a cut that is at its thinnest at the start of the tool working into the material and as the part is moved, so the thickness of the chip increase. This type of cutting is used to good effect in the woodworking industries where the material machines better using this method.

Tool Directions Dialogue

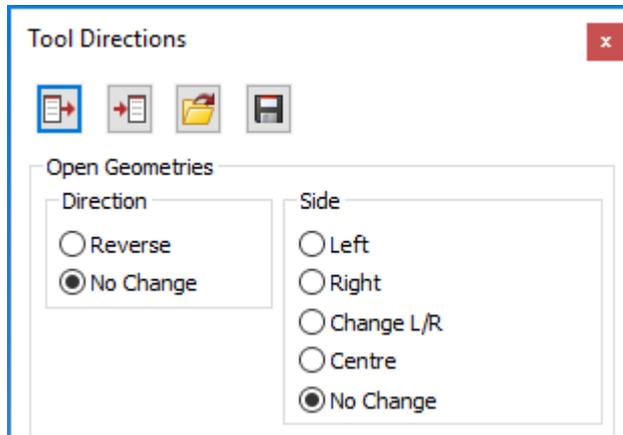


Figure 22 - Tool Directions for Open geometries

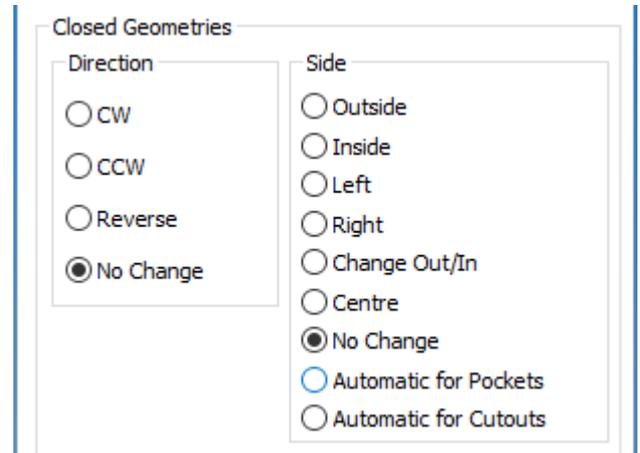


Figure 23 - Tool Directions for Closed geometries

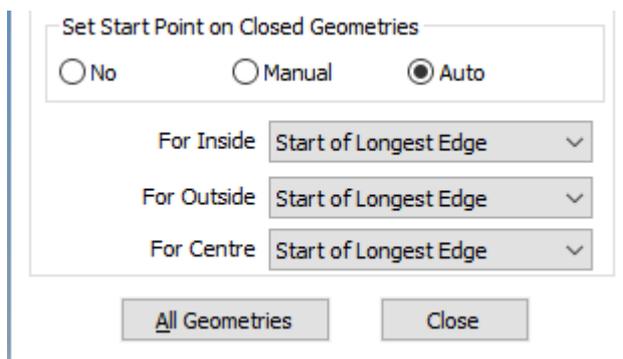


Figure 24 - Start point options

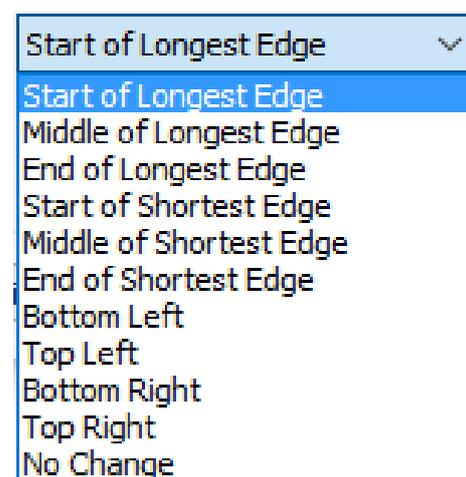


Figure 25 - Auto Start point locations

Set the options in the dialogue first, then pick the geometries to apply the options to.

If required, reset the options and pick the geometries to apply the new settings to.

When all tool directions have been applied select **[Close]** to remove the dialogue.

Hard Boundaries

In most instances, tools will be required to create machined profiles within specific areas, for example pocket areas or outer profile areas. The types of boundaries used in these methods of machining are referred to in ALPHACAM as **Hard Boundaries**.

Hard boundaries are those geometries which have their ghost tool set to one specific side of the geometry and the tool is not allowed to pass across this boundary.

Soft Boundaries

Another type of boundary is the **Soft Boundary**.

Soft boundaries are geometries whose ghost tool remains on the centre of the created geometry and the tool can pass onto this item.

When setting the ghost tools for **Open Geometries** use;

- Reverse to change the direction and start point.
- Left or Right to set the side for machining.

When setting the ghost tools on **Closed Geometries** use: -

- ⤴ CW Inside = Conventional machining.
- ⤵ CCW Inside = Climb Machining.
- ⤴ CW Outside = Climb Machining.
- ⤵ CCW Outside = Conventional machining.

Default Options

Using the series of icons along the top of the dialogue can assist in repetitive jobs.



Figure 26 - Setting Default options



Get Default Automatically loads the last pre-set options created using the Stored option.



Store Default Automatically stores the current settings to be used as the Default options.



Load Load a previously Saved set of options.



Save Saves the current options as a dedicated file that can be loaded when required.

Saved strategies are located in “.....LICOMDAT/ToolDirectionsData”

The format of the saved file uses the ***.atoldir file extension**.

Original Ghost tools

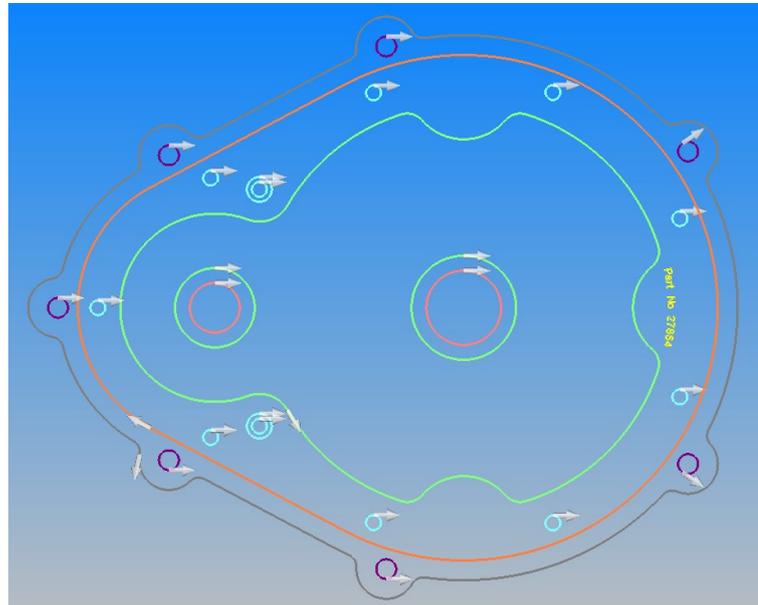


Figure 27 - Ghost tools as created on the Housing Plate

Set the Tool Directions

Select **Automatic for Pockets**, then select **[All Geometries]**.

Then set **Change Out/In** and select the outer profile 1 and the oval 2 to switch the Ghost Tool to the correct side.

Using the **Automatic for Pockets** option is a quick way to select many geometries and apply the settings. In some cases, though, it may be better to use the **Outside** or **Inside** options and select the geometries individually.

Selection of either **CW** or **CCW** for cutting directions is dependent on your material type and manufacturing processes.

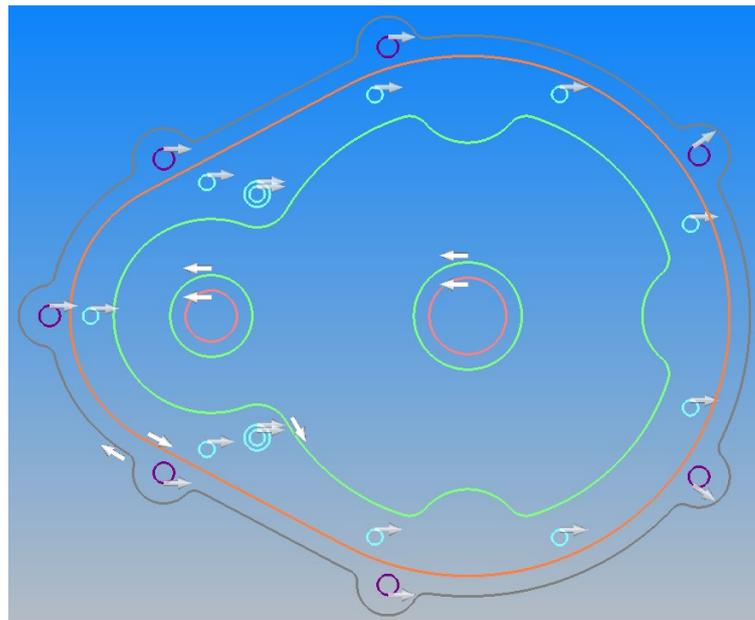


Figure 28 - Ghost tools set correctly for machining

Set the Start Points

Set the start points on the profiles to suitable positions.

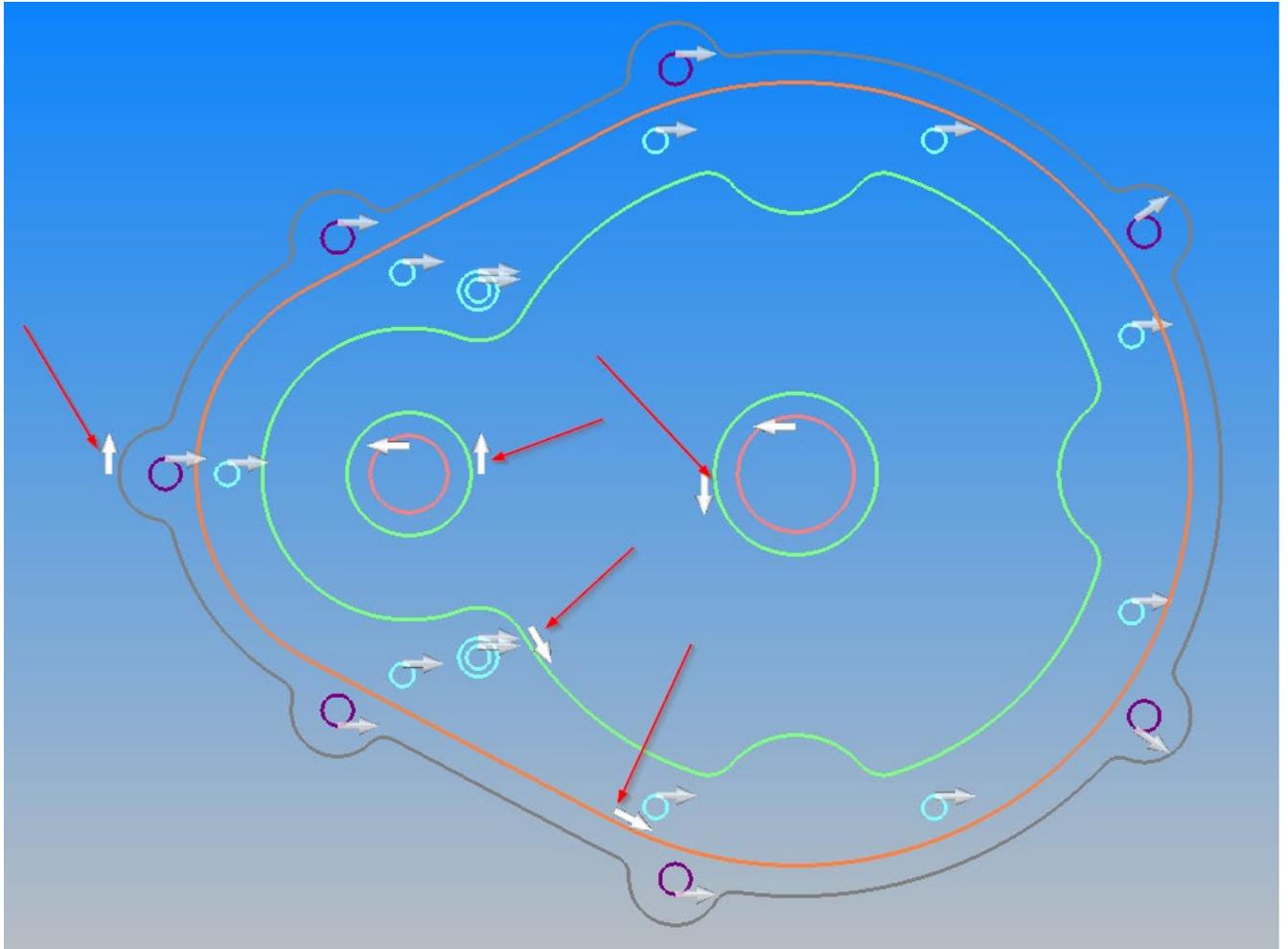


Figure 29 - Geometry start points set to suit good manufacturing



It should be noted at this point that any geometry feature that would be created using a drilling process, the $\text{Ø}8\text{mm}$ and $\text{Ø}6\text{mm}$ circles on the above example, do not require the tool side setting as drilling does not make use of the line side but of the centre of the circle.

Fundamentals Conclusion

From the previous set of general options, the process to follow becomes clear.

- Create a material if required.



Note that this option is not available in the Essential level of the product

- Set the part to the correct location for manufacture.
- Set the correct **Tool Direction** and **Side** to machine on.
- Set the **Start points** to a more beneficial location if required.

With the bottom two points in mind, if you know where you wish to start machining from, set the start location during the Tool Direction process.

If you are unsure, apply the toolpaths first, then run the simulation to see if there are any errors and if so, adjust the Start Point after applying the toolpaths.

The software is so flexible that even if you have forgotten to set the correct Tool Directions and have already applied a tool path, you can reset the Tool Directions and simply update the toolpaths after.

Common Geometry Problems

Red Geometry

When setting tool directions using Inside or Outside, ALPHACAM checks the geometry for gaps. If it is not a closed profile the system turns the profile **Red** (when using the default colour palette) and displays this error.

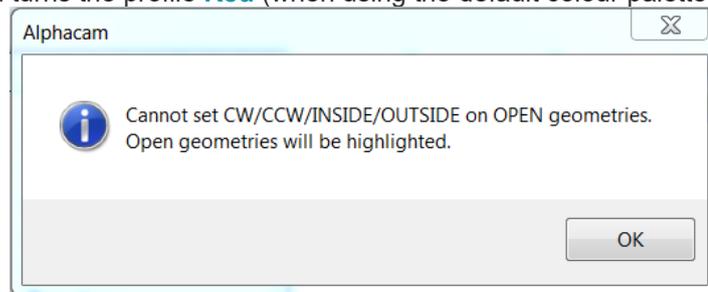


Figure 30 - Error displayed on Open geometries when setting Inside/Outside

A closed profile is one where the start and end points are coincident, the above error will occur when the start and end points are not the same. You can sometimes see the gap at the starting position, unless the ends overlap.

You can check the elements in a profile using the project manager.



If the geometries overlap, explode the profile and delete or trim the overlapping element, and then join all the geometries together to create one complete joined profile.

No Ghost Tools visible

This occurs when the geometries are not on the currently active work plane. Ghost tools are only displayed for the current work plane. elements assigned to the **Construction** layer will not display a ghost tool, as construction geometry cannot be machined.

Tools on Centre Error Message

The following dialogue is displayed when tool paths are applied to geometries when the tool side is set to centre line but the requirements for the cycle must be on one side or the other of a geometry.

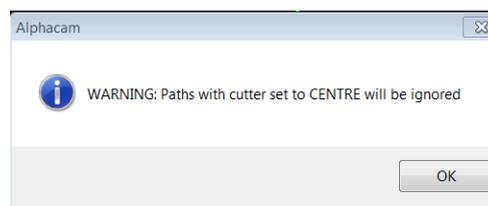


Figure 31 - Toolpath on Centre error dialogue

This means the tool will over cut the geometry because the tool centre will be positioned to the geometry. Pockets will be too big; outline profiles will be too small.

Area Clearance, Standard Pocketing

All area clearance tool paths are created with the **MACHINE > Pocketing**  options. Pocketing creates NC code to remove material from within a closed geometry boundary. The method used to create the tool paths is selected from the dialogue options.

Types

The first set of options allows you to select the area clearance method to be used.

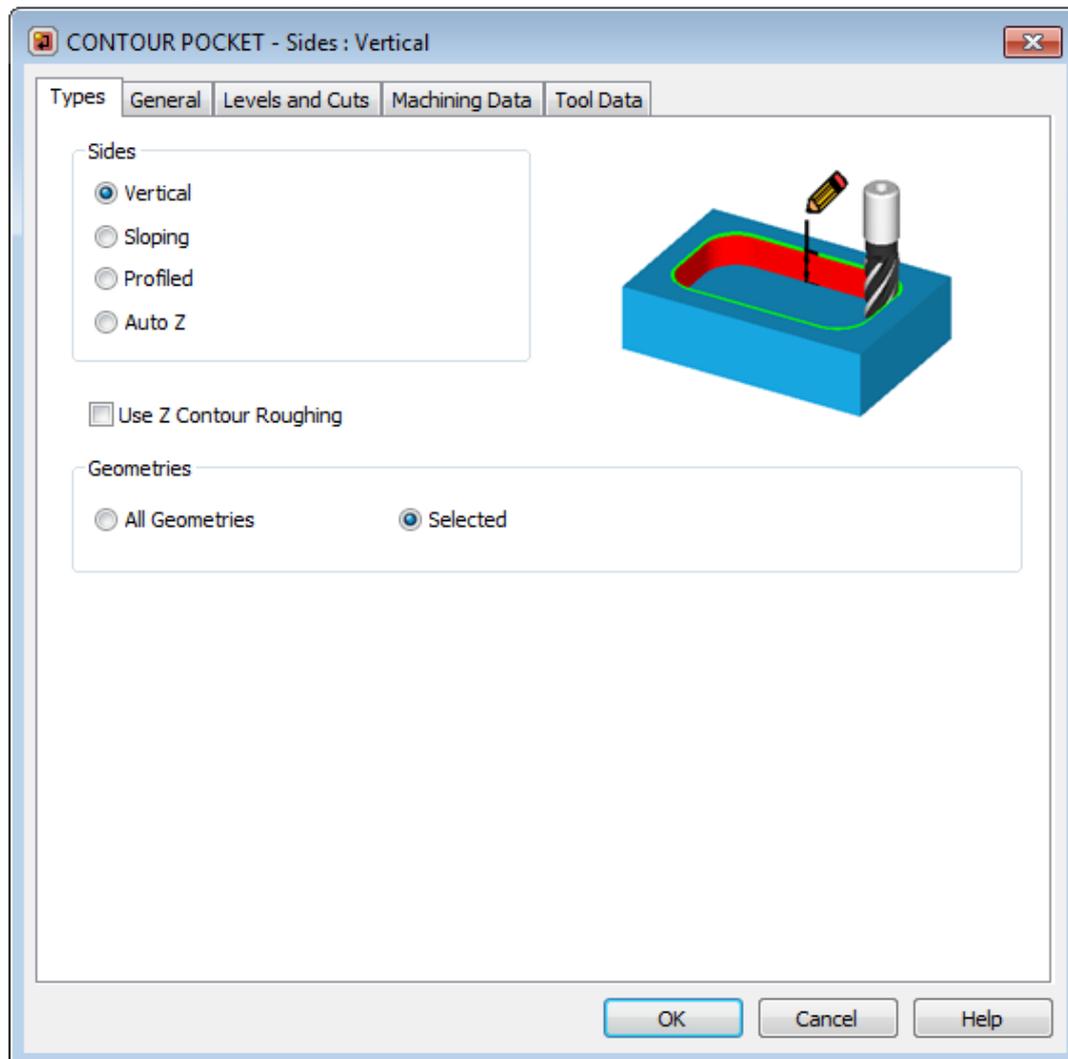
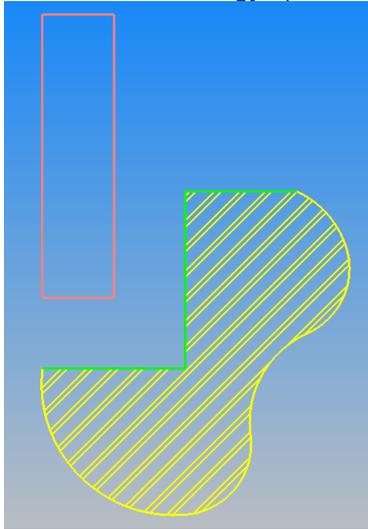


Figure 32 - Contour Pocket Types

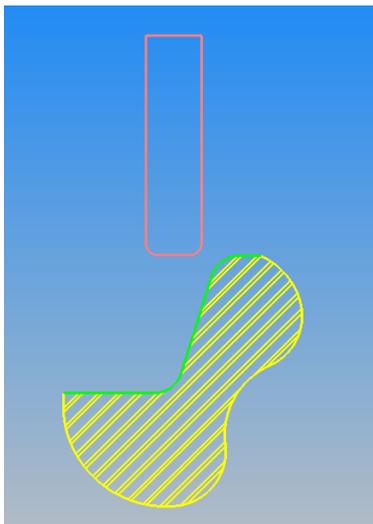
Firstly, you select the side and strategy option:

Vertical



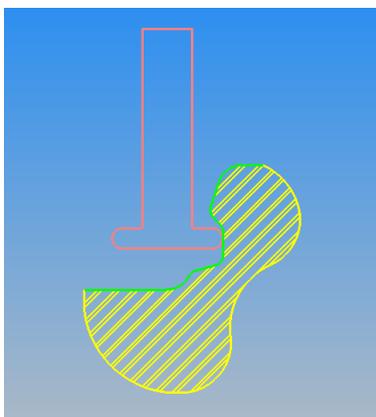
Walls are machined with the sides of the chosen tool and the depth values are assigned within the cycle. Usually created with a flat tool.

Sloping



Walls are machined with a defined draft angle to a specific depth assigned within the cycle. Created with either a Bull nose or Ball ended tool type.

Profiled



Walls are machined using a specifically created 2D geometry side profile, the depth of cut controlled via the size of the cross section. Created with either a Bull nose or Ball ended tool type or in certain qualifying circumstances, using the Special Drop down menu on the Tool Definition page.



Note that in certain instances, undercuts as shown cannot be created.

Auto Z

Walls are machined with the sides of the chosen tool and the depth values are assigned to the actual 2D geometries.
Usually created with a flat tool.

Use Z Contour
Roughing

Vertical, Sloping and Auto Z can all be enhanced by using this option which allows for greater customisation of the Pocket cycle.

 How to machine with sloping and profiled sides are covered later.

 Secondly, you select which geometries are to be used All Geometries or Selected ones.

Auto Z geometries are normally created from solids using a process known as **Feature Extraction**. Although the Z levels can be set on normally created geometries using

3D > Set Geometry Z Levels 

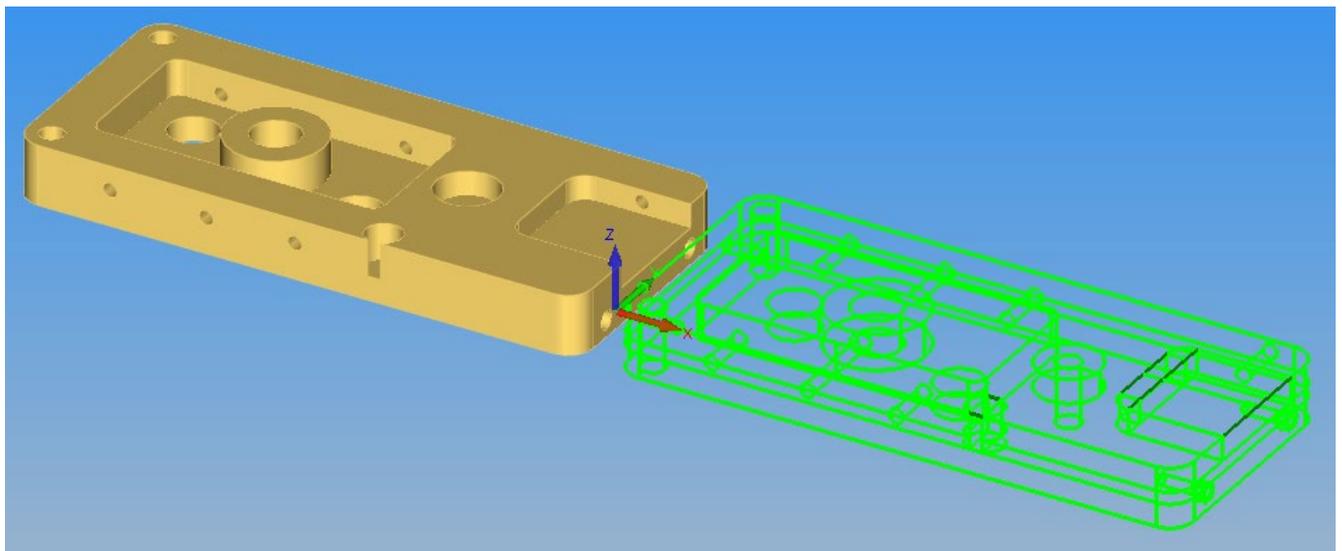


Figure 33 - Example of Feature Extracted geometries

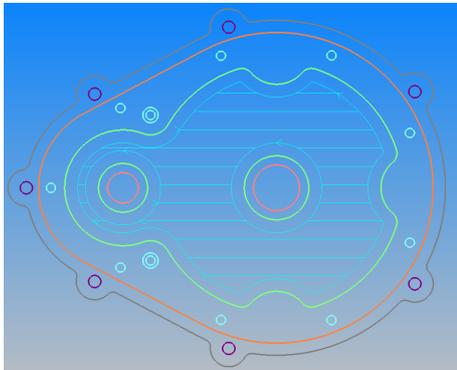
Depending on the type of work you are undertaking may have an influence on whether you use the **Vertical** option or the **Auto Z** option.

Examples of the Pocketing tool path patterns generated for each option.

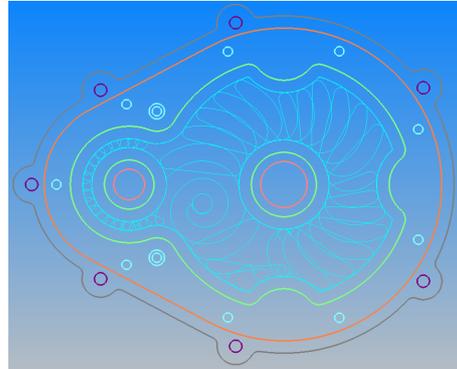
Use Z Contour Roughing

Z Contour Roughing

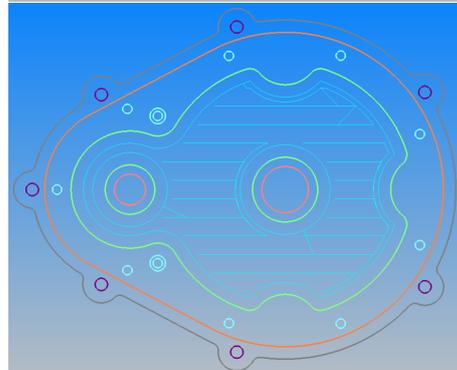
Linear



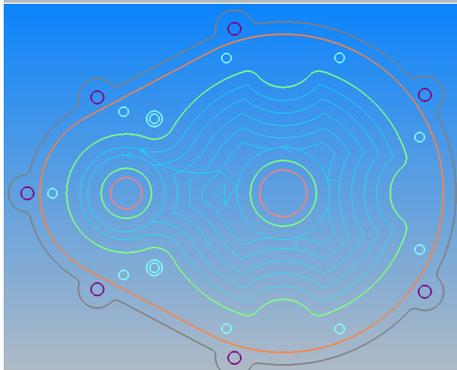
Waveform



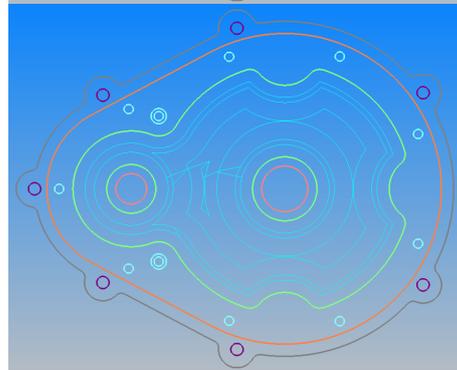
Linear



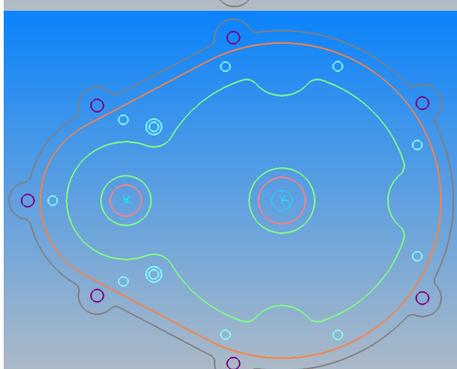
Contour



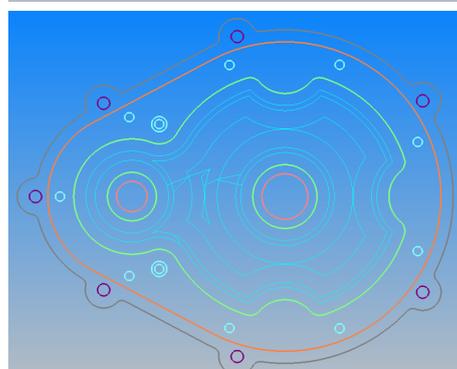
Contour



Spiral



Spiral



General

These are the settings which will be displayed if you are not using (or your module does not have) **Z Contour Roughing**.

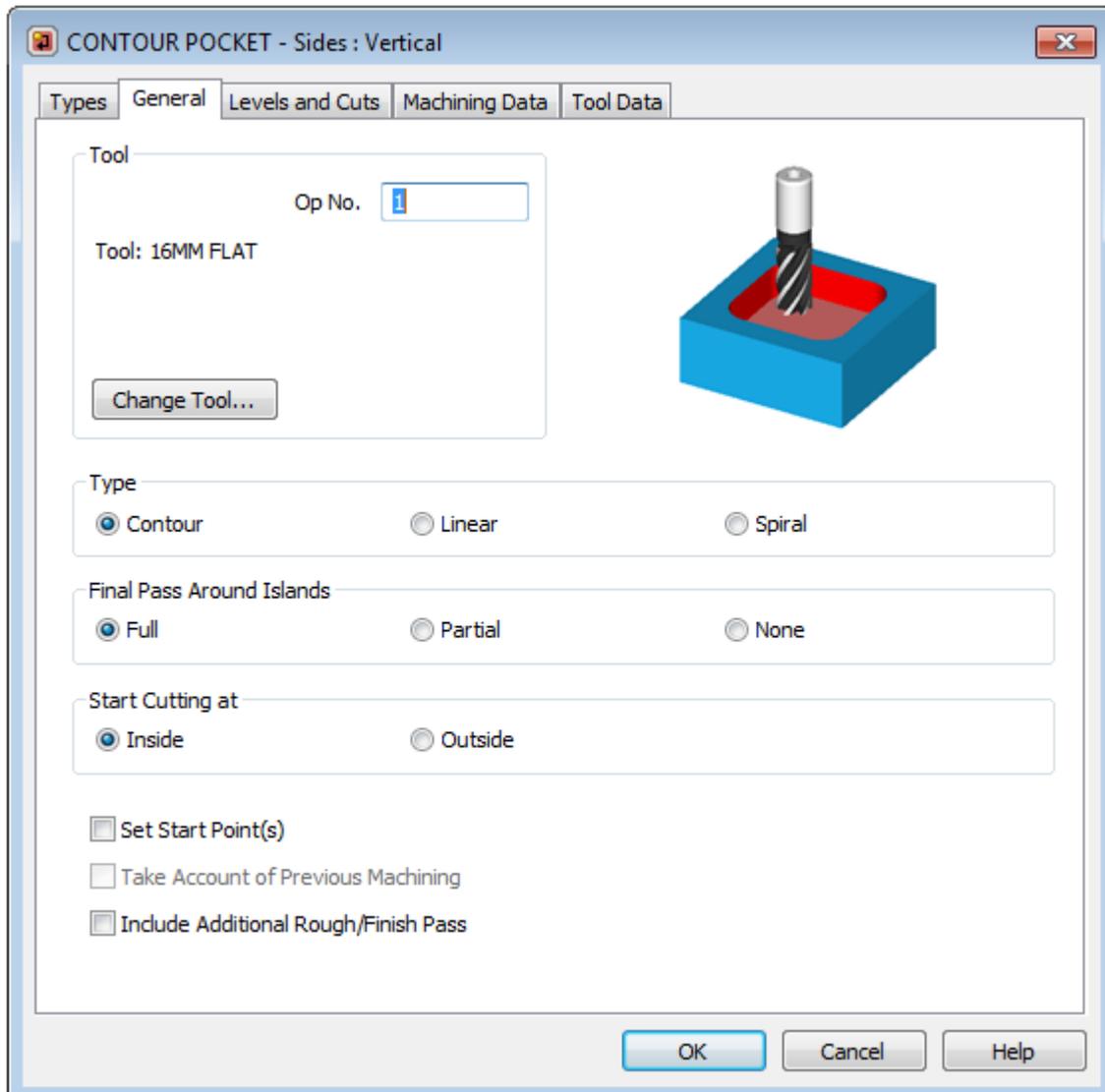


Figure 34 - General tab of the Pocketing cycle

This allows you to specify the pattern of the area clearance **Contour**, **Linear**, **Spiral**, how to treat islands **Full pass**, **Partial pass**, **None**.

There are additional items that allow you to pick a starting position where holes may have previously been created, and when the same area clearance is being actioned with smaller tools it takes into consideration the material already removed in a previous pocketing cycle, potentially saving time on the part.

Contour or Linear

Contour or Linear patterns with these options there can be many islands within the outer boundary; the islands can also have islands inside them.

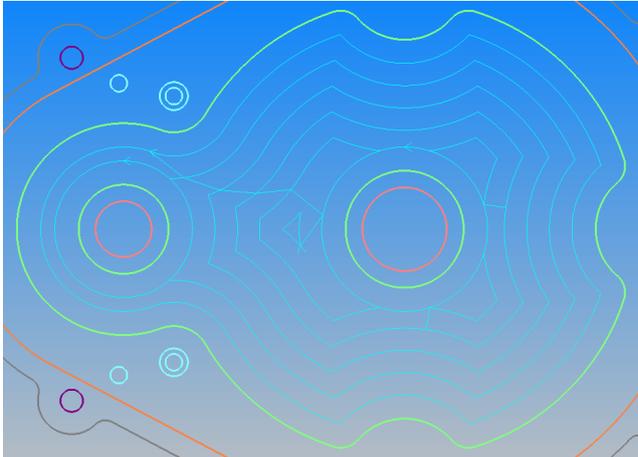


Figure 35 - Pocketing using the Contour option



Figure 36 - Pocketing using the Linear option

Spiral

The spiral pattern, performed by the standard calculating method can only be used to area clear circles without islands either starting from the inside working out (normal) or from the outside working in.

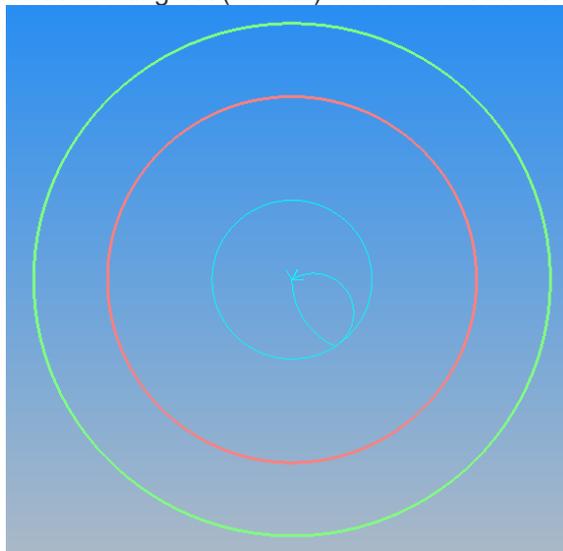


Figure 37 - Pocketing using Spiral option

Take account of Previous Machining

Checks what has been machined previously and only cuts where material is left.

Include additional Rough/Finish Pass

This option appends a **Rough or Finish** cycle to the end of the pocketing tool path, the tool can be changed if required and cutter compensation can be used.

Levels and Cuts

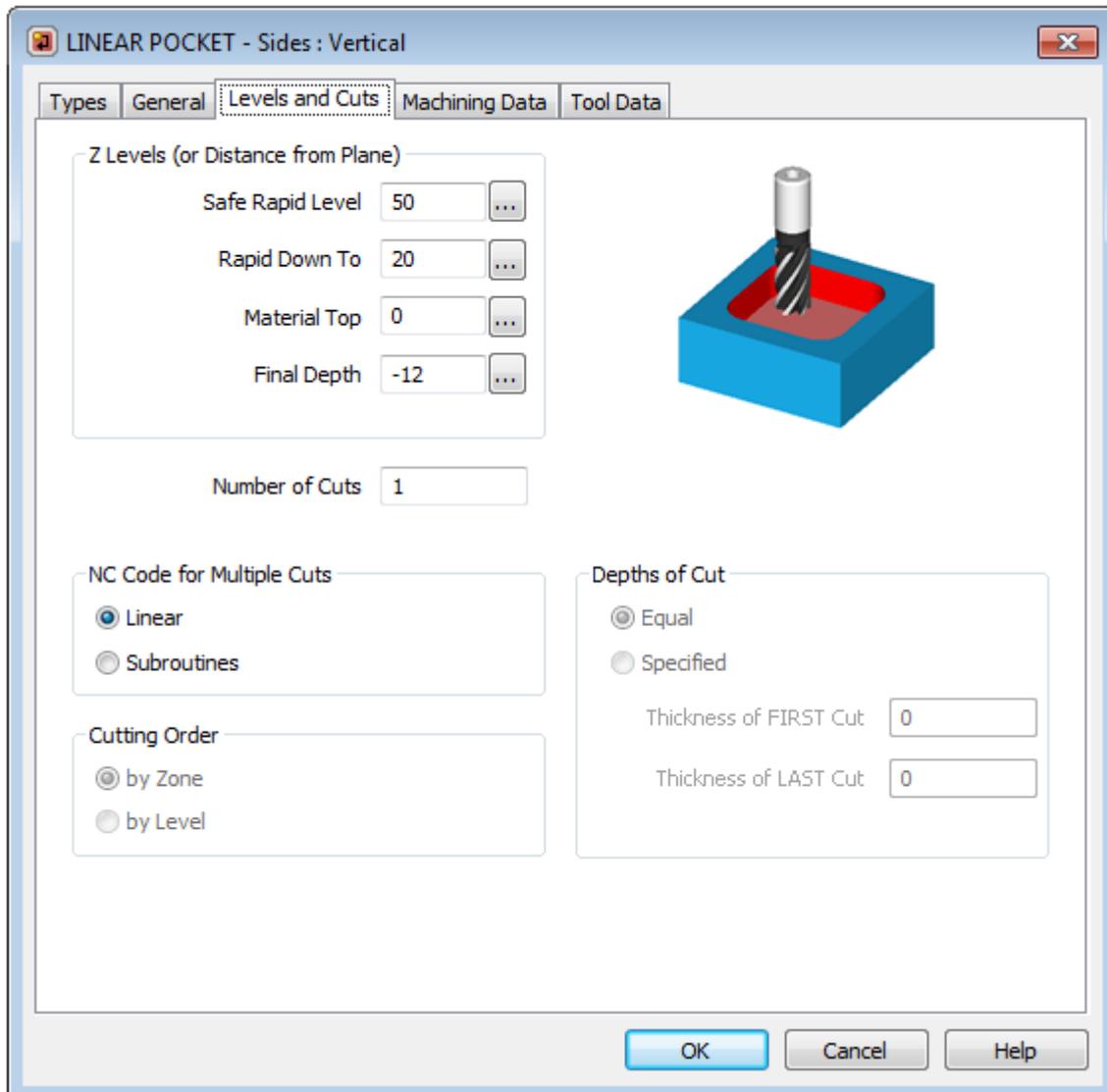


Figure 38 - Levels and Cuts tab of the Pocketing cycle

This allows you to set the;

- Z values relating to the pocketing cycle,
- The number of cuts,
- The NC code output options and cutting order,
- The depth control of multiple cuts.

Z Levels (or Distance from Plane)

Safe Rapid

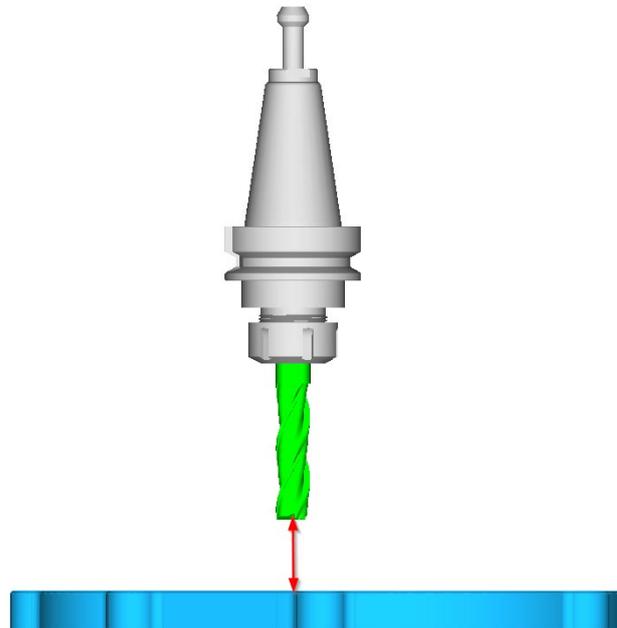


Figure 39 - Safe Rapid position in Z

The clearance distance above everything in the machining area so that the tool can pass across all items without any collisions.

To save entering the values time and again, consider setting a default option in **HOME > Configure > General** and look to the **Machining** entry dialogue options.

Rapid down To

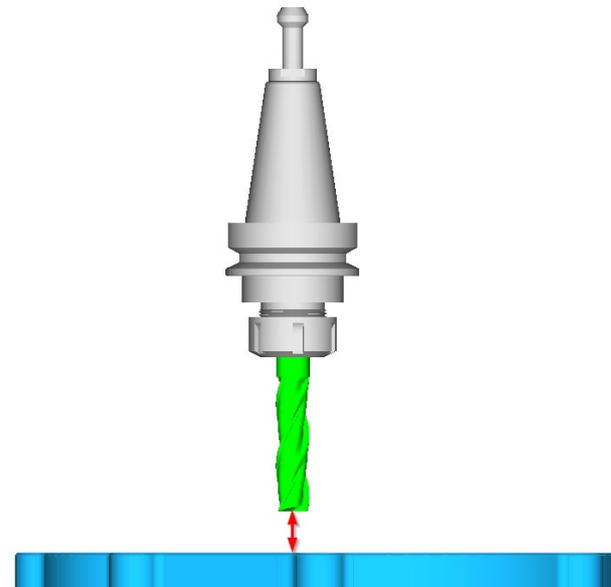


Figure 40 - Rapid Down To position in Z

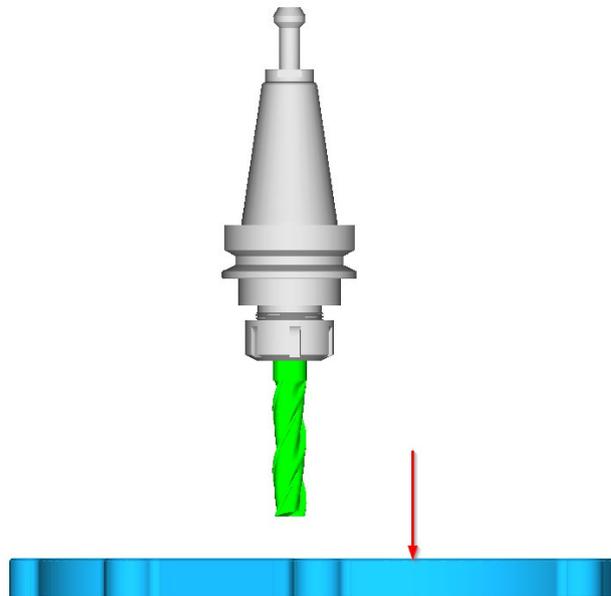
The positioning distance above the part to which the tool will fast traverse and position in the Z axis prior to the commencement of any machining.

To save entering the values time and again, consider setting a default option in

HOME > Configure > General and look to the **Machining** entry dialogue options.



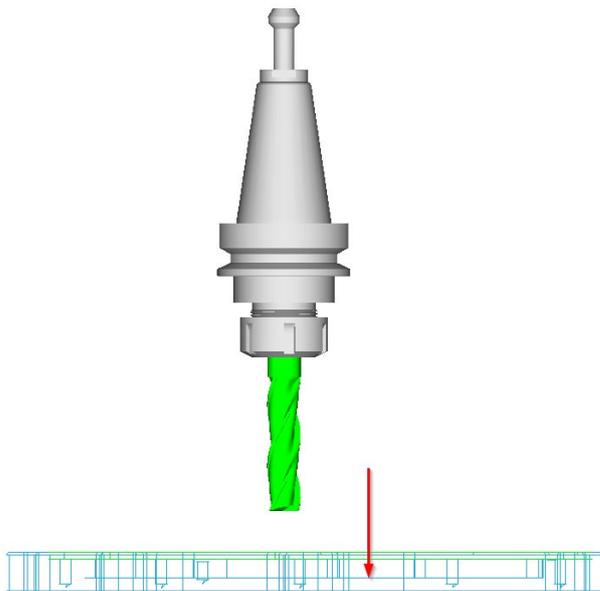
Material Top



The point in the Z axis from which the depths of the machining operation will be calculated from.

Figure 41 - Material top in Z

Z Final Depth



The position in the Z axis as an **Absolute** figure from the Z datum to machine the part to the correct depth values.

Figure 42 - Z Final Depth

Fixed Z options

Safe Rapid level	The Z level where the tool can safely rapid in X&Y without fouling any item or fixture.
Rapid Down to	The Z level that the tool Rapids to before it starts feeding into the material.
Material top	The Z level of the Top of the material being cut.
Final Depth	The Z level which specifies the final depth that the tool will cut at.

Auto Z options

Safe Rapid distance	The distance above the top Z value set on the geometry where the tool can safely rapid in X&Y without fouling any item or fixture.
Feed Down Distance	The distance above the top Z value set on the geometry that the tool rapids to before it starts feeding into the material.
Max Depth per Cut	The maximum depth of cut that will be used where possible. The value will be populated if it has been set in the tool definition.
Min Depth per Cut	The optional minimum depth of cut that will be used.

Number of Cuts

This is the number of step down cuts in the Z axis to be used to machine from the material top to the final depth value. This is not applicable when using Auto Z as the number of cuts is determined by the max depth of cut and the Z depths on the geometry.

NC Code for Multiple cuts

This option allows you to specify how the area clearance tool paths are output.

- Linear Creates NC code in a single program file. Each clearance area cut down in the Z axis is repeated.
- Subroutine Creates a separate sub section or program file for the pocket cycle and this is then called by the main program.

Subroutine output is only applicable with vertical sides, as all the other options cause the pattern at each cutting level to change. If this tool path is duplicated it will automatically be reset to linear code.



Also worth noting is that your machine control must support the use of Subroutines in any form to be able to use this method.

Cutting Order

The cutting order can be set to either **by Zone** or **by Level**.

The **by Zone** option will cause the area clearance tool paths to complete each area clearance to its full depth before creating the area clearance tool paths in the next area.

The **by Level** option will cause the area clearance tool paths to complete all areas at the first cutting level, then restarting with the first area completing all areas at the second level and so on. Any moves between areas will be at the Safe Rapid Z. The **by Level** methodology is normally used where there is a thin wall between pockets, or the pockets are deep and cutting each level allows the swarf/dust to be extracted.

The **by Level** methodology is used in the Aerospace Industry for machining items such as Wing Spars sometimes known as water line machining.

Depths of Cut

This option is only applicable when more than 1 cut has been entered in the **Number of Cuts**, each of the cuts may be made equal in depth if you click the **Equal** radio button.

If you wish to specify cut thicknesses for the first and last cuts, select the **Specified** radio button and enter the required values in the relevant entry fields.

If you have specified more than 2 cuts, the first and last will be as specified and the remaining cuts will divide the balance of the total cut depth into equal cuts in Z.

Slope and Profile Parameters

When using the Sloping and Profiled side options you are requested to set the **Maximum Error** (the step out between cuts) and the **Max depth of cut** (the maximum depth the tool can cut).



The Max Depth of Cut also has a direct impact on the finish of the part so using a large figure is best for roughing and a small figure for finishing.

Machining Data

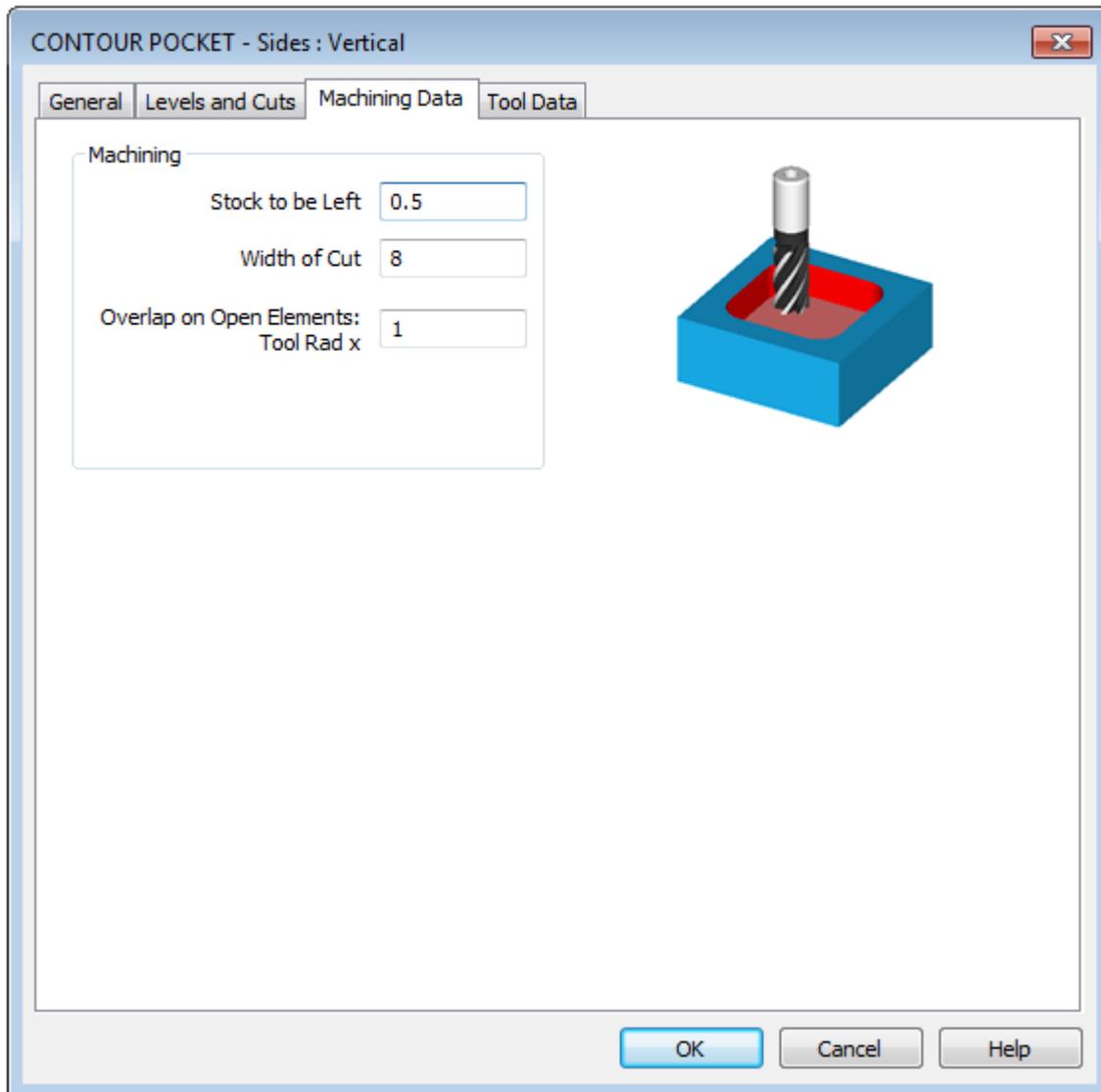


Figure 43 - Machining Data tab on Pocketing

Allows you to adjust the **Stock to be Left** (in X & Y), **Width of Cut** and, if being utilised, the option of how far to cut past a geometry that represents an open sided pocket using the **Overlap on Open Elements** option.

The Pocketing tool paths are calculated in a way to minimise tool lifts and rapid moves. In cases where the areas to be pocketed are complex, the tool can move up to 2x tool diameter along paths already cut, but if that distance is exceeded, the tool will lift to the Safe Rapid plane and rapid to the new position to start cutting.

Pocketing tool paths are always output to **Tool Centre, G41/42** compensation is not used, and tool lead-in/out cannot be added automatically. You can apply lead-in/out moves manually to all the cuts. If you require to apply finishing cuts using cutter compensation to the pocket sides you need to leave a stock allowance and use the **MACHINE > Rough or Finish** 

Set/Unset Open Elements

The **MACHINE > Set/Unset Open Elements**  command allows you to identify elements in a hard boundary that are to be treated as open elements.

A **Hard boundary** is one where the ghost tool is **not set on centre**, so that the cutting action will not overlap the geometry.

A **Soft boundary** is one where the ghost tool is **set on centre**, so that the cutting action overlaps the geometry.

The **Set/Unset Open Elements** command allows you to select elements in a hard boundary so that they allow the tool to pass across the elements. Typically, this is used in the case of open pockets, where one side of a pocket is designated as open so that the pocketing tool path will overlap at this side.

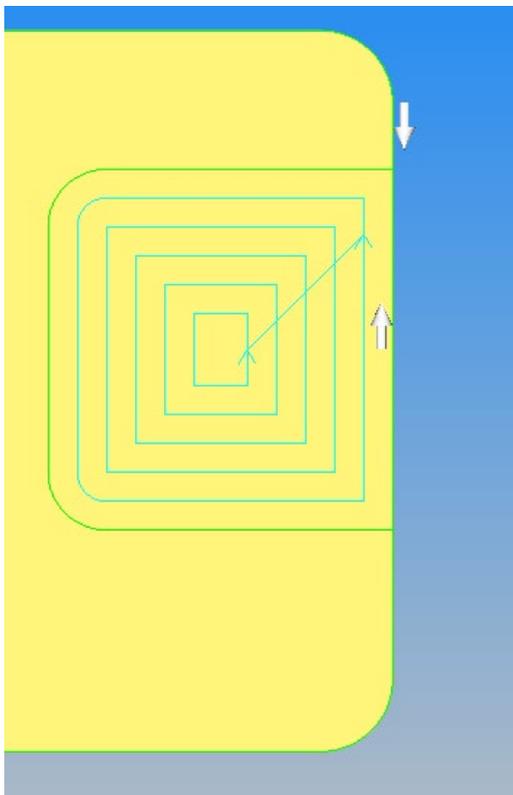


Figure 44 - Toolpath prior to using Set/Unset Open Elements

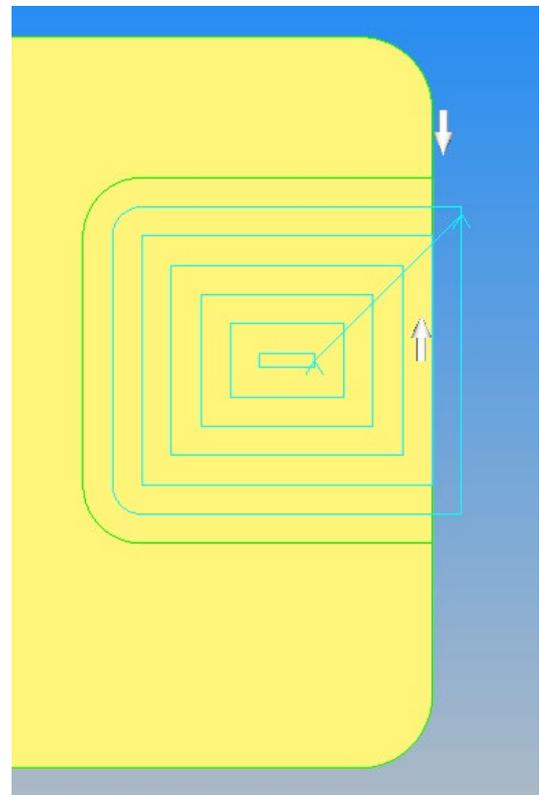


Figure 45 - Toolpath after to using Set/Unset Open Elements

The elements are made darker when set as open.

In the right hand image above, note that a larger value than the default 1 has been entered into the **Overlap on Open Elements** dialogue box, to force the tool to cross the geometry further. This will remove any possibility of there still being residual material left in the open sided section if the tool is worn or undersize.

Tool Data

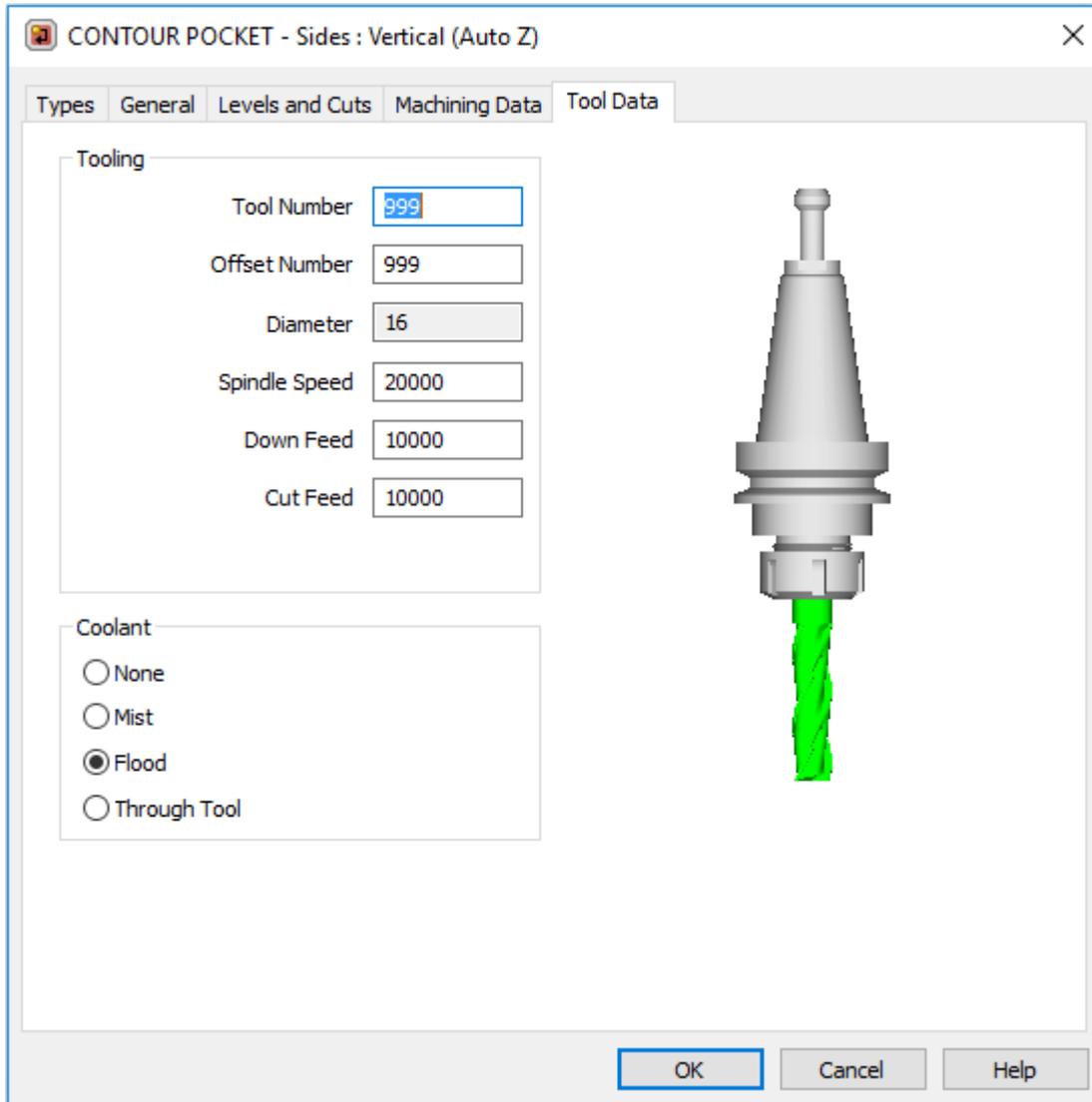


Figure 46 - Tool Data tab in Pocketing

Tool numbering and feed/speed options and any coolant options.

Area Clearance, Z Contour Roughing Options

All area clearance tool paths are created with the **MACHINE > Pocketing**  options. Pocketing creates NC code to remove material from within a closed geometry boundary. The method used to create the tool paths is selected from the dialogue options.

Types

The first set of options allows you to select the area clearance method to be used.

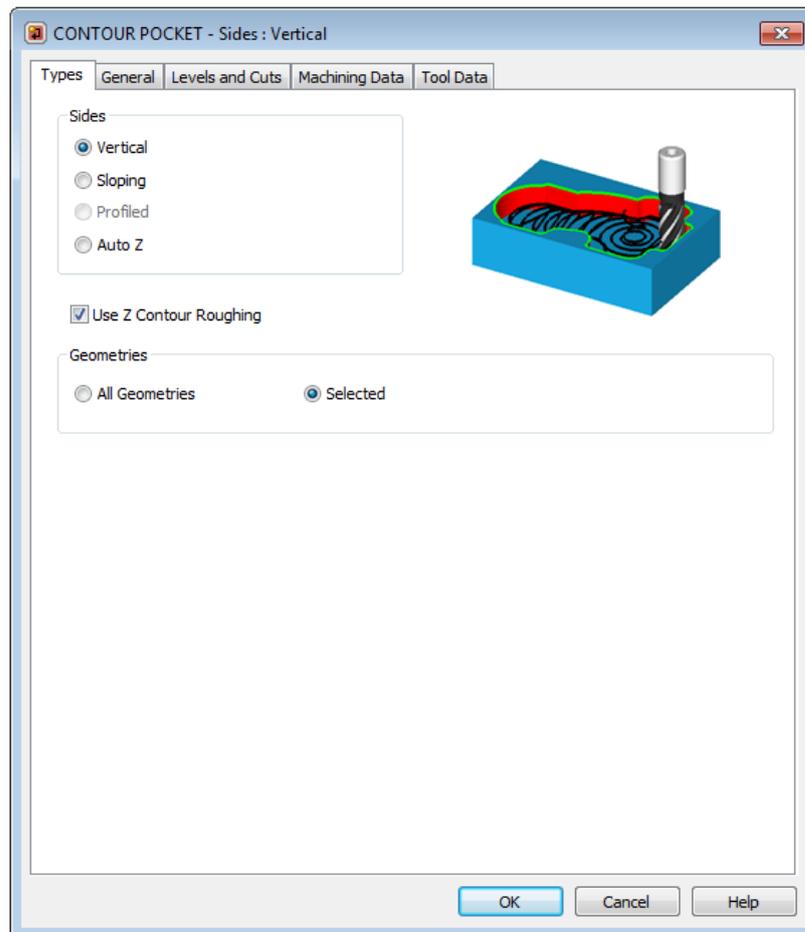


Figure 47 – Type tab of the Pocketing Cycle Use Z Contour Roughing activated

This dialogue section is the same for both types of Pocketing cycle, the only difference being that the **Use Z Contour Roughing** option is either active or not.

General

This dialogue is the same for Vertical, Sloping and Auto Z.

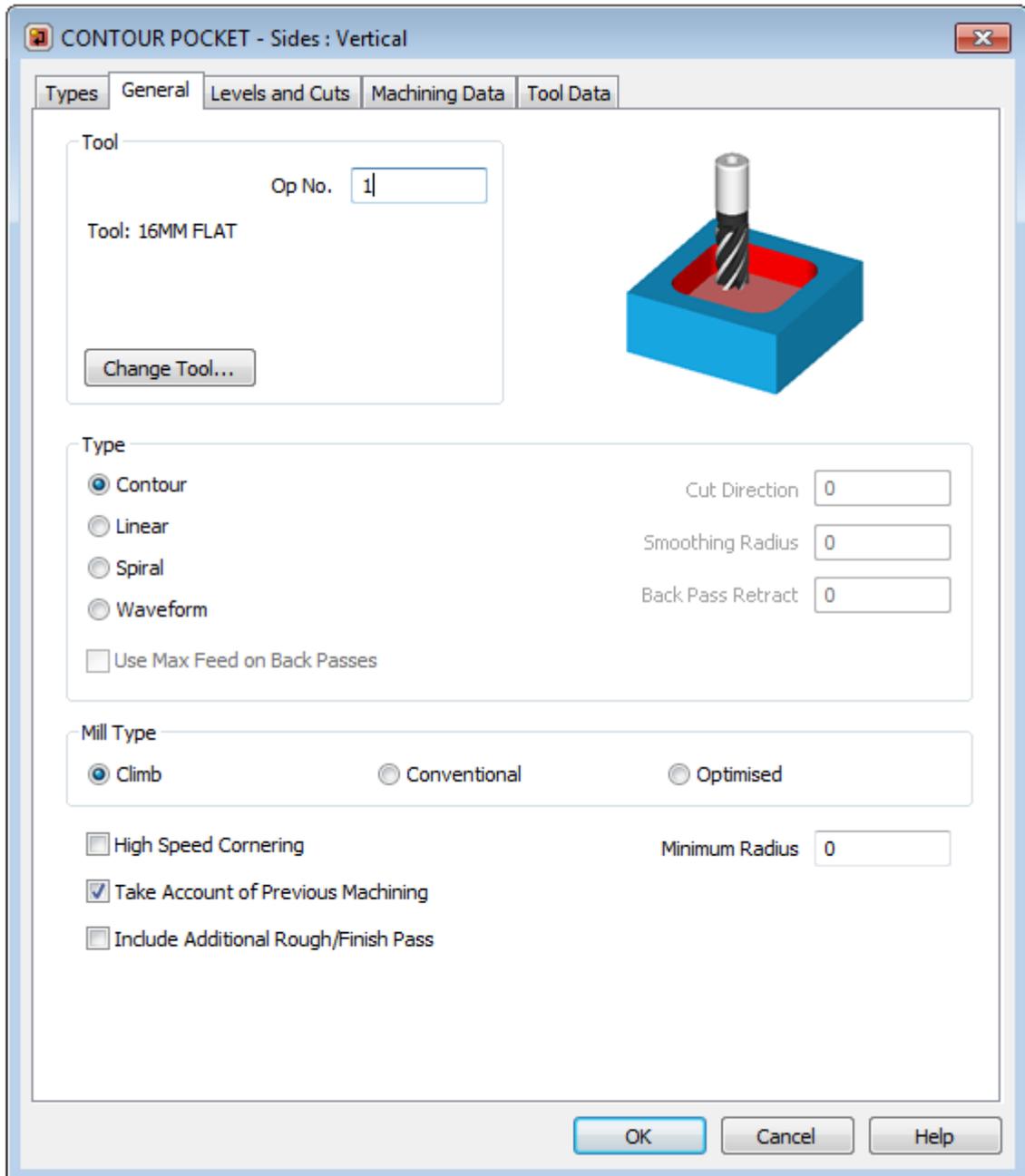


Figure 48 - General tab of the Pocketing cycle using Z Contour Roughing

Type

Contour

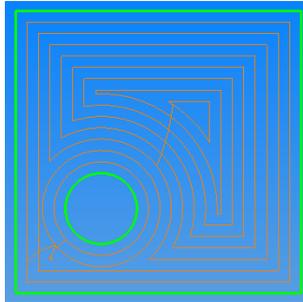


Figure 49 - Contour pocket using Z Contour Roughing

The machining calculations and options for these two types of path creation are exactly the same as when not using the Z Contour Roughing option.

Linear

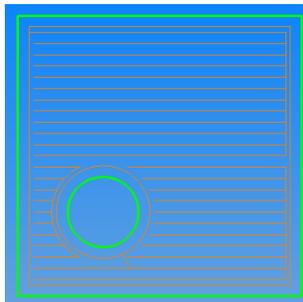


Figure 50 - Linear pocket using Z Contour Roughing

Spiral

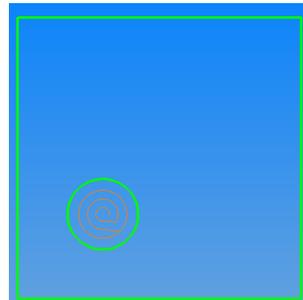


Figure 51 - Spiral pocket using Z Contour Roughing

When the **Use Z Contour Roughing** option is active, the Spiral option is now not restricted to only circles and can be applied to any form of geometry requiring an area clearance toolpath.

Waveform

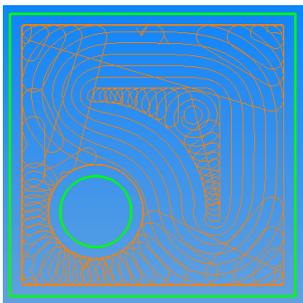


Figure 52 - Waveform pocket using Z Contour Roughing

The Waveform pocket strategy is only available when Use Z Contour Roughing is active. Waveform produces a tool path that provides constant load cutting.



Note that in some circumstances the Spiral setting will be overridden, and a concentric tool path created. This may occur for large step over values, e.g., where a spiral tool path would leave pegs of uncut material in corners.

Smoothing Radius (Waveform option only)

This controls the radius of tangential arcs between the linear moves that join the tool path when approaching into a corner.

By default, the smoothing radius is set to 25% of the tool diameter or equal to the minimum radius, whichever is greater. If specified, the radius must be at least equal to the minimum radius and no greater than the tool diameter.

Back Pass Retract (Waveform option only)



A large axial depth of cut can cause the tool to flex, so when disengaged on the back pass, the tool end will remove a small amount of material. This could blunt the bottom of the tool and generate unwanted heat in the component.

The Back Pass Retract is used to raise the tool by a small amount in the Z axis to avoid dragging the tool across the part surface on the back passes.

The tool starts to lift from the cut Z depth at the point where the tool disengages from the material and lifts over 5° of travel, at which point the tool will switch to a higher feed rate, if specified. The tool will descend back towards the Z cut depth 5° before the tool re-engages.

Use Max Feed on Back Pass (Waveform option only)

If checked, this option will use the maximum allowed feed rate on the back pass, where the tool is disengaged from the material over a large distance, i.e. where the tool is not cutting.

Max Feed is specified by

- \$753 in a template post processor.

or

- PostConfigure.FeedMax in VB type post processor.

The high feed rate becomes active 5° past the point where the tool disengages from the material. Then, at 5° before the tool re-engages the tool path switches back to the cutting feed rate.

Mill Type

The cutter can move along a profile on the left or right side. This, combined with the spindle direction, gives the type of cutting to be either Climb or Conventional.

- Climb** Tool cuts on the left of the profile (i.e. material is on the right).
- Conventional** Tool cuts on right of the profile.
- Optimised** Cycle is optimised to reduce link moves.
The tool may cut on the left or right side

High Speed Cornering

This setting only affects non-final passes; when active, a radius is introduced into sharp tool path corners. This is the default. This will lower machine stresses, reduce tool path length, maintain tool speed and reduced cycle time. It will, however, generate more NC code. When inactive, all tool path corners have an angle from the profile, however sharp.

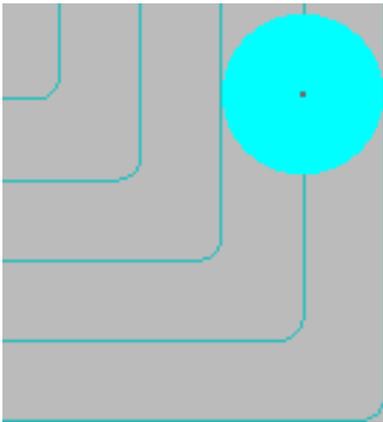


Figure 53 - High Speed Cornering Active

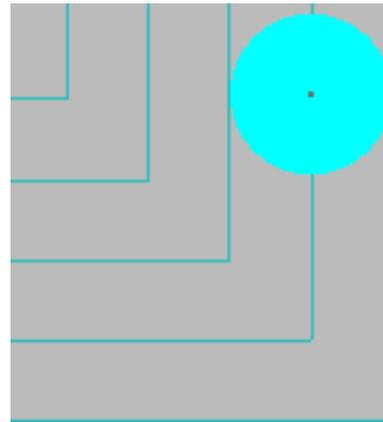


Figure 54 - High Speed Cornering Not active

Take Account Previous Machining

When active, the cycle looks for and avoids any areas of the part which have been previously machined at the same cutting depths using larger diameter cutters.

Settings for material top, final depth and allowances **MUST** remain constant to allow the calculations to be applied correctly. Any alterations to these three settings will result in the whole area being machined again with the smaller cutting tool.

Include additional Rough and Finish Pass

This option appends a rough and finish profile to the end of the pocketing tool path, the tool can be changed if required and cutter compensation can be used.

Levels and Cuts

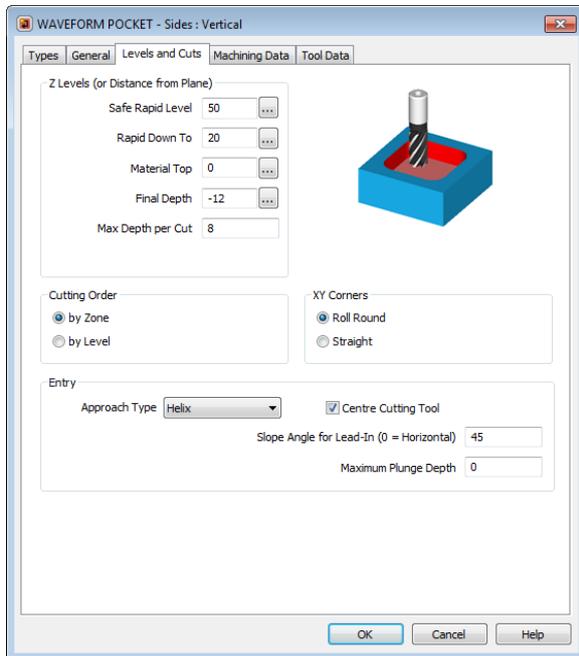


Figure 55 - Vertical dialogue options

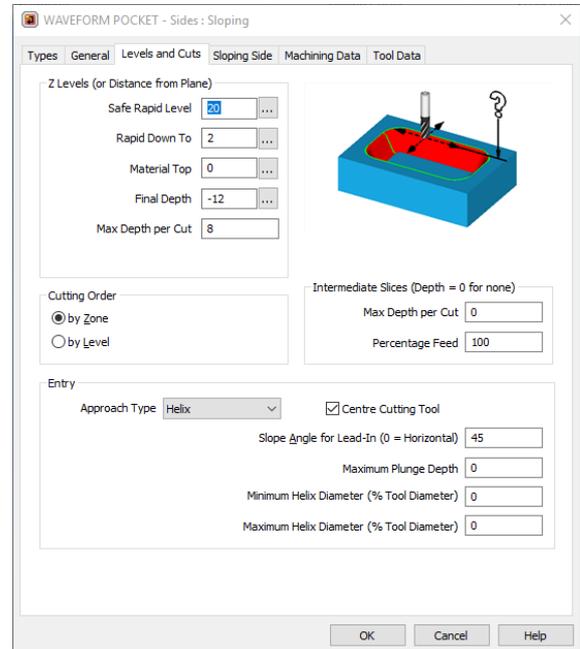


Figure 56 - Sloping Sides dialogue

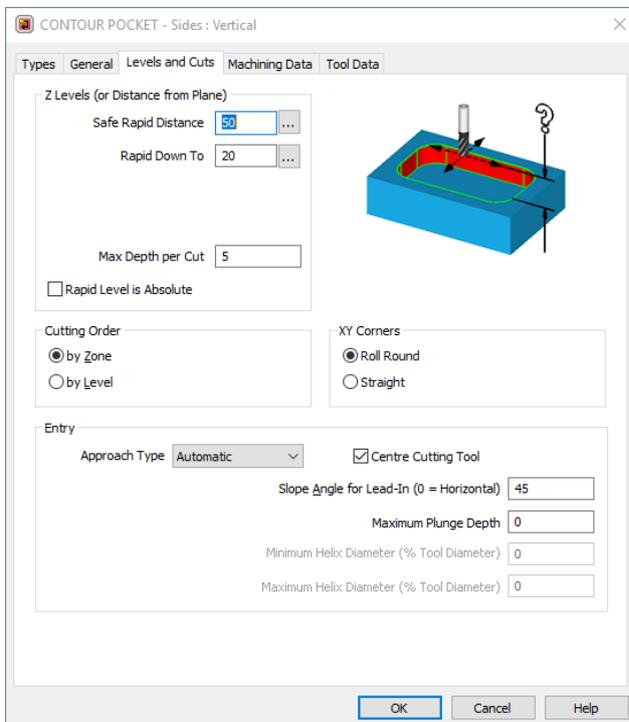


Figure 57 - Auto Z dialogue options

Z Levels (or Distance from Plane)

These dialogue options are the same as previously described in the standard form of pocket cycle.

Max Depth per Cut

Specifies the depth of each cut, this is used to calculate the number of cuts instead of applying a specific number of steps.

This value is populated from the Depth of Cut dialogue option on the Define Tool dialogue; refer to the Material and Tooling Definition manual for more information.

Intermediate Slices (Sloping Sides Only)

Intermediate slices may be used to reduce the size of the step left by the roughing cycle.

Only the step region is machined for intermediate slices. Intermediate slices will always be cut from bottom to top within each cut increment to reduce unnecessary cutting. This method roughs and semi-roughs the part in one cycle while allowing the tool to cut to its maximum depth for the main increments.

Max Depth per Cut

Specifies the depth of each successive cut (for intermediate slices). Please note that this defaults back to 0 after use to stop intermediate slices being activated by accident.

Percentage Feed

The feedrate for intermediate slices is specified as a percentage of the cutting feedrate.

X Y Corners (Vertical and Auto Z Only)

- ⊙ **Roll Round causes** the tool path centreline to arc around external sharp corners.
- ⊙ **Straight causes** the tool path centreline to create a sharp point at the corner.



Figure 58 - XY corners set to Roll Round

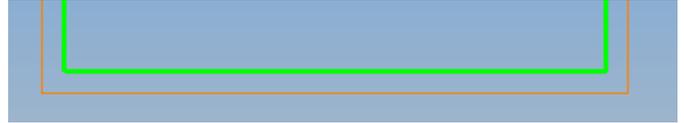


Figure 59 - XY corners set to Straight

Approach Type

Offers you the ability to start the cycle from four different options depending on the format of the chosen cycle.

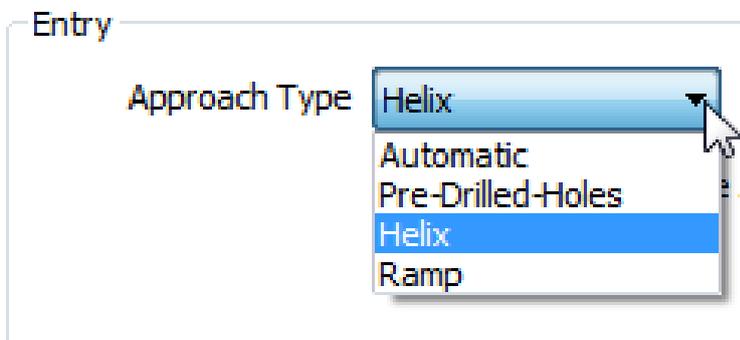


Figure 60 - Approach options

- | | |
|--------------------------|---|
| Automatic | The cycle uses a predetermined position dictated by the creation of the cycle and plunges the tool into the part vertically down the tools Z axis. |
| Pre-Drilled Holes | Specify a pre-defined circle as the point of entry. The geometry needs to have upper and lower Z values applied using either Feature Extraction or Geometry Z levels . The tool will rapid down the hole to the desired depth unless it is within the safe distance of the pre-drill point where it will feed to depth from the safe distance above the point. If Feed When Plunging is checked (see Machining Data), the tool will feed down the hole at the Down Feed value. |
| Helix | The cycle will place a helical approach in pocket areas from a position determined by the creation of the cycle. The tool entry will never create a full diameter path but instead follows a pre-calculated downward spiral, the slope of which is calculated using the Slope Angle option detailed later. |
| Ramp | This creates a ramp approach into pocket areas. The path will be a straight line cut of the diameter of the tool with the angle defined using the Slope Angle option. |



Note on the Helical option, your post processor must be written to generate the correct code output. Just setting the option in the cycle is no guarantee that you will necessarily generate the correct code for the machine to work with.

Slope Angle for Lead-In

The Slope Angle determines the angle of a Ramp or Helix entry move as it cuts into the stock material. The maximum ramp angle is calculated from the *Maximum Plunge Depth* and *Centre Cut* modifiers. This value will automatically be used if it is found to be less than the specified Ramp Angle. If the modifier is left blank the calculated ramp angle will be used.



Check against information supplied by the tooling manufacturer for the maximum slope angle.

Maximum Plunge Depth

The Maximum plunge depth specifies the maximum distance in the Z-axis that the tool can plunge into the material. The value will be used to check that the ramp move does not exceed this value. The maximum plunge is assumed to be equal to the Cut Increment when modifier is left blank.



If Maximum Plunge Depth is set to 0 the tool cannot plunge or ramp. Therefore, the cycle can only machine external areas or pocket areas that have pre-drilled holes.

Cutting Order

The cutting order can be set to either **by Zone** or **by Level**.

The **by Zone** option will cause the area clearance tool paths to complete each area clearance to its full depth before creating the area clearance tool paths in the next area.

The **by Level** option will cause the area clearance tool paths to complete all areas at the first cutting level, then restarting with the first area completing all areas at the second level and so on. Any moves between areas will be at the Safe Rapid Z.

The **by Level** methodology is normally used where there is a thin wall between pockets, or the pockets are deep and cutting each level allows the swarf/dust to be extracted.

The **by Level** methodology is used in the Aerospace Industry for machining items such as Wing Spars sometimes known as water line machining.

Helix Diameter

If you are using the Helix approach type when entering the part, then you are given the options to state a Minimum and Maximum helix diameter based on the Tool Diameter.

Minimum Helix Diameter (% Tool Diameter)

Maximum Helix Diameter (% Tool Diameter)

Sloping Side

When machining with sloping sides there is an extra dialogue that allows you to set the slope values.

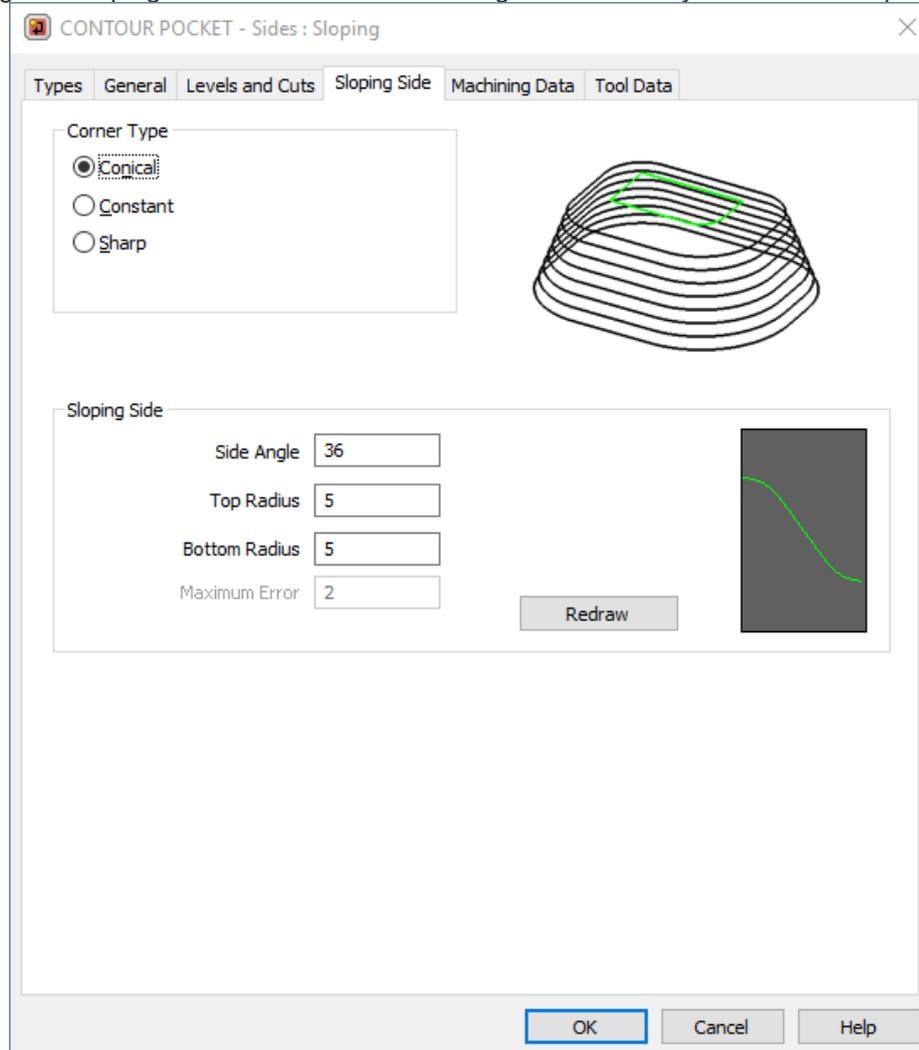


Figure 61 - Options only available using Sloping Sides

Corner Type

These options set the style of transition around external corners within a pocket environment

- Conical** Produces a flared style of corner on islands.
- Constant** Produces the correct geometry of the corner using a Roll Round style of corner tool path.
- Sharp** Produces the correct geometry of the corner using a Straight style of corner tool path.

The **Side Angle** sets the slope of the form where an angle of 0° would equal a vertical face. The two radius options for **Top and Bottom** allow for a blend radius to be generated between the sections.

Machining data

This dialogue is the same for Vertical and Auto Z.

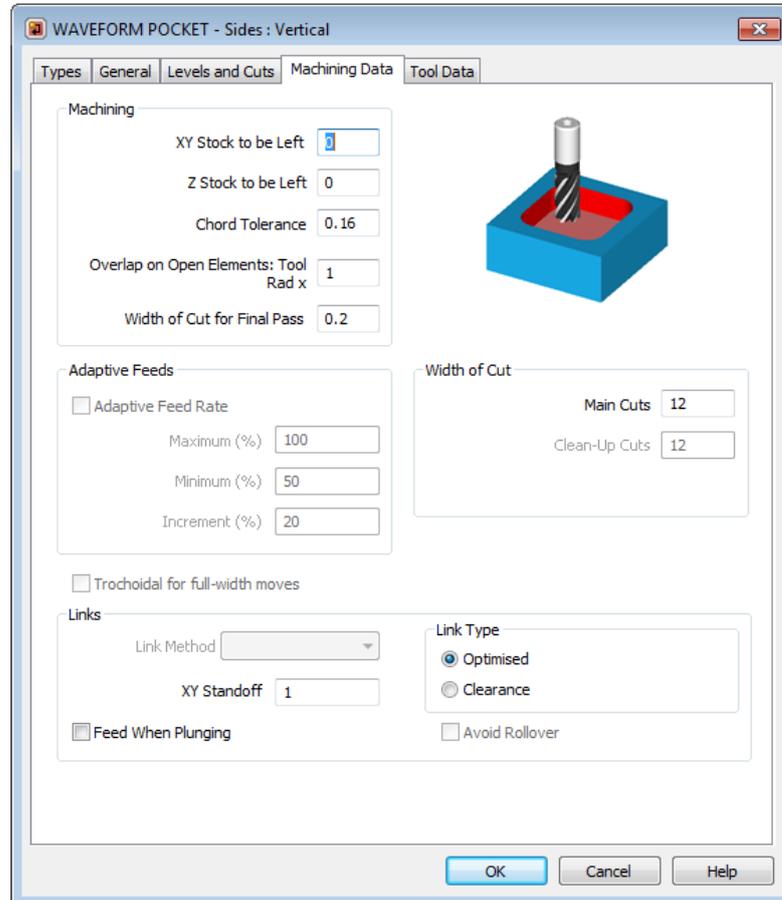


Figure 62 - Machining Data dialogue options

XY Stock to be Left The amount of extra material to be left on the walls of the pocket detail to allow for a finishing pass.

Z Stock to be Left The amount of extra material to be left on the pocket floor for a finish pass.

Chord Tolerance

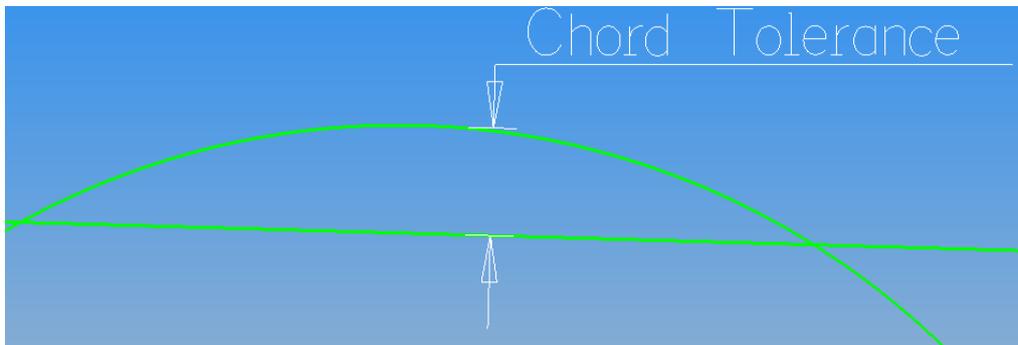


Figure 63 - Chord Tolerance

This option controls the accuracy of how a curved feature is followed by the toolpath.

As can be seen in the diagram above, a chord is any straight line drawn across a curve. ALPHACAM will ignore all the contact points on a curve between one effective intersection of the line and curve and the next in the chosen direction of travel.

The larger the chord tolerance is set, the more faceted a curved feature will appear, the smaller the value, the more accurately the curved is followed.

For a roughing cut, 0.25mm to 0.16mm is an acceptable value.

If you are using a Finishing strategy a practical value is between 0.1mm and 0.02mm.

Overlap on Open elements

This option is the same as discussed earlier on standard pocketing.

Width of Cut for Final Pass

This is the amount of material left on the side walls to allow for a semi finishing trim to take place around the profile of the pocket. Due to the nature in how the cycles are generated there may be areas of excess material due to tool moves that needs to be removed. This Final Pass option removes those instances.

Width of Cut

Main Cut

The value of the step over that each pass will take whilst generating the primary pocket cycle. This value is set as a percentage option of the cutter in use and can be altered individually on each pocket cycle or globally using the **HOME > Configure > General** options and on the **Machining** entry section, adjusting the percentage value in the Contour Pocket option.

Clean-Up Cuts

This value is assigned to the widths of any subsequent step over cuts that may need to be applied in some pocketing options.

Trochoidal for full-width moves

A **Full Width Cut** is one where the move exceeds the specified **Main Cuts** step over value. This potentially overloads the tool, so ALPHACAM detects when a cut would be full width and offers this option. Note that in a narrow channel there may not be room for a trochoidal cut, in which case the channel is excluded from the cycle.

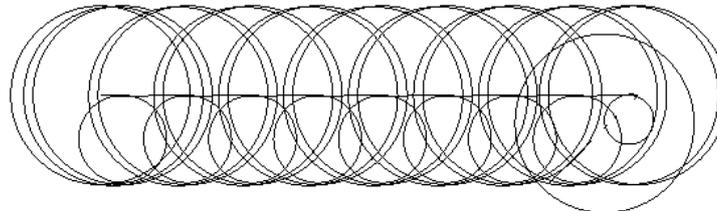


Figure 64 - Trochoidal toolpath example

This option is one more that is only available when not using the Waveform method.

Link Method

The Link Method drop down options is only available when not using the Waveform strategy. This linking scenario is used to traverse the tool from one location within a pocket area to another location within the same pocket area.

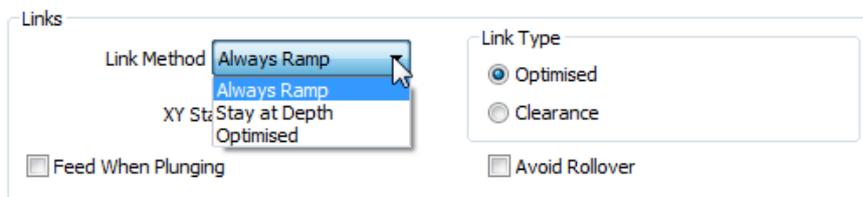


Figure 65 - Linking between different areas

Always Ramp There is a retraction for the link (rapid) move. (The height of the retraction is set by 'Link Type').

Stay at Depth The tool stays at the same depth and there is no retraction. This does not apply to links that clear geometry, for these links there is always a retraction to avoid gouging.

Optimised The faster of the two methods above is automatically selected.

Link Type

When a link is required, this setting controls the retract height.

Optimised This setting only applies to moves within the same machined out volume (as in a pocket in a boss in a pocket). For links that move between two separate volumes (two separate pockets for example), the tool always retracts to the Clearance height. These moves will be at the maximum feed rate specified in the post processor.

Clearance For the link the tool always retracts to the Clearance height.

XY StandOff

This allows you to specify the distance between the stock and the side of the tool when approaching the part walls.

Feed When Plunging

When checked, the tool will use the feed rate when moving down in the Z axis during the cycle. This is a safe option when the amount of stock material is unknown. When unchecked, the tool will rapid to the safe distance above the part then feed into position.

Avoid Rollover

Check this to eliminate the tool roll-round on the outer corners of open pocket geometry. This is mainly for use with wireframe and prismatic type solid geometry and may also only be applied when the correct conditions are met.

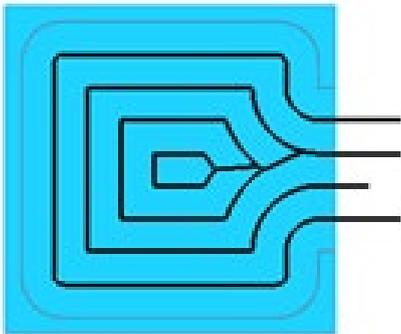


Figure 66 - Avoid Rollover Active

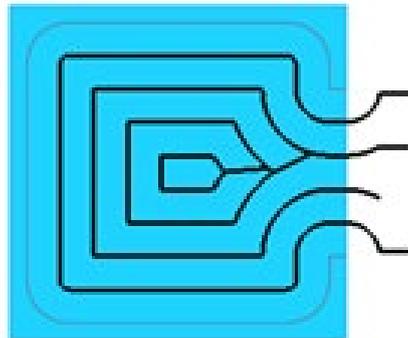


Figure 67 - Avoid Rollover Inactive

Adaptive Feed Rates

Adaptive Feeds

Adaptive Feed Rate

Maximum (%)

Minimum (%)

Increment (%)

Figure 68 - Adaptive feed rates options

Adaptive feed rates adjust the feed rate to even out loading on the tool; as such this is another option that is only available when not using the Waveform strategy.

For example, in a narrow channel the loading could be higher as the actual cut width increases above the specified Cut Width value; in this case the feed rate would be reduced. Conversely for smaller cuts at less than the specified %Step over, the feed rate would be increased to reduce machining times.

The adjustment is only made to XY feeds; it is not applied to any 'end cutting' plunge (ramp) moves.

The **Minimum (%)** is the percentage of the original specified feed rate to be used in a full width cut (where the effective step over = 100%). At intermediate effective step over's, the feed rate is adjusted proportionally.

For example, if the **Increment (%)** is 10 for a specified cut feed of 5500 and the specified **Cut Width is 50%** of the tool dia. i.e. 10, then, if the actual cut width created by the cycle is 12 i.e. 60% of the tool dia., the feed rate would be reduced by 10% to 4950.

If the cut width was 16 i.e. 80% of the tool dia., then the feed rate would be reduced by 30% to 3850. If the cut width was 20 i.e. 100% of the tool dia., then the feed rate would be reduced by 50% to 2750.

The **Maximum (%)** is the percentage of the original specified feed rate to be used as the effective step over reduces towards 0.

For example, if the **Increment (%)** is 10 for a specified cut feed of 5500 and the specified **Cut Width is 50%** of the tool dia. i.e. 10, then, if the actual cut width is 8 i.e. 40% of the tool dia., the feed rate would be increased by 20% to 6600.

If the cut width was 6 i.e. 30% of the tool dia., then the feed rate would be increased by 40% to 7700. If the cut width was 1 i.e. 5% of the tool dia., then the feed rate would be increased by 95% to 10725.

The feed rate can only be adjusted to certain values. This **Increment (%)** is the size of the steps between these values, as a percentage of the original feed rate.

 Specifying low values helps produce tighter control over the feed rate, but with potentially more CNC code and slower processing. Specifying high values helps reduce the CNC code and speed processing, but with less accurate control over the feed rate.

The **Cut Width** is specified as a width of cut value in the Z contour roughing dialogue the system internally uses the width of cut and the tool diameter to calculate the relevant percentages.

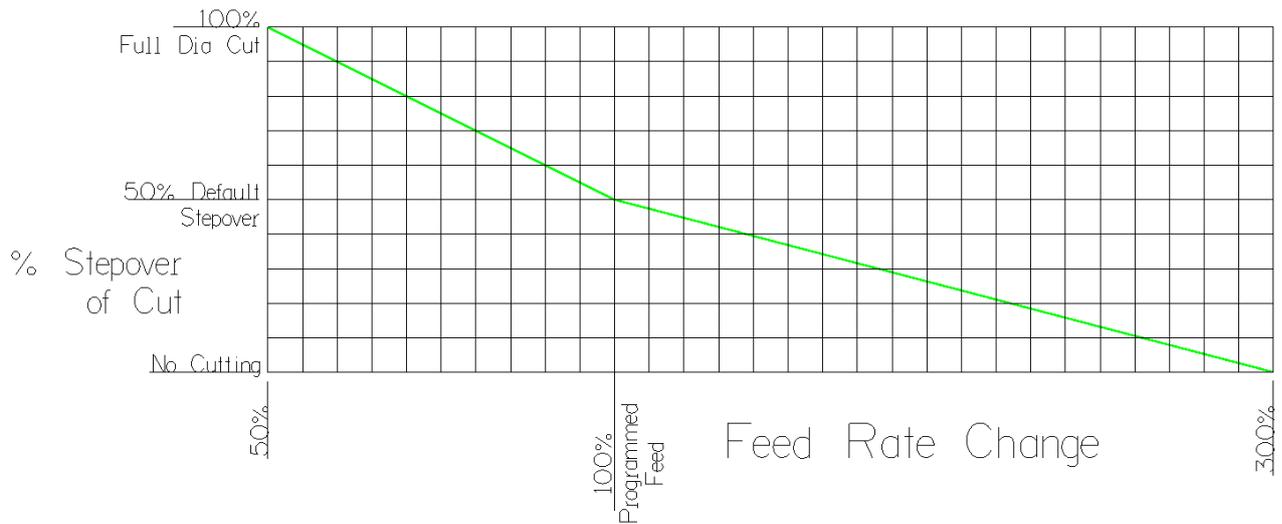


Figure 69 - Feed rate changes when using Adaptive Feed Rates

Tool Data

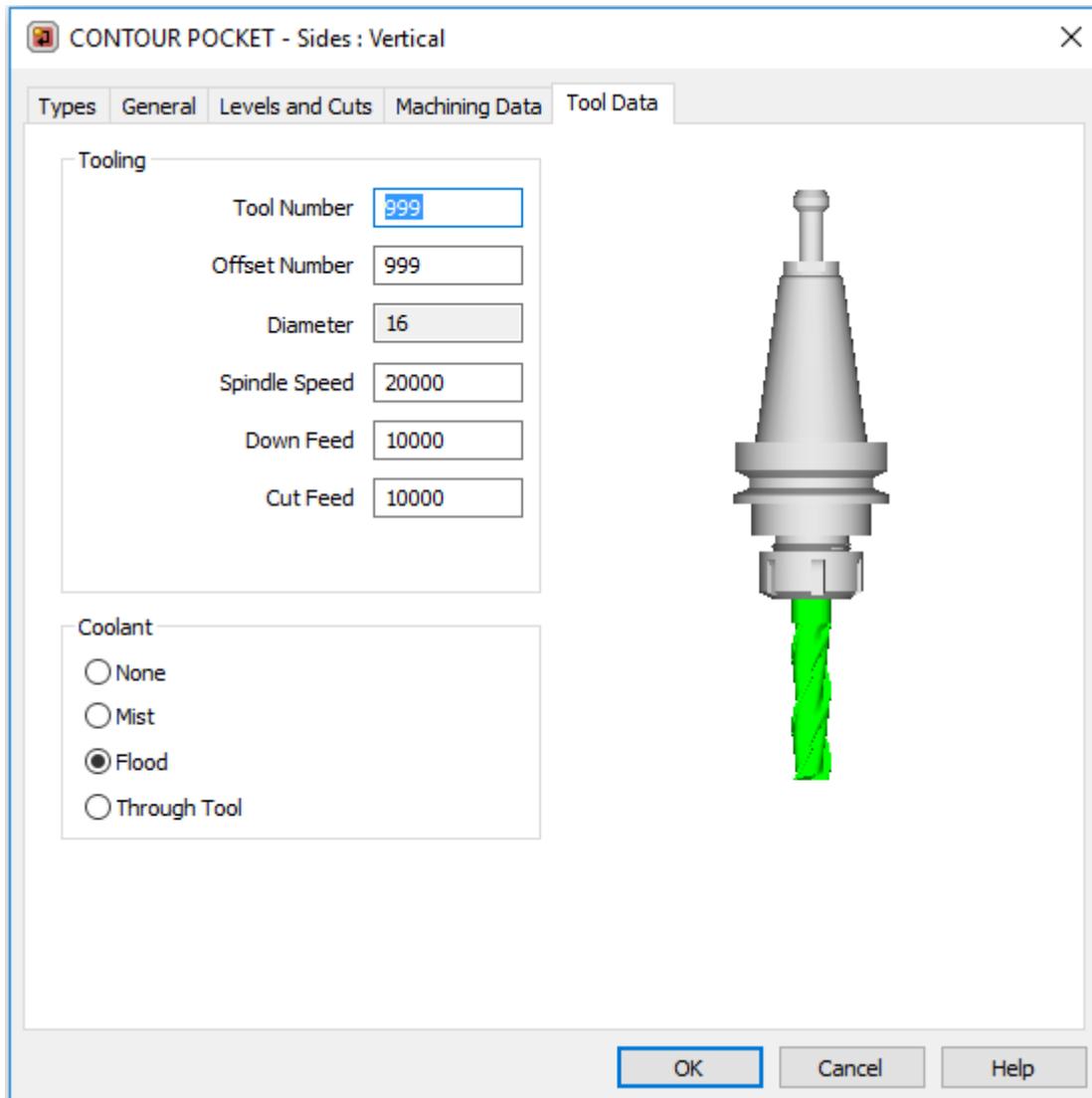


Figure 70 - Tool Data

Tool numbering and feed/speed options and any coolant options.

Rough or Finish, Vertical Sides.

Types

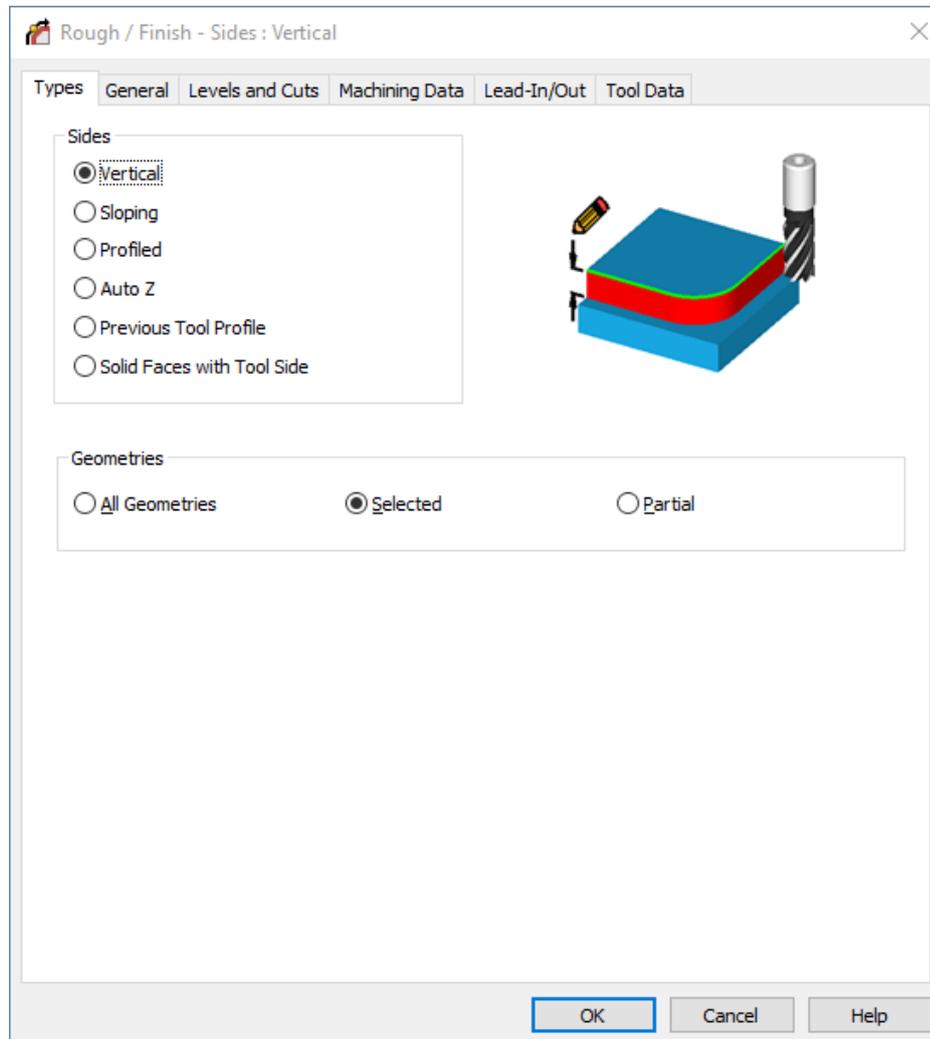


Figure 71 - Types tab for Rough or Finish cycle

The first four **Sides** options, when using the **Rough or Finish** cycle, are the same as those described for the Pocket cycle, with the addition of **Previous Tool Profile**.

This option, described in detail later, allows for the comparison of non-regular geometric side walls for residual machining.

Previous Tool Profile allows for direct 2D vertical machining on Planar faces without the need to use the Feature Extraction process.

The **Partial** option allows you to choose to only machine part way around an open or a closed geometry profile. The command will prompt you to select the points on the profile you want to machine between. You **MUST** select the points with respect to the direction of cut.

General

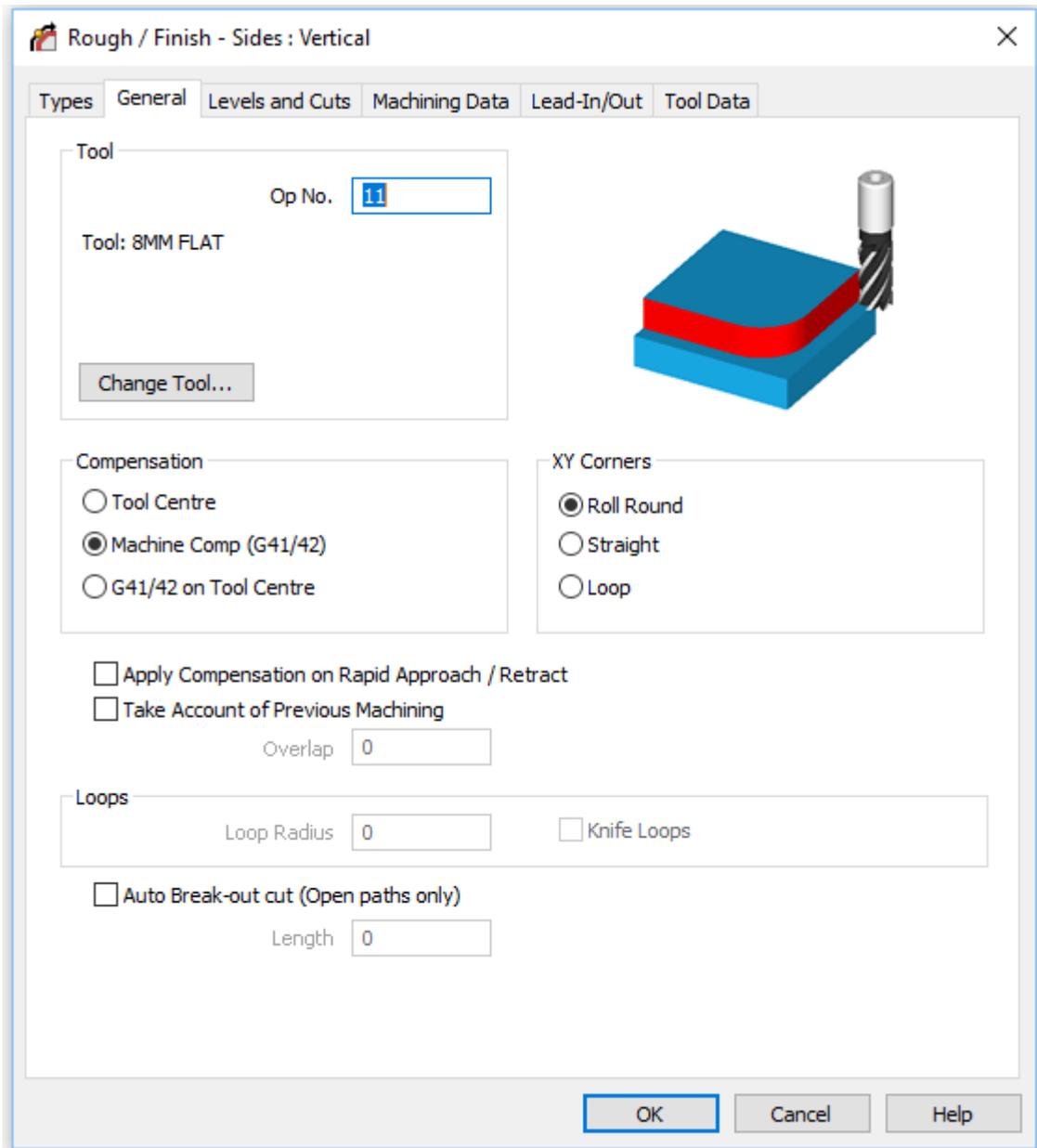


Figure 72 - General tab

Tool Compensation Overview

There are three options for cutter compensation, this a process on the machine tool that automatically adjusts the cutting process for the tool diameter/radius that has been set in the machine.

- ⊙ **Tool Centre**

Sets the output code to be the position of the tool centre line on the screen; the machine tool cannot make any alterations for the tool size.

- ⊙ **Machine Comp G41/G42**
(Referred to as **Radius Offset Compensation**)

Sets the output code to be the position of the geometry as drawn on the screen; the machine tool will automatically compensate for the tool size as defined on the machine using its working radius or diameter depending on the machine control.
Compensation is normally applied and cancelled along the lead in / out lines.

- ⊙ **G41/G42 on Tool Centre**
(Referred to as **Wear Offset Compensation**)

Sets the output code to be the position of the tool centre line on the screen; the machine tool can automatically compensate for the variation in tool size between that defined in ALPHACAM and that defined on the machine. If the tool on the machine and the tool in ALPHACAM are the same, then the tools' offset register will be set to 0.

Compensation ☉Tool Centre

With this option you are restricted to the size of the tool on the machine creating the correct size of part. Should the tool wear or be re-ground/sharpened to a differing diameter, then you will need to recreate the tool in ALPHACAM and program the part again using this new tool.

The NC code generated by ALPHACAM, automatically adds or subtracts the radius of the chosen tool to produce the correct locations for the actual tool centre in relation to the selected geometry.

In the following image you see a $\varnothing 16$ mm tool machining around a 100mm square. The locations in the image are the actual values sent to the machine control to produce the part.

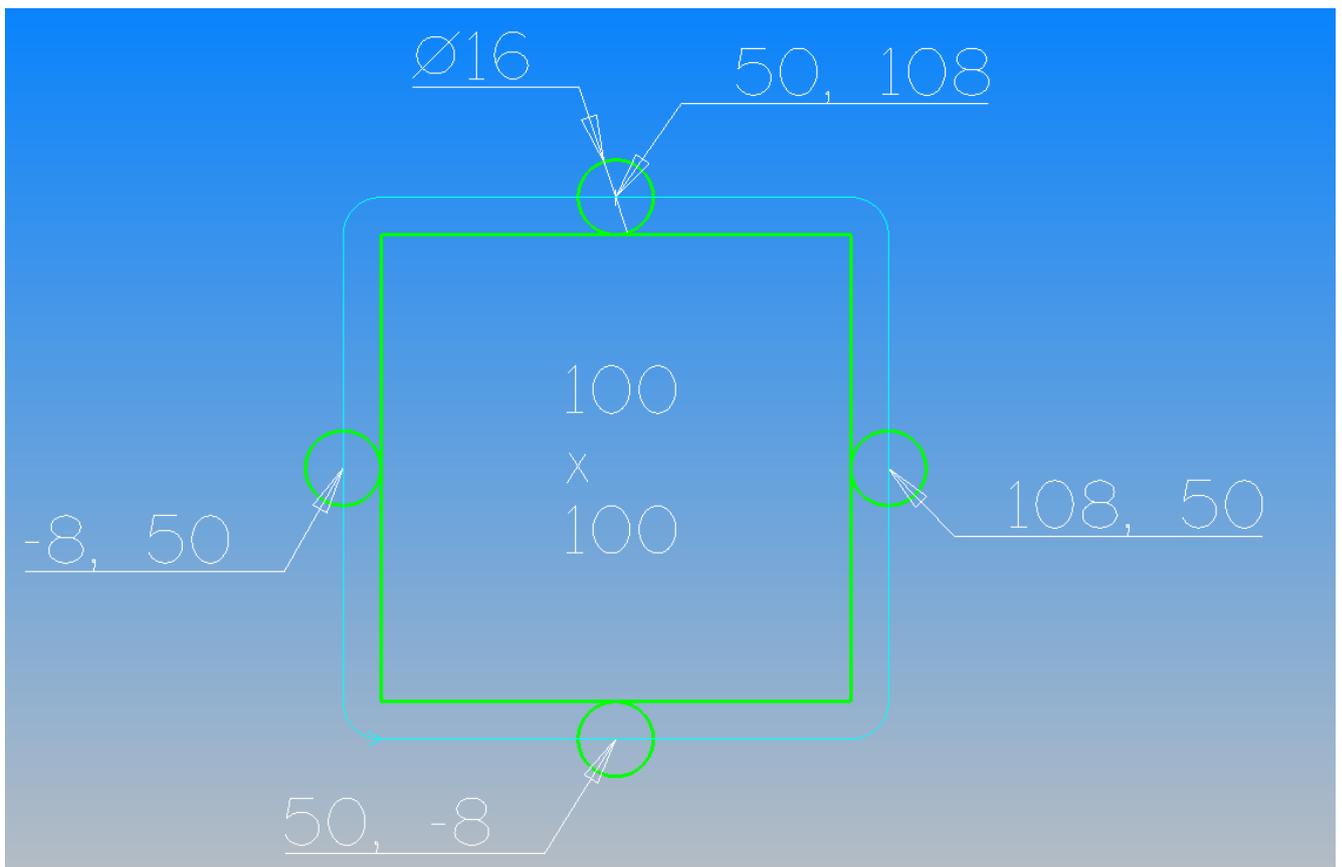


Figure 73 - NC code actual locations on sample part

With this option in use, if the part varies in size then your only option is to reprogram in ALPHACAM using a newly created tool that matched the exact size of the tool on the machine.

Compensation ⓄG41/42 On Tool Centre

This option is similar to the previous detailed option in that the NC code generated by ALPHACAM, automatically adds or subtracts the radius of the chosen tool to produce the correct locations for the actual tool centre in relation to the selected geometry.

In the following image you see a $\varnothing 16$ mm tool machining around a 100mm square. The locations in the image are the actual values sent to the machine control to produce the part.

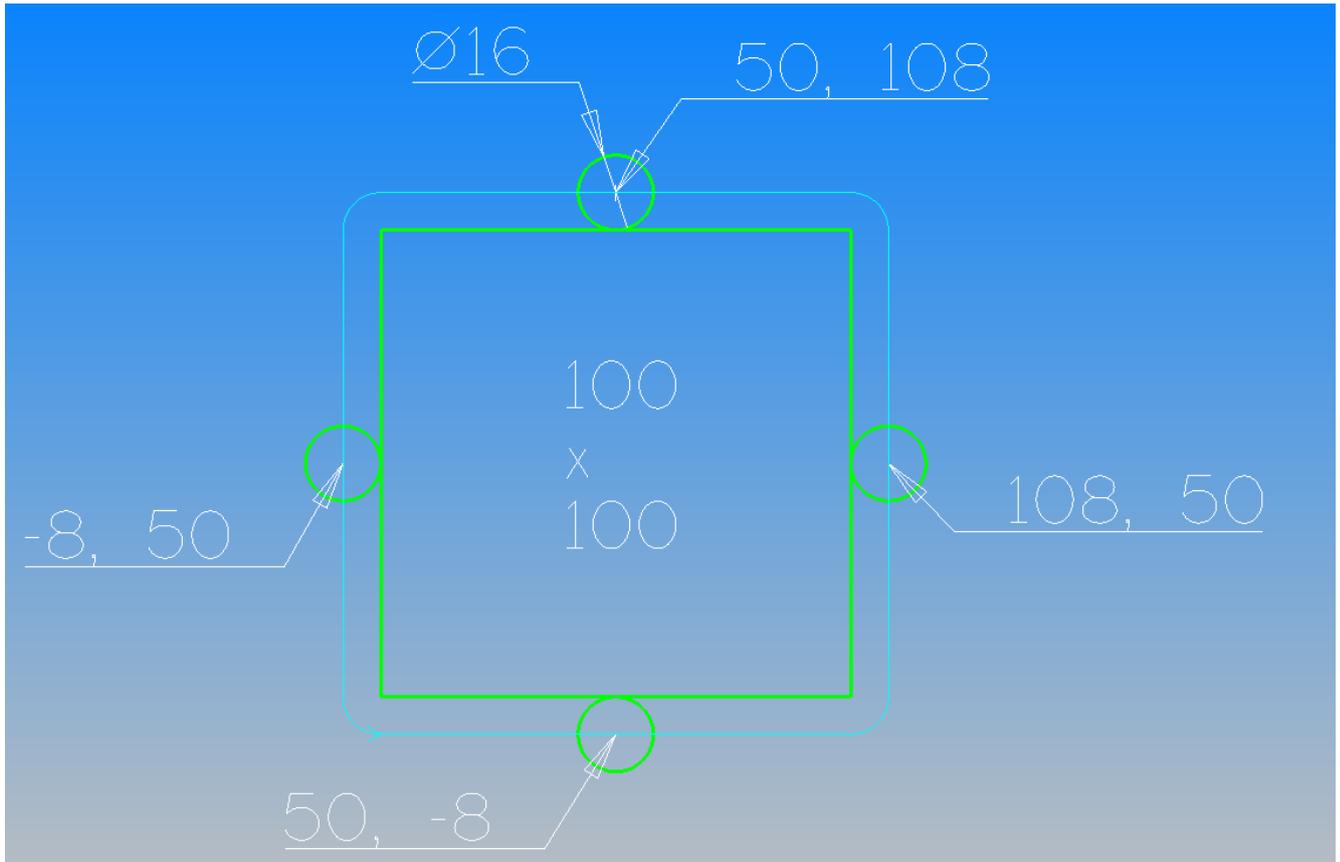


Figure 74 - NC code actual locations on sample part

Now, however, with this option you can adjust the size of the tool in the machine control to adjust the size of the part.

Using this method of Tool Compensation, you adjust a wear offset value on the machine by however much the part is over or under size.

A brand new tool would have a wear offset value of 0 (zero).

For example, should the part measure 101mm x 101mm, in the machine control you would set a wear offset value of ± 0.5 mm to adjust the position of the tool so that it cuts to the required dimensions.

The \pm symbol is stated because it depends on the machine control whether a positive or negative value is entered to adjust the size of the part.

Compensation ☉Machine Comp (G41/42)

This option differs from the previous two in the manner of how the NC code is generated.

In the following image you see a $\varnothing 16$ mm tool machining around a 100mm square. The locations in the image are the actual values sent to the machine control to produce the part.

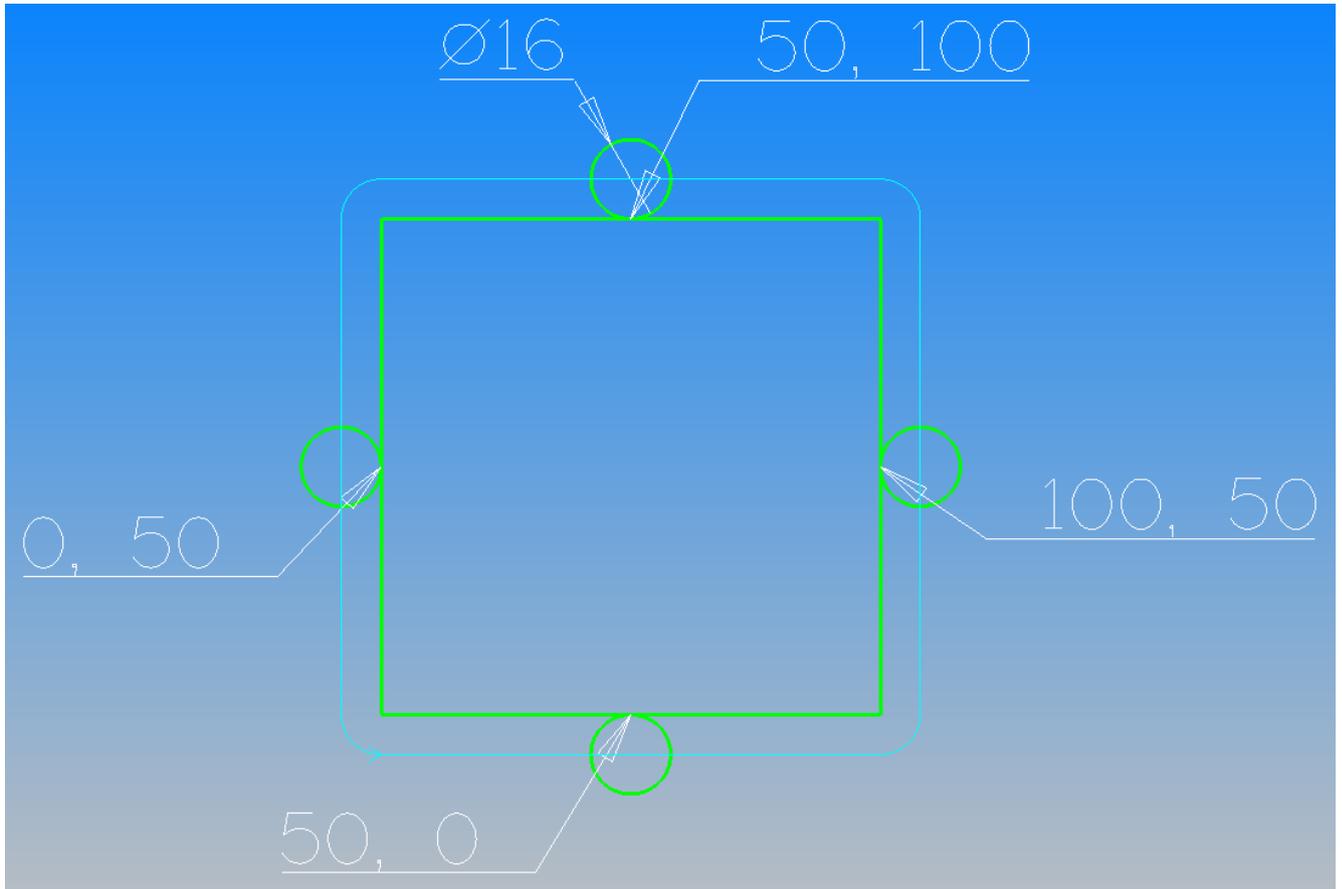


Figure 75 - NC code actual locations on sample part

With this option you can adjust the size of the tool in the machine control to adjust the size of the part.

Using this method of Tool Compensation, you adjust the actual Radius or Diameter of the tool according to the amount the part differs from the actual size required

A brand new tool would in this instance would be listed as R8 or $\varnothing 16$ in the machine control

Should the part measure 101mm x 101mm, in the machine control you would change either the Radius value to 8.5 or the diameter value to 15, depending on the machine control options, to adjust the position of the tool so that it cuts to the required dimensions.

Apply Compensation on Rapid Approach/Retract

If the machine tool supports this type of move where any offset options are applied during the rapid moves towards or away from the actual path to be machined and the post processor chosen supports these moves, ticking the option will structure the resulting NC code to suit such machining.

If your machine does not support this type of move then you will need to position the cutter at the correct location first, feed down to the required Z depth, and then apply the compensation move after that Z move.

In most post processor settings, a warning will be given to advise whether this option is to be used or not, along the lines of “Your Post Must...” Or “Your Post does not support...”

Take Account of Previous Machining

Checks what has been machined previously and only cuts where material is left.

When using this option, the upper and lower Z values and any XY stock amount **MUST** match to allow the calculation to function correctly. Any differences and the option is disregarded resulting in the entire profile being machined in full.

When in use, **Take Account of Previous Machining** also allows the use of the **Overlap option**.

This value is applied to the tool in use so that the new toolpath commences by this value before the required area to be machined and then continues past the area to finish past by the same value. This can prevent dwell marks or burning on customer facing products and lead to a better quality product.

X Y Corners

<input checked="" type="radio"/> Roll Round		<p>Causes the tool path centreline to arc around external sharp corners. The arc radius being the same as the tool radius.</p>
<input checked="" type="radio"/> Straight		<p>Causes the tool path centreline to create a sharp point at the corner.</p>
<input checked="" type="radio"/> Loop		<p>Causes the tool path centreline to create and external loop of a specified radius at sharp corners.</p>

Figure 76 - Examples of XY Corners

Loops

Loop Radius Knife Loops

Apply loop if corner angle is less than

Figure 77 - Loop value options

When using the **Loop option**, the values are entered in the **Loop Radius** dialogue and an **Angle value** can be specified if the loop is only required on angled corners that are less than the entered value.

The **Knife Loops option** is used to force two differing methods of corner loops; this is usually specific to dedicated Knife cutting machines.

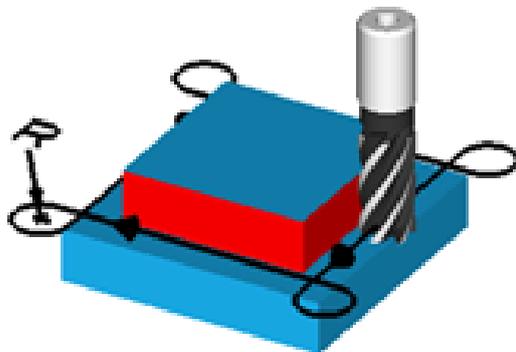


Figure 78 - Knife Loops not active

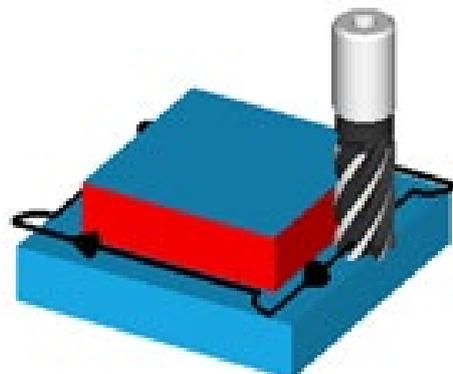


Figure 79 - Knife Loops active

Auto Break-out (Open paths only)

Auto Break-out cut (Open paths only)

Length

Figure 80 - Auto Break-out options

The Auto Break-out option allows for a short cut to be applied at one end of an Open geometry prior to the full cut across the geometry.

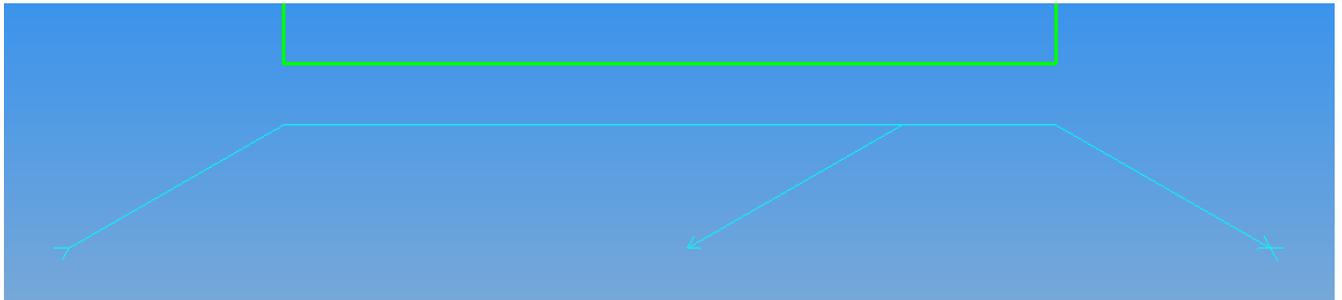


Figure 81 - Auto Break-out toolpaths

This type of cut is suited to grained material where a full pass only may split the grain where the tool exits the actual part. Performing the exit cut first in the opposite direction to the applied Ghost Tools can alleviate this issue.

Levels and Cuts

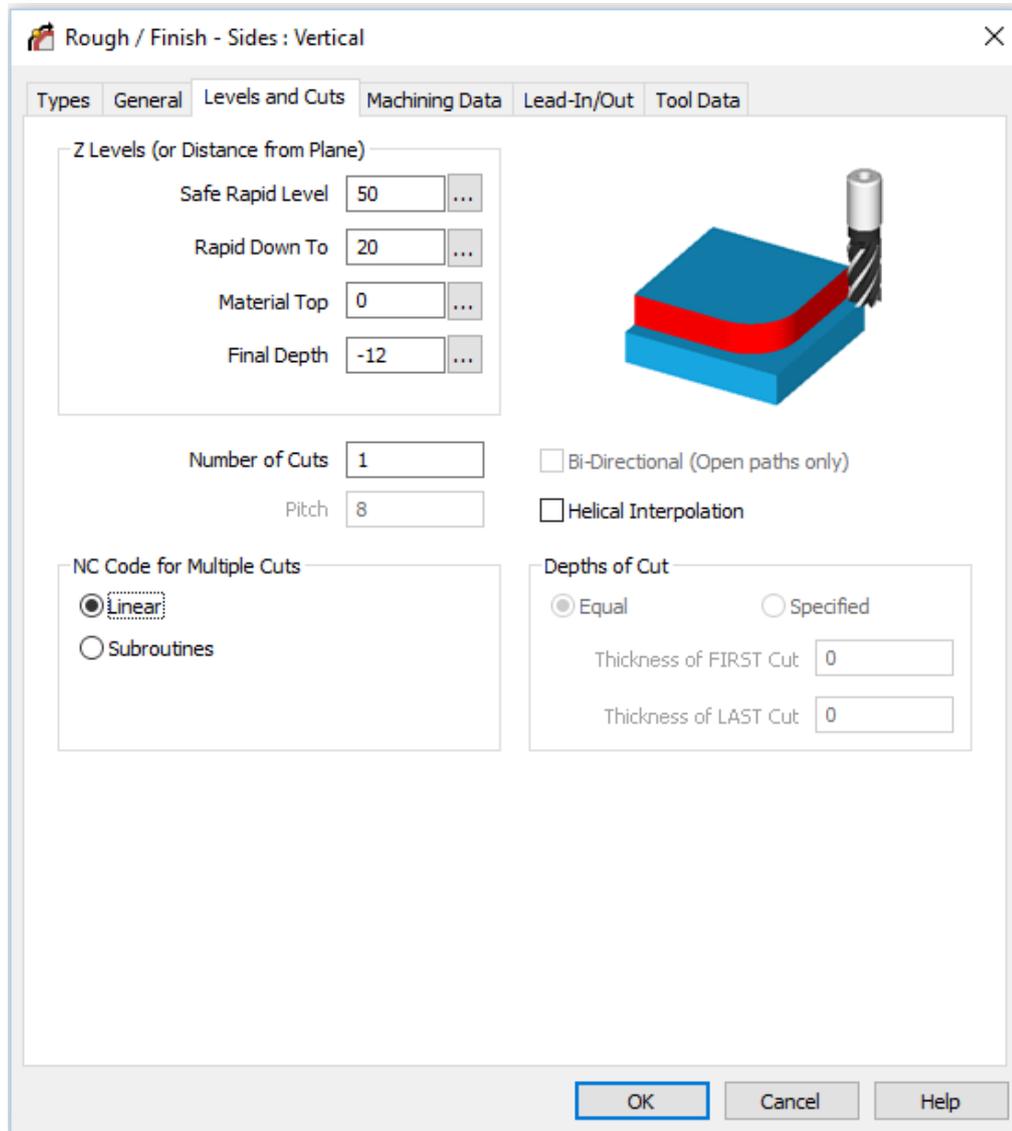


Figure 82 - Levels and Cuts

Z Levels (or Distance from Plane)

These dialogue options are the same as previously described in the standard form of pocket cycle.

Number of Cuts

The number of cuts in the Z direction to machine from the **Material Top** to the **Final Depth**.

Bi-Directional (Open Paths only)

When machining a single section of geometry, it may prove more beneficial to make the cuts run in both directions instead of returning to the same start location. If the material and tooling accept this type of machining, then setting the Bi-Directional option active will be of benefit.

□ Helical Interpolation

With this option active, the Number of Cuts entry is disabled, and the **Pitch** option made active.

Instead of machining a part at set Z depths using defined depths of cut, Helical Interpolation allows the toolpath to be generated in a spiral form that for every pass around the chosen profile the Z axis progressively drops by the amount set in the Pitch option.

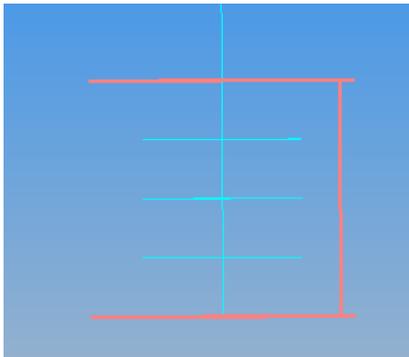


Figure 83 - Helical Interpolation not active

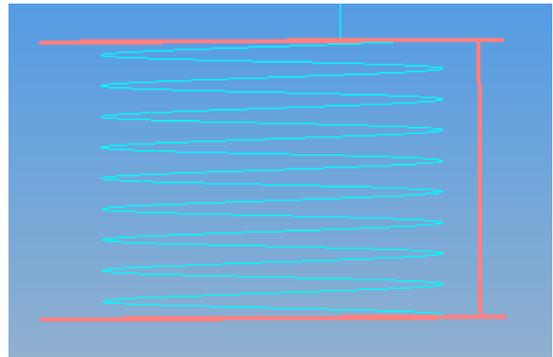


Figure 84 - Helical Interpolation active



Note that your post processor and machine need to support simultaneous 3 axis movement to be able to use the Helical option.

NC Code for Multiple cuts

This option allows you to specify how the area clearance tool paths are output.

- Linear** Creates NC code in a single program file. Each cut down in the Z axis is repeated.
- Subroutine** Creates a separate sub section or program file for the pocket cycle and this is then called by the main program.

Subroutine output is only applicable with vertical sides, as all the other options cause the pattern at each cutting level to change. If this tool path is duplicated it will automatically be reset to linear code.



Note that your machine control must support the use of Subroutines in any form to be able to use this method.

Depths of Cut

This option is only applicable when more than 1 cut has been entered in the **Number of Cuts**, each of the cuts may be made equal in depth if you click the **Equal radio button**.

If you wish to specify cut thicknesses for the first and last cuts, select the **Specified** radio button and enter the required values in the relevant entry fields.

If you have specified more than 2 cuts, the first and last will be as specified and the remaining cuts will divide the balance of the total cut depth into equal cuts in Z.

Machining Data

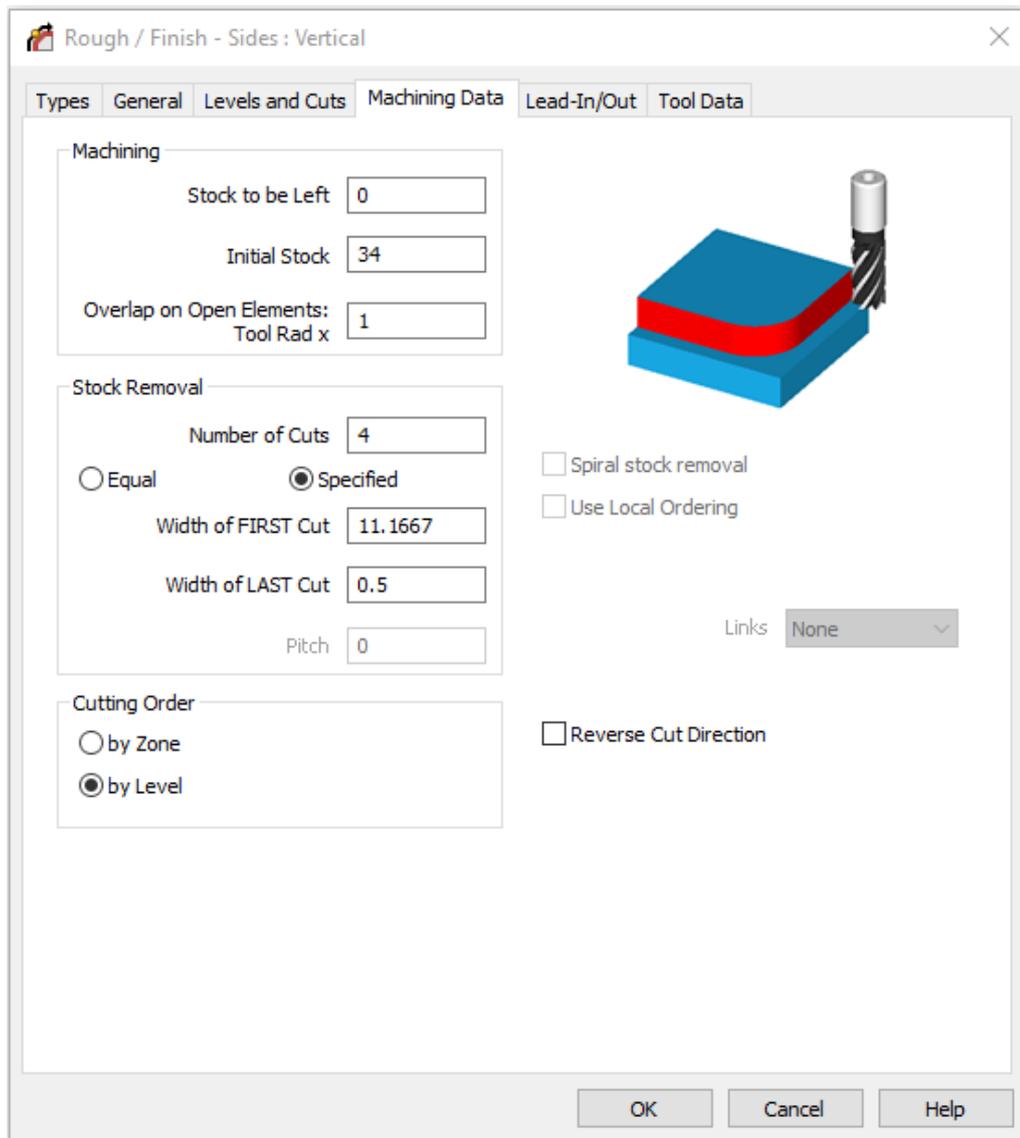


Figure 85 - Machining Data tab on Rough or Finish

Machining Options

Stock to be left	The amount of material not cut by the profiling cut.
Initial Stock	Is the distance from the geometry where the profiling process begins.
Number of Cuts	This is the number of cuts in XY to approach the geometry from the Initial stock value.
Overlap on Open Elements	The amount the tool overlaps open elements. See (MACHINE > Set/Unset Open Elements).

Widths of Cut

- Equal All XY cuts equal.
- Specified Allows you to set the width of the first and last cut in XY; any additional cuts are split equally with the amount left.

Cutting Order

- by Zone Machines the cuts in Z first prior to stepping across to the next cut in XY on the same geometry.
- by Level Machines the cuts in XY first prior to returning to the outer start point and then machining to the next Z depth on the same geometry.

Spiral stock removal

Only available when using the **Tool Centre** compensation option from the General tab. The machining method applies a constant Z level toolpath that spirals towards the selected geometry in the XY axis moves. The amount of step over for the toolpath is controlled using the Pitch dialogue. For each travel around the selected geometry profile when the tool path reaches the initial start point, the tool will have moved across in the XY plane by the Pitch value.

Use Local Ordering

Only applicable when more than one profile is to be selected. Works in a similar method to Cutting Order but applies the calculations to the multiple profiles.

Reverse Cut Direction

Setting this option active allows you to cut the same geometry in opposite directions without having to alter the assigned **Ghost Tool** direction, which could have a serious impact on already applied machining.

Links

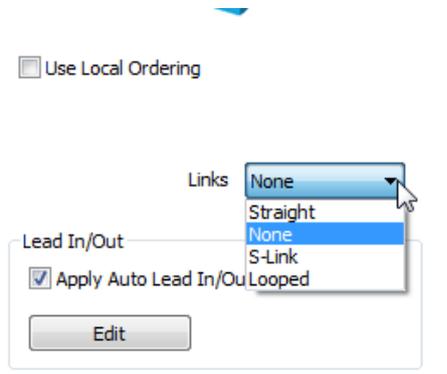


Figure 86 - Links drop down options

Links are only available when **Cutting Order** is set to **by Level**. The drop-down selector allows for different transition moves between the XY cuts.

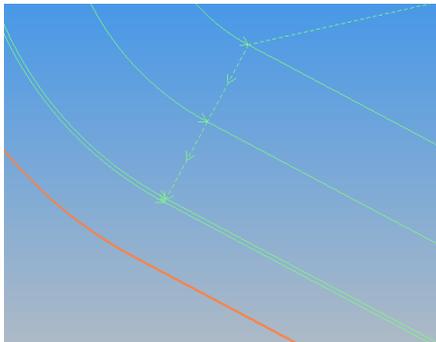


Figure 87 - Link set to None

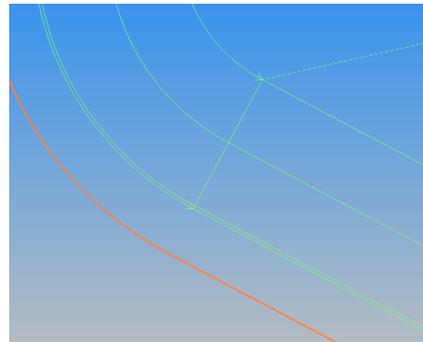


Figure 88 - Link set to Straight

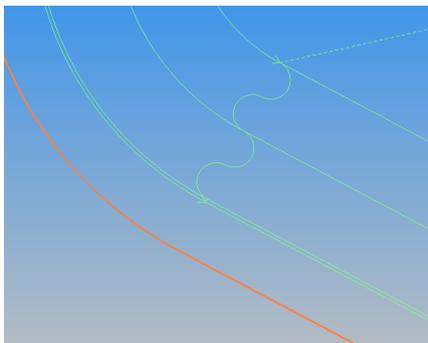


Figure 89 - Link set to S-Link

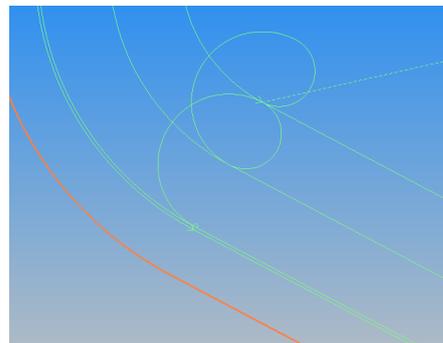


Figure 90 - Link set to Looped

If **Lead-In/Out** moves are applied, the **Lead-In** is applied to the start of the first path and the **Lead-Out** is applied to the end of the last linked path.

The **Straight** and **S-Link** options can only be applied if **Compensation** is set to **Tool Centre**. Otherwise, the following message is displayed.



Figure 91 - Warning if using Links with the wrong Machine Compensation option

Lead-In/Out

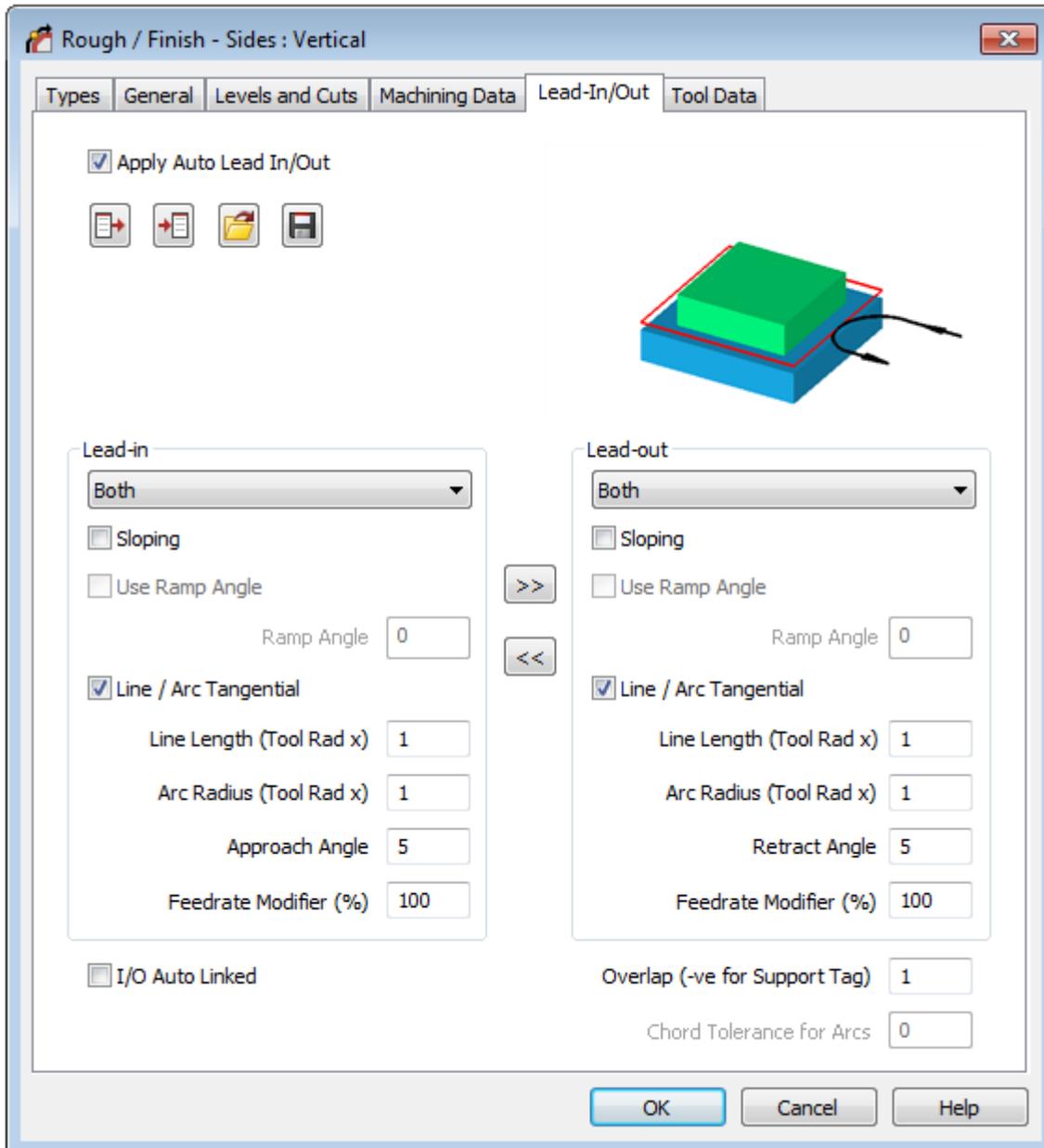


Figure 92 - Lead In/Out tab on Rough or Finish

By activating the Lead-In/Out option **Apply Auto Lead In/Out** you can assign specific approach and departure paths to the applied complete toolpath.

When Rough or Finish profiling it is normal to apply a lead in and/or out to the tool path. It is normally mandatory when using cutter compensation (G41/42) option.

Lead in / Out options can be applied retrospectively either by editing the operation in the project manager or in the case of pocketing tool paths using the **MACHINE > Tool Lead in/out** 

The lead in/out command provides a dialogue so that you can select the style of lead in and the style of lead out you to apply on the selected tool path.

Lead-In/Out options

Apply Auto Lead In/Out



Figure 93 - Lead-In/out data file options

- | | | |
|---|------------------------------|--|
|  | Get Defaults | Allows the user to load up the default options commonly used during day to day manufacturing. |
|  | Save Defaults | Allows the user to Save the current applied options as a prebuilt strategy to use in the day to day manufacturing processes. |
|  | Load Lead-In/Out Data | Allows for the loading of a Saved set of Lead-In/Out instructions for specific conditions where the Default options are unsuitable or not appropriate. |
|  | Save Lead-In/Out Data | Saves the current Lead-In/Out options so that they can be used with the above command. |

Style of Lead-In/Out

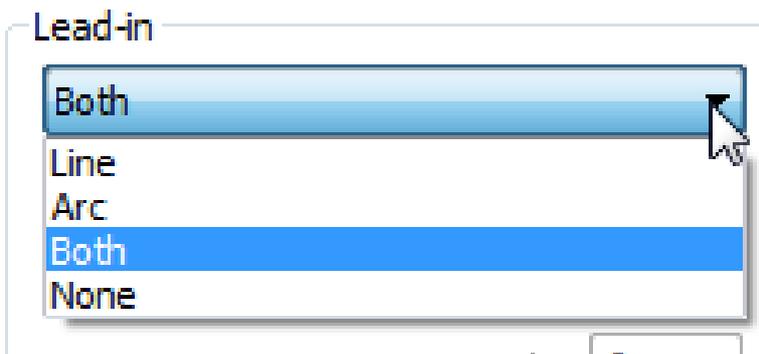


Figure 94 - Drop down selector for the type of Lead-In

The drop-down selector for both Lead-In and Out allows various combinations of approach and departure on the tool path.

Tool Entry, Lead In/Out options

Line

Lead-in

Line

Sloping

Use Ramp Angle

Ramp Angle

Line / Arc Tangential

Line Length (Tool Rad x)

Arc Radius (Tool Rad x)

Approach Angle

Feedrate Modifier (%)

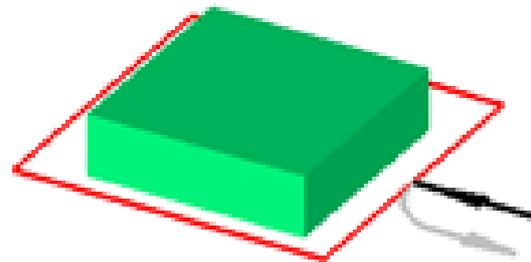


Figure 96 - Line Lead-In using a 90° approach angle

Figure 95 - lead-In/Out using Line

The **Line** options allow for a single straight line Lead-In or Lead-Out to be applied to the part.

Line Length (Tool Rad x) dictates the length of the approach/departure line.

Approach Angle specifies the angle by measuring against the actual face to be machined, in the image above a value of 90 is being used.

The **Feedrate Modifier (%)** can slow down or speed up the programmed feed depending on the machining conditions.

Sloping will create a ramping style of line that creates a calculated angle for the slope based on the **Line Length** and the **Rapid Down** to values.

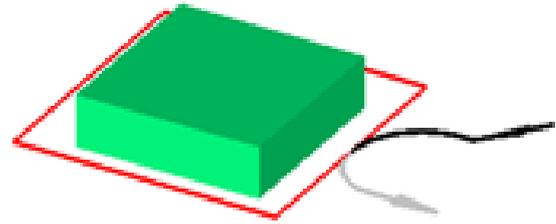


Figure 97 - Sloping option active

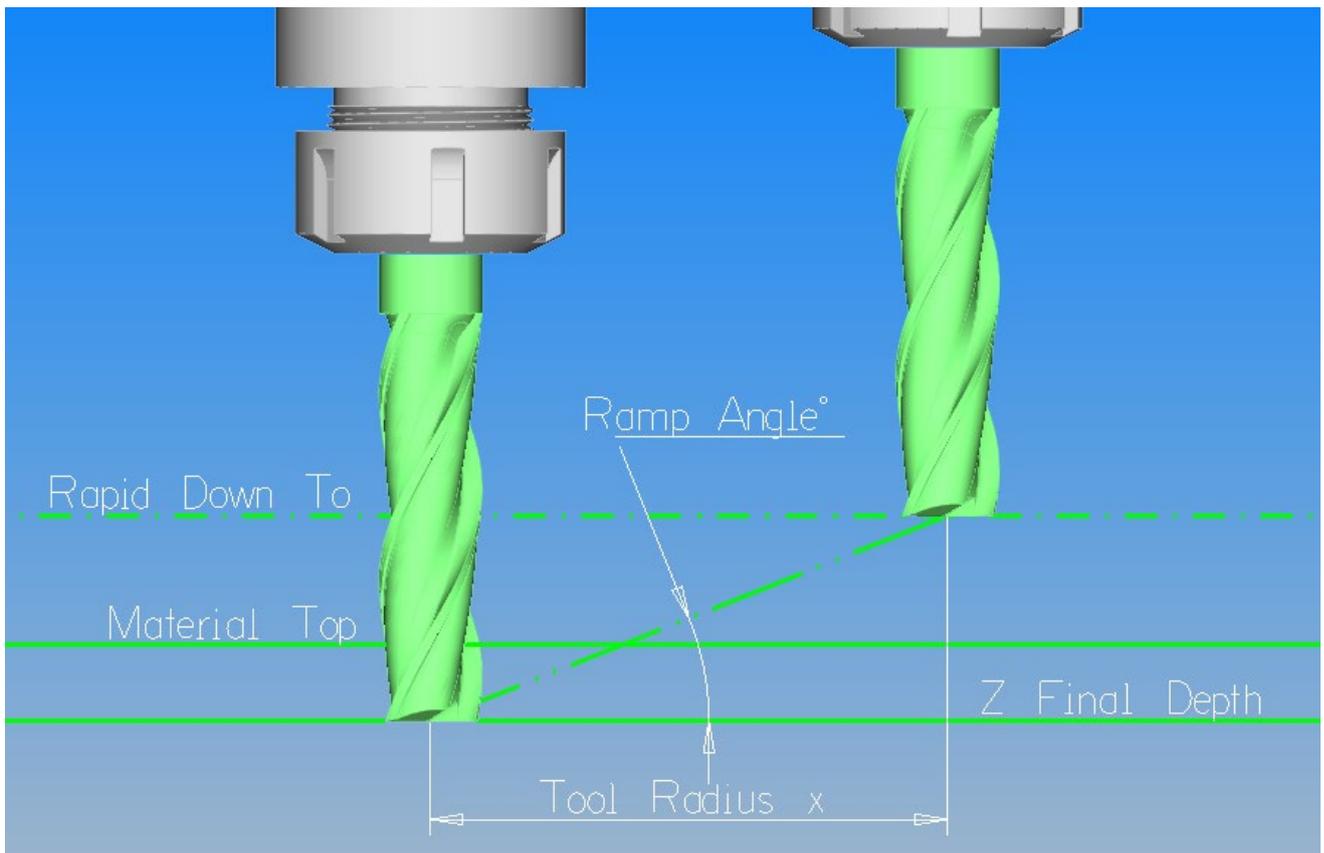


Figure 98 - Calculation of the Slope Angle

Use Ramp Angle will create a ramping style in the same manner as shown above except that the user can define the actual angle value in the **Ramp Angle** dialogue.

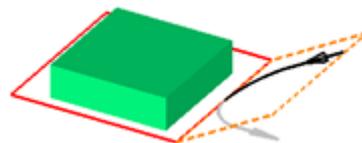


Figure 99 - Use Ramp Angle active

Arc

Lead-in

Arc

Sloping

Use Ramp Angle

Ramp Angle

Line / Arc Tangential

Line Length (Tool Rad x)

Arc Radius (Tool Rad x)

Approach Angle

Feedrate Modifier (%)

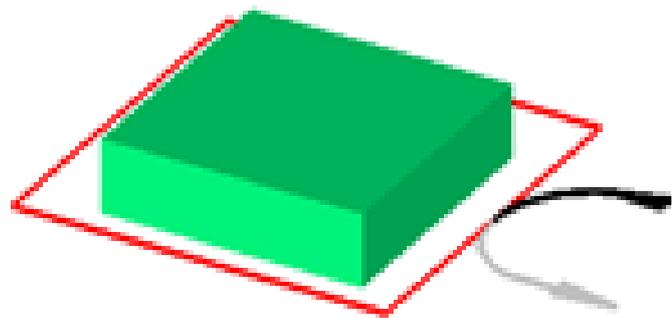


Figure 101 - Arc Lead-In using a 90° approach angle

Figure 100 - lead-In/Out using Arc

The **Arc** options allow for a single arc Lead-In or Lead-Out to be applied to the part.

Arc Radius (Tool Rad x) dictates the length of the approach/departure line.

Approach Angle specifies the angle by measuring against the actual face to be machined, in the image above a value of 90 is being used.

The **Feedrate Modifier (%)** can slow down or speed up the programmed feed depending on the machining conditions.

In the same manner as described previously, the **Sloping** and **Use Ramp Angle** options are also permissible when using the Arc option.

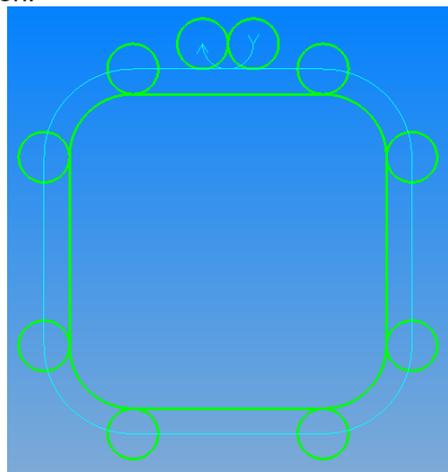


Figure 102 - Arc Lead-In and Out



Most machines **CANNOT** use the Arc only option when using compensation. Make sure your machine can apply Compensation on arcs **ONLY** or on rapids before using this option.

Both

Lead-in

Both ▼

Sloping

Use Ramp Angle

Ramp Angle

Line / Arc Tangential

Line Length (Tool Rad x)

Arc Radius (Tool Rad x)

Approach Angle

Feedrate Modifier (%)

I/O Auto Linked

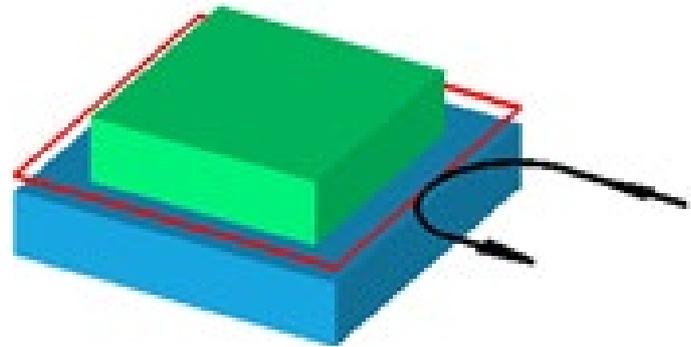


Figure 104 - Both Lead-In using a 90° approach angle

Figure 103 - lead-In/Out using Both

The **Both** options allow for a combination of both previous described options of Lead-In or Lead-Out to be applied to the part.

Line Length (Tool Rad x) dictates the length of the approach/departure line.

Arc Radius (Tool Rad x) dictates the length of the approach/departure line.

Approach Angle specifies the angle by measuring against the actual face to be machined, in the image above a value of 90 is being used.

The **Feedrate Modifier (%)** can slow down or speed up the programmed feed depending on the machining conditions.

In the same manner as described previously, the **Sloping** and **Use Ramp Angle** options are also permissible when using the Arc option.

Line / Arc Tangential creates a smooth flowing Lead-In/Out when active, if turned off, the Line will be at 90° to the Arc.

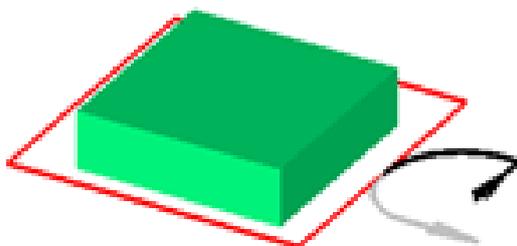


Figure 105 - Tangential option off

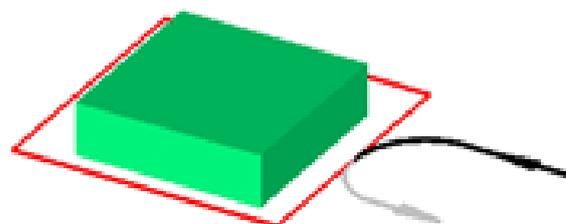


Figure 106 - Tangential option on

I/O Auto Linked allows the user to maintain a form of control over the retraction of the Z axis during machining where a toolpath is worked in a number of cuts down the Z axis.

With the Auto Link option off, on multiple depth toolpaths the Z axis will retract to the **Feed Down** distance after each cut and then return to the toolpath start location.

With the option active, the Start and Finish point of the toolpath, where applicable, are made coincidental and between Z depth cuts the tool will not retract to the **Feed Down** distance.

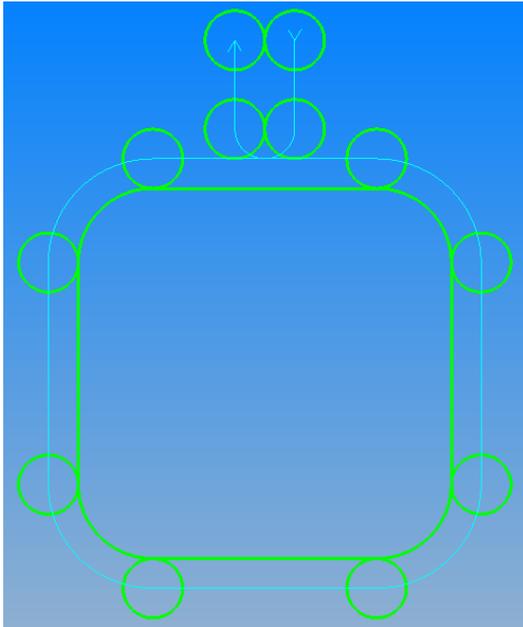


Figure 107 - Auto Link off

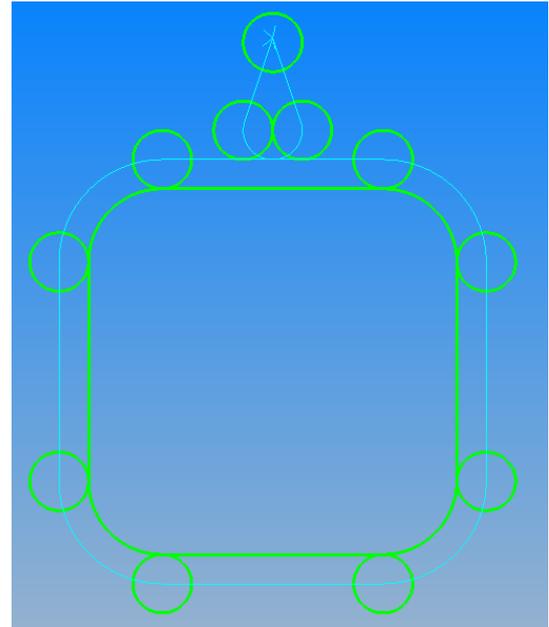


Figure 108 - Auto Link on, Tangential On

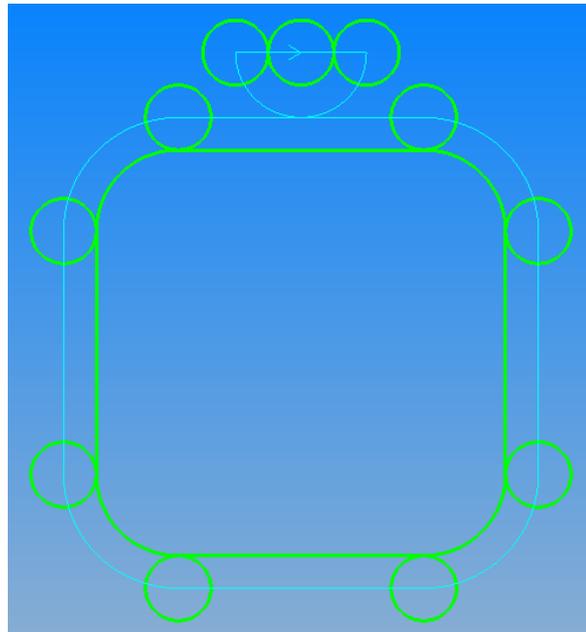


Figure 109 - Auto Link On, Tangential Off

Overlap

Overlap (-ve for Support Tag)

Figure 110 - Overlap value options

On all styles of **Lead-In/Out** it is possible to add an overlap value to allow the toolpath to continue past the original start point by the value applied to the dialogue box.

This can reduce the appearance of dwell or burn marks on parts where the start and finish of the toolpath are in the same location.

Using a positive value will cause the tool to pass over the original start point; conversely using a negative value will stop the toolpath short of the original start point and create a supporting link to the original material.

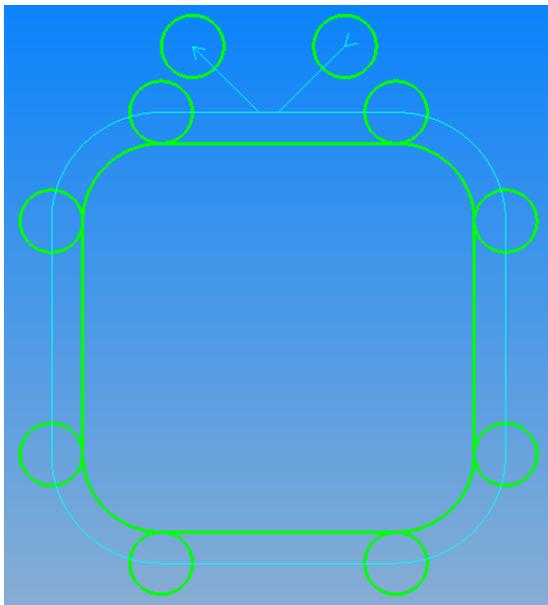


Figure 111 - Using a positive overlap value

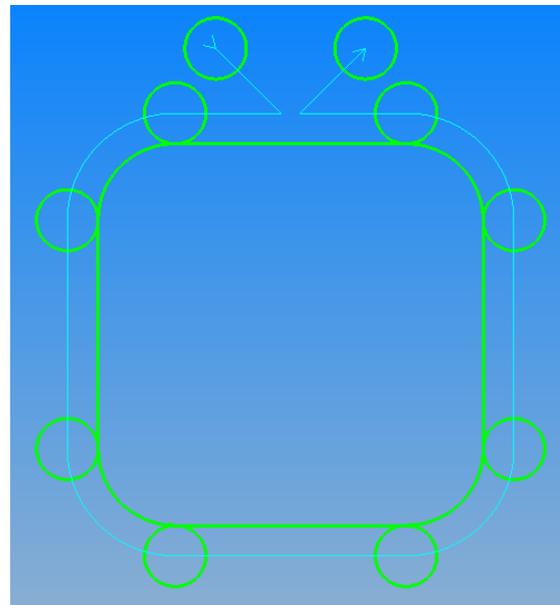


Figure 112 - Using a negative overlap for a support tag

Tool Data

This page is common to all option choices.

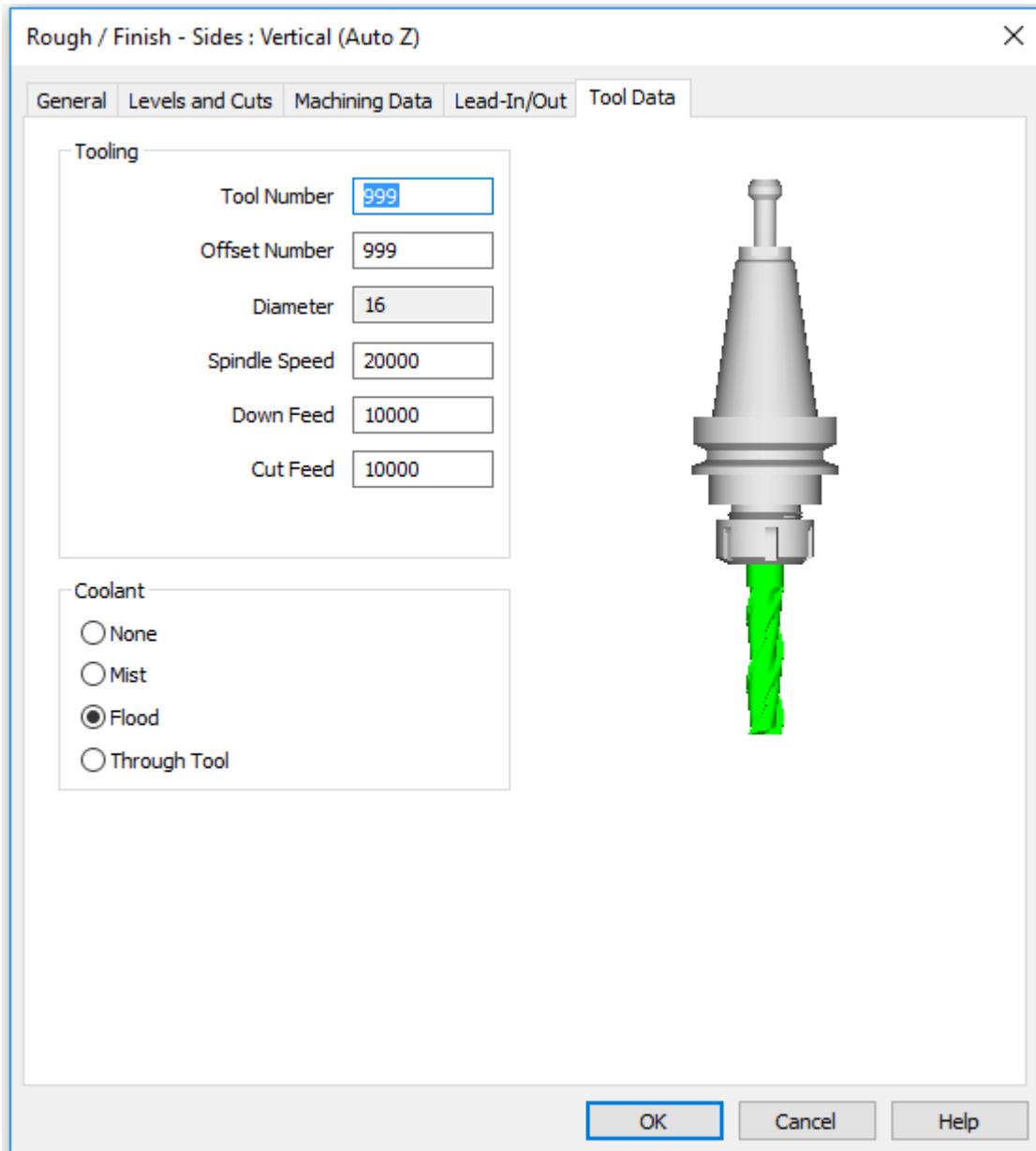


Figure 113 - Tool Data tab

Non Vertical Sides (Standard and higher modules)

Allows you to assign a specific uniform angled side to a geometric profile with options for blend radii on the top edge and the bottom edge.

This type of additional manipulation allows for draft angles to be applied to vacuum mould forms for example.

Pocketing using Sloping

After the primary dialogue for selecting the pocketing method in standard pocketing and the secondary dialogue when using the Contour Roughing option, one of the following dialogues will appear.

Standard Pocketing

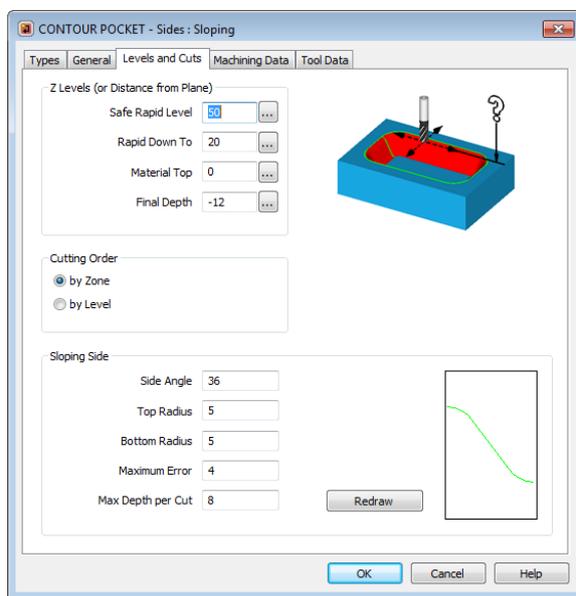


Figure 114 - Sloping Sides Levels and Cuts tab standard pocketing

Z Contour Roughing Pocketing

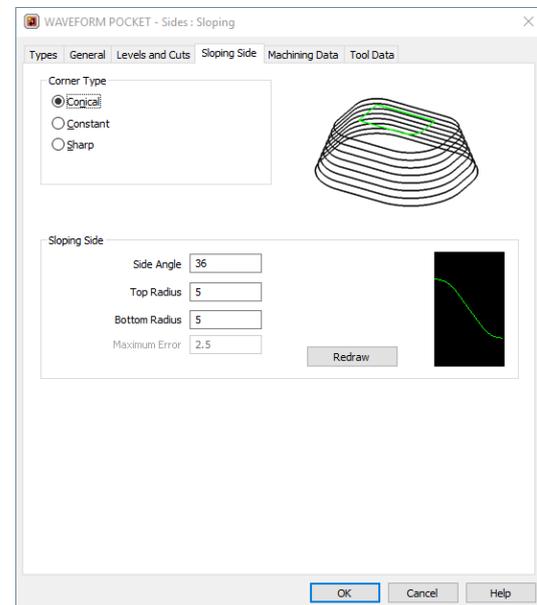


Figure 115 - Sloping Sides tab Z Contour Roughing pocket

Side Angle The required slope angle for the part in question where 0° would be a vertical wall.

Top Radius A blend between the material top and the angled side, 0 gives a sharp corner.

Bottom Radius A blend between the angled side and the lower depth of cut value, 0 gives a sharp corner. By default, this is set to the radius of the chosen cutter. The dialogue also displays a graphic of the form for verification from the user.

On the Standard dialogue use the **Maximum Error** for setting the width of step over for depth of cut calculation and the **Max Depth per Cut** for the step down in the slope direction should the Maximum Error be exceeded.

The Contour Roughing dialogue also allows for corner transitions using a Conical, Constant or Sharp option. These allow the user to alter the style of transition in the same manner as the XY cornering options in the Rough and Finish cycles when islands are present within the pocket.

Rough and Finish using Sloping

The Sloping Sides Finishing options are the same when it comes to settings as the Pocketing option with the addition of an option to cut **Bi-Directionally** if the tool path is created on an Open geometry.

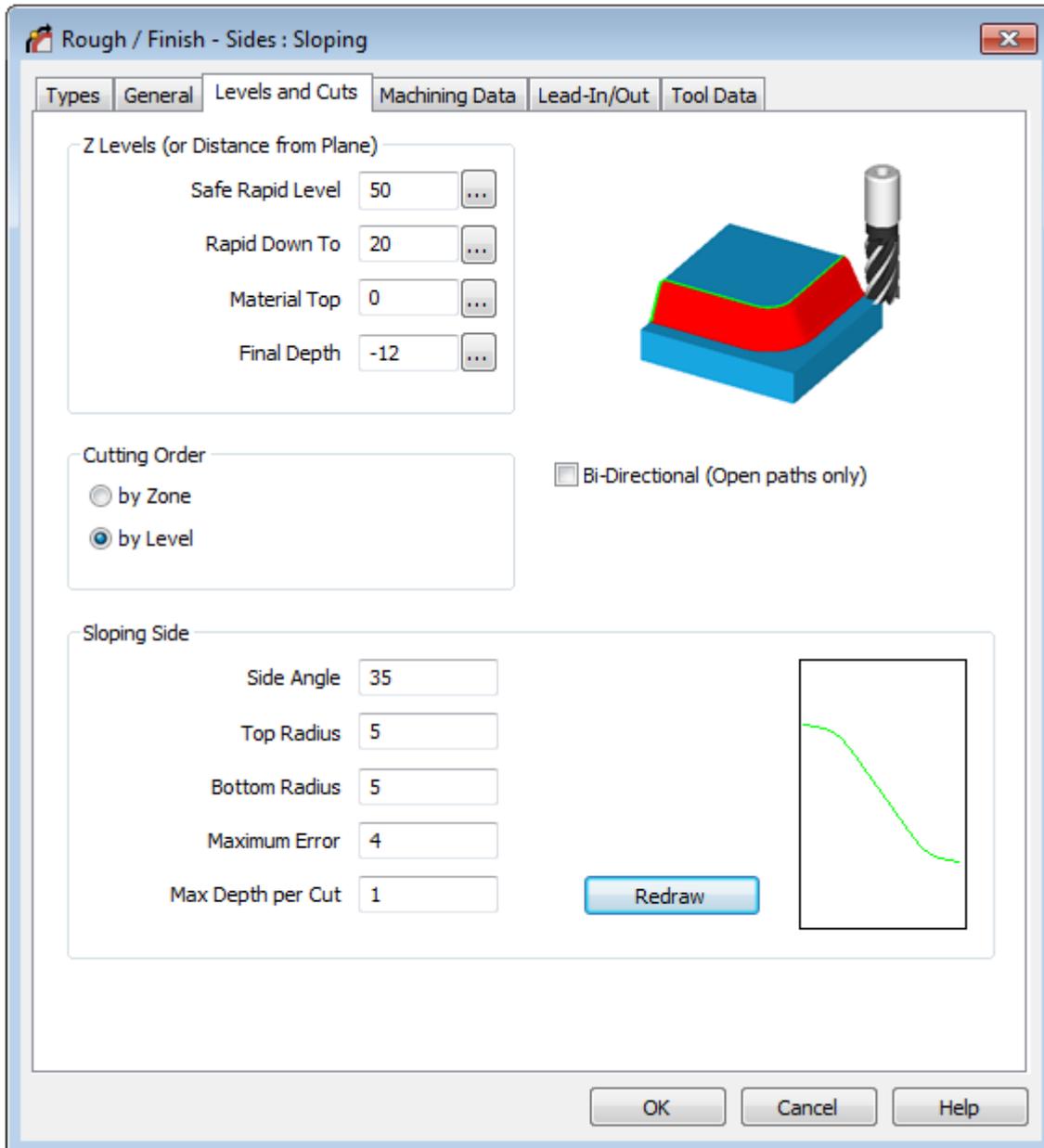


Figure 116 - Levels and Cuts tab using sloping sides

Profiled

Allows you to assign a geometric cross section to a geometric profile that can then create a 3D machined part without the use of 3D machining.

The two requirements for this are;

- The geometric profile that you wish to assign to.
- The actual profile shape of the final part.

Using **Profiled** allows you to create side profiles that would require a form cutter or User Defined cutter on parts where the cost of the tooling may be too expensive, one off production items, or you are wishing to create a sample for a client prior to investing cost into a specific tool prior to a full production run.

Pocketing using Profiled

After selecting the Pocketing cycle and choosing the method of the pocket, you are then shown the following dialogue option.

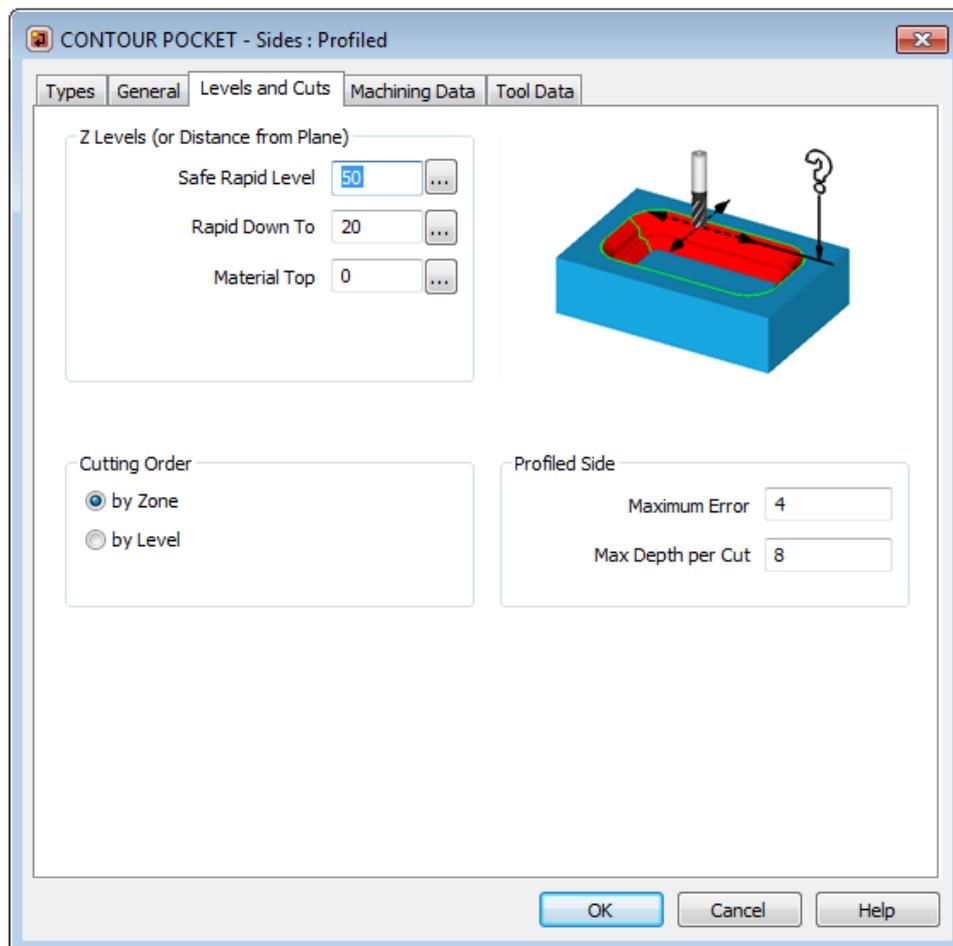


Figure 117 - Levels and Cuts tab using profiled sides on pocketing

Note that there is no requirement for a **Z Final Depth** as this is controlled by the side profile form, however long it is, that is how deep you will cut.

Once all of the requirements in the remaining dialogue boxes have been completed you are then requested to **Pick Side Profile**.

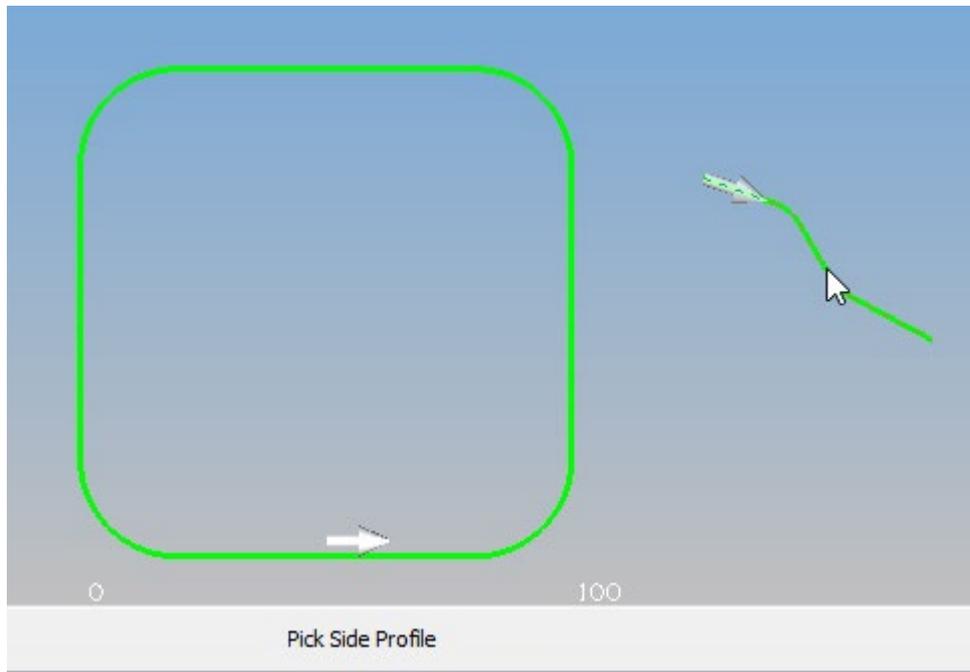


Figure 118 - Prompt to pick the cross section profile

The side Profile in the image is the item to the right side of the rectangle. After **<LClick>** on the profile you are asked for an application or Reference point.

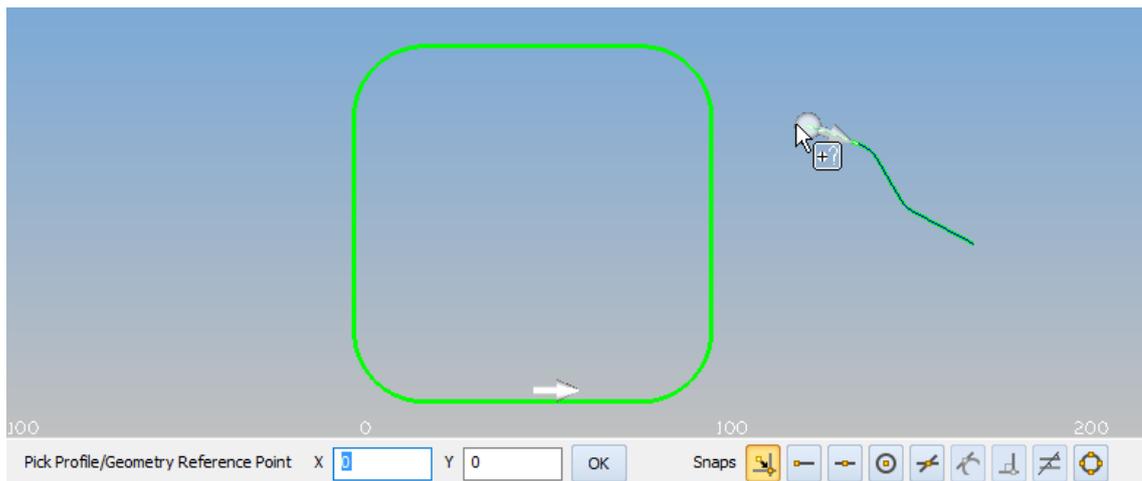


Figure 119 - Prompt for the application point of the cross section

This Reference Point is the location on the cross-section profile that will be attached to the driving profile to create the finished part.

Once the Reference Point is selected, you are prompted for the geometries to apply the side profile to.

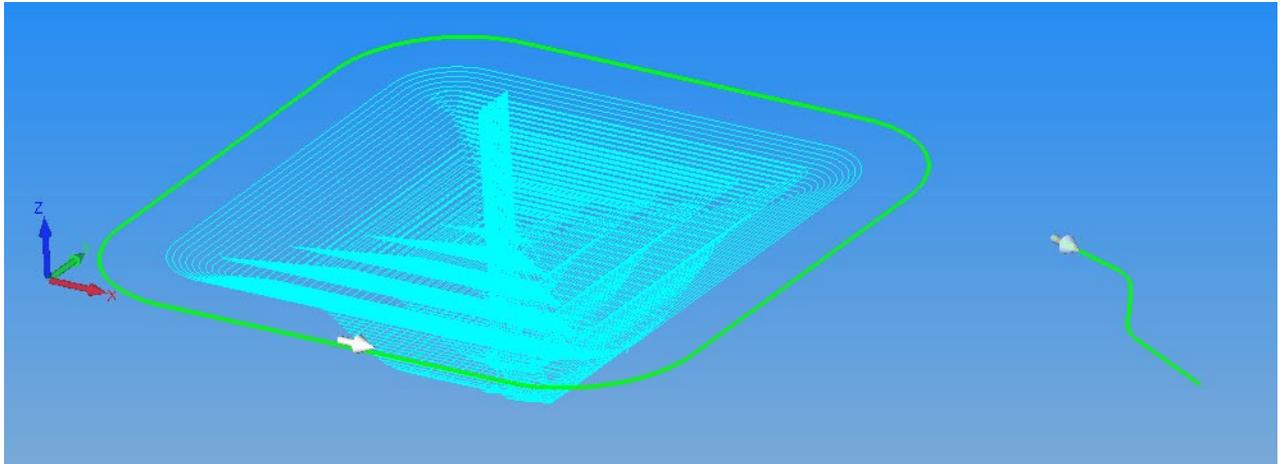


Figure 120 - Wireframe toolpath of Profiled Sides pocket

The pocket toolpath is created.

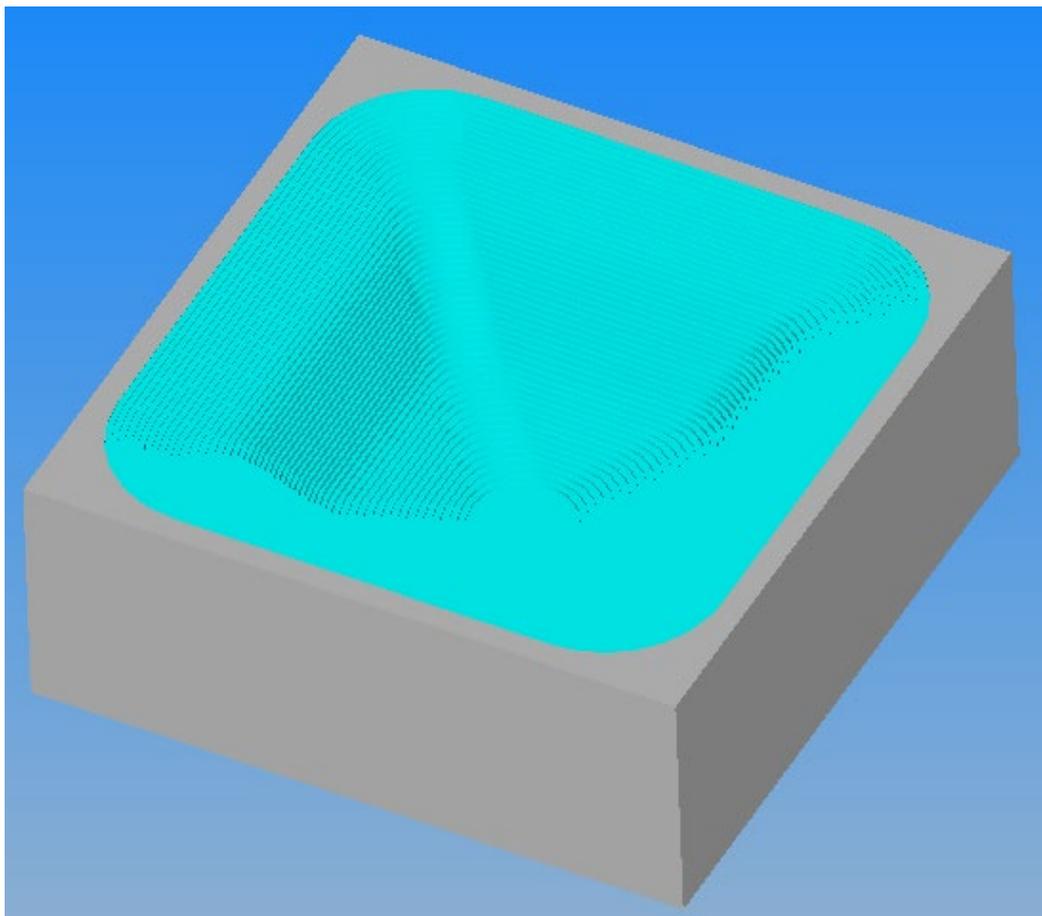


Figure 121 - Simulated Profiled Sides pocket

Rough and Finish using Profiled sides

The **Profile Sides** finishing options are the same when it comes to settings as the Pocketing option with the addition of an option to cut **Bi-Directionally** if the tool path is created on an Open geometry.

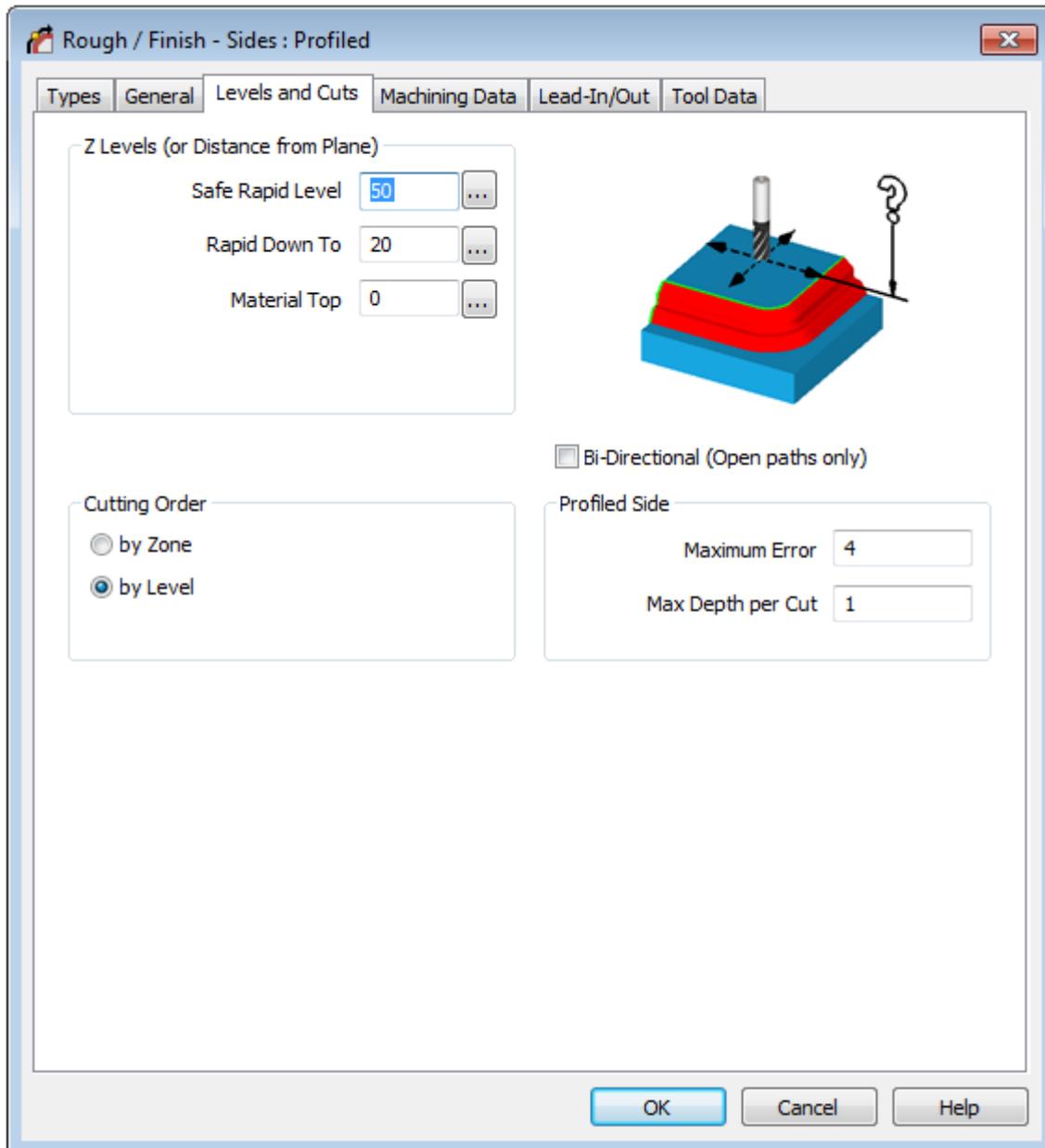


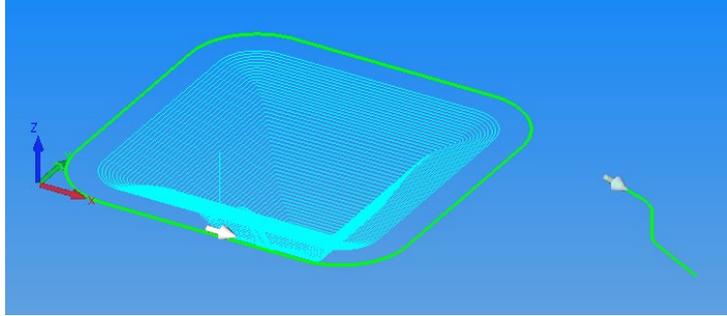
Figure 122 - Levels and Cuts tab using Profiled Sides on Rough or Finish cycle

Tool Direction and Reference Points are more important during the Rough and Finish cycle when using the Sloping option.

Depending on the form required the Tool direction still dictates whether the profile will work Inside, Outside, Left or Right of a line, but the Reference point then dictates if the profile machines below the selected driving geometry or above the driving geometry.

Tool Direction

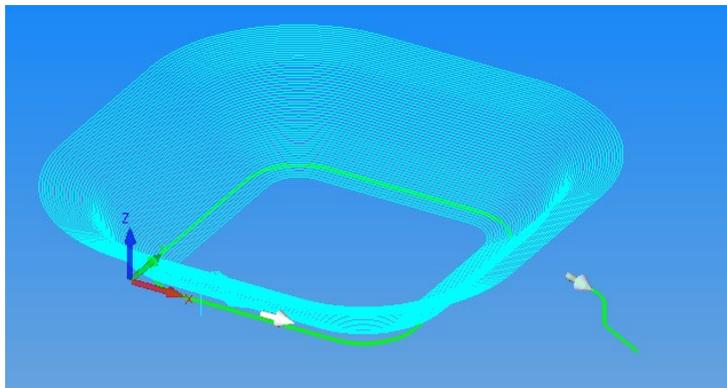
Inside.



Reference Point

On Ghost Tool arrow.

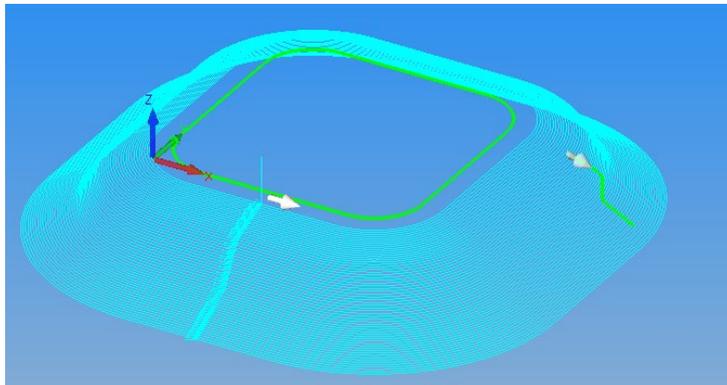
Inside.



Opposite Ghost Tool arrow.

Tool Direction

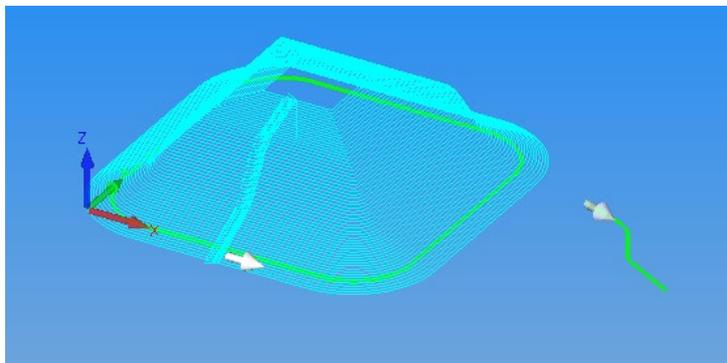
Outside.



Reference Point

On Ghost Tool arrow.

Outside.



Opposite Ghost Tool arrow.

If the following dialogue appears whilst you are applying the side profile.

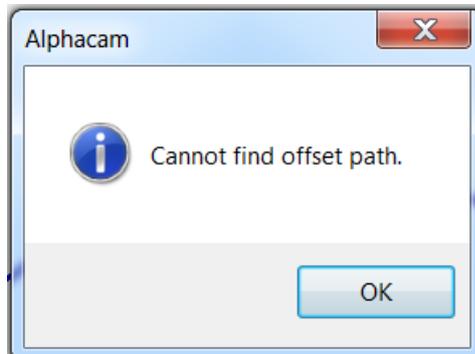


Figure 123 - Error displayed when a cross section profile would overlap itself

This will indicate that the side profile will intersect itself and the form cannot be created as currently drawn.

You may need to redesign the cross section or manipulate the final shape geometry to allow the shape to be created.



In the same manner that applying an actual form cutter to a part would destroy the job if the shape overlapped, ALPHACAM is letting you know that the tool path will not work prior to you even applying it.

Previous Tool Profile



Figure 124 - Previous Tool Profile application

In the same way **Take Account of Previous Machining** allows for a smaller cutter to machine areas left by a much larger cutter, **Previous Tool Profile** checks against the preceding cutter.

Should this be a simple tapered ball end cutter as the image shows, then the smaller ball end cutter will machine the areas in which the larger cutter was unable to reach.

This option can also be applied to more complex form tools in the same manner.

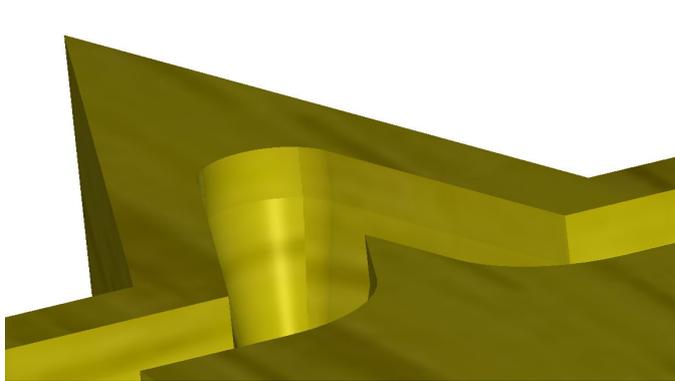


Figure 125 - Original Applied Rough cycle

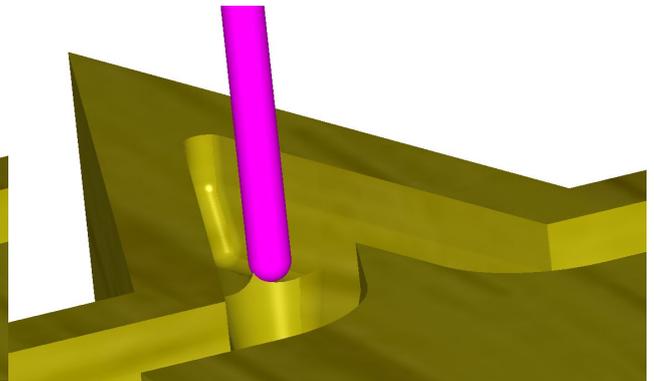


Figure 126 - Using the Previous Tool Profile option

Chamfer Profiling

When producing chamfers using a profiling cut (Rough or Finish), there are a few conditions that should be met depending on the working practices in use.

- The Chamfer stock allowance is calculated at the specified Material Top.
- The Chamfer stock allowance could be **negative** by the X/Y chamfer distance.
- Ensure the depth of cut does not exceed the tools edge length.
- Ensure the depth of cut does not foul any faces.

These conditions will need to be met if using a tool path that is set to Tool Centre.

User Defined tooling can also be used to generate chamfer or bevel profiles using specific location on the actual tool profile for the diameter reference and the Z level reference.

You should refer to the **Material and Tooling** creation documentation for more information regarding setting up **User Defined** tools for this purpose.

Drill/Tap Holes Options.

From the **Machine Section** of the ribbon you have the following options.

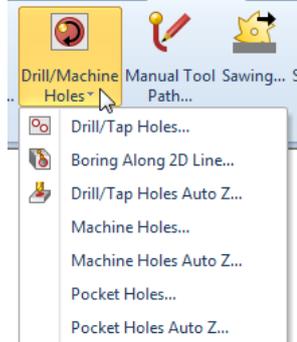


Figure 127 - Drilling options



Note that on some levels of ALPHACAM all these options are not available, and the cycle will be actual icons on the ribbon bar instead.

The **Drill/Tap Holes** options allow specific commands to be used to apply those types of cycles to any selected circles on the drawing.

The **Machine Holes and Pocket Holes** options allows for the same selection process for circles on the drawing but then instead of applying a drilling style cycle, the Pocket or Rough and Finish cycles are accessed instead.

As shown in the image above, **Boring Along 2D Line** is an Add-In which allows side boring to take place in the Essential module of ALPHACAM which has no means of using Work Planes.

General

The first dialogue allows you to select the machining type options: Drilling, Pecking, Tapping, Boring or Chamfering, plus several ways of choosing which geometries will be selected to be machined.

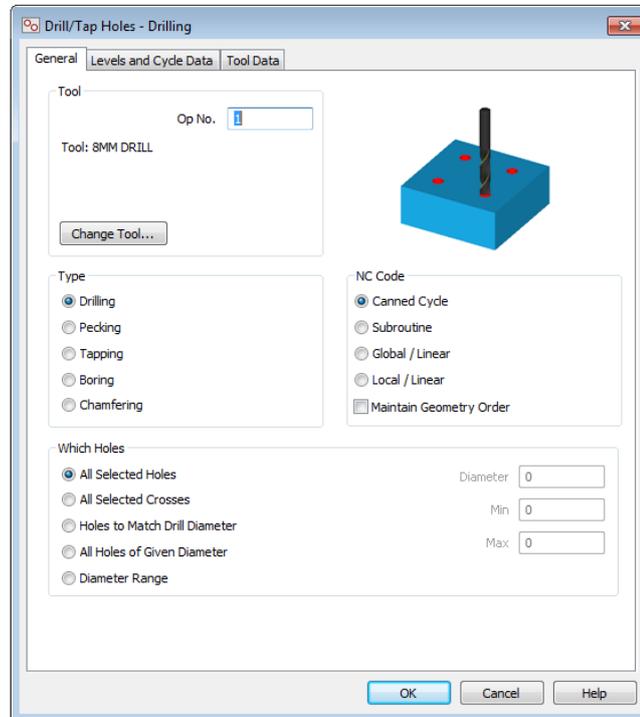


Figure 128 - General tab for all Drilling

Type

- Drilling** A single hit cycle that drives the drill bit from the start location to the finish depth in a single movement.
- Pecking** A multiple hit cycle that employs a rise and fall motion to clear swarf during the drilling cycle.
- Tapping** Allows for the creation of threads in parts with the option to using a pecking style if required.
- Boring** Creates a cycle like the drilling option but allows for a spindle stop and a small move from the part wall to allow the tool to be retracted without marking the part.
- Chamfering** A single hit cycle for the creation of defined depth chamfers.

NC Code

- Canned Cycle Generates an NC code where all the cycle information is contained within a single line.
- Subroutine Assigns the entire listing of code to a separate program or section to allow multiple tools to call the same section without repeating the values within the main NC program.
- Global/Linear Generates a complete 5 axis toolpath with the required vector options to control the head or table.
- Local/Linear Generates a 5 axis toolpath using the 3+2 method to control the head or table.
- Maintain Geometry Order If any re-ordering of toolpaths has been applied, using this option will revert the cycle back to the original drawn geometry order.



It should be noted that the pair of Linear options are only available from the Advanced module and higher of ALPHACAM.

Which Holes

- All Selected Holes Every circle that the user chooses using the <LClick> option will have the cycle applied to it regardless of size.
- All Selected Crosses Should a drawing use crosses to mark hole centres, these can be selected instead of circles.
- Holes to Match Drill Diameter Circles which are the same size as the current active drill will be chosen.
- All Holes of Given Diameter Using this option and the value box that becomes available allows for the choice of specific sizes of circles irrespective of the drill size.
- Diameter Range Like the option above, but allows for an upper and lower limit on the circles to be chosen.

Levels and Cycle Data

In the second dialogue, you set the Z level parameters for the differing options based on the cycle chosen. These are different when using the Auto Z machining options.

Drilling Fixed Z

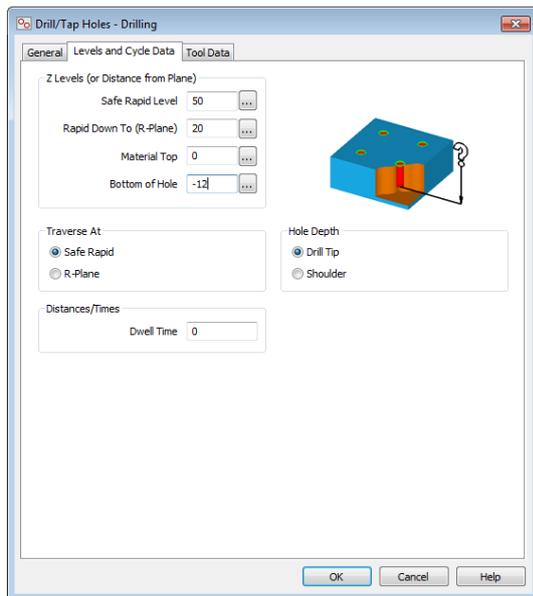


Figure 129 - Levels and Cycle Data tab Fixed Z Drilling

Drilling Auto Z

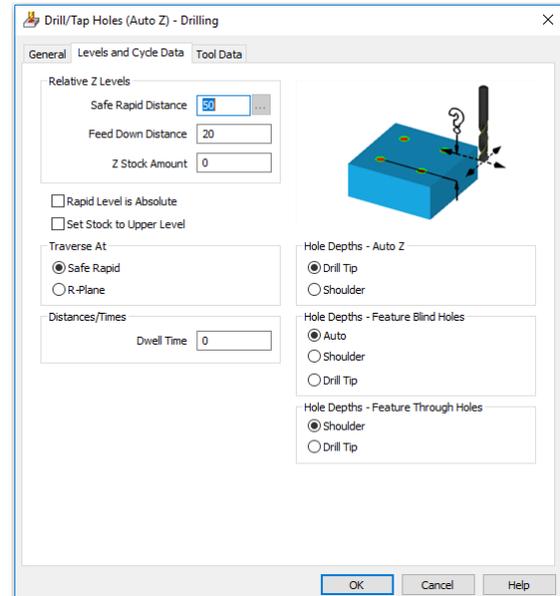


Figure 130 - Levels and Cycle Data Auto Z Drilling

The options for the depth of the hole differ slightly within the cycles.

Fixed Z requires the user to input the **Material Top** and **Bottom of Hole**, whereas the **Auto Z** method only requires a **Z Stock Amount** for any allowances that may need to be considered.

Traverse At

- Safe Rapid** Forces the tool to rise to the full Safe Rapid Level after completion of each cycle prior to moving to the next circle to be drilled.
- R-Plane** Lifts the tool only as far as the **Rapid Down to** or **Feed Down Distance** to save time.

Hole Depth

- Drill Tip** The value stated in the Bottom of Hole or by the lower geometry Z level is used as the final Z value that the end of the drill will position itself.
- Shoulder** ALPHACAM will calculate the required position of the end of the drill using the angle value assigned in the Tool Definition for the drill so that the full diameter of the drill meets the required depth stated in the dialogues.



Note that if using **Fixed Z** drilling and the **Shoulder** option to generate the cycle, should you re-visit the cycle for editing, ALPHACAM automatically resets the option to **Drill Tip** with the correct **Bottom of Hole** value. Auto Z options are not affected.

Pecking Fixed Z

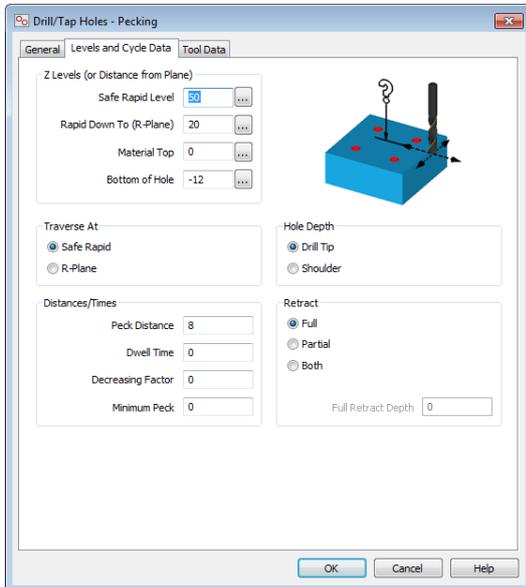


Figure 131 - Levels and Cycle Data Pecking Fixed Z

Pecking Auto Z

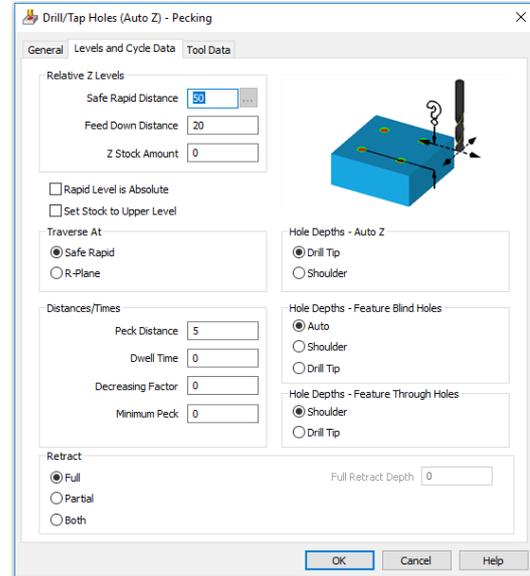


Figure 132 - Levels and Cycle Data Pecking Auto Z

In addition to the same options found in the Drilling cycle, Pecking adds the options for retracting the drill during the cycle via the **Peck Distance** option. The **Retract** options allow for the following;

- Full** The drill is removed from the hole by the full amount it has already machined.
- Partial** The drill is retracted by a small fraction, usually dictated via the actual cycle generated by the post processor, this can save time.
- Both** A combination of both options which after several partial retractions, the drill is completely removed from the hole to allow the material build up to clear prior to continuing drilling. Usually employed on deep holes.

The **Decreasing Factor** can allow for calculations to take place to reduce the depth of a pecking cycle as the hole gets deeper, down to a **Minimum Peck** distance that will be employed until the final depth of the drilled hole is reached.



The **Decreasing Factor** option, whilst available in the cycles, is not a standard option in supplied post processors. You should contact your ALPHACAM support team should you wish to employ this function as a post processor modification will be required.

Tapping Fixed Z

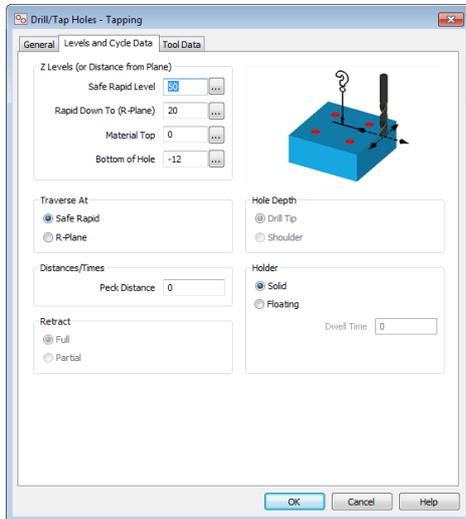


Figure 133 - Levels and Cycle Data Tapping Fixed Z

Tapping Auto Z

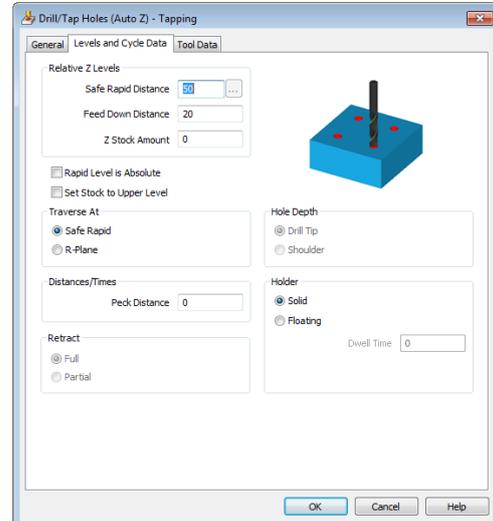


Figure 134 - Levels and Cycle Data Tapping Auto Z

The only additional option applied to **Tapping** is the ability to state whether a self-reversing **⊙ Floating** head type unit is being used, or Rigid Tapping is the chosen method via the **⊙ Solid** option.

Boring Fixed Z

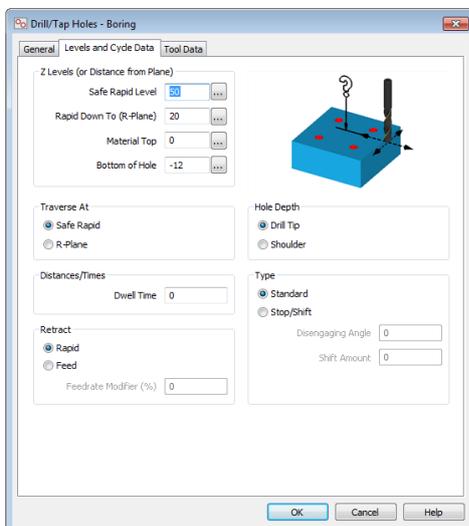


Figure 135 - Levels and Cycle Data tab Boring Fixed Z

Boring Auto Z

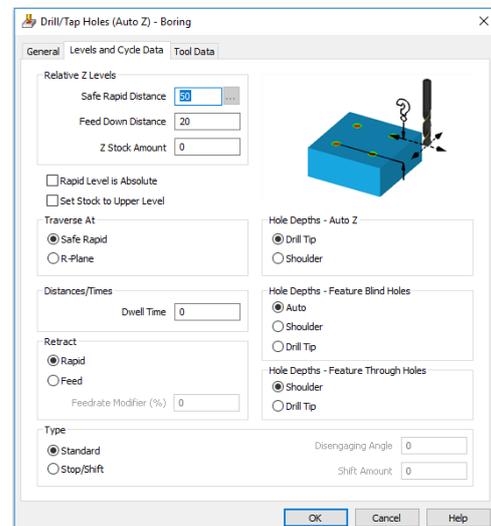


Figure 136 - Levels and Cycle Data tab Boring Auto Z

Additional options for Boring via the **Type** section.

⊙ Standard

In this method, the boring bar is driven to the Final Z depth of the hole, and then under the same Z feed rate the tool is retracted from the hole.

⊙ Stop/Shift

This method will stop the boring bar once the hole depth has been reached, rotate the boring bar to a **Disengaging Angle** and then step off the part by the **Shift Amount**.

Chamfering Fixed Z

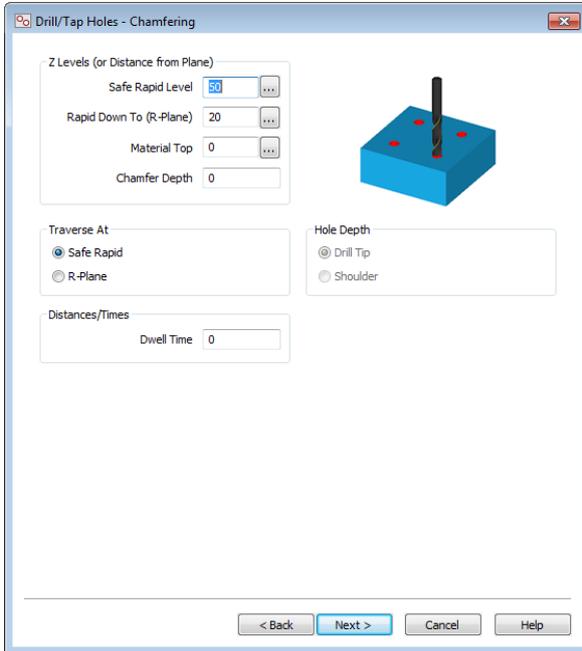


Figure 137 - Levels and Cycle Data Chamfering Fixed Z

Chamfering Auto Z

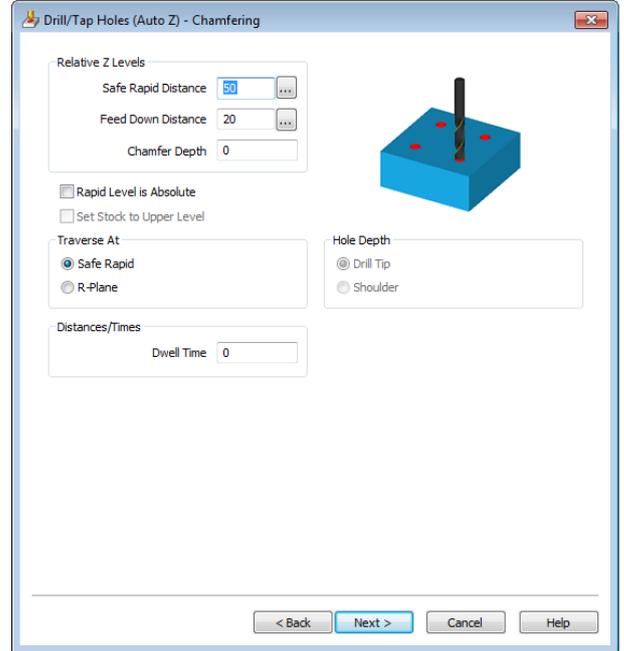


Figure 138 - Levels and Cycle Data Chamfering Auto Z

Common options on all Drill/ Tap Holes Auto Z options.

Auto Z Machining is like the machining methodologies previously described; the difference being that the machining depths are taken from the Geometries' associated Z levels instead of being manually input in the operation.

To use Auto Z machining the geometries MUST have their associated Z Levels defined.

To associate the Z level attributes to geometries, use **3D > Set Geometry Z Levels**.
Select the geometry(s), **<RClick>**, enter the profile top and bottom Z level values.



Two options can be applied on all the Auto Z hole creation options listed above;

Rapid Level is Absolute

This option will alter the **Feed Down Distance** and **Safe Rapid Distance** from being Relative to the current active work plane to be in relation to the Global Origin. This can allow extra clearance when working on multi plane parts.

Set Stock to Upper Level

During all the above cycles, the final depth of the hole has been based on the lower geometry Z value applied to the chosen geometry, selecting this option will now set the depth of the hole to be relevant to the higher geometry instead.

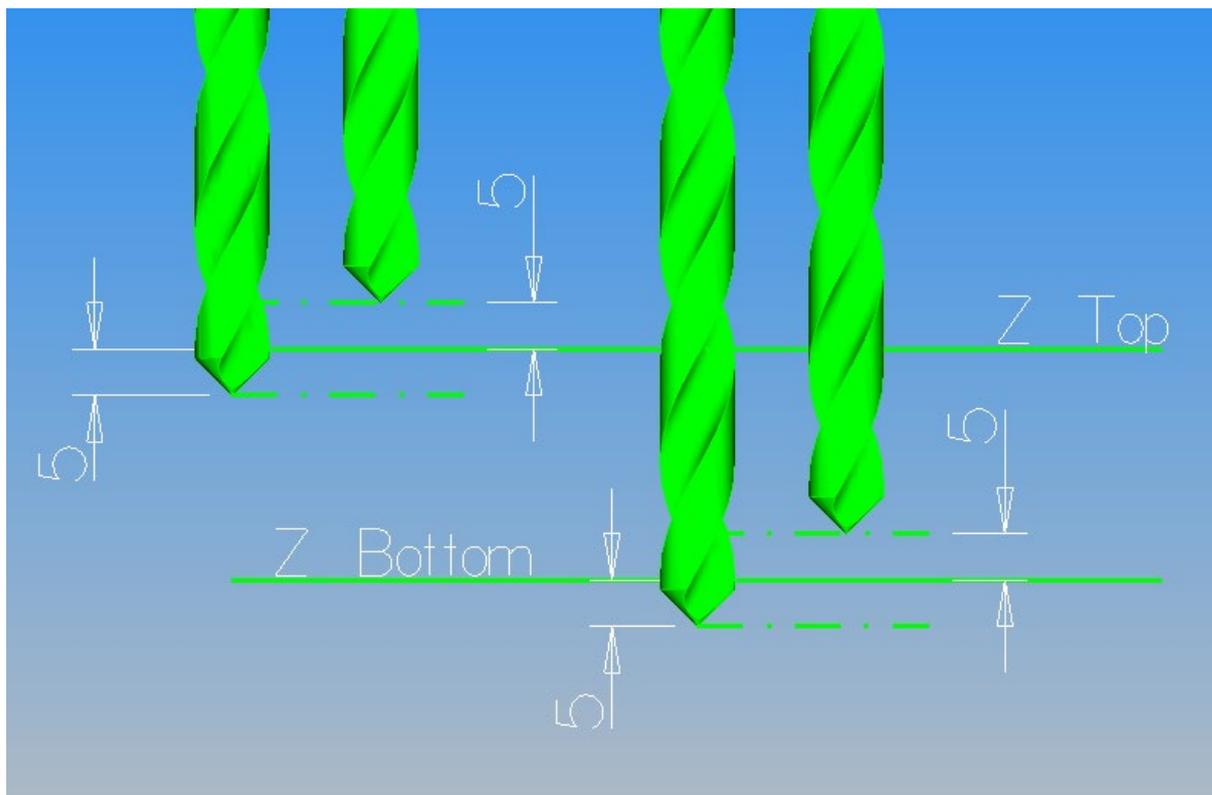
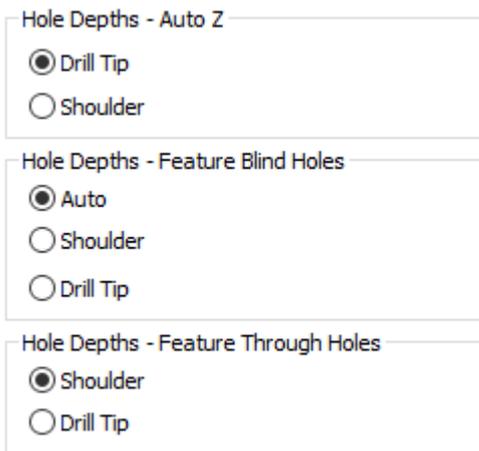


Figure 139 - Setting Stock options in Auto Z Drilling operations

This makes it easier to apply Auto Z hole spot drilling and hole chamfers.

Auto Z Hole Depth Selection



Hole Depths - Auto Z

Drill Tip

Shoulder

Hole Depths - Feature Blind Holes

Auto

Shoulder

Drill Tip

Hole Depths - Feature Through Holes

Shoulder

Drill Tip

Figure 140 - Auto Z Hole Depth Options

As detailed earlier, Auto Z still uses the Drill Tip or Shoulder option to allow control over the final depth of a hole but it also allows for finer control on more complex 3D Solid Models.

On **Feature Blind Holes** (Feature Extracted Holes that do not pass through a part) the default option is set to Auto.

ALPHACAM will check the described drill angle from the Tool Definition dialogue against the Feature Extracted geometries and stop the drill when the first point of contact is made. You can force Shoulder or Drill point should you wish.

On **Feature Through Holes**, the default option is set to the shoulder.

Tool Data

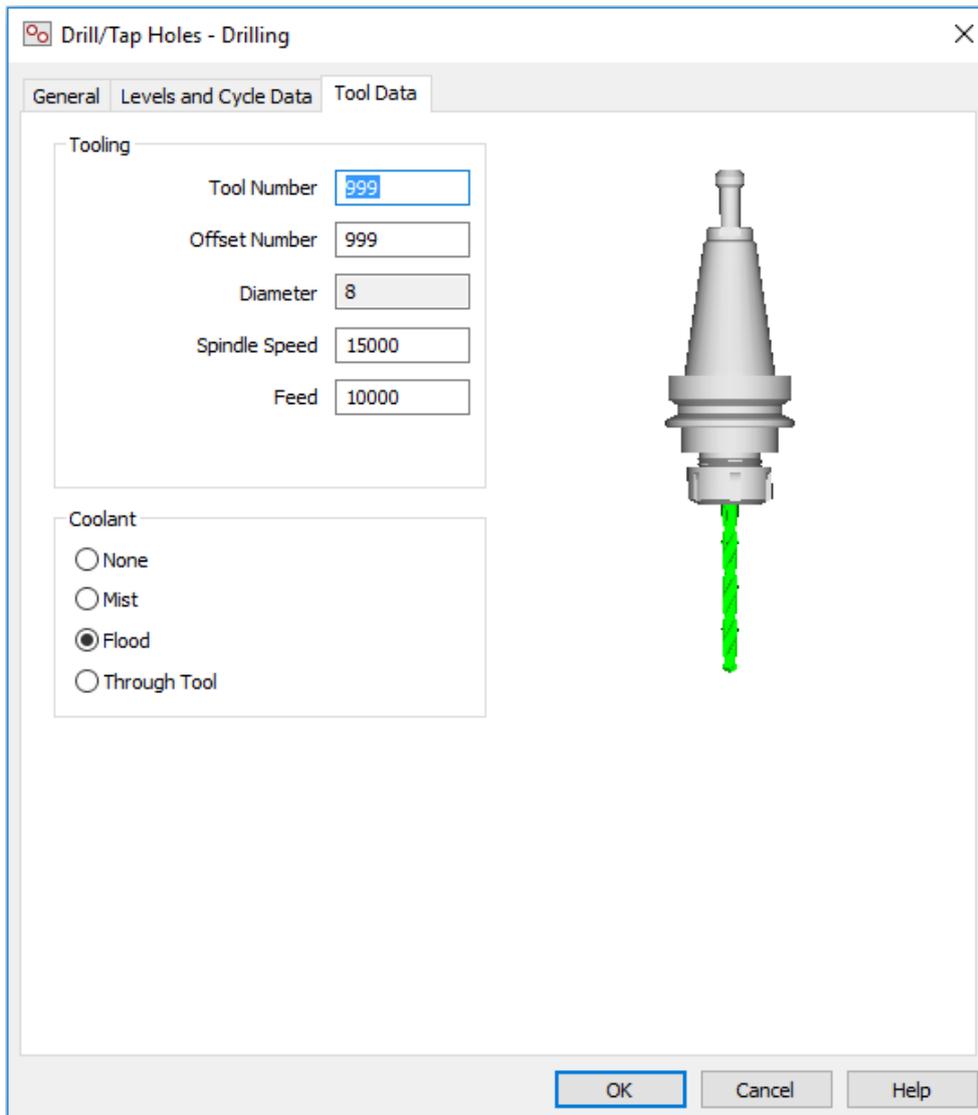


Figure 141 - Tool Data Drill/Tap Holes

Tool numbering and feed/speed options and any coolant options.

Selecting Geometries

Geometries that are circles can be selected by picking or windowing.

Geometries that are not circular can be selected but will be discarded when the selection is finished. Holes drawn as broken circles (e.g., tapped holes) will be accepted if each part is an arc with the same radius, and all the arcs have the same centre.

A cross is drawn at the centre of each hole showing the size of the drill. If any of the holes have diameter different from the drill diameter, they will be indicated, and you will be given a warning, E.G. when you are pilot drilling.

If you use the 3D Views option in the **VIEW** menu, the tool size cross is shown at the R (retract plane) level, and smaller crosses are shown at the bottom of the hole and at each peck level.

Chamfering cycle

The Chamfering cycle automatically calculates the required depth from a given circle diameter. This cycle supports drill type and flat type tools defined with taper angles. All selected holes, including Auto Z support, will be analysed to ensure the tool can enter the hole where there is a flat on the bottom of the tool, and that the tool diameter is sufficient to complete the full depth of chamfer.

If only the chamfer width is known i.e. the difference between the hole radius and the chamfer radius, then the chamfer depth can be calculated by entering the depth as:

$$- \tan(\vartheta) * Cw$$

Z Levels (or Distance from Plane)	
Safe Rapid Level	50
Rapid Down To (R-Plane)	20
Material Top	0
Chamfer Depth	$-\text{TAN}(30)*1$

Figure 142 - Chamfer Depth Calculation Fixed Z

- where

ϑ is the chamfer angle

Cw is the chamfer width

i.e. difference between the hole radius and the chamfer outer radius

Relative Z Levels	
Safe Rapid Distance	50
Feed Down Distance	20
Chamfer Depth	$-\text{TAN}(30)*1$

Figure 143 - Chamfer Depth Calculation Auto Z

If the outer diameter circle depicting the Chamfer is drawn, then the chamfer depth is set to 0 and the cycle will automatically calculate the depth required for the tool to match the geometry.

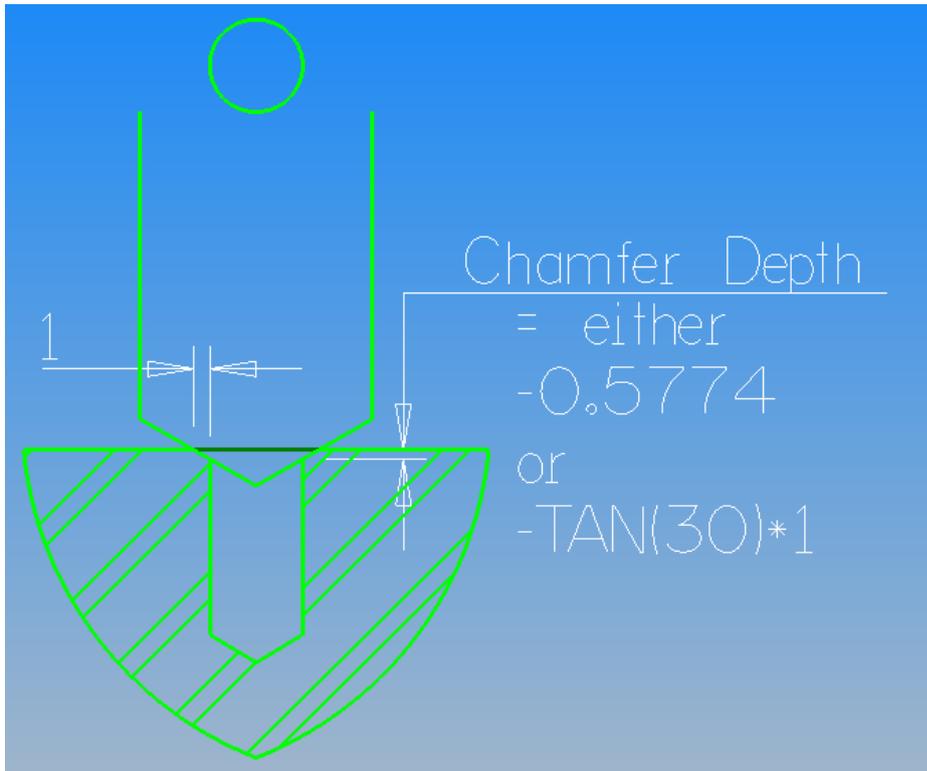


Figure 144 - Chamfer calculation for no chamfer outer diameter

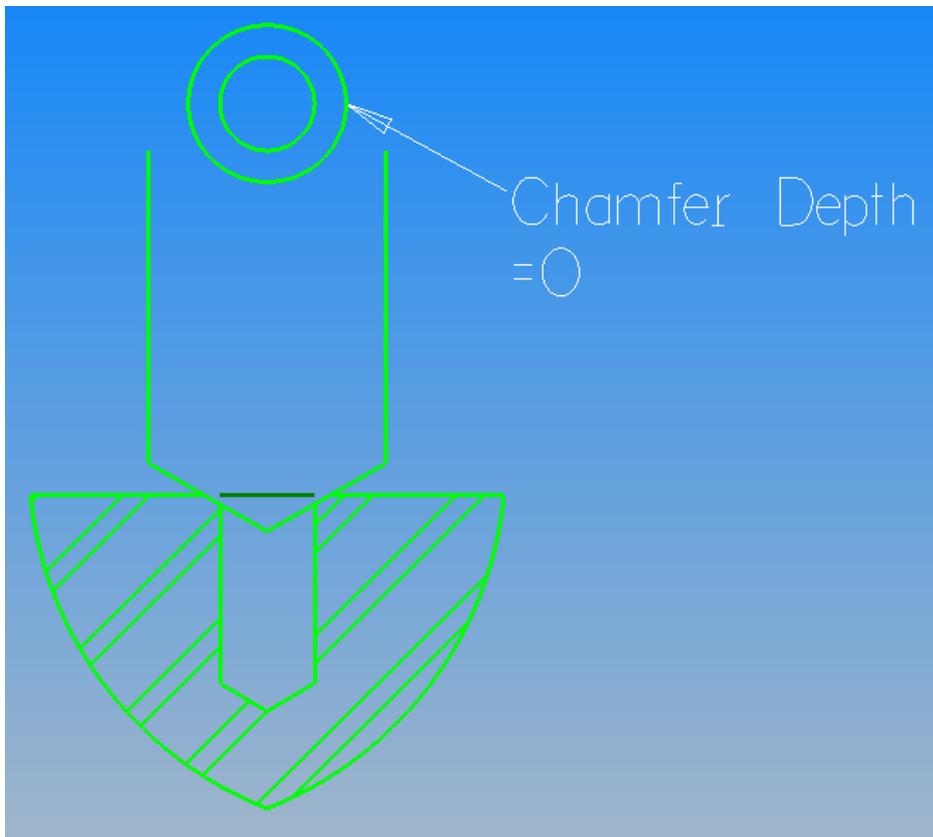


Figure 145 - Chamfer calculations when given the outer chamfer diameter

Machining Holes

This option allows you to use the same circle selection options as the **Drill/Tap Holes** process for circles that are to be machined using the Rough or Finish machining methodology.

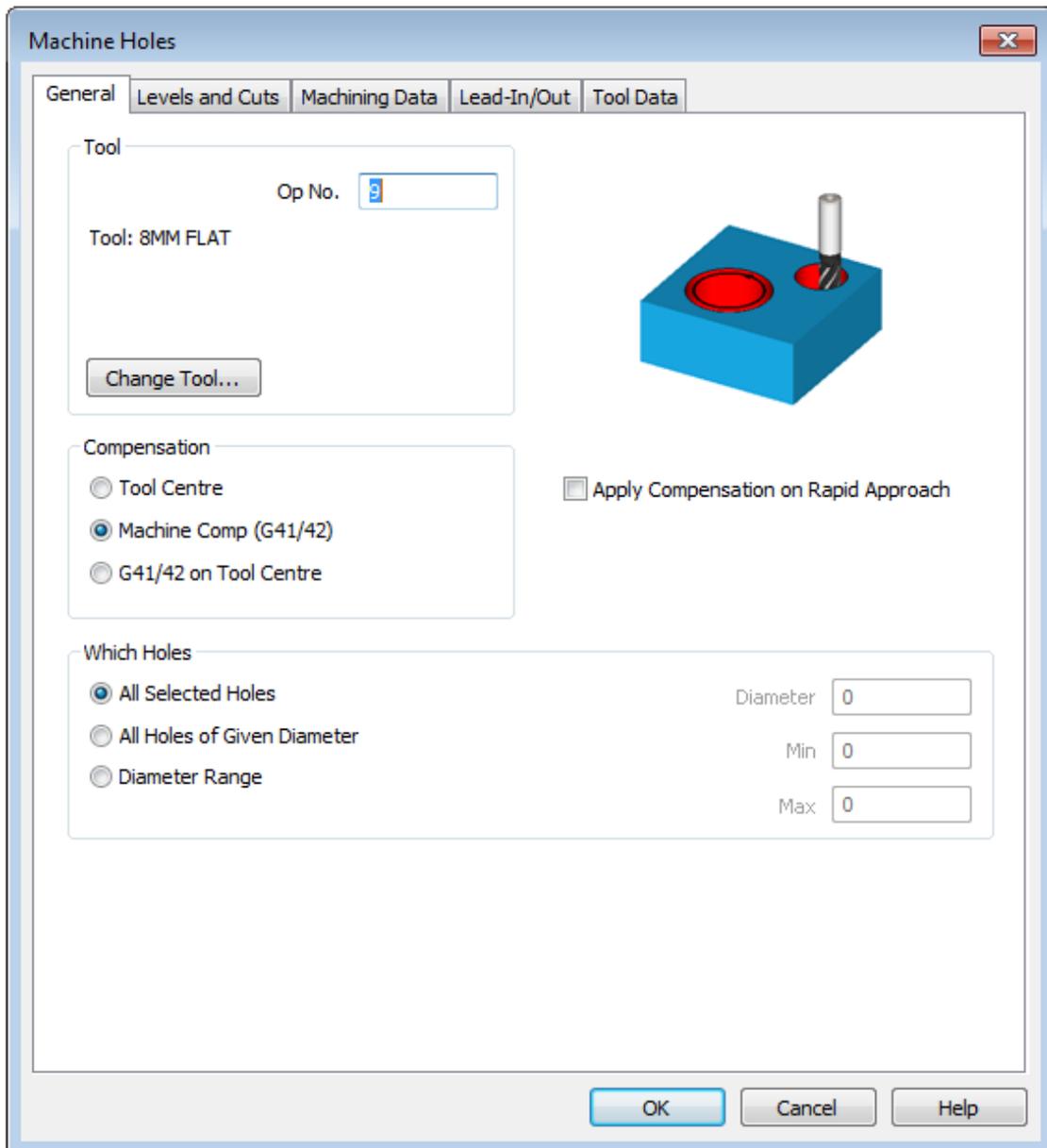


Figure 146 - General tab using Machine Holes

Only the General tab is different in the manner of using the **Which Holes** section to select the circles, the rest of the dialogues are the same as detailed earlier in this manual when dealing with the Rough and Finish cycle.

Pocket Holes

This option allows you to use the same circle selection options as the **Drill/Tap Holes** process for circles that are to be machined using the Pocketing machining methodology.

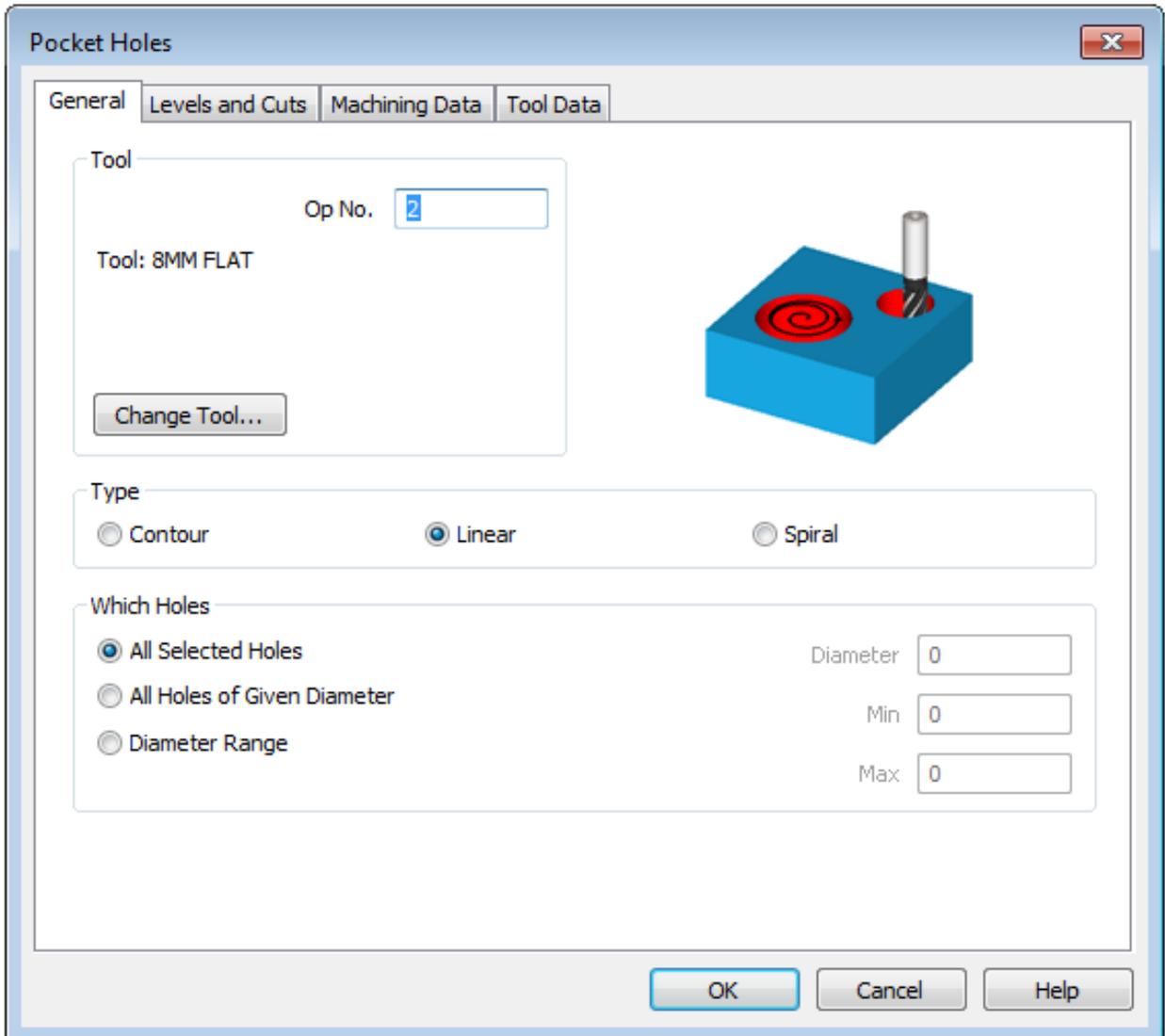


Figure 147 - General tab using Pocket Holes

Only the General tab is different in the manner of using the **Which Holes** section to select the circles, the rest of the dialogues are the same as detailed earlier in this manual when dealing with the Pocketing cycle.

Operation Manipulation

There are several options that can be used to modify machining operations.

To access the options **<RClick>** on the operation in the project manager page to display the popup menu.

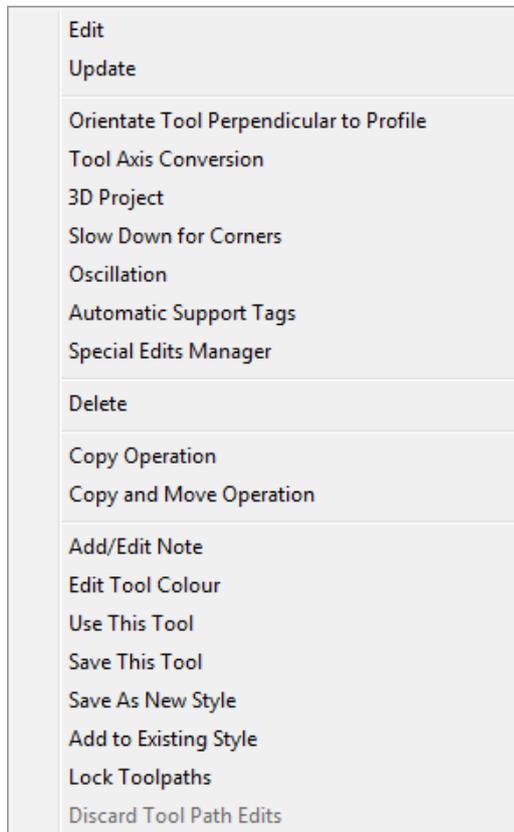


Figure 148 - **<RClick>** options for cycles in Mill/ Router/ Stone

Add and Remove geometries in machining strategies.

Adding or Re-selecting Geometries in a cycle

In Mill/Router/Stone modules, geometries can be added or removed from machining operations. To access this, select the '▷' against the operation then **<RClick>** on the word **Geometries** to display the popup menu options.

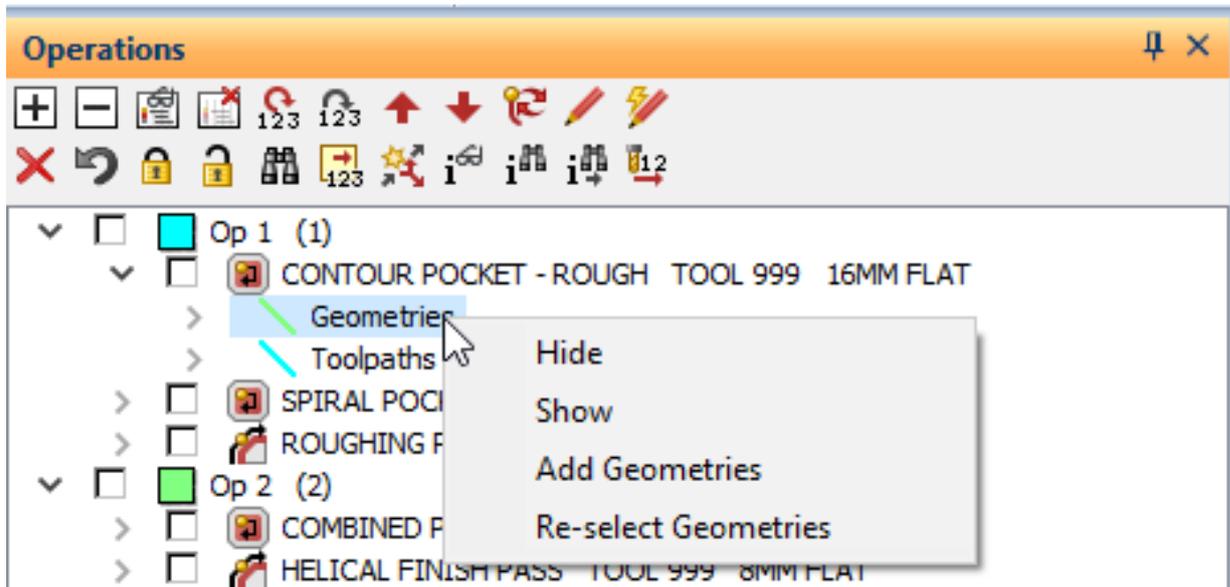


Figure 149 - accessing the ability to Add or Re-select Geometries on a cycle

This feature gives you the ability to **Add Geometries** to a machining strategy after it has been created or to **Re-select Geometries** if any errors have been made. When the geometries have been added, the strategy is automatically updated, and tool paths are added to the new geometries.

Removing Geometries from a cycle

Selecting the symbol against the word “Geometries” will show those geometries associated with the cycle and allow further manipulation to remove items chosen incorrectly, or separate specific parts to their own separate operation sequence.

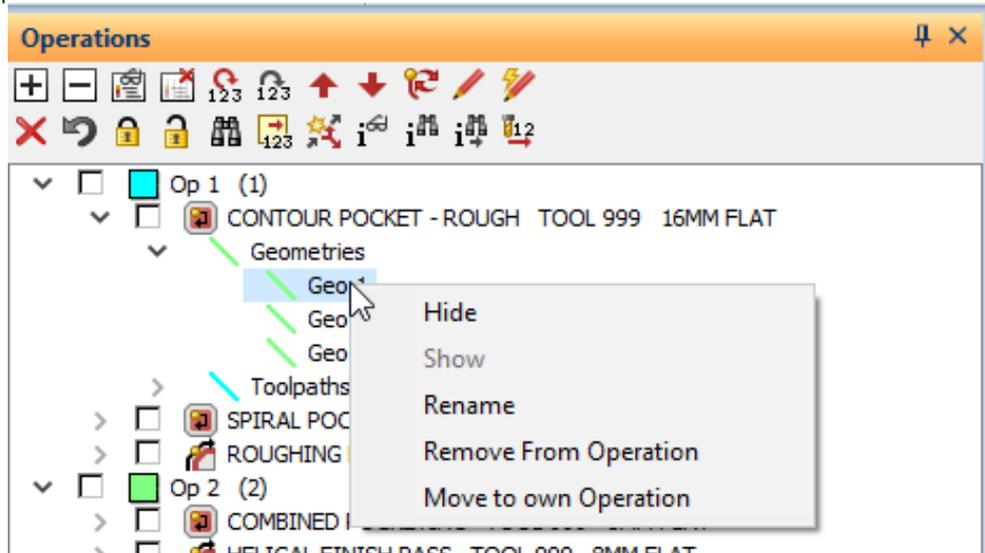


Figure 150 - <RClick> options to remove geometries from operations

Move to own operation

When tool paths are created together, and it is required to modify the tool paths applied to only one of the geometries, it is possible to split this geometry and subsequent geometries in the list out to their own operation.

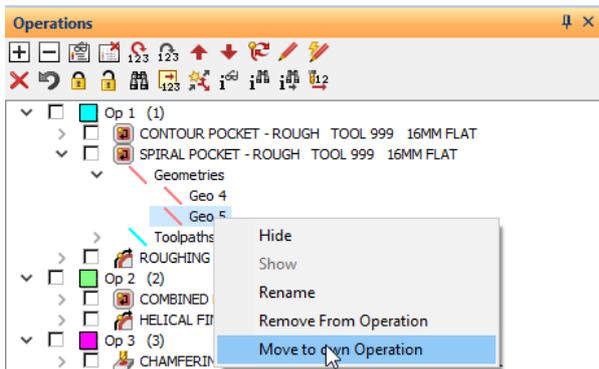


Figure 151 - Moving a geometry from a single cycle into two separate cycles

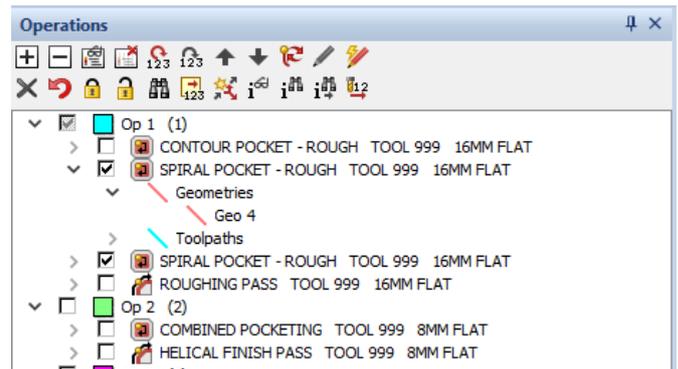


Figure 152 - New listing with two separate cycles

Setting and Using the Solid Simulation

When you have created your machining operations you will want to check the result by using the Solid Simulation, to use simulation, you must first define a material.



It must be noted that the **Material** and **Solid Simulation** options are not available in the **Essential** module of ALPHACAM.

There is a free macro that will create a material with an X & Y stock value.



Activate **ADD INS / Macros > Add ins...**

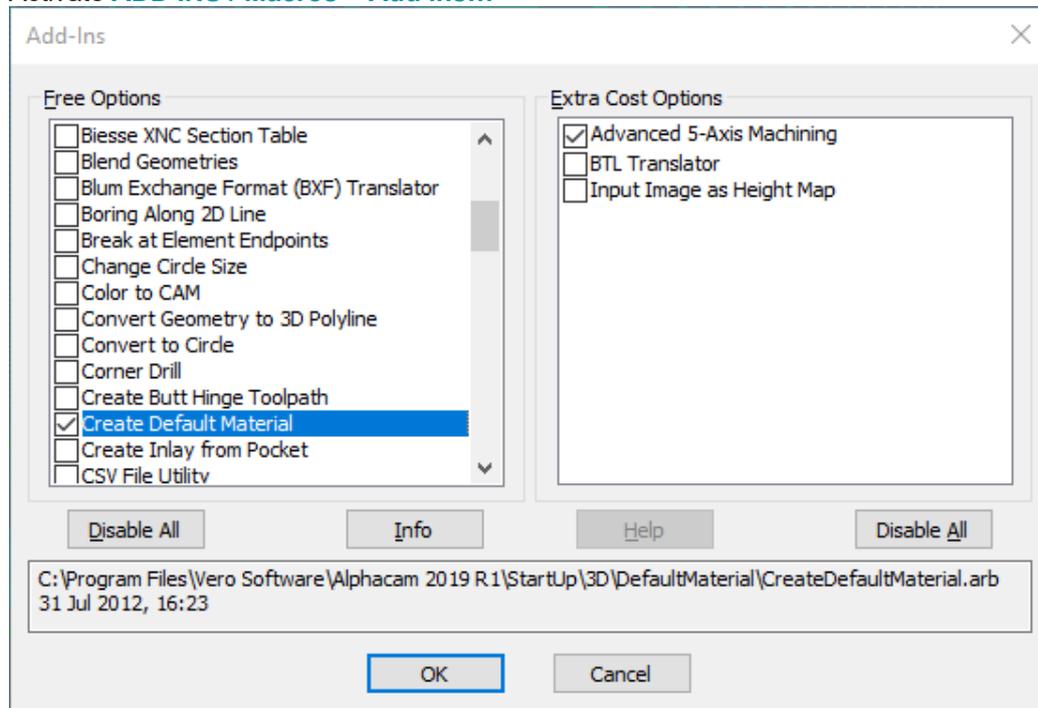


Figure 153 - Add-Ins Manager

Select the option **Create Default Material** from the dialogue list.

This will create the **3D > Auto Set Material**  command.

Setting up the Material

Using Set Material

If you have not previously setup a material when the simulation is activated a dialogue is displayed offering to setup a default material.

If you select **[OK]**, ALPHACAM will create a material where the size is taken from looking at the machining parameters and create a cuboid material that will encompass all machining operations; this will add extra material to the Length, Width, Height and Depth to also include the safety clearance options and the lowest Z value.

This may not provide a true graphical representation of the finished machining as the tools will not go through the material.

To define a material of any shape, follow the steps outlined below.

1. Create the geometry profile for the material.

2. Select **3D > Set Materials**



3. Select the edge of the default material or the geometry that you want to use for the material, and the following dialogue will appear.

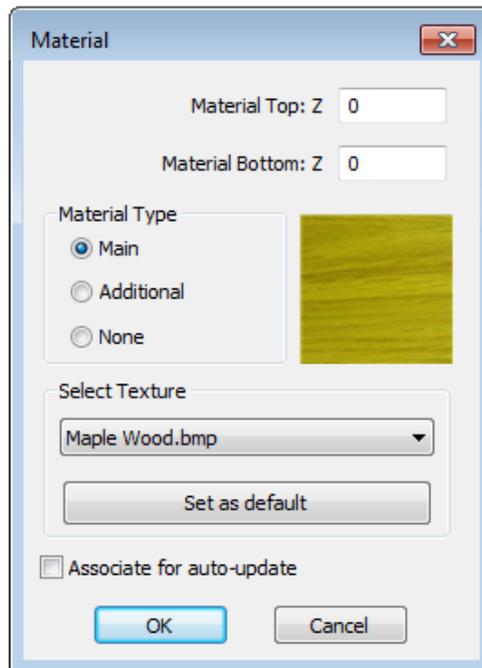


Figure 154 - Set Materials option dialogue

4. This gives you the option to set the top and bottom thickness of your material, and to select the material type.
5. There is also an option to allow the addition of an image file to represent the actual real world material in the simulation process to gain a more realistic impression of the finished part within ALPHACAM.
6. **Associate for auto-update** allows the material to alter its original described form to give a true reflection of the current job on subsequent operations.



It is preferable not to select the outer geometry of your shape as the material, because this will convert it from Geometry into Material and this will then have an adverse effect on your applied toolpaths.

To convert material back to construction, use the same procedure as for setting the material and set the material type to **None**, or to turn a material back to geometry use the

EDIT > Change  command setting the options **From** **Material** – **To** **Geometry** and select the material edge.

Using Auto Set Material

Using the Auto Set Material allows for the creation of a bounding rectangle around the entire part with allowance offsets applied in the X&Y directions.

Where Set Material can be used for any shape you can create, Auto Set Material can only set rectangular boundaries.

To define an **Auto Set Material** follow the steps outlined below.

1. Select **3D > Auto Set Materials**



2. The following dialogue will appear.

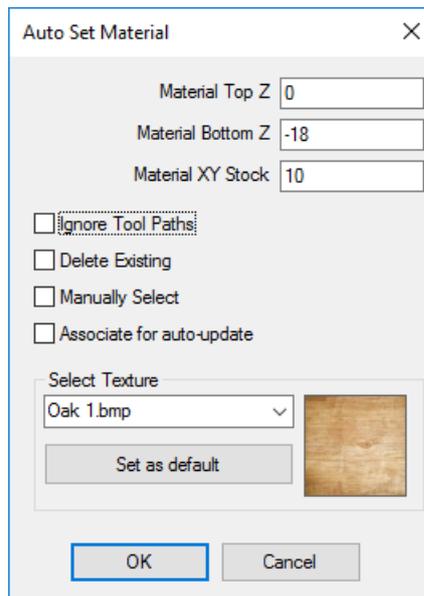


Figure 155 - Auto Set Material option dialogue

3. This gives you the option to set the top and bottom thickness of your material, and to set the excess material in X&Y.
4. **Ignore Tool Paths**, allows for the inclusion or not of any existing tool paths for the enclosing rectangle. Note that the material Top will include any Safe Rapid value in the cycles and the lower Z limit of an outside profile. The Material Stock will also be applied to around the outside of the outer profile tool.
5. **Delete Existing** allows for the updating of material stock without resorting to the Delete command.
6. **Manually Select** allows you to choose a geometric shape and apply the rectangular material around only the chosen item, rather than the entire part. For example, if you were performing a repair to an item and were only machining a section of a full part.
7. **Associate for auto-update** allows the material to be used on further operations and is updated to suit machining already applied, this will then remove the need to create new materials based on previous machining and will remove the unwanted machining of "Fresh Air" when creating complex parts and machining on multiple sides of a part.
8. There is also an option to allow the addition of an image file to represent the actual material in the simulation process to gain a more realistic impression of the finished part within ALPHACAM.

Simulation

To start the simulation, select the **VIEW > Simulation**  or **VIEW > Solid Simulation**  or select the Simulation Project Manager Page .

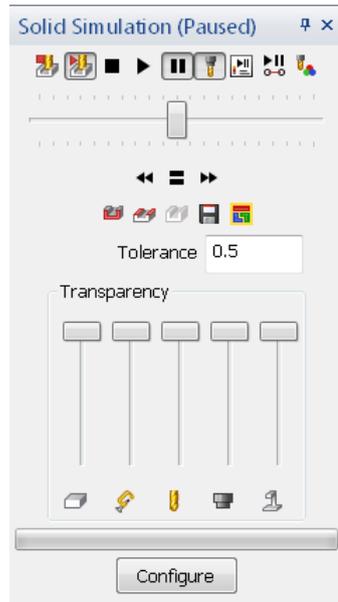


Figure 156 - Simulation controls from the Project Manager

	Close Simulation	Closes the simulation process and returns to the drawing/programming environment.
	Shade/Solid Simulation	Displays the simulation in Wire Frame if the option is not active, and a shaded solid simulation if active.
	Stop Simulation	Stops the simulation and leaves the simulator active.
	Run Simulation	Runs the simulation to view the programmed operations.
	Pause Simulation	Pauses the simulation.
	Show Tools	Toggles the visibility of the tooling during the simulation.
	Operation by Operation	The simulation automatically stops at the end of each programmed operation group.
	Single Step	Steps through the simulation one line at a time, the Run button must be used in conjunction with the Single Step option.
	Show Tool colours	Makes the applied colour of the tool take precedence over the default simulation colour. The default setting is for this to be active.

Simulation speed



Figure 157 - Simulation speed slider controls

Using the slider option or the Slow Down, Speed Up controls underneath affects the speed at which the simulation runs.



The quality of your pc and its graphics card will have a direct bearing on the speed and visual appearance of the results in the simulation.

Miscellaneous options



Vertical Section

Allows a cross section of the part vertically from Flat-Land.



Horizontal Section

Allows a cross section across the part parallel to Flat-Land.



Remove Sectioning

If any sections have been created, this option will remove them.



Save Stock

Saves the resulting simulation file as an STL which can be imported for use as a reference material for subsequent operations.



Split into Separated Parts

Identifies individual parts via differing colours, for example, multiple parts on a nested sheet layout

Tolerance

Tolerance

Figure 158 - Simulation Tolerance

The tolerance option allows the user to define how good the resulting simulation will appear by adjusting the value in the **Tolerance box**.

A large number will display a rougher appearance than one with a very small value.

It should also be noted that the smaller the value, the longer the simulation will take to process and, in some cases on computers that are not adequately equipped, may fail to run correctly.

Transparency

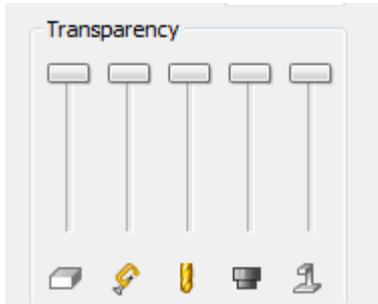


Figure 159 - Simulation transparency sliders

These sliders all perform the same function as each other but are applied to the following options if available in the ALPHACAM module in use. The higher the slider the more opaque an item will be in ALPHACAM.



Material transparency slider



Clamping and Fixture transparency slider



Tooling transparency slider



Tool Holder transparency slider



Machine transparency slider

Configure

The Configure button allows the user to set up the various colour options and precision of the simulation process in the same menu options as accessing **HOME > Configure > General** and using the **Simulation** options.

Solid Simulator

There is an additional simulator which is run in a separate window. This means it does not have to be closed to be able to return to the normal graphics screen to add or modify tool paths.

Send To Simulator



This sends the current ALPHACAM data to the Solid Simulator.

Send to Rapid Result



This sends the current ALPHACAM data to the Solid Simulator and displays the tool paths process without displaying any tooling or machines. (This is only applicable to 3D ball end machining).

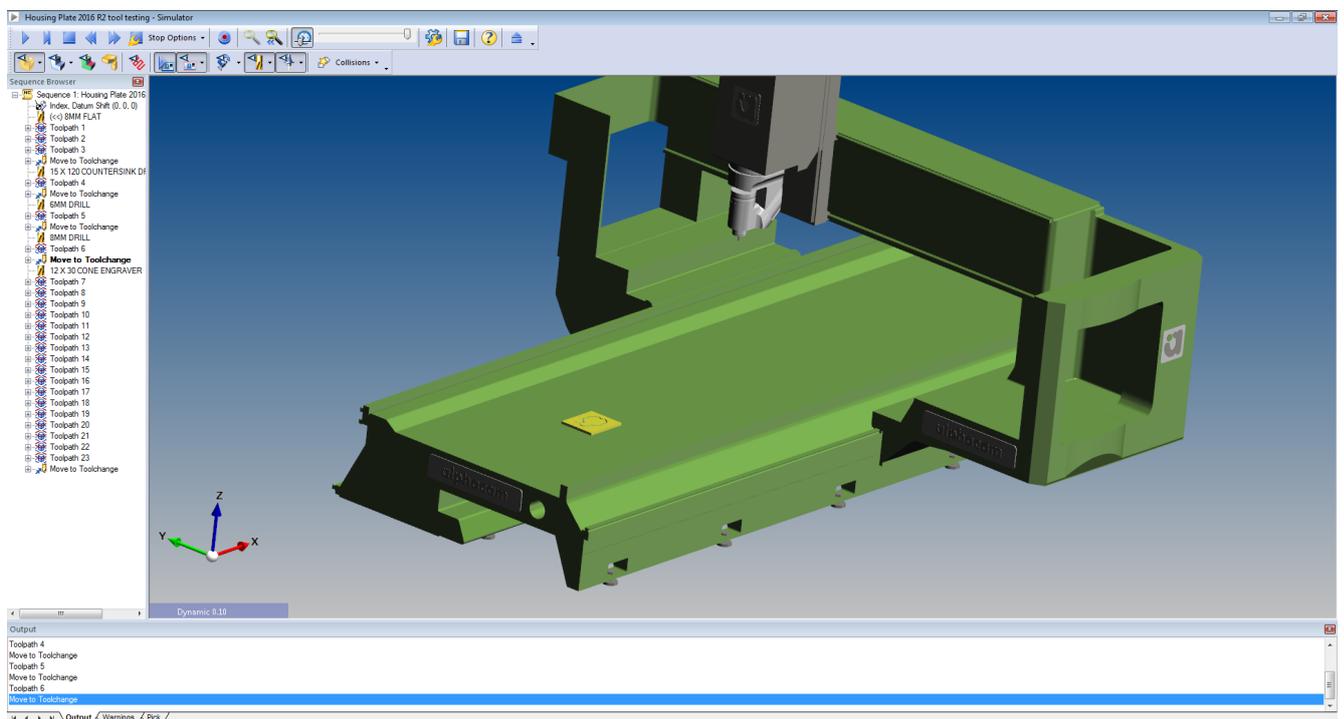


Figure 160 - External simulator window

As well as similar selection options for the visibility of material, tooling etc. this simulator shows where in the toolpaths any collisions are using the scrolling left hand pane on the above image. This can be extremely useful in more complex machining to identify problem areas within a complex toolpath.

Sawing

To use the sawing option, you must select a saw tool.

When creating a Saw tool, it is beneficial to add a simple tool holder to denote the motor side should the machine have 4 or 5 axis movement abilities, so that you can see if the tilt/twist angles are in the correct attitude for the machine head.

When you select **MACHINE > Saw**  the first dialogue is displayed.

Types

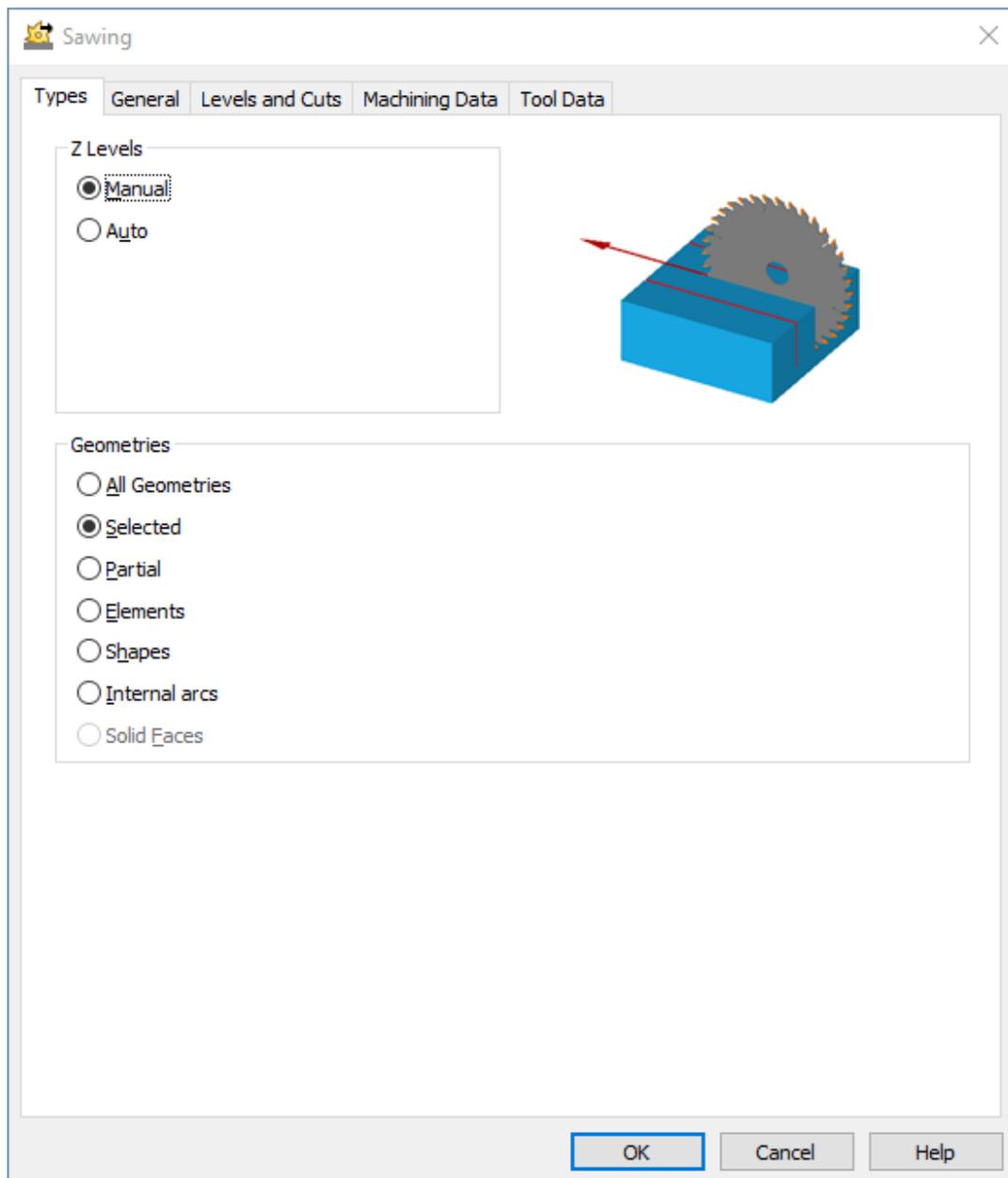


Figure 161 - Types tab on Sawing cycle

Z Levels

- Manual** The user enters the values in the dialogue.
- Automatic** Takes information directly from the geometries in the same manner as Auto Z.

Geometries

- All Geometries** Select all visible items.
- Selected** Use only the chosen items selected by the user.
- Partial** Use only the selected sections of a profile (points must be selected with respect to the cut direction).
- Elements** Only selected single sections of a complete profile.
- Shapes** An area clearance style saw cut pattern to be added by placing a range of overlapping saw cuts within the chosen geometry area.
- Internal Arcs** Allows the use of a 5 Axis sawing head to be used to create curved sections on an internal radius. Note that this option is to be used within the confines of your machine axis movement and the size of the saw being used.
- Solid Faces** Allows for the direct selection of planar faces from a solid model without the need to use the Feature Extraction command, **Edges for Sawing**.

When selecting multiple elements (a profile) the saw automatically retracts to the **Safe Rapid** height to orientate to the new angle.

General

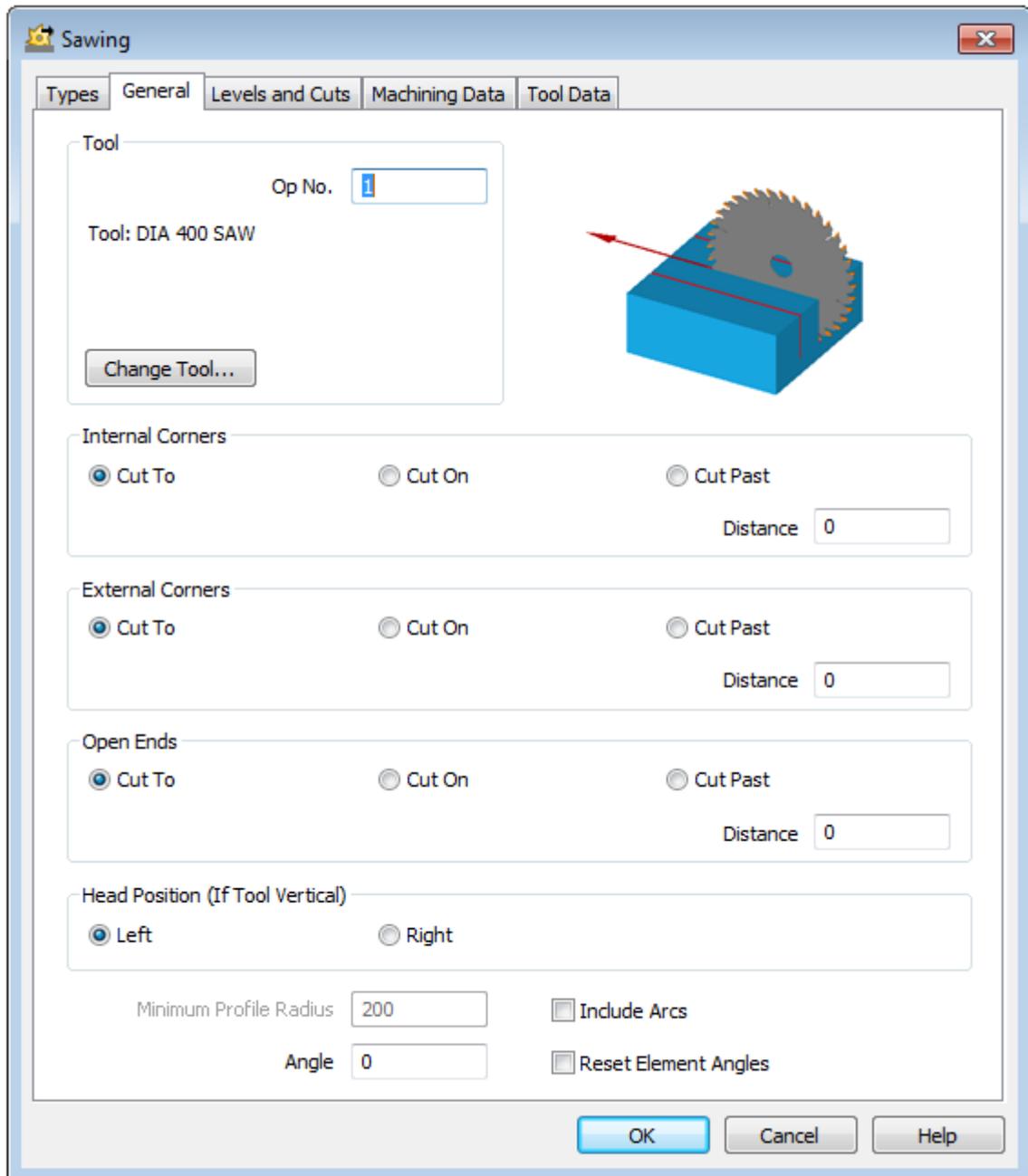


Figure 162 - General tab on Sawing

This dialogue allows you to set how the tool path is created with respect to **Internal, External corners** and at **Open ended** geometry/elements. You can also select the head side of the saw with respect to the geometry and any tilt angle.

Cut To

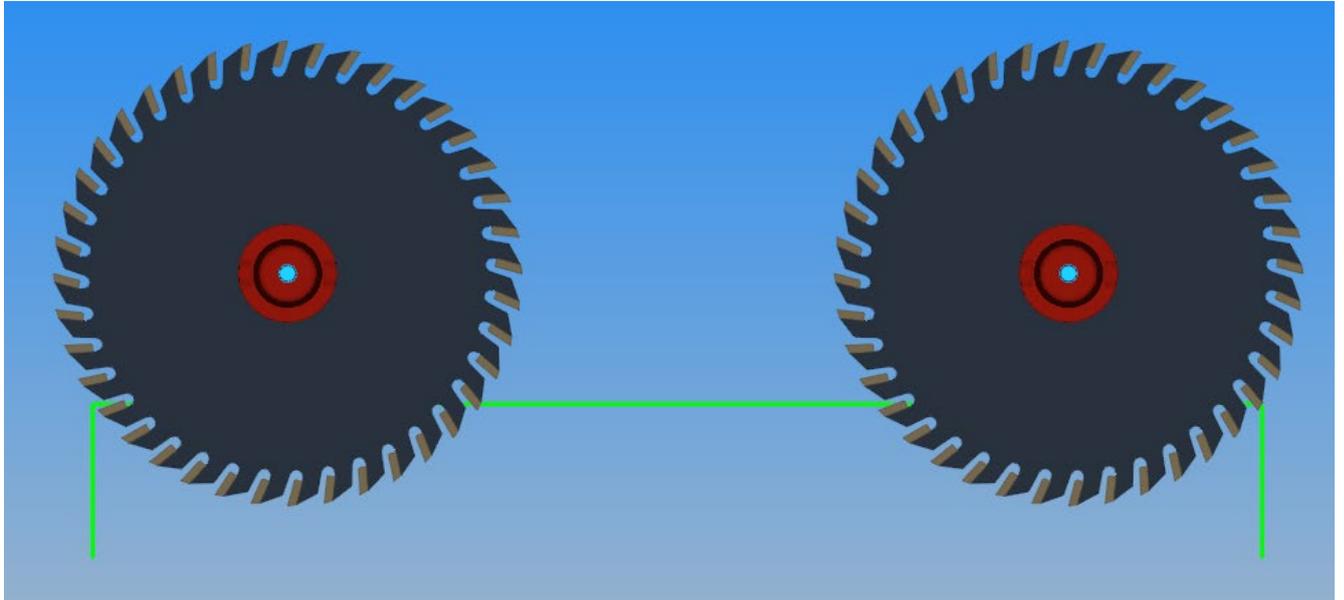


Figure 163 - Explanation of Cut To

Using the **⊙ Cut To option**, the Saw will cut the required geometry or Solid Face from a point where the peripheral edge of the saw is tangent to the chosen item, to the same tangent point at the opposite end of the cut.

Cut On

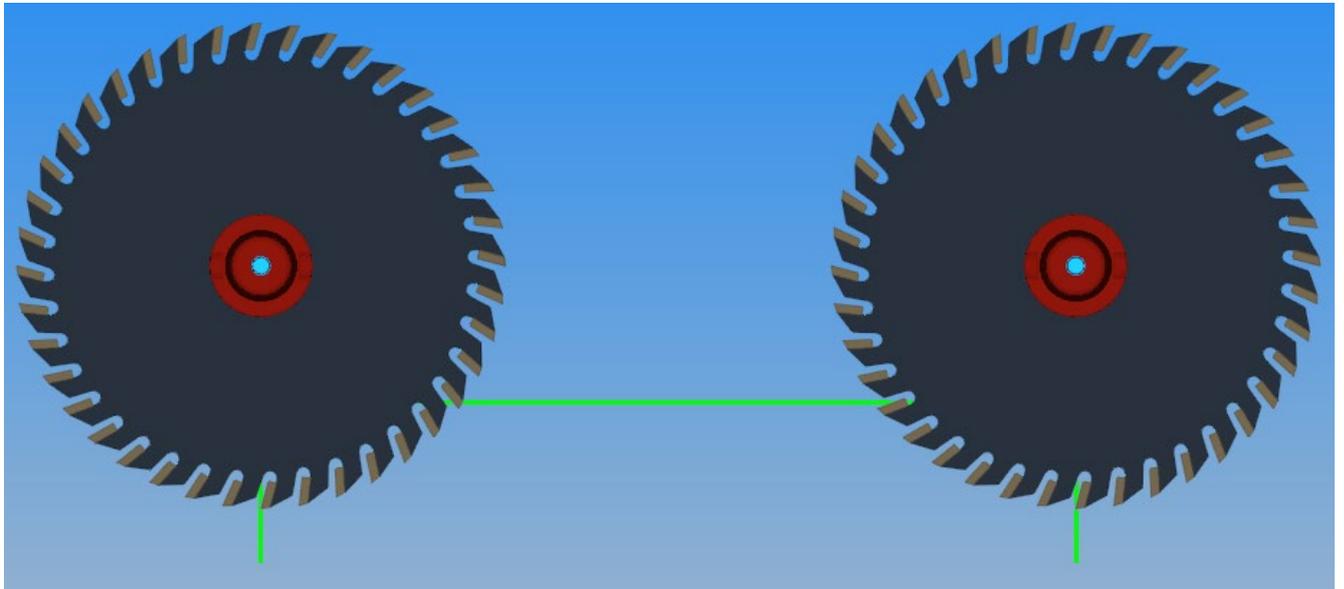


Figure 164 - Explanation of Cut On

Using the **⊙ Cut On option**, the Saw will cut the required geometry or Solid Face from a point where the centre line of the saw is in line with the chosen item, to the same point at the opposite end of the cut.

Cut Past

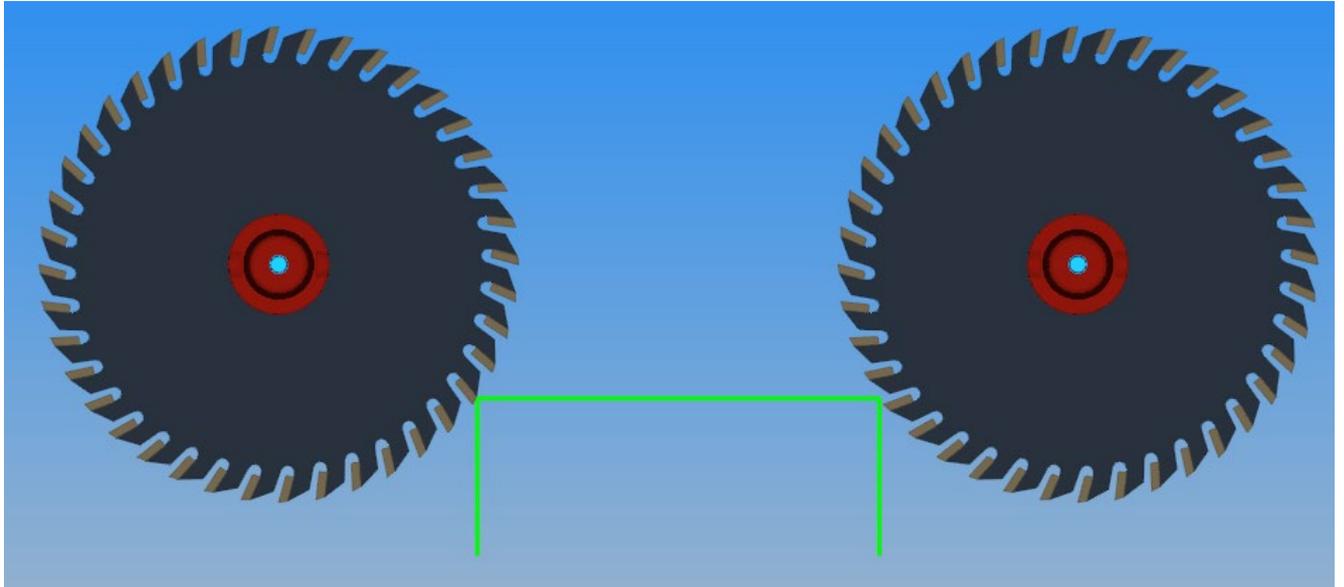


Figure 165 - Explanation of Cut Past

Using the **⊙ Cut Past option**, the Saw will cut the required geometry or Solid Face from a point where the peripheral edge of the saw is tangent to the chosen item on the outer edge of the disc, to the same tangent point at the opposite end of the cut.

When using the **Cut Past** option, you also have the option to enter a value for how far past the chosen option the saw blade will commence and complete the cut.

Levels and Cuts

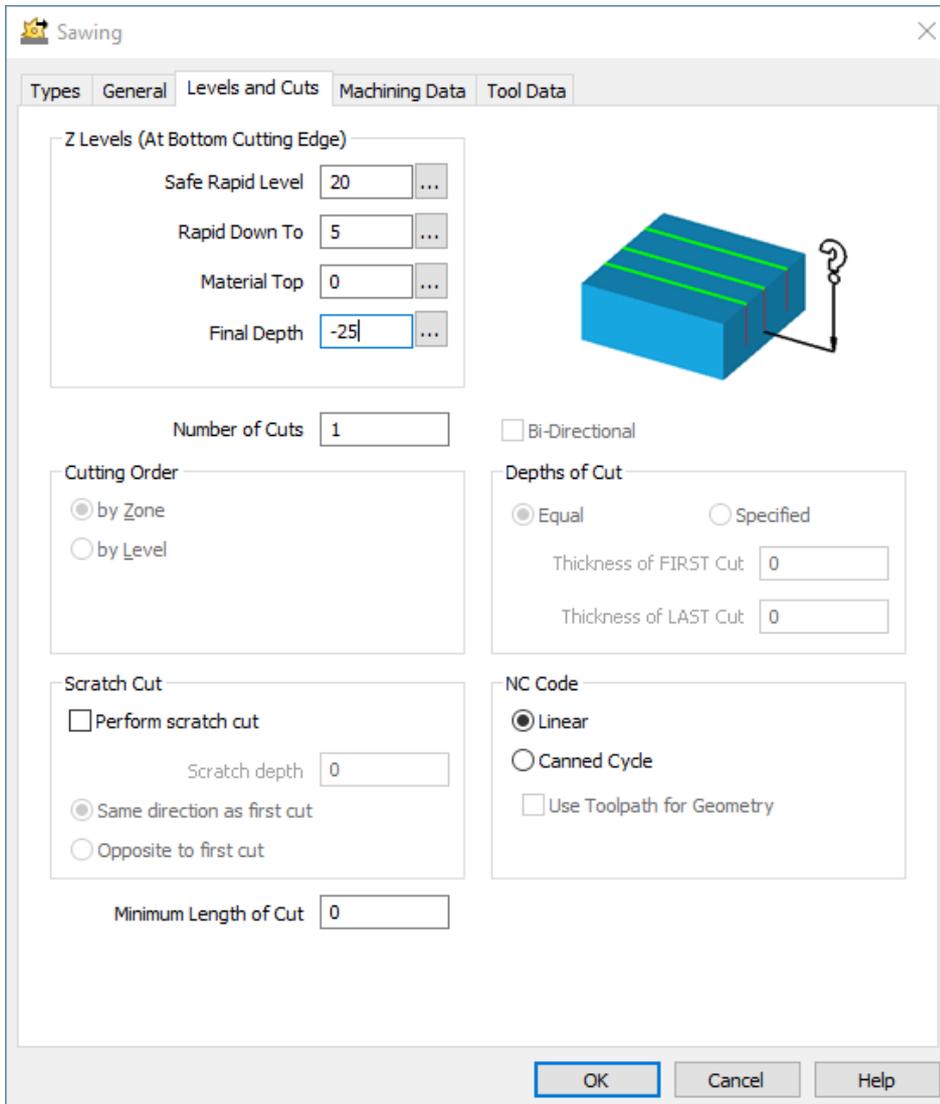


Figure 166 - Levels and Cuts tab on Sawing

Most this dialogue is the same as those used in the Pocket cycle and Rough and Finish with respect to the Z options and the style of cuts.

Scratch Cut

Activating the **Perform scratch cut** option allows the user to input a depth of cut which is independent of the main set up for depths and number of cuts. This could be used as an example, on veneered material to carefully mark through the veneer prior to cutting the actual material underneath.

Minimum Length of Cut

This option will allow a value to be entered and any cuts that are shorter than this will be discarded when the cycle is generated.

Machining Data

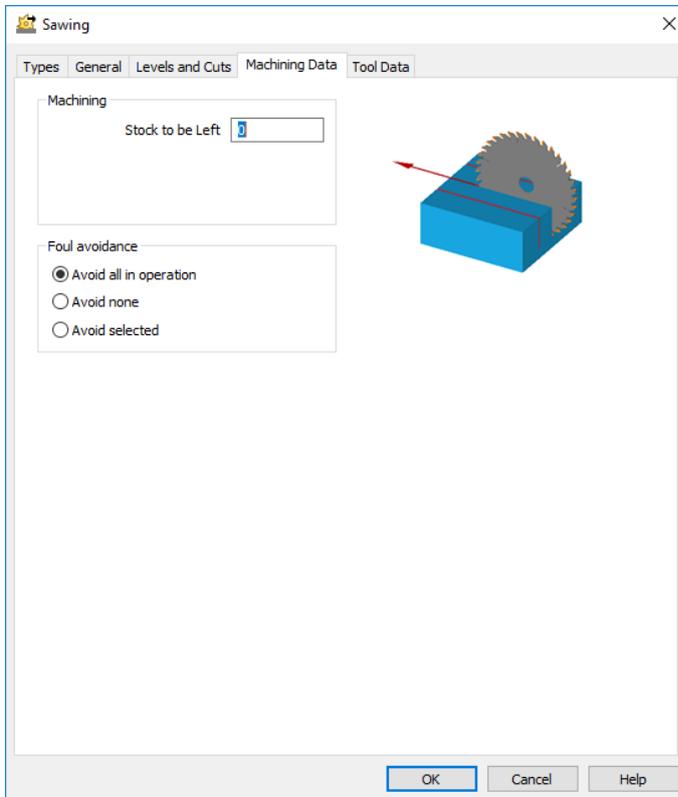


Figure 167 - Machining Data tab on Sawing

Allows for side wall options in the same manner as a Pocketing or Rough & Finish cycle does, as explained earlier.

Foul Avoidance

- Avoid in All operations** Automatically avoids applying cuts that would foul other geometries cut by the same operation. Foul avoidance cannot be used in a canned cycle.
- Avoid none** Does not perform any avoidance.
- Avoid Selected** Avoids any user selected geometries or Solid Faces within the same cycle.

Tool Data

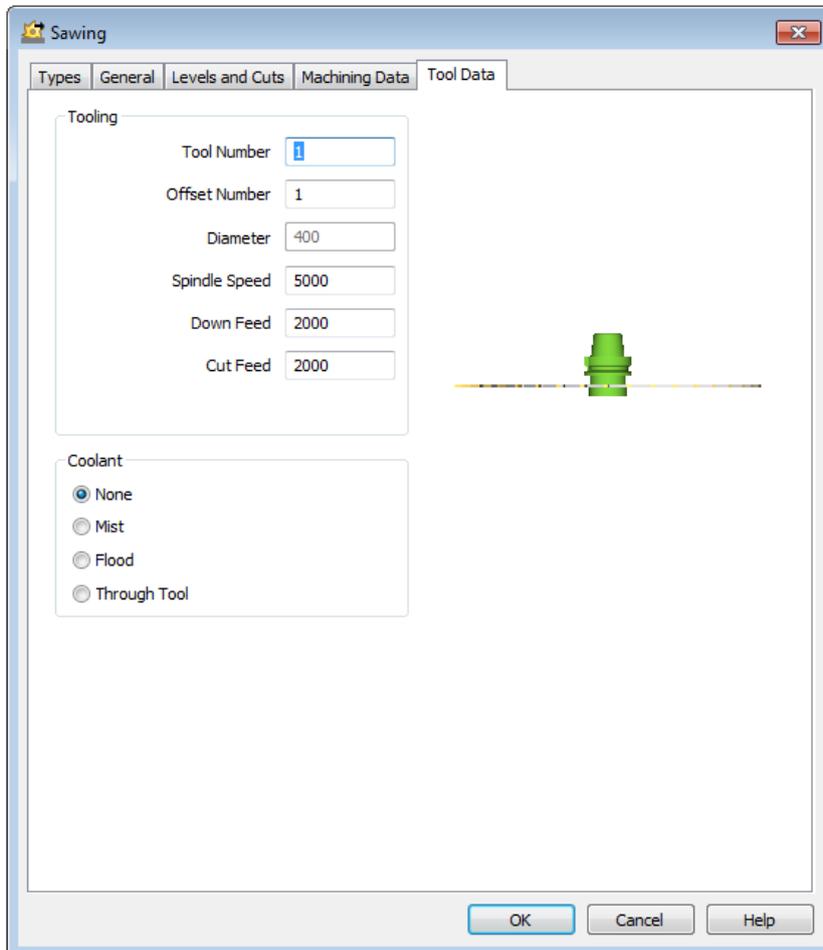


Figure 168 - Tool Data tab on Sawing

Options for the Speed and Feeds.

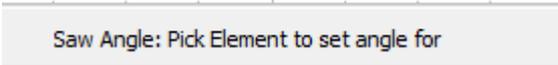
You are advised to apply a **Tool Holder** when creating saw type tools so that you can visualise which side of the blade the head unit will be working if you are performing multi axis sawing operations.

This will then allow you to set the option **Left** or **Right** on the **General** page to alter the head unit's location should the saw be placed incorrectly during the cycle creation stage.

Set Element Saw Angle

The Saw angle can be applied on the cut dialogue however this applies to all elements selected. If different elements require to be cut with different angles, then the **Set Element Saw Angle** should be used.

Select **MACHINE > Set Element Saw Angle**  the system will prompt you for the required element to alter.



Saw Angle: Pick Element to set angle for

Figure 169 - Set Element Saw Angle prompt

The Set Angle Dialogue is displayed allowing you to enter the angle required.

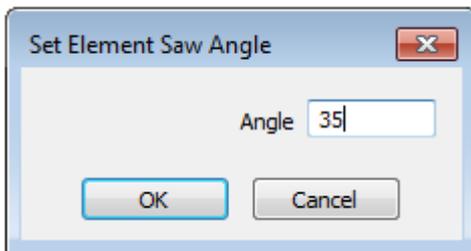


Figure 170 - Angle entry dialogue

 The Saw is tipped Counter Clockwise if this is the wrong direction Set the Element angle to a negative value and apply **MACHINE > Update Tool Paths**  the toolpath will be updated tip angle will be clockwise.

 Note the use of the sawing option may require a post processor update/upgrade.

3D Cut/Engrave

3D engraving is profiling procedure, like Rough or Finish, that creates tool paths that lift out at internal corners. The lift out action creates a 3D tool path.

3D engraving will work with any form tool however it is only practical to use continuous concaved forms. 3D engraving can be used on both Engraved (carved in lettering) and Embossed (raised lettering) geometries. When engraving Engraved (carved in lettering) forms such as Text, the process will consider both sides of the tool and restrict the depth of cut where the tool would foul on the back side.

General

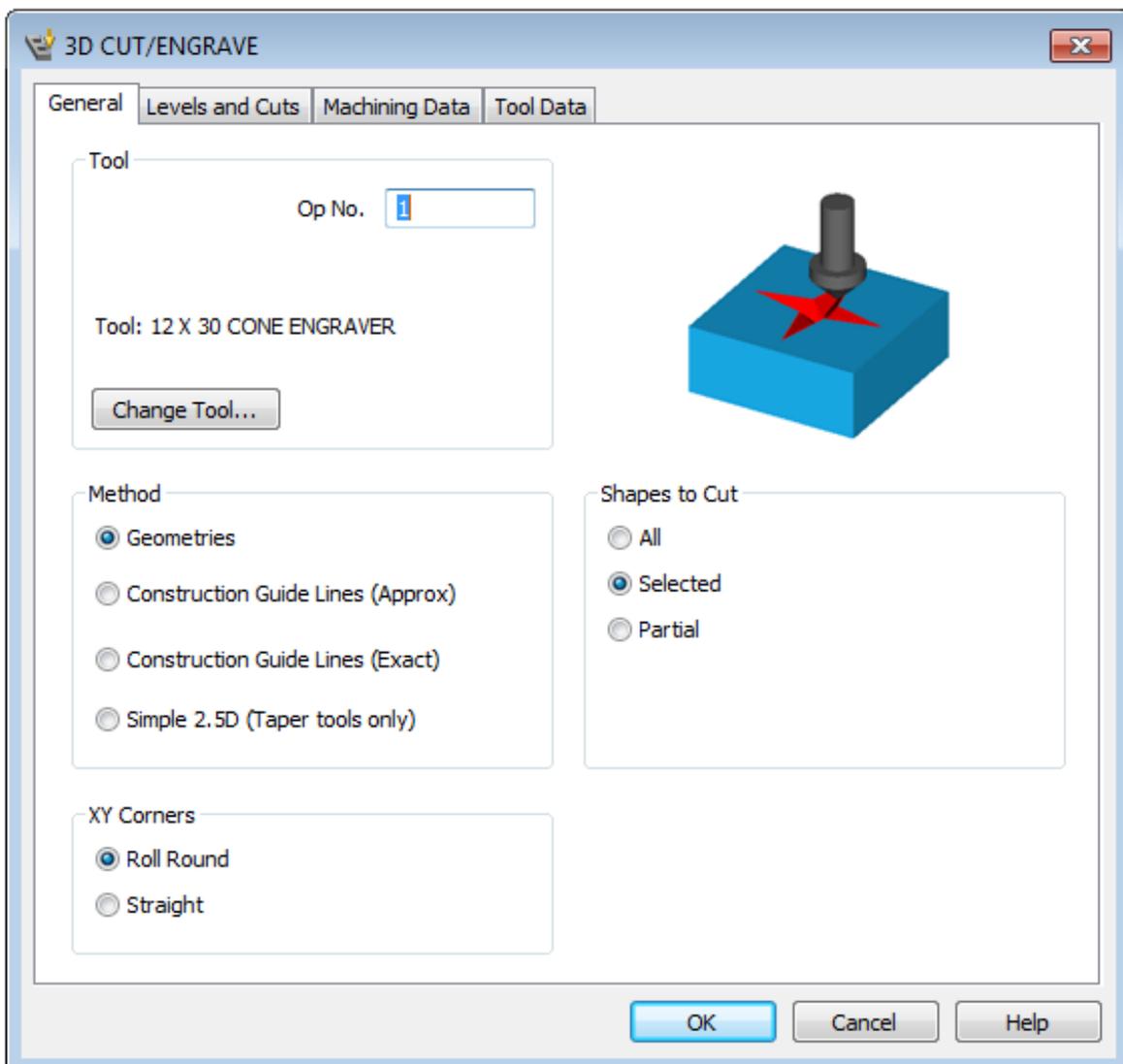


Figure 171 - General tab 3D Cut/Engrave

The options for items to cut and the XY Corners are the same as those used in the Rough and Finish cycle.

Method

- Geometries
 Ensures that the tool follows the selected Geometries exactly as drawn.

- Construction Guide Lines (Approx)
 Allows the use of Construction geometry to guide the tool in an approximation of the drawn shape. Useful for items such as Waveboards where uniformity is not an issue.

- Construction Guide Lines (Exact)
 Same as above but follows the Construction geometry exactly, or as close as possible, as drawn.

- Simple 2.5D (Taper tools only)
 Uses the same method as Geometries but gives true ramping in the corners but within specific parameters as described at the end of this section.

Levels and Cuts

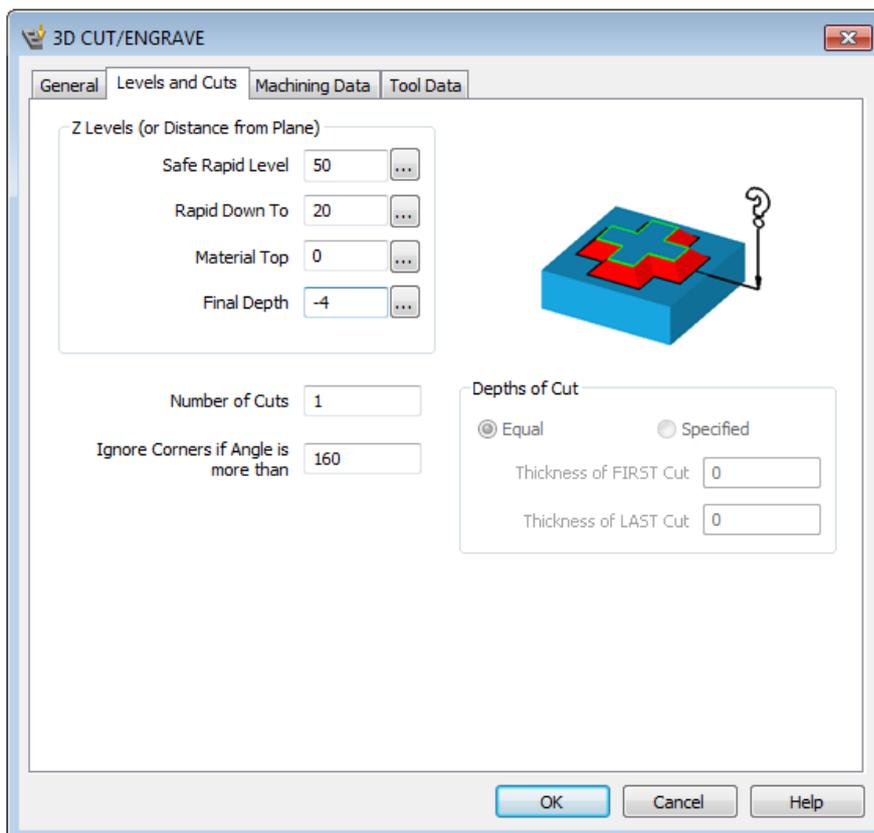


Figure 172 - Levels and Cuts tab

All information is the same as previously described cycles with the addition of **Ignore Corners if Angle is more than**.

This option removes the ramping from all corners larger than the stated angle where there would be no requirement for this move.

Machining Data

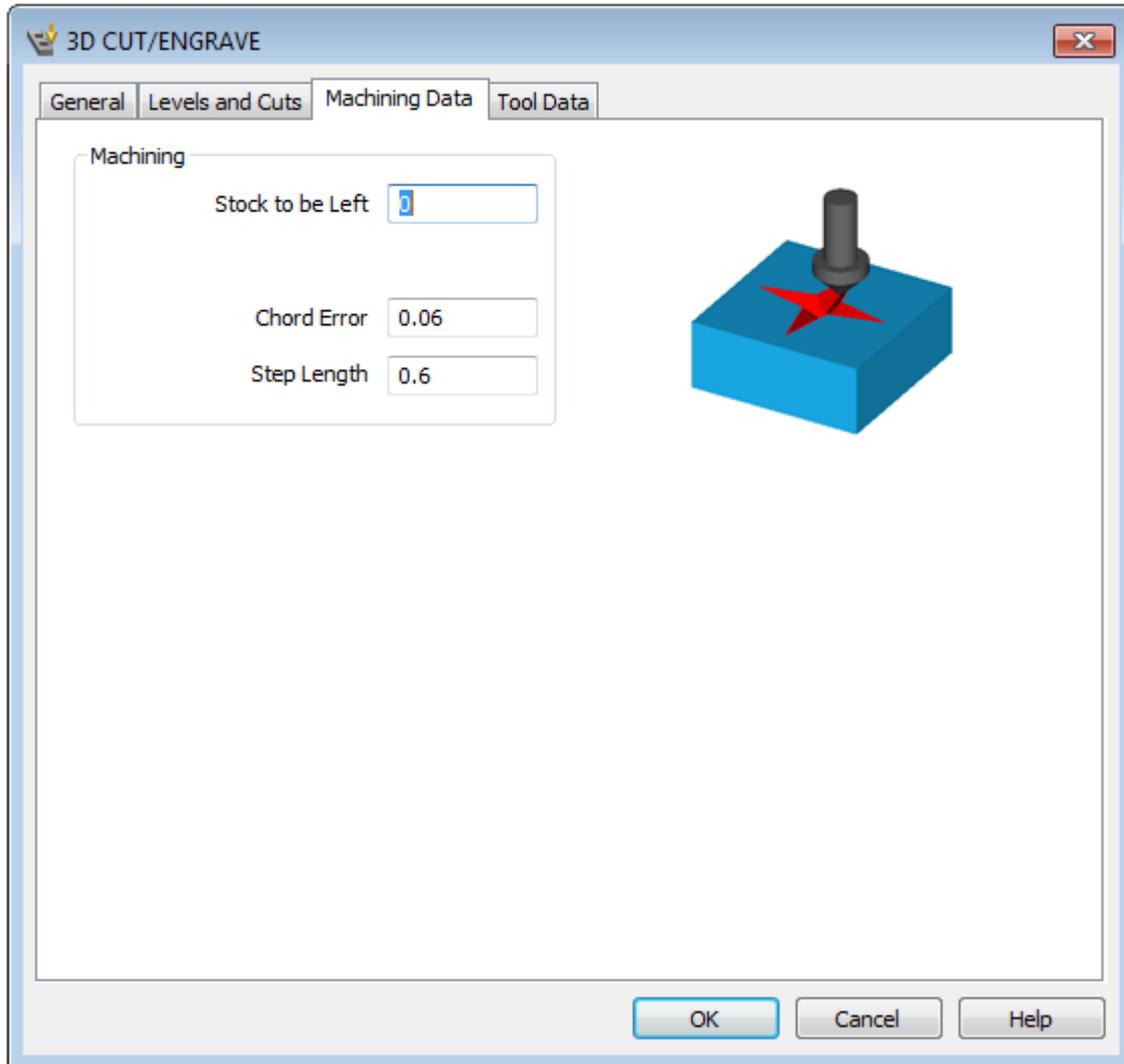


Figure 173 - Machining Data tab

Stock to be Left allows for a roughing pass using the cycle.

Chord Error Alters the precision which is used to follow the actual geometry selected to machine.

Step Length Adjusts how large the machining moves around arcs are.



All engraving toolpaths are generated from small XYZ moves, making the **Chord Error** and **Step Length** values smaller will give better accuracy for following complex shapes.

Tool Data

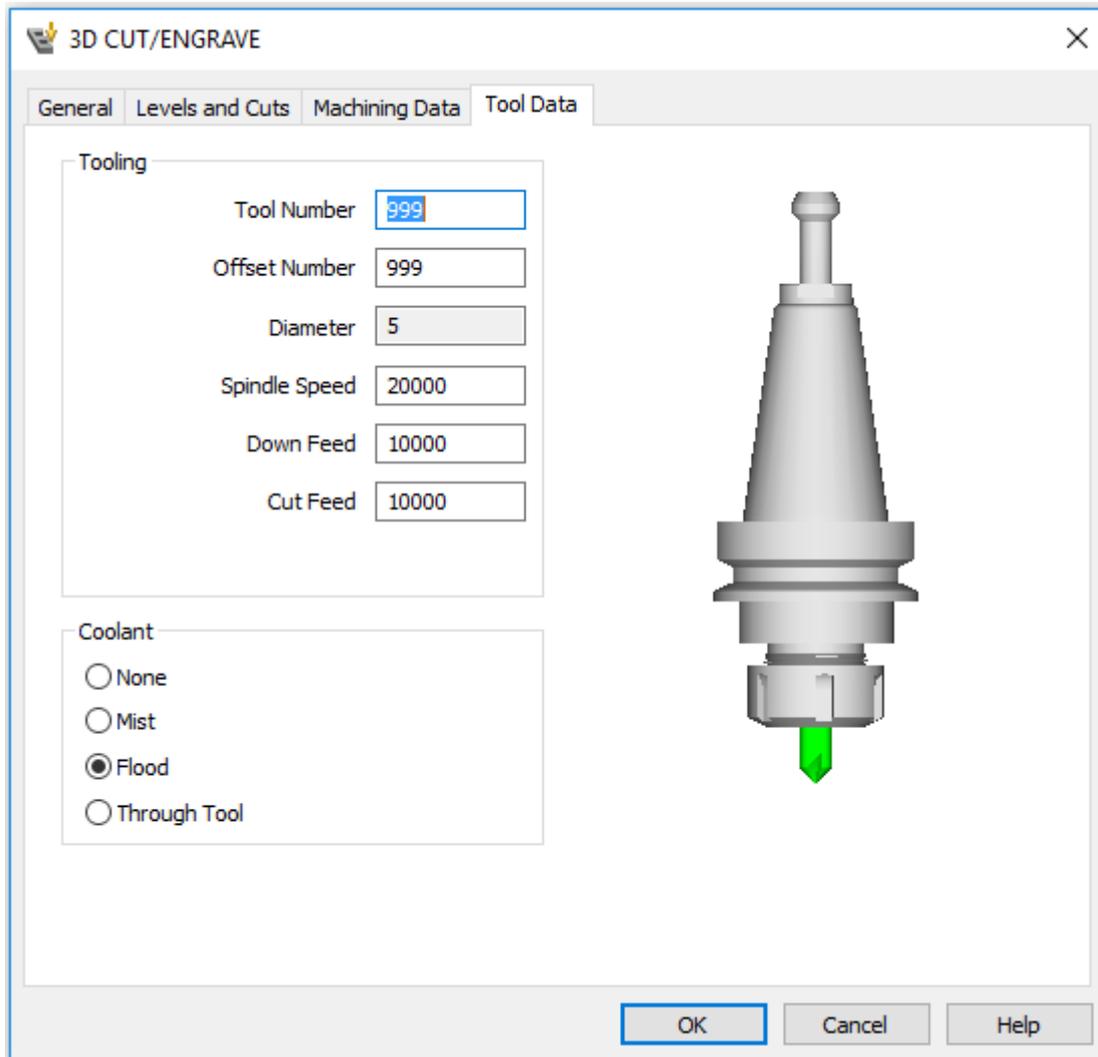


Figure 174 - Tool Data tab

Options for Speed and Feed.

Simple 2.5D Engraving

2.5D engraving is an addition to the 3D engraving option which provides direct tool path ramping at qualifying corners.



Simple 2.5D Engraving can only be used with tools defined as Flat taper tools.

The rule to determine qualifying corners is simple.

The tool follows around the profile at the stated depth and when a corner is reached, stops.

The tool contacts on the profile at two points.

The profile is extended tangentially from these points to intersect each other and, if the intersecting point is outside the actual geometry profile then the ramp would foul the part and is discarded.

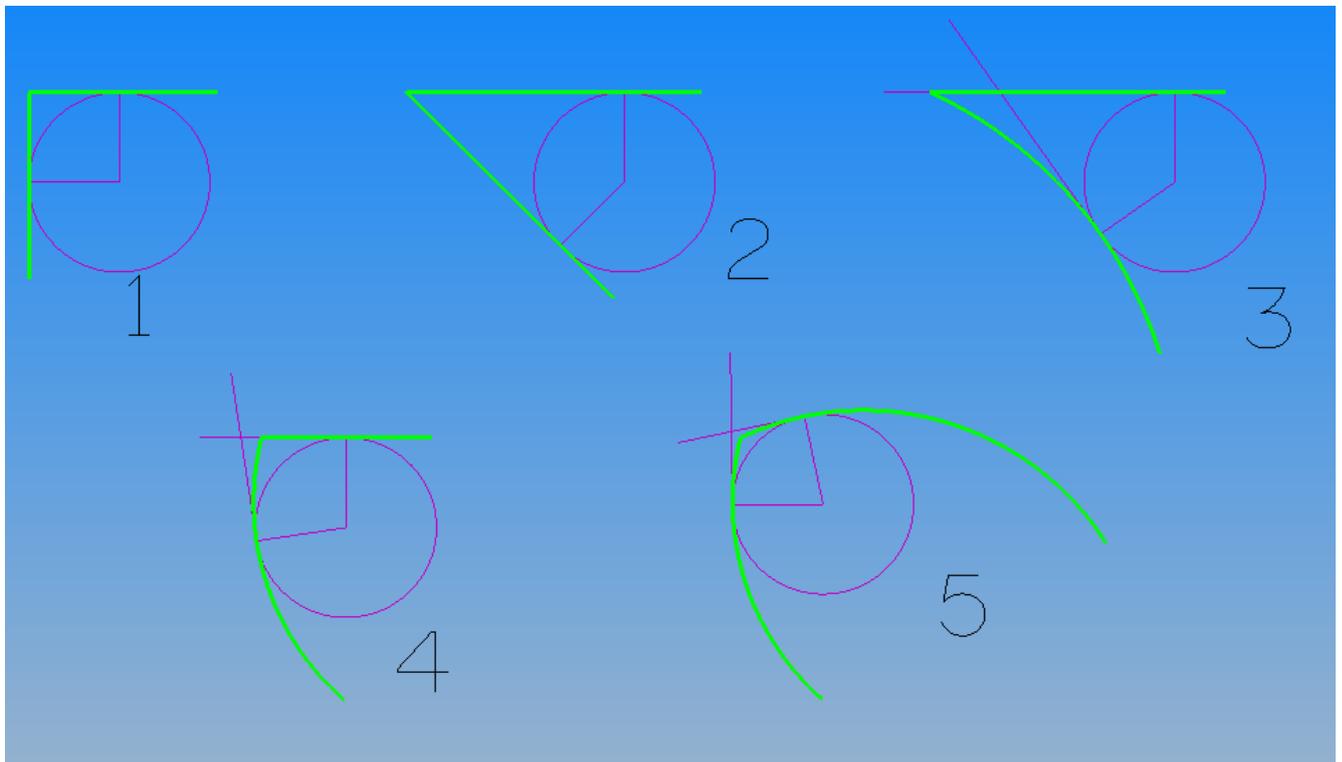


Figure 175 - Engraving using 2D Exact Engraving

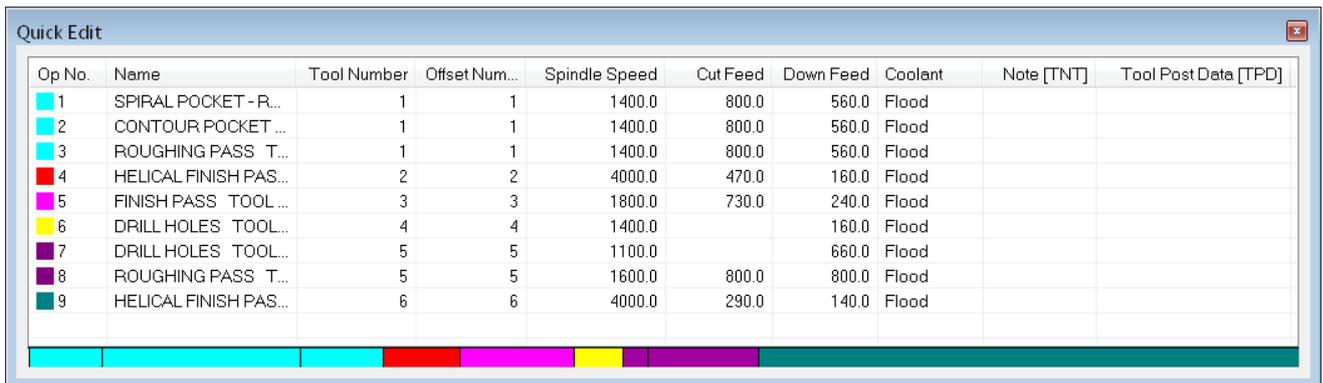
In the images above, corners 1, 2 & 3 are qualifying corners, 4 & 5 are NOT.

To engrave corners 4 & 5 you must use the  **Geometries** option instead.

Quick Edit

The quick editor allows you to quickly change Tool Numbers, Offset Numbers, Spindle Speeds and Cut Feeds and Down Feeds.

To use the Quick Editor, all items that need to be altered must be ticked on in the Operations page of the Project Manager. If you cannot see it, you cannot edit it.



Op No.	Name	Tool Number	Offset Num...	Spindle Speed	Cut Feed	Down Feed	Coolant	Note [TNT]	Tool Post Data [TPD]
1	SPIRAL POCKET - R...	1	1	1400.0	800.0	560.0	Flood		
2	CONTOUR POCKET ...	1	1	1400.0	800.0	560.0	Flood		
3	ROUGHING PASS T...	1	1	1400.0	800.0	560.0	Flood		
4	HELICAL FINISH PAS...	2	2	4000.0	470.0	160.0	Flood		
5	FINISH PASS TOOL...	3	3	1800.0	730.0	240.0	Flood		
6	DRILL HOLES TOOL...	4	4	1400.0		160.0	Flood		
7	DRILL HOLES TOOL...	5	5	1100.0		660.0	Flood		
8	ROUGHING PASS T...	5	5	1600.0	800.0	800.0	Flood		
9	HELICAL FINISH PAS...	6	6	4000.0	290.0	140.0	Flood		

Figure 176 - Quick Edit window

This is ideal for multiple machine changes, where you have the same job on different machines, just change the values in the Quick Editor and this will change all the operations.

To make a change **<RClick>** on the value to be changed and the popup dialogue is displayed where the change can be made and how it is to be applied.

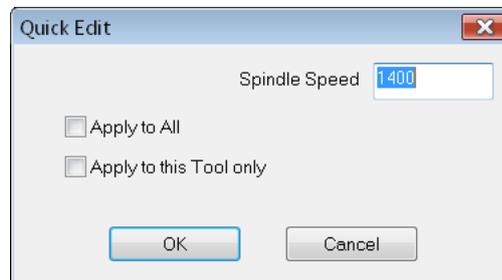


Figure 177 - Quick Edit alteration dialogue

Apply to All changes every instance in the Quick Edit dialogue listing to the new option in the dialogue form.

Apply to this Tool only sets only the same corresponding tool within the operations list to the new value.

Machining Styles

Using Machining Styles is the first step toward automated NC code production and is a sequential process.

Therefore, it is important that you can manually apply tool paths repeatedly and fully understand the process that you wish to automate.

Machining Styles are single or multiple machining operations that are stored without associated geometry. Machining Styles can be applied to applicable geometries in new jobs therefore improving the production of the NC program.

Care must be taken when creating/using machining styles as to tool numbers used.

Machining Styles Project Manager

The Machining Styles display shows a tree structure of all the Machining Styles available in the module and all the machining operations that they contain.

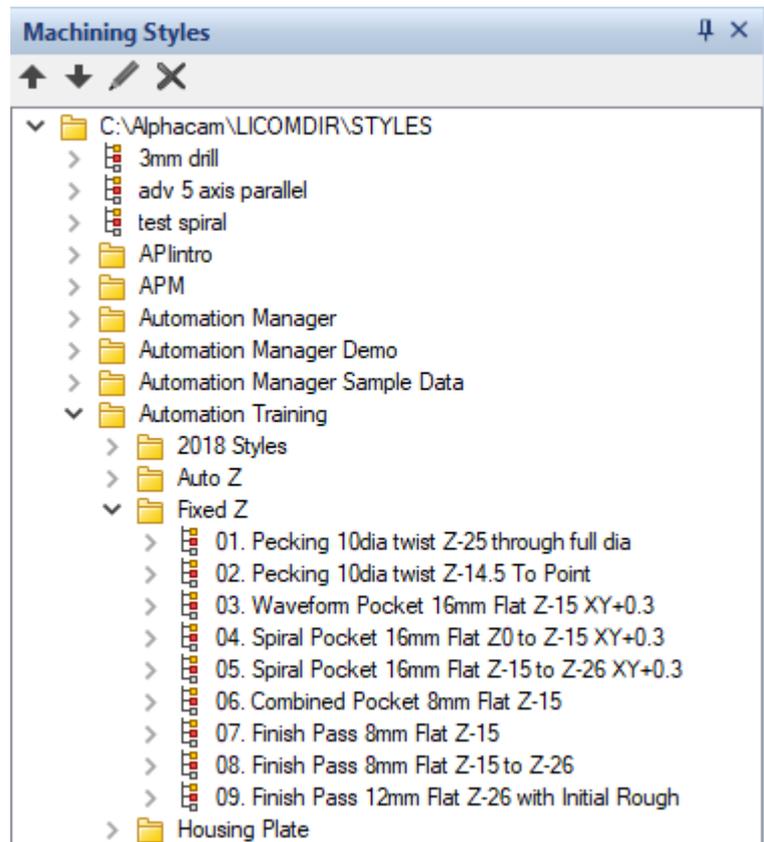


Figure 178 - Machining Styles Project Manager page

A machining style can contain one or more 2D and or 3D machining operations which can be individually applied to selected geometries or automatically to geometries on assigned layers.

The machining operations contained within a style can be modified prior to their application or the Operation can be altered under the Operations Tab after it has been applied to the geometries.

Creating a style

Styles are created from existing machining operations.

This is achieved by using the **<RClick>** menu option on the required cycle and choosing either **Save as New Style** or **Add to Existing Style** from the popup menu.

The fixed or calculated feed rates and spindle speeds existing in the operation will be stored in the style.

Organising Machining Styles

When saving styles, you should organise them into different folders, this will enhance the management and ease of access for similar groups of strategies. E.G. separate folders for fixed depth machining, and Auto Z machining or specific component production Windows, Doors, etc.

Manipulation and Duplication of Machining Styles

Styles and Operations can be copied; the order of the operations within a style can be adjusted. These options are accessed from the styles project manage popup menu.

Applying Machining Styles

The Machining Style popup menu allows you to:

Apply Style to Selected Geometries

This will apply all the machining operations stored in the style to the geometries selected.

Apply Style to Selected Layer

This will apply all the machining operations stored in the style to the geometries which are on a user layer. The user layer is selected from a list of current user layers.

Assign Layers for Geometries

This will assign the style to the layer name. The layer can either be selected from the list of current layers; although the name can be specified if it does not exist this command will not be actioned. To remove the assigned layer from a style, delete the assigned layer name in the dialogue box.

Create Layers for Styles.

This creates any layers which have been assigned to the style and operations that do not currently exist in the job.

Assign Layer for Boundaries

This is only applicable to styles containing 3D surface machining operations and allows you to specify the layer on which the boundary geometries are stored.

Assign Layer for Points

This is only applicable to styles containing 3D surface machining operations that require user input to select a start point. The layer should only contain one circle whose centre will be used as the start point.

Auto Styles Macro

The auto styles macro allows a collection of Machining Styles to be automatically applied to geometries on different user layers. This is a more advanced method of applying machining and is covered in detail in the Automation course **ALP TRG 202**.

NC Program Production Process

1. Study the component drawing or solid and produce a process plan for the Job. This should identify which features are to be machined, with what tool(s) and in what order. From the process plan you will identify the relationship between the feature geometry and the tool position, start point and direction of cut.
2. Construct Geometries using techniques shown by your tutor i.e. Lines, Arcs, etc. Trim, Break, etc. Make sure the geometry is the correct scale, it is in the correct place relative to the machine datum, and the geometry is in the form of joined profiles i.e. one ghost tool for each profile. Alternatively, correctly manipulate and orientate the solid then extract the geometries for the features to be machined and the geometry is in the form of joined profiles i.e. one ghost tool for each profile.

3. Display all the **Ghost Tools** , and select **MACHINE > Tool Directions** .
4. Set the options which define which side of the profile the tool is to be positioned when cutting process is applied. Pick the geometry onto which the setting is to be applied. Repeat this process until all the ghost tools have been positioned correctly. Then select **[Close]** on the dialogue.
5. Use **EDIT > Start Order > Start Pt**  and select the position where you require the cutting process to start if it is not already at the right place.



This can be done at the same time as setting the tool directions.

6. Use **HOME > Select Post**  and pick the post processor for the machine tool to be used.



If you only have one machine it is not necessary as your post processor will be set as default.

7. If the tool has been defined to calculate feeds and speeds, select **MACHINE > Select Material**  and select the applicable material and cutting conditions from the drop-down list.

8. Use **MACHINE > Select Tool**   from the dialogue, select the tool library and highlight the tool that is to be used for the first operation.
9. Select the machining strategy Pocketing, Rough or Finish, Drilling etc. Set the relevant parameters in the dialogues. Then select the geometries to which the strategy is to be applied. Ensure you select **[Finish]** or **<Esc>** to action the tool path creation.

Repeat from 7 until all the machining operations have been created.



Tool lead in and out options are mandatory for profile tool paths using cutter compensation G41/G42 on machines where compensation cannot be applied on the rapid motion.

10. Use one of the Simulation options to see the machining process; remember to define material unless you wish to let the system create a default material.



If there are errors evident in the simulation correct them; modify the geometry and update the tool paths or edit the tool path operation in the Operations project manager page.

11. Save the ALPHACAM Job file.



12. Use **HOME > Output NC** using to generate the NC code output.



If the machining operations are typical processes in your part production you can, once the operations have been proved, use them to create machining styles. Machining styles can be used to apply proven machining strategies to future geometries and therefore save time in programming.



Version amendments

V	Amendment Description	A	Software Version	Amended Date
13	Minor text formatting alterations.	1	2020.1	03/10/2019
13	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

