RhinoCAM 2.0

Training Manual

© MecSoft Corporation

Table of Contents

Introduction 6
Course Objective 6
Configurations of RhinoCAM 8
Machining Process 9
Understanding RhinoCAM 10
Programming Workflow 12
Machining Methods 13
Overview of Machining Methods 14
User Interface 16
Application Screen 16
Menus and Toolbars 18
Part Units 19
Machining Regions 20
Create Flat Area Regions 20
Create Surface Boundary 21
Create Section Curves 21
Editing Geometry 22
Reverse Curve 22
Change Start of Curve 23
Preparing the part for machining 25
MOps Browser - Setup Tab 25
Machine Setup (3Axis) 26
Post Options 28
Preferences 29
Color Preferences 29
Machining Preferences 30
Simulation Preferences 32
Utilities 34
G-Code Editor/Analyzer 34
Post Processor Generator 35
Orient Geometry 36
Create Stock 38
Box Stock 38
Part Box Stock 39
Locate WCS 40
Align Part and Stock 42
Cutting Tools Browser 44
Create/Edit Tools 44
Adjust Register 45
Cutcom Register 45
Z Offset 46
User Defined Tools 46
Load Tool Library 47
Select Tool from Library 48
List Tools 49
Save Tool Library 49
Selecting Machining Features / Regions 49
Select Curves as Regions 49
Select Flat Area Feature 52
Cutter Compensation 55
Create 2 ½ Axis Toolpaths 57
Profiling 57

Notes:

Cut Parameters 57
Cut Levels 59
Entry/Exit 60
Advanced Cut Parameters 61
Sorting 63
Clearance Plane 63
Facing 65
Cut Parameters 65
Cut Levels 66
Pocketing 67
Cut Parameters 67
Entry / Exit 68
Hole Pocketing 70
Cut Parameters 70
Entry / Exit 71
Engraving 73
Cut Parameters 73
Entry/Exit 74
V-Carving 76
Cut Parameters 76
V-Carve Roughing 77
Cut Parameters 77
Chamfering 78
Cut Parameters 79
Thread Milling 81
Cut Parameters 81
Entry / Exit 82
Re- Machining 83
Cut Parameters 83
Simulating Toolpaths 84
Tool Animation 86
Cut Material Simulation 86
Editing Machining Operations 87
Editing operations Associatively 88
Post Processing Machining Operations 91
Create 3 axis Toolpaths 94
Horizontal Roughing 94
Cut Parameters 94
Cut Levels 96
Engage / Retract 98
Advanced Cut Parameters 98
Clearance Plane 99
Parallel Finishing 100
Cut Parameters 100
Z containment 101
Entry / Exit 102
Horizontal Finishing 104
Cut Parameters 105
Optimized Machining 105
Entry / Exit 106
Radial Machining 108
Cut Parameters 108
Spiral Machining 110
Cut Parameters 110

Notes:

Create Hole Machining 112
Drilling 112
Hole Features 113
Cut Parameters 114
Sorting 115
Tapping 116
Cut Parameters 117
Boring 118
Cut Parameters 118
Reverse Boring 119
Advanced 3 axis Toolpath Methods 120
Horizontal Re-Roughing 120
Plunge Roughing 121
Cut Parameters 121
Engage /Retract 122
Plunge Re-Roughing 123
Projection Pocketing 124
Cut Parameters 124
3d Offset Pocketing 126
Cut Parameters 126
3D Offset Profiling 127
Cut Parameters 127
Pencil Tracing 128
Cut Parameters 128
Advanced Cut Parameters 129
Valley Remachining 132
Cut Control 132
Cut Parameters 133
Plateau Machining 136
Cut Parameters 136
Parallel Hill Machining 138
Cut Parameters 138
Horizontal Hill Machining 140
Cut Parameters 140
Curve Machining 142
Cut Parameters 142
Between 2 Curves Machining 144
Machining Features / Regions 145
Cut Parameters 146
Reverse Post Milling 148
Cut Parameters 148
Toolpath Editor 150
Global Edits 151
Transform Toolpath 151
Move Toolpath 151
Rotate Toolpath 152
Mirror Toolpath 153
Scale Toolpath 154
Reverse Toolpath 155
Instance Toolpath 155
Z Instance Toolpath 156
Fit Arcs to Toolpath 157
Linearize Toolpath 158
Selection Edits 158

Notes:

Move Selection in Z 159
Knowledge Base 160
Fixture Offsets 162
Create 4 axis Toolpaths 166
Machine Setup (4Axis) 166
4th Axis Options 167
Rotate Table 167
Incremental Rotation 169
4th Axis Facing 172
4th Axis Pocketing 173
4th Axis Profiling 174
4th Axis Roughing 175
Cut Parameters 175
Step Down Control 178
Clearance Plane 179
4th Axis Finishing 180
4th Axis Engraving 181
Cut Parameters 182
4th Axis Hole Making 183
4th Axis Drilling 184
4th Axis Tapping 184
4th Axis Boring 185
4Axis Reverse Boring 185
Create index 5 axis Toolpaths 186
Machine Setup (5Axis) 187
SetMCS 188
Align MCS With 189
SetMCS Origin 191
Using SetMCS for Orientation 193
Post Processor Generator 197
Customizing Post Processor 197
Main Editor 199
Variable List Dialog 201
General Tab 202
Motion Tab 205
Circle Tab 206
Helix/Spiral Tab 208
Spindle Tab 209
Feed Rate 211
Miscellaneous Tab 212
Start/End Tab 214
Tool Change Tab 215
Multi Axis Motion tab 216
Cycles Tab 218
Variables Tab 219
Editing Post Processor Variables 219

Notes:

Chapter 1

Notes:

Introduction

RhinoCAM is a unique CAM product plug-in that runs inside of Rhinoceros 4.0. This fully integrated product seamlessly integrates Rhino's CAD functionality with toolpath generation and cutting simulation/verification, in one package that is both easy and fun to use. RhinoCAM's machining technology capabilities enable you to produce toolpaths that you can send to the machine with utmost confidence.

You can work with the native Rhino design data as well as use any of the data types that can be imported into Rhino such as solids, surfaces and meshes. Then you can use RhinoCAM with its wide selection of tools and toolpath strategies to create machining operations and associated toolpaths. These toolpaths can then be simulated and verified, and finally post-processed to the controller of your choice.

RhinoCAM offers fast gouge free solids/surface model machining technology coupled with cutting simulation/verification capabilities running on standard Windows hardware. These features take the uncertainty out of the toolpath programming process and allow you to produce toolpaths that you can send to the machine tool with utmost confidence. A simple and well thought out user interface makes this system one of the most intuitive and easy to use milling systems in the market.

RhinoCAM supports milling operations from simple 2 ½ axis milling to complex 3 axis milling. Additionally drilling operations are supported in three axis mode. Most of the popular machine tool controllers are supported out of the box. Additionally RhinoCAM also includes a post-processor generator product that can be used to tailor RhinoCAM's output to virtually any machine tool controller in the market.

Course Objective

During the course, you will learn to:

- Utilize the features of Rhino's user interface
- Create basic geometry

- Understand Machining Methods
- Generate 2 ½ Axis, 3 axis 4 axis Toolpaths
- Edit Toolpaths
- Simulate toolpaths
- Post Process toolpath to output G-code

Notes:

Configurations of RhinoCAM

Notes:

RhinoCAM 2.0 comes in 3 different configurations to suit the varied machining needs of our customers. These configurations are as follows:

Standard Configuration

This is a general purpose machining program targeted at the general machinist. This product is ideal for the rapid-prototyping, hobby and educational markets where ease of use is a paramount requirement. Packed with sufficiently powerful manufacturing methods this easy to use package is not only effective but also attractively priced for the budget conscious or entry level buyer.

4th Axis Add-on

Add-On module can be added on to the RhinoCAM 2.0 STD product. Allows 4th Axis Indexed and continuous roughing and finishing operations in RhinoCAM 2.0. Curve based continuous machining such as 4 Axis Facing, Pocketing, Profiling & Engraving as well as Surface/solids/meshes based machining such as Roughing, Finishing and Curve Projection machining are available.

PRO Configuration

This powerful package is ideal for mold, die and tooling, wood working rapid-prototyping and general machining markets. This product boasts of powerful toolpath generation strategies coupled with tools for efficiently controlling the cutting tool for detailed machining capabilities, while not sacrificing ease of use. Suitable for demanding users with sophisticated manufacturing requirements. (This product configuration includes the 4th Axis Module described above)

Machining Process

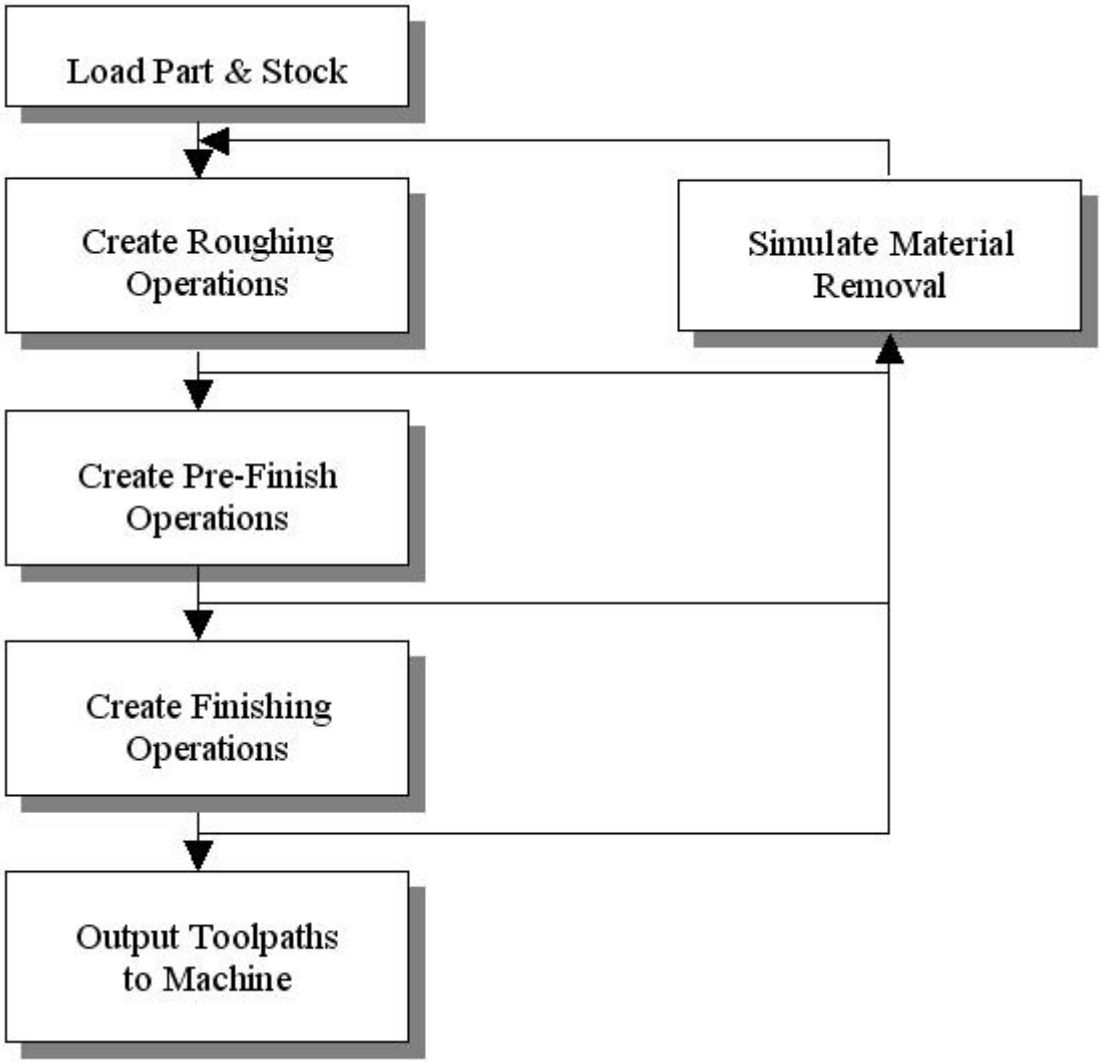
Notes:

The manufacturing process aims to successively reduce material from the stock model until it reaches the final shape of the designed part model. To accomplish this, the machinist or programmer utilizes a machining strategy. A typical machining strategy employed in the manufacturing industry is to use larger cutters to perform bulk removal of material early on in the manufacturing process. These operations are called roughing operations. This is then followed by operations employing successively smaller cutters removing proportionately smaller amounts of material from the stock model. This is done until the part has a uniform amount of stock left. These operations are called pre-finishing operations. This is then followed by finish operations. Here the uniform stock remaining on the part is removed by using a small cutter removing a constant amount of material with every motion to produce the net shape.

Understanding RhinoCAM

Notes:

The standard workflow of RhinoCAM mimics this process and can be represented by the flow chart shown below.



Part geometry is imported into RhinoCAM via the various data interfaces provided in Rhino. A stock model representing the raw stock from which the part needs to be manufactured can then either be created using the various tools provided in RhinoCAM or imported.

The user then determines the machining strategy to be used in manufacturing the part. This can be done by loading a previously saved manufacturing operation sequence or by creating a new one. This manufacturing strategy is represented by a sequence of machining operations in RhinoCAM. To create a new machining strategy, the user simply selects the tools and the machining operations in sequence and generates toolpaths. The system automatically records this sequence. This record can be archived as an operation list that can be retrieved for later use.

To create a new machining operation, the user selects a tool followed by the type of toolpath to be created. The user then selects the parameters to use for machining and then generates the toolpath.

In RhinoCAM a 3D part is typically machined using 3 axis operations. It can be rough machined initially by employing a horizontal roughing operation, followed by a finishing operation. The horizontal roughing toolpath will typically be generated using a relatively large flat end mill or an end mill with a corner radius to efficiently remove bulk of the material from the raw stock. This machining operation could be followed up with another horizontal roughing operation either with the same tool or with a smaller tool to remove more material.

The part can then be pre-finished by employing either the parallel finishing operation or the horizontal finishing operation. Pre-finishing and finishing operations typically employ ball end mills with or without a side angle.

Finally a parallel finishing operation using a small radius ball tool and employing a fine step-over is then used to finish the part to net-shape. This may or may not be followed by other re-finishing operations to reduce handwork on the part.

For prismatic parts that have straight walls, it might be more expedient to use 2-1/2 axis toolpath methods rather than the 3 axis methods described above. The strategy followed however will be similar where in bulk material removal or roughing will be performed by a combination of facing and pocketing operations. These will then be followed up with one or more profile finishing operations. It should be noted that for 2-1/2 axis operations it is not necessary to have the complete 3D geometry defined. Curves that represent the sides of the part are sufficient to define the geometry for machining.

Once all of the operations are completed the user can then go back and review the operation sequence, re-order operations if desired and output the toolpath for post-processing. The "Operations Browser" can be used to manage these operations.

Notes:

Notes:

Programming Workflow

Once the part is loaded, the typical workflow is reflected in the layout of the tabs of the Mops Browser window. The workflow is designed to allow the user to work starting from the left most tab and ending at the right most tab. As each tab is accessed a toolbar with functions specific to the tab chosen will be displayed just below the tab. The functions in each of the toolbars corresponding to each tab are also best accessed in order from left to right.

Thus the user typically would start with the **Setup** tab and access each of the buttons, optionally, in the toolbar that appears when this tab is selected in sequence from left to right. Once the setup functions are completed, the user will then proceed to the **Create** tab to commence programming the part. Once machining operations are completed the user can simulate the operations by selecting the **Simulate** tab before finally sending the operation codes to the machine tool.

Step 1: Setup before programming

Step 2: Create machining operations

Step 3: Simulate machining operations

Once the machining operations have been created and verified, they can be post processed to create G-code files. These G-code files can then be sent to the controller of the machine tool to drive the actual machine tool.

Machining Methods

There are two major classes of machining operations that can be created in RhinoCAM - milling and drilling. Milling operations are used to mill out material to form shapes. Drilling operations are used to create holes. Both classes are essential in any manufacturing industry.

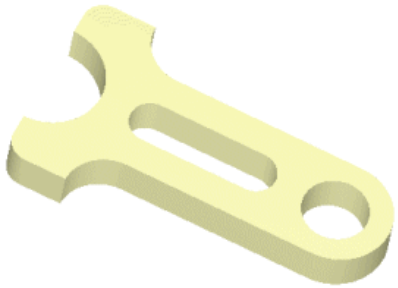
Notes:

Overview of Machining Methods

Notes:

Milling operations can be categorized as 2½ axis, 3 axis, 4th axis, and 5th axis milling.

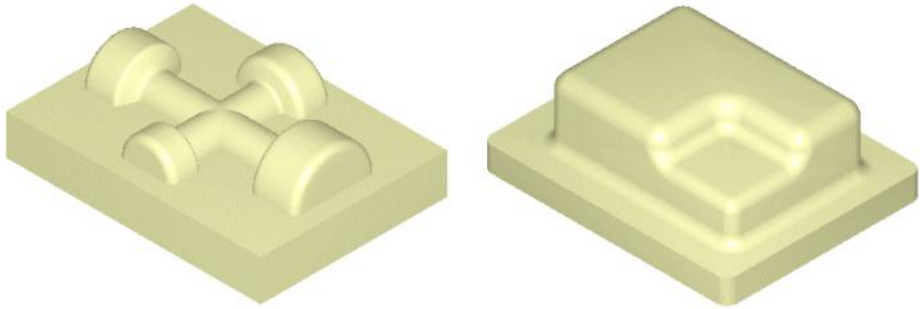
2½ Axis: The tool can move in X and Y directions, while the Z level is fixed at set locations. Because 2½ toolpaths do not relate to either part or stock geometry, machining regions must be selected; these define the boundaries of tool motions.



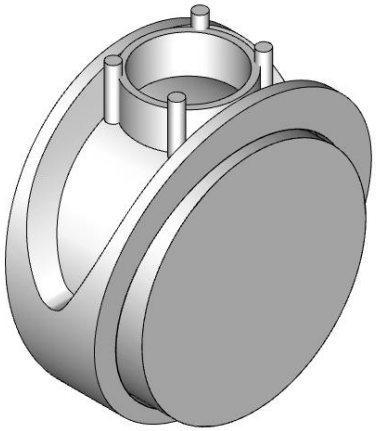
This type of machining is useful for machining prismatic parts – extrusions of curves along the Z axis. Because of its straight sides, a prismatic part can be machined by locking the tool at the first Z level, performing XY motions, then repeating for subsequent Z levels.

Using this class of machining, you can machine parts that are defined only by 2D curves. 3D part geometry can be present, though it is not necessary.

3 Axis: The tool can move simultaneously in all three directions. This is appropriate for parts that have complex, curved, and non-vertical surfaces.



4 Axis: Fourth Axis operations are used to machine parts that cannot be machined with simple 2 ½ axis or 3 axis machining operations. Since the tool moves only up and down along the Z axis during 3-Axis milling, areas that cannot be seen from above cannot be cut.



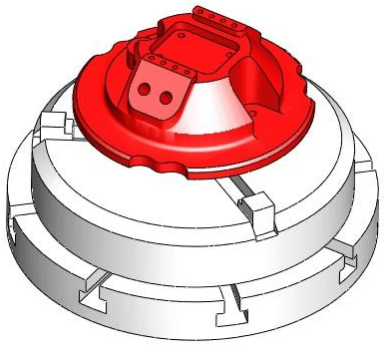
- Index 4 Axis: RhinoCAM's implementation of indexed fourth axis milling allows the user to do this automatically if the machine tool is equipped with a rotating head or table. Indexing refers to the ability of rotating the part about the X or Y axis and then performing machining in a 3 Axis fashion with the part locked at this new orientation.

- In the continuous mode, the tool axis is always pointed normal to the rotation axis of the machine tool. Rotary axis can be set to X (A) axis or Y (B) axis.

5 Axis: Index 5-axis machining, also known as 3+2 machining. Fifth axis machining enables you to change tool direction relative to any set of axes.

Notes:

Notes:



RhinoCAM's implementation of indexed fifth axis milling allows the user to do this automatically if the machine tool is equipped with a head/head, table/table or a head/table configuration. Indexing refers to the ability of rotating the head/head or the table/head on the machine tool and then performing machining in a 3 Axis fashion with the part locked at its orientation.

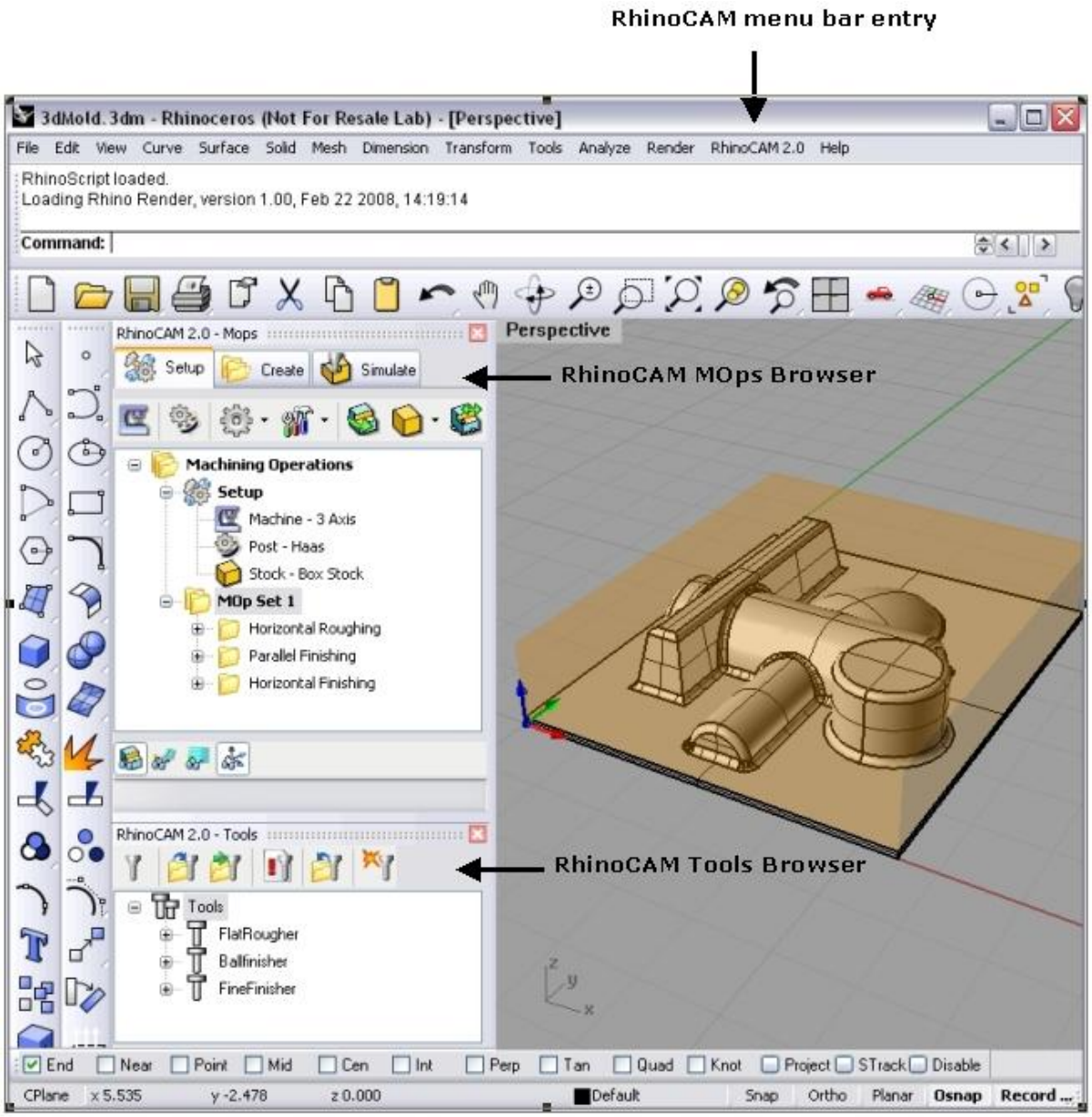
User Interface

RhinoCAM adheres to the Windows standard for user interface design and integrated into Rhinoceros 4.0 environment seamlessly.

Application Screen

There are 3 main interface objects created when RhinoCAM is loaded.

1. RhinoCAM menu bar item in the Rhinoceros 4.0 menu bar
2. RhinoCAM Mops Browser window
3. RhinoCAM Tools Browser window


RhinoCAM menu bar entry
3dMold.3dm - Rhinoceros (Not For Resale Lab) - [Perspective]
File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render RhinoCAM 2.0 Help
RhinoScript loaded.
Loading Rhino Render, version 1.00, Feb 22 2008, 14:19:14
Command:
RhinoCAM 2.0 - Mops
Setup Create Simulate
RhinoCAM MOps Browser
Machining Operations
Setup
Machine - 3 Axis
Post - Haas
Stock - Box Stock
MOp Set 1
Horizontal Roughing
Parallel Finishing
Horizontal Finishing
Perspective
RhinoCAM 2.0 - Tools
RhinoCAM Tools Browser
Tools
FlatRougher
Ballfinisher
FineFinisher
End Near Point Mid Cen Int Perp Tan Quad Knot Project STrack Disable
CPlane x 5.535 y -2.478 z 0.000 Default Snap Ortho Planar Osnap Record ...

Notes:

Menus and Toolbars

Notes:

When RhinoCAM is invoked it will add a menu bar item, titled **RhinoCAM 2.0** to the main Rhino menu bar. Selecting this menu bar item will create a drop down menu as shown below.

Each of the menu items is described below:

Machining Operations Browser - This is used to toggle the display of the Mops Browser window from the Rhino user interface

Cutting Tools Browser - This is used to toggle the display of the Cutting Tools Browser window from the Rhino user interface

Toolbars contain buttons that provide shortcut to commands. You can float a toolbar anywhere on the screen, dock it at the edge of graphics area.

Part Units

RhinoCAM works in either **Inches** or **Millimeters**. When units are switched from Inches to Millimeters or vice versa, machining operations and cutting tools information are not transferred. It is recommended that the user checks model units before programming with RhinoCAM. To check model units select Tools->Options. This brings up the Rhino Options dialog box. Click on Units under Rhino Options to verify the model units.

Notes:

Chapter 2

Notes:

Machining Regions

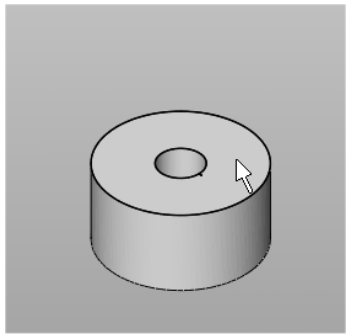
RhinoCAM allows the user to select open and closed curves (called regions) in the XY plane to control the areas to be machined. This is a very powerful feature and affords the user fine control over the manufacturing process. Multiple regions can be defined and selected in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. Thus there are two operations that the user should be aware of when working with regions.

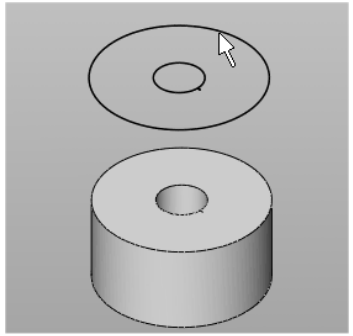
Create Flat Area Regions

Create a curve forming the boundary of a flat-area region. The region could be a face of a solid or a plane. If there are holes in the area, multiple curves will be made, one for the outer boundary, and one curve each for the inner boundary of the holes.

Use one of the methods under **Curve ->Curve from Objects**

Duplicate Face Border, Duplicate Border





Class Room Exercise: Open CreateFlatAreaRegions.3dm and create Flat Area curves using Duplicate Face Border.

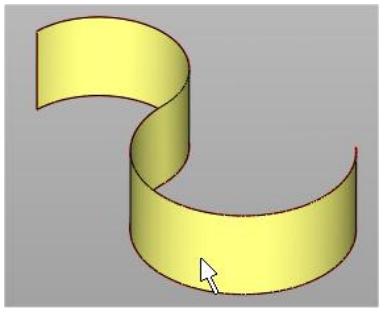
Notes:

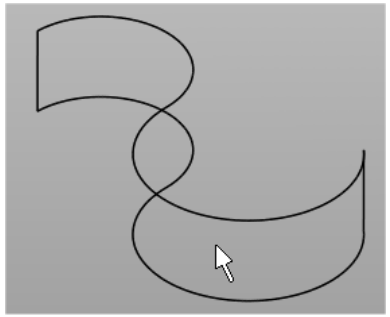
Create Surface Boundary

Create a single curve from the outer boundary of a surface.

Use one of the methods under **Curve ->Curve from Objects**

Duplicate Border, Duplicate Edge





Class Room Exercise: Open CreateSurfaceBoundary.3dm and create Surface Boundary using Duplicated Edge / Duplicate Border

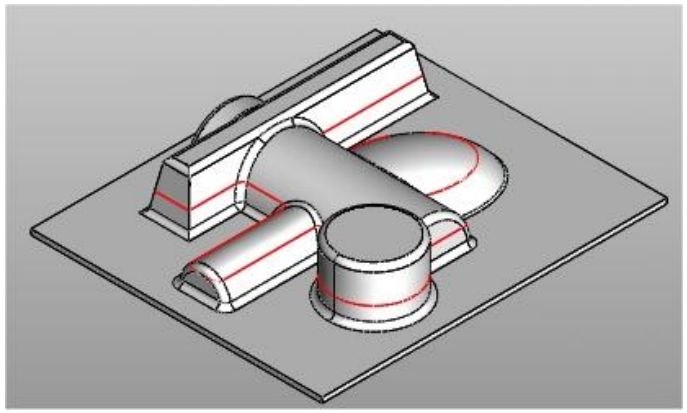
Create Section Curves

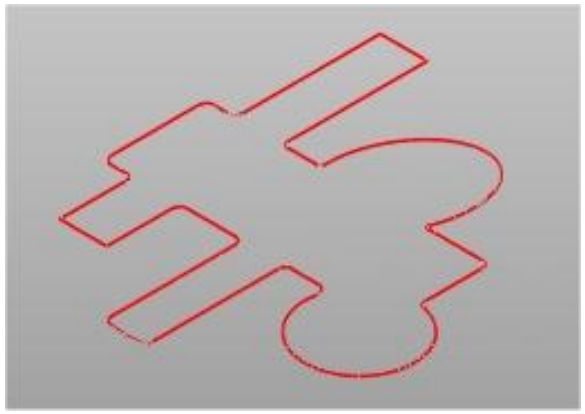
Curve ->Curve from Objects->Section

Creates cross-section curves of all solids. The curves will be independent once created.

Creates a planar curve or points resulting from the intersection of a defined cutting plane through curves, surfaces, polysurfaces, or meshes.

A plane intersects all geometry, and the curves of intersection are the cross-sections. The plane is parallel to the active construction plane.





Notes:

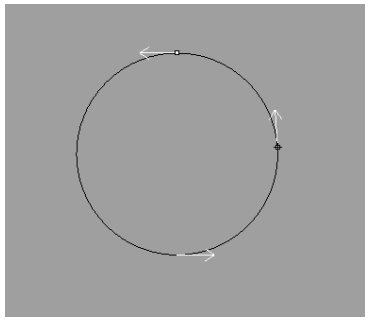
Class Room Exercise: Open 3Axis_Example1.3dm and create a section curve.

Editing Geometry

Reverse Curve

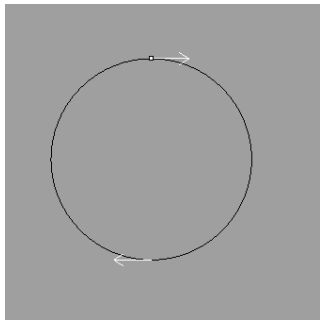
Reverse the orientation of curves.

1. Pick a curve or multiple curves.



2. **Analyze -> Direction**
3. Select Flip on under the Clickable options in the Command window and this flips the direction of curve.

Notes:



Press Enter to exit the command.

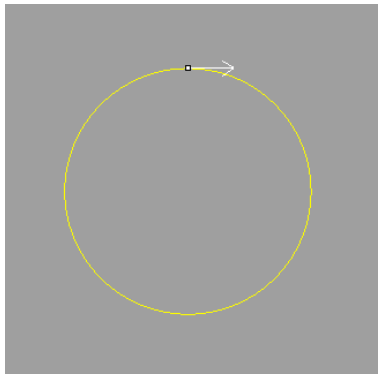
Alternatively you can select the curve and type **Flip** in the command window to reverse curve.

The direction of the curve is shown by the little arrow on the circle. It is now going the other way.

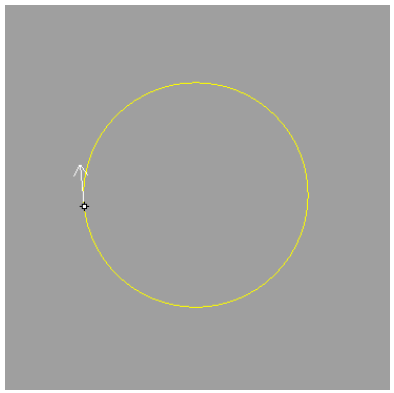
Change Start of Curve

Changes the starting point of a closed curve. It has no effect on open curves.

Curve->Curve Edit Tools->Adjust Curve Seam



The white dot is the start of the curve before it is changed. Left Click and move the start point on the circle to change the start point.



Press Enter to Exit the command.

The point of selecting the curve is where the start will be put.

Class Room Exercise: Open EditCurves.3dm

- Select the 1st curve and change start of curve
- Select the other curve and reverse the curve direction.

Notes:

Chapter 3

Notes:

Preparing the part for machining

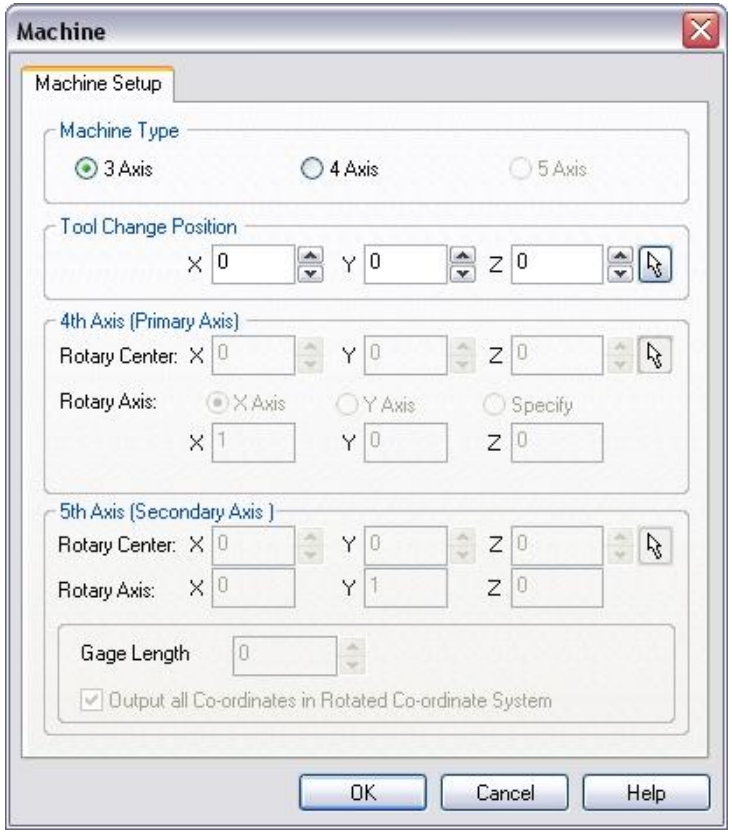
The Browser is a dock-able window that allows management of various entities or objects that can be created in RhinoCAM. There are 2 browsers in RhinoCAM – MOps and Cuttings Tools. MOps browser has three main modes of operation represented by tabs at the top of the window. These are Setup, Create and Simulate. Each tabbed view also incorporates a toolbar at the top. These toolbars group all of the functions associated with the type of object in the tab.

MOps Browser - Setup Tab

Selecting the Setup tab in the Browser provides access for specifying Machine type, Post Processor, Part and Stock Geometry.

Machine Setup (3Axis)

Sets the Machine for 2 ½ axis / 3 axis, 4 axis and 5 axis operations and Tool Change Position.



Machine Type: Allows the user to specify the machine type.

- Select 3 axis for 2 ½ and 3 axis Machining methods.
- Select 4 axis for index and continuous 4 axis machining methods.
    - Make sure to set the Rotary Axis to X if the part rotates about the X (A) axis or Y if the part is rotating about Y (B) axis.

Notes:

- Make sure to set the rotary center. The part geometry must pass thro' the rotary center. The rotary center is indicated with a Green arrow when the Machine Setup Dialog is selected and machine type is set to 4 Axis.

- Select 5 axis for index 5 axis machining methods.
  - Set the Primary Axis under the 4th axis Rotary Axis.
    - If the Primary Axis is A set the Rotary axis as X Axis.
    - If the Primary Axis is C axis, select Specify and set the Value of Z =1.
  - Set the Secondary axis under 5th Axis Rotary Axis.
    - Set X =1, Y=0, Z=0 for A axis,
    - Set X =0, Y=1, Z=0 for B axis,

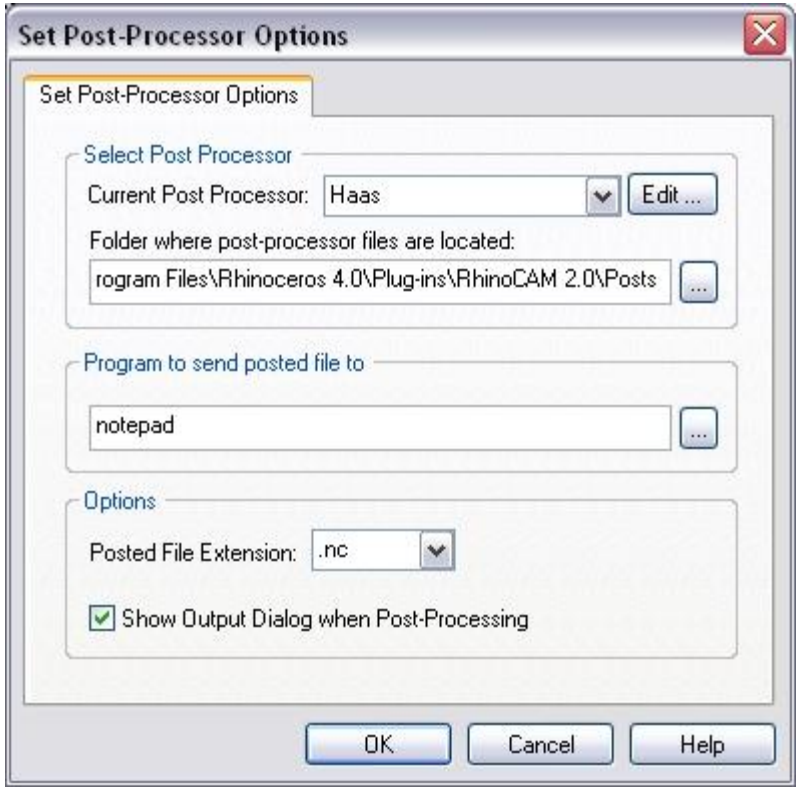
    - **Gage length:** This parameter is used for 5 axis toolpath computation. The distance from the tool tip to the pivot point determines the gage length.
    - **Output all Co-ordinates in Rotated Co-ordinate System** (MCS) –
      - This option is selected if the G codes need to be output in the local coordinate system.
      - Uncheck the box to output the G-code with respect to the World Coordinate System (WCS).

Tool Change Position: User can specify a coordinate location in X, Y & Z. RhinoCAM will take this coordinate values and output it for every tool change. The tool change variables must be configured in the post processor.

Notes:

Post Options

Allows the user to set the Current Post Processor, posted file extension, program to display the posted file.



Current Post Processor - User can change the default post processor by selecting a post from the list of available post processors under Current Post Processor.

Folder where post-processor files are located - RhinoCAM uses macro files with a .spm extension to handle post-processing to different controllers. These files are typically located in the "Posts" directory under the RhinoCAM installation folder.

Notes:

RhinoCAM by default looks in this directory to build the list of available post-processors shown under the Current Post. User can change the post processor by selecting a post from the list of available post processors under Current Post.

To change the post processor file location, user can specify the folder to find the Post-Processor macro files by selecting the "Browser for Folder" button in the dialog.

Program to send the Posted file to – This feature allows the user can specify a program to display the posted file. This could a text editor like notepad.

Posted File extension: - User can select a posted file extension from the list or add an extension to the list. Example (.cnc)

RhinoCAM by default performs interactive post-processing. That is, when the user selects a toolpath for post-processing RhinoCAM launches the post-processor and waits for the post-processor to complete. The user also has the option to turn off display of the output dialog (post & save dialog) for each post processing operation.

During interactive post-processing, RhinoCAM launches the notepad editor to view the output file. The user can specify a different text editor to use by selecting the editor name in the corresponding field in this dialog.

Preferences

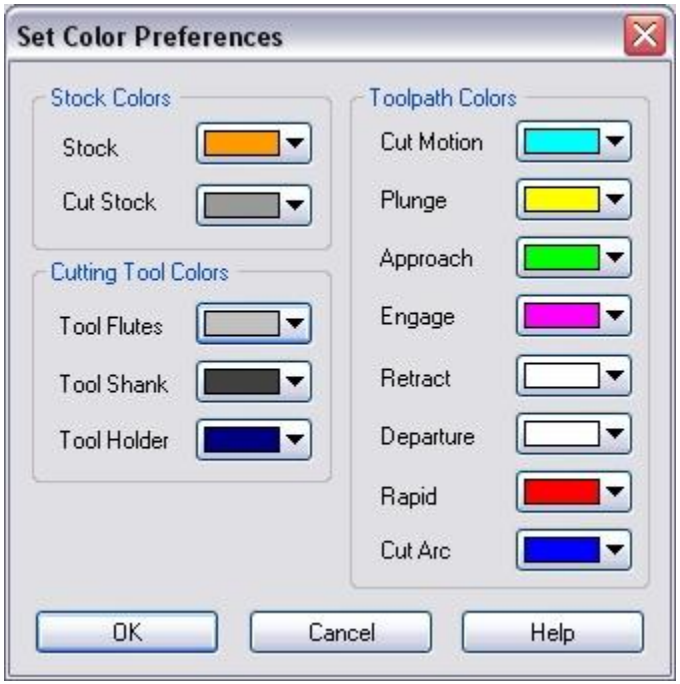
Provides access to specify Color, Machining and Simulation Preferences.

Color Preferences

Users can set the colors to display various objects in RhinoCAM using this dialog. To change each of the color settings in this dialog select the colored button next to the item of interest. This will bring up the color selection dialog, which can be used to choose the color needed. Once a color has been selected the button will change its color to the selected one.

Notes:

Notes:



Cutting Tool Colors - Set the tool's flute (cutting portion) and the shank's (non-cutting portion) color in this section.

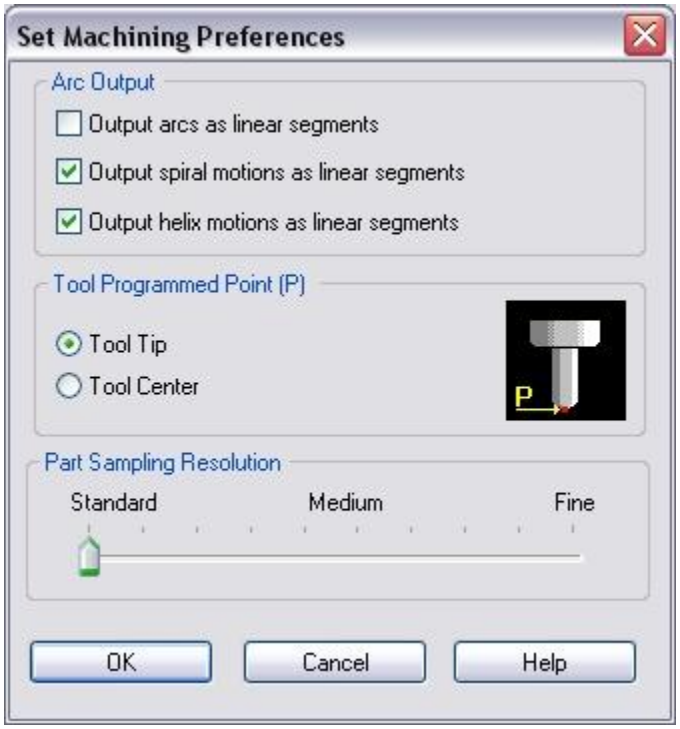
Stock Colors - Set the stock colors. The user can differentiate between cut and non-cut areas by specifying different colors for them here.

Toolpath Colors - The user can set the colors of various types of motions of the toolpath in this section. The types are classifies as plunge, approach, cutting, engage, retract, departure, rapid or transfer motions. Cutting motions can further be classified as linear or circular. Each of these types can be assigned different colors using the corresponding buttons in this section.

Machining Preferences

Used to select the machining preferences such as outputting arcs as line segments etc

Users can set the machining preferences using this dialog.



Arc Output: Some of the controllers do not have arc, spiral and helical output (for example G2, G3). For such type of controllers, the arcs that are generated in the RhinoCAM toolpath can be output as linear segments using this option.

Tool Programmed Point (P): The toolpath can be output as the tool tip or the tool center. If the output is set to be the tool center, the toolpath will be offset by the difference in the height of the tool tip and tool center. The default value is the tool tip.

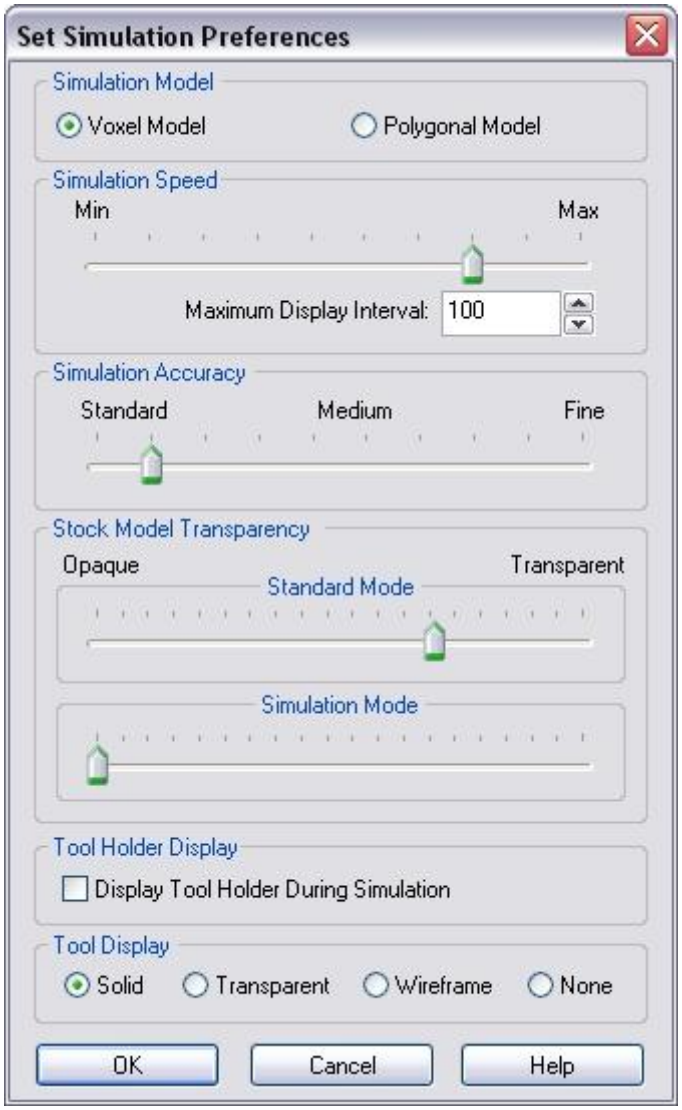
Part Sampling Resolution: This setting is used to control the quality of display of the simulated model. RhinoCAM also allows the stock model to be faceted to aid in its toolpath cut simulation computations. The user can control the accuracy of this model as well by selecting from Standard, Medium or Fine. The finer the stock faceting tolerance, the more accurate the toolpath cut simulation will be. However, this also results in slower performance. For large parts, the user is recommended to use the Standard or Medium options, while for smaller parts Medium or Fine options would work satisfactorily.

Notes:

Simulation Preferences

Used to select the simulation preferences such as simulation speeds, accuracy, stock model transparency etc

Users can set the simulation preferences using this dialog.



Notes:

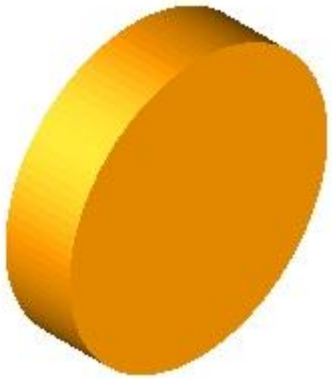
Simulation Model: In RhinoCAM the user can choose between two simulation models. One is called the Voxel Model and the other the Polygonal Model.

(**The Polygonal model is only available in the 4th Axis add-on option and PRO product**).

The Voxel model is a fast simulation model that is primarily used for 3 axis applications. It is especially useful when there are large amounts of toolpath blocks to be simulated. This model is fast but suffers from some accuracy limitations near vertical walls. The display quality of this simulation might also be insufficient for some applications especially when simulating near vertical walls. The Polygonal Model on the other hand is a high quality simulation model. This model uses more accurate simulation algorithms at the expense of speed. The speed of this simulation can be relatively slow when compared to the Voxel model. Additionally only the Polygonal Model of simulation can be used for 4 and 5 axis simulations. The Voxel model is limited strictly to 3 Axis applications.

Here is an example of a cylinder stock model representation with Voxel and Polygonal model.

Simulation Model - Voxel



Simulation Model – Polygonal

Simulation Speed: User can control the speed of the simulation using the slider bar and the Maximum display interval.

Simulation Accuracy: This setting is used to control the accuracy of display of the simulated model. The user can control the accuracy of the stock model by selecting from Standard, Medium or Fine. The finer the stock model accuracy results in slower performance and increases the simulation time.

Notes:

Stock Model Transparency: User can control the stock model transparency under standard mode and under simulation mode.

Tool holder Display: User can turn on /off the display of tool holder during simulation.

Tool Display: The cutting tool can be displayed either as a sold, Transparent, wireframe or can be turned off during simulation.

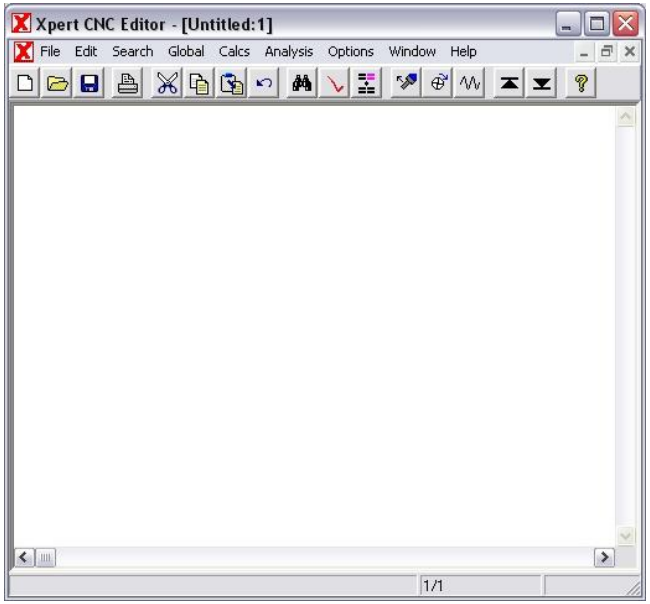
Notes:

Utilities

Provides access to G Code Editor and Post process generator.

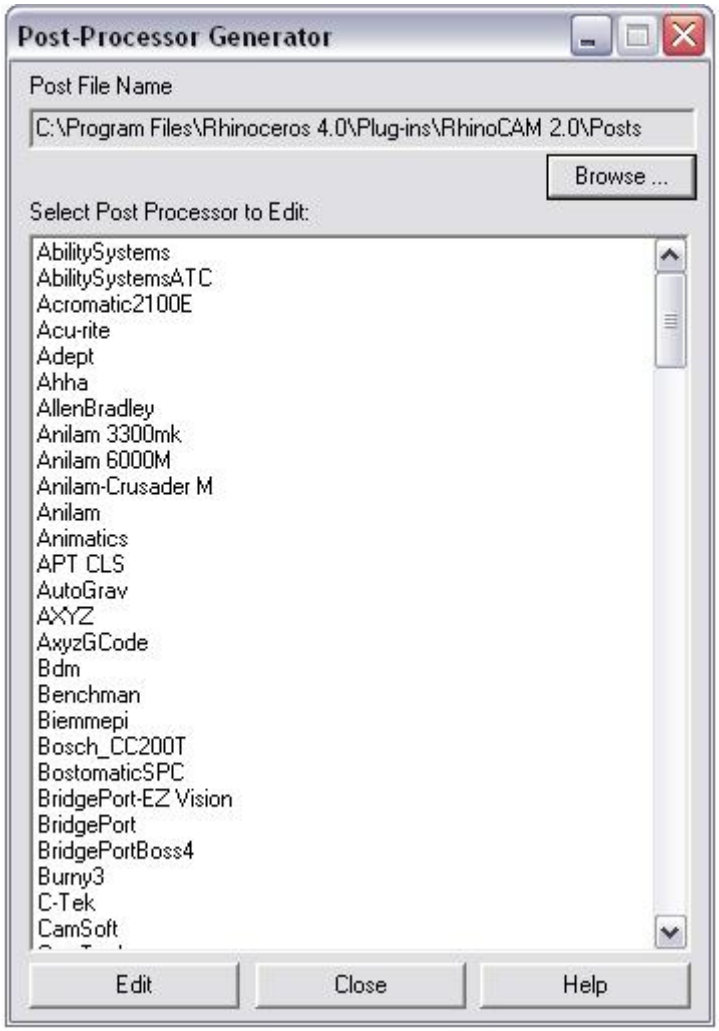
G-Code Editor/Analyzer

RhinoCAM bundles a 3rd party G-code editor program. This program can be used to edit the G-code files that are created by the post-processor.



Post Processor Generator

This utility can be used to set up post-processors to be used in RhinoCAM.



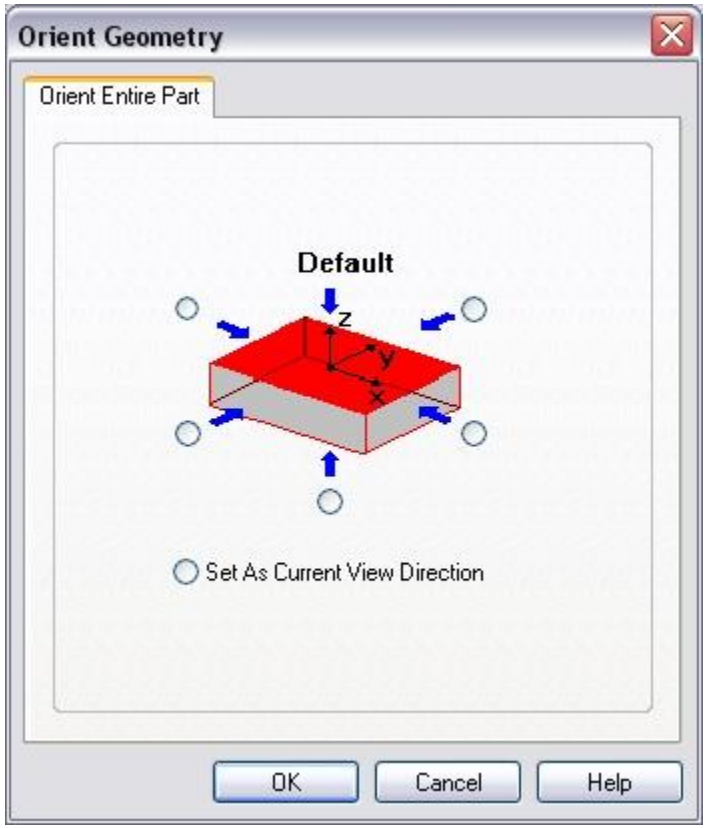
Notes:

Orient Geometry

The geometry can be oriented in any of the principal directions (+X, -X, +Y, -Y, +Z, -Z) using this option. The world coordinate system and the machine coordinate systems are aligned with the new cutting direction.

Once part geometry is loaded, users can set the cutting direction if the part is not oriented correctly with respect to how the part would be fixtured on the machine tool for cutting.

This dialog offers a convenient way of selecting each of the six principal directions to set the cutting direction.



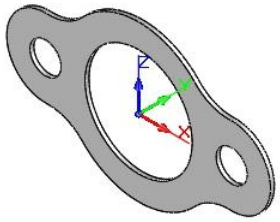
Notes:

Notes:

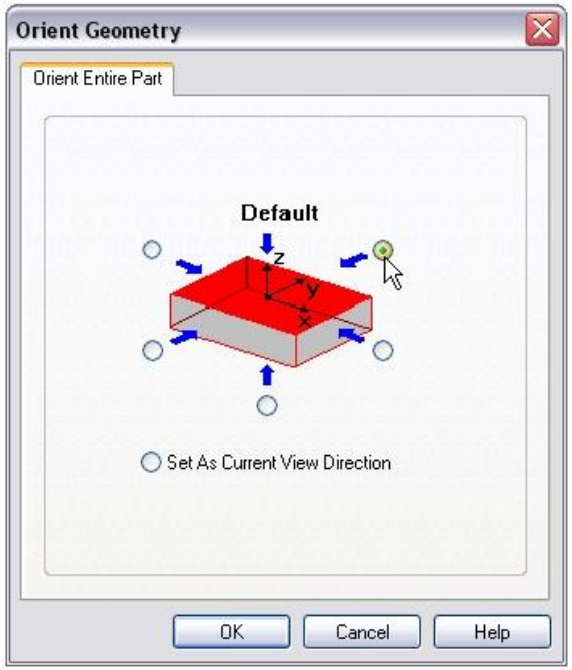
The cutting direction is considered as the direction of the tool axis. Once the user selects a direction all of the geometry will be rotated to an orientation where the selected cutting direction becomes the negative Z axis.

The steps to orient geometry are as follows:

1. Load the Part

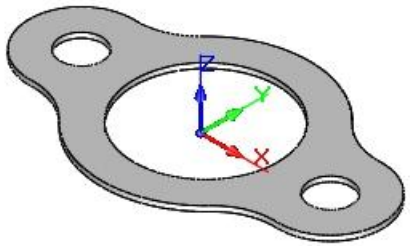


2. Select Orient Geometry from the Setup Tab under the RhinoCAM-MOps browser.
3. Now click on the Radio button next to the arrow corresponding to Y axis and click OK.



Notes:

4. The part is now oriented as shown below.



Create Stock

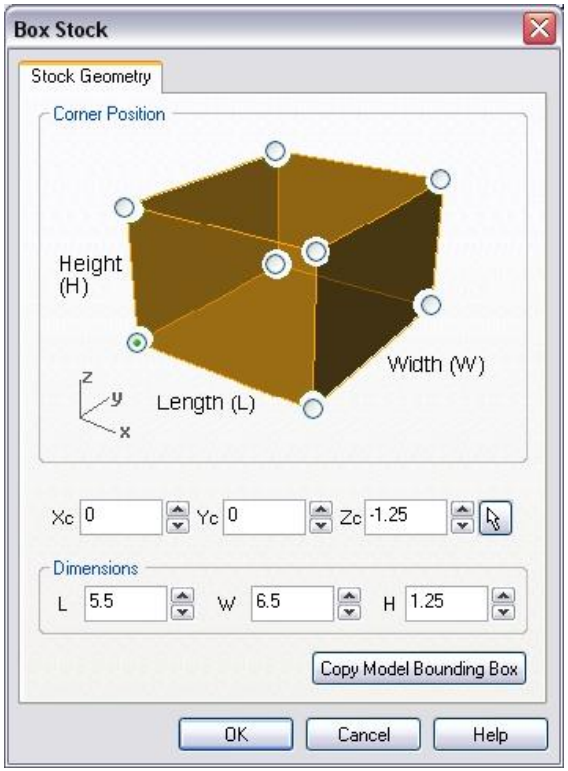
Stock Geometry represents the raw stock from which the design part needs to be manufactured. RhinoCAM allows the definition of various kinds of stock models.

The methods that are currently available to the user under STD configuration are Box Stock and Part Box Stock.

Box Stock

The user can define the raw stock model as a simple box by selecting the "Box Stock" option from the "Create/Load Stock" under the Setup tab in RhinoCAM-MOps Browser.

The user can define the box by simply typing in the length, width and height of the box in the corresponding edit boxes of the dialog. The corner position (origin) of the box can also be repositioned by picking the desired coordinate or by specifying the coordinate values in the corresponding edit boxes in the dialog (Xc, Yc, Zc). When the user clicks on the OK button, a stock model based on the user definition will be created and displayed.

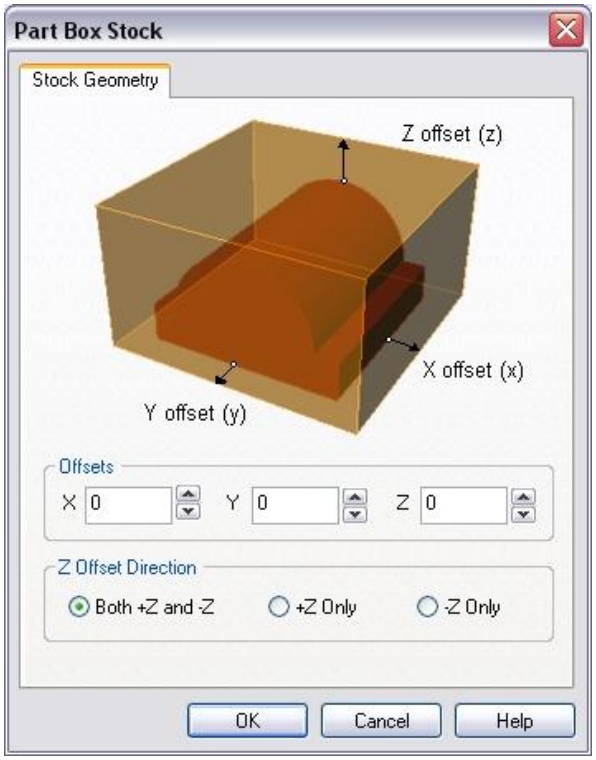


Part Box Stock

The user can define the raw stock model as a simple box by selecting the "Part Box Stock" option from the "Create/Load Stock" under the Setup tab in MOps Browser.

The system calculates the bounding box of the part model as the XYZ extents of geometry of the part model. The user can then define offsets in any of the three coordinate directions to apply to the computed bounding box. The system will expand the bounding box by the offset amount in each of the coordinate directions. When the user clicks on the OK button, a stock model based on the user definition will be created and displayed.

Notes:



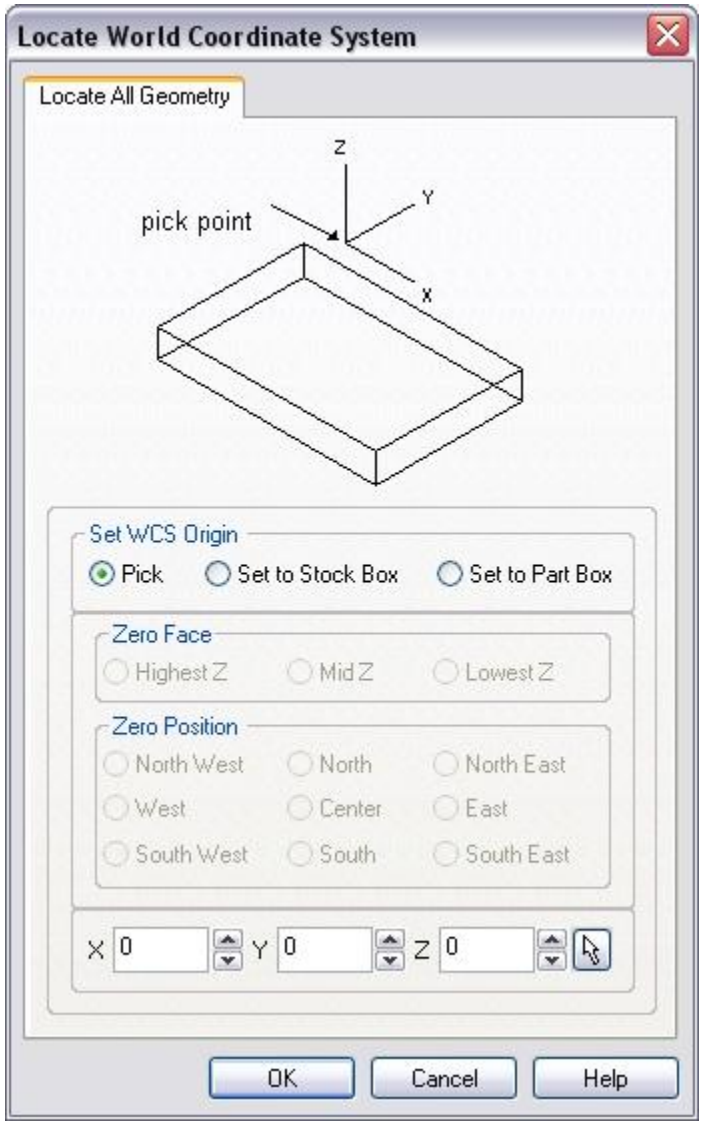
Locate WCS

The Machine Zero of the part & Stock can be set using this option. The world coordinate system and the machine coordinate systems are aligned with the new machine zero.

Once part geometry is loaded into Rhino, users can set the location of the World Coordinate System (WCS) Origin with respect to the geometry. An alternative way of thinking about this is to transform all loaded geometry to an appropriate location. This Locate WCS dialog offers the user a variety of ways of accomplishing this.

This dialog can be invoked by selecting Orient Geometry from the Setup tab under the RhinoCAM-MOps Browser.

Notes:



The user can set the origin by explicitly picking a point or can set it with respect to the Part or Stock geometry bounding boxes.

Notes:

Pick: If the user selects the Pick option, the button with the pick cursor close to the bottom of the dialog will be activated. The user can then click on this button to graphically select a point to align the WCS origin to.

Set to Stock Box: Selecting this item will activate the Zero Face and the Zero Position sections of the dialog. The user can then select the Z and the XY locations, with respect to the bounding box of the stock geometry, by choosing the appropriate selections in the dialog.

Set to Part Box: Similar to the previous selection, selecting this item will activate the Zero Face and the Zero Position sections of the dialog. The user can then select the Z and the XY locations, with respect to the bounding box of the stock geometry, by choosing the appropriate selections in the dialog.

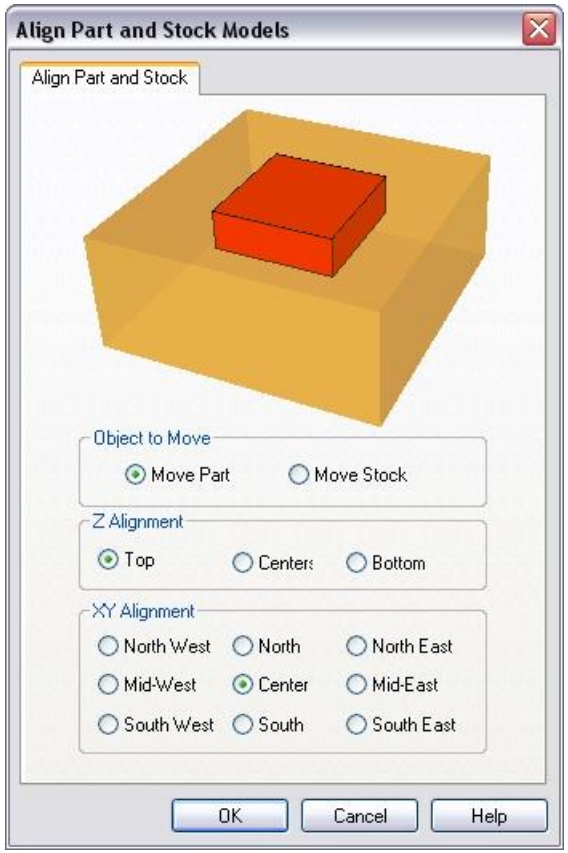
Align Part and Stock

This button is used to locate that part model within the stock model. This function is especially useful when the part model and the stock model are created without regard to their respective positional locations.

It is typical for users to need the ability to position part geometry in some geometric relationship with the stock geometry. A typical scenario is that the user is working with pre-sized stock. The user loads the stock geometry in the desired location and sets the machine zero. Now the part geometry is loaded. In such cases it would be desirable to locate the part with respect to the already positioned stock without having to go through actually calculating the transformation delta values. This dialog offers a convenient method of relative positioning based on the features of the part and stock geometry.

Once both part and stock geometry are loaded into RhinoCAM, use this dialog to perform the relative positioning. Both Z and XY alignment of different faces of the part with respect to the stock are possible. Select the necessary alignment options using the appropriate radio buttons in this dialog. The user also has the option of moving either the Part geometry or the Stock geometry to perform the alignment operation.

Notes:



The status bar on the Setup tab has the following controls

Stock Model Visibility: Turn on/off stock model

Toolpath Visibility: Turn on/off toolpath display

Z Level Display: Displays tool path by Z levels

MCSYS Visibility: Turns on/off of Machine Coordinate System display.

Notes:

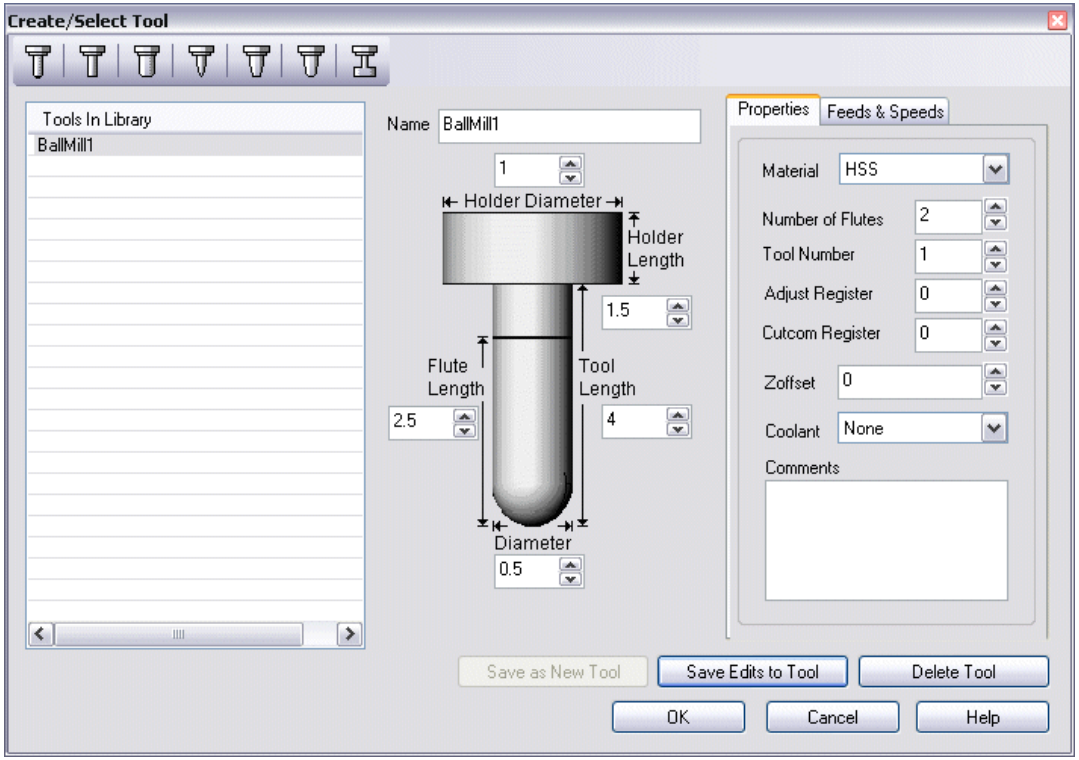
Cutting Tools Browser

Notes:

The tool manager lists all of the tools currently defined as well as the tools that are in use in machining operations. Users can edit a tool by double clicking the tool button in the browser. A tool can be deleted by selecting the tool from the Tools browser, right click cut or use the delete key from the keyboard.

Create/Edit Tools

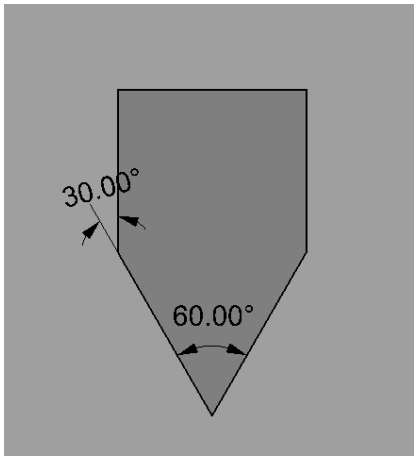
This button brings up the tool dialog that enables the creation and saving of the desired tool. All milling, drilling and user defined tools can be created here.



Notes:

The geometry definition of the tool contains edit boxes for the diameter, corner radius, taper angle, flute length and the tool length. These definitions are standard APT parameters for the tool definition. The flute length denotes the cutting length while the tool length denotes the total length of the tool to the tool holder.

Taper Angle is set for VeeMill, ChamferMill and TaperMill. This angle is the included angle. For a 60 degree taper tool, the Taper Angle is set as 30.



Adjust Register

This is used to set the Tool Length Offset (an integer). Generally this is set the same as Tool Number. The posted code would output H<#> and the # corresponds to the offset value in the controller's tool table. Note the post processor needs to be configured to output the Adjust Register

For example

N20 T1 M6

N30 G43 H1 Z0.25

Where H1 points to the controllers tool table for tool length compensation.

Cutcom Register

This is used to set the Tool Diameter Offset (an integer) for cutter compensation / tool wear compensation at the controller. Generally this is set the same as Tool Number. The posted code would output D<#> and the # corresponds to the offset value in the controller's tool table.

For example

N30 G41 X 2.0 Y 1.0 D1

Where D1 points to the controllers tool table for diameter compensation.

Note the post processor needs to be configured to output the Cutter compensation. Refer to cutter compensation for detailed description.

Z Offset

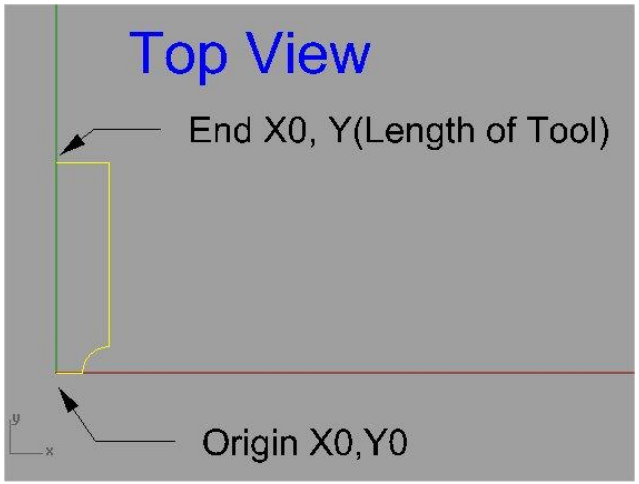
This parameter offsets the Z value in the posted g-code by the specified value. This can be set to a positive or negative value and can be an integer or decimal.

Use of Lollipop cutter and Dove Tail tool is limited to 2 ½ Axis Profiling and Engraving operations. This feature is available in the PRO configuration.

User Defined Tools

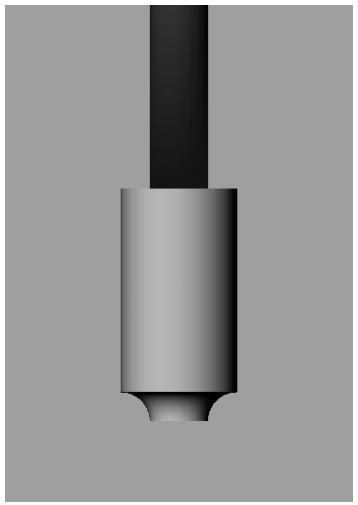
RhinoCAM allows creation of special purpose tools like form tools. These can be defined under user defined tool in the create/select tool dialog.

Steps to create a user defined tool



Notes:

1. Draw half the tool profile from the top view as shown in the picture.
2. Join the curves to form a single curve.
3. Make sure one end of the curve is at origin (0,0) and the other end at X0, Y<value>
4. Select the curve and Export selected as a Rhino (*.3dm)
5. Go to Create/Select Tool and select User Defined Tool
6. Browse to select the tool profile.
7. Specify the Maximum diameter and the tool length. Click Save as New Tool.



Notes:

Load Tool Library

The load tool library button enables the loading of a previously saved tool library. Double click on the desired file to load it into RhinoCAM. The loaded tool list will be seen under the tool button in the browser. To perform the edit, rename, cut, copy or paste operations on any of these tools, hit the right mouse button while highlighting the desired tool.

RhinoCAM supports 2 types of tool library file format *.csv and *.vkb.

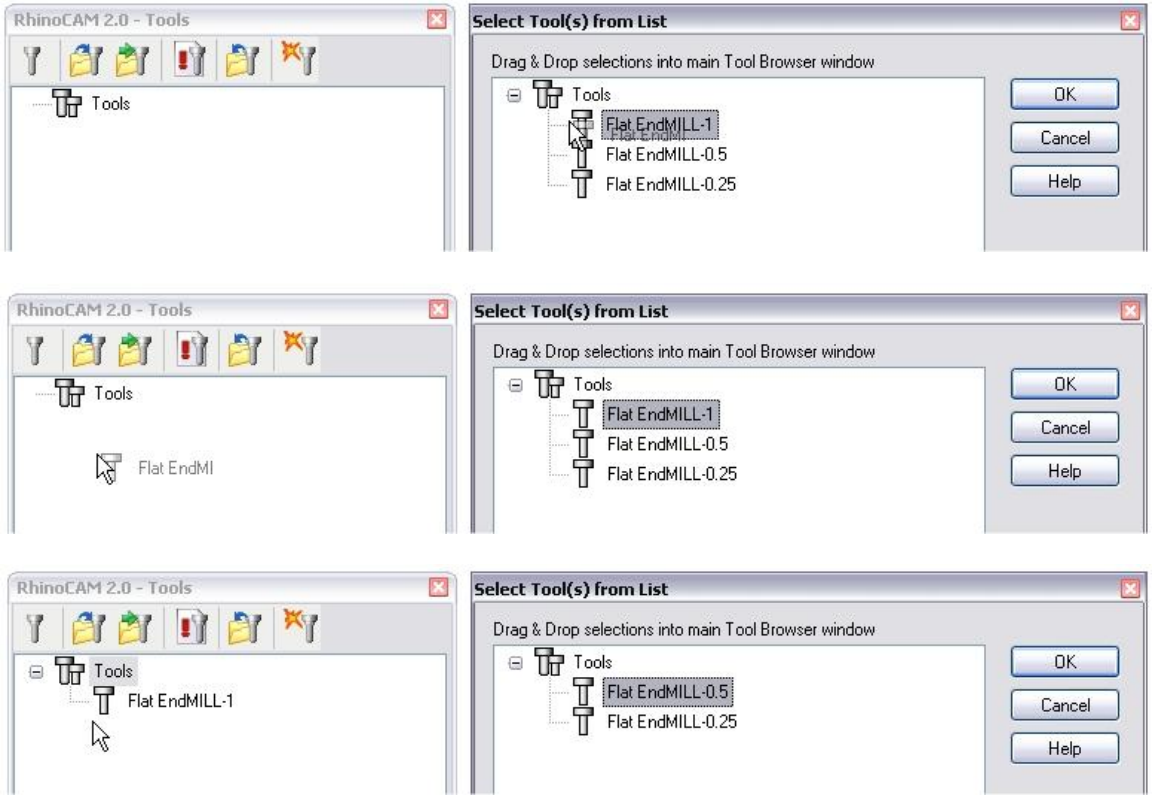
*.csv format saves and loads tools without the feeds/speeds assigned for each tool.

*.vkb format saves and loads tools with the feeds & speeds assigned for each tool.

Notes:

Select Tool from Library

The select tool library button enables users to select tools from a previously saved tool library. The list of tools will now be displayed on a pop up dialog and the user can drag and drop the tools from the selecting list to the cutting tools browser. To perform the edit, rename, cut, copy or paste operations on any of these tools, hit the right mouse button while highlighting the desired tool.



List Tools

The button brings up all the tool properties associated with the tools currently recorded in the current RhinoCAM session.

Save Tool Library

This button enables the created tools to be saved in a tool library file. The file can be saved in the desired directory and read in when required.

Selecting Machining Features / Regions

Select Curves as Regions

This feature allows the user to select curve as limiting region and the toolpath would stay on to the selected region.

Criteria for selecting Curves as regions

- Open and closed curves (Lines, Polylines, Arcs, Circles, Polycurves) can be selected as regions for 2 ½ axis Profiling, Engraving and Chamfering operations.
- Only closed curves (Polylines, Circles, Polycurves) can be selected as regions for 2 ½ Axis Facing, Pocketing, V-Carve Roughing, V-Carving, and all 3 axis operations.
- Points and circles can be selected for Hole Making (Drilling, Tapping, Boring and Reverse Boring), Hole Pocketing and Thread Milling operations.
- There is no limit on the number of curves that can be selected as region.
- Self-intersecting curves cannot be selected as regions.

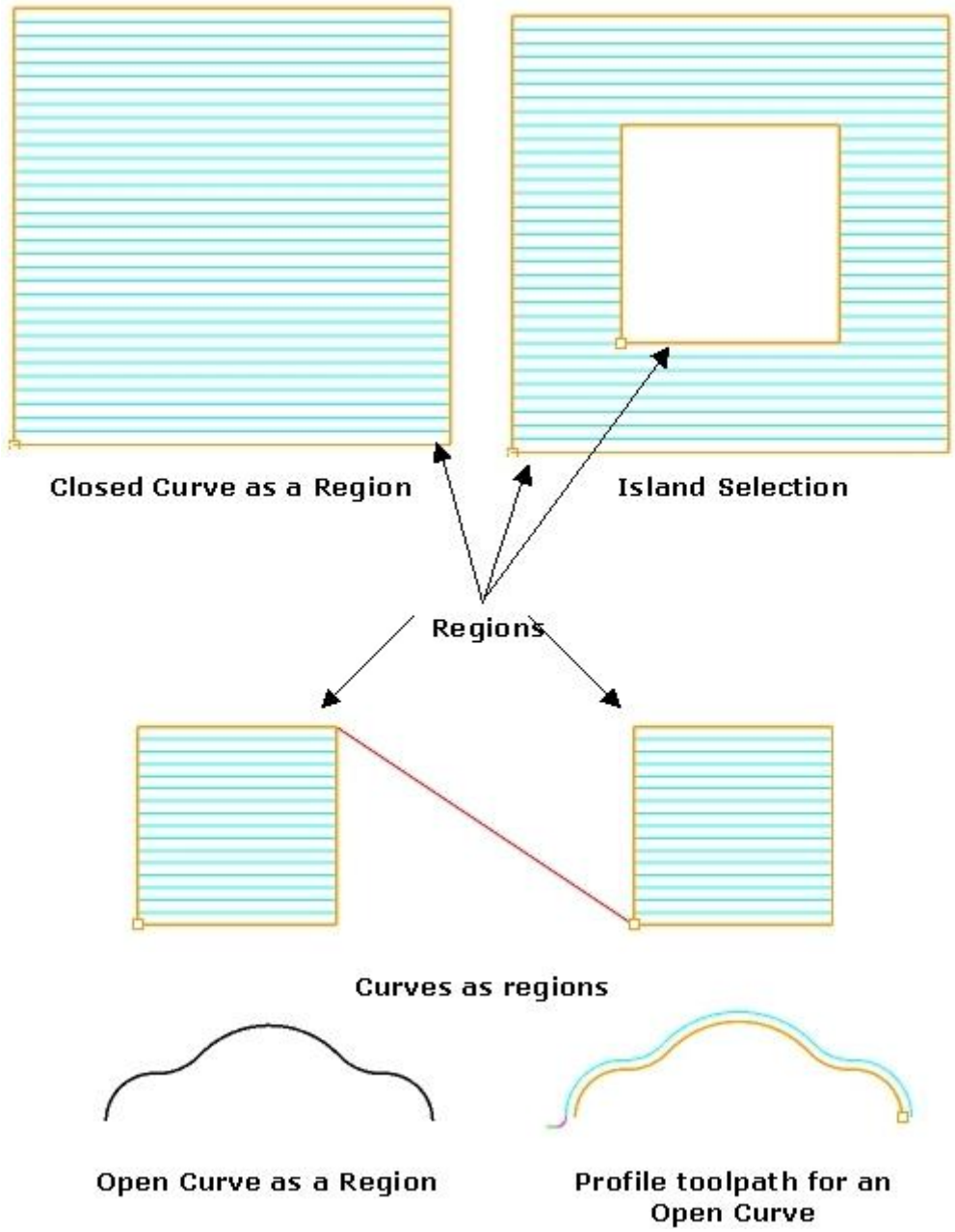
Notes:

- A curve inside another would be treated as an island for toolpath computation.

Open Loops Found - Selecting a region that is not a closed a curve would result in open loops. To resolve this error the selected curve must be chained / joined to form a closed curve either by adding a line segment or by joining two or more curves.

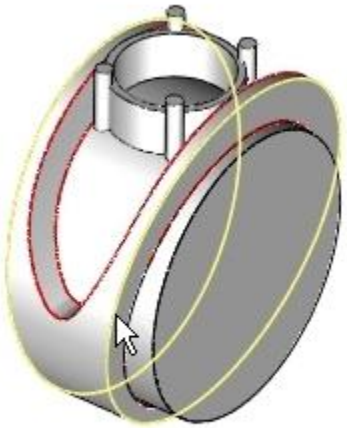
In the picture below, the toolpath is restricted to the selected region(s)

Notes:

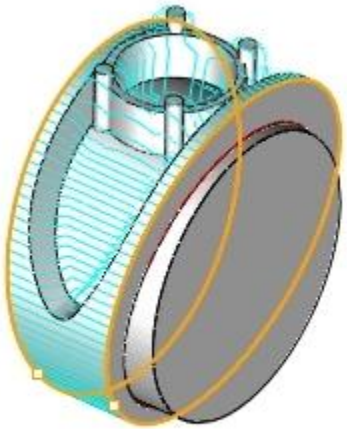

Closed Curve as a Region
Island Selection
Regions
Curves as regions
Open Curve as a Region
Profile toolpath for an Open Curve

Notes:

Notes:



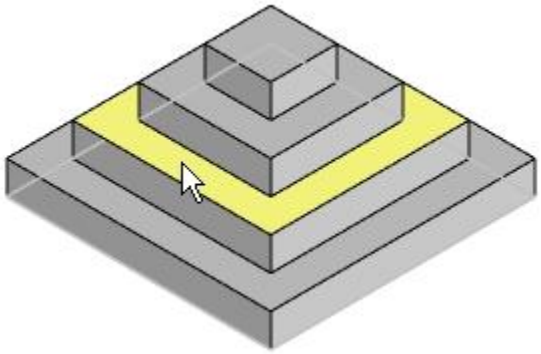
Selecting closed curves as regions for 4axis Machining Operations



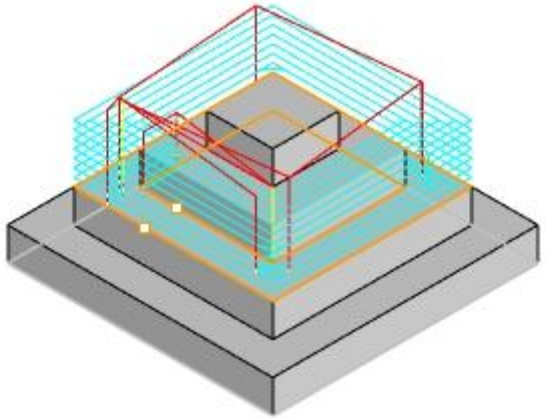
Generated toolpath for the selected regions

Select Flat Area Feature

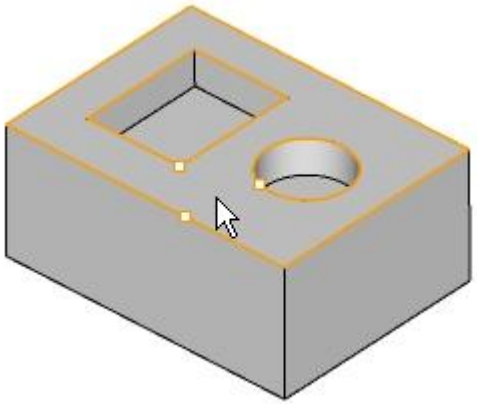
This feature allows the user to select flat areas as limiting regions. Examples of flat area features are shown below.



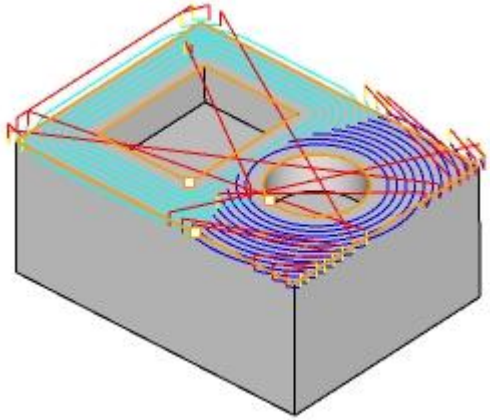
Selecting a Flat area Feature for Roughing



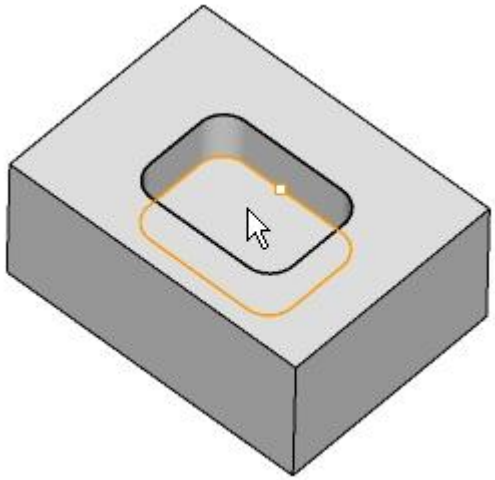
Roughing Toolpath generated for the Flat Area Feature



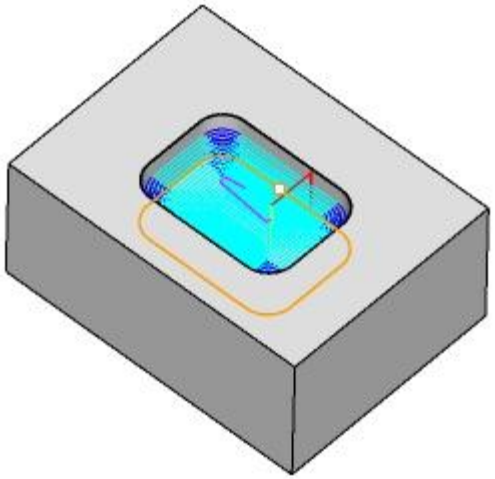
Selecting a Flat area Feature for Facing



Facing Toolpath generated for the Flat Area Feature



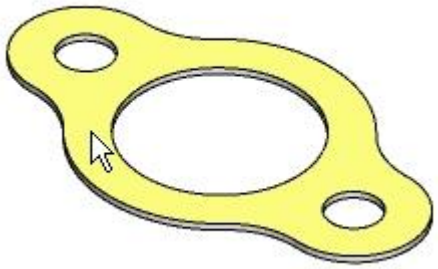
Selecting a Flat area Feature for Pocketing



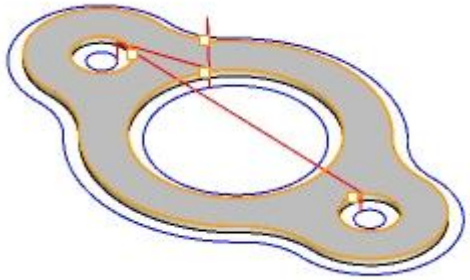
Pocketing Toolpath generated for the Flat Area Feature

Notes:

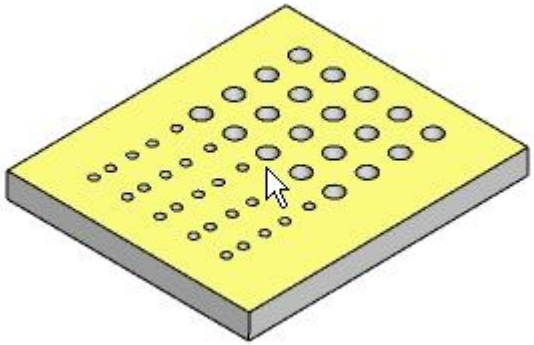
Notes:



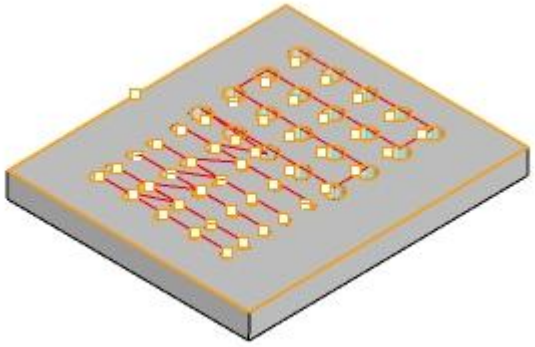
Selecting a Flat area Feature for Profiling



Profiling Toolpath generated for the Flat Area Feature



Selecting a Flat area Feature for Hole Making (Drilling, Tapping, Bore & Rev Bore)



Drilling Toolpath for the Flat Area Feature

#	Selected Machining Region(s)
1	Region 1
2	Region 2
3	Region 3

Remove All – Removes all the Selected Curves and Flat Area from the Selected Machining Region(s) list.

Remove Active – This allows the user to remove a region from the selection list. The user needs to select a region from the list of Selected Machining Region(s) and click Remove Active.

Cutter Compensation

Notes:

Cutter compensation is used typically to compensate for the difference in the dimensions of the actual cutter used in machining and the cutter used for programming in RhinoCAM. For example, if the cutter used in programming in RhinoCAM is 0.25 inches and due to tool wear the actual cutter is only 0.24 inches in size, the user can compensate for this in the controller rather than having to re-program the operation in RhinoCAM again.

In order to do this the user needs to do the following:

1) Turn cutter compensation on in the Operation Set Compensation to Auto/ON or Control/ON.

2) Specify the cutter compensation value and the compensation register in the controller (the controller needs to be capable of doing this)

A few things to watch out for:

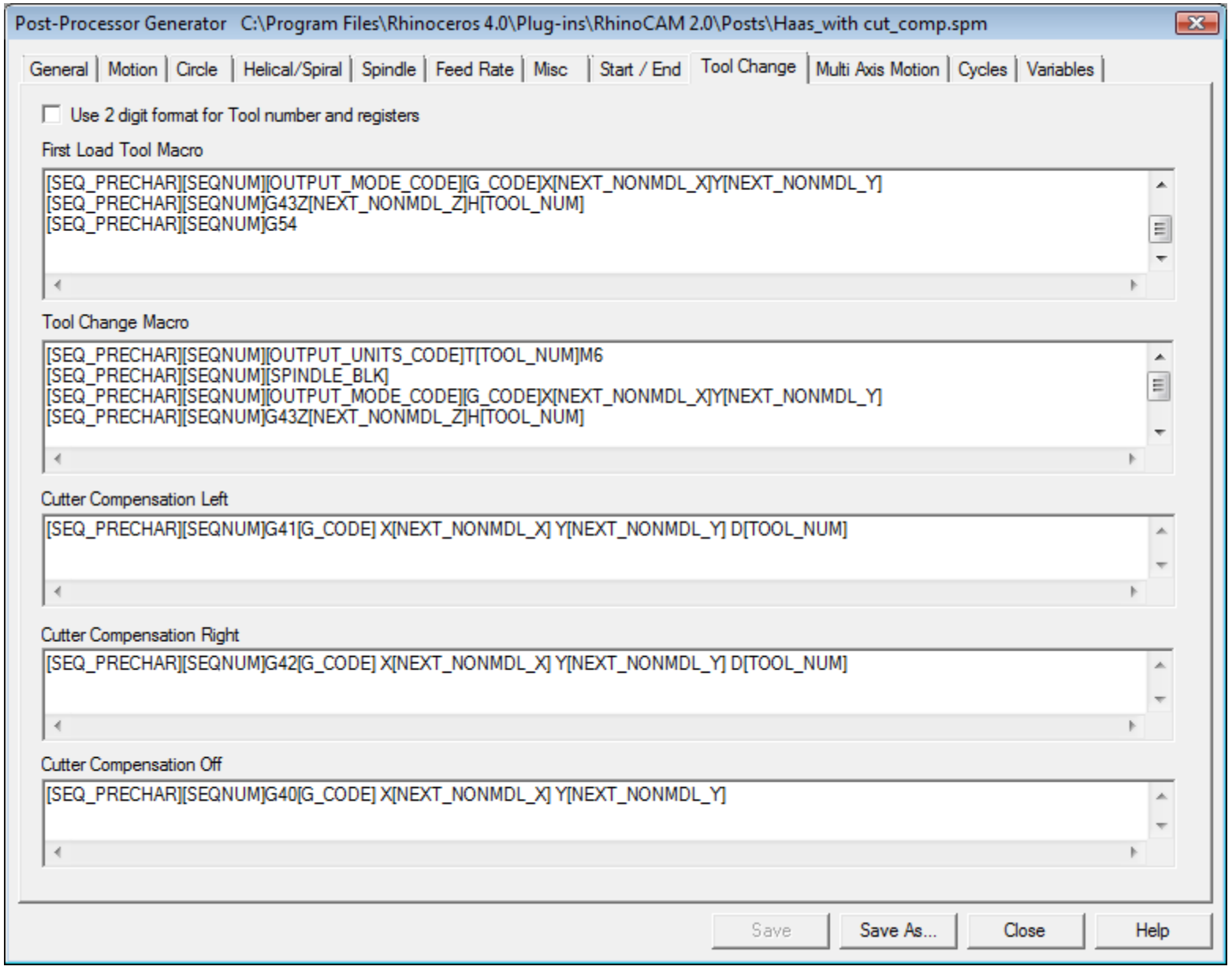
1) Cutter compensation makes sense only in 2-1/2 axis operations. If you are using roughing (pocketing & facing) the compensation will be turned on only in the final passes.

2) Make sure you are not using Zig-Zag cut traversal in any of the methods that you want to turn compensation on.

3) Make sure you have a linear motion for the controller to turn on the compensation value on. If your first motion is an arc the controller will not be able to turn on the compensation. Thus, in 2-1/2 axis profiling, make sure there is a linear entry motion for the controller to be able to turn compensation on.

Note: Setting the Compensation to Auto/ON or Control/ON has the same behavior in RhinoCAM.

Select the Post Processor from the Setup tab in RhinoCAM browser by selecting Utilities and Post Processor generator.


Post-Processor Generator C:\Program Files\Rhinoceros 4.0\Plug-ins\RhinoCAM 2.0\Posts\Haas_with cut_comp.spm
General
Motion
Circle
Helical/Spiral
Spindle
Feed Rate
Misc
Start / End
Tool Change
Multi Axis Motion
Cycles
Variables
Use 2 digit format for Tool number and registers
First Load Tool Macro
[SEQ_PRECHAR][SEQNUM][OUTPUT_MODE_CODE][G_CODE]X[NEXT_NONMDL_X]Y[NEXT_NONMDL_Y]
[SEQ_PRECHAR][SEQNUM]G43Z[NEXT_NONMDL_Z]H[TOOL_NUM]
[SEQ_PRECHAR][SEQNUM]G54
Tool Change Macro
[SEQ_PRECHAR][SEQNUM][OUTPUT_UNITS_CODE]T[TOOL_NUM]M6
[SEQ_PRECHAR][SEQNUM][SPINDLE_BLK]
[SEQ_PRECHAR][SEQNUM][OUTPUT_MODE_CODE][G_CODE]X[NEXT_NONMDL_X]Y[NEXT_NONMDL_Y]
[SEQ_PRECHAR][SEQNUM]G43Z[NEXT_NONMDL_Z]H[TOOL_NUM]
Cutter Compensation Left
[SEQ_PRECHAR][SEQNUM]G41[G_CODE] X[NEXT_NONMDL_X] Y[NEXT_NONMDL_Y] D[TOOL_NUM]
Cutter Compensation Right
[SEQ_PRECHAR][SEQNUM]G42[G_CODE] X[NEXT_NONMDL_X] Y[NEXT_NONMDL_Y] D[TOOL_NUM]
Cutter Compensation Off
[SEQ_PRECHAR][SEQNUM]G40[G_CODE] X[NEXT_NONMDL_X] Y[NEXT_NONMDL_Y]
Save
Save As...
Close
Help

Notes:

Chapter 4

Notes:

Create 2 ½ Axis Toolpaths

Profiling

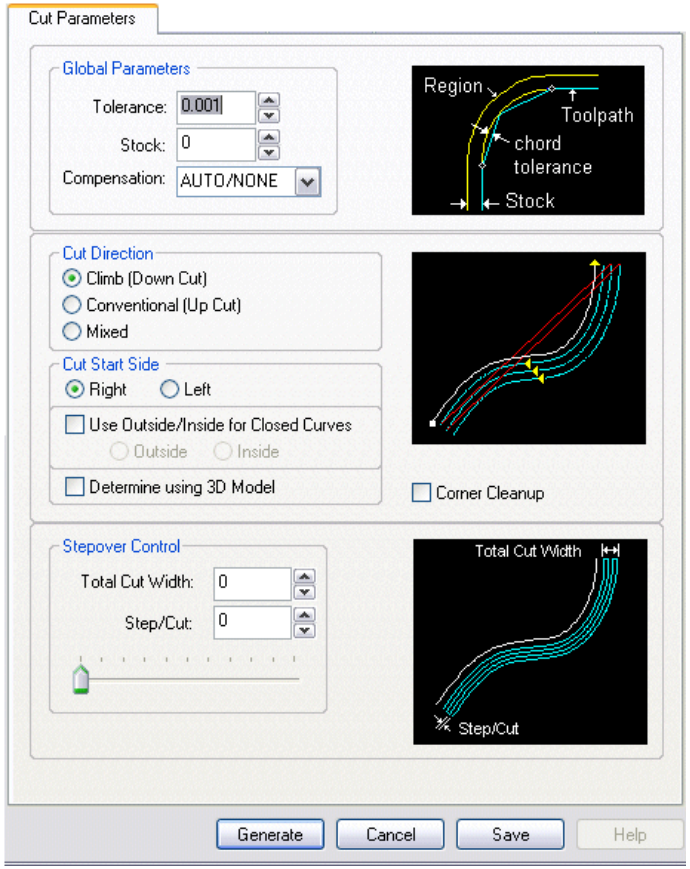
This method machines open and closed regions (curves) by tracing along one side of their contours. You can define offsets so that the tool makes multiple passes relative to the regions. Profiling can be used as a finishing operation after a **Pocketing** or **Facing** toolpath, or it can be used alone.

These regions are treated as the tops of vertical walls spanning from the Z values of the regions down to the user specified minimum Z.

Cut Parameters

The user has the ability to set the Global Cut Parameters and the Cut Direction via this property page of the dialog.

The **Global Cut Parameters** section allows the user to set the tolerance value to be used in machining. Any uniform thickness or stock to be left of the part can be specified here.



Tolerance is the allowable deviations from the actual part geometry plus the Stock layer (if any).

Stock is the thickness of the layer that will remain on the side of the part after the toolpath is complete. Roughing operations generally leave a thin layer of stock, but for finishing operations this value is zero.

The **Cut Direction** allows the user to specify the Climb or Conventional or Mixed.

The Cut Start Side determines the side of the curve to be cut. For example, for a clockwise circle, the right side would be inside the circle and the left side would be outside the circle. If a 3d model is in use, the "Determine using the 3d model" box can be checked to automatically determine the start side.

The **Stepover Contro**l section allows the user to define the spacing between the cuts. The total width to be cut can be specified along with the step size for each cut by entering the value or using the sliding bar.

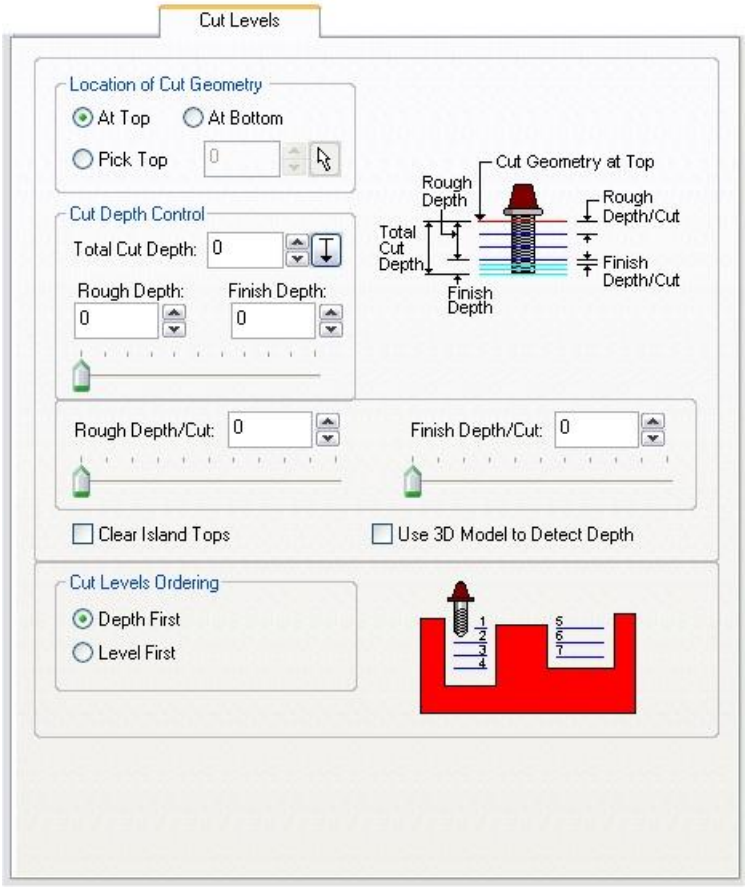
Notes:

Cut Levels

Notes:

The **Location of Cut Geometry** can be set to the top Z level or the bottom Z level by picking the option desired. Optionally the user could choose to enter location, a certain distance from the top either by typing in the value or by selecting the pick button and selecting the point on the part.

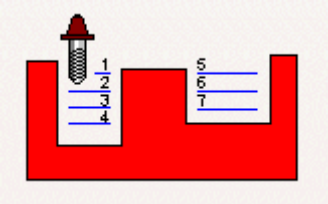
The **Cut Depth Control** enables the user to define the depth of the cut. You can define a Rough Depth and a Fish Depth by either typing in the desired values or by using the slider bar provided below. Similarly the user can define the depth per cut for both the Roughing and the Finishing paths.



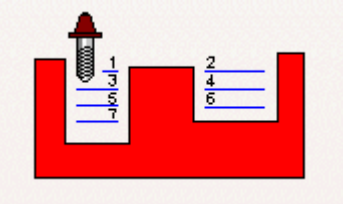
Checking the **Clear Island Tops** will add a cut at the island levels (inner regions).

3d Model to Detect Depth should be used only when there is a 3d model is being used. This option when checked will detect the bottom most level of the pocket (if there are multiple depths) and will add toolpaths until that last level. Only the finish depth needs to be specified. The roughing depth is automatically determined by the difference between the total depth and the finishing depth.

Cut Level Ordering can be set to either Depth first or Level first as shown below



Depth First



Level First

Entry/Exit

The user has the ability to set the tool Entry and Exit parameters using the following property page dialog.

The **Entry motion** consists of **Approach Motion** and an **Engage Motion**. The user can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion. Similarly the **Exit motion** consists of a **Retract motion** followed by a **Departure motion**. The Retract motion can be either a radial (arc) motion or a linear motion at an angle. The departure motion is a linear motion.

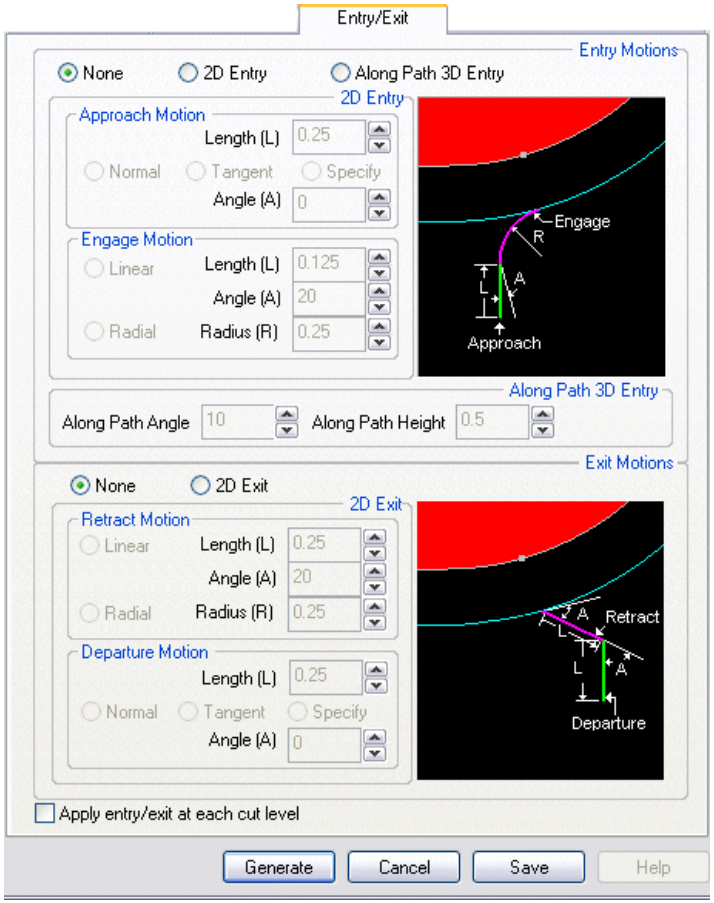
The user can choose to define the 2D Entry and Exit path of the tool. Depending on the stock material and cut pattern the Approach/Departure Motion can be either normal, tangential or at a specified angle with respect to the stock.

The Engage/Retract Motion can be set to Linear or a Radial. In the Linear Motion the cutter follows a linear ramp motion, ramping back and forth from a user specified height to the engage point. The length of this move, as well as the angle of this motion can be specified by the user.

The Entry path can also be defined as a 3D Entry along a specified path angle with a path height.

Notes:

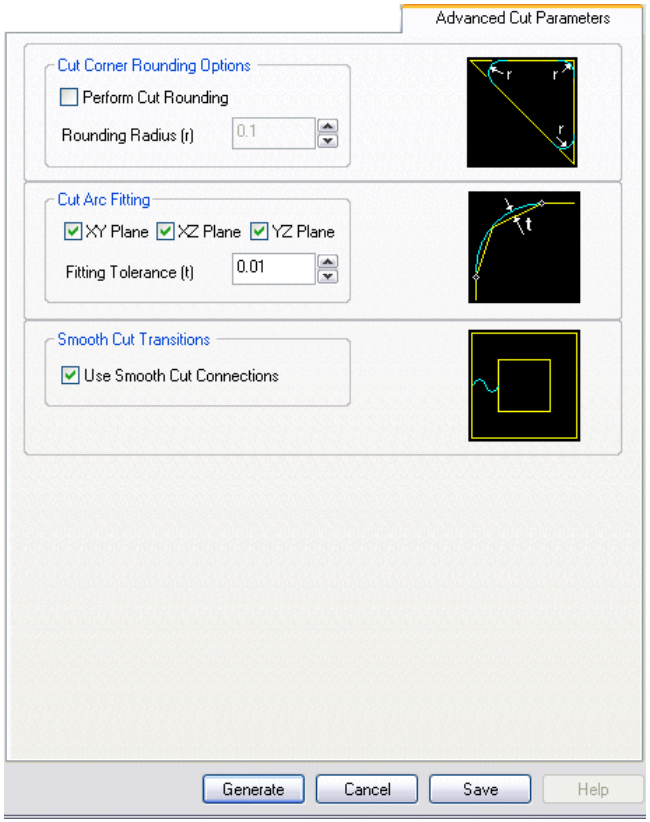
Notes:



The Entry/Exit Motions can be set for each cut level by checking the box at the bottom of the screen.

Advanced Cut Parameters

The advanced cut parameters page is used to control the cuts for high speed machining. All of the options in this section are designed to reduce rapid acceleration and deceleration of the machine during the cutting process. These parameters allow smoothing of the toolpaths by introduction of arcs. The user can use these parameters even if the controller does not support arcs. In this case, make sure that the output is set to linear output. This can be set in the Machining Preferences dialog in located in the RhinoCAM MOps in the Set Up tab from the Preferences icon.



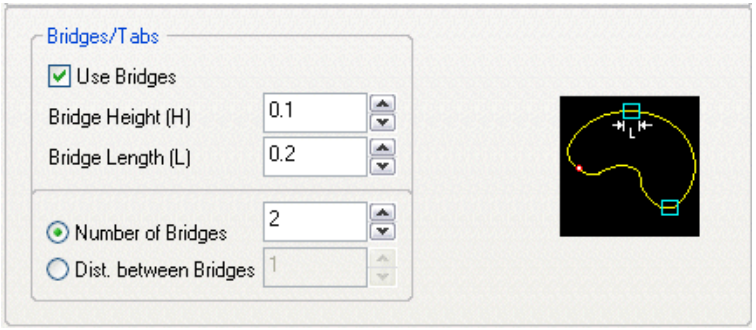
Cut Corner Rounding Options: This option is used to round sharp corners in the toolpath. The user can specify a rounding radius; fillets of the specified radius will be introduced in sharp corners if possible. These fillets will only be introduced in planes parallel to the XY plane.

Cut Arc Fitting: This option can be used to fit arcs to the toolpath. Arc fitting can be accomplished on planes parallel to the XY, XZ, and the YZ planes. The user specifies an arc fitting tolerance and the system attempts to fit arcs to the computed toolpaths. Fitting arcs to toolpaths serves to make the toolpath smoother as well as reducing toolpath size.

Smooth Cut Transitions: This option can be used to introduce S shaped or C shaped cut transitions between two successive offset cuts. These transitions are introduced only in offsets that are generated in planes parallel to the XY plane. These transitions allow the cutter to transition from one cut to the next in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.

Notes:

A very useful setting added to this dialog for the Profiling operation is the use of Brides or tabs. The user can define the use of bridges along the part boundary. These tabs are used to hold the part on the table during the actual machining operation. The dimensions of the bridges, their number and the distance between them can be defined here as seen below.



Sorting

Allows the user to specify a sorting method.

No Sort – Does not use a sorting method and uses the order in which the geometries are created or order in which the machining regions were selected.

Minimum Distance Sort – Sorts based on minimum distance of start points between 2 or more geometries. The start point can be set to Lower Left, Lower Right, Upper Left or Upper Right.

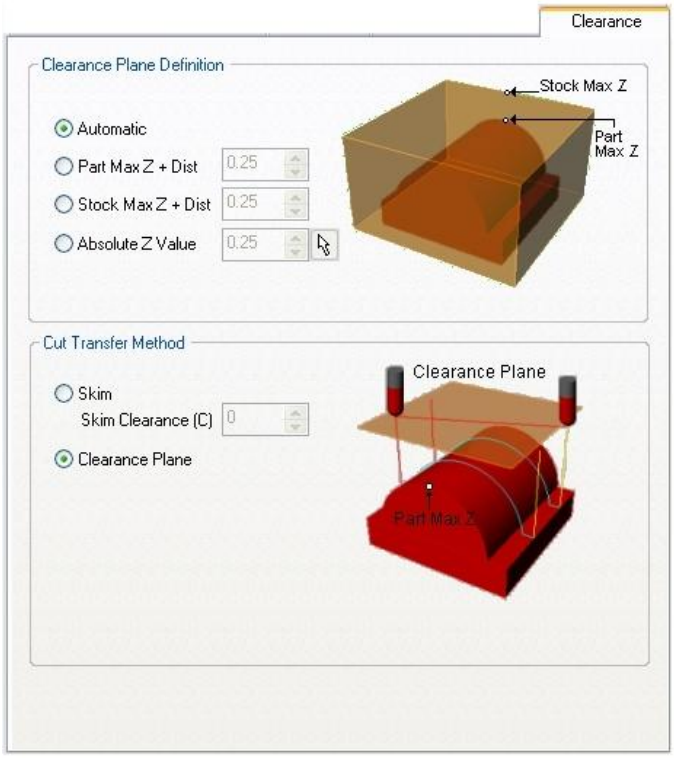
Directional Sort - The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort direction is defined by the Start angle. The Secondary Sort Direction is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or Zig Zag (two ways).

Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. Typically the user would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Notes:

Cut Transfer - The user can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can either be instructed to move to the clearance plane and then perform the transfer motion to the next cut location or it could do a skim motion. In the skim motion, the system automatically determines the safe height by taking into consideration the condition of the part and stock model and using this Z value as the height to perform the transfer motions.



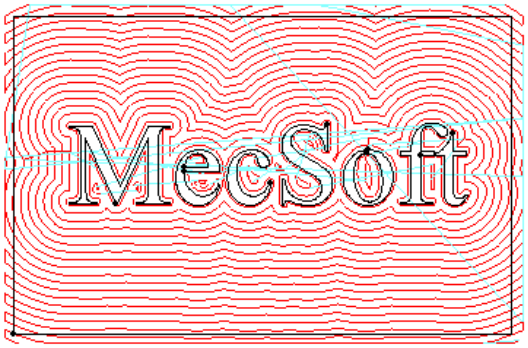
Class Room Exercise – Open Gasket.3dm and create profiling operation.

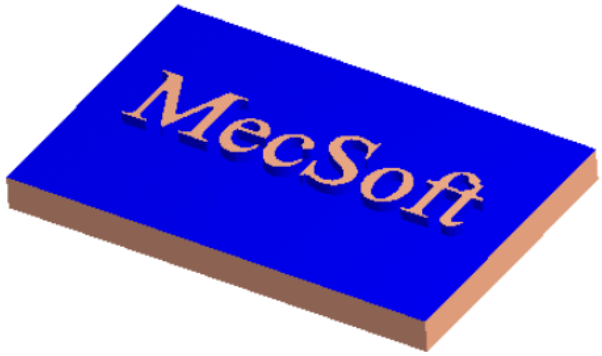
Notes:

Facing

Notes:

Facing is a method of generating planar toolpaths using regions as the part geometry limits. Here the 3-D part geometry model is not considered in the toolpath generation. Here the outermost region is used to construct a virtual 3-D stock geometry model that has the same shape as the region and has vertical walls spanning from the region down to the bottom Z level. All other the input regions are considered to belong to part geometry. The toolpath begins at the top Z value and stops at the bottom Z depth defined by the user. The cut patterns that used in this type of machining can either be a linear cut pattern or an offset cut pattern.





Cut Parameters

The **Global Parameter**s section allows the user to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can be specified here.

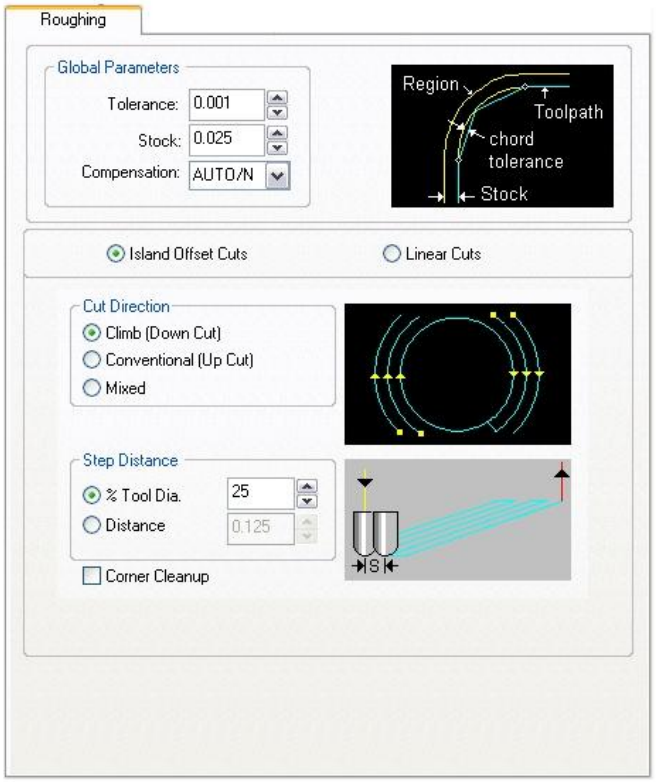
Tolerance is the allowable deviation from the actual part geometry plus the Stock layer (if any).

Stock is the layer of material that will remain around the part after the toolpath is completed. Generally Roughing operations leave a thin layer of stock, unlike finishing operations where this value is usually set to zero.

Compensation stands for cutter compensation. The user can turn this on by selecting from the drop down menu. The cutter compensation direction, left or right, is determined by the Cut Direction defined (Climb or Conventional).

Cut Pattern defines the type of path that the tool will follow at each Z level. The user can choose a linear cut pattern where the tool will always traverse in linear cuts or an Island Offset cut pattern where the tool will traverse in successive uniform offsets of the part shape.

Notes:



Cut Direction can be specified as Climb, Conventional or Mixed.

The **Step Distance** section allows the user to define the spacing between the cuts. The spacing can be specified either as a percentage of tool diameter or at a specified distance.

Checking the **Corner Cleanup** parameter will cause automatic detection of all the corners that the tool could not reach between each pass. It will then add a toolpath based on the uncut area detected; either a linear cut in case of smaller areas or a cut that travels along the shape of the uncut area, when the area is large.

Cut Levels

This is similar to profiling allows the user to set the Location of Cut Geometry and Cut Depths.

The **Cut Depth Control** enables the user to define the depth of the cut. You can define a Rough Depth and a Fish Depth by either typing in the desired values or by using the slider bar provided below. Similarly the user can define the depth per cut for both the Roughing and the Finishing paths.

Checking the **Clear Island Tops** will add a cut at the island levels (inner regions).

Note: Entry/Exit, Advanced Cut Parameters and Clearance Planes are set similar to Profiling operation.

Pocketing

Pocket machining is a method of generating planar toolpaths using regions as the part geometry limits. Here the 3-D part geometry model is not considered in the toolpath generation. Instead the input regions are used to construct a virtual 3-D part geometry model that has vertical walls with the top edge defined by the regions Z values or a user specified value and has an XY shape defined by the regions geometry. The toolpath begins at the top Z value and stops at the bottom Z depth defined by the user. The cut patterns that used in this type of machining can either be an offset, linear, radial or a spiral cut pattern.

Cut Parameters

The user can set the Global Cut Parameters, the Cut Pattern and the Step Over Control via this property page.

The Cut Parameters section allows the user to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can be specified here.

Tolerance is the allowable deviation from the actual part geometry plus the Stock layer (if any).

Notes:

Stock is the layer of material that will remain around the part after the toolpath is completed. Generally Roughing operations leave a thin layer of stock, unlike finishing operations where this value is generally set to zero.

Compensation stands for cutter compensation. The user can turn this on by selecting from the drop down menu. The cutter compensation direction, left or right, is determined by the Cut Direction defined (Climb or Conventional).

The **Cut Pattern** section allows the user to define the type of cut pattern that the tool will follow when it is at each Z level. Currently the user can choose from the following cut patterns

- **Offset** cut pattern where the tool will traverse in successive uniform offsets of the part shape,
- **Linear** cut pattern where the tool will traverse in a linear cuts always,
- **Spiral** cut pattern which can go from inside out or outside in,
- **Radial** cut pattern.

The **Cut Direction** can be specified as Climb, Conventional or Mixed.

Start Points for the Offset, Spiral and Radial cuts can be set to the inside or the outside. In case of the Linear cuts the user could choose to start at the bottom right corner or at the top left corner.

The **Step Distance** section allows the user to define the spacing between the cuts. The spacing can be specified either as a percentage of tool diameter or at a specified distance.

Checking the **Corner Cleanup** parameter will cause automatic detection of all the corners that the tool could not reach between each pass. It will then add a toolpath based on the uncut area detected; either a linear cut in case of smaller areas or a cut that travels along the shape of the uncut area, when the area is large.

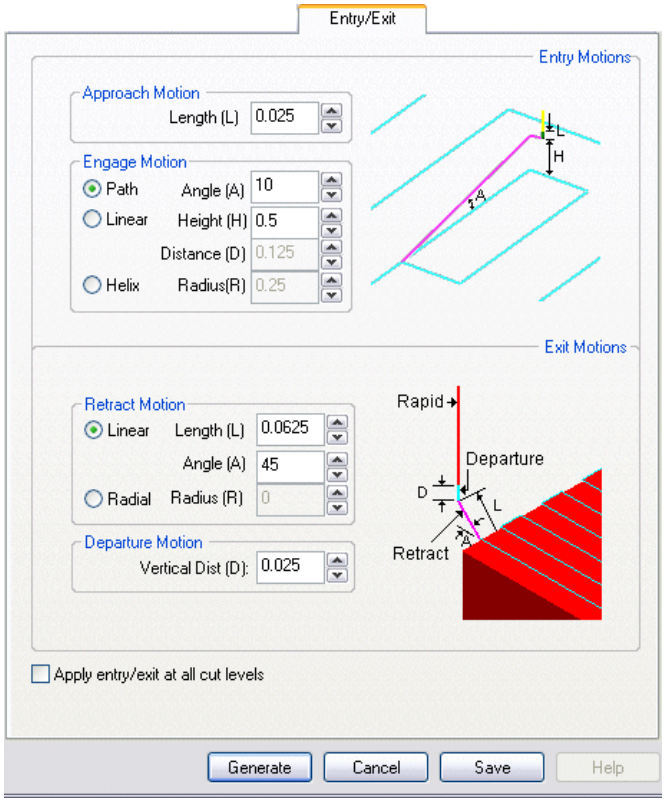
Entry / Exit

The **Entry motion** consists of Approach and Engage motions. The user can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

The **Engage Motion** parameters allow the user to define how the cutter would engage into material when forced into such a situation. This can happen when machining a cavity or pocket. The user has the option of ramping the cutter or engaging vertically down. The Ramp Engage motion will typically be used when machining with flat or corner-radius end mills.

Notes:

Notes:



In the **ramp** option, the cutter can ramp in one of the following 3 ways:

Path: In this method the cutter follows the contour of the part in a ramping motion on the outside until it hits the cutting start point. The user can control the angle of descent and the length of this engage motion by specifying these parameters.

Linear: Here the cutter follows a linear ramp motion, ramping back and forth from a user specified height to the engage point. The length of this move, as well as the angle of this motion can be specified y the user.

Helix: Here the cutter follows a helix as it descends from a user defined height to the first cut point. The angle of the helix as well as the radius of the helix can be specified by the user.

The **Exit motion** consists of a Retract motion followed by a Departure motion. The retract motion can be either a radial (arc) motion or a linear motion at an angle. The departure motion is a linear motion.

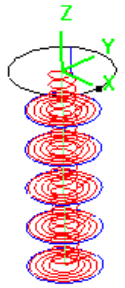
Note: Cut Levels, Advanced Cut Parameters and Clearance Planes are set similar to Facing & Profiling operations.

Class Room Exercise – Open Shaft Base.3dm and create Facing and Pocketing operations.

Notes:

Hole Pocketing

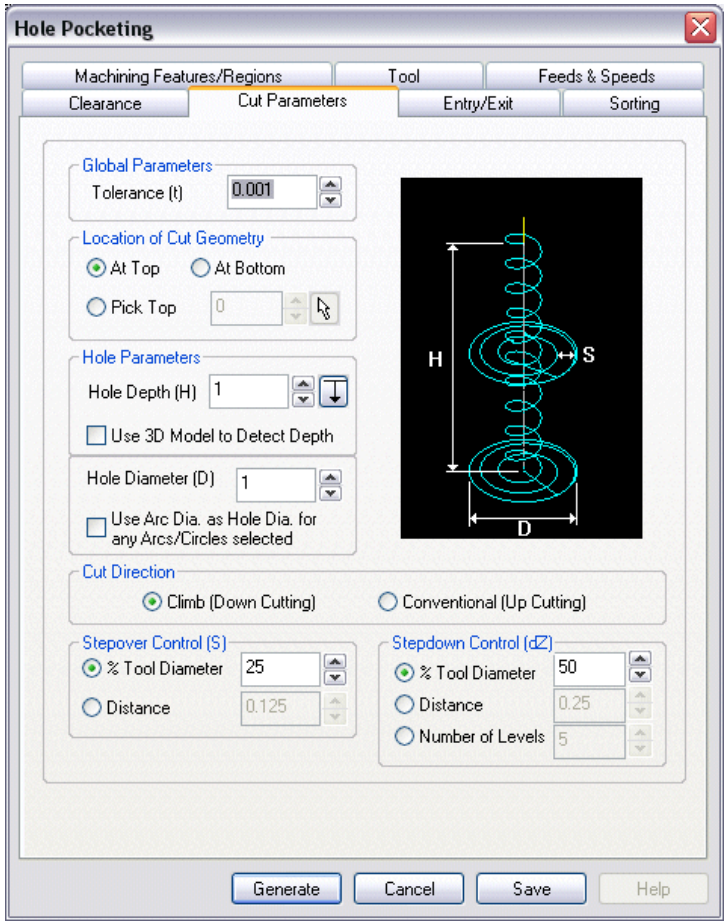
This method is used to cut large holes as a milling operation, rather than drilling. Engage can be specified as a helix with height and angle or pitch. Output can be a helix cycle (for machines with this capability) or a series of linear moves. After the helix engage, the hole is cut to the outer diameter using a spiral motion, followed by a circular motion to clean up the hole.



Cut Parameters

Tolerance is the allowable deviations from the actual part geometry. **Cut Geometry Location** can be either manually picked or set to either the top or the bottom of the hole.

The **Hole Depth** and the **Diameter** can be specified for the Hole using the corresponding edit boxes in the dialog box. The user can also choose to have the depth and the diameter automatically selected by the program using the part geometry.



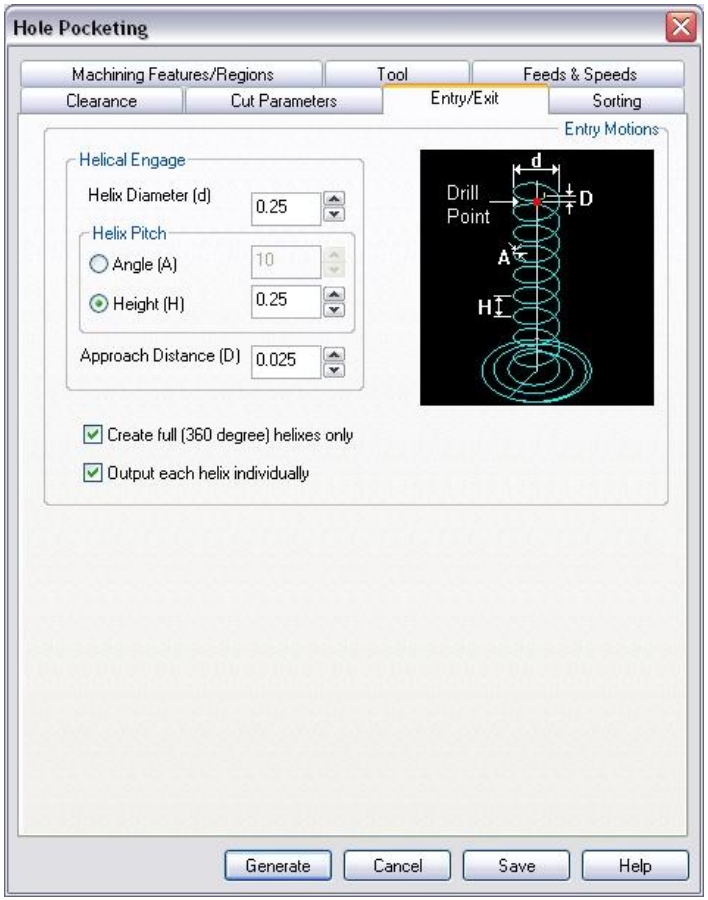
The **Cut Direction** allows the user to specify the Climb direction which cuts down or Conventional direction which cuts up.

The **Stepover Control** section allows the user to define the spacing between the cuts. The step over can be a distance or a percentage of the tool diameter.

The **Stepdown Control** defines the spacing at different levels of the cut, similarly defined as specified distance, a percentage of the tool diameter or set number of levels.

Entry / Exit

Notes:



The cutter approaches and engages and the type of transfer motions during the cutting cycle can be specified here. The engage can be a helical engage, which is achieved by specifying the helix diameter and the Helix pitch as an angle or height.

The helix diameter must be less than the difference of the hole diameter to the tool diameter.

Selecting Create full helixes only – converts a partial helix to a full helix. When this option is checked the user can choose to split the helixes into individual helix of 360 each in the posted code.

Note: Sorting and Clearance plane Settings are set similar to Profiling Operation.

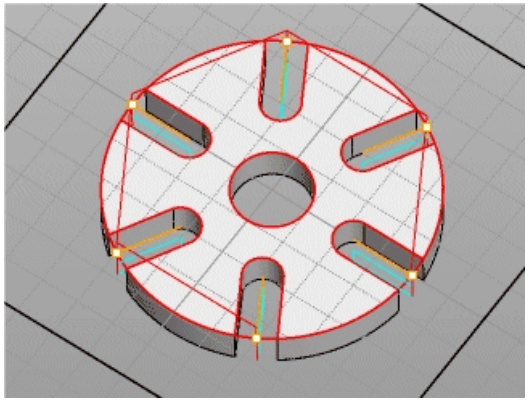
Class Room Exercise – Open Shaft Base.3dm and create Hole Pocketing Operation.

Notes:

Notes:

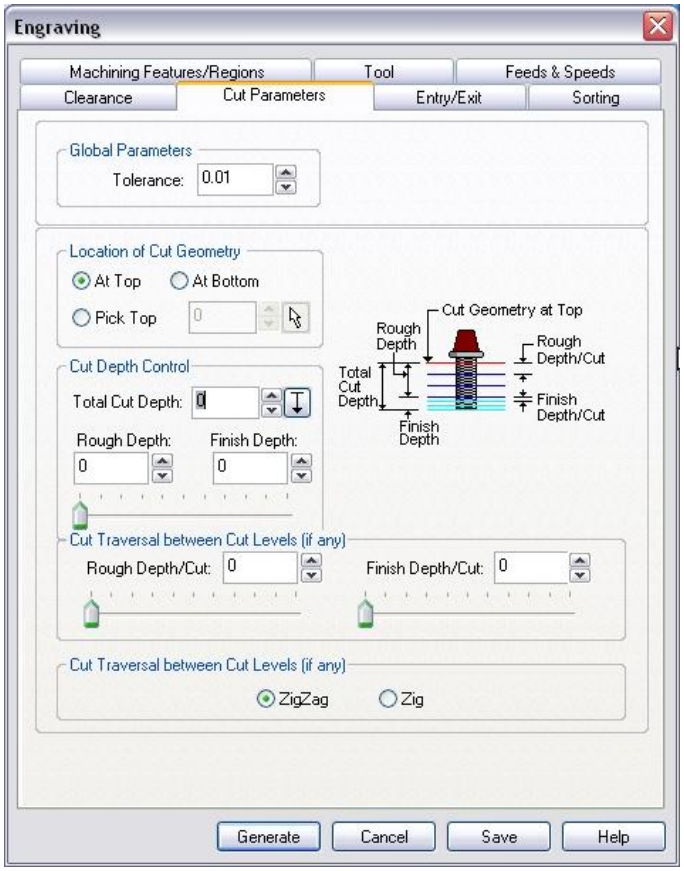
Engraving

Engraving allows the user to select open or closed regions to engrave. In addition to 2D regions, 3D regions can also be chosen. Multiple depths can also be specified for better control of the engraving operation. This method is especially suited for engraving text and logos to part geometry. Unlike the 3 Axis Curve Machining method, the cutter is not projected to the surfaces below. It merely follows the specified regions.



Cut Parameters

The **Location of Cut Geometry** can be set to the top Z level or the bottom Z level by picking the option desired. Optionally the user could choose to enter location, a certain distance from the top either by typing in the value or by selecting the pick button and selecting the point on the part.



The **Cut Depth Control** enables the user to define the depth of the cut. You can define a Rough Depth and a Fish Depth by either typing in the desired values or by using the slider bar provided below. Similarly the user can define the **Depth per Cut** for both the Roughing and the Finishing paths.

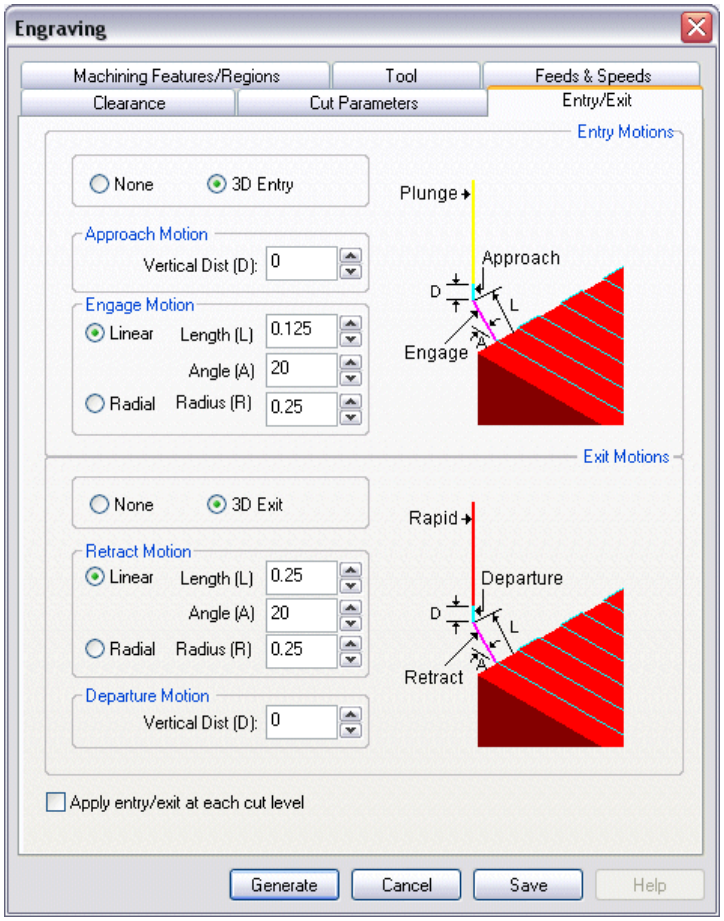
Cut Traversal between Cut Levels can be set to Zig or ZigZag. Setting the cut traversal to Zig generates cuts along the direction of the curve, retracts the tool to cut transfer plane and starts at the same start location for the next level. Setting the cut traversal to ZigZag cuts both directions.

Entry/Exit

When the user selects this tab in the dialog the following property page is displayed. The user has the ability to set the Entry/Exit parameters via this dialog.

Notes:

The Cut Entry section can be used to control how the cutter enters during the cutting process. The user can choose to approach vertically and specify a vertical approach or retract by choosing the **None** option. Alternatively, a **3D Entry** can be specified for the Engage and Retract motions.



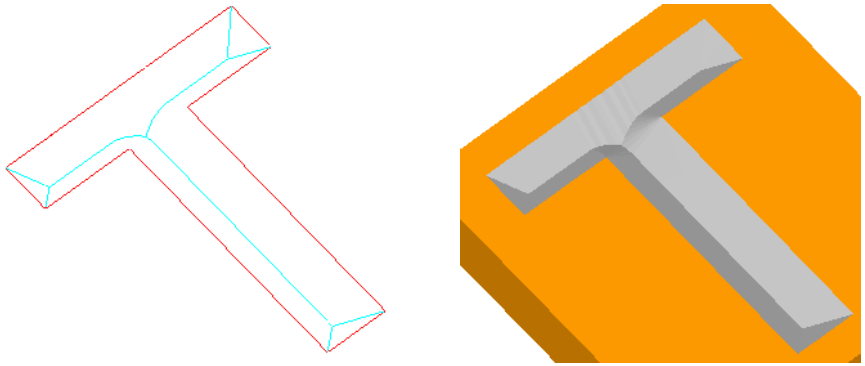
Note: Clearance plane settings are set similar to Profiling Operation.

Class Room Exercise – Open Gear.3dm and create Engraving Operation.

Notes:

V-Carving

This is used to create cutter paths with a V-cutter (tapered cutter) that is especially suitable for lettering and sign making.



The unique feature of this type of toolpath is that the V-cutter creates a chiseled look at the corners by moving the cutter tangent to both converging sides of the corner.

A typical V-carve finishing toolpath is shown above. Here the inside of the letter T is being machined with a V cutter. Notice that the cutter path at the corners of the geometry actually is coincident with the geometry, while further away from the corner it moves down in Z. This feature of the toolpath is what causes the cutter to create a chiseled look at these corners as the simulation picture illustrates.

Cut Parameters

The V-Carving cutter can be set to cut inside the curve or trace the outside of the curve. The **Location of Cut Geometry** can be set to the top Z level or the bottom Z level by picking the option desired. Optionally the user could choose to enter location, a certain distance from the top either by typing in the value or by selecting the pick button and selecting the point on the part.

The **Cut Depth Control** enables the user to define the depth of the cut. You can define a Rough Depth and a Fish Depth by either typing in the desired values or by using the slider bar provided below. Similarly the user can define the **Depth per Cut** for both the Roughing and the Finishing paths.

Note: Clearance Plane & Sorting parameters are set similar to Profiling operation.

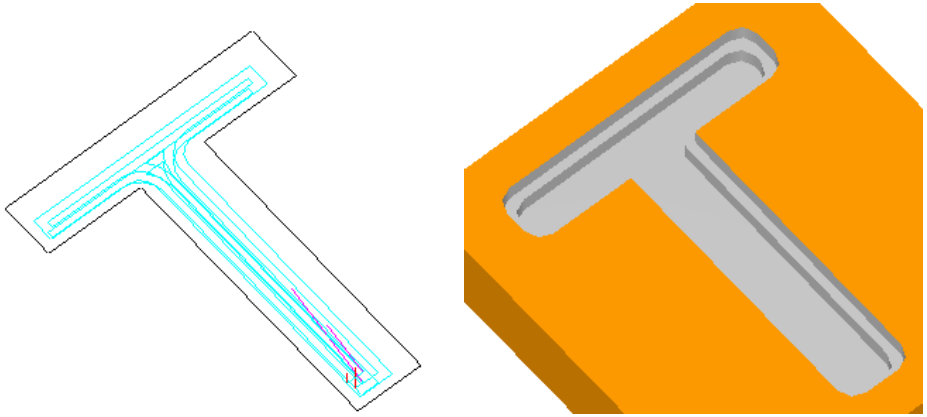
Class Room Exercise – Open V-Carve1.3dm and create V-carving toolpath.

Notes:

V-Carve Roughing

Notes:

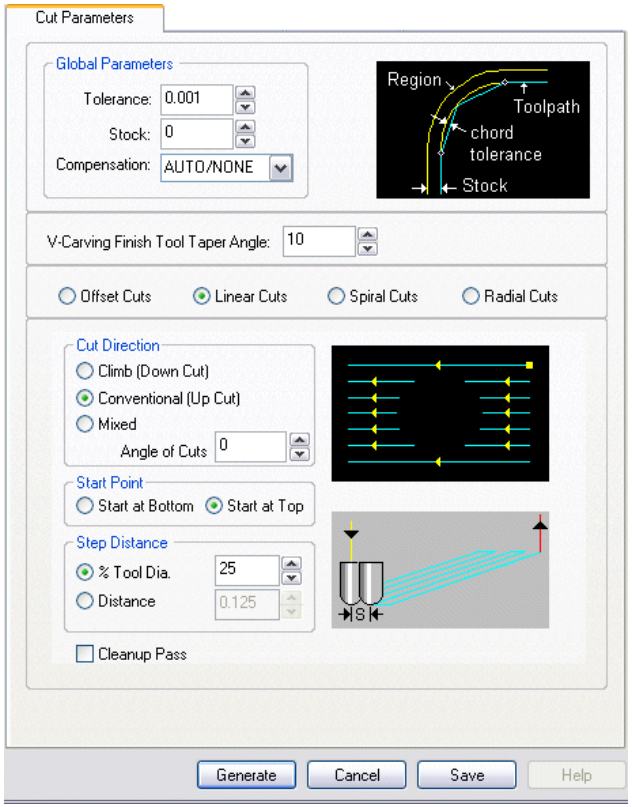
This toolpath method is used to remove material in-between the V-crave finishing toolpaths, typically with a straight cutter. However the operation compensates for the taper of the finish cutter when creating the roughing toolpath as shown below.



Cut Parameters

Cut parameters for V-Carve roughing is similar to the pocketing operation. Detailed description of each parameter can be obtained from the pocketing section.

V-Carve Finishing Tool Taper Angle- - This represents the Taper angle (included angle) of the V-bit that would be used after the roughing operation is generated. A Flat End Mill is used for V-Carve Roughing toolpath.



Note: Cut Levels, Entry/Exit, Advanced Cut Parameters is set similar to Pocketing Operation.

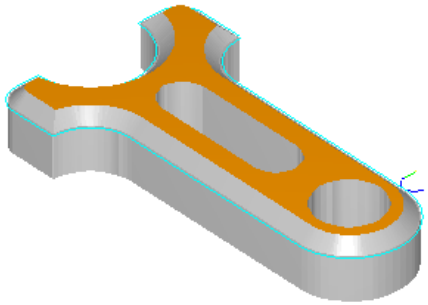
Class Room Exercise – Open Embossing.3dm and generate V-carve Roughing Operation.

Chamfering

The chamfering toolpath is used to create a sharp chamfered corner at the edges the geometry as shown below. The toolpath is created with a taper end mill of the same taper as the desired chamfer. User can specify the chamfer width; maximum depth/cut & tool tip clearance.

Notes:

Notes:



Cut Parameters

The **Global Cut Parameters** section allows the user to set the tolerance value to be used in machining. Any uniform thickness or stock to be left of the part can be specified here.

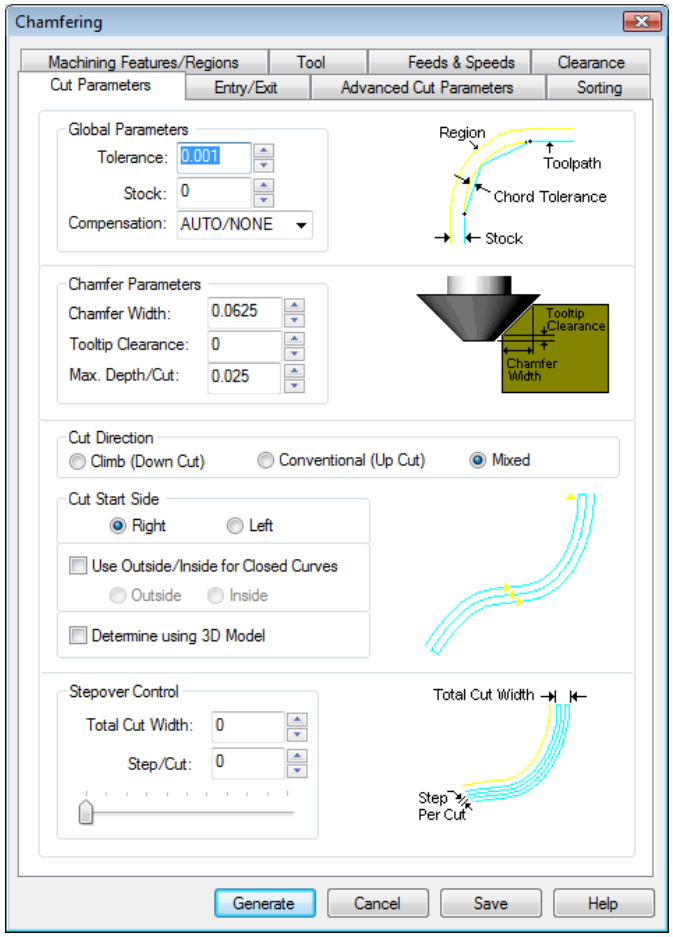
Tolerance is the allowable deviations from the actual part geometry plus the Stock layer (if any).

Stock is the thickness of the layer that will remain on the side of the part after the toolpath is complete. Roughing operations generally leave a thin layer of stock, but for finishing operations this value is zero.

The **Chamfer Width** is used to define how wide the chamfer is. Since the chamfer shape is determined by the taper angle of the cutter used it is only necessary to define the width of the chamfer. Additionally a vertical clearance called **Tooltip Clearance** can be defined to make sure the tip of the tool clears the chamfer completely so as not to leave any material on the bottom sides of the chamfer. Setting **Maximum Depth/Cut** allows the chamfer to be cut in multiple Z levels.

The **Cut Direction** can be specified as Climb, Conventional or Mixed.

The **Cut Start Side** can be used to control on which side of the curve the cutter is located for machining. For open curves the location is best defined by specifying either **Right** or **Left** of the curve. For closed curves it might be easier to check the **Use Outside/Inside for Closed Curves** and choose either **Outside** or **Inside** for locating the cutter with respect to the machining regions selected for machining.



Optionally the user can choose to request the system to compute the best side to locate the cutter based on an underlying 3D representation of the model to be machined. This is done by checking the **Determine Using 3D Model** check button.

The **Stepover Control** enables the user to define the total width of cut. This is the total width of material to be removed. This width of cut will be removed by multiple XY passes. The distance between each successive cut is defined by the **Step/Cut** parameter. Set both values to be zero if you just want one pass.

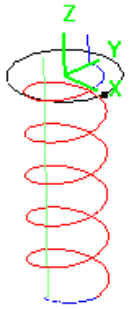
Note: Entry/Exit, Clearance Plane, Advanced Cut Parameters are set similar to Profiling Operation.

Class Room Exercise – Open Chamfer.3dm and generate Chamfering toolpath.

Notes:

Thread Milling

This method is used to cut threads using a tread mill. The pitch is defined in the thread mill tool definition. Thread milling options include internal or external threads, and right or left threads. The threads can be cut in a single pass or over multiple passes with a step over distance.



Cut Parameters

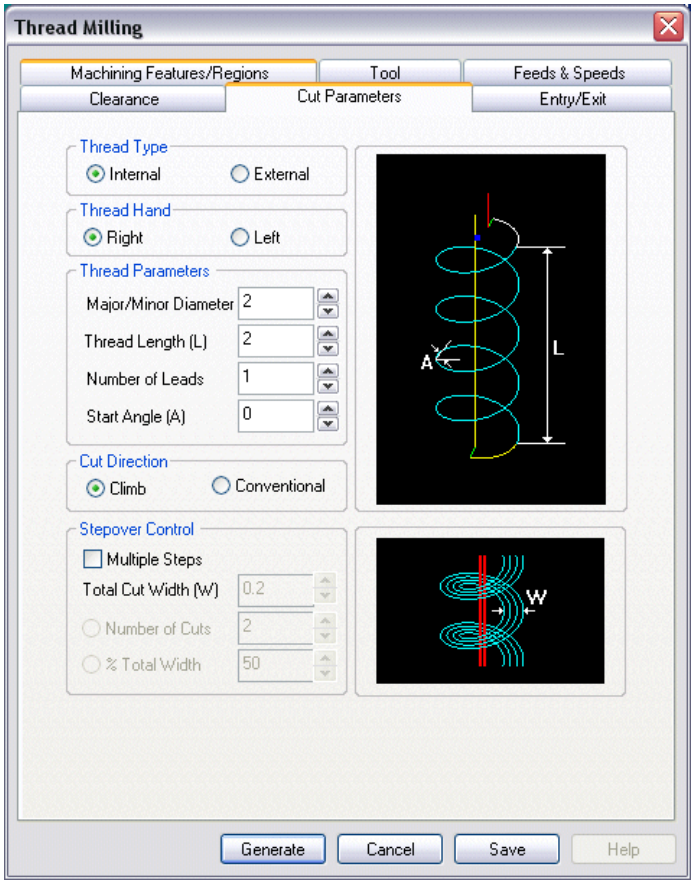
The user has the ability to select the **Thread Type,** either internal or external threads and then select the Threading **direction**, either left or right hand threads.

Using the **Thread parameters** section, the user can set the diameter of the thread, the thread length, number of leads and the start angle.

The **Cut Direction** as usual can either be climb or conventional.

The **Stepover Contro**l for Thread Milling can be defined by the number of passes – single or multiple. The Total cut width can be either a percentage of the total width or number of cuts.

Notes:



Entry / Exit

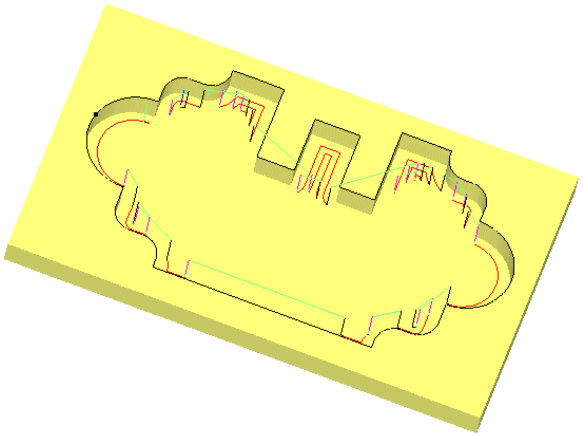
Then the user selects the Entry/Exit tab the following property page is displayed. The approach and departure can be linear and the Engage and Retract motions can be radial (arc) toolpaths.

Note: Clearance Parameters can be set similar to Profiling Operation or other 2 ½ axis machining operations.

Notes:

Re- Machining

This method uses a smaller tool to remove uncut material left after a previous operation. The previous operation and the previous tool diameter need to be specified for this operation. The previous operation can be facing, pocketing or profiling. Given these parameters, all the uncut areas are automatically calculated. These uncut areas are processed similar to a facing operation.



Notes:

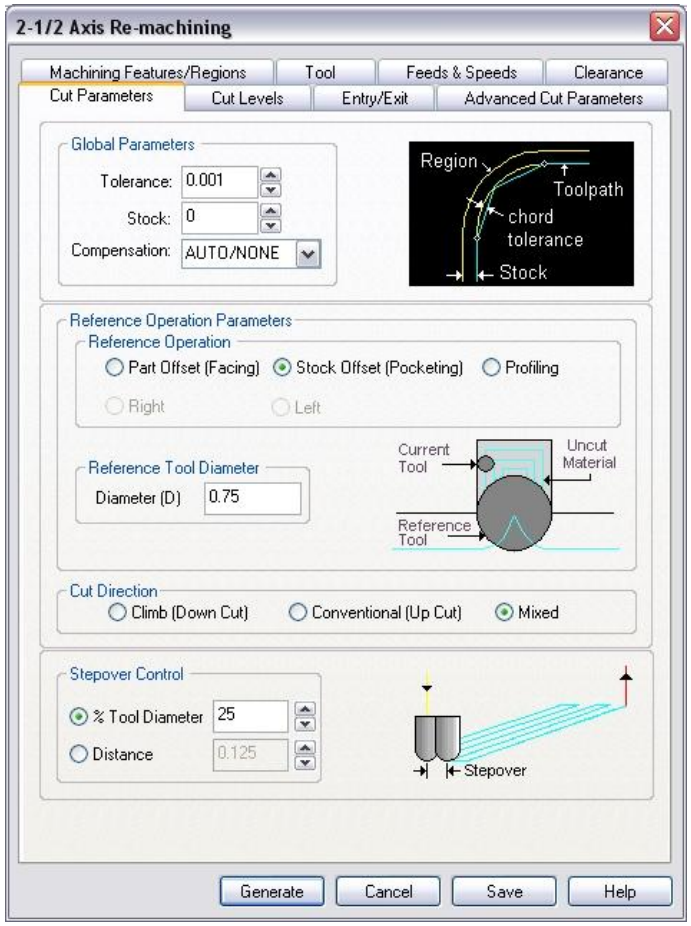
Cut Parameters

Global cut parameters can be set similar to other 2 ½ Axis operations.

The **Stepover Contro**l section allows the user to define the spacing between the cuts. The total width to be cut can be specified along with the step size for each cut by entering the value or using the sliding bar.

Reference Operation Parameters: The Reference operation parameters section is used for determining the uncut areas. The Reference operation which can be either a Facing, Pocketing or a Profiling operation needs to be specified along with the tool that was used for the reference operation. Once the Uncut areas are determined, they will be cut using the Facing kind of toolpath.

The **Cut Direction** allows the user to specify the Climb or Conventional or Mixed.



Note: Cut Levels, Entry/Exit, Advanced Cut Parameters and Clearance are set similar to profiling & facing operations.

Notes:

Simulating Toolpaths

RhinoCAM offers very powerful cut material simulation functionality to allow users to simulate actual machining of the generated toolpaths. The output of this simulation is a true 3D cut model. This 3D model can be rotated, zoomed and manipulated at will by the user. This cut model can be visually compared with the part model to show areas of

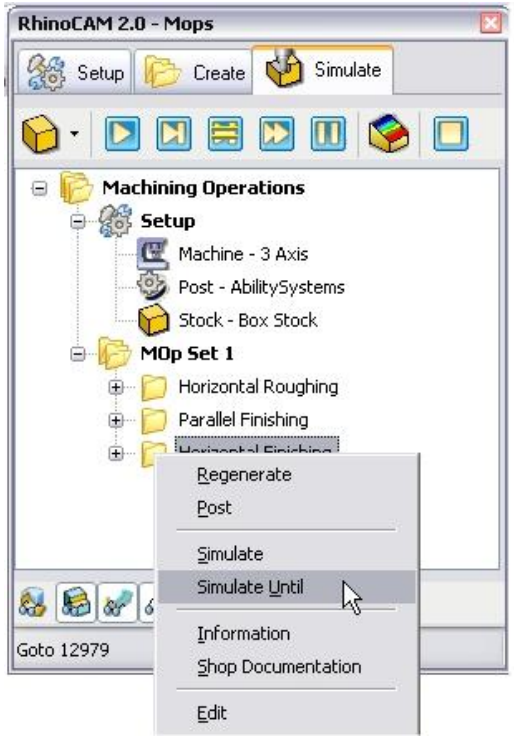
uncut material and/or areas of over-cut material using this component. The simulation features allow the early detection and correction of programming errors. The following section describes the material removal simulation functionality available in RhinoCAM.

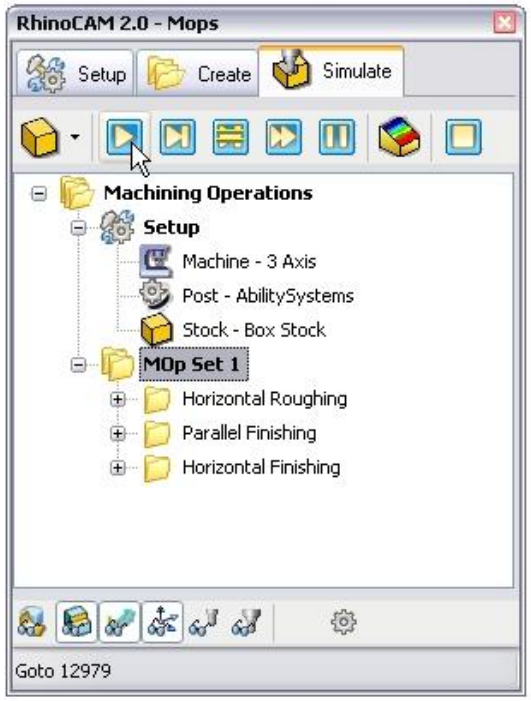
There are two kinds of toolpath simulation available in RhinoCAM. These are Tool Animation and Cut Material Simulation

In both modes of operation, the simulation can be performed either on the currently active machining operation or on multiple operations.

The active operation is the one that is selected and shown highlighted in the Machining Operations Browser. Typically, this would be the last toolpath that was generated. To simulate any operation, select the operation in the browser and click **Simulate** from simulate tab of the browser or by using **right click** and simulate.

To perform simulation on multiple operations select the last operation, right click and choose **Simulate Until**. Alternatively you can select a Mop Set and select **Simulate** to simulate all the operations within a Mop Set





Notes:

Tool Animation

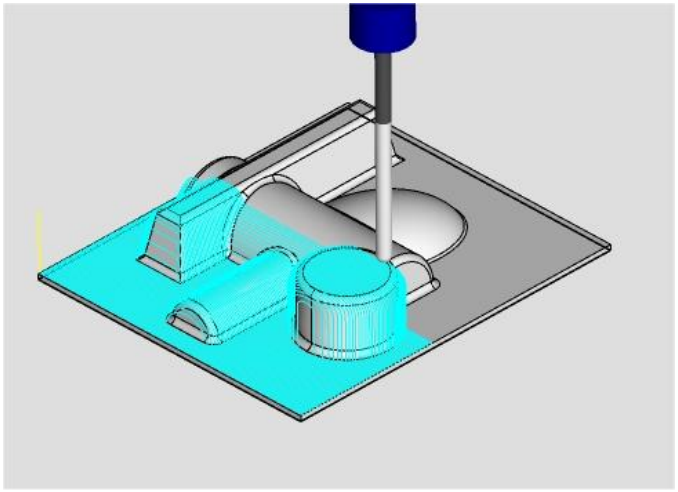
Notes:

Simple tool animation can be carried out in RhinoCAM by using the controls on the Simulate Browser. If there is no stock loaded or if the stock is loaded and the stock visibility is turned off then the tool can be animated to follow the toolpath by setting the step increment to the desired value and clicking on the Simulate button on the Simulate tab of the browser or by selecting an operation and choosing right click to simulate.

The user can also choose to display the toolpath as the tool is being animated. This is a powerful new function that allows the user to actually watch the toolpath being displayed on the screen incrementally. To do this make sure the toolpath visibility is turned on before starting the tool animation along the toolpath.

Stock Visibility

Toolpath Visibility

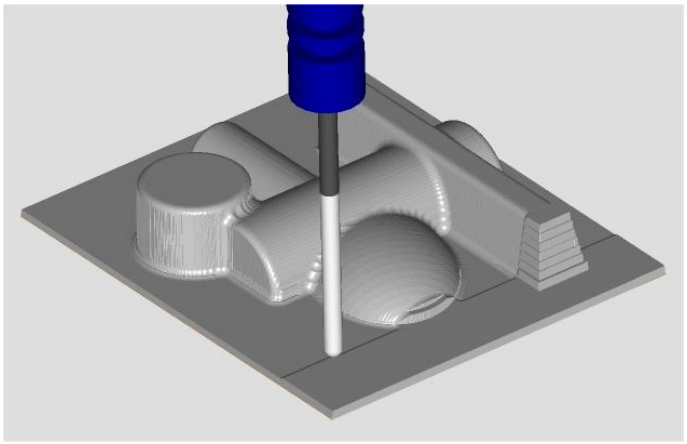


Cut Material Simulation

RhinoCAM offers a powerful cut material simulation functionality to allow users to simulate actual machining of the generated toolpaths. To perform cutting simulation, a stock model must be loaded and displayed and a machining

operation must be active. Then using the controls in the Simulate tab of the RhinoCAM -Browser the user can perform the cutting simulation.

The output of this simulation is a true 3D cut model. This 3D model can be rotated, zoomed and manipulated at will by the user. This cut model can be visually compared with the part model to show areas of uncut material and/or areas of over-cut material using this component. An example of cut material simulation is shown below.



It should be noted that in some cases, especially when simulating cutting of vertical walls (as is typically done in 21/2 axis machining), the simulation model leaves visual artifacts at these areas. The reason for this is that the simulation model display resolution is not very high along the Z-axis. This causes jagged areas to be displayed under these circumstances. It should be emphasized that these are purely visual artifacts and do not represent the true output that would be generated on the machine tool. An example of this visual artifact is shown above.

Editing Machining Operations

Once a Machining operation is created it is listed under the RhinoCAM -MOps browser. By default all the operations are created under Mop Set 1. A mop set can hold several machining operations and users can create additional MOP Sets. The can be edited in a couple of ways. Changes can be made to any of the objects that make up the operation such as the machining features, tool, feeds/speeds, clearance parameters and machining parameters. This type of editing is called associative editing. This is because the edits made to the operation are saved with the operation and upon regeneration the changes would be effected.

Notes:

Editing operations Associatively

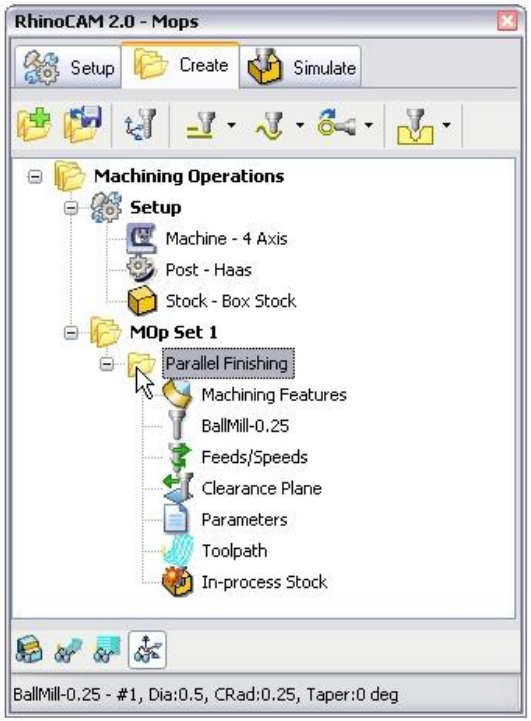
Notes:

Machining Operations can be edited by using the RhinoCAM -MOps Browser. Each machining operation is represented as a folder in the browser. In the expanded state of this folder icon, five icons representing five different objects that make up the operation are displayed. These are the machining features, tool, feeds/speeds, clearance plane and parameters. Double clicking on any of these icons gives the user an opportunity to edit the object.

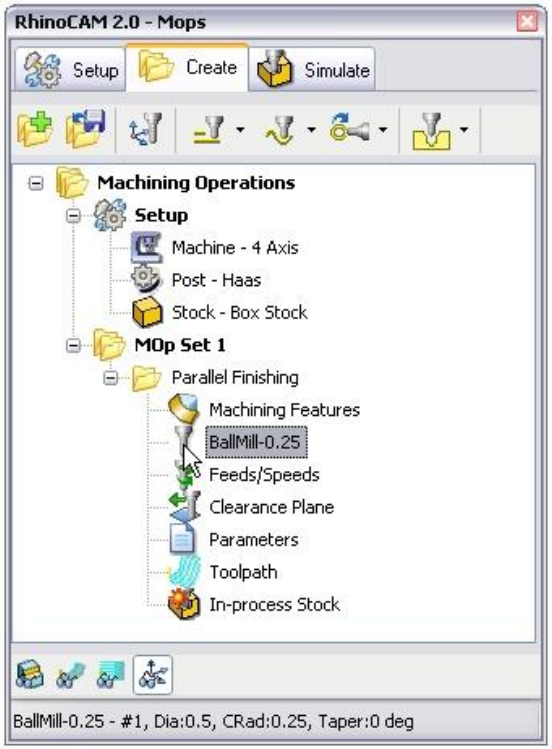
Double clicking on the operation name would allow the user to change one or more objects

- Machining features,
- Tool,
- Feeds/Speeds,
- Clearance Plane,
- Parameters

For the selected operation.



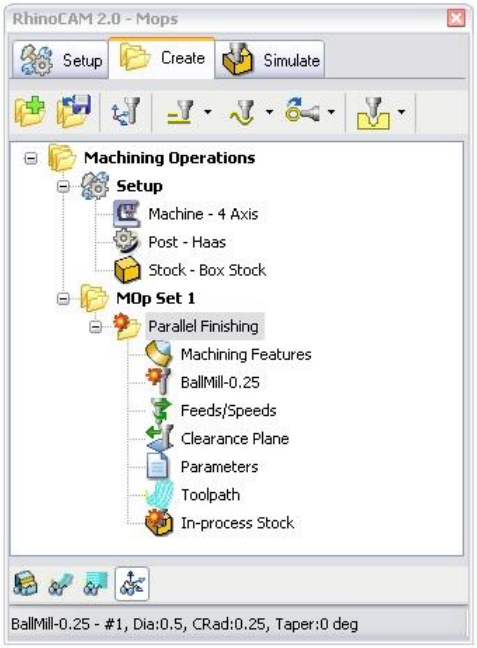
Right mouse click or **double clicking** a specific icon for example the Tool icon would bring up the Tool Creation dialog, upon which the user can substitute the current tool with another or edit the parameters of the current tool.



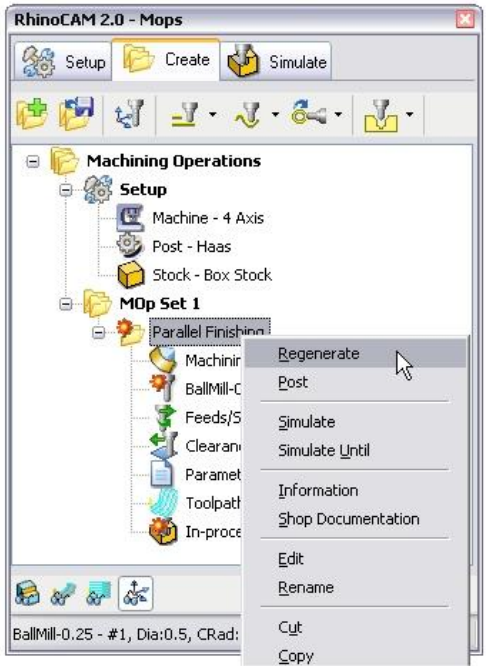
If any of the seven objects that make up the operation were to be edited after the toolpath was generated for the operation, the operation will be flagged as needing re-computation. RhinoCAM indicates such a condition by adding a red marker to the operation folder. The object that necessitated this re-computation is also displayed with a red marker. An example of this is shown below.

Notes:

In this case the tool used in the operation was edited after the machining operation was created and so is shown differently, as is the operation.

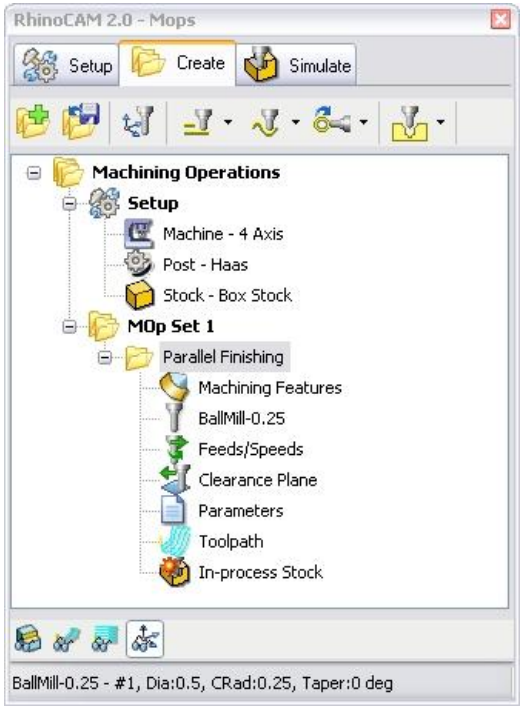


In order to regenerate the operation that is flagged with a red mark, user would have to select the operation right click and select Regenerate.



Notes:

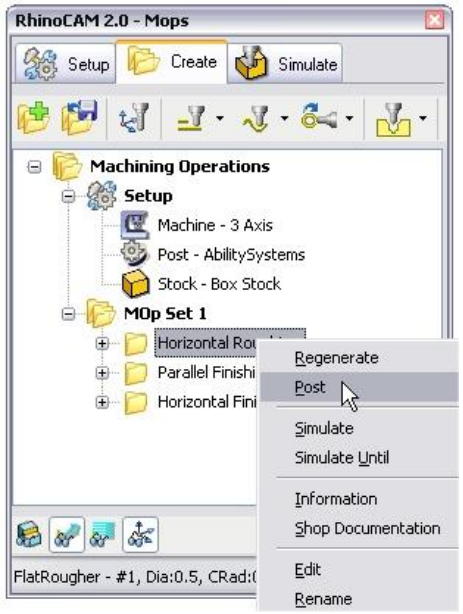
The toolpath is now generated with the modified settings.



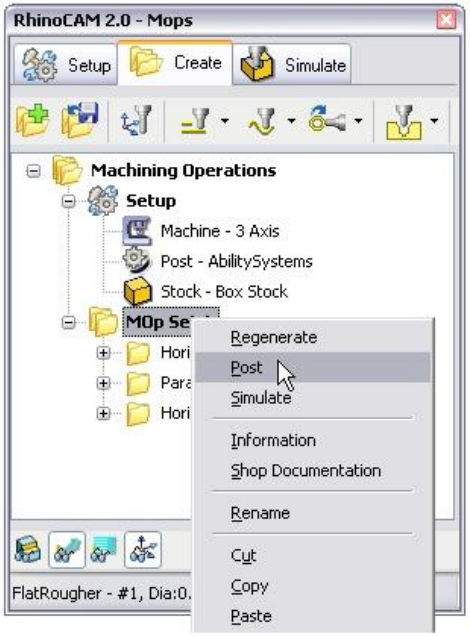
Post Processing Machining Operations

Once machining operations are created they can be post processed to a specific machine controller. To post process a machining operation, select the operation in the browser or the MOp Set, right click and select Post button in the Create MOps tab.

Notes:



Posting a machining operation



Posting a MOp Set

This would display the Post & Save Dialog.

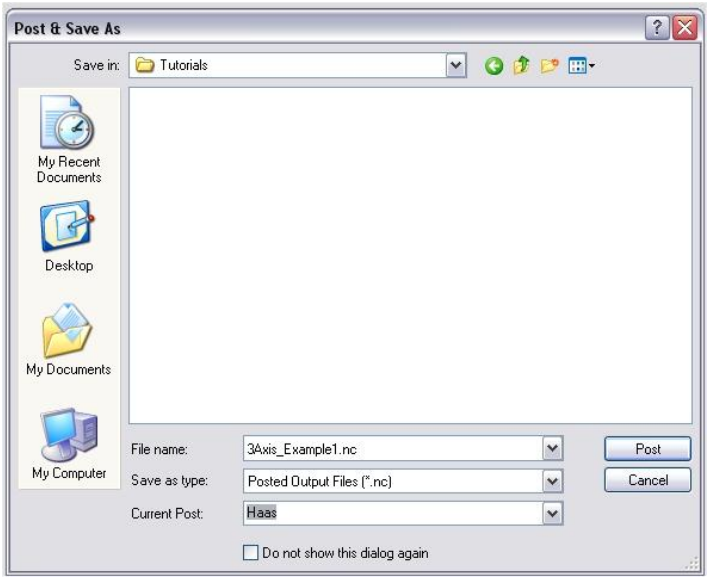
The following are the default settings when the post and save dialog is called.

- File->Save dialog points to the folder location where the part geometry is located.
- Save as type – refers to post file extension. This information is obtained from the set post options dialog.
- Current Post - refers to the **controller/post processor** to post process the toolpath. This information is also obtained from the set post options dialog.
- Program to send the posted file to - notepad

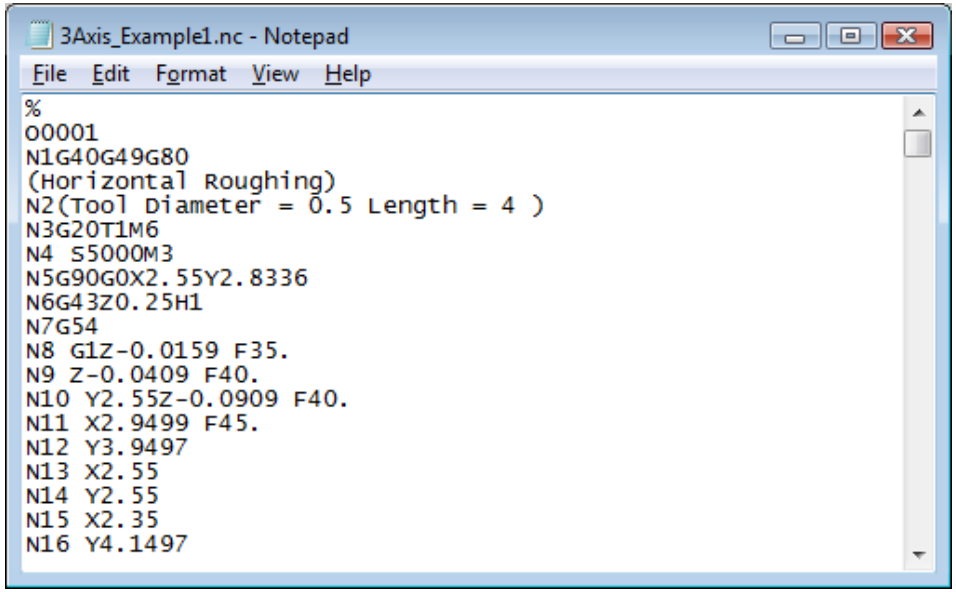
The user can override the default settings under the Post & Save Dialog.

Once the user clicks on the "Post" button in the dialog, post processing will begin and the posted file is located under the specified folder.

Notes:



The posted file is now displayed in a notepad.



Notes:

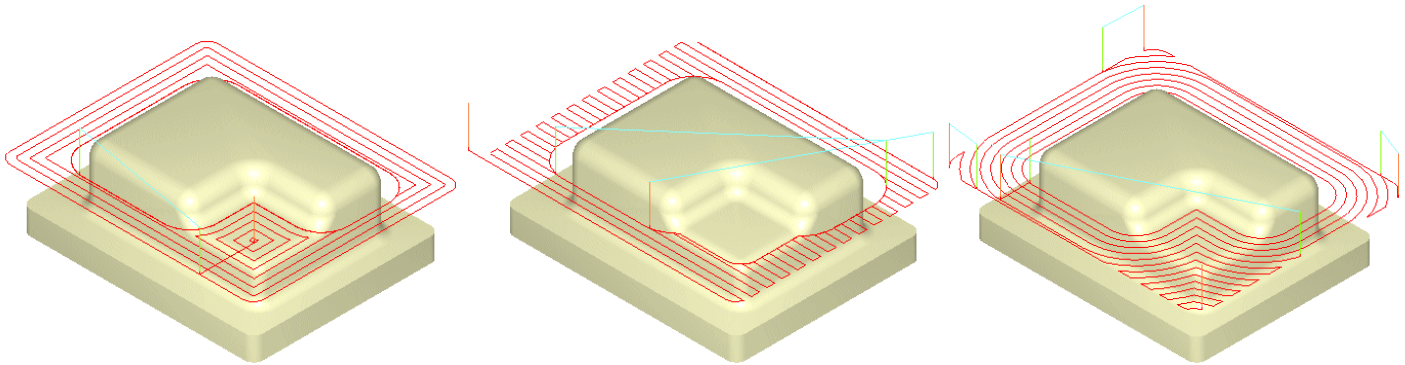
Chapter 5

Create 3 axis Toolpaths

Horizontal Roughing

This is RhinoCAM's principal method of roughing, also known as waterline or constant Z cutting, in which the material is roughed out in horizontal layers. This type of machining is very efficient for removing large volumes of material, and is typically performed with a large tool. Roughing is typically followed by semi-finishing or finishing toolpaths.

Both part and stock geometry are used to determine the regions that can be safely machined. Three types of cutting patterns are available: Linear (parallel, zigzag lines), Stock Offset (spiral pattern within stock and part), and Part Offset (spiral pattern outside the stock and outside the part). Tool motions are shown for single Z levels in the pictures below.



Cut Parameters

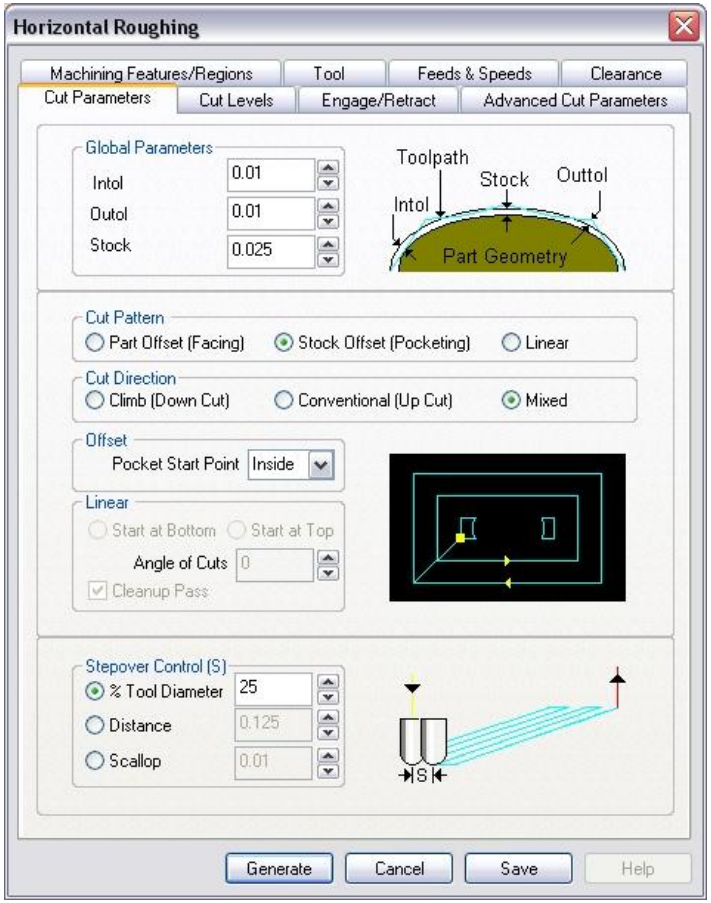
Stock: The thickness of the layer that will remain on top of the part after the toolpath is complete. Roughing operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Intol and Outol is allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any).

Notes:

Intol: Inward tolerance - the maximum thickness of material that can be removed from the Stock layer.

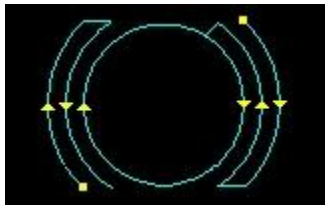
Outol: Outward tolerance - the maximum thickness of material that can remain above the Stock layer.



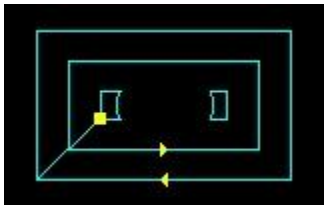
Cut Pattern: Allows the user to define the type of cut pattern that the tool will follow when it is at each Z level. There are three types of cuts that the user can choose. The first two are an offset cut type (stock offset or part offset) and the third is a linear cut type.

Notes:

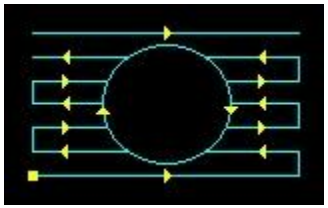
Notes:



Part Offset



Stock Offset

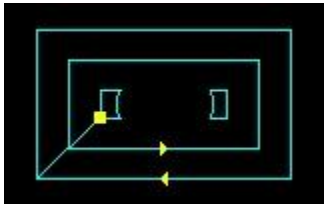


Linear

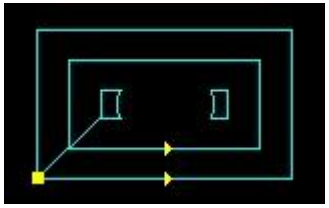
In the offset cut type, the cut regions are successively offset until there is nothing left to machine. This type of cutting is sometimes called spiral machining. The user can specify the cut start point to be either the inside of the cut regions or from the outside.

Cut Direction can be controlled by specifying either climb or conventional or mixed.

Pocket Start Point can be set to inside or outside. This is applicable for Part offset and Stock Offset cut patterns.



Inside



Outside

Cleanup Pass parameter will cause automatic detection of all the corners that the tool could not reach between each pass. It will then add a toolpath based on the uncut area detected; either a linear cut in case of smaller areas or a cut that travels along the shape of the uncut area, when the area is large. This is applicable for Linear cut patterns.

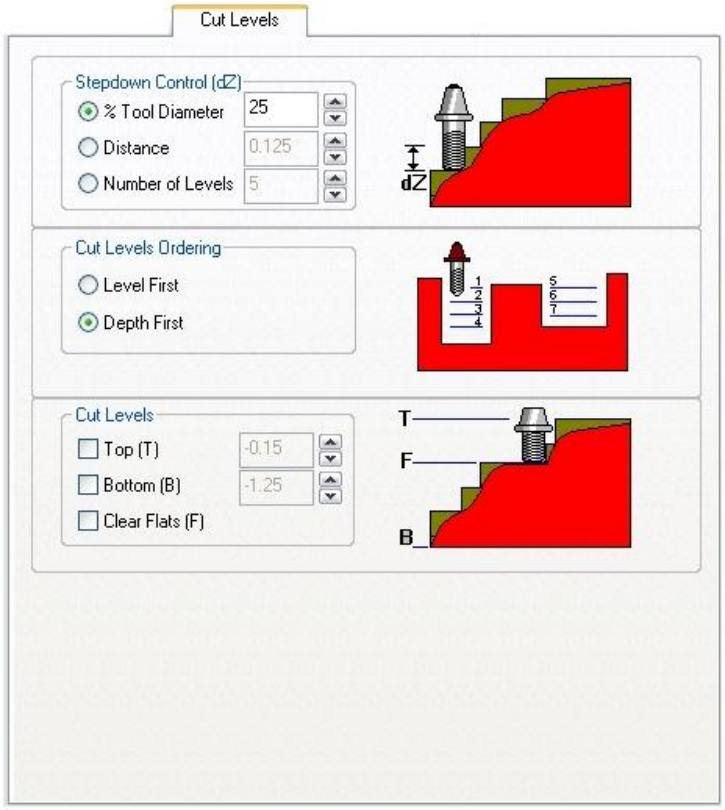
The **Stepover Control** section allows the user to define the spacing between the cuts. The spacing can be specified either as a percentage of tool diameter, a specific distance or as the scallop height.

Cut Levels

The **Stepdown Control** section allows the user to define the spacing between the horizontal cut levels for the roughing operation. The spacing can be specified either as a percentage of tool diameter, a specific distance or as the total number of levels desired.

The **Cut Levels Ordering** section allows the user to order the cut regions. The order of cutting can either be specified as level first or depth first. In the level first option cut regions are ordered such that all cut regions in a single Z level or machined first before the next cut level is machined. In the depth first option, the regions in successive Z regions that form a single pocket or machining feature are machined first before the next such feature is machined.

The **Cut Levels** section allows the user to optionally specify the top and bottom cut levels. If the top cut level is not specified, the maximum Z value of the stock model is used as the start Z value for computing the cut levels. If the bottom cut level is not specified then the minimum Z value of the stock model is used as the end Z value for the cut levels.



Notes:

Notes:

Clear Flats: The user can also optionally tell the system to clear flat areas automatically. If this is chosen then the system will insert a cut level when it detects a flat area in the part model. The spacing of the next lower cut level will then continue from this cut level.

Engage / Retract

The user will be able to specify how the cutter engages and retracts when forced to engage into material. The user also has the ability to specify the engage conditions when engaging from the outside of the part. The user can also specify the type of transfer motions to perform while cutting.

The **Engage in Material** section allows the user to define how the cutter would engage into material when forced into such a situation. This can happen when machining a cavity or pocket. The user has the option of ramping the cutter or engaging vertically down. The ramp engage motion will typically be used when machining with a flat or corner-radius end mills.

Setting the Engage/Retract in material to Vertical approach applies a vertical plunge into the material.

The **Engage in Air** section allows the user to define how the cutter would engage to start cutting when starting from outside of the part. This can happen when machining a core or performing a facing operation. In such cases the user has the option of specifying the cutter to start from the outside, a certain distance away from the cut start point. Or optionally a straight vertical engage can be specified.

Advanced Cut Parameters

The advanced cut parameters page is used to control the cuts for high speed machining. All of the options in this section are designed to reduce rapid acceleration and deceleration of the machine during the cutting process. These parameters allow smoothing of the toolpaths by introduction of arcs. The user can use these parameters even if the controller does not support arcs. In this case, make sure that the output is set to linear output. This can be set in the Machining Preferences dialog in located in the RhinoCAM Mops in the Setup tab from the Preferences icon.

Cut Corner Rounding Options: This option is used to round sharp corners in the toolpath. The user can specify a rounding radius; fillets of the specified radius will be introduced in sharp corners if possible. These fillets will only be introduced in planes parallel to the XY plane.

Cut Arc Fitting: This option can be used to fit arcs to the toolpath. Arc fitting can be accomplished on planes parallel to the XY, XZ, and the YZ planes. The user specifies an arc fitting tolerance and the system attempts to fit

arcs to the computed toolpaths. Fitting arcs to toolpaths serves to make the toolpath smoother as well as reducing toolpath size.

Smooth Cut Transitions: This option can be used to introduce S shaped or C shaped cut transitions between two successive offset cuts. These transitions are introduced only in offsets that are generated in planes parallel to the XY plane. These transitions allow the cutter to transition from one cut to the next in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.

Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. Typically the user would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Cut Transfer - The user can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can either be instructed to move to the clearance plane and then perform the transfer motion to the next cut location or it could do a skim motion. In the skim motion, the system automatically determines the safe height by taking into consideration the condition of the part and stock model and using this Z value as the height to perform the transfer motions.

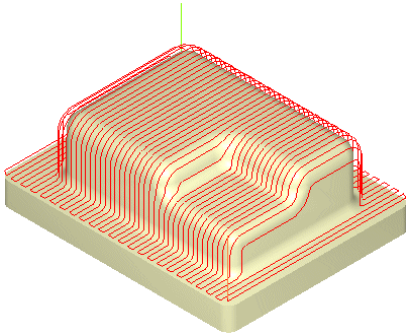
Notes:

Parallel Finishing

Notes:

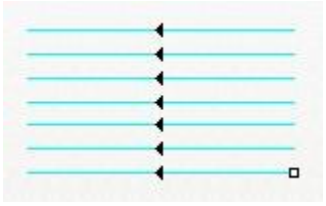
This is an efficient method of finishing or pre-finishing, typically used when part surfaces are relatively flat. A 2D linear zigzag pattern is generated on the XY plane above the part geometry. The tool moves along this cut pattern, following the contours of the part geometry below.

The orientations of these vertical planes about the XY plane are constant and can be defined by an angle about the X axis. As the cutter follows these vertical planes, it can either form a Zig or ZigZag cut pattern. In the Zig cut pattern, the cutter always goes in a constant direction while in the ZigZag cut pattern, the cutting direction alternates between two successive parallel planes. This is one of the most commonly used cut methods for pre-finishing and finishing parts. The tools typically employed in this operation are ball end mills.

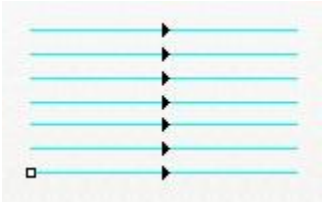


Cut Parameters

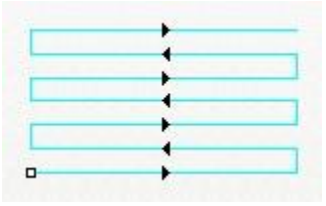
Global Parameters are set similar to Horizontal Roughing Operation. The **cut direction** can be controlled by specifying either climb or conventional or mixed.



Climb



Conventional



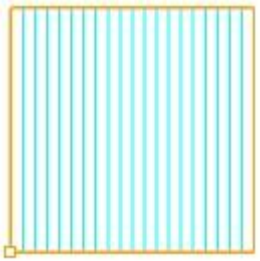
Mixed

The **start side** can be specified as from Top or Bottom

The **angle of cuts** can be specified as degrees (0-360).

Angle = 0

Angle = 45

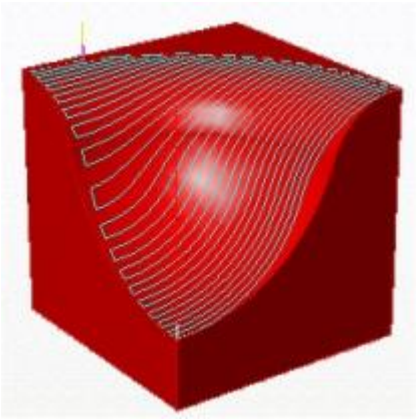


Angle = 90

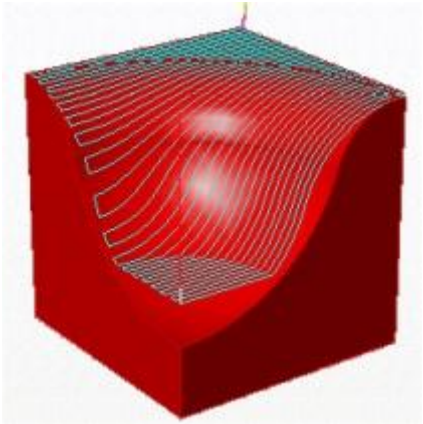
The Stepover Control section allows the user to define the spacing between the cuts. The spacing can be specified either as a percentage of tool diameter, a specific distance or as the scallop height.

Z containment

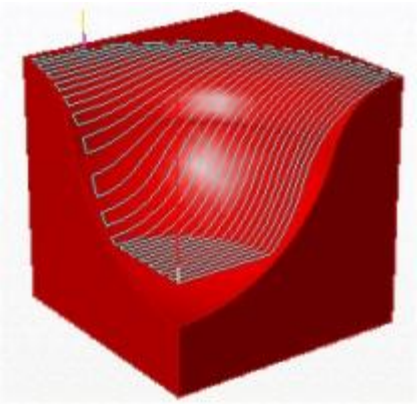
This feature allows the user to contain the toolpath in Z level by specifying High Z and Low Z values either by inputting the values or by picking the Z height from the 3D model. Specifying a Z level containment limits the parallel finishing toolpath to high z or low Z values. User can specify a high and a low z value. If the Z level containments are not defined parallel finishing toolpath takes the models Z height into consideration.



Specifying Highest Z



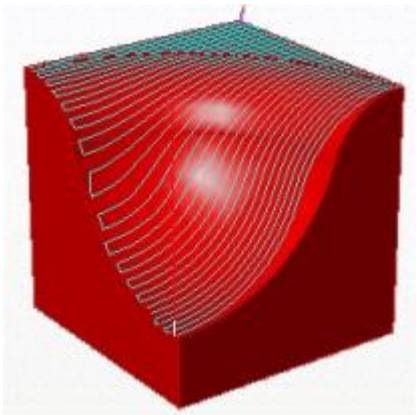
Specifying Lowest Z



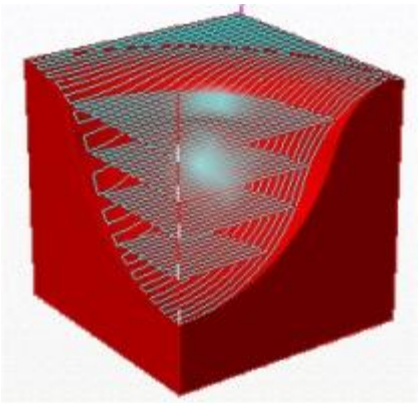
Highest & Lowest Z

Notes:

User can also insert **multiple step-downs** in Z cuts for parallel finishing. This can be accomplished by checking the Insert multiple step-down Z cuts. The step down Z distance can be specified as a percentage of tool diameter, distance or as number of levels.



Without Multiple Step-down



With Multiple Step-down

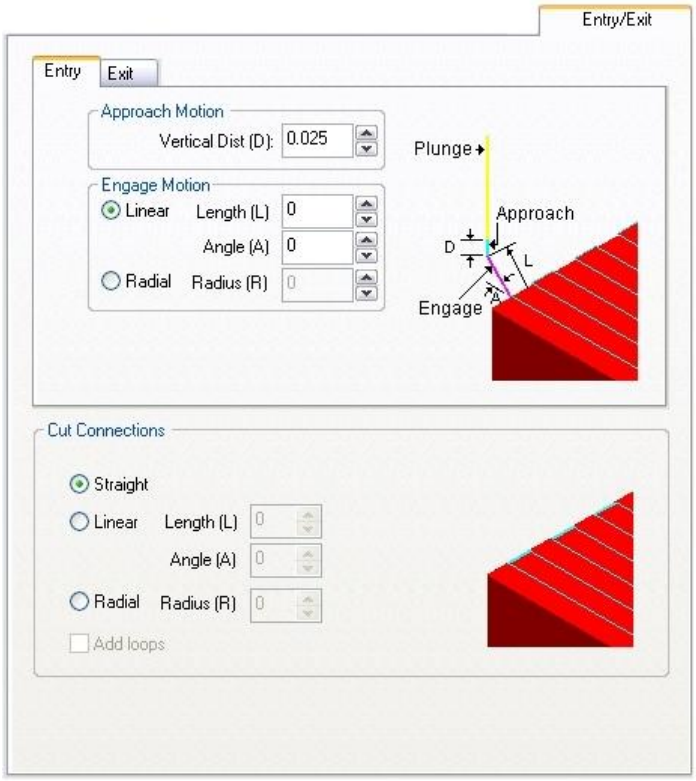
Entry / Exit

The Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

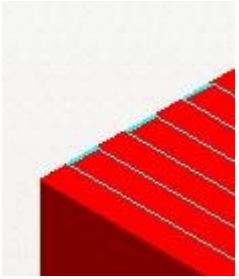
In this option the approach and departure distance can be specified. This distance is the distance from the cut start point where the cutter switches from an approach feed rate to an engage feed rate for entry and from retract feedrate to departure feedrate for exit motions.

The Approach and Retract motions can be Linear or Radial. The cutter uses a ramp motion with the defined angle and distance away from the cut start point to engage and retract from the cut start or end point. The ramp angle is measured from the horizontal. In the radial engage/retract option, a circular arc motion is used to engage and retract from the cut start or end point. The radius of this arc can be controlled by the user.

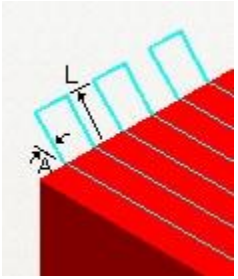
Notes:



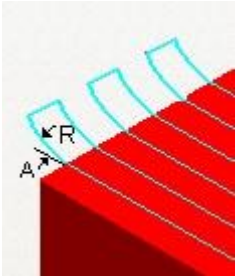
Cut Connections can be turned on for smooth transition between cuts (stepover). The transfer between cuts takes place at the end of each cut. User can specify the Angle and Length or Radius for the cut connections. Cut connections can be Straight, Linear or Radial.



Straight



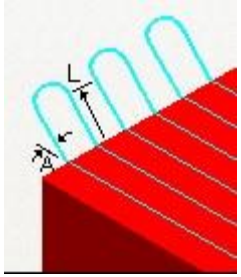
Linear



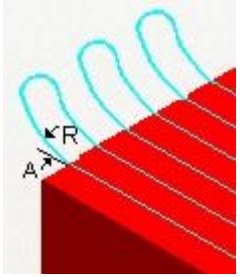
Radial

Notes:

Loops can be added to Linear or Radial cut connections. These transitions allow the cutter to transition from one cut to the next in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.



Linear Entry

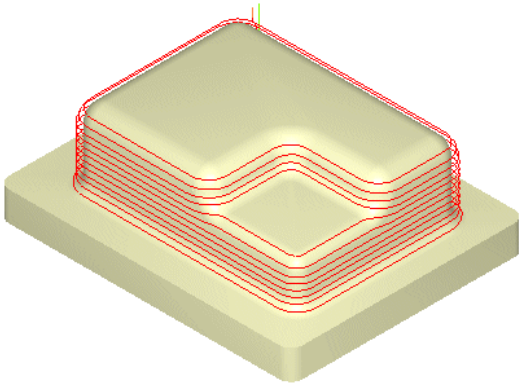


Radial Entry

Clearance Plane is set similar to Horizontal Roughing operation.

Horizontal Finishing

Horizontal Finishing is a toolpath method that can be used either as a pre-finishing operation or as a finishing operation. This method is similar to Contour Finishing. The major difference is the type of cutting that is performed. Here, the cutter finishes in constant Z planes. This type of cutting is suitable for parts with have steep walls while Contour Finishing is more suitable for shallow parts.



The tool types commonly used in this method are ball tools. As the cutter follows these horizontal planes, it can either maintain climb/conventional/mixed type of machining. In climb or conventional, the direction of cutting is maintained so as the corresponding cutting condition is maintained on the part. In the mixed type of machining

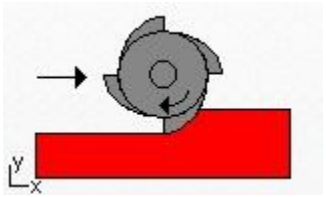
Notes:

however, the direction of cutting is alternated between each parallel plane, similar to the ZigZag machining in Contour Machining.

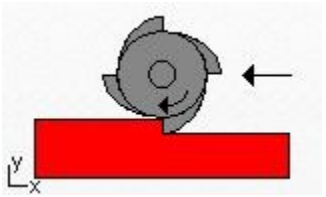
Cut Parameters

Global parameters are set similar to Horizontal Roughing and Parallel Finishing operations.

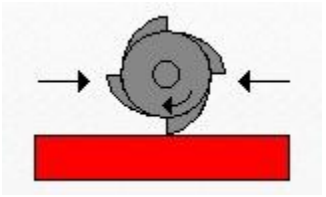
The Cut Direction of the toolpath can be specified either as Climb, Conventional or Climb/Conventional by selecting the appropriate radio buttons. As mentioned before, in climb or conventional, the direction of cutting is maintained so as the corresponding cutting condition is maintained on the part. In the Climb/Conventional type of machining however, the direction of cutting is alternated between each parallel plane.



Climb Cut



Conventional Cut


Climb/Conventional Cut

Cut Levels are specified similar to Horizontal Roughing operation.

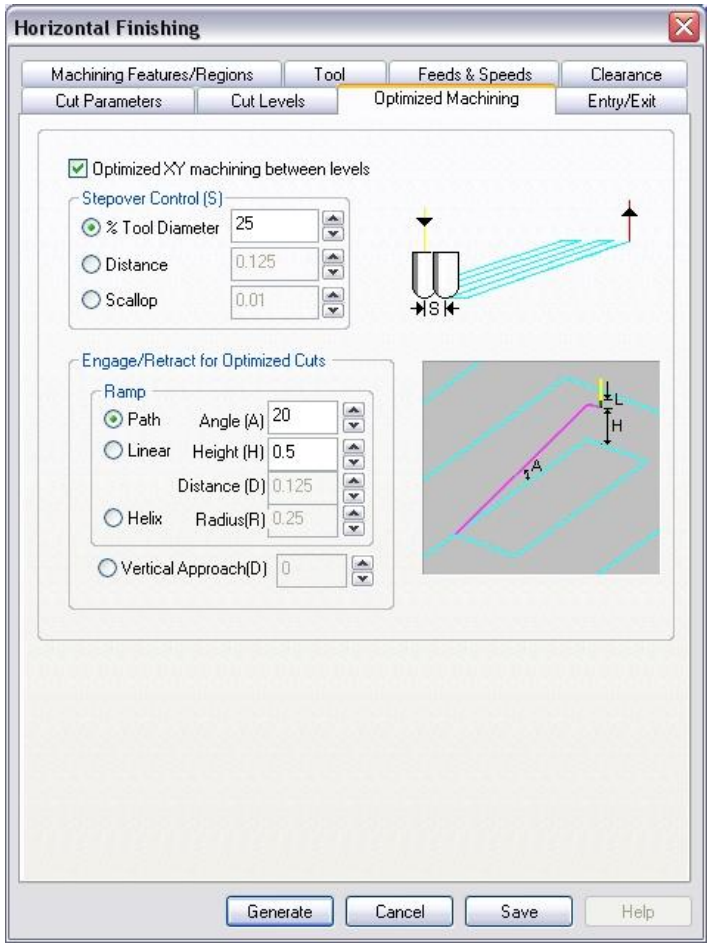
Optimized Machining

Optimized machining is used when the user wants to maintain almost constant scallop height during machining with the Horizontal Finishing method. In this method, machining takes place in constant Z planes spaced a constant Z distance from each other. Due to this constant Z spacing, machining relatively flat areas will leave areas un-machined. The size of these un-machined areas increase with the degree of the flatness.

Optimized machining automatically recognizes these flat areas and inserts projection toolpaths, similar to 3 Axis pocket machining to clean out un-machined areas. This is a highly effective way of maintaining almost uniform scallop height on the part.

The user can control the spacing of these optimized pocket toolpaths as well as the engage and retracts for these pocket toolpaths. It should be noted that optimized machining would work only between closed cuts. The uncut material between open cuts will be ignored.

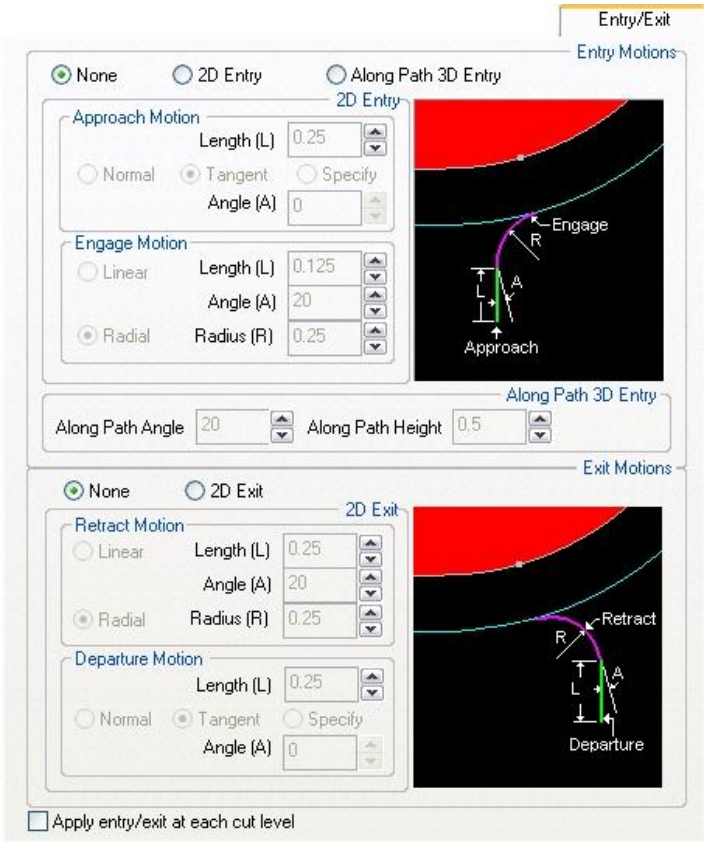
Notes:



Notes:

Entry / Exit

The Entry motion consists of **Approach Motion** and an **Engage Motion**. The user can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion. Similarly the Exit motion consists of a **Retract motion** followed by a **Departure motion**. The Retract motion can be either a radial (arc) motion or a linear motion at an angle. The departure motion is a linear motion.



The user can choose to define the 2D Entry and Exit path of the tool. Depending on the stock material and cut pattern the Approach/Departure Motion can be either normal, tangential or at a specified angle with respect to the stock.

The Engage/Retract Motion can be set to Linear or a Radial. In the Linear Motion the cutter follows a linear ramp motion, ramping back and forth from a user specified height to the engage point. The length of this move, as well as the angle of this motion can be specified by the user.

The Entry path can also be defined as a 3D Entry along a specified path angle with a path height.

The Entry/Exit Motions can be set for each cut level by checking the box at the bottom of the screen.

Clearance Plane parameters can be set similar to other roughing and finishing operations.

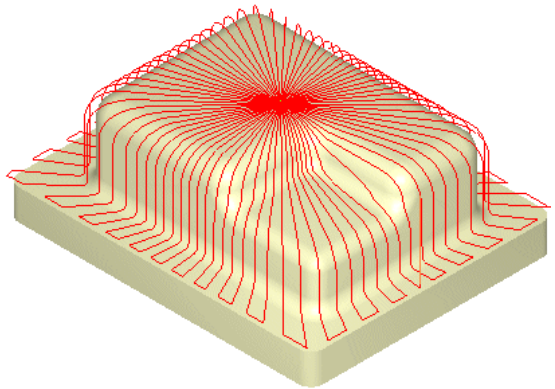
Notes:

Class Room Exercise – Open 3Axis_Example1.3dm and create Horizontal Roughing, Parallel Finishing & Horizontal Finishing operations.

Notes:

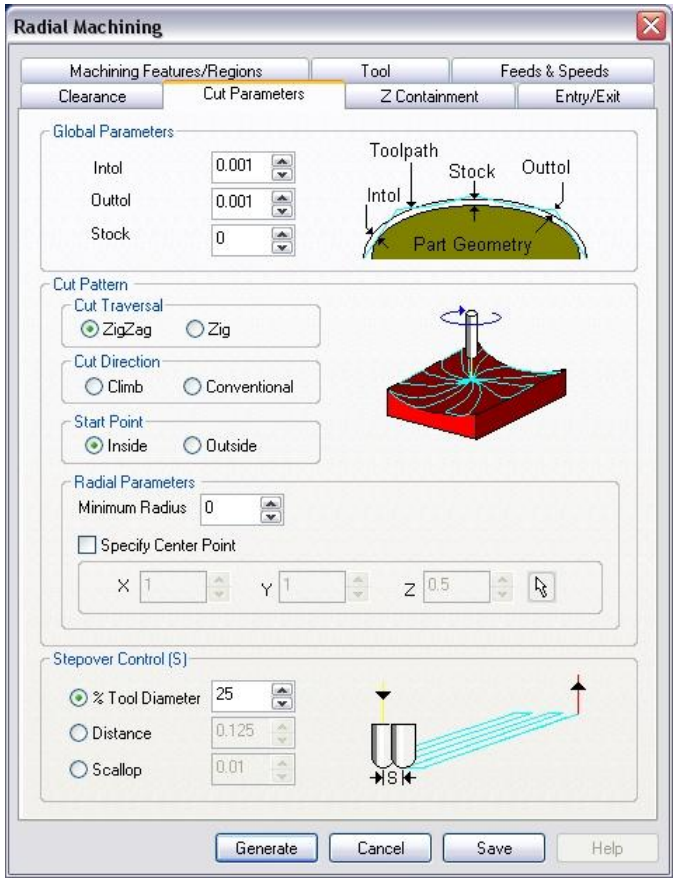
Radial Machining

Radial Finishing is a toolpath method that can be used either as a finishing operation for regions that have circular or near circular characteristics such as pocket bottoms. In this cut method, linear cuts are generated inside an enclosed machining region, extending from a center point. Machining regions are necessary to be active for this cut method to work. There is no limitation on the number of regions or the number of nesting's of the region.



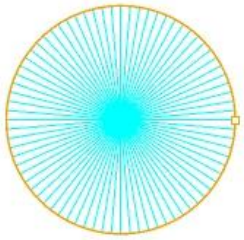
Cut Parameters

As the cutter follows these linear cuts, it can either form a Zig or ZigZag cut pattern. In the Zig cut pattern, the cutter always goes in a constant direction while in the ZigZag cut pattern, the cutting direction alternates between two successive linear cuts. The start point of the cuts can also be changed from outside to inside or inside to outside.

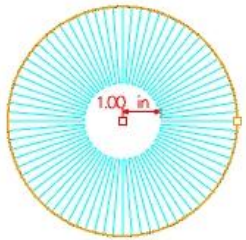


Radial Parameters – This allows the user to specify the minimum radius for the Radial Machining toolpath and the center point for the radial toolpath.

Specifying a minimum radius eliminates the toolpath from the center point to the specified radius.



Min Radius = 0



Min Radius = 1

Notes:

Notes:

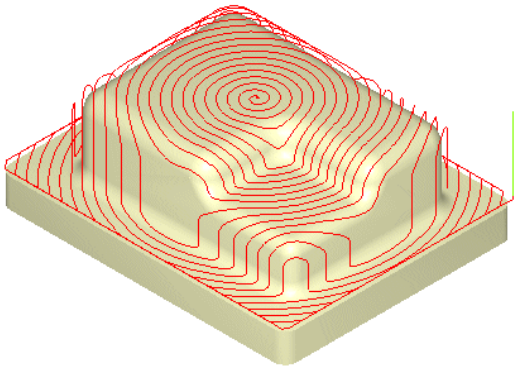
The user can specify the center point or by default the system calculates the optimum center point.

The Stepover Control section allows the user to define the spacing between the linear cuts in either the Zig or the ZigZag cut pattern. The spacing can be specified either as a percentage of tool diameter, a specific distance or as the total number of linear cuts desired. The stepover is computed on the circumference of a circle centered about the center of the machining region and enclosing all of the machining regions completely. This guarantees that the desired stepover will never be exceeded at any time during the cut operation.

Z Containment, Entry / Exit & Clearance Plane parameters are set similar to Parallel Finishing operation.

Spiral Machining

Spiral machining is a method of generating a spiral toolpath. It can be used efficiently for circular regions. Single/multiple regions must be selected and activated to generate the spiral toolpath. The toolpath will be generated only within the activated regions.



Cut Parameters

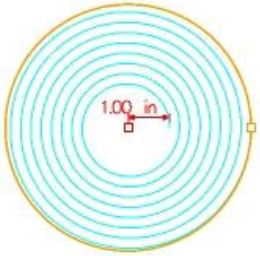
The cut patterns generated could be in the climb or the conventional direction. In the climb directional the cutter always moves in the counter-clockwise direction and in the conventional direction the cutter follows a clockwise path.

The cut pattern allows the user to define the start point of the toolpath. The two available options are: Inside or Outside the selected region. Selecting the "Inside" option causes the toolpath start point to be the centroid of the selected region. Selecting the "Outside" option causes the start point of the toolpath to be on the largest circle with the centroid as the center.

Radial Parameters – This allows the user to specify the minimum radius for the Spiral Machining toolpath and the center point for the Spiral toolpath.

Specifying a minimum radius eliminates the toolpath from the center point to the specified radius.

Min Radius = 0



Min Radius = 1

The Stepover Control section allows the user to define the spacing between the cuts in either the Inside or the Outside cut pattern. The spacing can be specified either as a percentage of tool diameter, a specific distance or as the total number of linear cuts desired. The stepover is computed on the radius of a circle centered about the center of the machining region and enclosing all of the machining regions completely. This guarantees that the desired stepover will never be exceeded at any time during the cut operation.

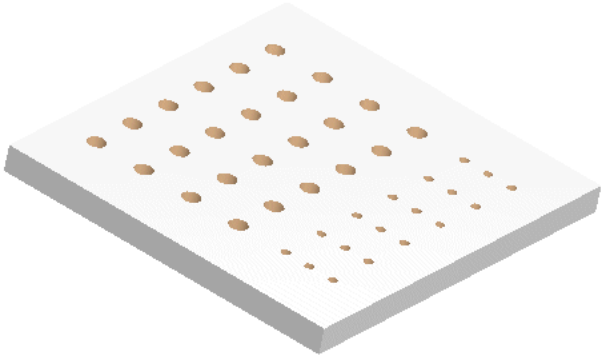
Z Containment, Entry / Exit & Clearance Plane parameters are set similar to Parallel Finishing operation.

Notes:

Chapter 6

Create Hole Machining

These operations are used to create holes in the part, including drill holes, counter sunk holes and through holes. Tapped and bored holes can also be created.



Drilling

The Drill cycle is used to cut holes in the part. The drilling toolpath method is invoked by picking the Create folder in the RhinoCAM MOps and selecting the Holes menu.

The following drill cycles are available:

- **Standard**: Used for holes whose depth is less than three times the tool diameter.
- **Deep**: Used for holes whose depth is greater than three times the tool diameter, especially when chips are difficult to remove. The tool retracts completely to clean out all chips.
- **Counter Sink**: Cuts an angular opening at the end of the hole.
- **Break Chip**: Similar to Deep drilling, but the tool retracts by a set clearance distance.

Notes:

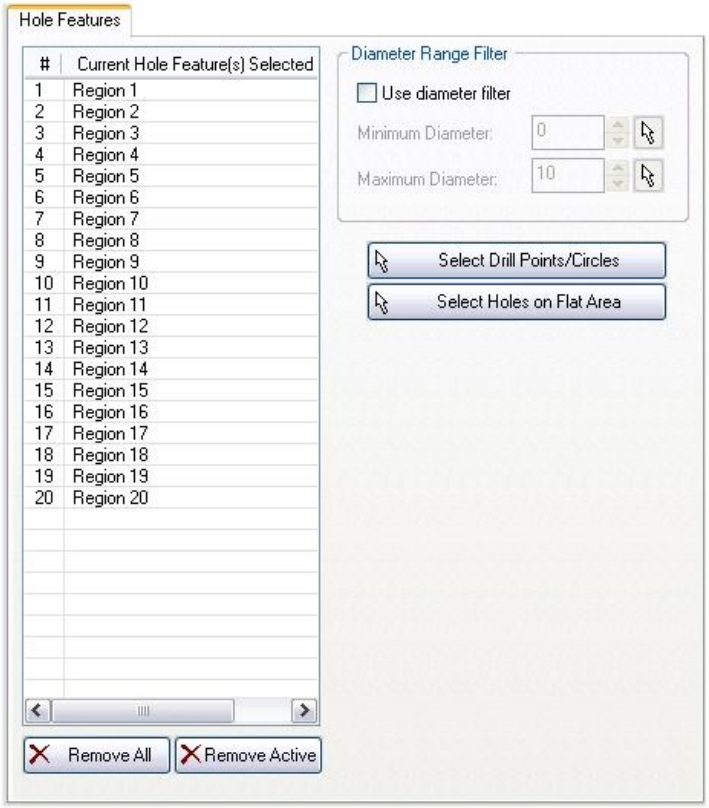
Hole Features

Notes:

Hole Features are nothing but circular regions that must be selected for the drill operation. The user is given the flexibility to select drill points and select all holes on a Flat area for the operation.

The **Use Diameter Filter** box can be checked to select specified holes. This allows the user to select only the holes within the specified diameter range.

The holes selected will be displayed on the left hand side of the dialog box. After the selection the user can choose to delete the selection of any/all of the holes using the **Remove All** and **Remove Active** buttons at the bottom of the dialog box seen below.



Tools - The Drill tool dialog limits the user to pick only the tool that can be used for the drilling operation namely, the Drill and the Center Drill bits seen below.

Cut Parameters

The drill cycles supported are:

- Standard
- Deep
- Breakchip
- Countersink
- User defined Drill cycles

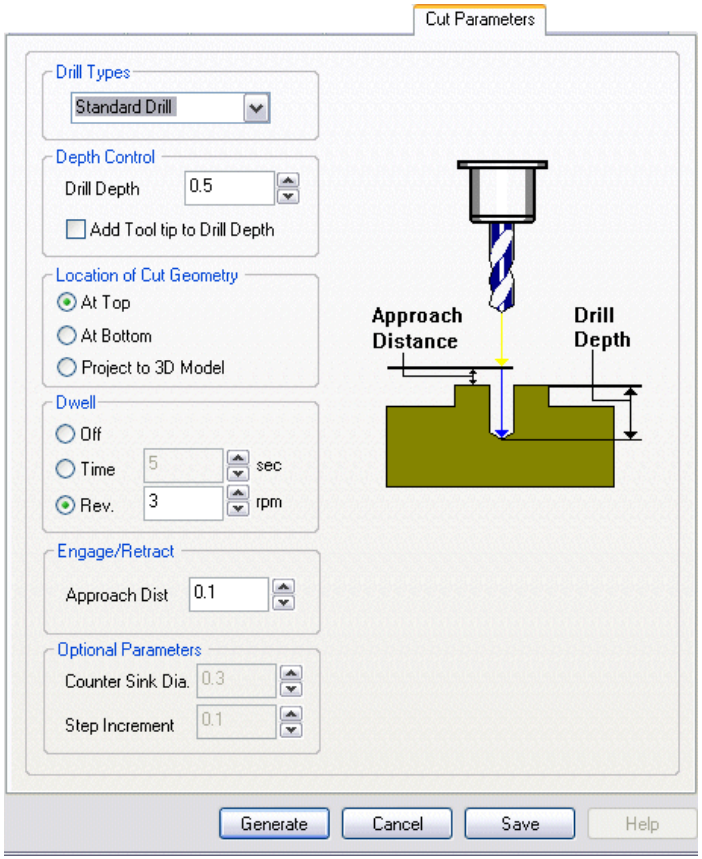
The **standard drill** cycle is used to cut holes shorter than three times the tool diameter.

The **Deep Drill** cycle is usually used to cut deep holes with depths greater than three times the tool diameter, especially when the chips are difficult to remove. The tool retracts completely to clean out all the chips. The **Breakchip drill** cycle is also used for deep drilling, but the tool retracts only by a calculated clearance distance. **Countersink drilling** is used when cutting of an angular opening at the end of the hole is required.

Drill depth refers to the hole depth. It needs to be specified for Standard, Deep and Breakchip drilling.

Notes:

Notes:



Cut Geometry Location – allows the user to specify if the selected region is at the top or bottom of the part geometry / hole feature.

Dwell is an optional parameter that allows a machine delay of either 't' seconds or 'n' revolutions of the spindle.

The **CounterSink Diameter** is required only for the CounterSink Drill type operation. RhinoCAM will automatically calculate the drill depth. "Step Increment" specifies the tool retract distance. It needs to be specified for Deep and Breakchip drilling.

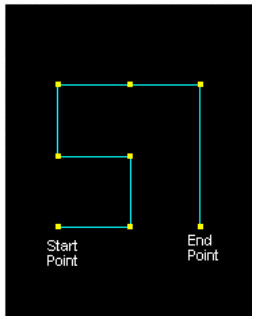
The user can set the **Engage** and **Retract approach distance.**

Sorting

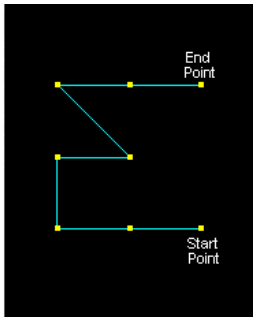
Allows the user to specify a sorting method.

No Sort – Does not use a sorting method and uses the order in which the geometries are created or order in which the machining regions were selected.

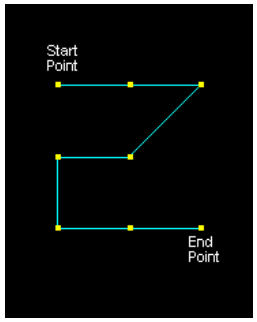
Minimum Distance Sort – Sorts based on minimum distance of start points between 2 or more geometries. The start point can be set to Lower Left, Lower Right, Upper Left or Upper Right.



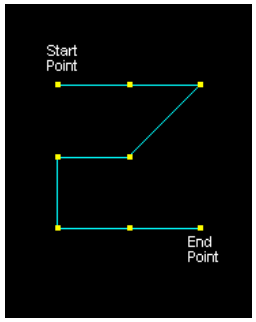
Lower Left



Lower Right

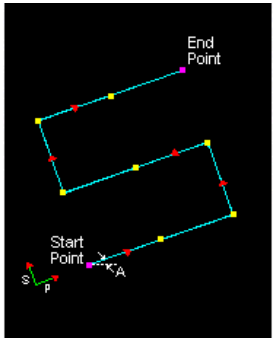


Upper Left

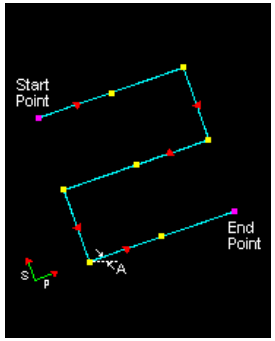


Upper Right

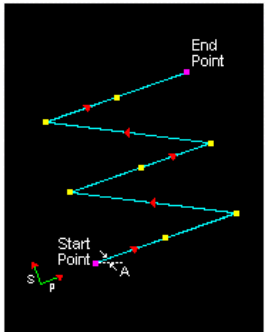
Directional Sort - The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort direction is defined by the Start angle. The Secondary Sort Direction is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or Zig Zag (two ways).



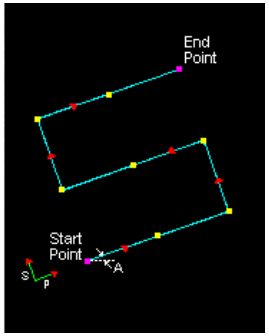
Low to High



High to Low



Zig Pattern



ZigZag Pattern

Clearance Plane can be set similar to any 2 ½ Axis machining operations.

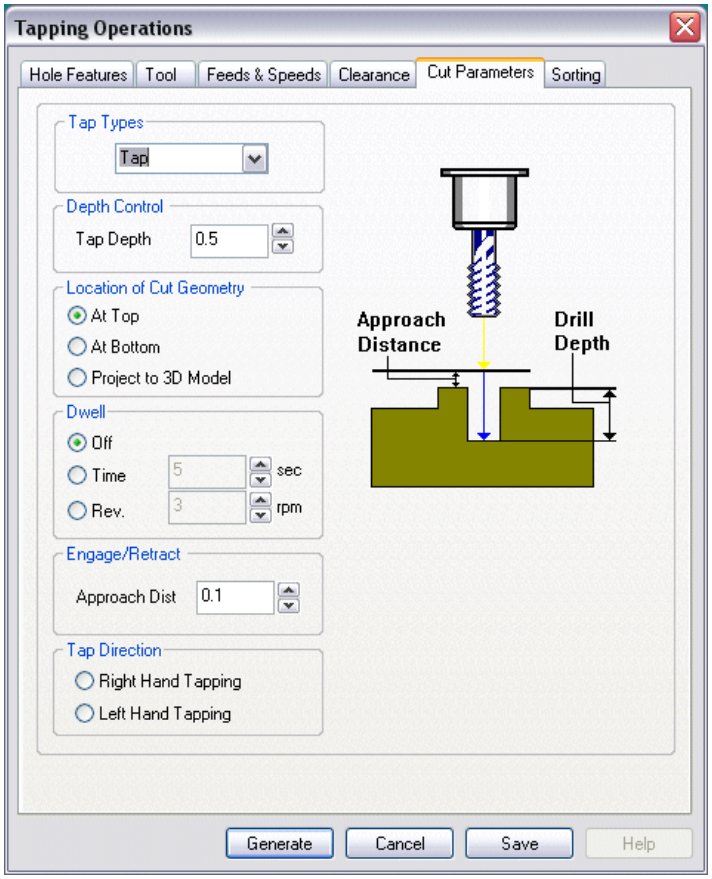
Tapping

Notes:

A Tap cycle is used to drill threaded holes in the part, clockwise or counter-clockwise.

Cut Parameters

The Tap parameter settings are similar to the Drill parameter settings.



Just as in the drilling operation, User defined Tap cycles can be created by choosing the Tap1 or Tap2 cycle type. Additionally the user can set the direction of tap to be either Right handed or Left Handed.

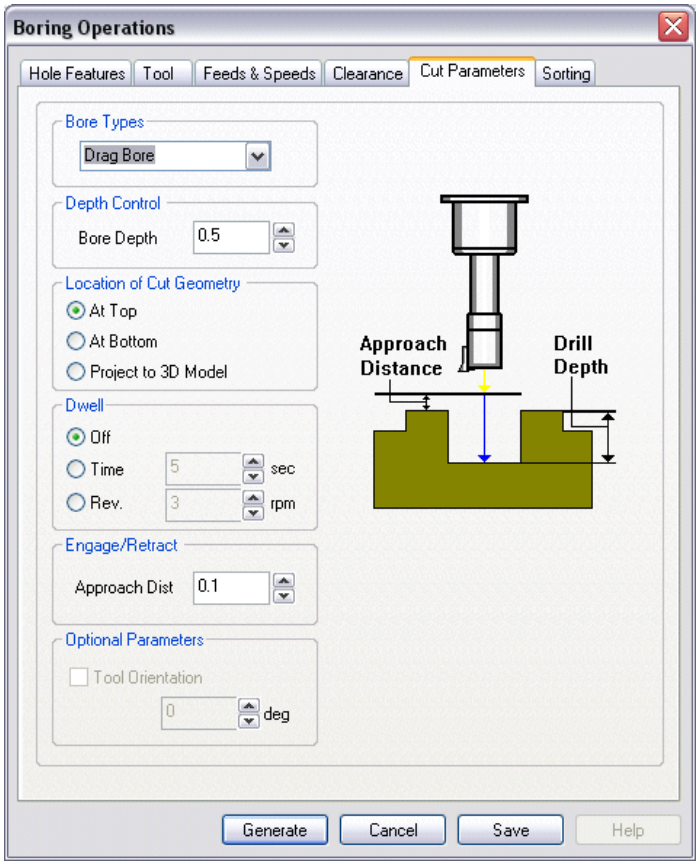
Note: Sorting, Hole Features parameters can be set similar to Drilling Operation.

Notes:

Boring

Cut Parameters

A Bore cycle is used to form shapes inside a hole. The following boring cycles are available:



- **Drag**: The tool is fed to the specified depth at the controlled feed rate. Then the spindle is stopped and the tool retracts rapidly.
- **No Drag**: The tool is fed to the specified depth at the controlled feed rate. It is then stopped to orient the spindle, moved away from the side of the hole and then retracted.

Notes:

- **Manual**: The tool traverses to the programmed point and is fed to the specified depth at the controlled feed rate. Then the tool stops and is retracted manually.

Note: Sorting, Hole Features parameters can be set similar to Drilling Operation.

Reverse Boring

This is just a Bore cycle in the reverse direction. The spindle is oriented to the specified angle and moves rapidly to the feed depth and moved to the part. The spindle is turned on and the cycle is started.

Cut Parameters, Hole Features and Sorting parameters are set similar to the Bore operation.

Class Room Exercise – Open Bitholder.3dm and create Drilling operations.

Notes:

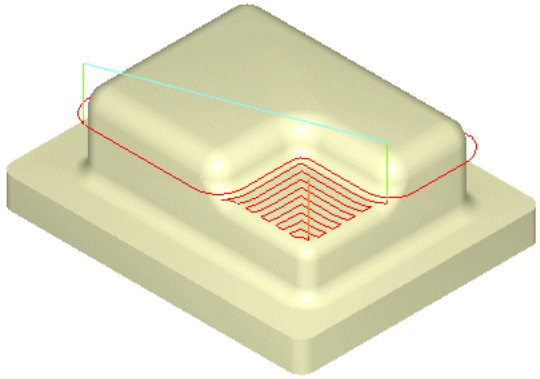
Chapter 7

Advanced 3 axis Toolpath Methods

These toolpath methods are available with the PRO configuration of the product.

Horizontal Re-Roughing

This is used to create toolpaths in areas that were not machined by previous operations. Unmachined areas are determined by comparing the part to the stock remaining after the previous operation. Machining is performed in constant Z levels, one of which is shown below:



This ensures that successive roughing operations do not move the tool in areas that have already been machined thereby eliminating wasteful air motions. The way RhinoCAM determines these un-machined areas is by considering the cut model (result of the cut material simulation) corresponding to the previous operation as the stock model for the subsequent roughing operation. Machining is performed in constant Z levels as in horizontal roughing.

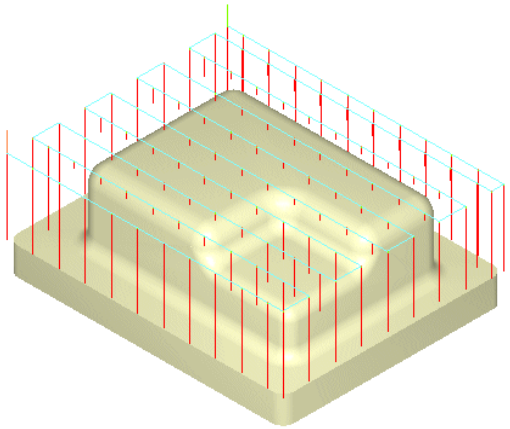
Note: Horizontal Re-Roughing takes into account the only the stock left until the previous operations.

The toolpath parameters are specified similar to Horizontal Roughing operation.

Notes:

Plunge Roughing

Also called drill roughing, the tool can cut in the Z direction only, not in X and Y. The tool makes a series of overlapping plunges to remove cylindrical plugs of material.



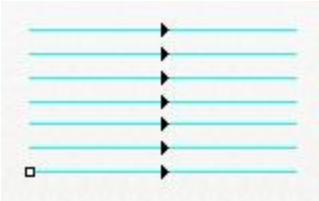
RhinoCAM 's plunge roughing algorithms can be used to create toolpaths that remove material by feeding only in Z and not in X and Y. The tool makes a series of overlapping, drilling-like plunges to remove one cylindrical plug of material after another with each pass. The user can control the spacing of these plunge motions as well as the federates of the plunge and traversal motions.

Cut Parameters

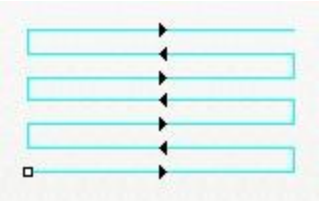
Global parameters are set similar to Horizontal Roughing operation.

The **cut direction** can be controlled by specifying either climb or conventional or mixed.

Climb



Conventional



Mixed

Notes:

The **start side** can be specified as from Top or Bottom

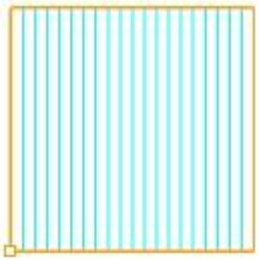
Start at Bottom

Start at Top

The **angle of cuts** can be specified as degrees (0-360).

Angle = 0

Angle = 45



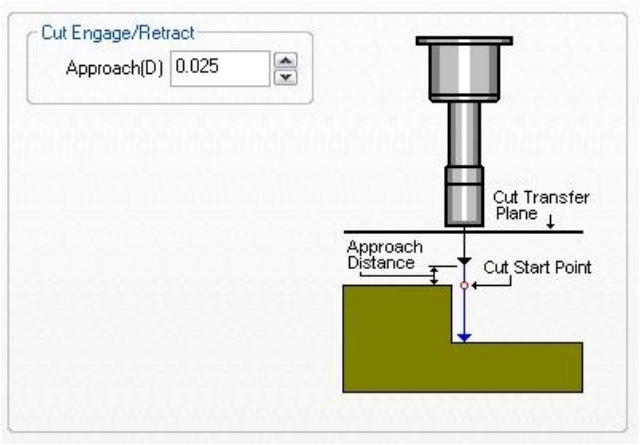
Angle = 90

The Stepover Control section allows the user to define the spacing between the cuts. The spacing can be specified either as a percentage of tool diameter, a specific distance or as the scallop height.

Engage /Retract

The Engage / Retract works similar to a drill operation. The user can specify the Approach Distance under Cut Engage/Retract parameters.

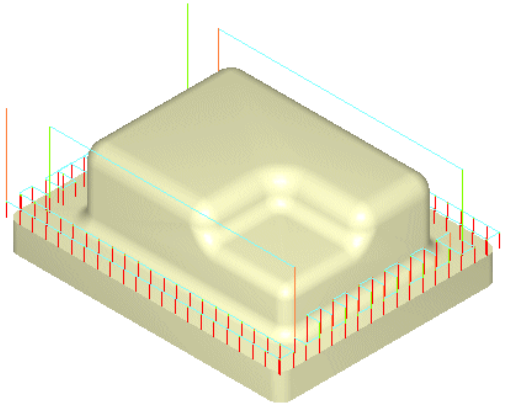
Notes:



Clearance Plane can be set similar to other 3 axis operations.

Plunge Re-Roughing

As in Horizontal Re-roughing, RhinoCAM uses the cut model of the previous operation as the stock model to determine the areas wherein to plunge cut. The tool traversal is controlled in the same manner as in Plunge Roughing.



Note: Plunge Re-Roughing takes into account the only the stock left until the previous operations.

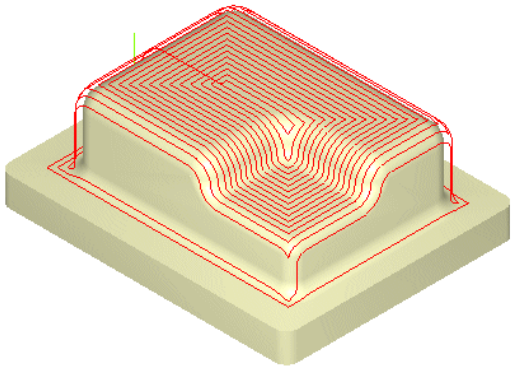
Cut Parameters are set similar to Plunge Roughing operation.

Notes:

Projection Pocketing

Notes:

Projection pocketing operation can be used for Pre finishing or fine finishing surface bottoms in closed regions. In this method, the toolpath is generated in 2D and then projected down to the surfaces below. Machining regions are necessary to be active for this cut method to work. There is no limitation on the number of regions or the number of nesting's of the region.

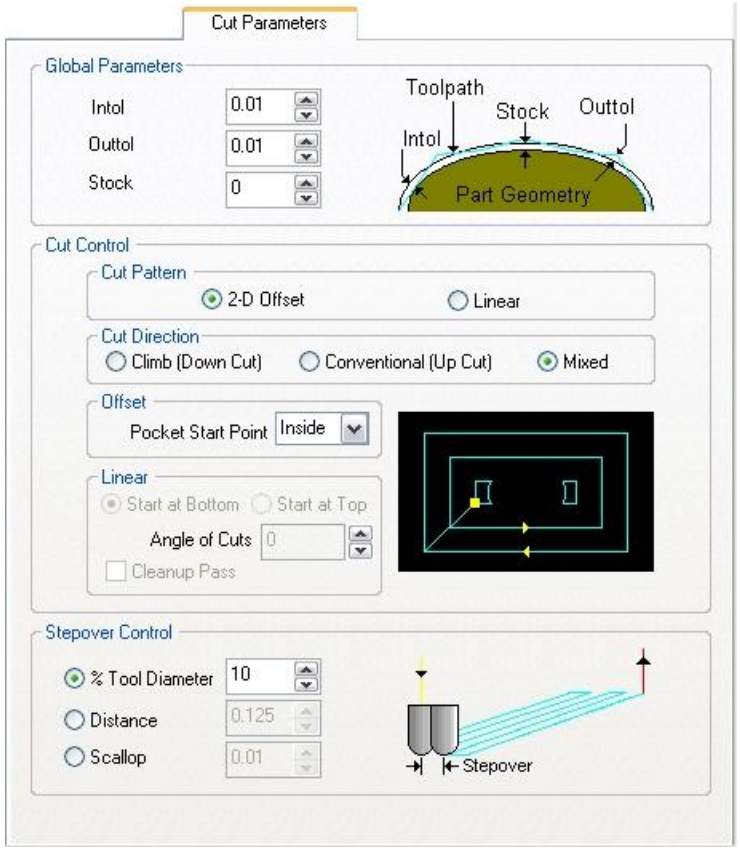


Cut Parameters

Global Parameters are set similar to other 3 axis operations.

Cut Control: The Cut Pattern section allows the user to define the type of cut pattern that the tool will follow when it is at each Z level. There are two types of cuts that the user can choose. The first one is an offset cut type (2D offset) and the second one is a linear cut type.

In the offset cut type, the cut regions are successively offset until there is nothing left to machine. This type of cutting is sometimes called spiral machining. The user can specify the cut start point to be either the inside of the cut regions or from the outside.



Cut Direction can be controlled by specifying either climb or conventional or mixed.

Pocket Start Point can be set to inside or outside. This is applicable for Part offset cut patterns.

Cleanup Pass parameter will cause automatic detection of all the corners that the tool could not reach between each pass. It will then add a toolpath based on the uncut area detected; either a linear cut in case of smaller areas or a cut that travels along the shape of the uncut area, when the area is large. This is applicable for Linear cut patterns.

The **Stepover Control** section allows the user to define the spacing between the cuts. The spacing can be specified either as a percentage of tool diameter, a specific distance or as the scallop height.

Entry/Exit are set similar to Parallel Finishing operation.

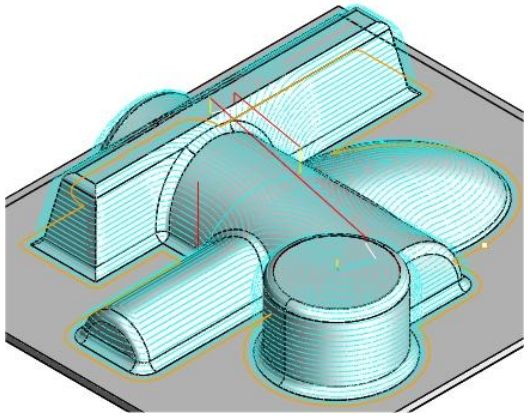
Notes:

Notes:

Advanced cut parameters and Clearance plane parameters can be set similar to 3 axis Horizontal Roughing.

3d Offset Pocketing

This method is used for pre-finishing and finishing of pockets with sculpted bottoms and/or sides. The pockets are defined by regions, and successive inner offsets of these outer regions are generated. The tool moves along these offset curves with a constant 3d step over while following the contours of the part below.



Cut Parameters

Global parameters (intol, outol & stock) are set similar to other 3 axis operations.

Cut Direction: Can be controlled by specifying either climb or conventional or mixed.

Pocket Start Point can be set to inside or outside. This is applicable for Part offset cut patterns.

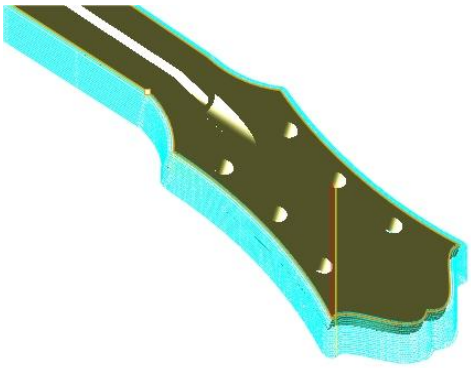
The **Stepover Control** section allows the user to define the spacing between the cuts. The spacing can be specified either as a percentage of tool diameter, a specific distance or as the scallop height.

Entry/Exit and Clearance plane parameters are set similar to Parallel Finishing operation.

3D Offset Profiling

Notes:

Suitable for machining isolated areas or shapes, this method machines along a curve. You must specify a curve as machining region, direction and cut pattern. The tool simultaneously follows the region and the contours of the part below in constant 3d.



Cut Parameters

Cut Control:

The **Cut Type** section allows the user to define the type of cut pattern that the tool will follow. The two types of cut patterns that are available are ZigZag and Zig. In the ZigZag cut pattern, the cutter traverses back and from along the cuts. In the Zig cut pattern the cutter traverses the cuts in a single direction.

Cut **Start Side** could be along the curve within specified bandwidth either to the left, right or on both sides of the curve. It can also be set to move across the curve within specified band width either to the left, right or on both sides of the curve.

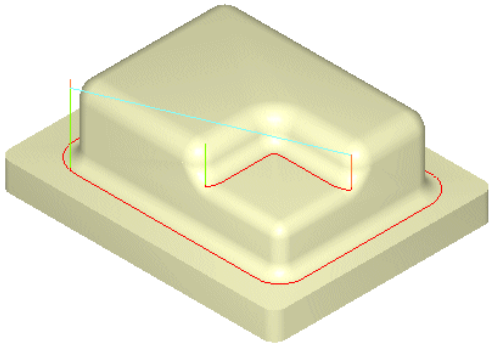
Left **Right** **Both Sides**

The **Stepover Control** section allows the user to define the spacing between the linear cuts in either the Zig or the ZigZag cut pattern. The spacing can be specified either as a percentage of tool diameter, a specific distance or as the scallop height. The stepover is computed by the cutting bandwidth. This guarantees that the desired stepover will never be exceeded at any time during the cut operation.

Entry/Exit and Clearance plane parameters are set similar to Parallel Finishing operation.

Pencil Tracing

Used either for roughing, re-machining, or cleanup, the tool is driven along valleys and corners of the part. The system identifies all double contact or bi-tangency conditions based on the tool radius. It then creates cutting paths along these locations.



When used as a roughing operation, valleys and corners are relieved so that subsequent operations will not encounter large amounts of material in these regions, thereby reducing tool deflection and wear. When used as a cleanup operation, scallops that remain after finishing operations are removed.

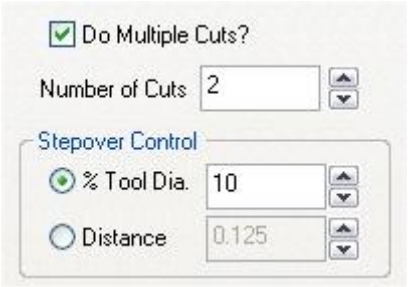
Cut Parameters

Global parameters (intol, outol & stock) are set similar to other 3 axis operations.

Cut Direction of the toolpath can be specified either as Climb or Conventional by selecting the appropriate radio buttons.

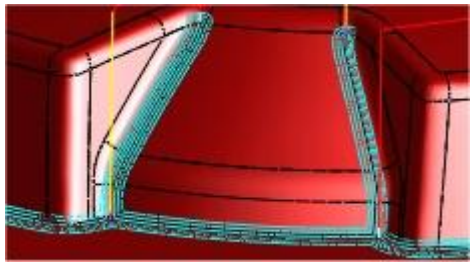
Notes:

Notes:



Multiple Cuts: Allows the user to 2 or more cut passes and the distance between the cuts can be specified using Stepover control. Specifying the Number of cuts as 2 results in 2 cuts on both sides of the pencil trace.

Single Cut



Multiple Cuts (3 cuts)

Advanced Cut Parameters

This section of the dialog is used to control the cuts based on the slope of the cuts.

Split Cuts at Angle - If the user checks this then the pencil trace cuts can be controlled based on the subsequently input slope angle based parameters. If the user does not check this then all of the computed pencil trace cuts will be output. The user can then specify the following parameters

Cuts Split Angle - The cuts will be examined and split at this specified angle. This angle is computed with respect to the XY plane

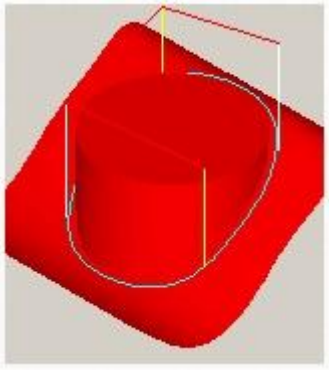
Cut Split Limiting Length - In some cases the split angle could cause small cut segments to be output. This can happen when the pencil trace cuts trajectories bounce up and down the specified cuts split angle. If the split segment is below a certain length, the user might not want to split the curve at that location. This parameter controls this limiting length of the cut

Notes:

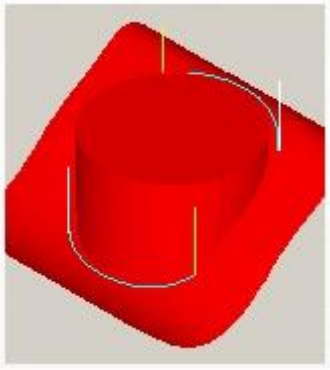
Output Flat & Steep Cuts - Allows the output of both the flat and steep cuts after the splitting is done at the split angle location

Output Flat Cuts Only - Allows the output of the flat cuts only after splitting

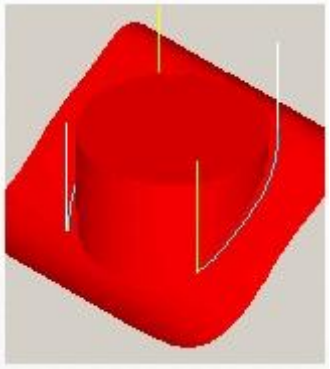
Output Steep Cuts Only - Allows the output of only the steep cuts after splitting



Flat & Steep Cuts

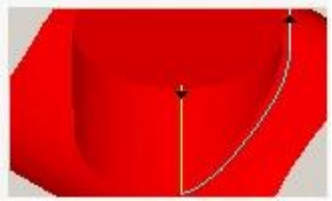


Flat Cuts Only

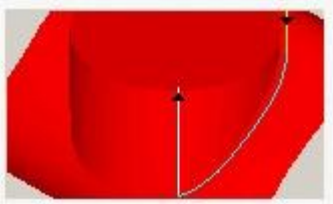


Steep Cuts Only

Steep Cuts Direction: The user can also control the direction of the steep cuts by choosing either Ascending or Descending. In the former case all steep cuts will be ordered such that the cutter goes from low Z value to high while in the other case in the opposite direction.



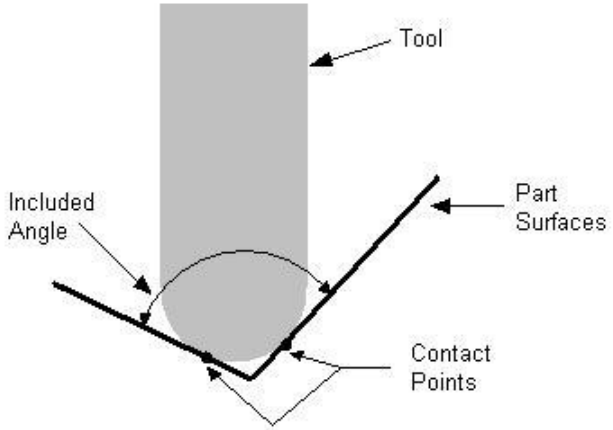
Ascending



Descending

Additionally the user can chose to ignore cuts steeper than a certain angle. This is usually done to avoid cutting very steep vertical areas in the part. The default value is 90 degrees, which means all cuts will be output. The user needs to specify something lower than this if he wants to prevent cutting very steep areas. This angle is also computed with respect to the XY plane.

The user can also control the output of cuts based on the included angle. The included angle is the angle between the two tangent lines at the bi-tangency points of the pencil traces. Note that the pencil trace cuts are the locations of all points where the tool is contacting the part-geometry in two or more tangential positions. Thus, the larger this angle the shallower this area. This angle has a range from 90 degrees to 180 degrees. At 90 degrees the tool is contacting at the side of a vertical wall. Close to 180 degrees the configuration of the two surfaces are quite shallow. The picture below shows how the included angle is computed.



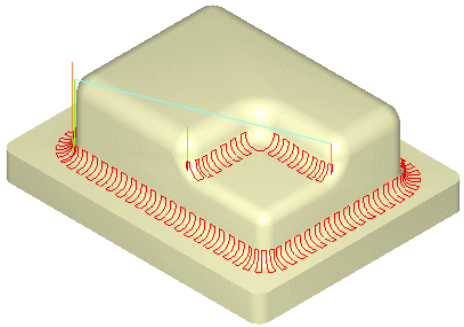
Entry/Exit and Clearance plane parameters are set similar to Parallel Finishing operation.

Notes:

Valley Remachining

Notes:

Valley Re-machining is a toolpath method that can be used as a re-finishing operation to clean up regions in a part that could not be reached by a larger tool. Such areas would typically be found in the valleys and corners of the part. In this toolpath method the user specifies the previous or reference tool diameter and a tool that is smaller than this tool.



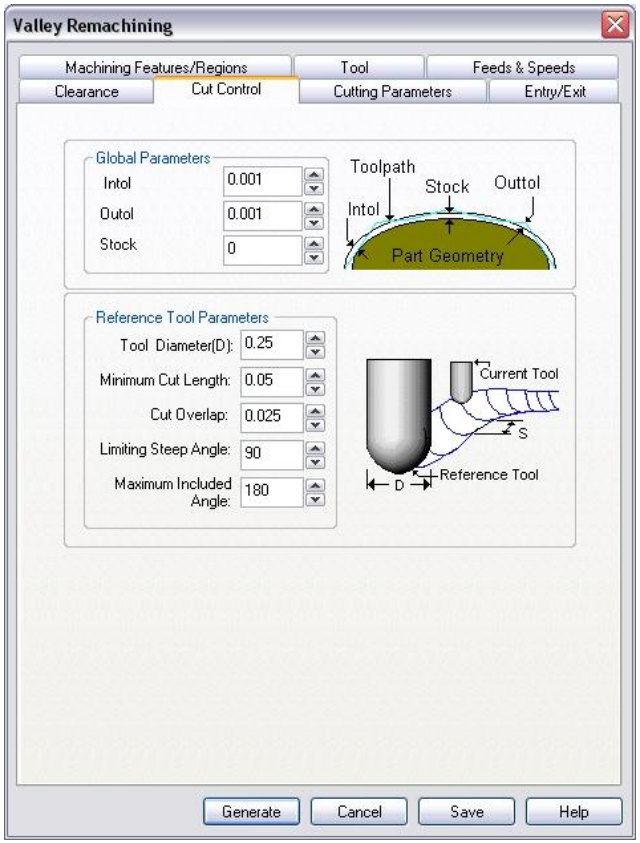
The system computes all areas that would are inaccessible to this reference tool and applies the smaller tool to machine only in these areas. This is a very efficient method to remove remaining material from valleys and corners of a part, significantly reducing bench-work on the part.

Cut Control

The Global parameters are set similar to other 3 axis machining operations.

The **reference tool parameters** section allows the user to specify the reference tool diameter. This diameter must be larger than the current tool diameter.

Additionally the user can chose to ignore cuts steeper than a certain angle. This is usually done to avoid cutting very steep vertical areas in the part. The default value is 90 degrees, which means all cuts will be output. The user needs to specify something lower than this if he wants to prevent cutting very steep areas. This angle is also computed with respect to the XY plane.



The user can also control the output of cuts based on the included angle. The included angle is the angle between the two tangent lines at the bi-tangency points of the pencil traces. Note that the re-machining cuts are the locations of all points where the tool is contacting the part-geometry in two or more tangential positions. Thus, the larger this angle the shallower this area. This angle has a range from 90 degrees to 180 degrees. At 90 degrees the tool is contacting at the side of a vertical wall. Close to 180 degrees the configuration of the two surfaces are quite shallow.

Cut Parameters

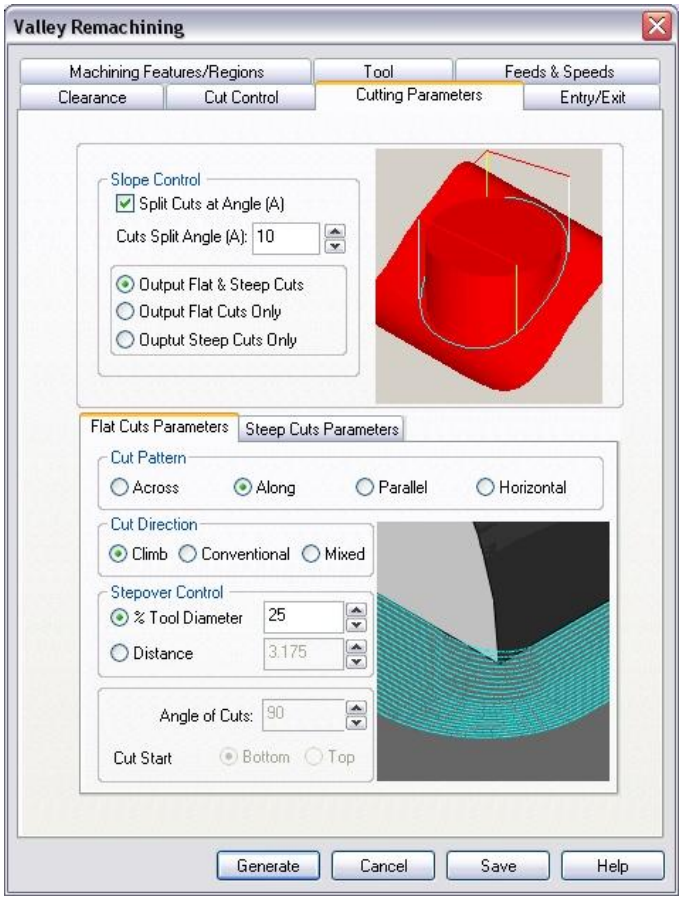
This section of the dialog is used to control the cuts based on the slope of the cuts.

Notes:

Split Cuts at Angle - If the user checks this then the remachining cuts can be controlled based on the subsequently input slope angle based parameters. If the user does not check this then all of the computed pencil trace cuts will be output. The user can then specify the following parameters

Cuts Split Angle - The cuts will be examined and split at this specified angle. This angle is computed with respect to the XY plane

Cut Split Limiting Length - In some cases the split angle could cause small cut segments to be output. This can happen when the pencil trace cuts trajectories bounce up and down the specified cuts split angle. If the split segment is below a certain length, the user might not want to split the curve at that location. This parameter controls this limiting length of the cut



Notes:

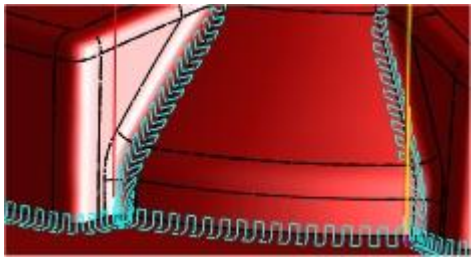
Output Flat & Steep Cuts - Allows the output of both the flat and steep cuts after the splitting is done at the split angle location

Output Flat Cuts Only - Allows the output of the flat cuts only after splitting

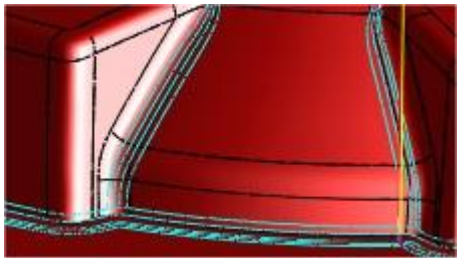
Output Steep Cuts Only - Allows the output of only the steep cuts after splitting

Cut Pattern: The cut patterns specified as Along, Across, Parallel or Horizontal. User can specify the step over for the above cut pattern.

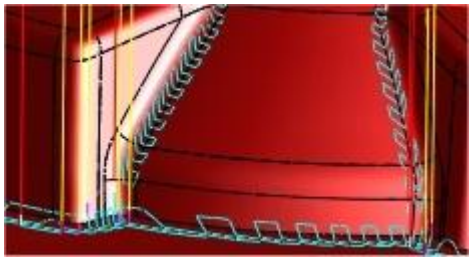
With Parallel cut pattern, the user can specify the Angle of Cut and the cut start as bottom or top.



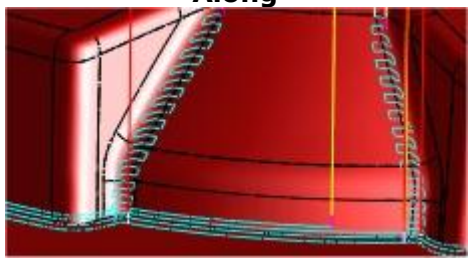
Across



Along



Parallel



Horizontal

Cut Direction can be set as Climb, Conventional or Mixed

The cut patterns for Flat and Steep cuts can be independently specified as Along, Across, Parallel or Horizontal

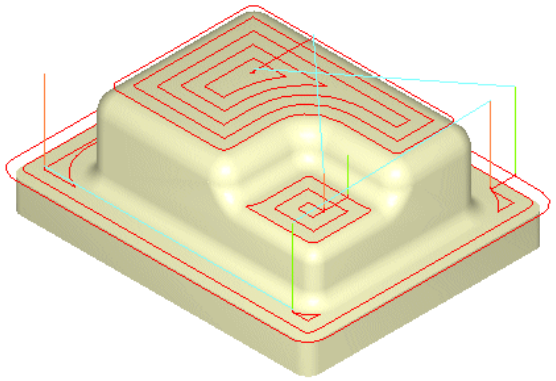
Entry/Exit and Clearance plane parameters are set similar to Parallel Finishing operation.

Class Room Exercise – Open 3Axis_Example2.3dm and create Pencil Tracing & Valley Remachining operations.

Notes:

Plateau Machining

Plateau Machining is a toolpath method which can be used either as a finishing or re-finishing operation. In this cut method, the cutter is restricted to machine areas in the part that are shallower than a user specified angle from the horizontal XY plane. Once the system identifies these regions then a Contour Finish type of machining is employed to machine these regions. This type of machining is used to finish or re-finish areas that were not machined completely by a Horizontal Finishing operation.



Cut Parameters

Global parameters are set similar to other 3 axis operations.

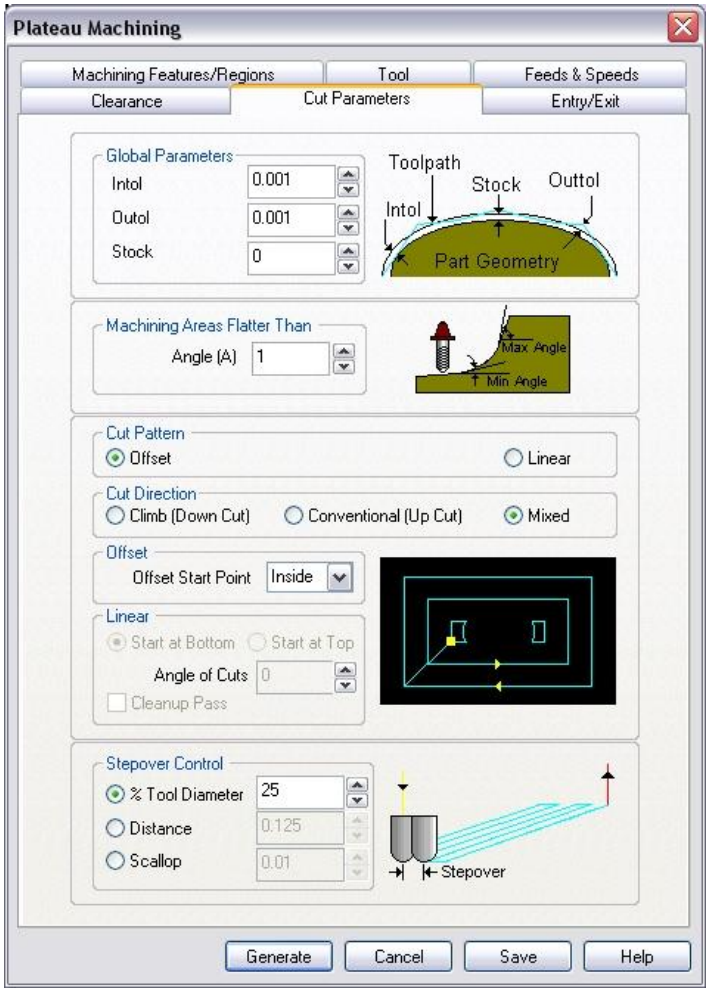
Machining Areas Flatter Than - allows the user to define the plateau angle. This angle controls which regions will be machined. The system identifies all areas of the part that are shallower than this angle from the horizontal and creates machining regions that are used in the subsequent machining process.

The **Cut Pattern** section allows the user to define the type of cut pattern that the tool will follow when it is at each Z level. There are two types of cuts that the user can choose. The first one is an offset cut type (2D offset) and the second one is a linear cut type.

In the offset cut type, the cut regions are successively offset until there is nothing left to machine. This type of cutting is sometimes called spiral machining. The user can specify the cut start point to be either the inside of the cut regions or from the outside.

Cut Direction can be controlled by specifying either climb or conventional or mixed.

Notes:



Pocket Start Point can be set to inside or outside. This is applicable for Part offset cut patterns.

Checking the **Cleanup Pass** parameter will cause automatic detection of all the corners that the tool could not reach between each pass. It will then add a toolpath based on the uncut area detected; either a linear cut in case of smaller areas or a cut that travels along the shape of the uncut area, when the area is large. This is applicable for Linear cut patterns.

The **Stepover Control** section allows the user to define the spacing between the cuts. The spacing can be specified either as a percentage of tool diameter, a specific distance or as the scallop height.

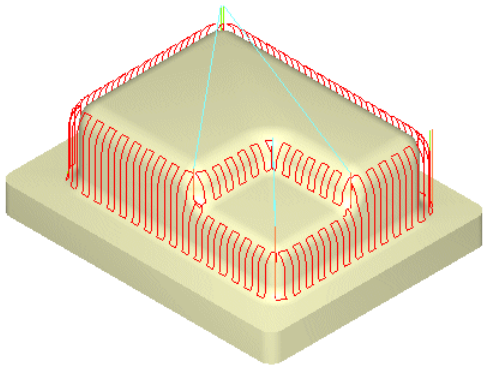
Notes:

Entry/Exit and Clearance plane parameters are set similar to Parallel Finishing operation.

Notes:

Parallel Hill Machining

Steep Contour Re-finishing is a toolpath method, which can be used as a re-finishing operation that is applied after a Contour Finish operation. In this cut method, the cutter is restricted to machine areas in the part that are steeper than a user specified angle from the horizontal XY plane. Once the system identifies these regions then a Contour Finish type of machining is employed to machine these regions. Additionally the system automatically chooses between two cut angles to machine these areas so as to maintain proper scallop height control on the part. One of these two angles is the cut angle used in the previous Contour Machining operation (this is input by the user) and the other angle is 90 degrees to this angle.



Optionally, the user can specify to cut only those regions that were not properly machined by the previous Contour Machining operation. These regions would be steep areas in the part that were parallel to the cutting angle of the previous operation. In this case the toolpath will only be created in the cut direction that is 90 degrees to the previous cut angle.

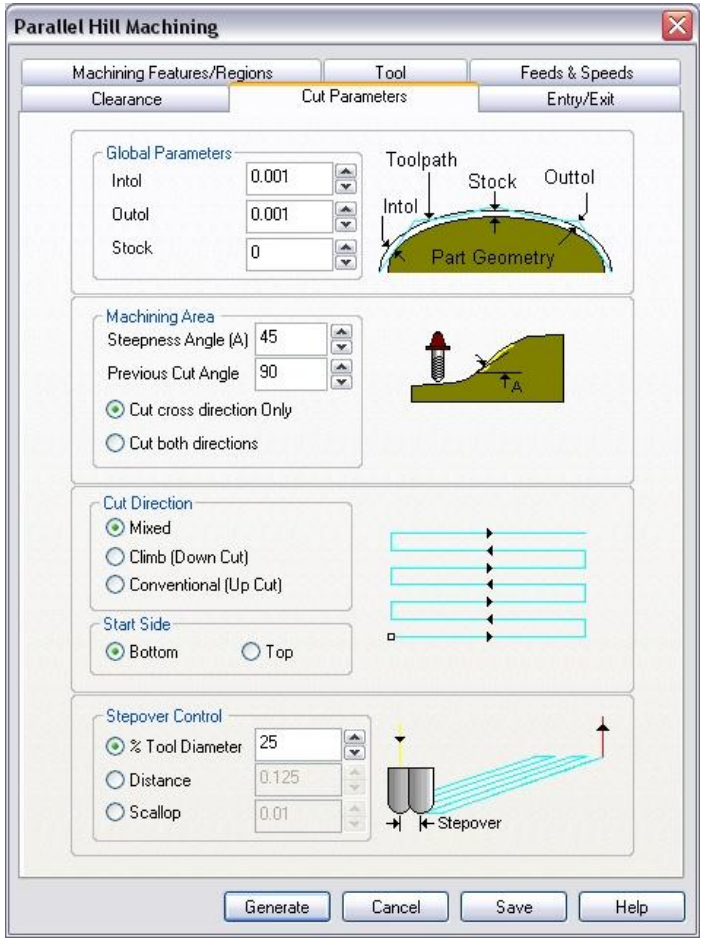
Cut Parameters

Global parameters are set similar to other 3 axis operations.

The **Machining Area** section allows the user to define the steepness angle. This angle controls which regions will be machined. The system identifies all areas of the part that are steeper than this angle from the horizontal and creates

machining regions that are used in the subsequent machining process. It additionally alters these regions based on whether the user wants to cut only in the cross direction of the previous contouring operation or in both directions. In the former case the cut angle that will be used in this operation will be at 90 degrees to the previous cut angle. In the latter case the a cut angle along and a cut angle 90 degrees to the previous cut angle will also be used to machine the detected steep regions.

Notes:



The **cut direction** can be controlled by specifying either climb or conventional or mixed.

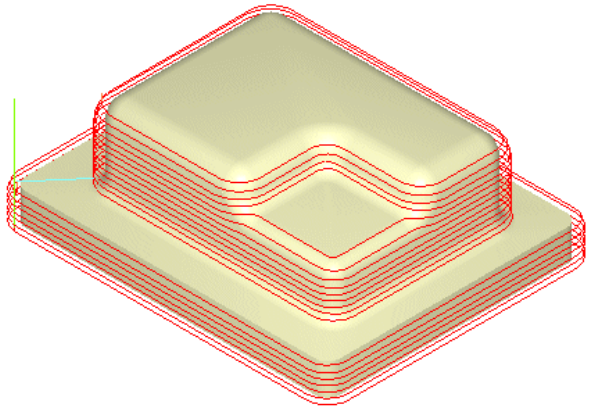
The **start side** can be specified as from Top or Bottom

The Stepover Control section allows the user to define the spacing between the cuts. The spacing can be specified either as a percentage of tool diameter, a specific distance or as the scallop height.

Entry/Exit and Clearance plane parameters are set similar to Parallel Finishing operation.

Horizontal Hill Machining

Steep Horizontal Re-finishing is a toolpath method, which can be used as a re-finishing operation that is applied after a Contour Finish operation. In this cut method, the cutter is restricted to machine areas in the part that are steeper than a user specified angle from the horizontal XY plane. Once the system identifies these regions then a Horizontal Finish type of machining is employed to machine these regions.



Optionally, the user can specify to cut only those regions that were not properly machined by the previous Contour Machining operation. These regions would be steep areas in the part that were parallel to the cutting angle of the previous operation. In this case the toolpath will only be created in the regions that are steeper than the user defined angle and parallel to the previous cut direction.

Cut Parameters

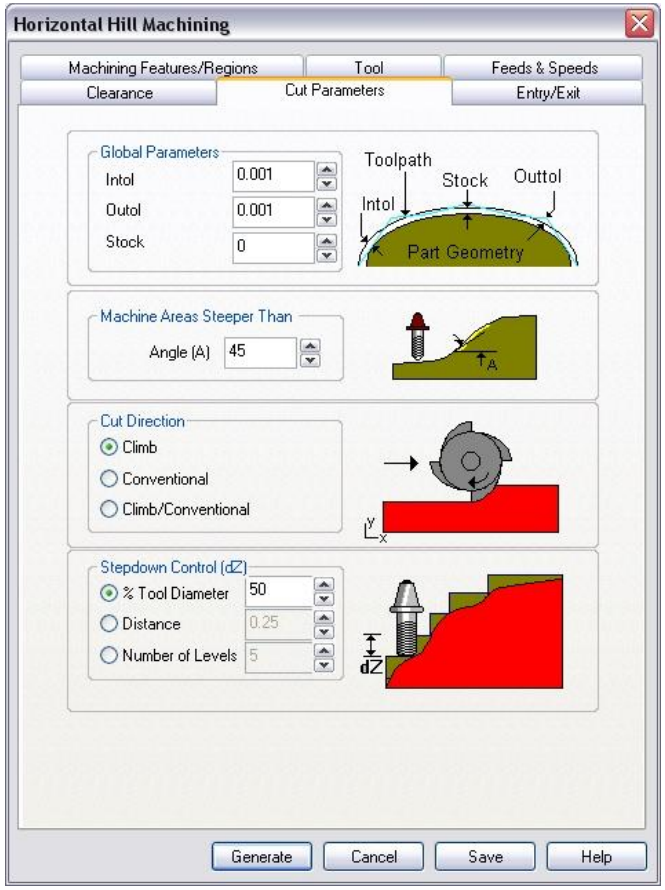
Global parameters are set similar to other 3 axis operations.

The **Machining Areas Steeper Than** control section allows the user to define the steepness angle. This angle controls which regions will be machined. The system identifies all areas of the part that are steeper than this angle from the horizontal and creates machining regions that are used in the subsequent machining process. It additionally

Notes:

alters these regions if the user wants to cut only in the cross direction of the previous contouring operation and applies the horizontal cutting toolpath.

Notes:



The **Cut Direction** of the toolpath can be specified either as Climb, Conventional or Climb/Conventional by selecting the appropriate radio buttons. As mentioned before, in climb or conventional, the direction of cutting is maintained so as the corresponding cutting condition is maintained on the part. In the Climb/Conventional type of machining however, the direction of cutting is alternated between each parallel plane.

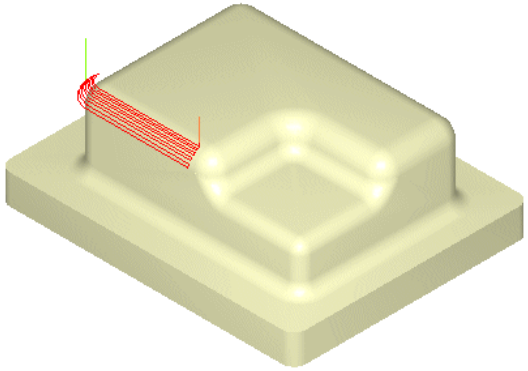
The Stepdown Control section allows the user to define the spacing between the horizontal cut levels for the roughing operation. The spacing can be specified either as a percentage of tool diameter, a specific distance or as the total number of levels desired.

Entry/Exit and Clearance plane parameters are set similar to Parallel Finishing operation.

Notes:

Curve Machining

Suitable for machining isolated areas or shapes, this method machines along a curve. You must specify one or more machining regions, direction and cut pattern. The tool simultaneously follows the region and the contours of the part below.

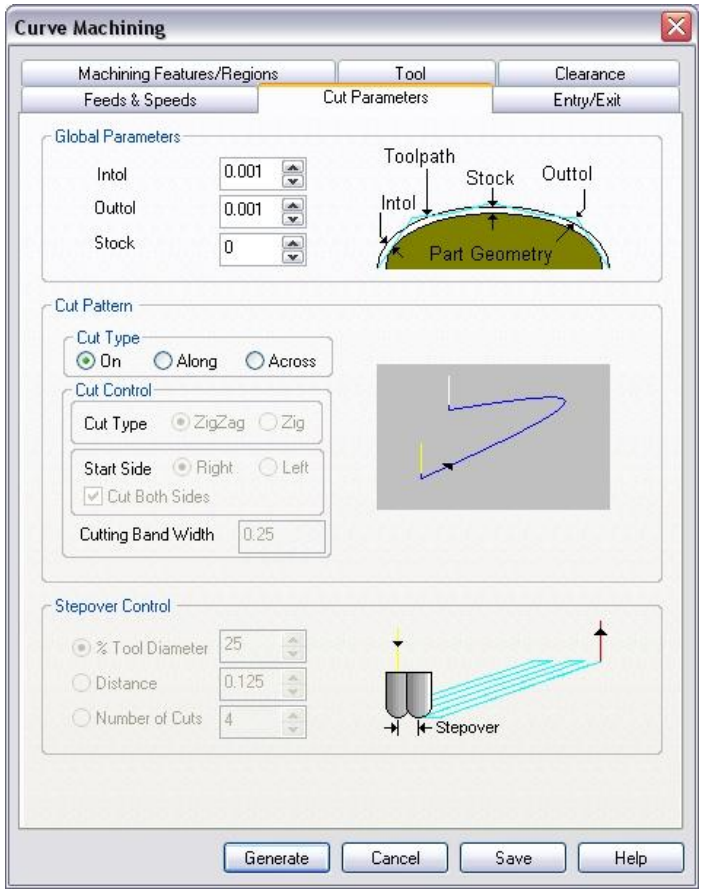


In this method, the toolpath can be generated on the curve, along the curve or across the curve. Machining regions are necessary to be active for this cut method to work. There is no limitation on the number of regions or the number of nestlings of the region.

As the cutter follows these linear cuts, it can either form a Zig or ZigZag cut pattern. In the Zig cut pattern, the cutter always goes in a constant direction while in the ZigZag cut pattern, the cutting direction alternates between two successive linear cuts. The cutter can be made to follow either or both sides of the curve.

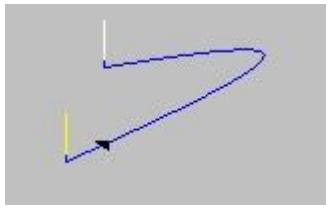
Cut Parameters

Global parameters are specified similar to other 3 axis operations.

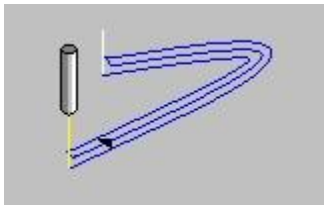


Cut Pattern:

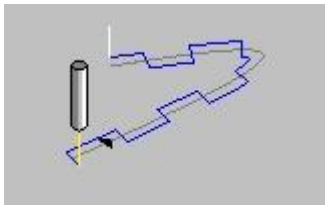
The **Cut Type** can be specified to be one of the following three types. It can be exactly on the curve; or it could be along the curve within specified bandwidth either to the left, right or on both sides of the curve. It can also be set to move across the curve within specified band width either to the left, right or on both sides of the curve.



On



Along



Across

Notes:

Notes:

The **Cut Control** section allows the user to define the type of cut pattern that the tool will follow. The two types of cut patterns that are available are ZigZag and Zig. In the ZigZag cut pattern, the cutter traverses back and from along the cuts. In the Zig cut pattern the cutter traverses the cuts in a single direction. In the ZigZag mode of traversal the user can choose to have the cutter start at the outside of the regions or from the inside. In the Zig traversal, the user can control the cut direction as either to start on the outside and go inwards or vice versa.

Cut **Start Side** could be along the curve within specified bandwidth either to the left, right or on both sides of the curve. It can also be set to move across the curve within specified band width either to the left, right or on both sides of the curve.

Left

Right

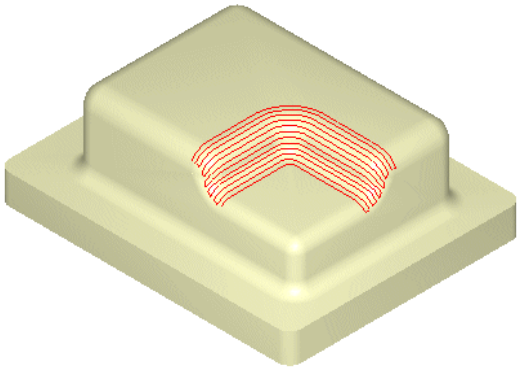
Both Sides

The **Stepover Control** section allows the user to define the spacing between the linear cuts in either the Zig or the ZigZag cut pattern. The spacing can be specified either as a percentage of tool diameter, a specific distance or as the scallop height. The stepover is computed by the cutting bandwidth. This guarantees that the desired stepover will never be exceeded at any time during the cut operation.

Entry/Exit and Clearance plane parameters are set similar to Parallel Finishing operation.

Between 2 Curves Machining

Sometimes called flowline machining, this method machines between two open or closed curves. Using a pattern either parallel or normal to the curves, the toolpath makes a gradual transition from one curve to the other. This creates a blended toolpath that can be used to efficiently finish complex shapes.



The created toolpath will make a gradual transition from one curve to the other depending on the geometric form of the two curves.

Machining Features / Regions

The machining regions for Between 2 Curves machining need to be 2 curves only.

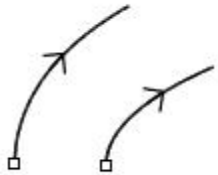
The 2 curves

- Need to be either open or closed.
- User cannot select an open curve and the other as closed.
- Need to be in the same direction
- Start point needs to be at the same location on both the curves.

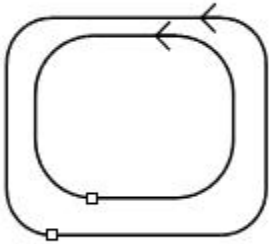
Following types of curves can be used as Machining Regions

Notes:

Notes:

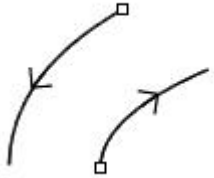


Open curves- same direction

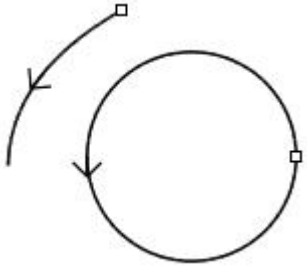


Closed Curves same direction and start points near each other

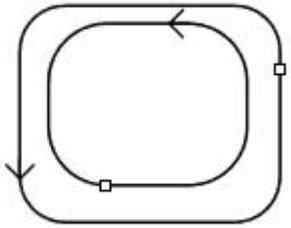
The following types of selections **cannot** be used as Machining Regions



Open curve – opposite directions



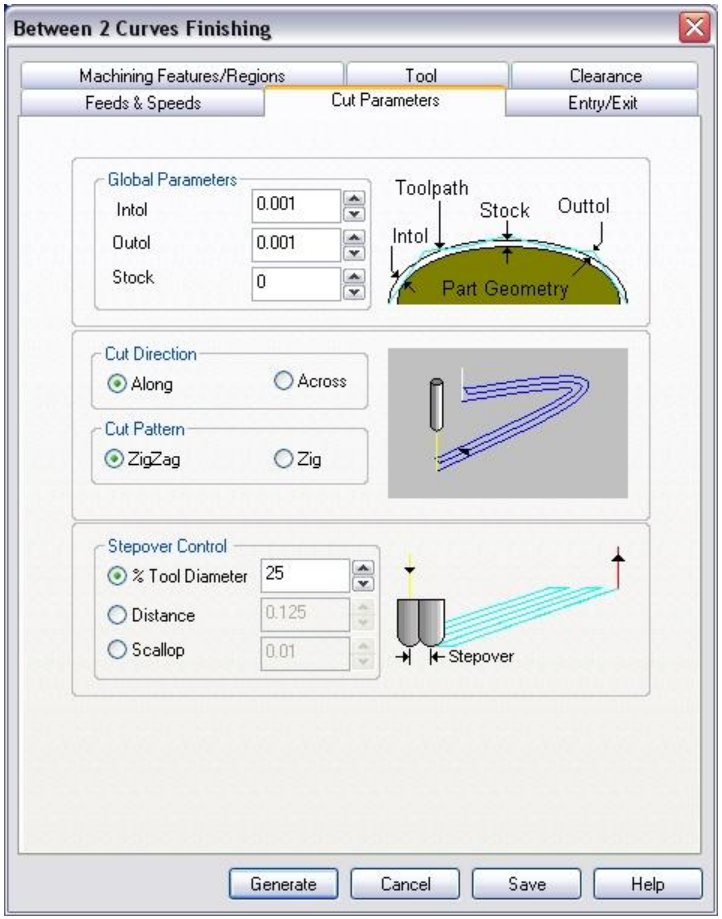
Open and closed curve



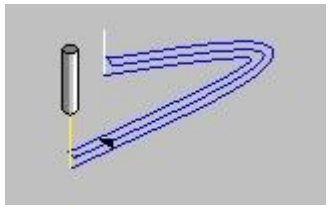
Closed curve start points not near

Cut Parameters

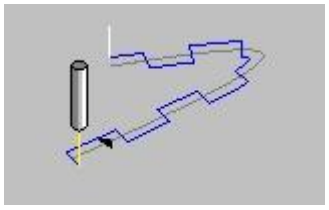
Global parameters are specified similar to other 3 axis operations.



Cut Direction can be specified as along or across



Along

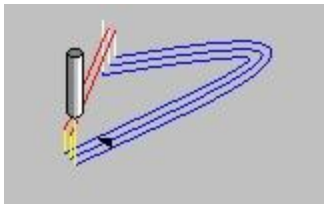


Across

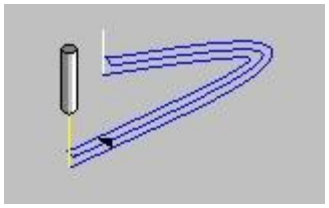
Notes:

Notes:

The **Cut Pattern** section allows the user to define the type of cut pattern that the tool will follow. The two types of cut patterns that are available are ZigZag and Zig. In the ZigZag cut pattern, the cutter traverses back and from along the cuts. In the Zig cut pattern the cutter traverses the cuts in a single direction.



Zig



Zig Zag

The **Stepover Control** section allows the user to define the spacing between the cuts. The spacing can be specified either as a percentage of tool diameter, a specific distance or as the scallop height.

Entry/Exit and Clearance plane parameters are set similar to Parallel Finishing operation.

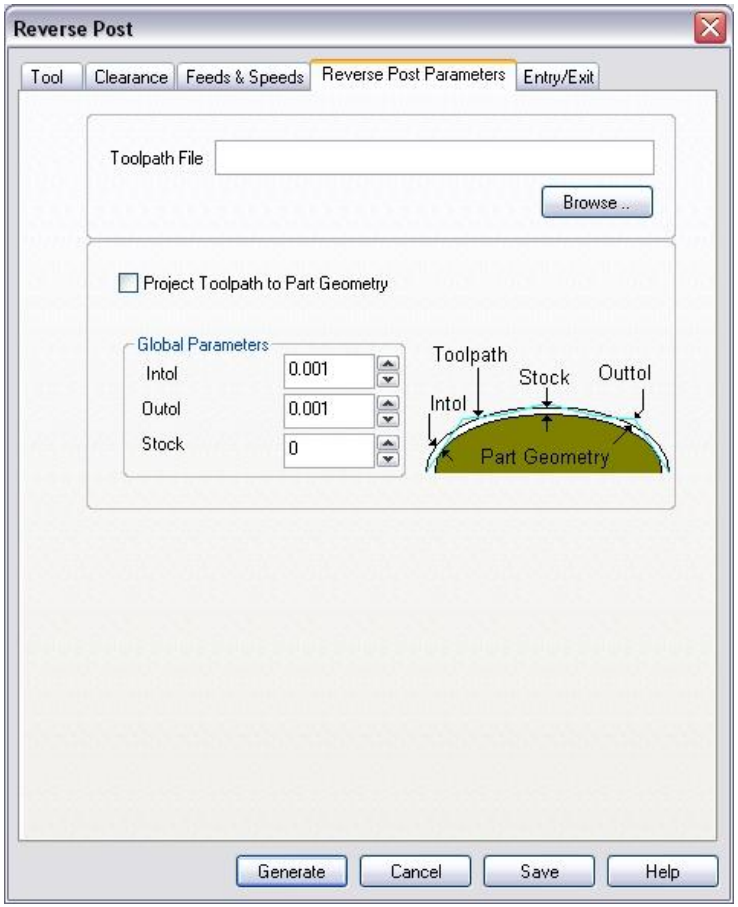
Reverse Post Milling

The Reverse Post Milling operation can be used to create machining operations with existing CLS files or ISO-G code files. The user can simply load these files in or use the loaded toolpath and project the cutter down to surfaces below.

Cut Parameters

The user can load a toolpath file using the edit field in the top of this dialog. The Browse button allows the user to browse for the toolpath files. Two types of files can be reverse posted. One is standard APT CL text files. These files should have a .cls or .apt extension on them. The other type is ISO standard G Code file. This file can have any extension other than the ones used for APT CL files.

The user can choose to project the toolpath to part geometry below by checking the check box.



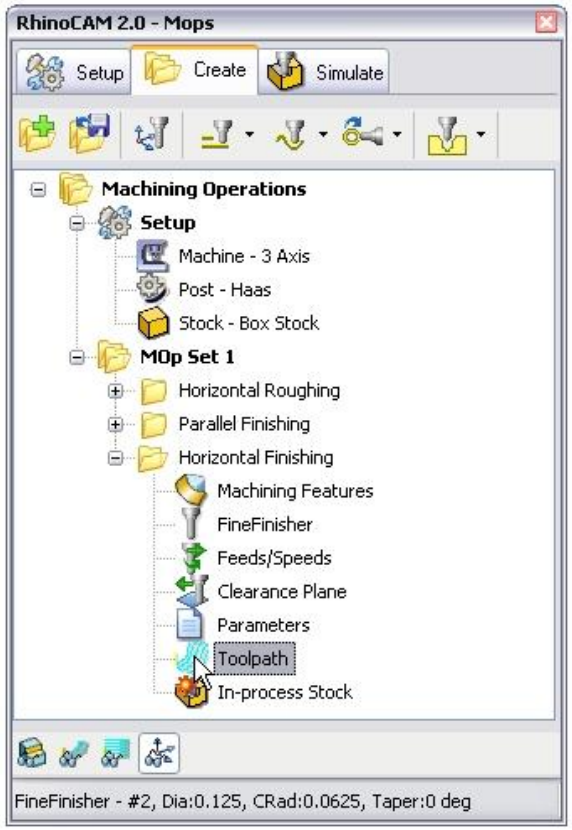
Entry/Exit and Clearance plane parameters are set similar to Parallel Finishing operation.

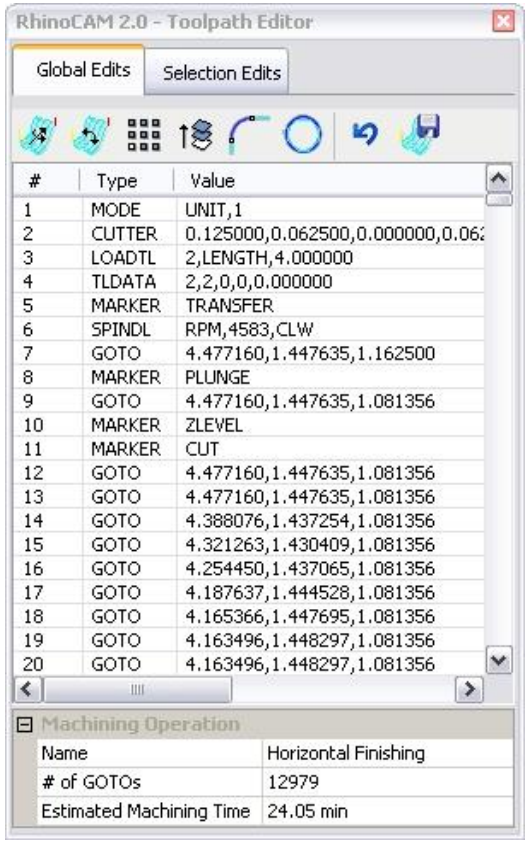
Notes:

Chapter 8

Toolpath Editor

Once a machining operation is created, the toolpath can be edited. To bring up the toolpath editor, the user double clicks on the toolpath icon of the operation in the Browser. The toolpath editor is a dockable dialog bar that will be initially docked to the left hand side of the frame window of RhinoCAM. This is shown below.





Notes:

The dialog shows 2 tabs Global Edits and Selection Edits on the top with a list box that lists the tool motions that can be edited. Below this list box there is another list box that displays the operation name, the number of toolpath points as well as the estimated machining time. The user can remove individual tool motions from the tool path by selecting the desired lines.

Editing the toolpaths using the toolpath editor flags the Machining Operation folder to red indicating that the toolpaths have been edited graphically.

Global Edits

These edits apply the changes to the entire toolpath.

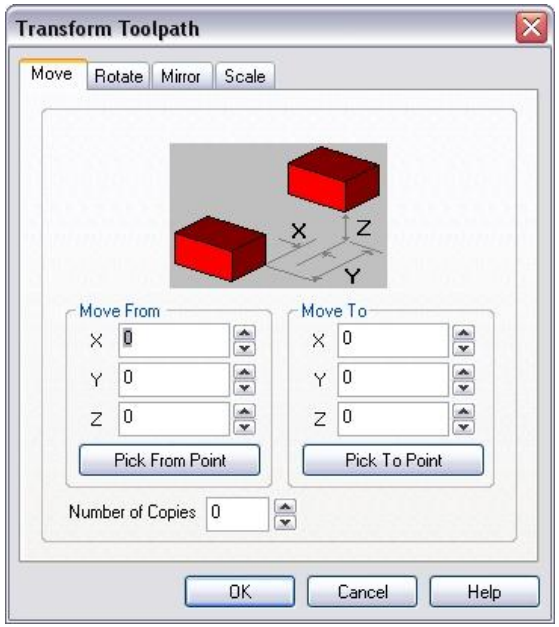
Transform Toolpath

Transform allows the user to Move, Rotate, Scale and Edit Toolpath.

Move Toolpath

Allows the user to translate the toolpath by specifying From and To points for the move. The user can either specify these points by coordinate entry in the corresponding edit fields or could graphically pick the points by selecting the pick buttons. Once the points are specified, selecting the apply button will perform the move operation

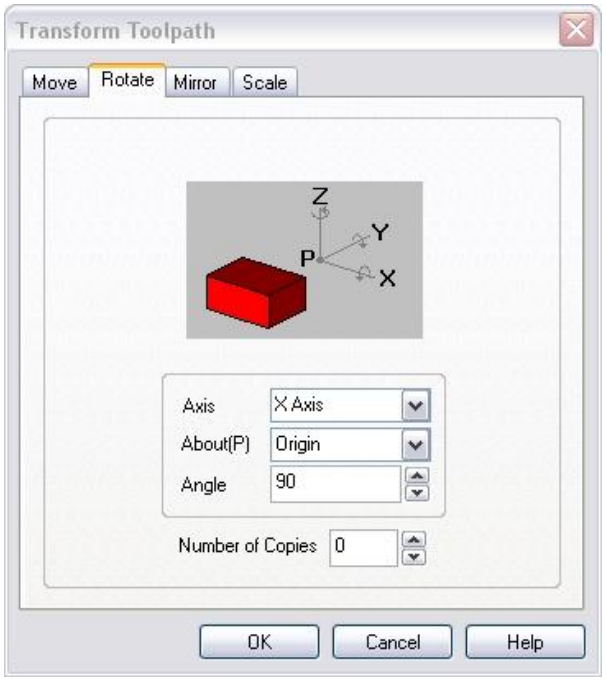
Notes:



Rotate Toolpath

Selecting the Rotate tab allows the user to rotate the toolpath. Rotation can be performed around any of the three principal axes. The angle of rotation as well as the point and Axis of rotation can be specified by the user. The point of rotation could be one of the origin, center of the part, minimum point of the part and the maximum point of the part. The Axis of rotation could be the X, Y or the Z Axis. The user can specify any arbitrary rotation angle.

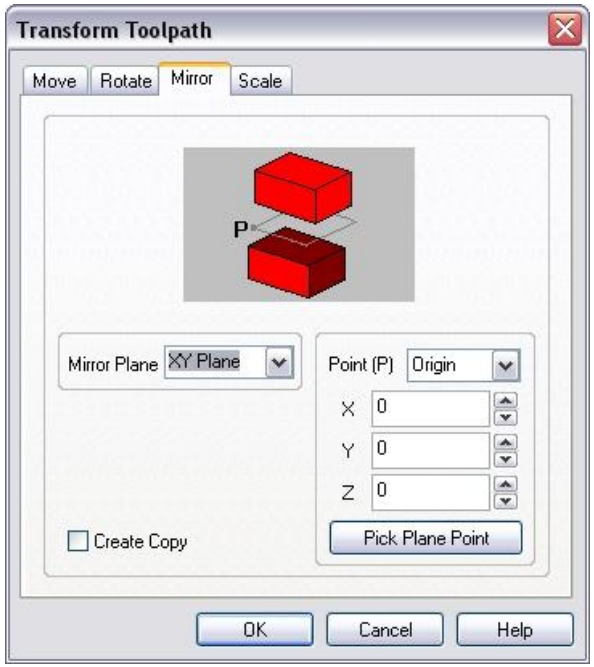
Notes:



Mirror Toolpath

Selecting the Mirror tab allows the user to Mirror the selected toolpath.

Notes:

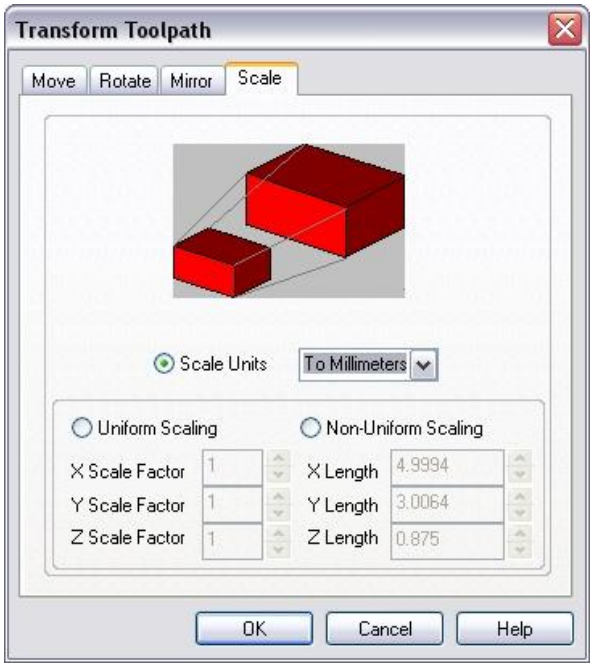


Notes:

Scale Toolpath

Selecting the scale tab allows the user to scale the selected toolpath. All scaling is performed about the zero of the coordinate axes. The user can optionally scale from mm to an inch or vice versa or by specifying a scale factor. The user can also scale the selections uniformly or perform non-uniform scaling in each of the three principal axes.

Notes:



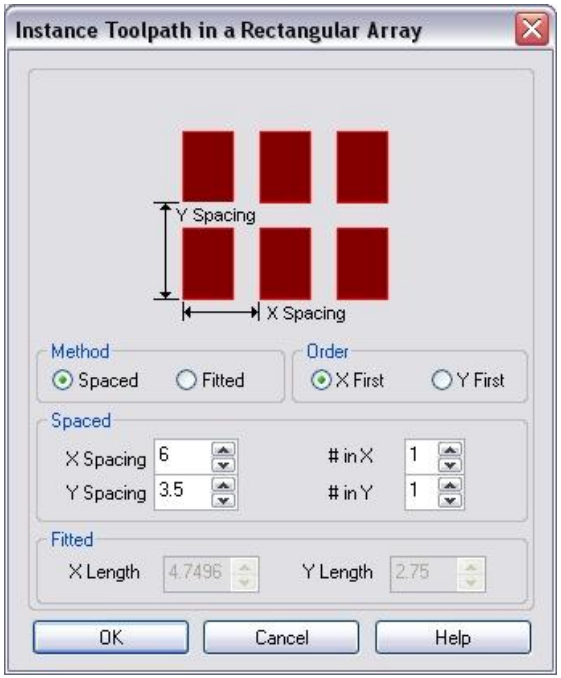
Reverse Toolpath

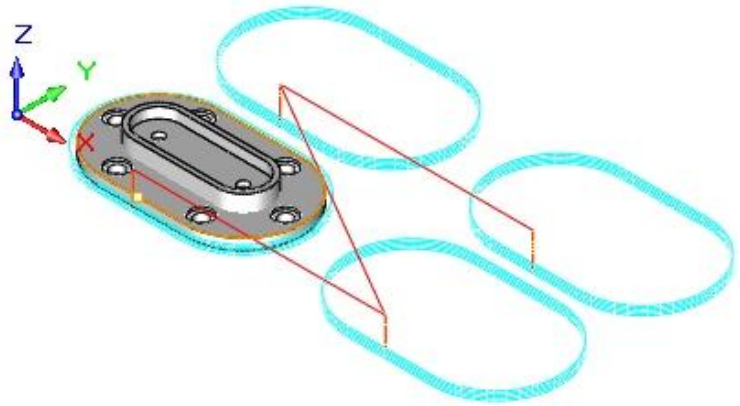
Allows reversal of direction of the toolpath. All engage motions are changed to retract motions and all retract motions are converted to engage motions.

Instance Toolpath

User can create multiple instances of the toolpath via this dialog. This can be useful in situations where the user is manufacturing multiple instances of the same part.

The user can specify multiple instances in the X and Y directions in two different ways. The first method, or the spaced method, the user specifies the independent X and Y spacing between the toolpaths. The spacing is then used and applied to the bounding box of the part geometry as shown in the picture on the dialog. In the second method, or the fitted method, the user specifies a XY rectangle in which the toolpaths will be fitted. Here again, the bounding box of the part geometry is considered when computing the number of toolpaths that can be fitted in the specified space. Additionally, the order of the instancing can also be specified to be either X first or Y first.



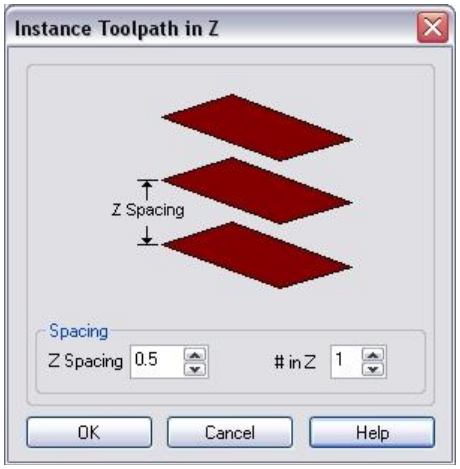


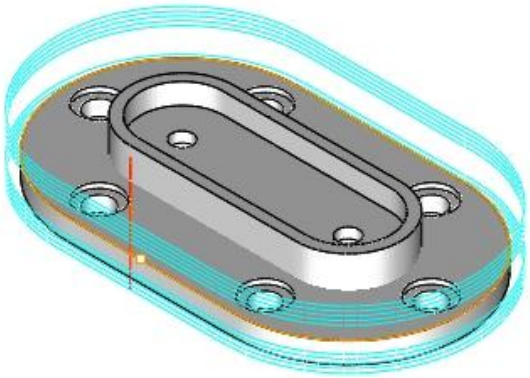
Notes:

Z Instance Toolpath

User can create multiple instances of the toolpath using this dialog. This can be useful in situations where the user is manufacturing multiple instances of the same part.

The user can specify the number of instances in Z as well as the spacing between the Z instances. The toolpath will be copied multiple times and the tool initial engage will start from the top-most instance and work its way to the bottom most instance. Negative values can be specified in the Z spacing. In this case the instances will proceed below the current location of the toolpath.

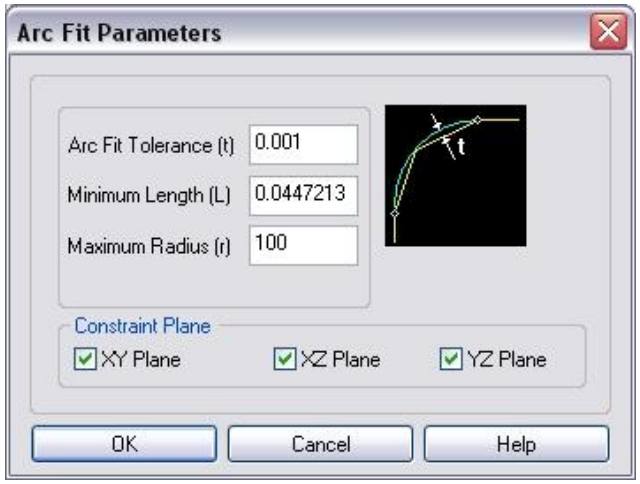


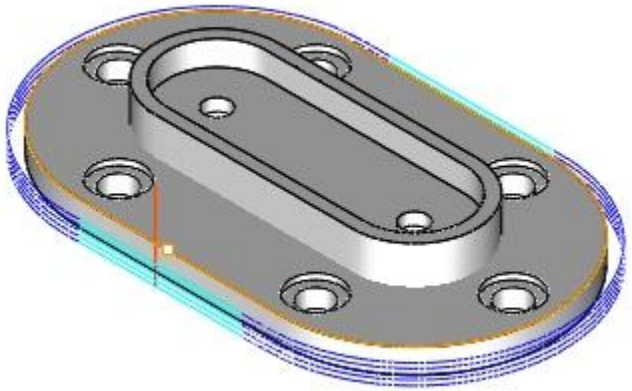


Notes:

Fit Arcs to Toolpath

Arc or circular motions can be fitted to point-to-point toolpath motions in the toolpath editor. This serves to reduce the size of the post-processed toolpath as well as to improve surface finish. To fit arcs to a toolpath, the user clicks on this button. The following dialog will be then be invoked.

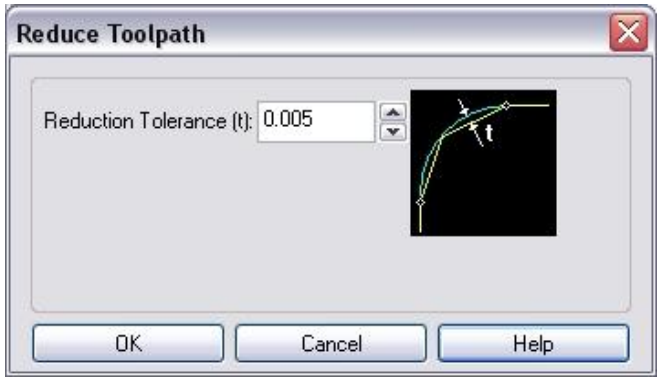




The user specifies an arc fitting tolerance, a limiting arc length as well as the smallest radius of the arc allowed. The arc motions can further be restricted to be created in any or all of the principal planes.

Linearize Toolpath

This option is used to optimize the toolpath. A tolerance value can be specified and this will remove any additional toolpath points, which are less than the tolerance specified.



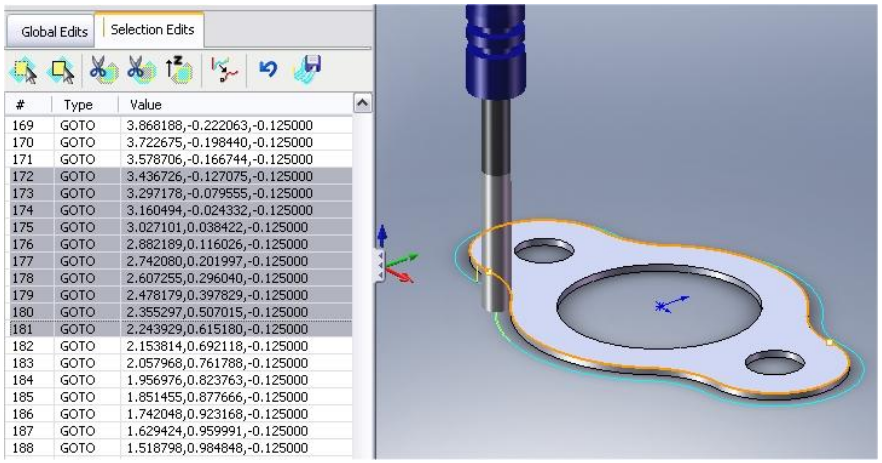
Undo Toolpath Edits: Clicking on this performs an undo operation and restores the toolpath back to its original un-edited state.

Save Toolpath Edits: Saves the edited toolpaths.

Selection Edits

These edits apply to the selected motions of the toolpath. The motions can be selected by selecting the rows of toolpath motions in the toolpath editor and the selections are highlighted on toolpath display.

Notes:



Select cuts by single pick or drag rectangle: Clicking on this tool allows user to graphically select sections of toolpath for editing.

Select closed curves to select cuts inside it: Allows user to select a closed curve as region to eliminate cuts inside it.

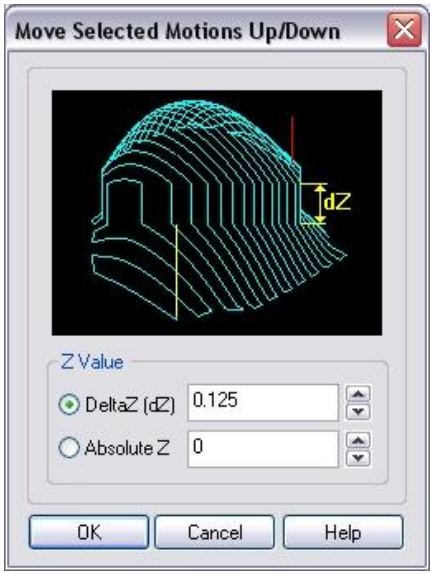
Cut Selections: Clicking on this button will remove all tool motions that are currently selected. It should be noted that in the case of tool motions that intersect with the selection regions the tool motions would be clipped against the selection regions and removed.

Isolate Selections: Clicking on this button will remove all tool motions other than those currently selected. Tool motions that intersect with the selection regions will be clipped and isolated.

Move Selection in Z

This dialog can be used to move up the selected motions up by either a delta amount or move the toolpath to an absolute Z coordinate. The user can select the entire toolpath or a section of the toolpath to be moved.

Notes:



In the first case the toolpath selection is merely shifted up in Z by the delta amount. In the second case, the starting point of the selection is moved to the specified absolute Z coordinate. All other selections will be shifted up by the same relative amount as the first point of the toolpath point.

Convert Selection to curve: The toolpath or a part of it can be selected using the selection commands. Then this command can be used to convert that toolpath into a Curve (region).

Undo Toolpath Edits: Clicking on this performs an undo operation and restores the toolpath back to its original un-edited state.

Save Toolpath Edits: Saves the edited toolpaths.

Knowledge Base

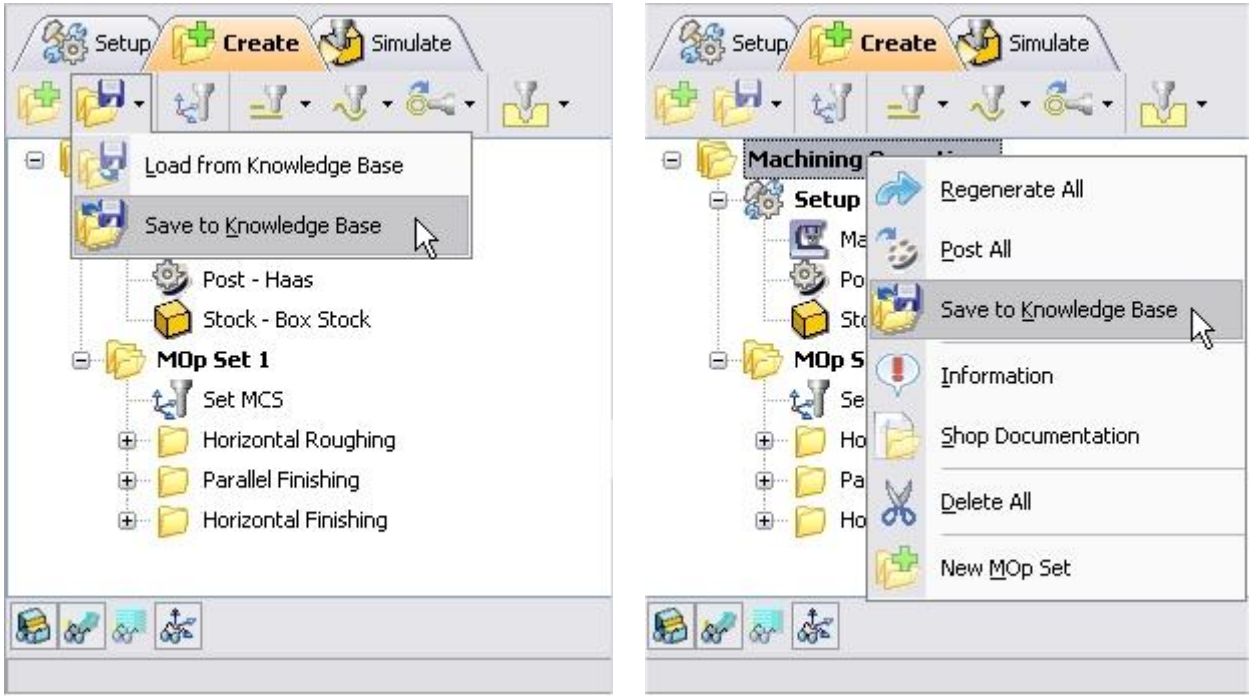
RhinoCAM introduces another powerful feature that makes "push button" programming a reality. Users can archive an entire machining strategy specific to a certain class of parts in a Knowledge Database or K-Base. This feature is a powerful feature and can be used in many situations. In family of part situations, where the same set of machining

Notes:

operations and tools can be applied to machine these parts, it would be most appropriate to archive this processes in a K-base file and then apply it across all of the parts in this family. Another situation where this feature can be used is in shop floor programming. Experienced programmers can determine the sequence of operations to be used to machine a certain class of parts and create a K-base file. Once these K-base files are thoroughly debugged, operators at the shop floor can then load and generate toolpaths almost automatically. Doing this not only increases the throughput but also the productivity of the entire manufacturing team, resulting in dramatic cost savings for the enterprise.

The machining strategy can include both, the sequence of machining processes used as well as the specific parameters used in each of these machining processes.

To create a Knowledge Base, simply start creating machining operations. Once created these machining operations can be re-sequenced if necessary. When completely satisfied with the machining operations used and their sequence, click Knowledge Base and Save to Knowledge Base from the create tab or pick the "Save to Knowledge Base" option by right click on Machining Operations and selecting Save to Knowledge Base.

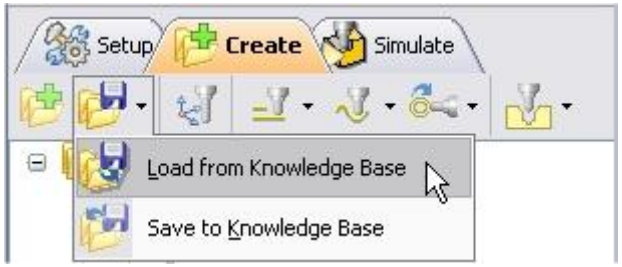


This will allow the user to save this entire sequence of operations and all of the associated operation parameters in an external VKB file.

Notes:

Notes:

Once such a file is created, users can load and re-use this K-base in any other part file that it may be appropriate. To load a K-Base file select Knowledge Base and "Load from Knowledge Base" option from the create operations browser.



Users can load one or more knowledge base files. These are automatically appended to the existing list of machining operations and they appear separate MOp Sets.

These K-base files do not have associated toolpaths. Once loaded, the user selects each of the machining operations, in the "Machining Operations Browser" and creates toolpaths based on the parameters saved in the K-base file.

Fixture Offsets

RhinoCAM offers fixture offsets where multiple machine coordinate systems can be set using work coordinate offsets.

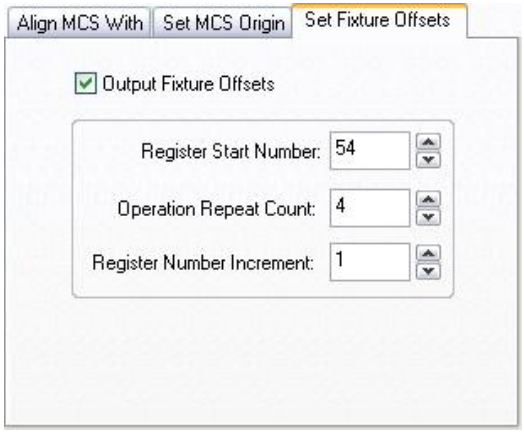
For machining repeat parts user can set the repeat count, register start number and the register increment in the Set Fixture Offset tab located in the Set MCS Dialog.

Register Start Number: This specifies the start register. Use 54 for G54

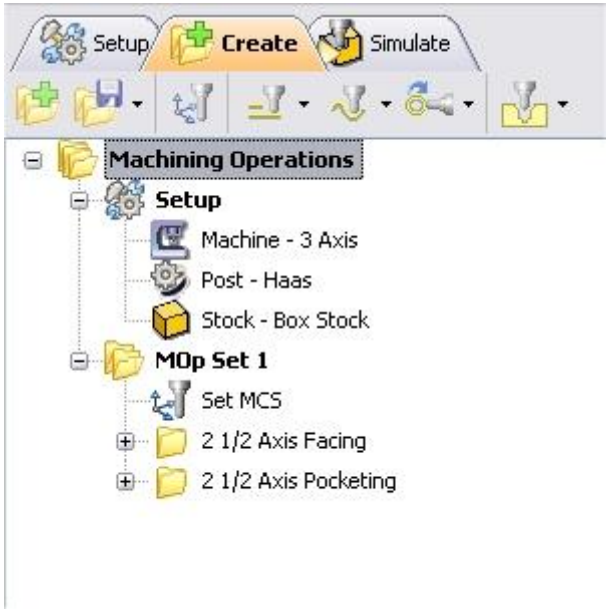
Operation Repeat Count: specify the number of repeats. To machine 4 parts set the repeat count = 4.

Register Number Increment: Setting this to 1 increment the work offset number by 1. (54,55,54 etc.)

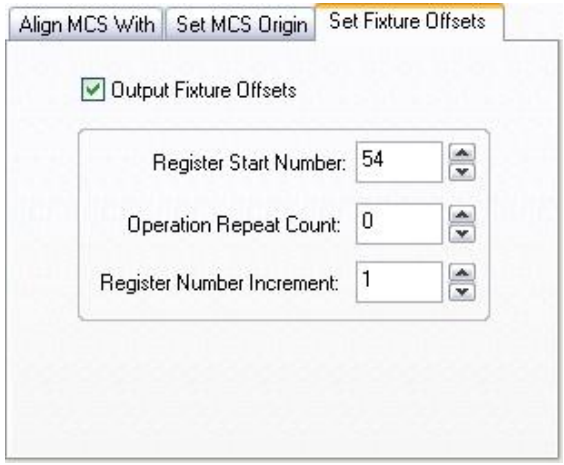
Notes:



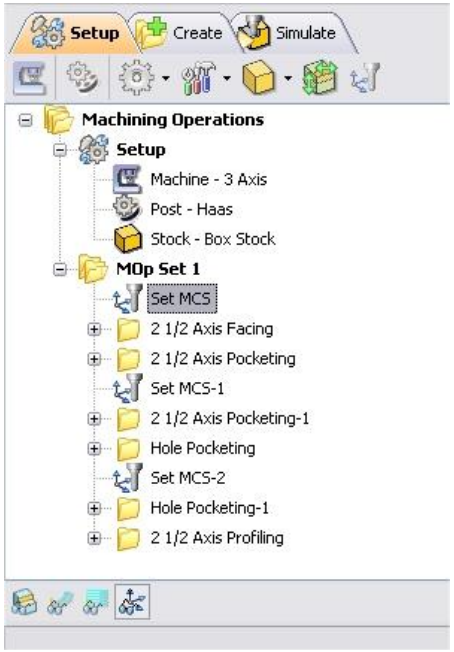
The machining operations that are generated and listed below the SetMCS with Fixture offset when posted by selecting the Mop Set above it will automatically generate the code by repeating itself by the number of repeat count and each copy would have the register number with the specified increment.

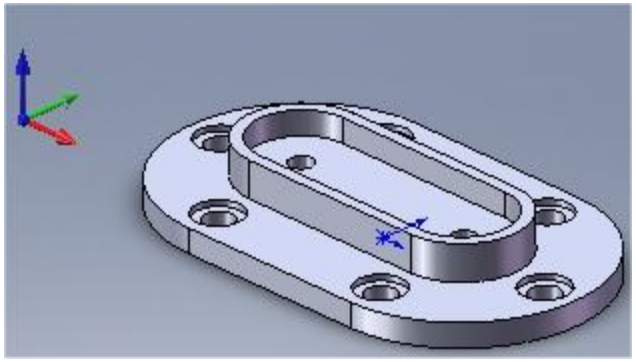


User can also use the fixture offset to set multiple machine coordinates for the same part. In this case the operation repeat count can be set at 0.

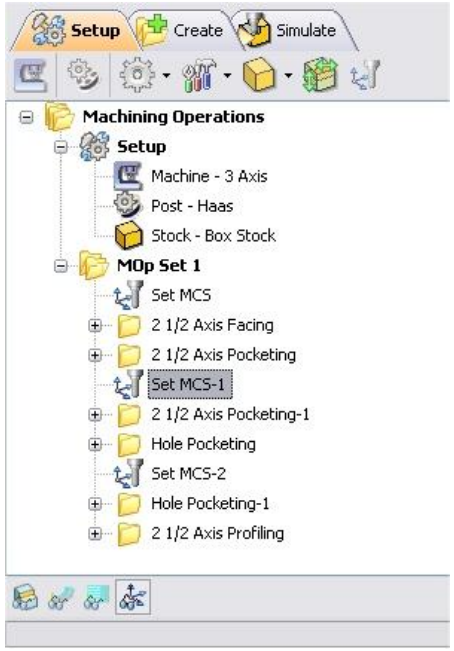


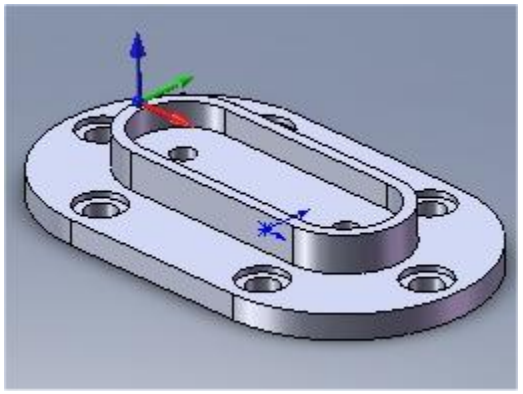
The below example shows 3 fixture offsets the first one being in line with Origin.

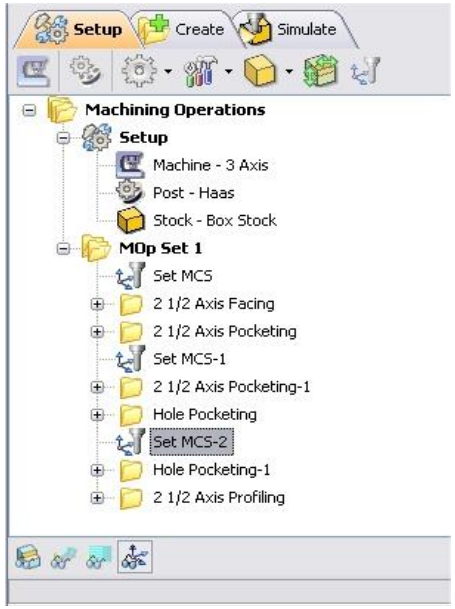


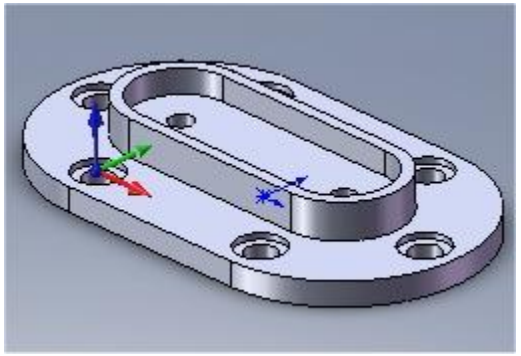


Notes:


Setup
Create
Simulate
Machining Operations
Setup
Machine - 3 Axis
Post - Haas
Stock - Box Stock
MOp Set 1
Set MCS
2 1/2 Axis Facing
2 1/2 Axis Pocketing
Set MCS-1
2 1/2 Axis Pocketing-1
Hole Pocketing
Set MCS-2
Hole Pocketing-1
2 1/2 Axis Profiling




Setup
Create
Simulate
Machining Operations
Setup
Machine - 3 Axis
Post - Haas
Stock - Box Stock
MOp Set 1
Set MCS
2 1/2 Axis Facing
2 1/2 Axis Pocketing
Set MCS-1
2 1/2 Axis Pocketing-1
Hole Pocketing
Set MCS-2
Hole Pocketing-1
2 1/2 Axis Profiling



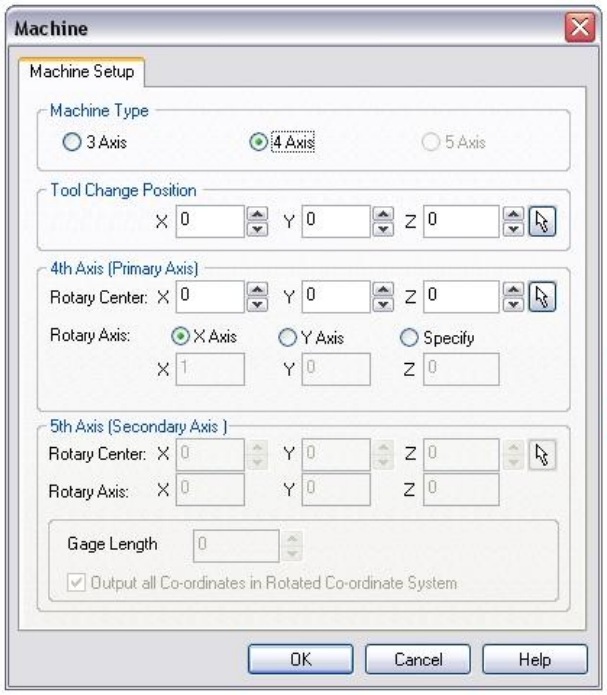
Notes:

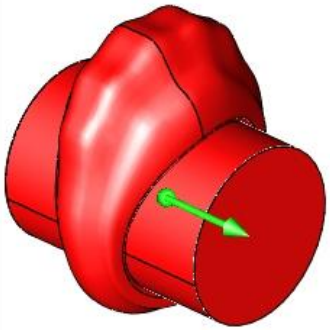
Chapter 9

Notes:

Create 4 axis Toolpaths

Machine Setup (4Axis)





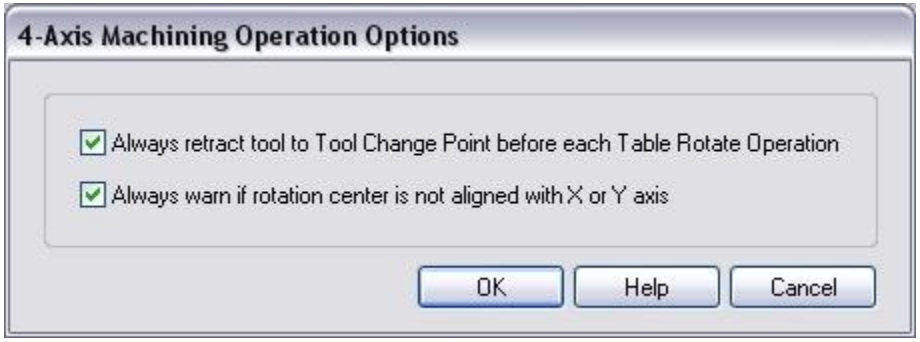
- Select 4 axis for index and continuous 4 axis machining methods.
  - Make sure to set the Rotary Axis to X if the part rotates about the X (A) axis or Y if the part is rotating about Y (B) axis.
  - Make sure to set the rotary center. The part geometry must pass thro' the rotary center. The rotary center is indicated with a Green arrow when the Machine Setup Dialog is selected and machine type is set to 4 Axis.

Tool Change Positing is set to retract tool to Tool Change Point before each table rotate operation.

4th Axis Options

Notes:

This dialog allows the user to set the preferences for index and continuous 4 axis operations.



Always retract tool to Tool Change Point before each table rotate operation: When the user selects this option, RhinoCAM will retract the tool to the tool change point. The tool change point is specified under Machine Setup. This can be located under the Setup tab in the RhinoCAM Mops Browser.

Always warn if rotation center is not aligned with X or Y axis: Selecting this option warns the user if the Machine Zero and the rotary center are not set at the same coordinate point. User can override this setting by un-checking this option.

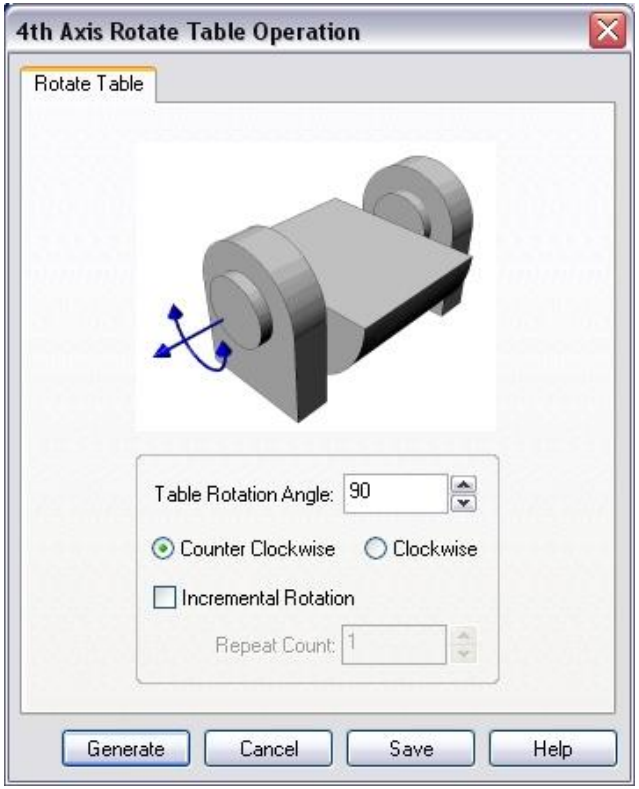
Rotate Table

Here the user can index the table to any specified angle and then lock the table in position. This can then be followed by any of the allowable 3 axis machining operations, thereby allowing access to the part from different orientations.

4th Axis indexing refers to the ability to rotate the part about the X (A) axis or the Y(B) axis and lock it in position in this new orientation and then perform standard 3 axis milling operations in this locked position. RhinoCAM allows the user to program these part rotation motions as well to create standard 3 and 21/2 axis machining operations in these new part orientations. Indexing is usually performed by fixturing the part in a rotary table.

Notes:

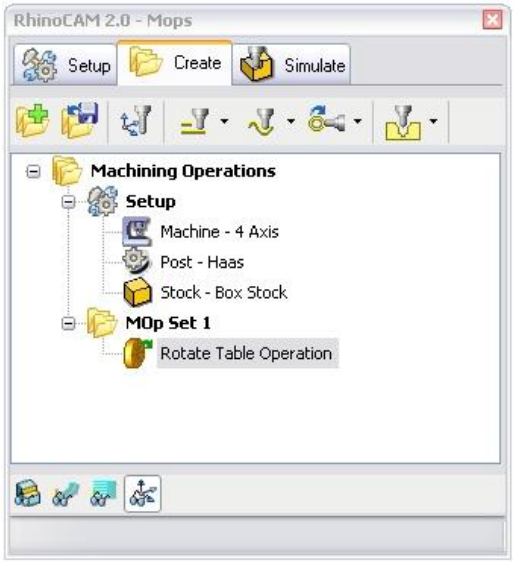
Rotary Axis must be set to X (A) or Y (B) axis before performing any 4-axis operations from the Machine Setup dialog located on the Setup tab.



This type of programming allows the machining of part geometry that has more than one side that needs to be machined. An example of a shape that cannot be machined with a simple 3 axis toolpath is a sphere. Only one half of the sphere can be machined using 3 axis operations. The other half needs to be approached from the flip side in order for this portion to be machined. Using an indexing head as the one shown will allow easy reorienting of the part. Note that in order to utilize 4th axis indexing you must have a CNC control that supports a fourth axis and a rotary table or native fourth axis that can be controlled by the CNC machine.

The table rotation angle is specified in absolute coordinate system. When a rotate table operation is performed, the Rotate Table operation is listed in the MOps browser under a MOp Set and the MCS (Machine Coordinate System) is now oriented to the rotated coordinate as shown below.

Notes:

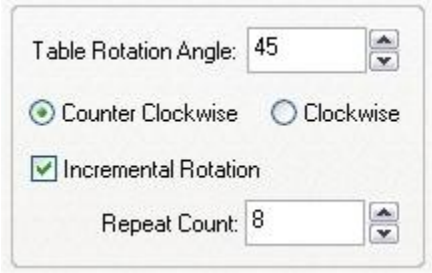


The picture below shows the rotation of MCS. The lines in Blue represent the Z axis, Red represents X axis and Green represents Y axis. The rotary axis is set to X (A) axis.

Rotate 0 **Rotate 90** **Rotate 180** **Rotate 270**

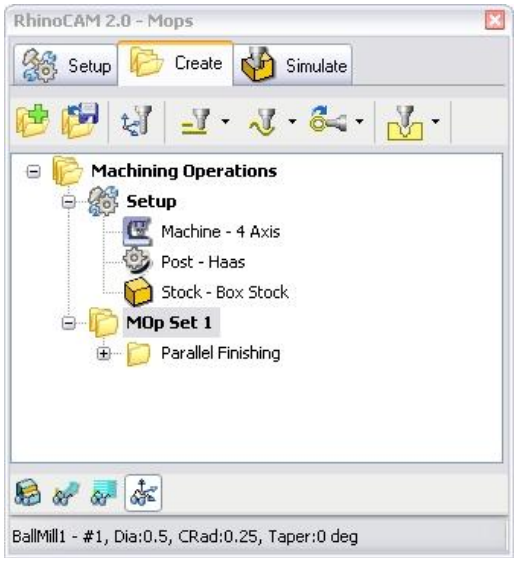
Incremental Rotation

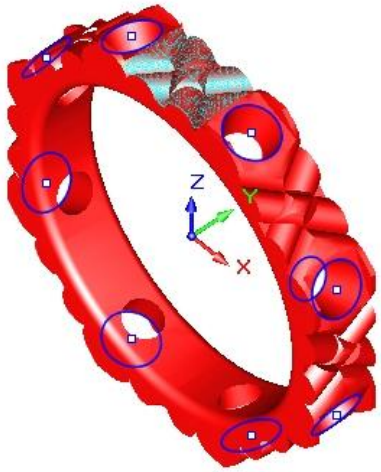
This feature allows the user to create a machining operation (s) and instance it along the X (A) or Y (B) axis by specifying the Rotation Angle and # of instances. This allows user to machine features that repeats at a constant increment along the part in X (A) or Y (B) axis. Ideal for machining repeat features on rings. Users can choose from a wider range 2 ½, 3 axis & 4 axis operations.



Steps to create incremental rotate operation.

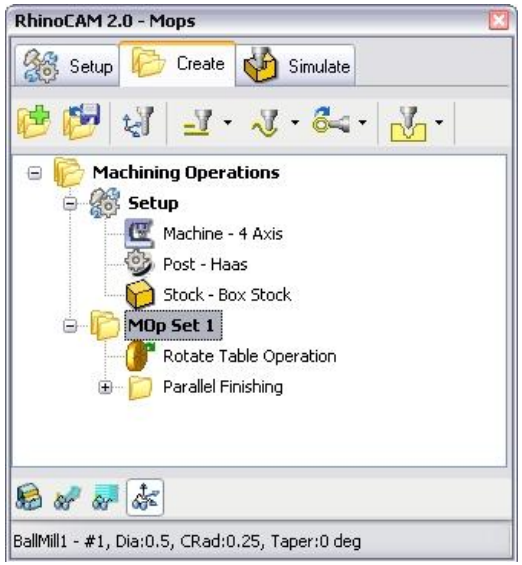
1. Set the Machine zero (WCS), Machine Setup and Rotary Center.
2. Generate a Machining Operation.

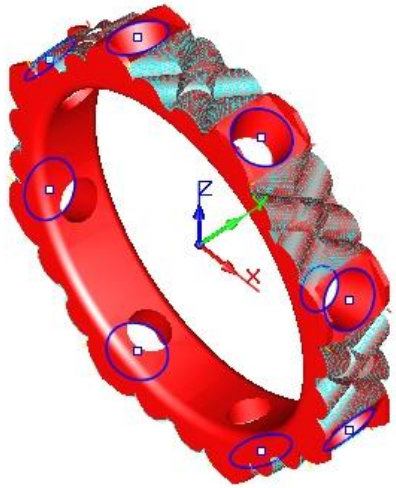




3. Create a Rotate Table Operation and set the Table Rotation Angle. Select Incremental Rotation and specify the repeat count.
4. Move the order of the Rotate Table Operation above the Machining Operation. **The Incremental rotation would now be applied to all the MOps that are under the Rotate Table operation.**

Notes:


RhinoCAM 2.0 - Mops
Setup
Create
Simulate
Machining Operations
Setup
Machine - 4 Axis
Post - Haas
Stock - Box Stock
MOp Set 1
Rotate Table Operation
Parallel Finishing
BallMill1 - #1, Dia:0.5, CRad:0.25, Taper:0 deg


Z
X

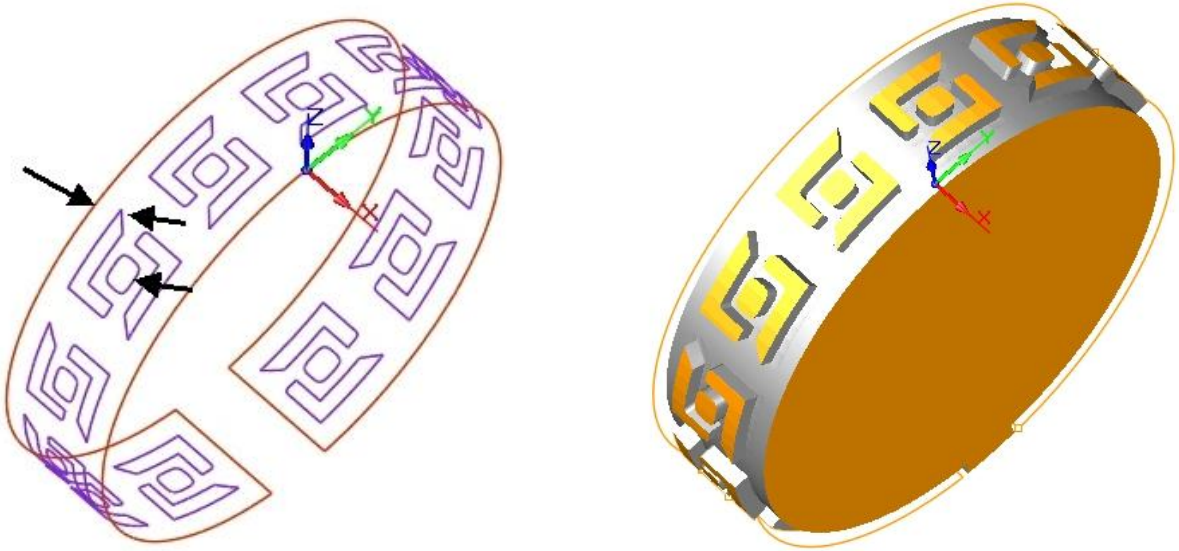
Notes:

4th Axis Facing

Notes:

This method is similar to 2 ½ Axis Facing. This method machines closed regions as if they were completely enclosing material to be removed. This means that the tool can approach the material from outside the outer regions, creating reverse pockets. The machining regions can be selected from a cylindrical surface.

This example uses multiple regions – the outer region is the curve, and features inside the curve are considered inner regions. Nested regions are treated as islands (areas to avoid).



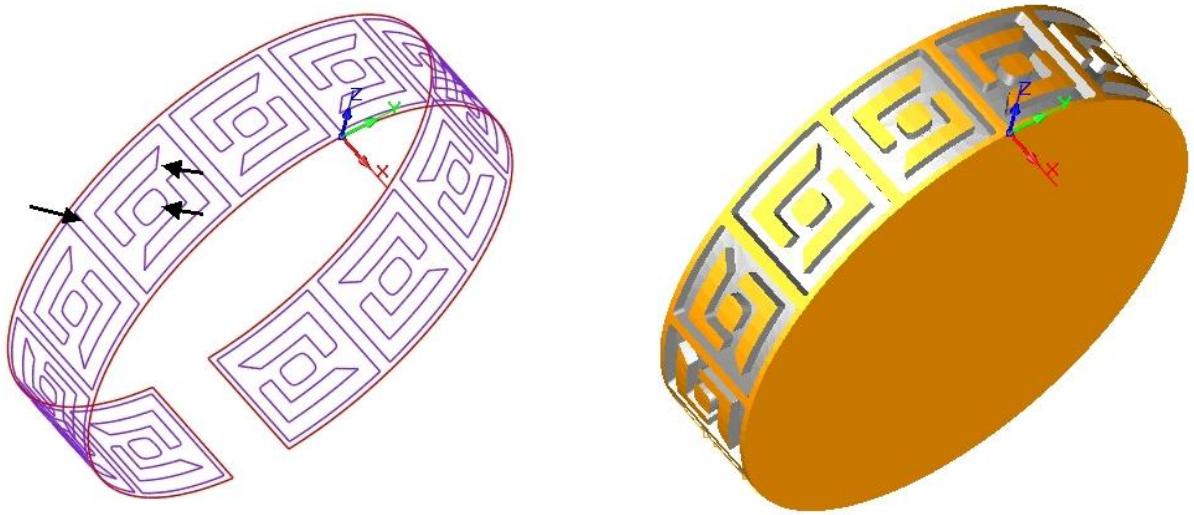
Cut Parameters, Cut Levels, Entry/Exit and Clearance Plane parameters are set similar to 2 ½ Axis Facing Operation.

4th Axis Pocketing

Notes:

This method is similar to 2 ½ Axis Pocketing. This method machines closed regions as if they were pockets - completely enclosed by inner and outer regions. The tool cannot go beyond the outer region, and cannot go within inner regions. This is unlike Facing, in which the outermost region is considered to enclose material to be removed. The machining regions can be selected from a cylindrical surface.

This example uses similar regions as the previous Facing example, but the outer region is within the stock limits.



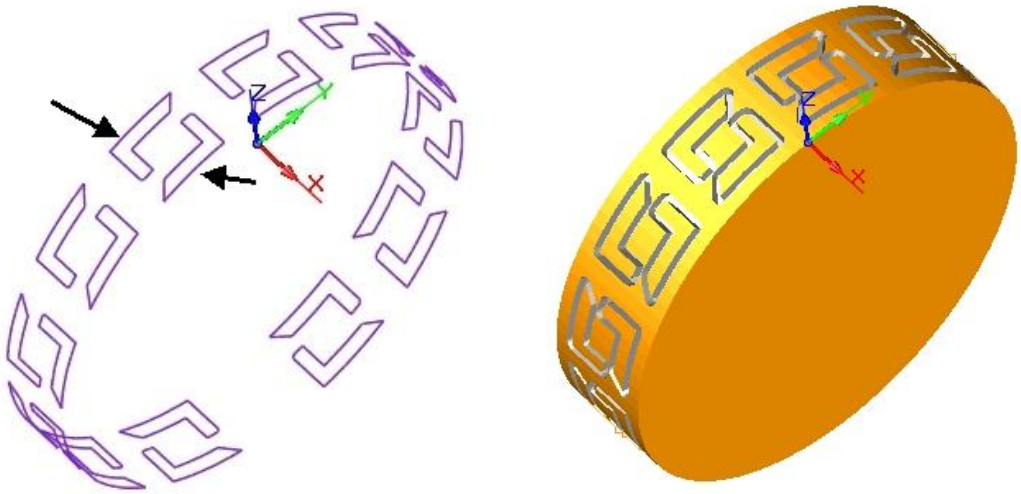
Cut Parameters, Cut Levels, Entry/Exit and Clearance Plane parameters are set similar to 2 ½ Axis Pocketing Operation.

4th Axis Profiling

Notes:

This method is similar to 2 ½ Axis Profiling. This method machines open and closed regions by tracing along one side of their contours. You can define offsets so that the tool makes multiple passes relative to the regions. Profiling can be used as a finishing operation after a **Pocketing** or **Facing** toolpath, or it can be used alone.

The machining regions can be selected from a cylindrical surface.



Cut Parameters, Cut Levels, Entry/Exit and Clearance Plane parameters are set similar to 2 ½ Axis Pocketing Operation.

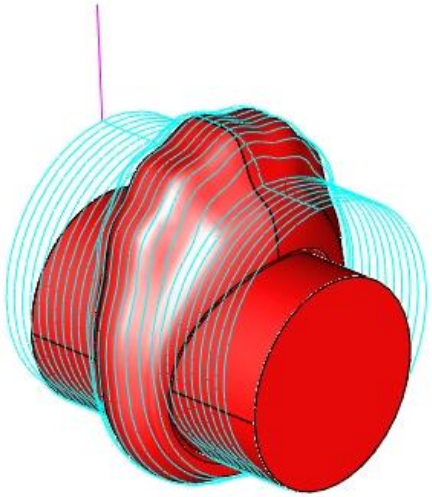
4th axis Facing, Pocketing and Profiling operations are limited to cylindrical features.

Class Room Exercise – Open 4AxisPocketing_1.3dm and create 4th Axis Drilling and Pocketing operations.

4th Axis Roughing

Notes:

Roughing is performed in continuous mode. Multiple levels can be defined and the tool works its way from the outermost level to the innermost level. This is similar to the 3 Axis Horizontal Roughing except that the levels in this case are concentric to the axis of rotation. The table rotation varies constantly as the tool is removing material.

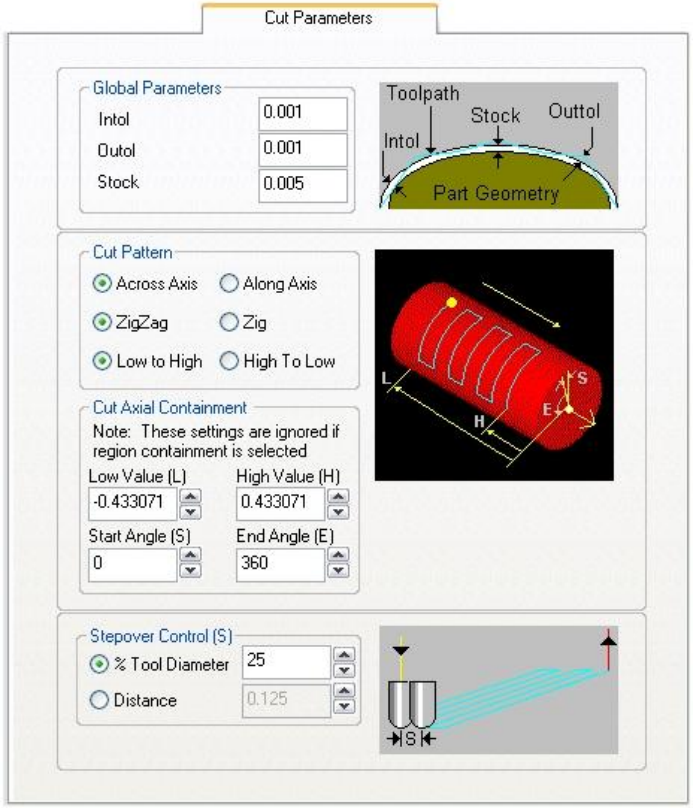


Cut Parameters

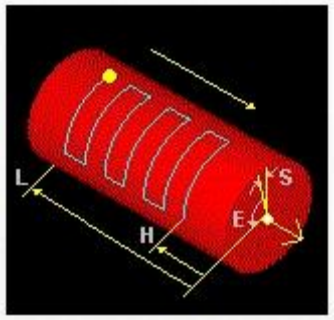
The toolpath can be computed along and across the axis of rotation. Along Axis will create toolpaths that traverse along the rotation axis. Selecting across Axis will generate toolpaths that traverse perpendicular to the rotation axis.

Global parameters can be set similar to other 3 axis machining operations.

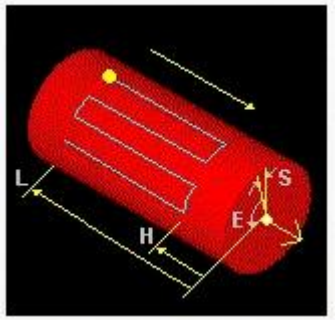
The cut type section allows the user to define the type of cut pattern that will be generated. Currently the user can choose Along Axis or Across Axis, Zig-Zag or Zig, High-to-Low or Low-to-High, and the Start and End Angles.



Selecting Along Axis will create toolpaths that traverse along the rotation axis. Selecting Across Axis will generate toolpaths that traverse perpendicular to the rotation axis.



Across Axis

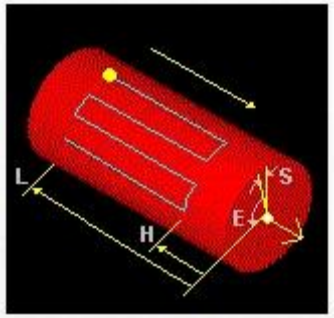


Along Axis

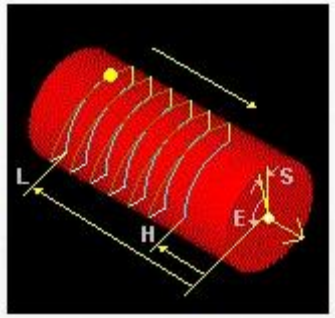
Notes:

Notes:

Selecting ZigZag will allow the toolpath to traverse back and forth while the Zig option will force the toolpath to be in one direction only.

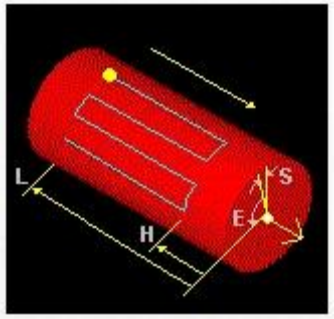


Zig Zag

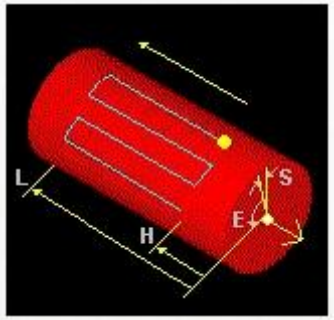


Zig

Selecting Low-to-High will make the toolpath start from the lower coordinate along the tool axis and proceed to the higher coordinate. Selecting High-to-Low will reverse this behavior.



Low to High

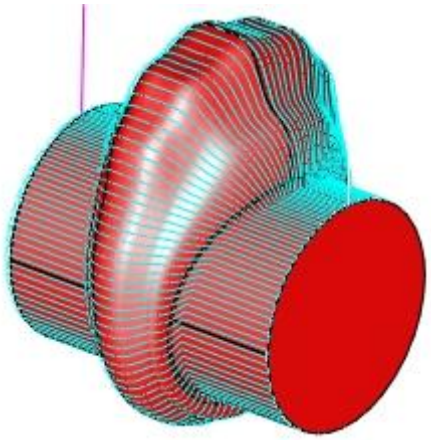


High to Low

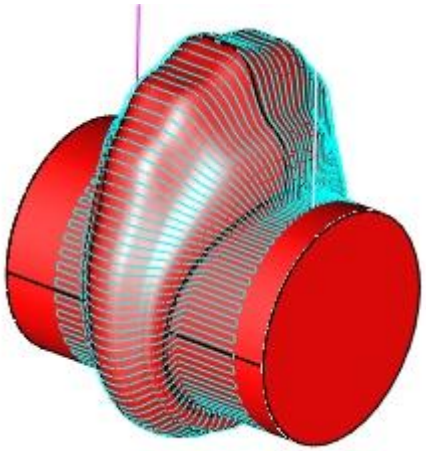
The user can also contain the toolpath both by specifying low and high coordinates along the cut axis as well as by specifying a low and a high rotation axis about the rotation axis.

Selecting **Do not cut past** allows the user to specify the radial distance of the last level from the rotary axis. (This option is available under the Step Down control tab for 4th Axis Roughing)

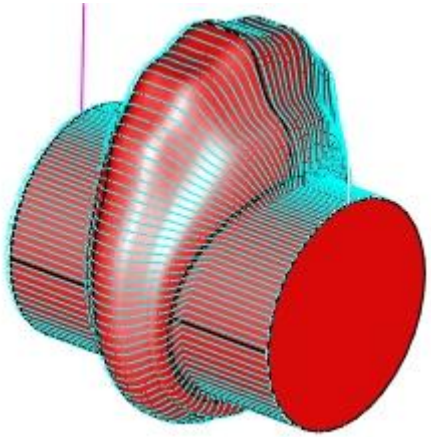
Notes:



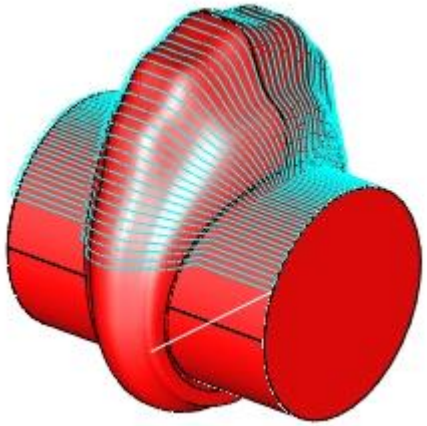
No containment Limit



With Containment Limit



No Axis Containment



With Axis Containment

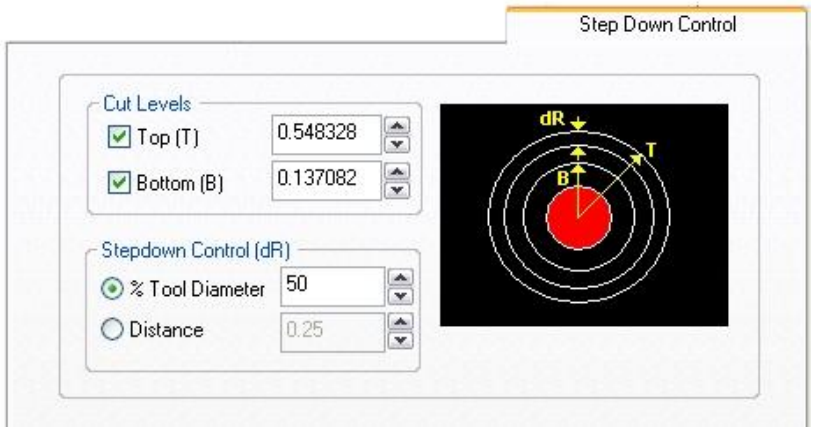
The Stepover Control section allows the user to define the spacing between the cuts. The spacing can be specified either as a percentage of tool diameter or a specific distance.

Step Down Control

Notes:

To create multiple levels the user needs to select the Step Down Control. This brings up the property page shown below.

Selecting the Top check box will allow the user to specify the radial distance of the top most level from the rotary axis. If this is not selected, the system will use the largest radial distance based on the part geometry. Selecting the Bottom check box will allow the user to specify the radial distance of the last level from the rotary axis. The system will then create cut levels starting from the Top level to the Bottom level. The spacing between each level is specified by the Step-down Control section. The spacing can be specified either as a % of Tool Diameter or as a Distance.



Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. Typically the user would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

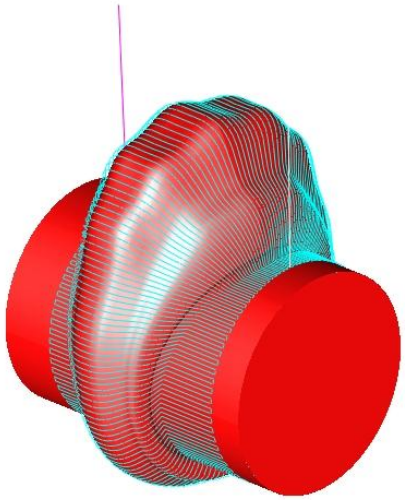
Cut Transfer - The user can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can either be instructed to move to the clearance plane and then perform the transfer motion to the next cut location or it could do a skim motion. In the skim motion, the system automatically determines the safe height by taking into consideration the condition of the part and stock model and using this Z value as the height to perform the transfer motions.

Notes:

4th Axis Finishing

In this method, finish cutting is performed by varying the table rotation continuously as the tool traverses along or across the rotary axis. The tool additionally can move up in the Z axis as it performs final finish machining on the part. This method is similar to the 3 Axis Parallel Finishing except that the cutting is performed in 4 axis mode.

Notes:

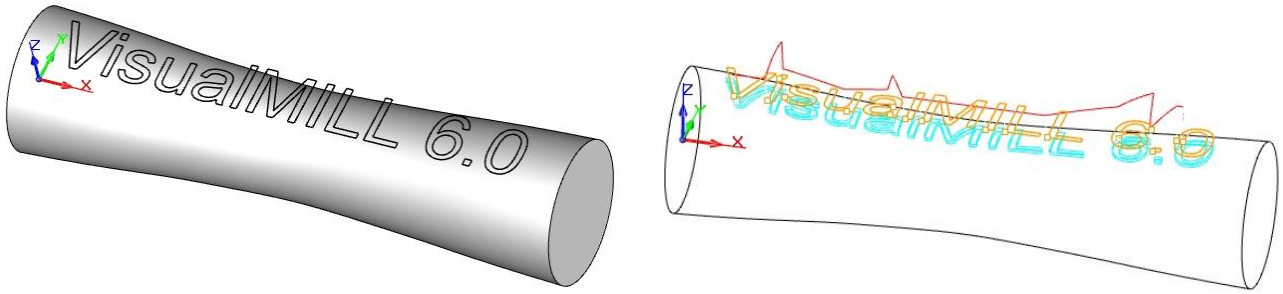


Cut parameters & Clearance Plane definition are set similar to 4th Axis Roughing operation.

Class Room Exercise – Open RingExample_1.3dm and create 4th Axis Roughing and Finishing operations.

4th Axis Engraving

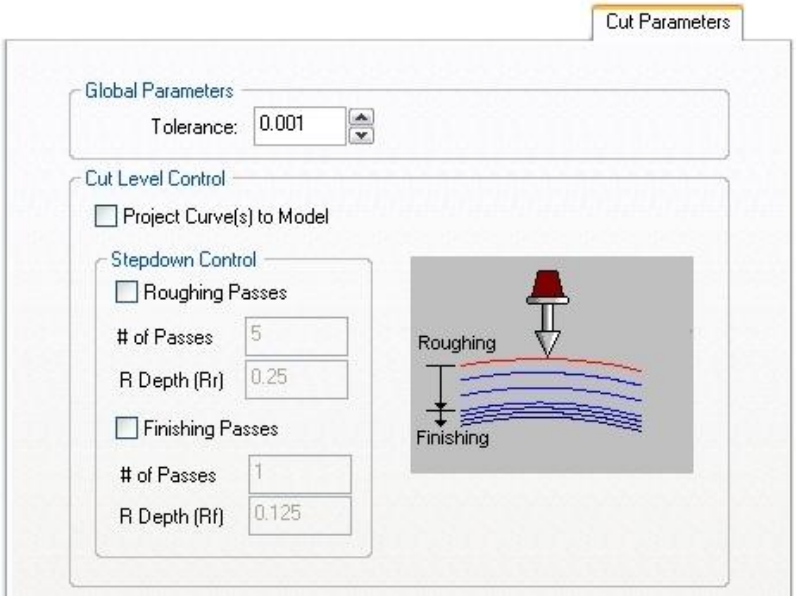
This method is allows engraving curves on a rotary axis. The tool follows the curve as the table rotates. User can specify multiple depths of cut for the engraving operation. Optionally the user can choose to project the curves to the surfaces below before machining. This method is useful in scribing text etc onto the part geometry in 4 axis mode.



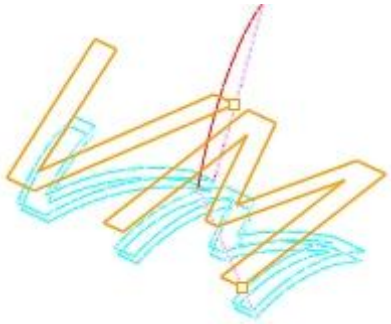
Notes:

Cut Parameters

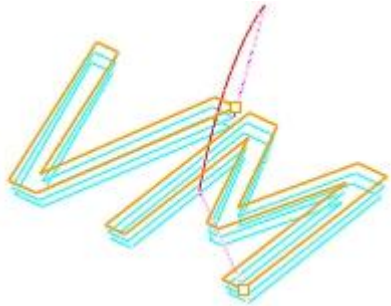
Tolerance is the allowable deviations from the actual part geometry



Cut Level Control – Allows the user to project the curves to the 3d model when the curves are on a 2D plane



Curves Projected to 3D model



No Projection

The **Step Down Control** section allows the user to select the Roughing and Finishing passes and the depth of each pass. The user can optionally specify both the Roughing passes and/or the Finishing passes. If the user does not specify either of them the cutter will simply follow the select region. It should be noted that the orientation of the cutter as it follows the region will always be maintained normal to the center of rotation of the table, thereby maintaining true 4th axis machining.

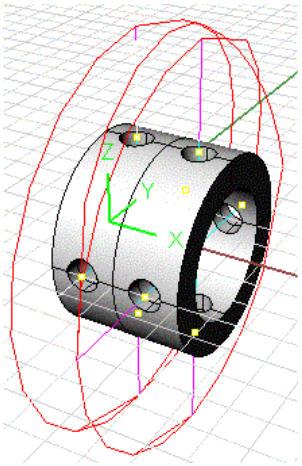
Clearance Parameters are set similar to 4th axis Roughing and Finishing operations.

4th Axis Hole Making

All of the hole creating machining operations available in 3 axis are also available in 4 axis mode. The 4 axis Options is invoked by selecting the **Create** tab and clicking on the Holes button in the RhinoCAM Mops browser.

These operations include Drilling, Tapping, Boring and Reverse Boring. As in any other 4th axis operation, the tool is positioned normal (perpendicular) to the rotary axis, as shown in the picture below. Once the holes (regions) are selected, the dialog boxes are similar to the 3-axis hole making operations. Sorting of holes is also possible to optimize the tool motion.

Notes:



Notes:

Cut Parameters, Entry/Exit and Sorting are set similar to 3 axis Hole Making operations.

Clearance plane parameters are set similar to 4th axis finishing operations.

4th Axis Drilling

The following drill cycles are available:

- **Standard**: Used for holes whose depth is less than three times the tool diameter.
- **Deep**: Used for holes whose depth is greater than three times the tool diameter, especially when chips are difficult to remove. The tool retracts completely to clean out all chips.
- **Counter Sink**: Cuts an angular opening at the end of the hole.
- **Break Chip**: Similar to Deep drilling, but the tool retracts by a set clearance distance.

4th Axis Tapping

A Tap cycle is used to drill threaded holes in the part, clockwise or counter-clockwise.

4th Axis Boring

A Bore cycle is used to form shapes inside a hole. The following boring cycles are available:

- **Drag**: The tool is fed to the specified depth at the controlled feed rate. Then the spindle is stopped and the tool retracts rapidly.
- **No Drag**: The tool is fed to the specified depth at the controlled feed rate. It is then stopped to orient the spindle, moved away from the side of the hole and then retracted.
- **Manual**: The tool traverses to the programmed point and is fed to the specified depth at the controlled feed rate. Then the tool stops and is retracted manually.

4Axis Reverse Boring

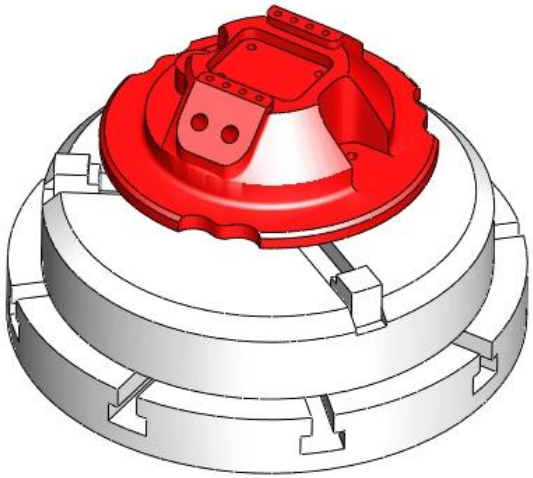
This is a Bore cycle in the reverse direction. The spindle is oriented to the specified angle and moves rapidly to the feed depth and moved to the part. The spindle is turned on and the cycle is started.

Notes:

Chapter 10

Create index 5 axis Toolpaths

Fifth axis machining enables you to change tool direction relative to any set of axes. Fifth Axis operations are used to machine parts that cannot be machined with simple 2 ½ axis or 3 axis machining operations. Since the tool moves only up and down along the Z axis during 3-Axis milling, areas that cannot be seen from above cannot be cut. In such cases, the object could be divided into multiple sections and machined separately. RhinoCAM's implementation of indexed fifth axis milling allows the user to do this automatically if the machine tool is equipped with a head/head, table/table or a head/table configuration. Indexing refers to the ability of rotating the head/head or the table/head on the machine tool and then performing machining in a 3 Axis fashion with the part locked at its orientation.



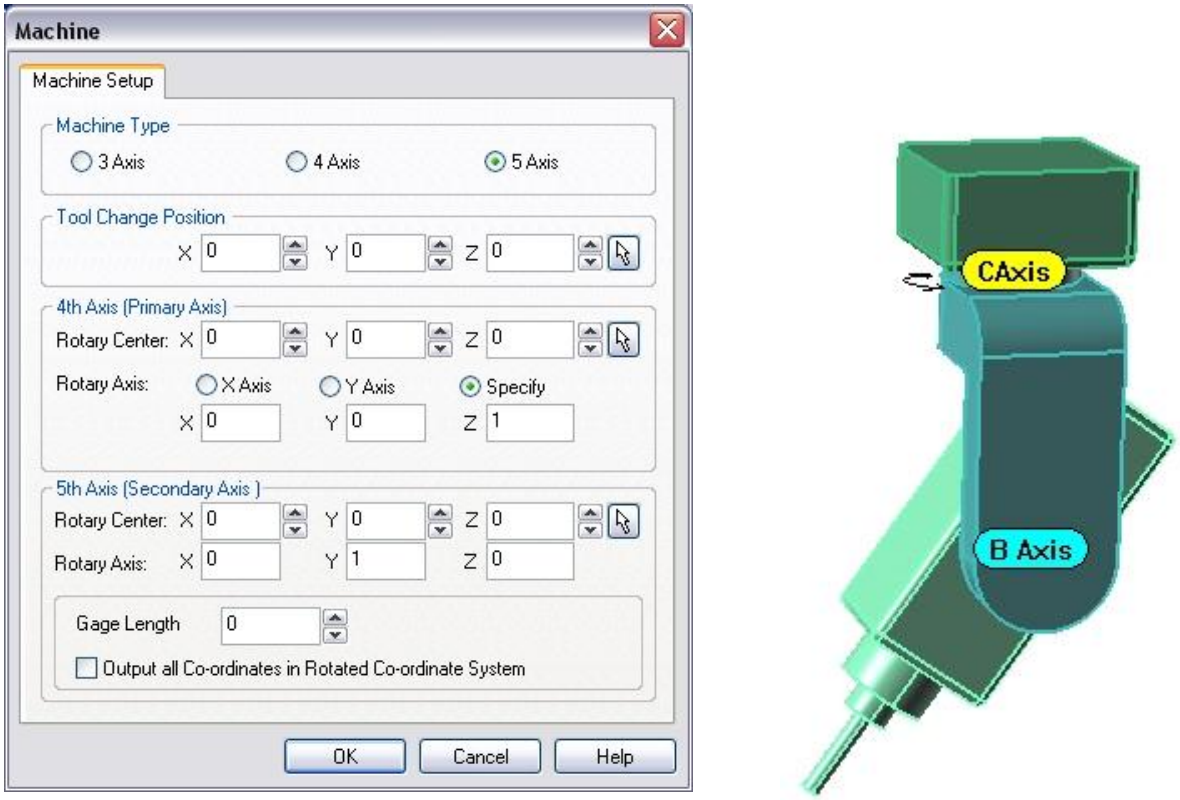
To successfully machine this part, it will use 2 or more different orientations by using set MCS operation to orient the tool normal to a planar surface.

Before creating 5th axis toolpaths, the user needs to specify to the system the primary and secondary rotation axis This can be done by selecting the Machine Tool button in the Setup window of the Browser.

Notes:

Machine Setup (5Axis)

Notes:



5 axis Machine Setup for a C and B configuration

- Select 5 axis for index 5 axis machining methods.
  - Set the Primary Axis under the 4th axis Rotary Axis.
    - If the Primary Axis is A set the Rotary axis as X Axis.
    - If the Primary Axis is C axis, select Specify and set the Value of Z =1.
  - Set the Secondary axis under 5th Axis Rotary Axis.
    - Set X =1, Y=0, Z=0 for A axis,
    - Set X =0, Y=1, Z=0 for B axis,

- **Gage length:** This parameter is used for 5 axis toolpath computation. The distance from the tool tip to the pivot point determines the gage length.
- **Output all Co-ordinates in Rotated Co-ordinate System** (MCS) –
  - This option is selected if the G codes need to be output in the local coordinate system.
  - Uncheck the box to output the G-code with respect to the World Coordinate System (WCS).

Notes:

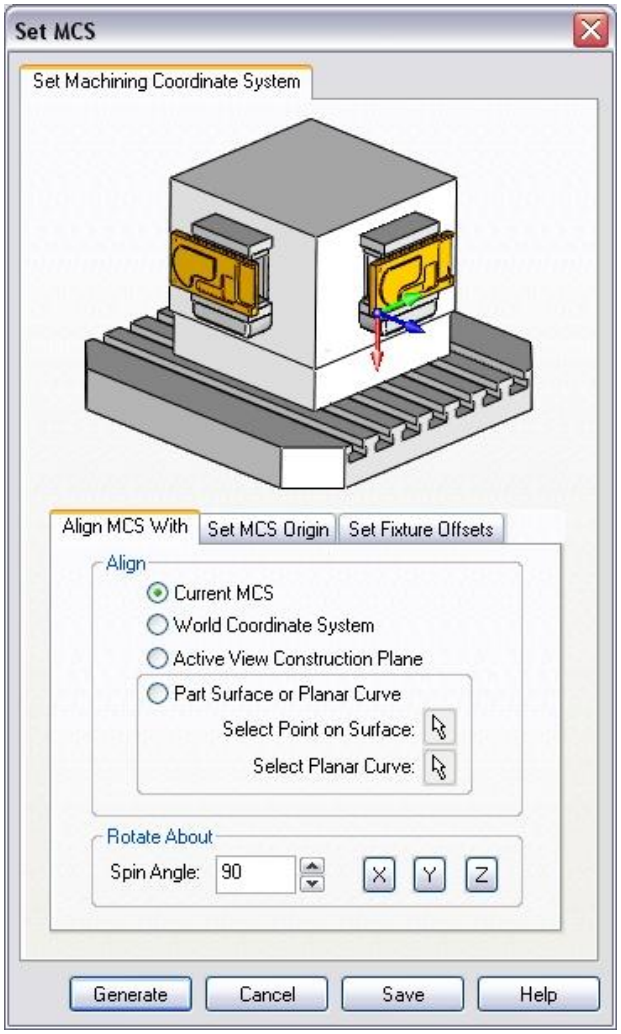
SetMCS

The Machine Coordinate System is the coordinate system that defines the tool orientation as well as the tool zero position.

The MCS is displayed as a triad with Blue line representing the Z-axis, Red representing X-axis and Green Representing the Y-axis. The WCS is displayed the same way as MCS with XYZ coordinates labeled on top of it.

MCS

WCS



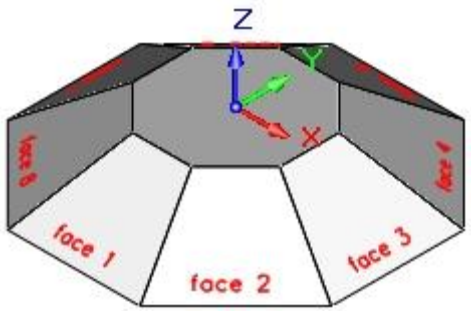
Align MCS With

This tab is used to define the MCS orientation by

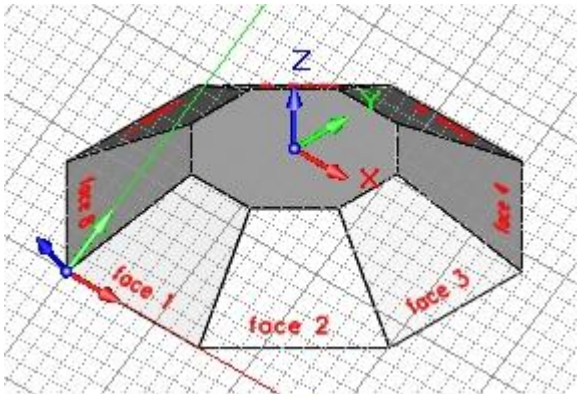
Notes:

- **Current MCS** - Setting the MCS to previously set MCS location

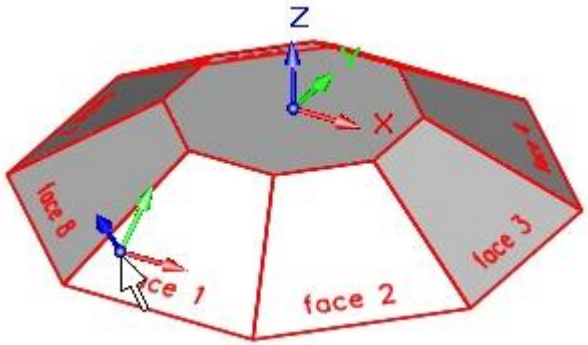
- **World Coordinate System** – Setting the MCS to the WCS. In this case MCS and WCS would have the same coordinate location.



- **Active View Construction Plane** – Setting the MCS to the orientation of the active viewport of the construction Plane.

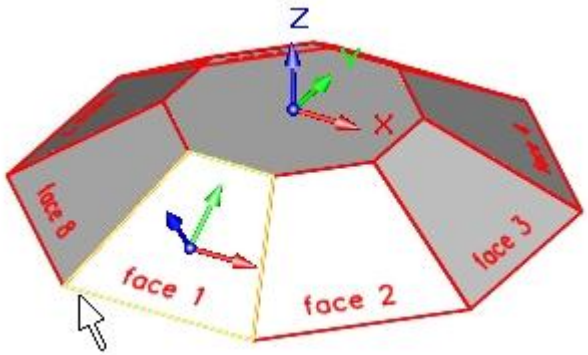


- **Part Surface or Planar Curve** – User can select a point of a surface or a planar curve to determine the alignment of MCS.

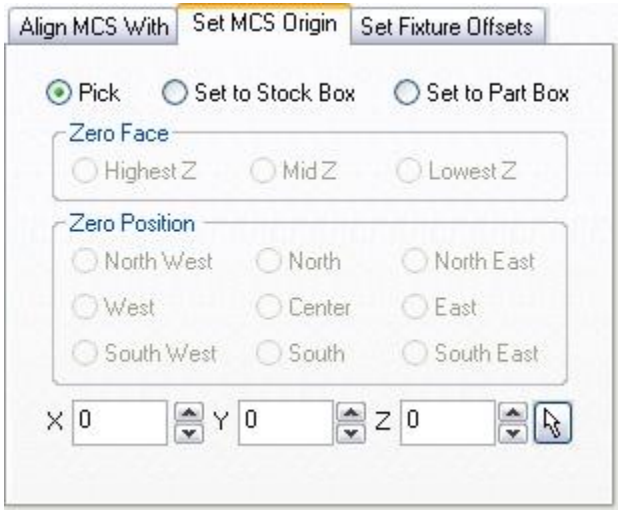


Notes:

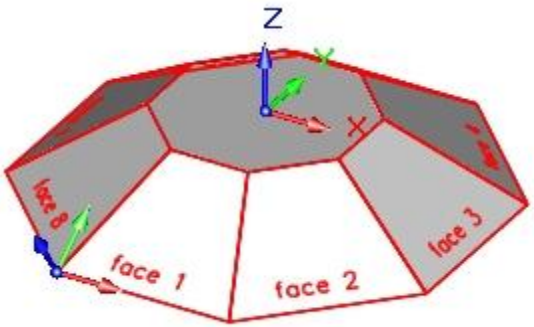
Notes:



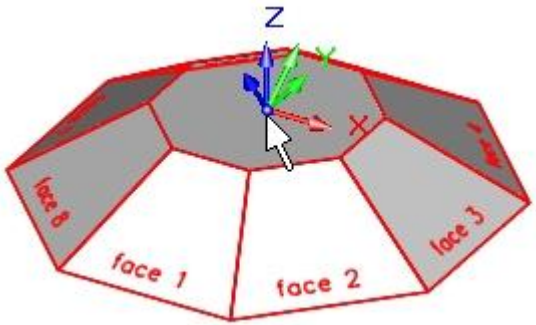
SetMCS Origin



Once the MCS alignment is set, user can switch to the SetMCS Origin tab to set origin point (home) for the MCS. This can be set to the same location as the WCS or at any location on the part geometry.



Align MCS



Set MCS origin to WCS origin using Pick option

Notes:

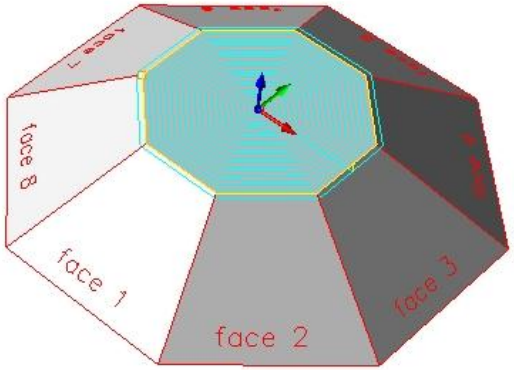
Using SetMCS for Orientation

Notes:

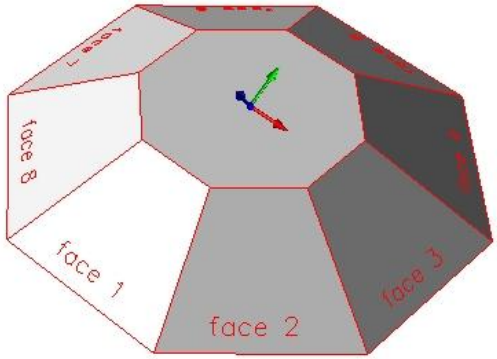
In order to perform an index 5 Axis Operation, the user first needs to use the orient the Machine Coordinate System (MCS) using the SetMCS operation. The Machine Coordinate System is the coordinate system that defines the tool orientation as well as the tool zero position. This can then be followed by any of the allowable 3 axis machining operations, thereby allowing access to the part from different orientations.

The example below shows steps for creating a 5-axis index toolpath for the octagon model.

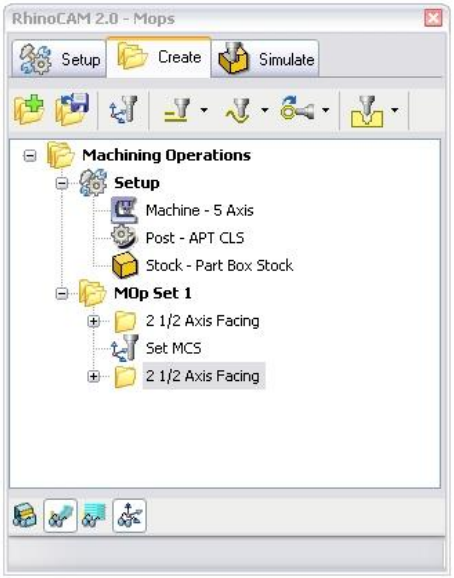
1. Use Locate WCS and set the WCS to the Highest Z, and center in XY.
2. Generate a 21/2 Axis Facing toolpath by selecting the Flat Area at the top.

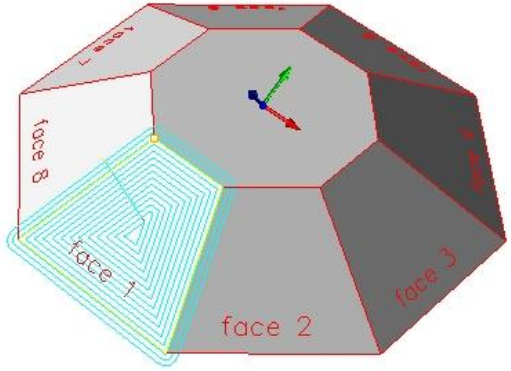


3. Use Set MCS from the Create operation tab and align the MCS to face1. Use Set MCS origin set this to 0,0,0 and click Generate.



4. Generate a 21/2 Axis Facing toolpath by selecting the region at Face1.

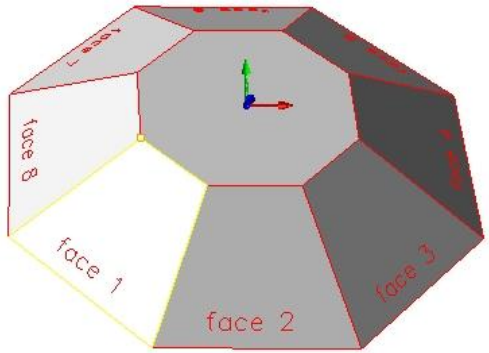




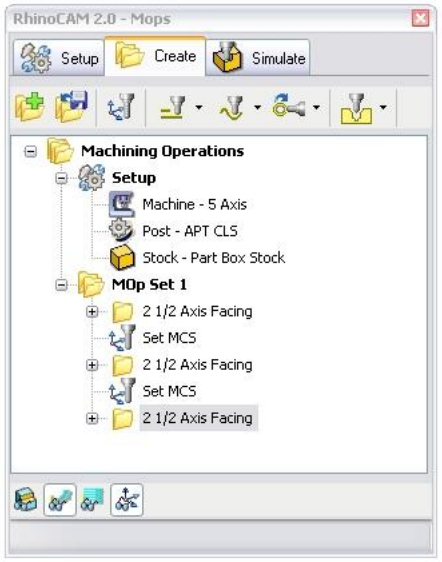
Notes:

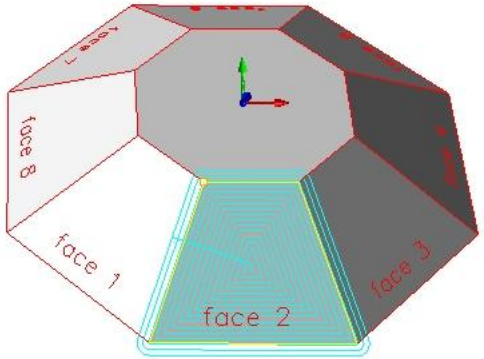
Notes:

5. Use Set MCS from the Create operation tab and align the MCS to face2. Use Set MCS origin set this to 0,0,0 and click Generate.

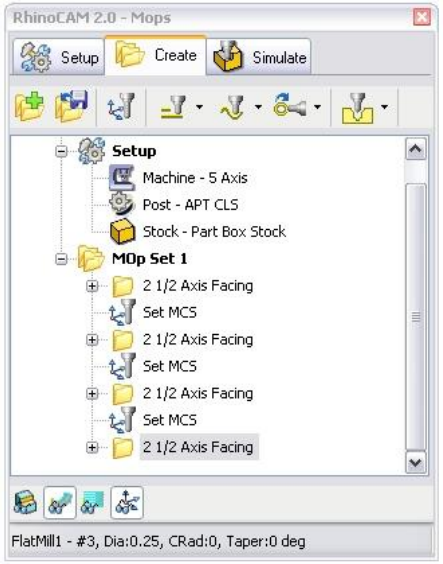


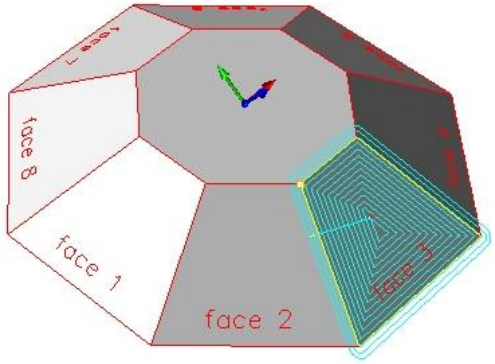
6. Generate a 21/2 Axis Facing toolpath by selecting the region at Face2.





7. Repeat the above steps for all the other face.





Notes:

Chapter 11

Notes:

Post Processor Generator

RhinoCAM's Post-Processor Generator is used to edit post processor files (SPM Files). These files are used by RhinoCAM during toolpath post-processing. RhinoCAM reads in a user specified SPM file, each file corresponding to a single machine tool controller, and generates the post-processed output using the rules resident in these files. Users have the ability to edit these files to modify these rules, thereby controlling the output that RhinoCAM generates.

Customizing Post Processor

Post Processor Generator can be launched from the setup tab under Utilities->Post Processor Generator.

Select the post processor file to edit from the File Select Dialog and click Edit. After selecting the file, it can be edited using the Editor dialog. The format of various output blocks, such as motion, feed rates, spindle etc., can be edited by selecting the appropriate tabs in this dialog.

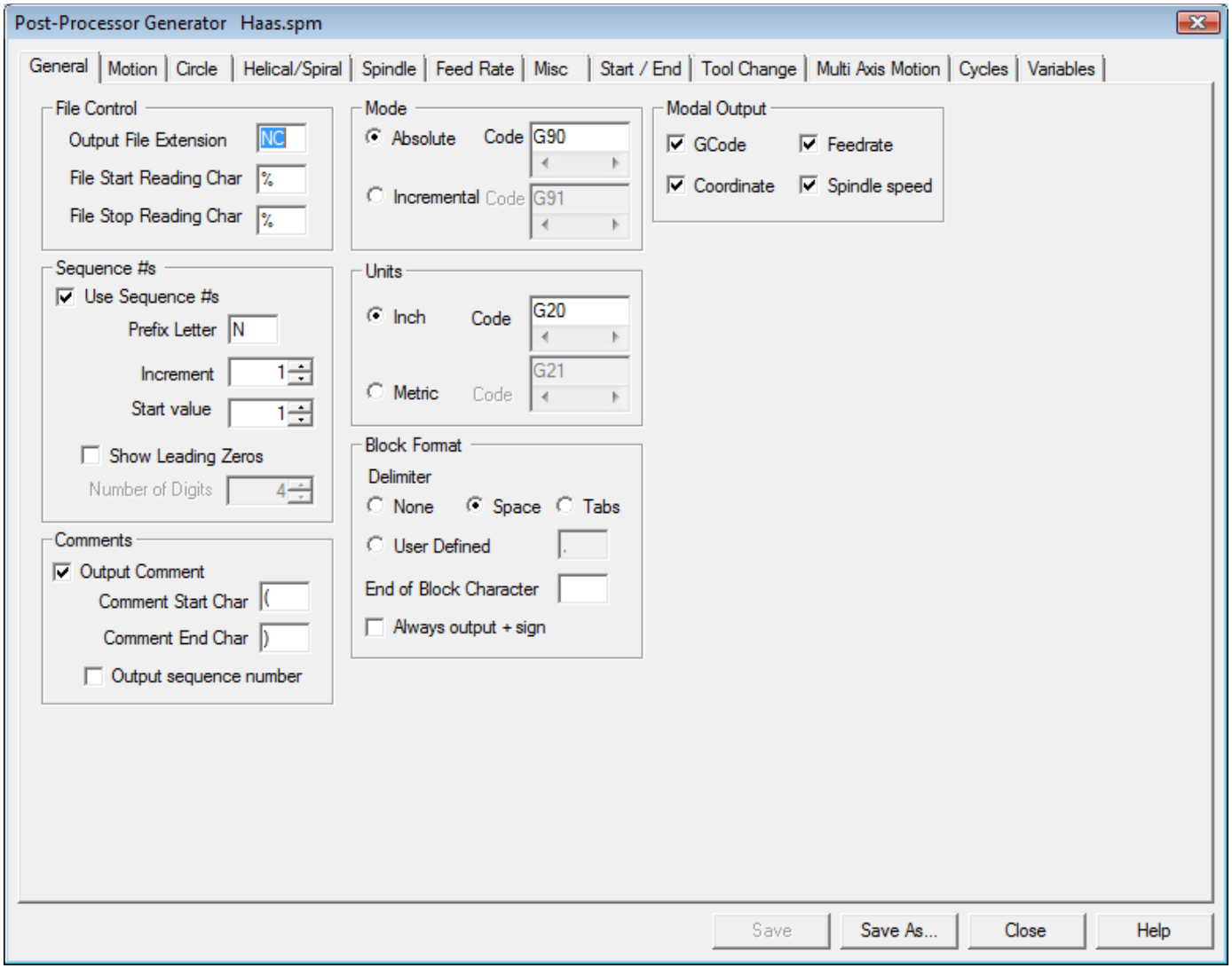
In addition to predefined block definitions, the user can add startup codes as well as termination codes specific to the controller and shop practices. These blocks can be user-defined statements that may contain built in variables.

Notes:

Main Editor

Notes:

The main editor dialog of the RhinoCAM Post-Processor generator is shown below. This dialog is a tabbed dialog with each tab allowing the user to define a block type. Each of the ten tabs seen is described in detail in the following sections.



Description of each tabbed dialog

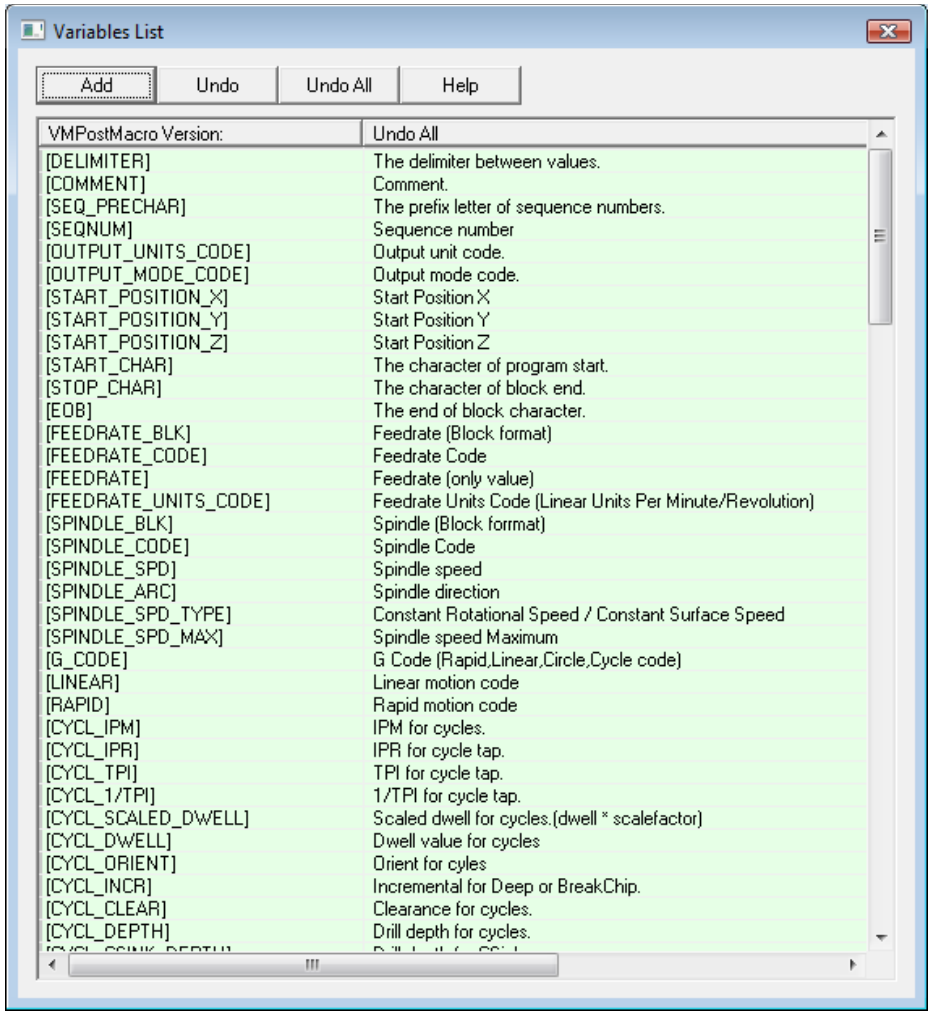
- **General:** This tab helps the user set up file information, G-Code format, mode and the units of operation.
- **Motion:** Motion block specifier
- **Circle:** Circle block specifier
- **Helical/Spiral:** Helical and Spiral motion block specifier
- **Spindle:** Spindle code specifier
- **Feed Rate:** Feed Rate specifier
- **Miscellaneous:** Coolant and compensation code specifier
- **Start/End:** Start and End code specifier.
- **Tool Change:** Load and Tool Change Macro specifier.
- **Multi Axis Motion:** 4th & index 5 Axis Motion specifier.
- **Cycles:** Cycle G-Code and format specifier.
- **Variables:** Lists variables and their values used in post-processing

Notes:

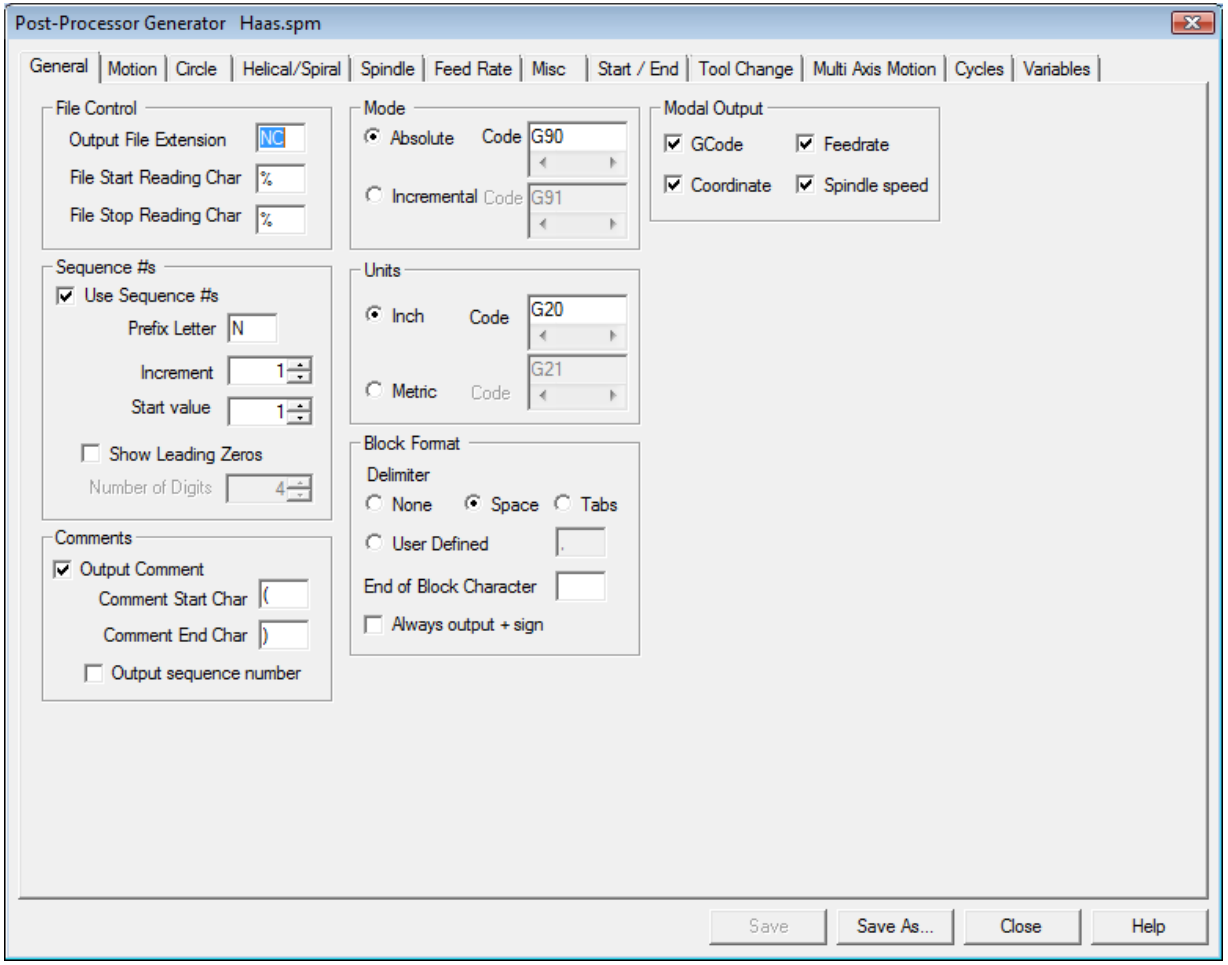
Variable List Dialog

Notes:

The Variable List dialog shown below can be invoked by pressing the right mouse button from within edit boxes that are used in setting up startup and termination code for a post-processor. This dialog can be used to add variables to the active edit box for macros.



General Tab



File Control Parameters pertaining to the output NC file.

- Sequence Number - Sequence number output control
- Modal Output - Output modality
- Mode - Output mode, incremental or absolute
- Units - Output file units
- Block Format - G-code format indicator
- Comments - Format of comments in the generated file

Notes:

Notes:

File Control

This section allows the user the set the default extension of the output file. The user can also set up an optional **file start** character and a **file end** character in the output file. This file start character will be the first character written to the output file. Similarly the file end character will be the last character written to the output file. Most standard controllers look for a percent sign (%) as this last character.

Sequence #s

The section allows the user to control the output of sequence numbers to the output file. This option is available for all G-code lines except Tool change Macro and Start and End Code. Sequence number output can be turned on or off. When on, a prefix letter can optionally be added at the beginning of every sequence number. Sequence numbers can also be output in increments rather than sequentially. This increment value can be specified here. In addition the number of digits output as well as presence/absence of leading zeros in the sequence numbers can be controlled.

Modal Output

The RhinoCAM Post processor generator allows the following parameters to be set as modal or non-modal. Modal output setting will output the value of a variable only if it is different from the value that was last output.

- **Gcode** G-Code modal optionsample
- **Coordinate** Coordinates modal optionsample
- **Feedrate** Feed Rate modal optionsample
- **Spindle Speed** Spindle Speed modal option sample

An example of non-modal data is shown below. The repeated values are shown in colored text.

S1000M03

G00 X1.0 Y2.0 Z0.0 F10

S2000M03

G01 X1.0 Y2.0 Z3.0 F20

G01 X1.0 Y3.0 Z3.0 F20

S2000M03

Mode - Coordinate values can be set to be either absolute or incremental. In the absolute mode, Coordinate values (X, Y, Z, I, J, K) are output as absolute values. In the incremental mode, coordinate values are output as incremental values.

Units - The user specifies to the post-processor the units of the output file by outputting a code, which can be defined here. Output units can either be in the English system (inches) or in the Metric system (mm).

Block Format

This section defines the general format of all output blocks. Each of the options is described below.

- **Delimiter** - Delimiter [D] used in the G-code. - example G01[D]X1.0[D]Y2.0[D]Z3.0[D]S3000M03[D]F20
- **None** - Delimiter absent in the output.
- **Space** - Space is output as the delimiter
- **Tab** - Tab is output as the delimiter
- **User Defined** - User specified delimiter character
- **End of Block Character** - End of block character to output
- **Always output +sign** - Display '+' for positive values.

Example (G01 X+1.0 Y+1.0 Z+1.0)

Comments

This section defines the general format of commentary blocks.

- **Output Comment** - Set parameter to output comments
- **Comment Start Characters** - Comment start character
- **Comment End Characters** - Comment end character
- **Output Sequence Number** - Toggle sequence number for comments

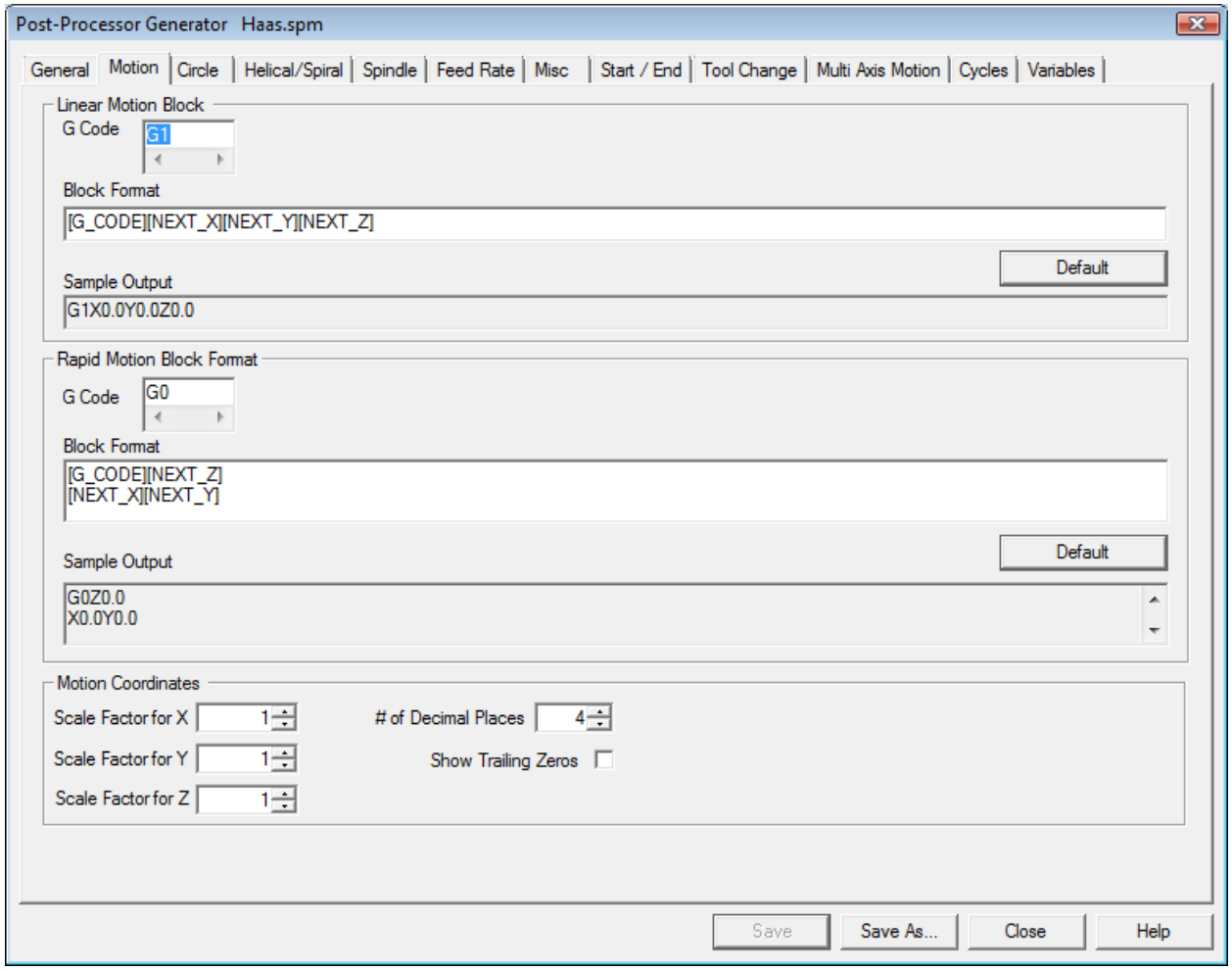
If you want the comments to be output just as it is without the Start Character and the End Character, then in **RhinoCAM**, when you are inserting a comment, type in a $ sign in front of it, like

$ Comment

Notes:

Motion Tab

This tab is used to define the linear motion outputs of a post-processor.



- Linear Motion Block Motion definition for linear cuts.
- Rapid Motion Block Motion definition for rapids.
- Motion Coordinates Method used to output the coordinate values

Notes:

Linear Motion Block - Used to define the output format for the cut motions (e.g.: G1). The sample output can be seen in the non editable Sample Output field

Rapid Motion Block - Used to define the output format for the rapid motions (e.g.: G0). The sample output can be seen in the non editable Sample Output field

Motion Coordinates

- **Scale Factor** - Scale factor of the coordinate values. (Includes circles and cycles)
- **#of Decimal Places** - Number of digits output after the decimal point
- **Show Trailing Zeros -** Show the trailing zeros (e.g. 5.4 is output as 5.4000)

Notes:

Circle Tab

This tab is used to define the circle block output.

This Tab contains

- G-Code Arc Direction Code
- Plane Code Circle Plane code
- Output Format Circle output format
- Arc Center [I, J, K] Calculation of the arc center coordinates.
- Circle Block Format Circle output format
- Sample Output Sample output code for the circle command.
- Limit Arcs

G-Code

- **Clockwise Arc Code** - Clockwise Arc Code
- **C-clockwise Arc Code** - Counter-Clockwise Arc Code

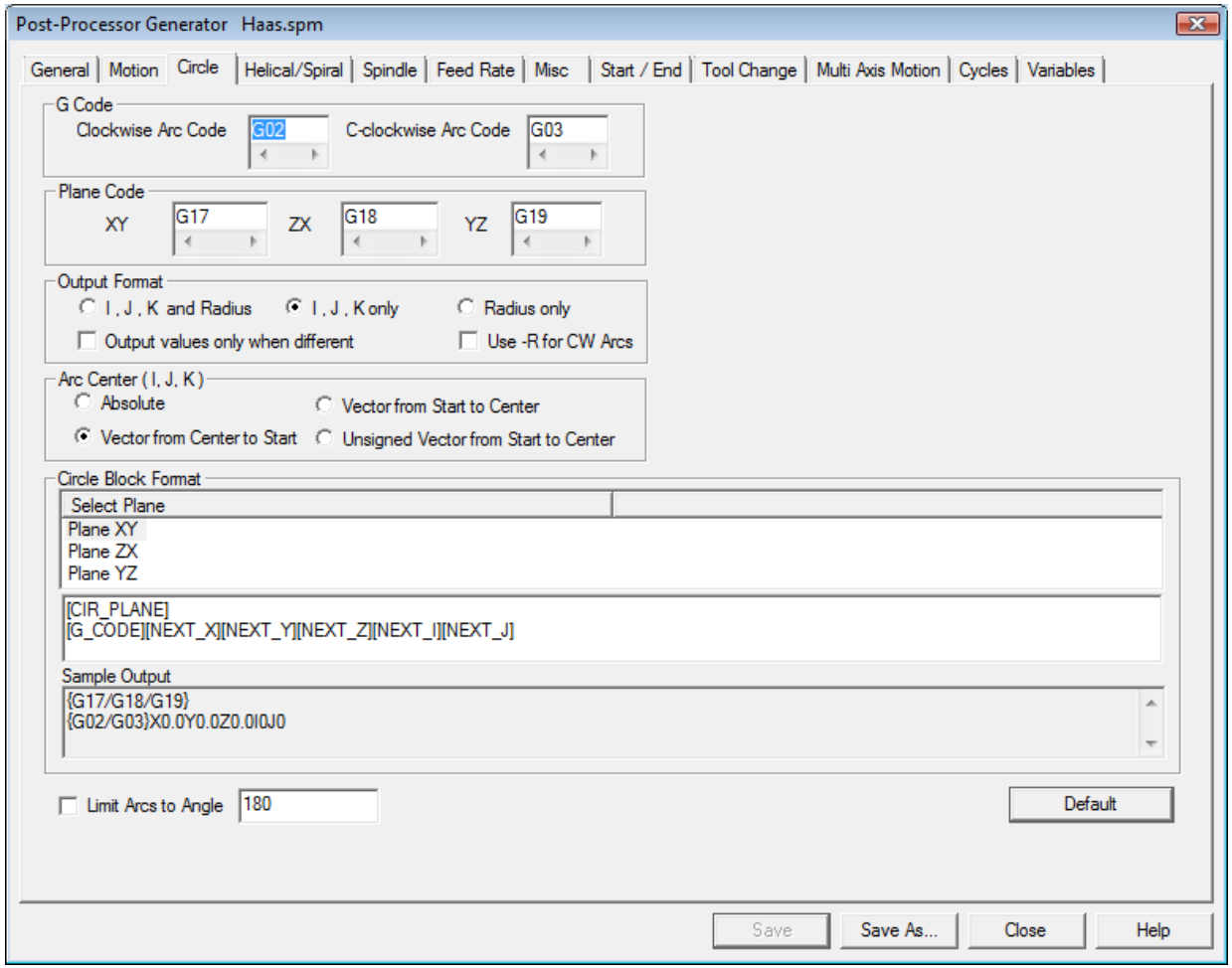
Plane Code - The G Code for the principal planes in which the Arc motion is output.

Output Format

Helps define the output format for the circle command.

- **Output I,J,K and Radius** Output I,J,K and Radius

- **Output Radius only** Output only Radius.
- **Output I,J,K only** Output only I,J,K.
- **Output values only when different** Output I,J,K,R values only when different from the previous values.



Arc Center [I, J, K] - Defines the calculation of the arc center coordinates. Absolute is the absolute center, Vector from Center to Start is (Center - Start), Vector from Start to Center is (Start - Center) and Unsigned vector is the unsigned distance between center and start.

Circle Block Format - Used to specify the block format for three different planes. Use the default button to see the default values and edit them if needed.

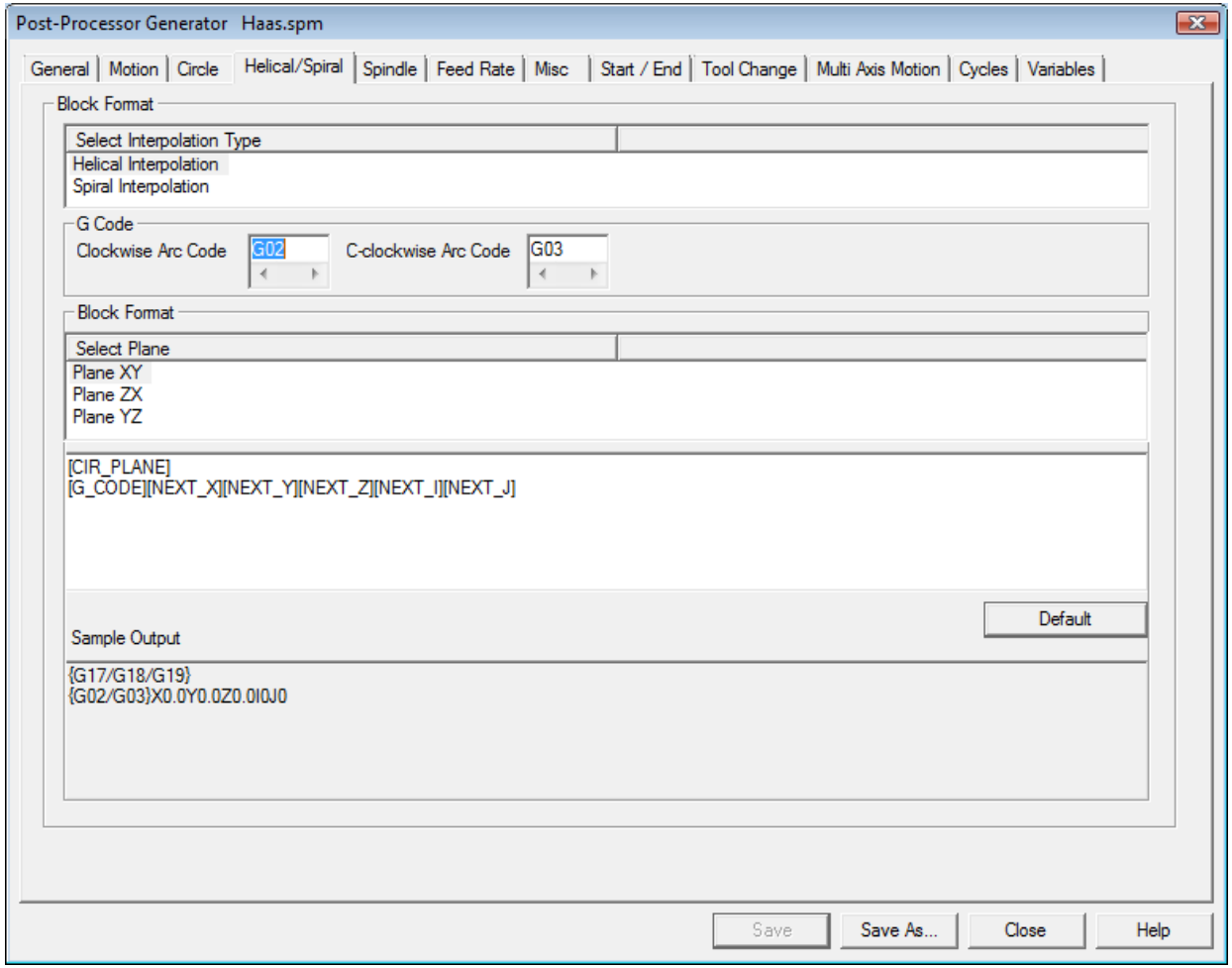
Notes:

Sample Output - This field displays sample output of the arc motion block. This field is non-editable. It indicates the parameters selected from the available options.

Limit Arcs - This is used to limit arcs to a certain angle. This is helpful for certain types of controllers which cannot output arcs greater than a certain angle

Notes:

Helix/Spiral Tab



For both these cycles, the Arc Center and the Plane code are the same as those defined in the Circle Section. Please define those first before defining these cycles.

Interpolation Type - Used to select the Interpolation type i.e. Define the parameters for the Helical Cycles or the Spiral Cycles.

G Code - Used to define the Clockwise and the Counter Clockwise Codes for the Helical or Spiral Cycles

Block Format - Used to specify the block format for three different planes. Use the default button to see the default values and edit them if needed.

Sample Output - This field displays sample output of the Helical/Spiral motion block. This field is non-editable. It indicates the parameters selected from the available options.

Notes:

Spindle Tab

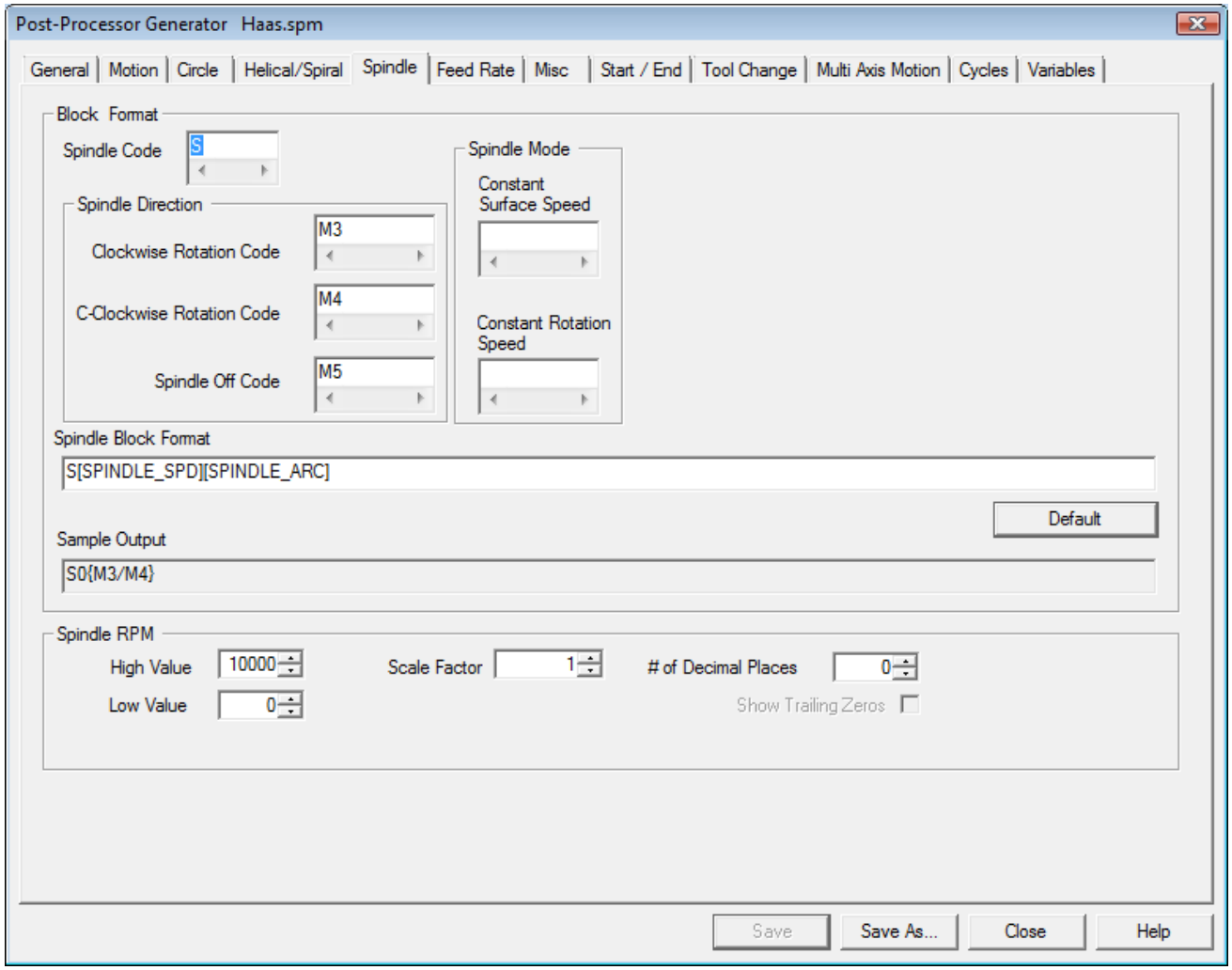
This tab sets the Spindle Speed parameters.

This Tab contains

- Spindle Block Characters for spindle block.
- Spindle RPM Parameter for the adjustment of spindle value.

Spindle Block

- **Spindle Code -** Register used for the feed rate value.
- **Spindle Direction -** Specify the direction code for the spindle
  - **Clockwise Rotation Code -** Clockwise spindle code
  - **C-Clockwise Rotation Code -** Counter clockwise spindle code
  - **Off Code -** Spindle off code
- **Block Format**
  - Sets the spindle block format
- **Sample output**
  - This filed displays sample output of the spindle block. It is a non-editable field.



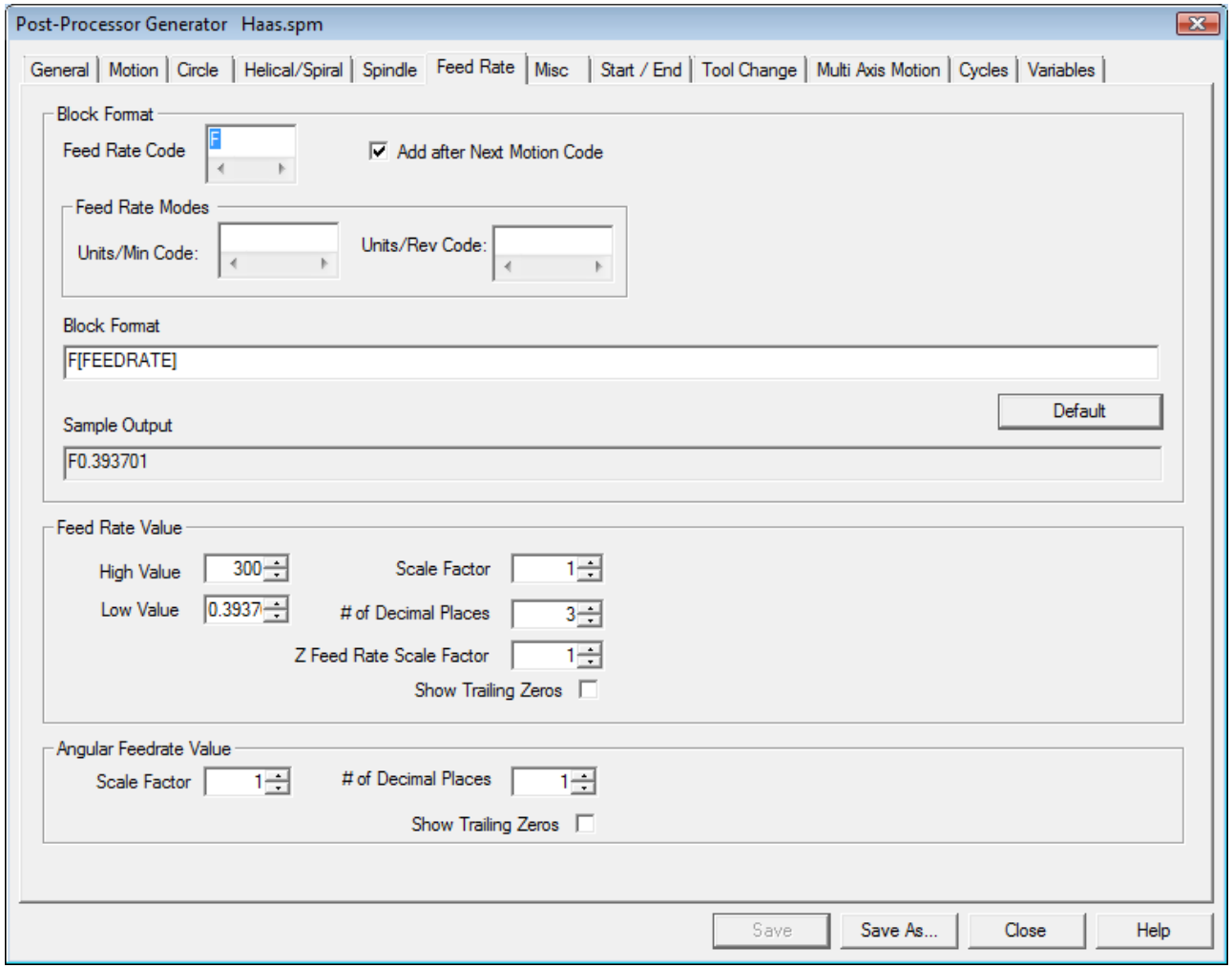
Spindle RPM

- **High Value** Maximum spindle value.
- **Low Value** Minimum spindle value.
- **Scale Factor** Scale factor of Spindle value.
- **# of Decimal Places** Number of digits output after the decimal point

Notes:

Feed Rate

This tab sets the feedrate parameters.



This Tab contains

- Feed Rate Block Characters for the feed rate block.
- Feed Rate Value Parameter for the adjustment of feed rate value.
- Sample Output Sample output of the feed rate block

Notes:

Notes:

Feed Rate Block

- **Feed Rate Code** Register used for the feed rate value.
- **Block Format**. Format for the feed rate block
- **Sample Output** Sample output displays sample code of the feed rate. It is a non-editable field.

Feed Rate Value

- **High Value** Maximum Feed rate value.
- **Low Value** Minimum Feed rate value.
- **Scale Factor** Scale factor of Feed rate value.
- **# of Decimal Places** Number of digits output after the decimal point
- **Z Feed Rate Scale Factor** Feed rate scale factor for Z Feed Rate (use [ZFEEDRATE] variable to get the Z feed rate)

Angular Feed Rate Value

- **Scale Factor** Scale factor for the Feed rate of 4th axis motions. (use [ROTATION_FEEDVALUE] variable to get the Angular feed rate)
- **# of Decimal Places** Number of digits output after the decimal point

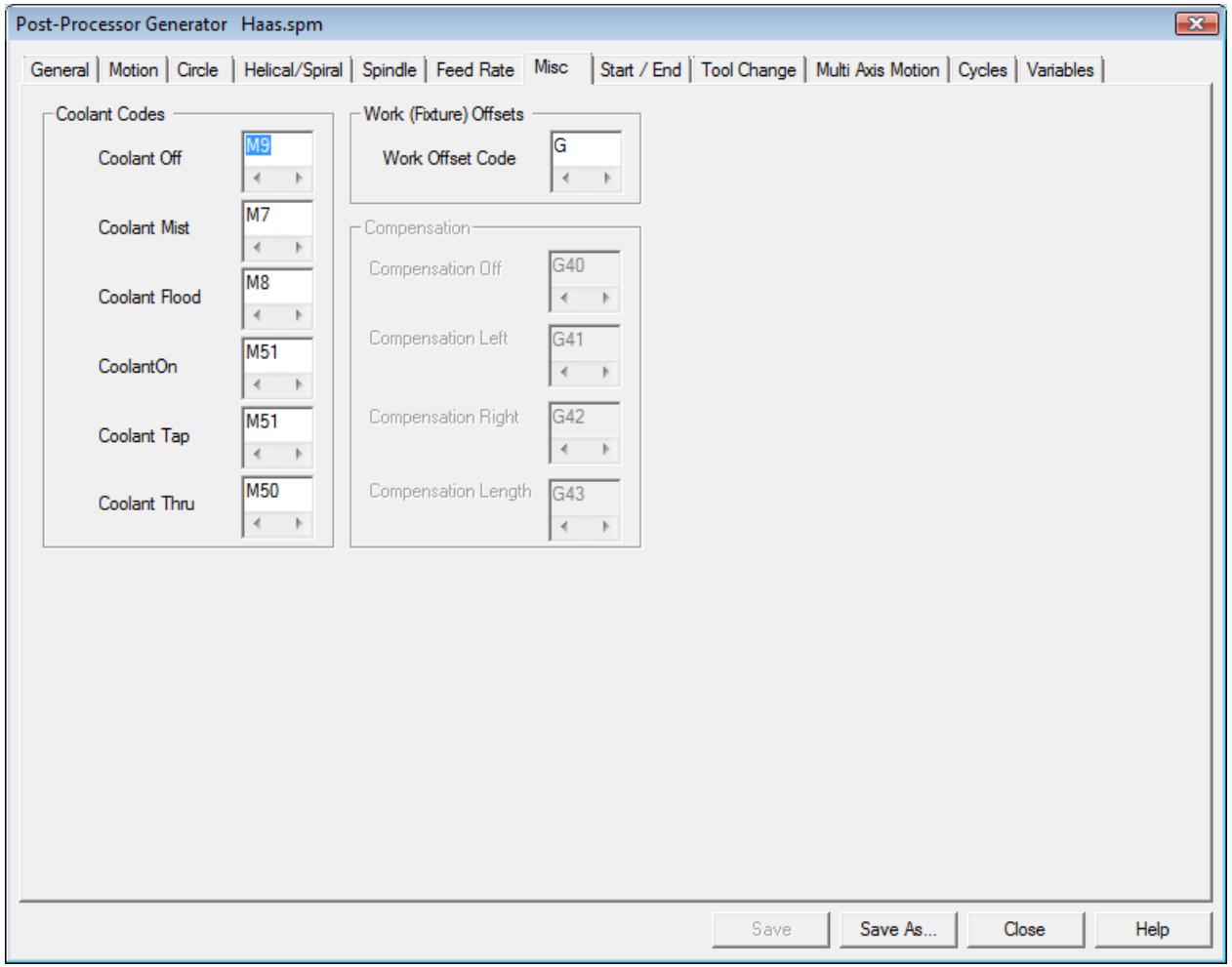
Miscellaneous Tab

This Tab contains the following parameters

- Coolant Code Characters for Coolant Block
- Compensation Codes of Compensation.

Coolant Codes

- **Coolant Off** Coolant Off code
- **Coolant Mist** Coolant Mist code
- **Coolant Flood** Coolant Flood code
- **Coolant On** Coolant On code
- **Coolant Tap** Coolant Tap code
- **Coolant Thru** Coolant Thru code



Work (Fixture) Offsets – Used to Specify the Work Offset Prefix code.

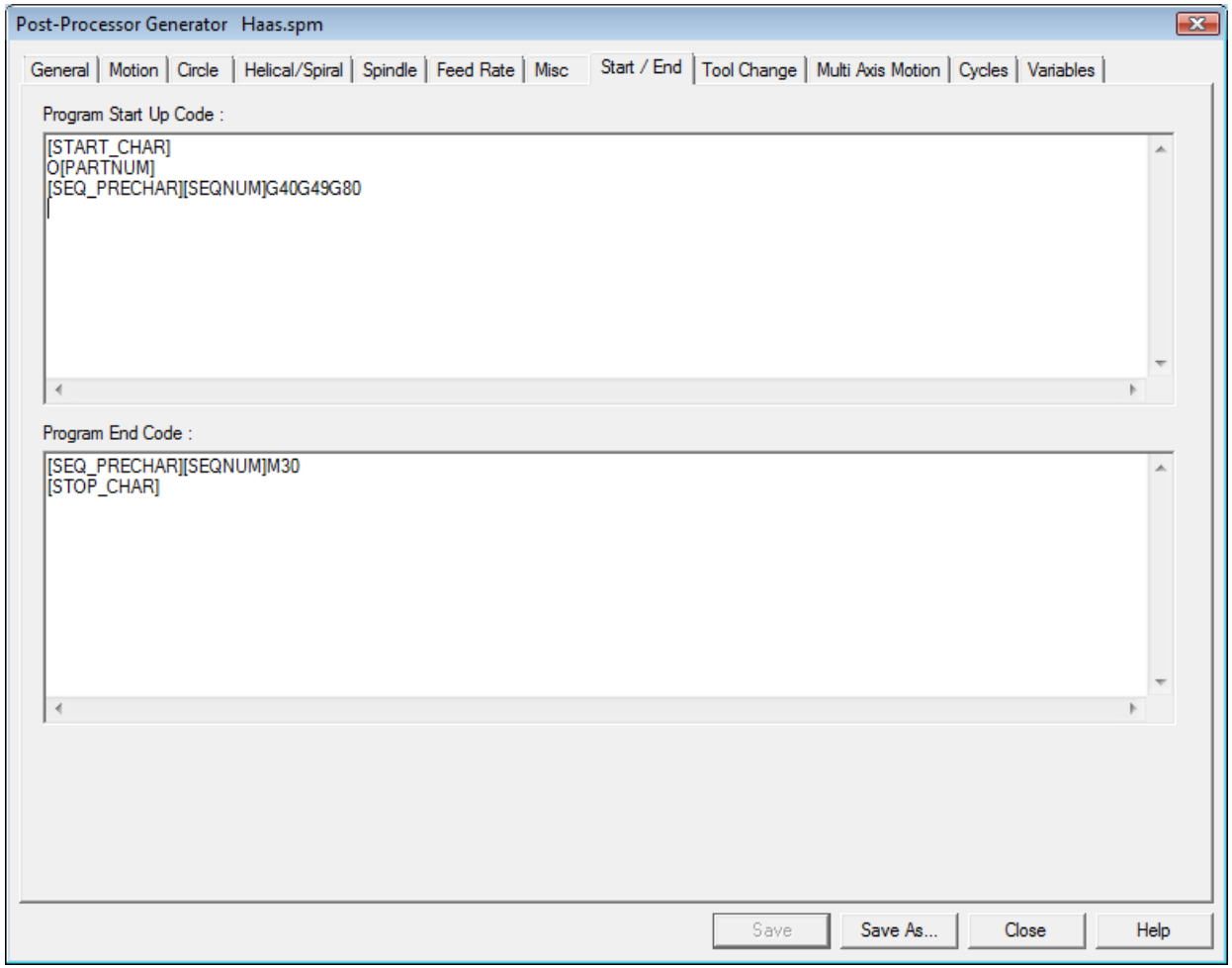
Compensation

- **Compensation Off** (This is not editable - reserved for future releases)
- **Compensation Left** (This is not editable - reserved for future releases)
- **Compensation Right** (This is not editable - reserved for future releases)
- **Compensation Length** (This is not editable - reserved for future releases)

Notes:

Start/End Tab

This tab is used to set the Program Start and End G-Code.



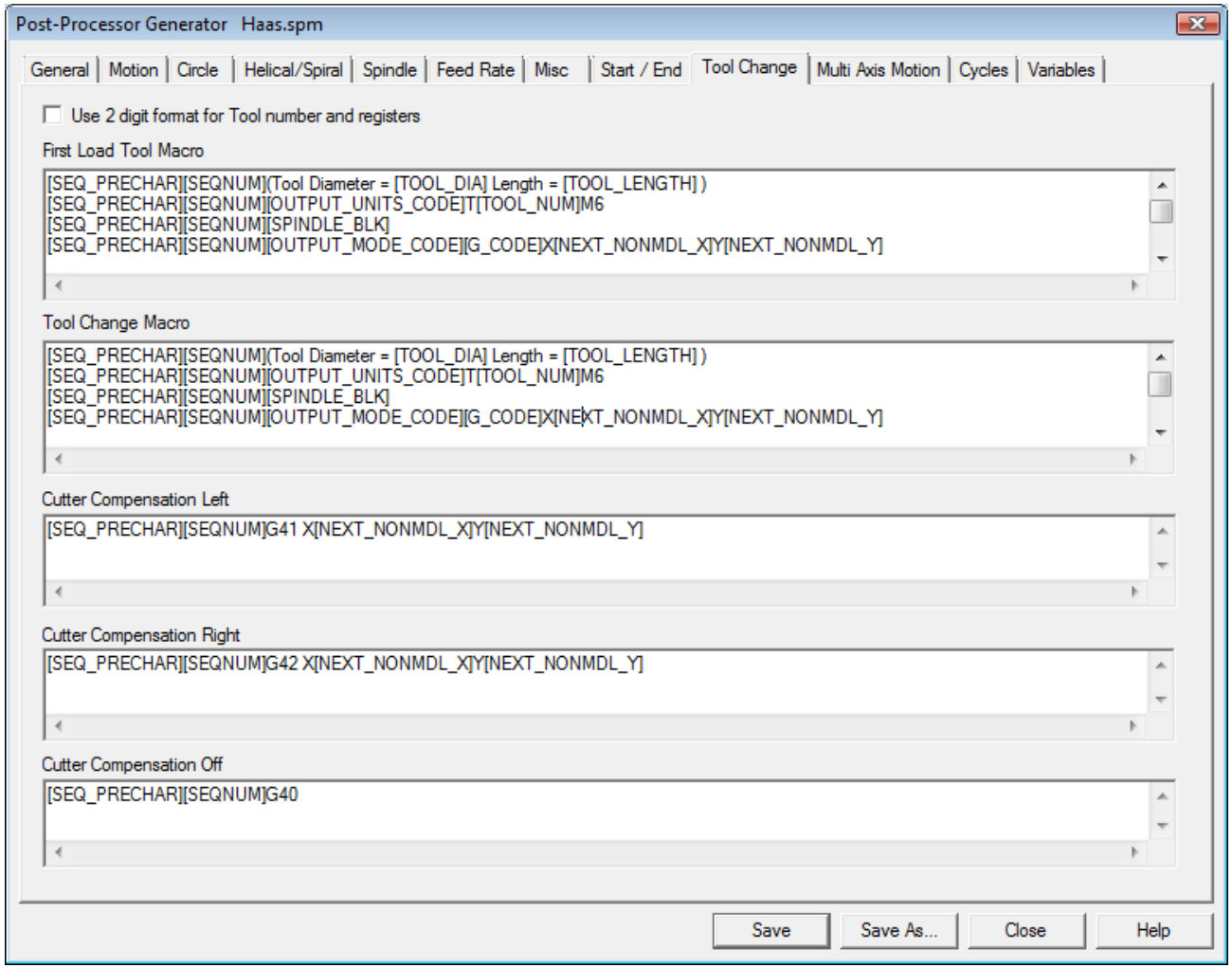
This Tab contains

- Program Start Up Code - First macro output in the generated NC file.
- Program End Code - Last macro output in the generated NC file.

Notes:

Tool Change Tab

This tab is used to set the **Load Tool** and **Tool Change** macros.



This Tab contains

- First Load Tool Macro: Macro for the first load tool command.
- Tool Change Macro: Macro for tool change command. (Not including the first load tool.)

Notes:

- Cutter Compensation Left Macro: Macro used to define the output when the cutter compensation left is detected in the output
- Cutter Compensation Right Macro: Macro used to define the output when the cutter compensation right is detected in the output
- Cutter Compensation Off Macro: Macro used to define the output when the cutter compensation cancel (off) is detected in the output

Notes:

Multi Axis Motion tab

This Tab contains the following parameters

- Rotation Axis Code Characters for Rotation Axis Code
- Rotation Direction Code Characters for Rotation Direction Code.
- Angle Value Angle Value for 4th Axis Motion.
- Motion Block Motion Code for 4th Axis Motion.
- Rapid Block Rapid Code for 4th Axis Motion.

Rotation Axis Code

- **A Axis** Rotation about A Axis
- **B Axis** Rotation about B Axis
- **C Axis** Rotation about C Axis

Rotation Direction Code

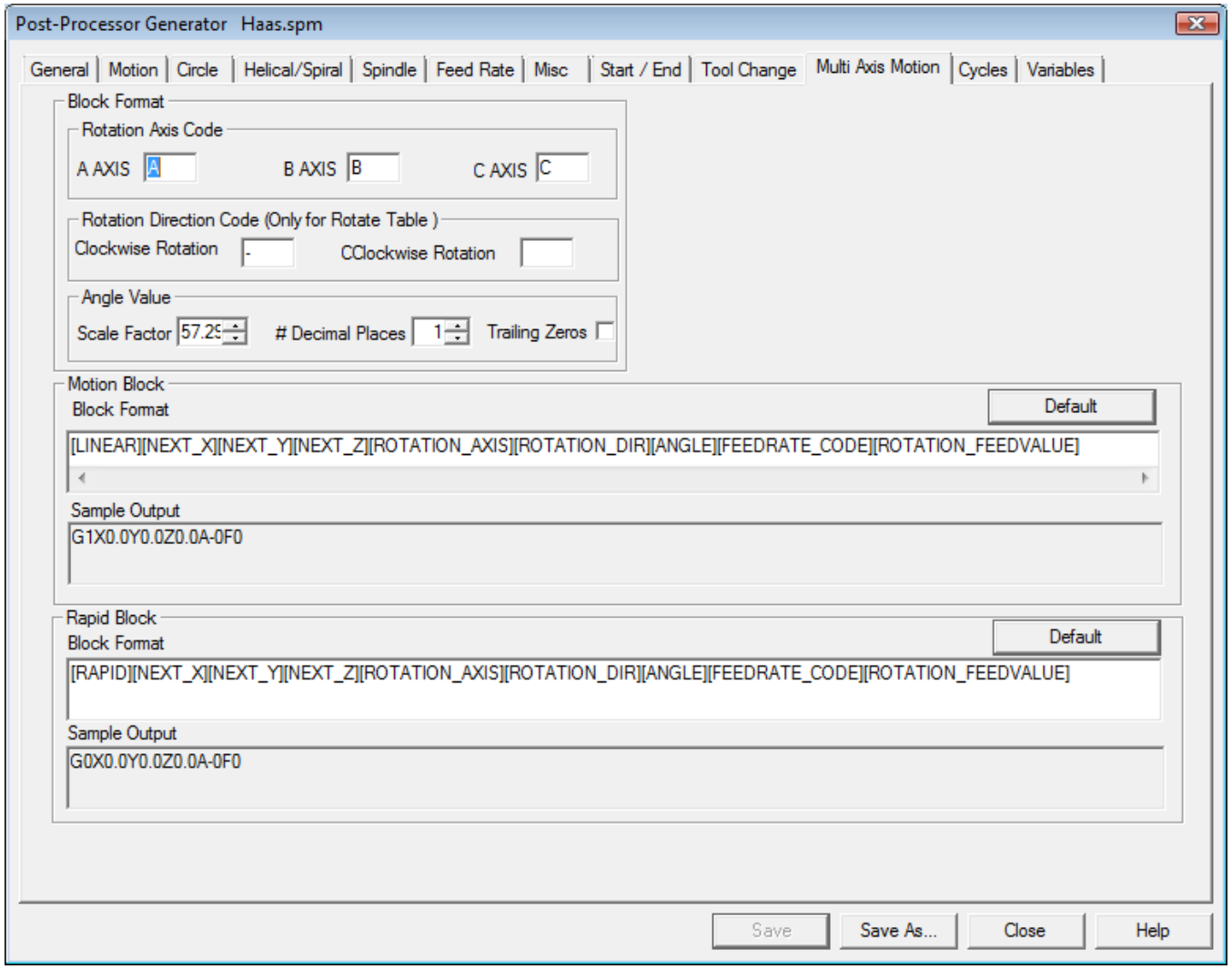
- **Clockwise Rotation** Clockwise rotation code
- **CClockwise Rotation** Counter Clockwise rotation code

Angle Value

- **Scale Factor** Scale Factor for Angle Value (the angle is in radians, to convert to degrees use a scale factor of 57.295779513082
- **# of Decimal Places** No of Decimal Places
- **Show Trailing Zeros** Number of Trailing Zeros after Decimal places

Motion Block Helps to define the output format for the 4th Axis motion code.

- Sample output displays sample code of the 4th Axis motion. It is a non-editable field.



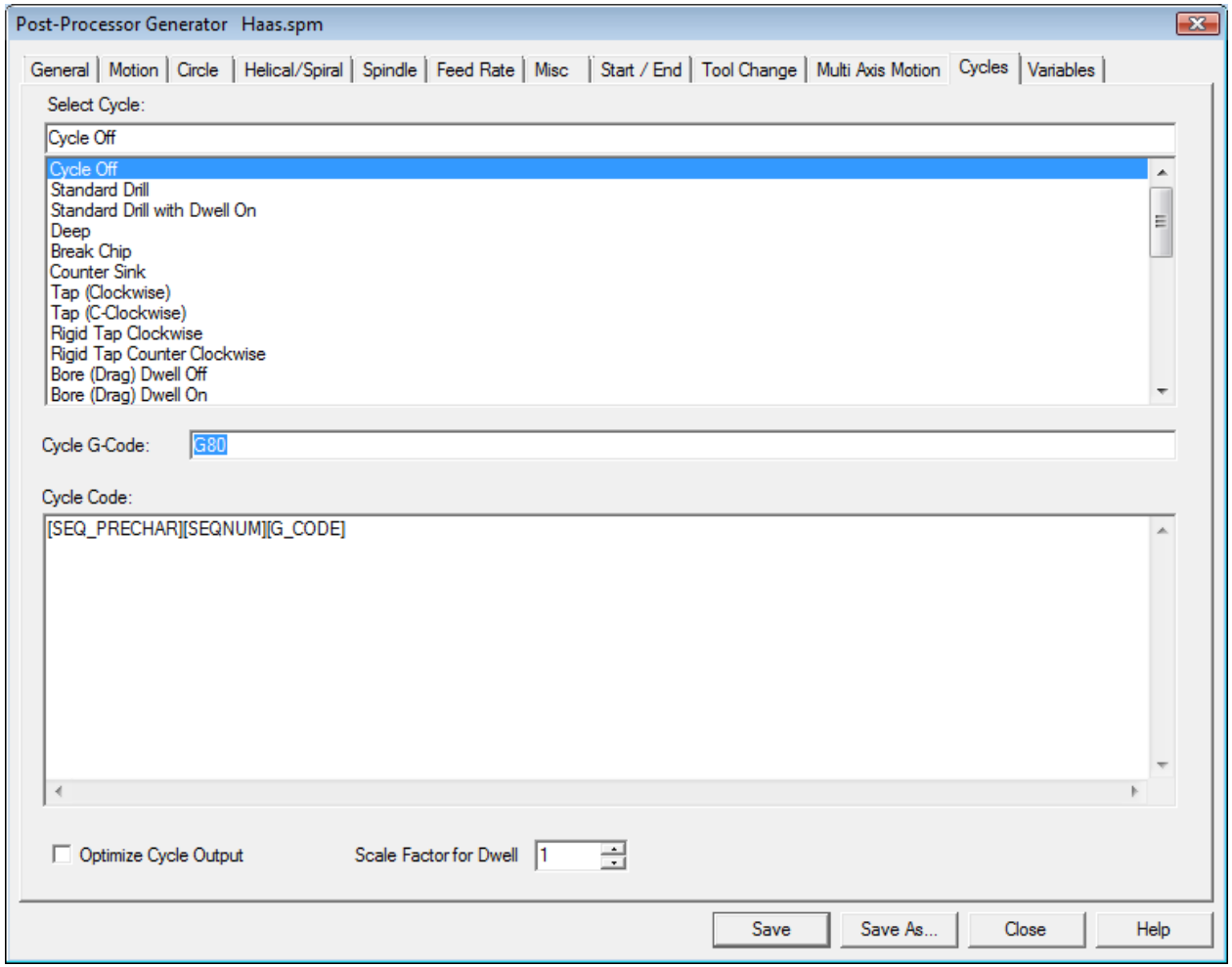
Rapid Block

- Helps to define the output format for the 4th Axis rapid code
- Sample output displays sample code of the 4th Axis rapids. It is a non-editable field.

Notes:

Cycles Tab

This tab is to set the **cycle** parameters in the RhinoCAM Post-Processor.



This Tab contains

- Cycle G-Code for cycles
- Cycle G-code Macros for the cycle commands. It represents the selected G-code value that defines each specific cycle. This value is displayed in the edit box below all the available options. It can be changed if required.

Notes:

Cycle

- **Drill Dwell** G-Code for Drill with a dwell value.
- **Drill No Dwell** G-Code for Drill without a dwell value.
- **Deep** G-Code for Deep Drill.
- **Break Chip** G-Code for Break Chip Drill.
- **Counter Sink** G-Code for Counter Sink Drill.
- **Tap Clockwise** G-Code for clockwise Tap.
- **Tap Counter Clockwise** G-Code for counter clockwise Tap.
- **Bore [Drag] Dwell On** G-Code for Drag Bore with a dwell value.
- **Bore [Drag] Dwell Off** G-Code for Drag Bore without a dwell value.
- **Bore [No Drag] Dwell On Orient On** G-Code for No Drag Bore with a dwell and an Orient value.
- **Bore [No Drag] Dwell On Orient On** G-Code for No Drag Bore with a dwell and without an Orient value.
- **Bore [No Drag] Dwell Off Orient On** G-Code for No Drag Bore without a dwell and with an Orient value.
- **Bore [No Drag] Dwell Off Orient Off** G-Code for No Drag Bore without a dwell and an Orient value.
- **Bore [Manual] Dwell On** G-Code for Manual Bore with a dwell value.
- **Bore [Manual] Dwell Off** G-Code for Manual Bore without a dwell value.
- **Bore [Reverse] Dwell On** G-Code for Reverse Bore with a dwell value.
- **Bore [Reverse] Dwell Off** G-Code for Reverse Bore without a dwell value.
- **Cycle Off** G-Code for Cycle Off
- **Scale Factor of Dwell** Scale factor for Dwell value.
- **Optimize Cycle output** will define the cycle format only once and will output the X,Y values for all the other holes. This will result in significant reduction in the file size for output.

Variables Tab

This tab lists all the **variables** used in the macros in the RhinoCAM Post Processor.

Editing Post Processor Variables

You can directly type the macro. With the exception of '[' and ']' characters as variables.

Notes:

Usage of Variable List Dialog

You can also add variables directly in the following manner.

	Operation	Dialog to operate
1	Click the right mouse button on the required edit box	Main Editor
2	Variable List Dialog is displayed.	Variable List Dialog
3	Select the position.	Main Editor
4	Select the variable to add. (This procedure is for adding) Double click the left mouse button on the variable list and the highlighted variable is added.	Variable List Dialog
5	Use the [Add] or [Undo] or [Undo All] buttons to perform the standard add and undo operations.	Variable List Dialog

Note: Multiple operations would require the user to repeat steps 3 to 5.

Class Room Exercise:

Select a post processor of your choice to edit.

- Add Cutter Compensation Macro's in the Tool Change tab
- Add Tool Change Coordinate position in the Tool Change tab

Notes: