Lista Studio srl
www.lista.it
Borgo Belvigo 33
36016 Thiene Vi ITALY
Tel. 0445,382056
info@lista.it

Welcome to Alibre CAM

MecSoft Corporation

Introduction

Welcome to Alibre CAM and thank you for choosing one of most powerful and easy to use complete CAD/CAM packages on the market today.

Alibre CAM is a unique CAM add-on product that runs inside of Alibre Design 10.0. This fully integrated product seamlessly integrates Alibre Design's 3D CAD functionality with toolpath generation and toolpath simulation. Alibre CAM's machining technology capabilities enable you to produce toolpaths that you can send to a machine with utmost confidence.

You can work with the native Alibre Design data as well as use any of the data types that can be imported into Alibre Design. Then you can use Alibre CAM 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.

Welcome to the Alibre CAM on-line help system. Please use the Table of Contents as a starting point to browse through the various topics of interest. You can also use the keyword search to locate topics of interest to you. Additionally, utilize the context sensitive help that is available through out the system to make your Alibre CAM sessions more productive.

This section contains a brief overview of Alibre CAM and the installation instructions. This is followed by the sections listed below:

Getting Started with Alibre CAM: This section briefly describes how to get started with Alibre CAM.

User Interface: This section describes each of the major user interface elements in Alibre CAM.

Overview

Alibre CAM is a unique CAM product plug-in that runs inside of Alibre Design 10.0. This fully integrated product seamlessly integrates Alibre Design's CAD functionality with toolpath generation and cutting simulation/verification, in one package that is both easy and fun to use. Alibre CAM'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 Alibre design data as well as use any of the data types that can be imported into Alibre Design such as solids, surfaces and meshes. Then you can use Alibre CAM 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.

Configurations of Alibre CAM 1.1

Alibre CAM 1.1 comes in four configurations – Alibre CAM, Alibre CAM Standard, AlibreCAM Professional and Alibre CAM Expert.

Alibre CAM Standard is a general-purpose machining program targeted at the typical machinist. It is ideal for the rapid-prototyping, general machining, hobby and educational markets. Alibre CAM Professional includes all the features of Alibre CAM Standard with the 4th axis Milling Capabilities. The Alibre CAM Expert product is an advanced version, suitable for mold, die and tool making, woodworking, and other complex applications. This configuration is suitable for machinists with sophisticated manufacturing requirements. The Expert version additionally supports 4th and 5th axis milling. All configurations support 2½ and 3 axis milling, as well as drilling.

Alibre CAM also comes in an Xpress configuration that includes 3 axis Parallel Finishing, toolpath animation and supports standard Mill tools (Ball End Mill, Flat End Mill & Corner Radius Tool)

The table below summarizes the toolpath generation features found in Alibre CAM 1.1.

	Alibre CAM	Alibre CAM Standard	Alibre CAM Professional	Alibre CAM Expert
2 1/2 Axis Milling				
Profiling	Yes	Yes	Yes	Yes
V-Carving	No	Yes	Yes	Yes
Hole Milling	No	Yes	Yes	Yes
Thread Milling	No	Yes	Yes	Yes
Engraving	No	Yes	Yes	Yes
Facing	No	Yes	Yes	Yes
Pocketing	No	Yes	Yes	Yes
Re-Machining	No	No	No	Yes
Advanced Pocketing	No	No	No	Yes
Advanced Profiling	No	No	No	Yes
Hole Making				
Drilling (includes Standard Drill, Deep Drill, Break Chip, Counter Sink)	Yes	Yes	Yes	Yes
Hole Sorting	Yes	Yes	Yes	Yes
User defined drilling cycles	Yes	Yes	Yes	Yes
Automatic hole Selection	No	Yes	Yes	Yes

Tapping	No	Yes	Yes	Yes
Boring	No	Yes	Yes	Yes
Reverse Boring	No	Yes	Yes	Yes
Post-Processors				
User customizable post-processor generator	Yes	Yes	Yes	Yes
User Defined Cycles	Yes	Yes	Yes	Yes
Arc Output	Yes	Yes	Yes	Yes
Simulate Cycles	Yes	Yes	Yes	Yes
Helix Output	No	Yes	Yes	Yes
Spiral Output	No	Yes	Yes	Yes
5-Axis	No	No	No	Yes (Extra Cost Item)
Toolpath Simulation				
Toolpath Animation	Yes	Yes	Yes	Yes
Tool Holder Simulation	Yes	Yes	Yes	Yes
Cut Material Simulation	No	Yes	Yes	Yes
Advanced Cut Material Simulation	No	No	Yes	Yes
Tool Holder Collision Detection	No	No	Yes	Yes
Part/Stock Comparison	No	No	No	Yes
Tools				
Standard Mill Tools (Ball, Flat, Corner Radius)	Yes	Yes	Yes	Yes
Standard Drill Tool	Yes	Yes	Yes	Yes

Taper Mill, Thread Mill, Face Mill	No	Yes	Yes	Yes
Center Drill, Reamer, Tap Tool, Bore & Rev Bore Tools	No	Yes	Yes	Yes
Dove Tail, Lollipop & Form Tools	No	No	No	Yes
4 Axis Milling				
4-Axis Indexed Machining	No	No	Yes	Yes
4-Axis Continuous Rotary Machining	No	No	Yes	Yes
4-Axis Continuous Rotary Engraving	No	No	Yes	Yes
4-Axis Parallel Roughing	No	No	Yes	Yes
3 Axis Milling				
Horizontal Roughing	Yes	Yes	Yes	Yes
Parallel Finishing	Yes	Yes	Yes	Yes
Horizontal Finishing	No	Yes	Yes	Yes
Plunge Roughing	No	No	No	Yes
Horizontal Re-roughing	No	No	No	Yes
Plunge Re-roughing	No	No	No	Yes
3 Axis Pocketing	No	No	No	Yes
Pencil Tracing, including Flat Mills	No	No	No	Yes
Valley Re-machining	No	No	No	Yes
Plateau Machining	No	No	No	Yes
Steep Area Parallel Machining	No	No	No	Yes
Steep Area Horizontal Machining	No	No	No	Yes

Curve Machining	**No**	**No**	**No**	**Yes**
Spiral Machining	**No**	**No**	**No**	**Yes**
Radial Machining	**No**	**No**	**No**	**Yes**
Between 2 Curves Machining	**No**	**No**	**No**	**Yes**
Reverse Post Machining	**No**	**No**	**No**	**Yes**
Horizontal Hill Machining	**No**	**No**	**No**	**Yes**
5 Axis Milling				
Indexing	**No**	**No**	**No**	**Yes**
Toolpath Editing				
Toolpath Graphical Editing	**No**	**No**	**No**	**Yes**
Toolpath Instancing	**No**	**No**	**No**	**Yes**
Toolpath Arc Fitting	**No**	**No**	**No**	**Yes**

System Requirements

- Intel Pentium compatible computer
- Windows 2000 SP2 or later, XP Professional or Home, or Vista
- At least 512 MB RAM.
- Alibre Design 10.0
- Approximately 100 MB of hard disk space.

Please keep in mind that if you are working with large models these minimum requirements might not be sufficient for adequate performance. You will have to upgrade either your computer's memory (RAM) capacity and/or your computer's main processor speed.

Installing Alibre CAM

To install Alibre CAM software, follow these instructions:

1. First make sure a valid copy of Alibre Design is installed on your computer
2. Double click on the Alibre CAM installation program. This will launch the installation wizard that will install Alibre CAM on your computer.

3. Please choose a destination folder to install Alibre CAM on your computer.
4. Once you select the installation folder the install program will automatically install Alibre CAM on your computer and initialize the environment for successfully running Alibre CAM.

Registering Alibre CAM

Upon successful installation, Alibre CAM will operate in Demo mode. In demo mode you will still be able to run Alibre CAM but will not be able to do the following:

1. Save CAM data to the Alibre Design files
2. Post-process created toolpaths
3. Export CAM data to VisualMill

To enable these functions, Alibre CAM needs to be registered with MecSoft and valid license codes obtained before it can become operable again.

To register Alibre CAM, launch the product. Once Alibre CAM is loaded and ready, you will see the **Enter License Codes** dialog shown below, if you have not already registered the product. You can alternatively access this dialog by selecting the Alibre CAM option in the Alibre menu bar and choosing Register Alibre CAM. The **Tries Left** field indicates the number of times you can run Alibre CAM before it starts operating in demo mode.

Lista Studio srl

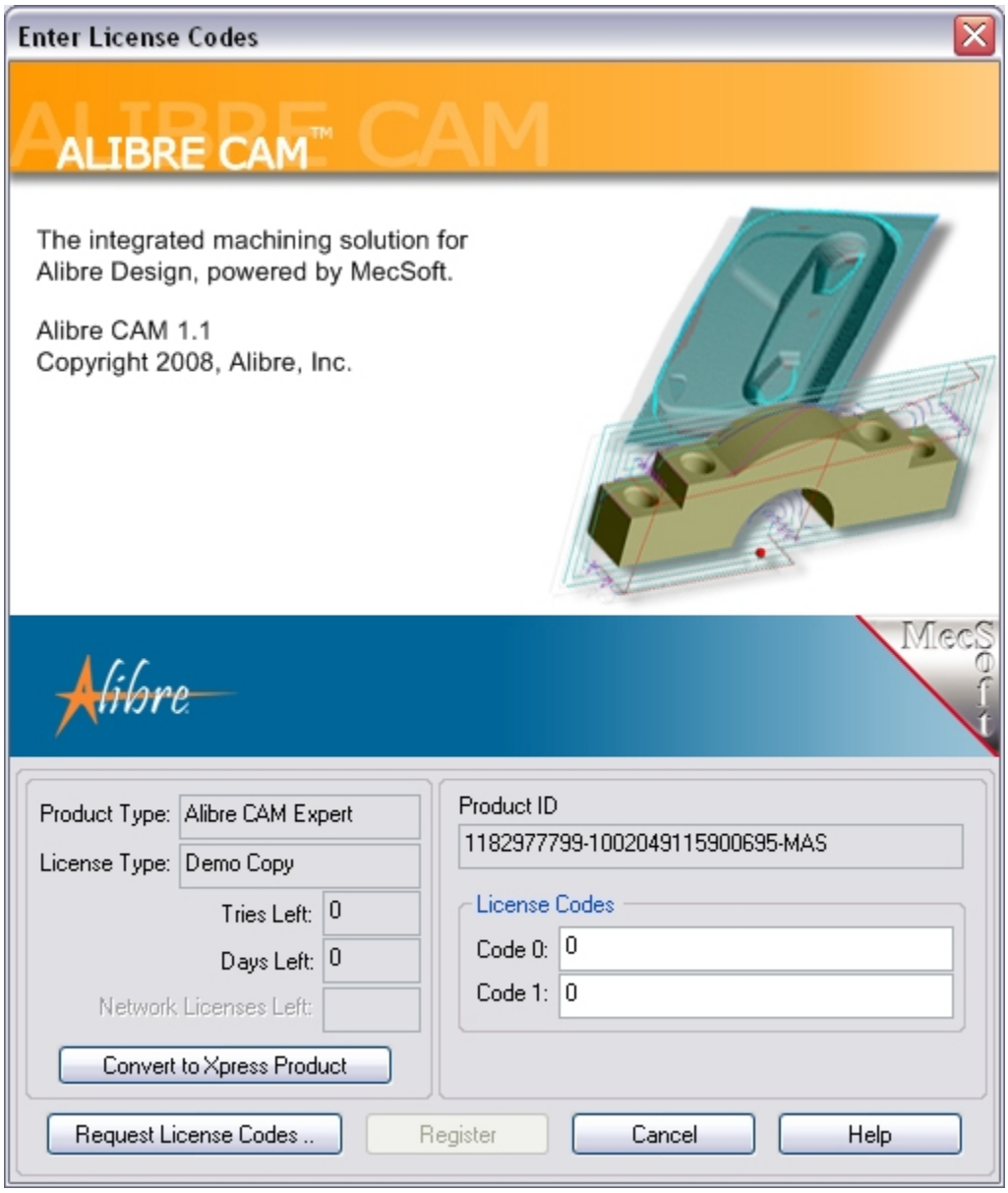
www.lista.it

Borgo Belvigo 33

36016 Thiene Vi ITALY

Tel. 0445,382056

info@lista.it



Note: *This registration dialog can also be invoked from the Alibre CAM item in the Alibre menu bar.*

To obtain license codes you must register the product using the Web form available at www.mecsoft.com. You can automatically launch this web form by selecting the **Request License Codes** button in the dialog. This will launch a web page that will present you with a form. Please make sure you provide all the required information and select the submit button to request license codes.

If you purchased this product from Alibre Inc., MecSoft Corporation or one of their resellers please make sure you submit the same email address that you used during your purchase process so that we can identify you as a valid customer. If you have purchased the product through an authorized MecSoft reseller, please obtain the license codes from your reseller. In addition to this information make sure you also provide the Product ID that is shown on the registration dialog shown above.

Once we identify you as a valid customer we will send you back an email with two codes, Code 0 and Code 1 that you need to input in the registration dialog shown above to register the product successfully. In order to expedite this process please make sure you specify accurately all the requested information in the form.

Alibre CAM can be optionally run using the Alibre CAM Xpress configuration without requiring license codes from MecSoft. You can convert your product to run the Xpress configuration at any time by simply selecting the **Convert to Xpress Product** button on the registration dialog. Once the product is running in the Xpress configuration, this button text will be changed to **Convert to Licensed Product**. Selecting this button again will revert the product back to its original configuration.

Running Alibre CAM

Click on the Windows **Start** button and select **Programs**. Point to the **Alibre Design** program. Once you load Alibre Design, select New->Part. This loads a New Part workspace. From the Menu bar select **Alibre CAM,** then **Browser**.

Troubleshooting Alibre CAM Installation

If you have followed the installation steps outlined in the installation section correctly and are unable to load and run Alibre CAM follow the troubleshooting steps outlined in the Appendix to correct the problem.

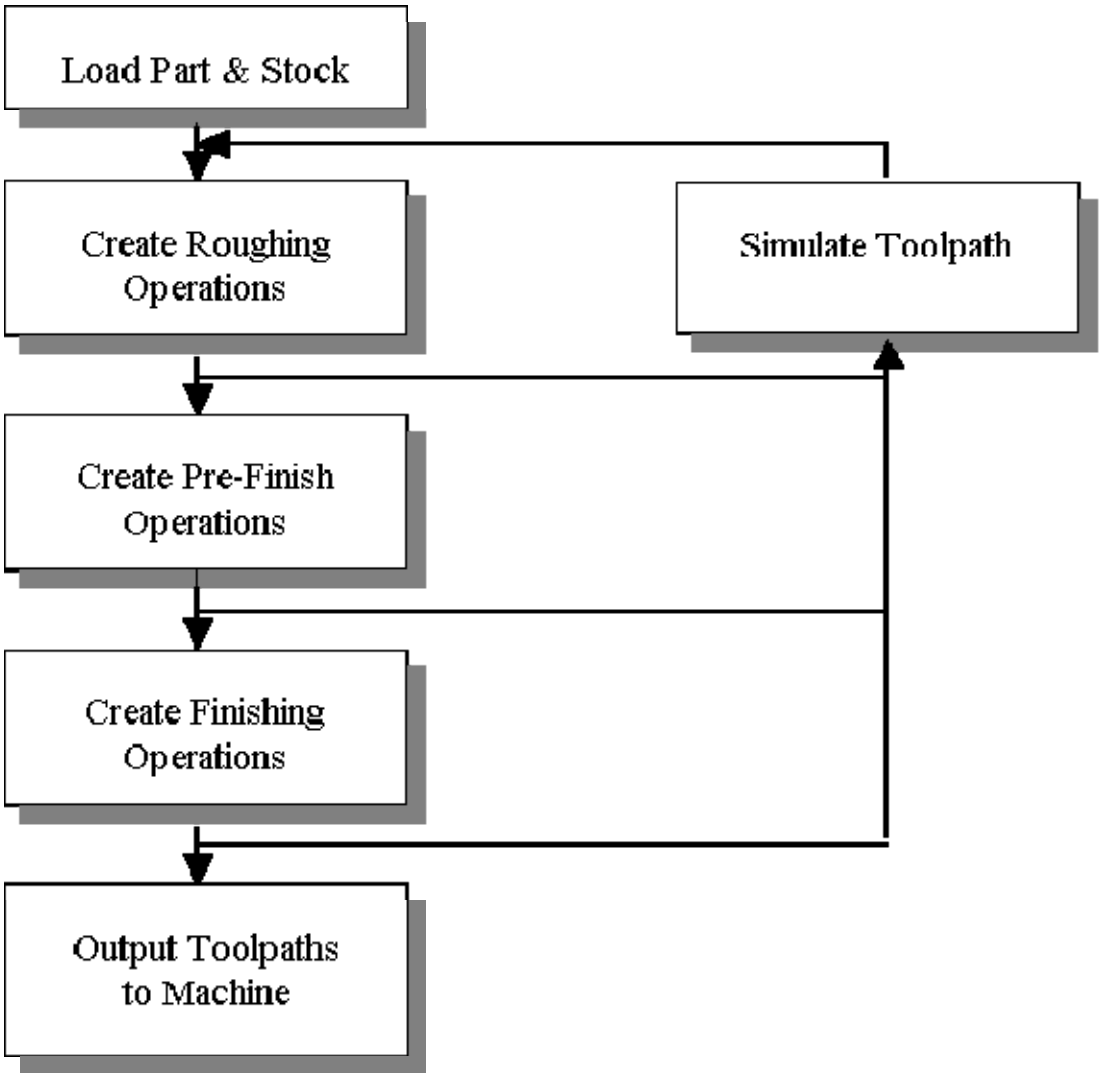
Getting Started with Alibre CAM

Welcome to Alibre CAM. By this time you have installed Alibre CAM on your system and are ready to run the product from within Alibre Design. Alibre CAM's installation directory contains a tutorials folder that contains part files that you can use to get yourself familiar with the product. Additionally, the online help system provides comprehensive help topics as well as context sensitive help to help you become a productive user of Alibre CAM.

Alibre CAM Workflow

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



Part geometry is loaded into Alibre CAM via the various data interfaces provided in Alibre. A stock model representing the raw stock from which the part needs to be manufactured can then be created using the various tools provided in Alibre CAM.

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 Alibre CAM. 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.

Typical Scenario

In Alibre CAM the part can be rough machined by employing a horizontal roughing operation, followed by a pencil trace 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. A pencil trace operation with a ball tool of the same radius as the roughing cutter can be used then to relieve the corners of the part before running a semi-finish operation.

The part can then be pre-finished by employing either the contour machining operation or the horizontal finishing operation. Pre-finishing and finishing operations typically employ ball end mills with or without a side angle. Re-machining operations to remove material from steep, flat and steep areas can then be employed to make sure that there is uniform material left on the part. Finally a finish contour 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 complex 3D parts, additional finish operations may be necessary. For example, **Parallel Finishing** operation can be defined using a small ball tool with a fine step-over value. Additionally certain areas of the part geometry may need some special machining requirements. For such conditions, containment regions can be used to restrict the area of milling.

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.

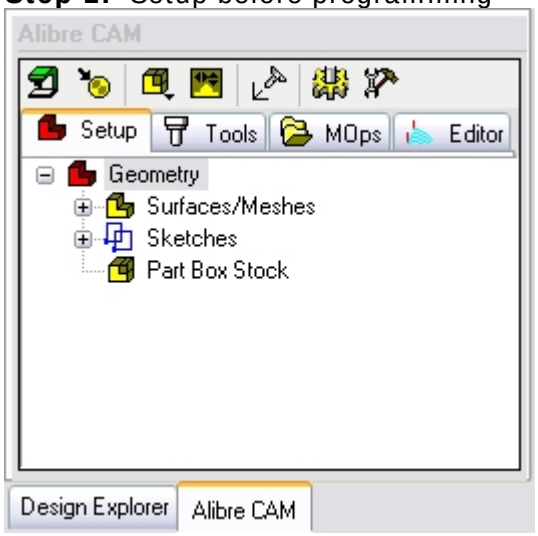
This machining strategy is what you program using Alibre CAM. You can also simulate toolpath to visualize how the cutter (Tool) moves around model at any time during the process. This provides valuable feedback that can help you choose the most appropriate machining strategy.

Programming Workflow

Once the part is loaded, the typical workflow is reflected in the layout of the icons in the **MOps** tab of the Browser. The workflow is designed to allow the user to work starting from the left most tab and ending at the right most tab. The functions in each of the toolbars corresponding to each tab are also best accessed in order from left to right.

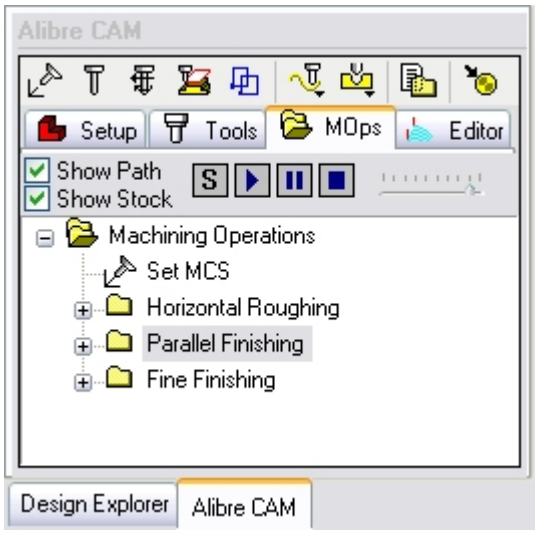
Thus the user typically would start with the **Setup** tab and access each of the buttons, optionally, in the toolbar that appears when this tab is selected in sequence from left to right. Once the setup functions are completed, the user will then proceed to the **Tools** tab to create, select and save tools to be used in the machining. After this the user will proceed to the **MOps** or **M**achining **Op**eration**s** tab and commence programming the part. Once a program is completed the user can simulate the toolpath on the **MOps** tab before sending it to the machine tool.

Step 1: Setup before programming

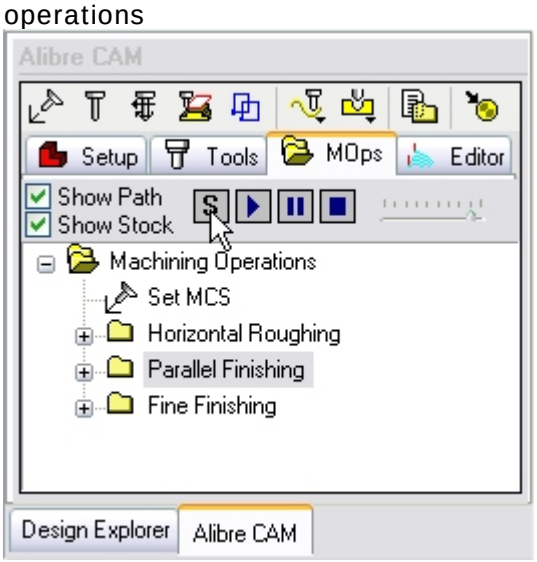


Step 2: Create, select and save tools

Step 3: Create machining operations



Step 4: Simulate machining operations

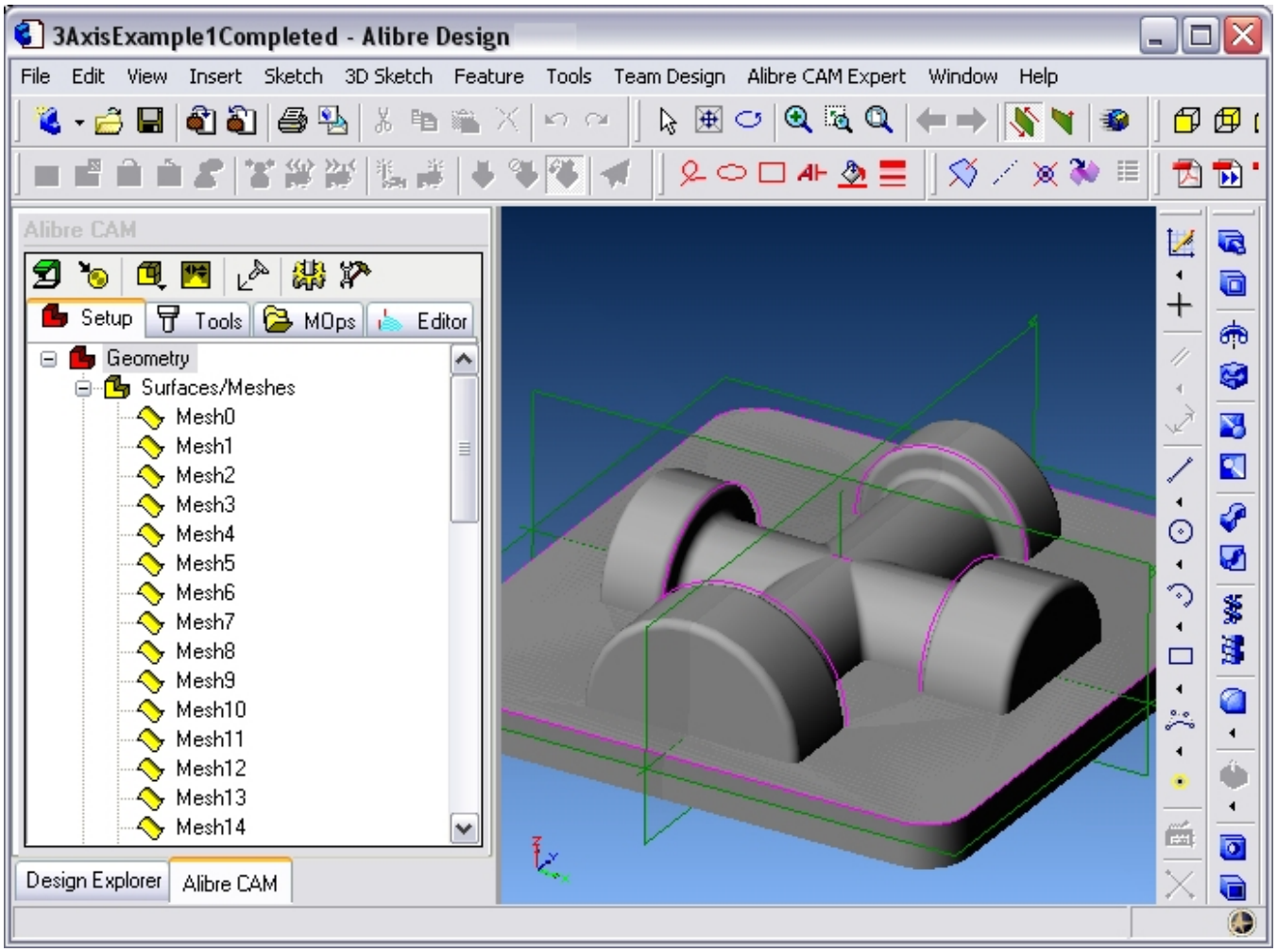


Post-Processing

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.

User Interface

Alibre CAM adheres to the Windows as well as the Alibre Design standard for user interface design. A screen shot of the Alibre CAM product plug-in running inside Alibre Design is shown below.

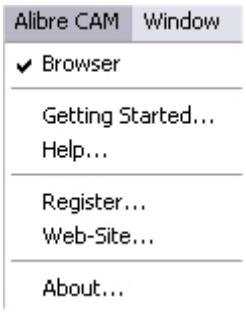


There are 2 main interface objects created when Alibre CAM is loaded

1. Alibre CAM menu bar item in the Alibre Design menu bar
2. Alibre CAM browser window

Alibre CAM Menu Bar Item

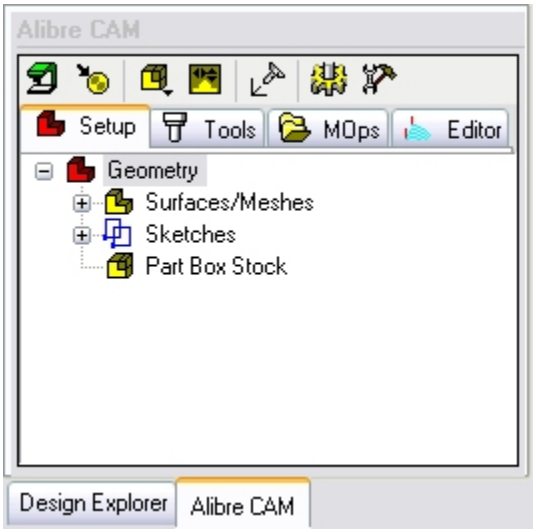
When Alibre CAM is invoked it will add a menu bar item, titled **Alibre CAM** to the main Alibre Design menu bar. Selecting this menu bar item will create a drop down menu as shown below.



Alibre CAM Browser Window

The Browser is docked inside the Alibre Design Explorer window and provides management of various entities or objects that can be created in Alibre CAM. Users can use the tabs to toggle between the Alibre Design Explorer and Alibre CAM. By default, this window will appear docked on the left hand side of the Alibre display when Alibre CAM comes up. This window can be resized by resizing the Design Explorer window.

This window has three main modes of operation represented by tabs at the top of the window. These are **Setup, Tool,** and **MOps**. Selecting each of these tabs allows different views of objects in the Alibre CAM database. In addition each tabbed view also incorporates a context sensitive toolbar at the top. These toolbars are groups of functions that are associated with the type of object(s) in the tab. In addition to the 3 tabs, Alibre CAM Expert Product includes a **Toolpath Editor** tab that allows the user to graphically edit a toolpath.



Users can switch between the Design Explorer and Alibre CAM by switching tabs at the bottom of the Alibre CAM browser.

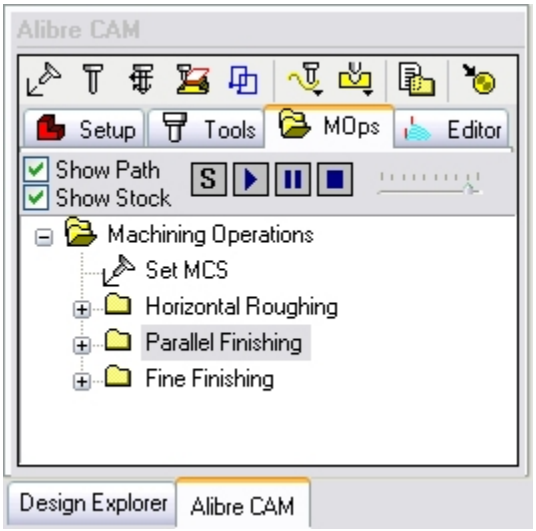
For an in-depth description of each of the buttons in the toolbars please refer to sections below.

Browser

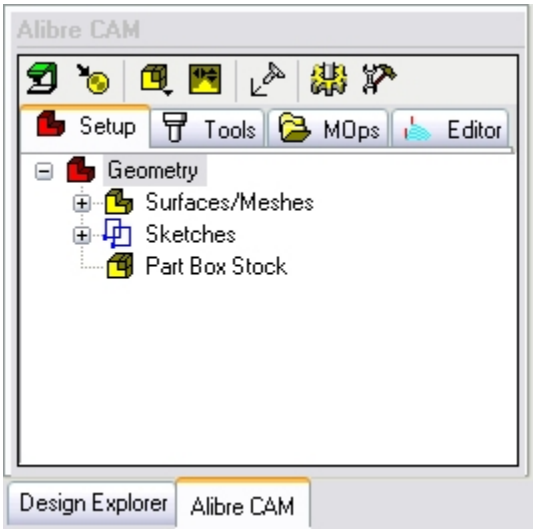
The Browser appears to the left of the screen by default. The Browser contains the following four tabs.

Note: Alibre CAM Standard and Alibre CAM Professional products include Setup, Tools and MOps tabs only. Toolpath Editor tab is available in the Alibre CAM Expert Product only.

As mentioned above, selection of each tab in the Browser window will locate a toolbar specific to the selection. This section describes the buttons in each of these toolbars.



Setup Tab Toolbar



The Setup manager displays the three types of geometry that can be created and manipulated in Alibre CAM: **Surfaces/Meshes**, **Sketches** and **Stock**

The first icon represents the **Part**. For an imported part, the full path is indicated. If the part consists of solids, each face is represented as a **Mesh**.

Sketches in the model are regions used to define machining boundaries. If you have created machining operations, the toolpath for the respective curve will appear underneath it.

Lastly, the **Stock** icon indicates the type of stock. You can double-click or right-click to create a different type of stock, or delete the stock. A red star next to this icon indicates that the work-in-progress stock model corresponding to this operation needs to be created.

 **Machine Setup:** Define the machine tool and specifying its tool change position

 **Set Post Options:** Set the path for the post processor files and program to be used for displaying the posted file

 **Create/Load Stock:** Create/load a stock material from the list of stock types

 **Locate Part within Stock:** Locate stock relative to part geometry.

 **Set MCS:** Set Machine Coordinate System (Machine Zero)

 **CAM Preferences:** Set up Machining Color and Simulation Preferences

 **CAM Utilities:** Access to Post Processor Generator, Knowledge base & more.

Tools Tab Toolbar

This tab lists all tools currently defined in the file. If you have created machining operations, the toolpath will appear in the **Tools** tab underneath the tool it uses. You can rename and delete tools, but you cannot delete a tool that is used in a toolpath. Double-click a tool icon to edit its parameters.

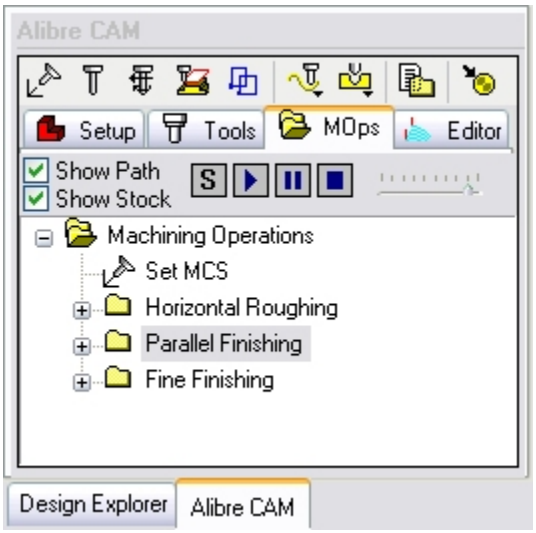
Create/Select Tool: Launches the Tool Creating dialog to create Milling and Drilling tools.

Load Tool Library: Load tools from External Library.

Tools Info: Lists information on all tools

Save Tool Library: Save current tools to library

MOps Tab Toolbar



"MOps" stands for Machining Operations. All toolpaths you create are listed here, in order of creation. Within each toolpath folder you can edit its various components, such as tool, regions, or cut parameters, by double-clicking the relevant icon. Right-clicking on a toolpath name provides several options, including simulation, generation, and post-processing.

If you make any changes to a toolpath's parameters, the yellow folder icon for that toolpath will turn red. This indicates that the toolpath needs to be regenerated.

Set MCS: Set Machine Coordinate System (Machine Zero)

Create/Select Tool: Launches the Tool Creating dialog to create/select Milling and Drilling tools.

Set Feeds/Speeds: Defines the feed and speed rates for cutting, rapid, approach, engage, retract, and depart tool motions.

Clearance Control: Sets the level away from the part for safe rapid tool motion.

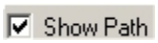
Select Regions: Provides several methods for selecting curves that will act as machining boundaries.

Milling Methods: Choose the type of toolpath you want to create.

Hole Machining: Choose the type of Hole machining toolpath you want to create.

Machining Operations Info: Displays information like machining operation name, cut feed, machining time for each machining operation.

 **Post Process:** Sends the toolpath code to the machine.

 **Show Path:** Displays Toolpath

 **Show Stock:** Displays Stock Geometry

 **Simulate**: Loads the Alibre CAM Stock Simulation window. (Available in Alibre CAM Standard, Expert and Professional versions only)

 **Animate**: Shows the Toolpath Animation

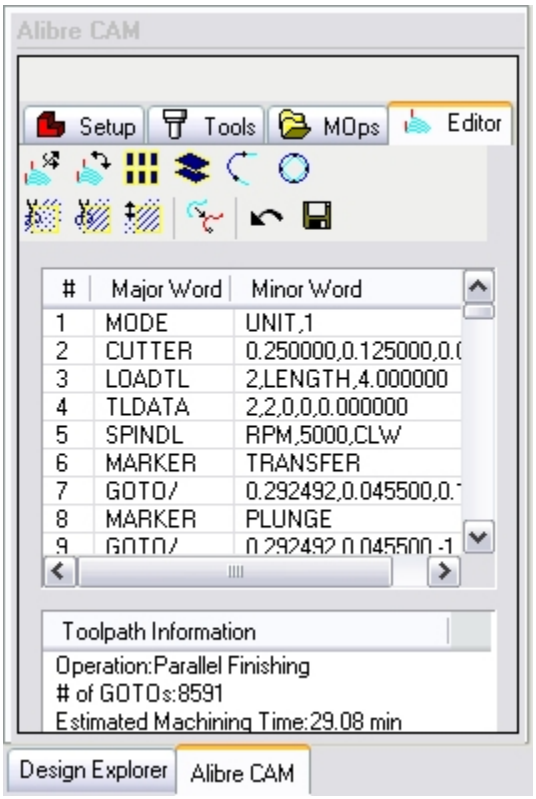
 **Simulation Speed:** Varies the simulation/animation speed

 **Pause:** Pause simulation/animation

 **Stop:** Stop and reset animation.

Toolpath Editor Tab Toolbar

Available in Alibre CAM Expert Only.



Refer to the Editing Toolpaths (Toolpath Editor) section for detailed description

Overview of Machining Methods

There are two major classes of machining operations that can be created in Alibre CAM - 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.

Milling operations can be categorized as 2½ axis, 3 axis milling.

- **2½ Axis:** The tool can move in X and Y directions, while the Z level is fixed at set location
- **3 Axis:** The tool can move simultaneously in all three directions.
- **4 Axis:** The tool stays normal to the rotary axis and the part rotates about X (A) axis or Y (B) axis.
- **5 Axis:** Index 5-axis machining, also known as 3+2 machining. 2 of the tool axis are oriented at a certain angle and use a combination 2 ½ & 3-axis machining.

These categories, and the available operations, within them are described in the sections to follow.

Two and a Half Axis (2½) Axis Operations

In 2½ axis toolpath, the tool can move in X and Y directions, but Z movements are limited to set levels. 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.

A typical machining operation would involve roughing then finishing. Unlike the more complex 3 axis parts, detailed finishing is typically not necessary for prismatic parts. Roughing is typically

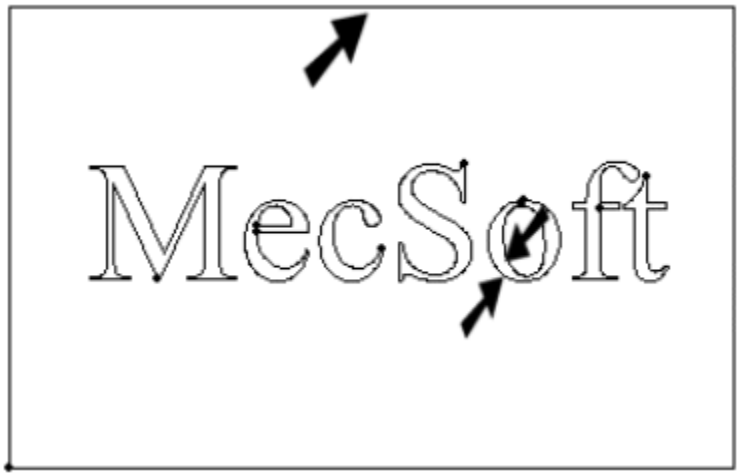
performed using a combination of Facing and Pocketing operations, and Profiling is used for finishing.

Facing

Available in Alibre CAM Standard, Professional and Expert versions only

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.

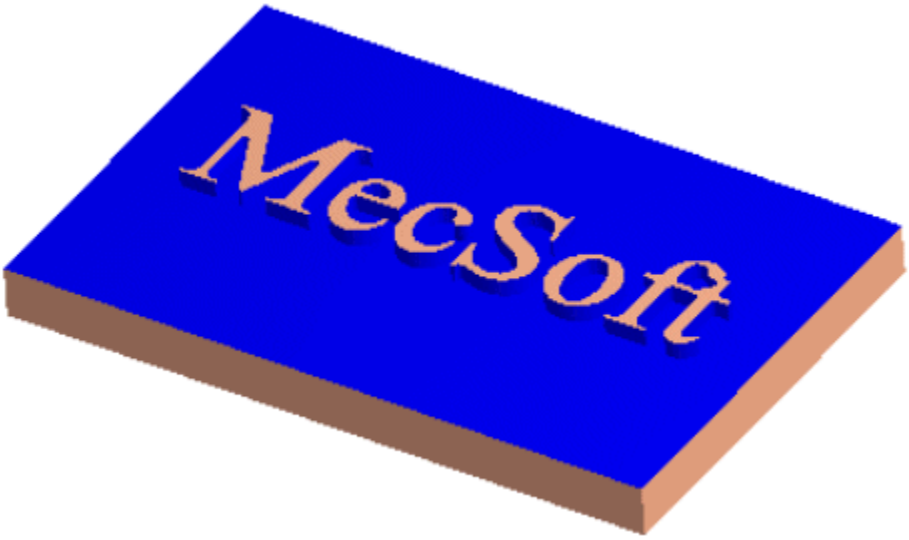
This example uses multiple regions – the rectangle is the outer region, and each letter is an inner region. Some letters have nested regions; these are treated as islands (areas to avoid).



Note: The outer region should encompass the stock. You can easily create this type of region by selecting Bounding Region from the Curves category of the Geometry bar.

The toolpath looks like the following:

The stock simulation:



Pocketing

Available in Alibre CAM Standard, Professional and Expert versions only

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.

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

The toolpath looks like the following:

The stock simulation:

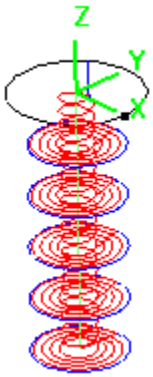
Profiling

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.

Hole Pocketing

Available in Alibre CAM Standard, Professional and Expert versions only

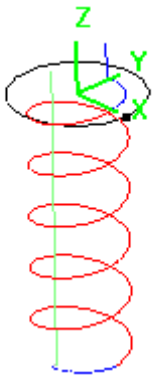
This method is used to cut large holes as a milling operation, rather than drilling. Engage can be specified as a helix with height and angle or pitch. Output can be a helix cycle (for machines with this capability) or a series of linear moves. After the helix engage, the hole is cut to the outer diameter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



Thread Milling

Available in Alibre CAM Standard, Professional and Expert versions only

This method is used to cut threads using a thread 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.



Engraving

Available in Alibre CAM Standard, Professional and Expert versions only

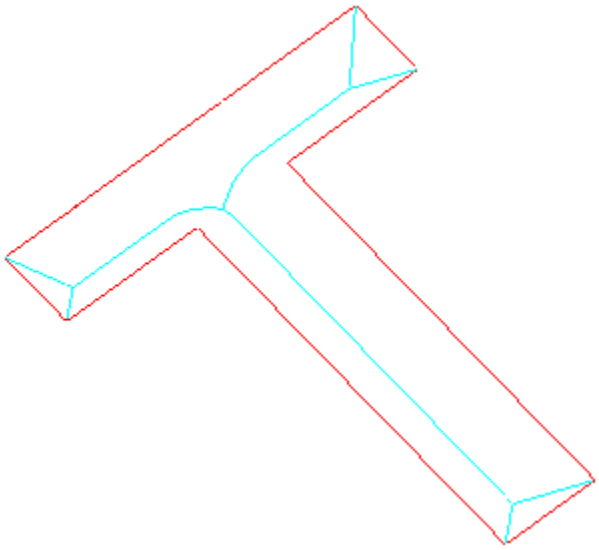
Engraving allows the user to select open or closed regions to engrave. In addition to 2D regions, 3D regions can also be chosen. Multiple depths can also be specified for better control of the engraving operation. This method is especially suited for engraving text and logos to part geometry. Unlike the 3 Axis Curve Machining method, the cutter is not projected to the surfaces below. It merely follows the specified regions. There are two ways to do the engraving operation. The on-condition engraving, in which the tool center follows the 3d curve exactly and the To condition in which the V-carving tool approaches from either left or right and to carve either inside

or outside of the letters, logos etc. For the To-condition to work the regions must be closed regions. The regions are flattened to 2D regions and the toolpath is created. The corners of the regions between two passes are automatically figured out and are cleaned up using 3D clean up pass.

V-Carving

Available in Alibre CAM Standard, Professional and Expert versions only

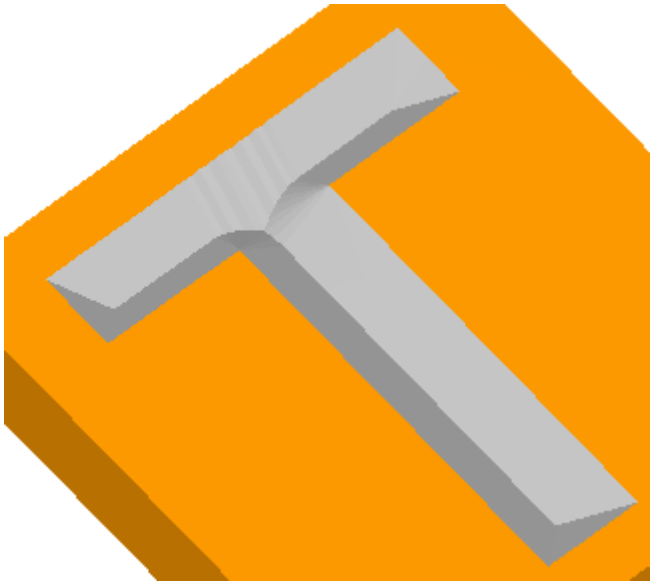
This is used to create cutter paths with a V-cutter (tapered cutter) that is especially suitable for lettering and sign making.



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.

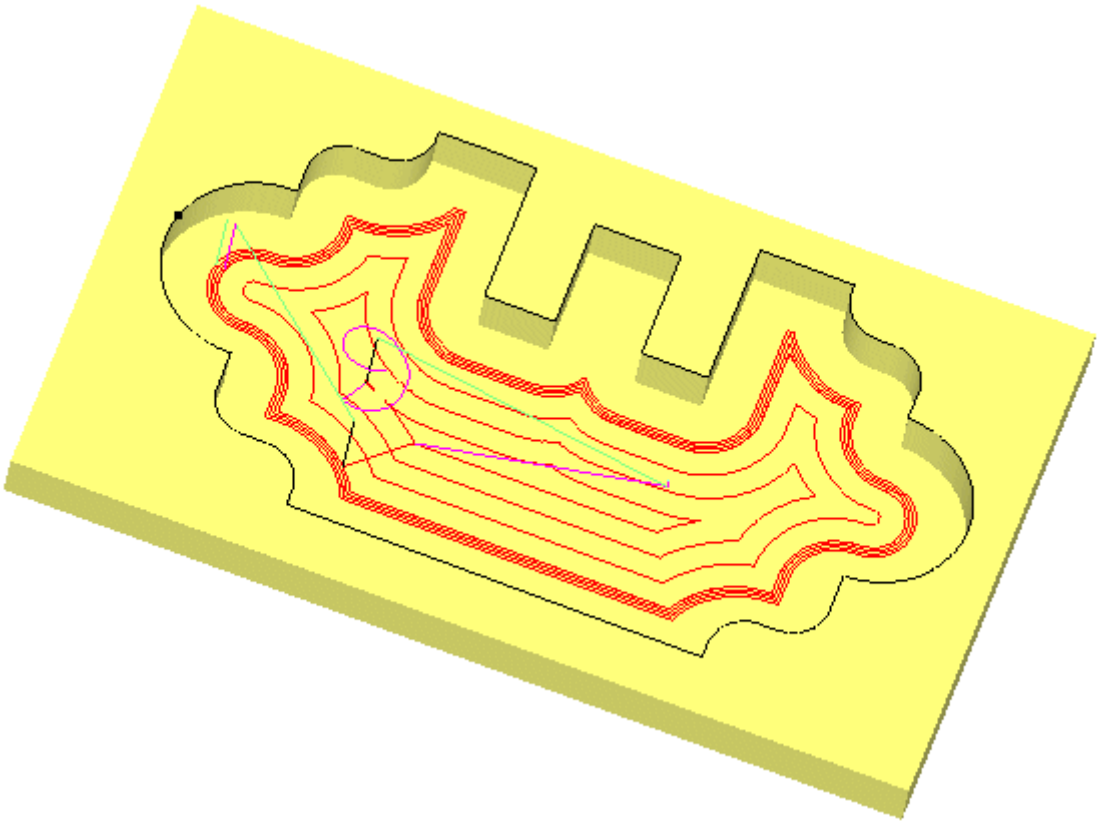
The stock simulation of the toolpath is shown here:



Advanced Pocketing

Available in Alibre CAM Expert Only

This machining operation encompasses the functionality of both pocketing and profiling operations. This allows the user to do roughing (pocketing) and finishing (profiling) in one operation and allows a greater degree of control. The stock left in the roughing can be cleaned up by the finishing operation, without the user having to know how much to cut in the finishing pass, since the stock to cut for finishing is figured out by the difference in the stock variable in roughing and the stock variable in finishing.



Advanced Profiling

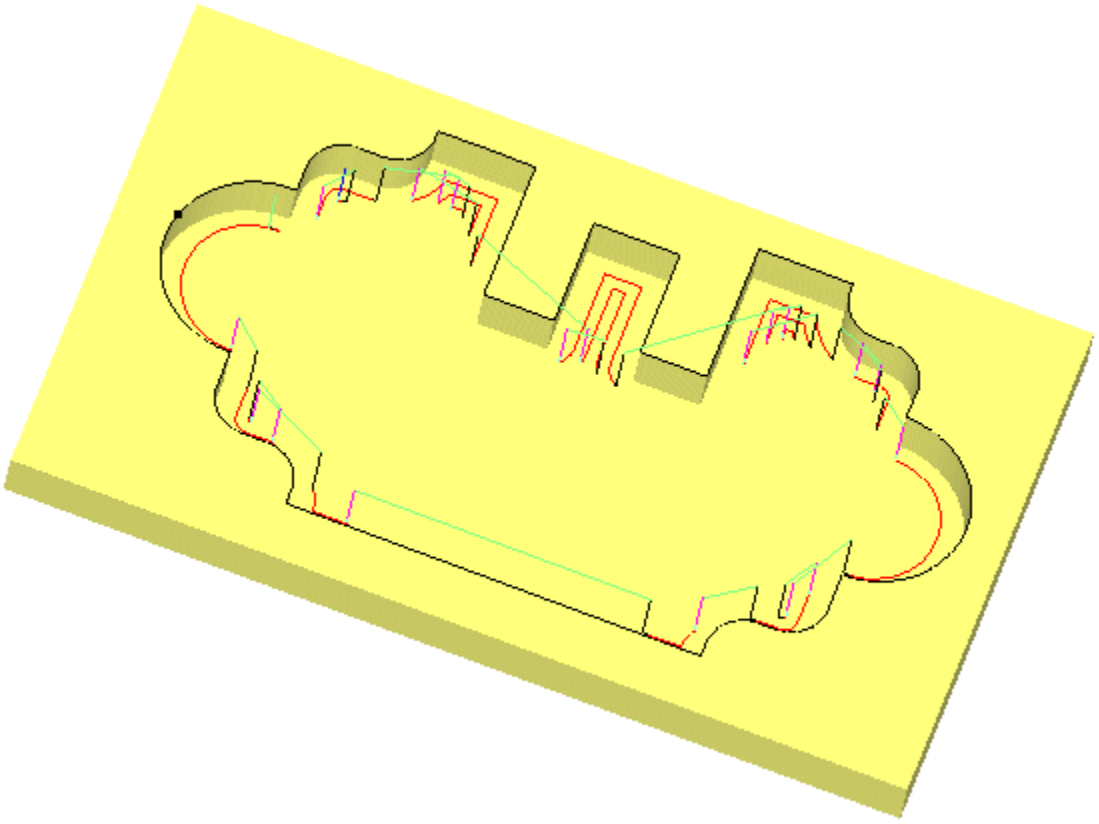
Available in Alibre CAM Expert Only

This machining operation can be used when multiple profiling passes are needed at varying widths with different step over distances. The width can be divided into roughing (larger step over) and finishing (smaller step over) passes. This also allows machining of closed as well as open regions by considering these regions as if they were vertical walls. This kind of operation is typically used for finishing vertical walls in a part. Multiple depths can be defined by specifying the bottom Z values and also the spacing between the cut levels.

Re-Machining

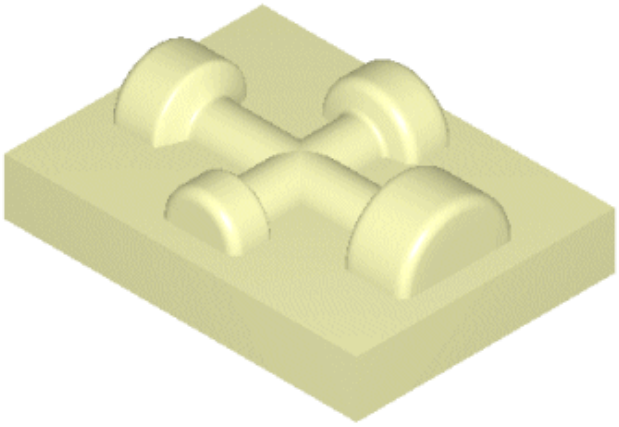
Available in Alibre CAM Expert Only

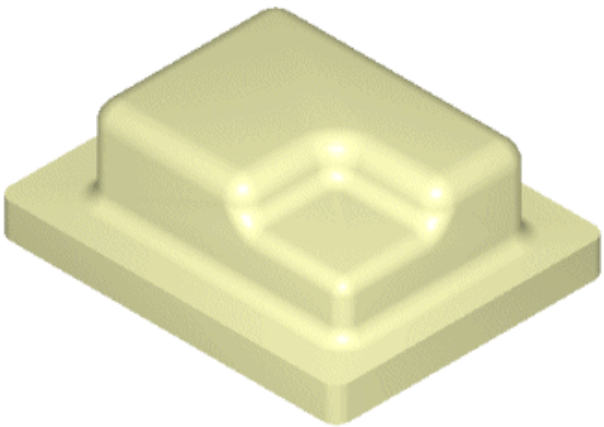
This operation can be used to cut the uncut material from a previous operation using a larger tool. The previous operation and the previous tool diameter need to be specified for this operation. The previous operation can be either facing, pocketing or profiling. Given these parameters, all the uncut areas are automatically calculated. These uncut areas are processed similar to a facing operation.



Three Axis (3) Operations

In this type of machining, the tool can move simultaneously in all three axes. This is appropriate for parts that have complex, curved, and non-vertical surfaces.

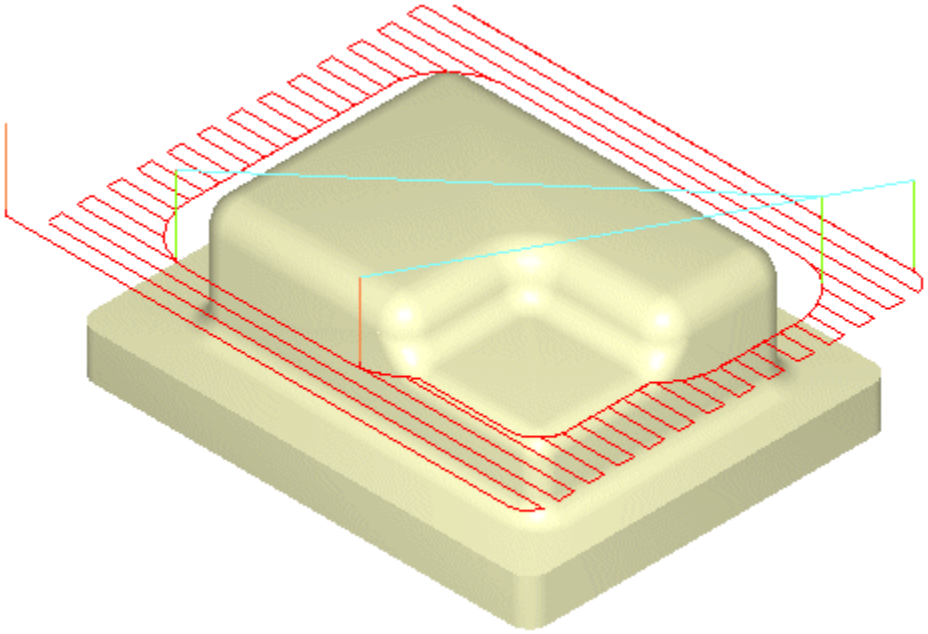




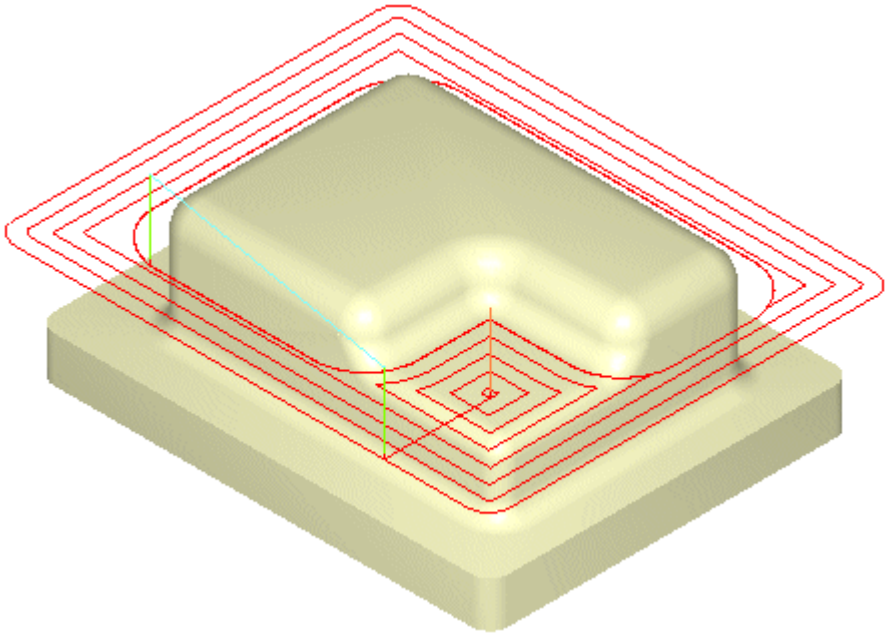
Horizontal Roughing

This is Alibre CAM's principal method of roughing, also knows 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.

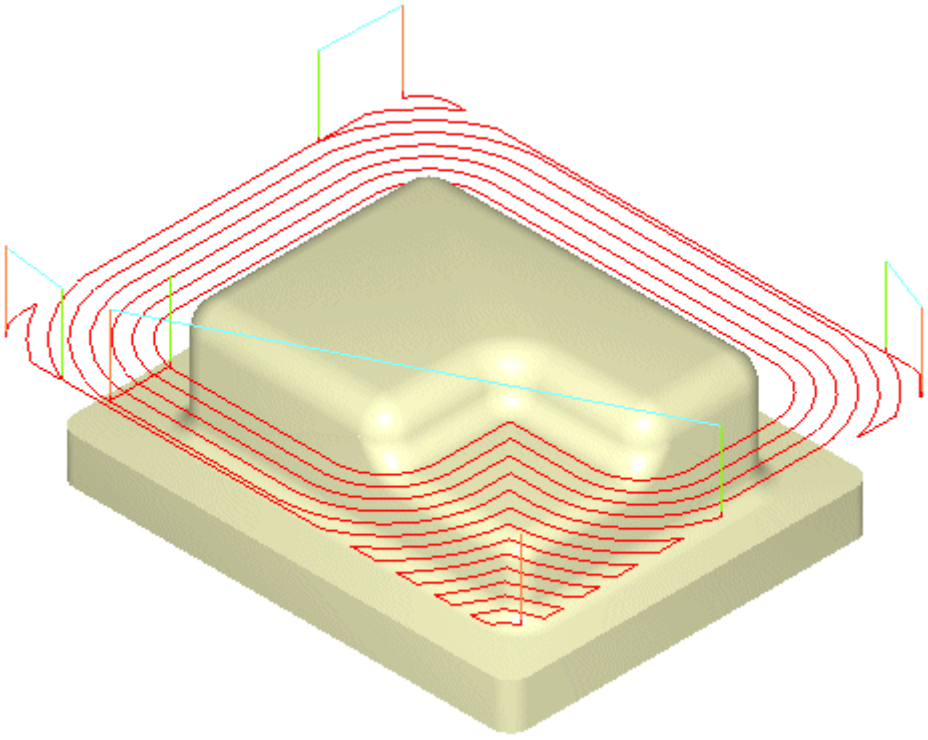
Both part and stock geometry are used to determine the regions that can be safely machined. Three types of cutting patterns are available: Linear (parallel, zigzag lines), Stock Offset (spiral pattern within stock and part), and Part Offset (spiral pattern outside the stock and outside the part). Tool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



Horizontal Roughing – Linear



Horizontal Roughing – Stock Offset (Pocketing)

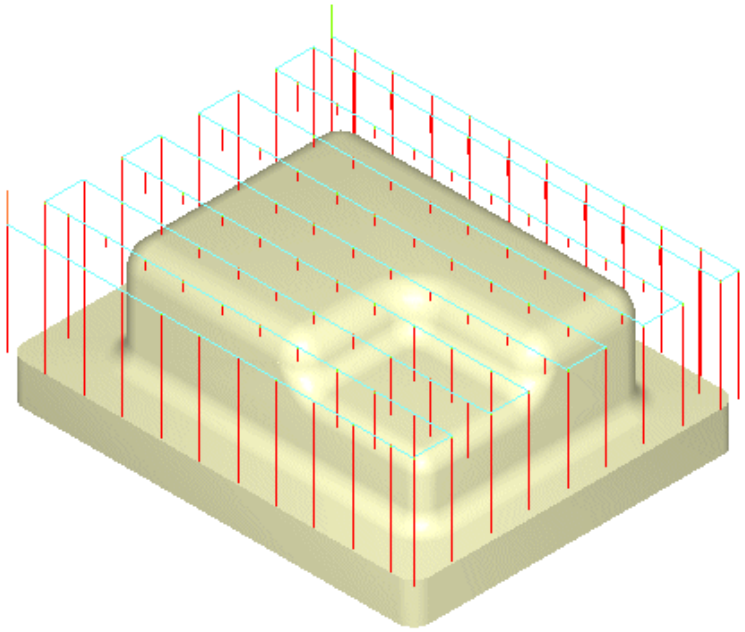


Horizontal Roughing – Part Offset (Facing)

Plunge Roughing

Available in Alibre CAM Expert only.

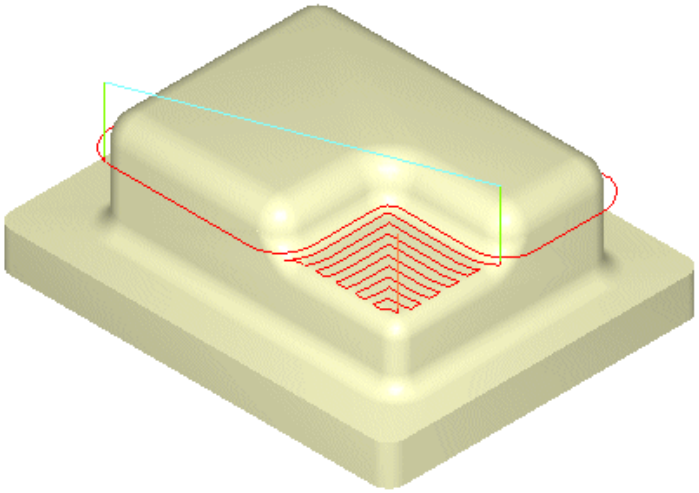
Sometimes called drill roughing, the tool can cut in the Z direction only, not in X and Y. The tool makes a series of overlapping plunges to remove cylindrical plugs of material.



Horizontal Re-Roughing

Available in Alibre CAM Expert only.

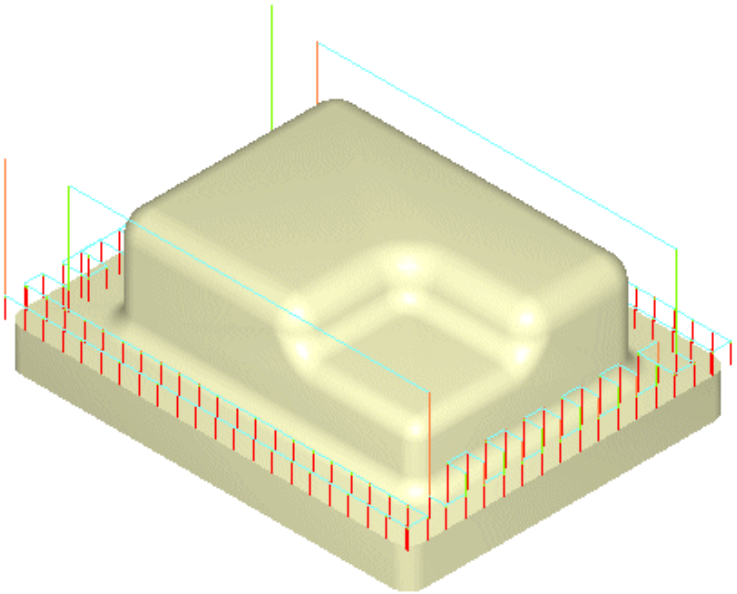
This is used to create toolpaths in areas that were not machined by previous operations. Unmachined areas are determined by comparing the part to the stock remaining after the previous operation. Machining is performed in constant Z levels, one of which is shown below:



Plunge Re-Roughing

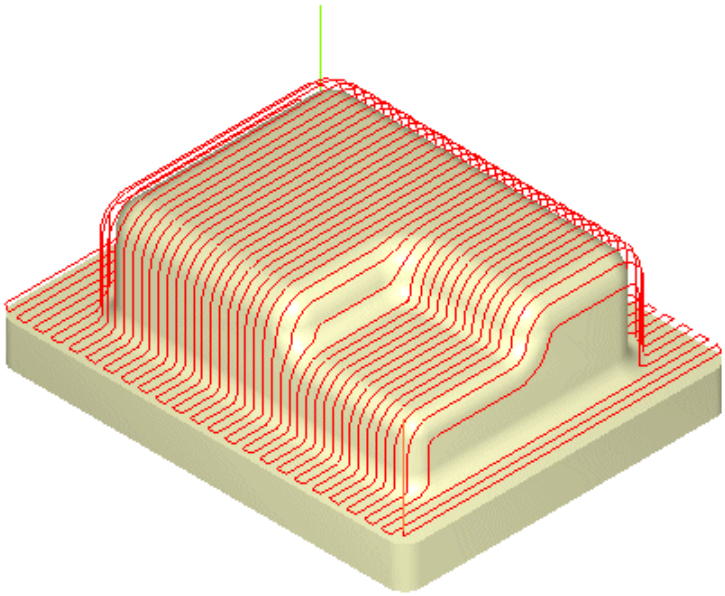
Available in Alibre CAM Expert only.

Similar to Horizontal Re-Roughing, this method uses plunge motions to machine areas that were not machined by previous operations.



Parallel Finishing

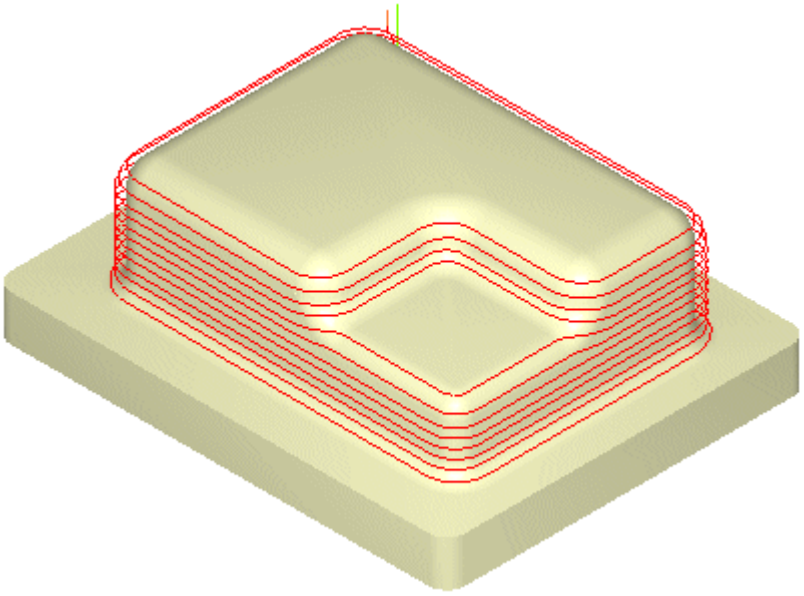
This is an efficient method of finishing or pre-finishing, typically used when part surfaces are relatively flat. A 2D linear zigzag pattern is generated on the XY plane above the part geometry. The tool moves along this cut pattern, following the contours of the part geometry below.



Horizontal Finishing

Available in Alibre CAM Standard, Professional and Expert versions only

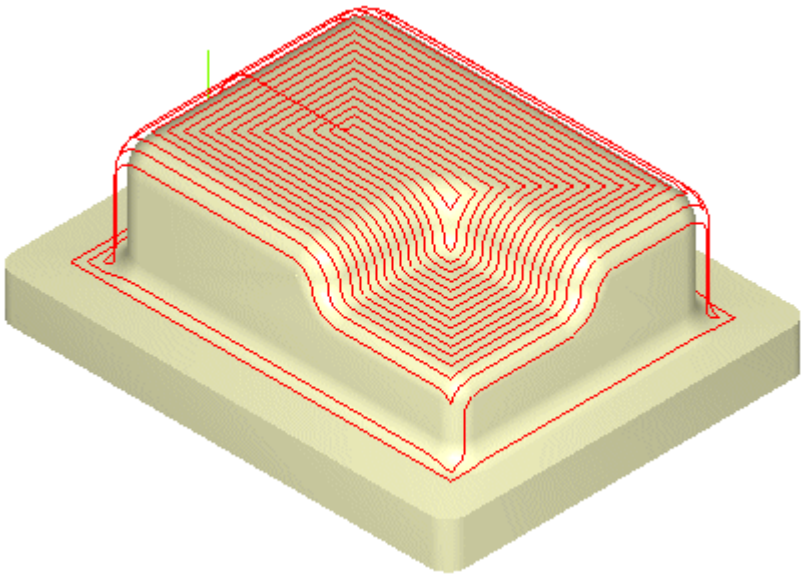
This method is used for pre-finishing or finishing in constant Z levels, typically used when the part has large vertical surfaces and when Parallel Finishing will not yield satisfactory results.



Three (3) Axis Pocketing

Available in Alibre CAM Expert only.

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 while following the contours of the part below.

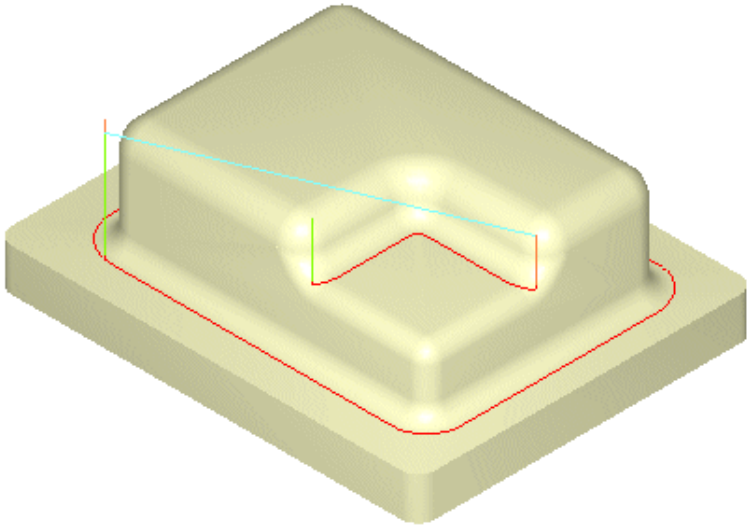


Pencil Tracing

Available in Alibre CAM Expert only.

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.

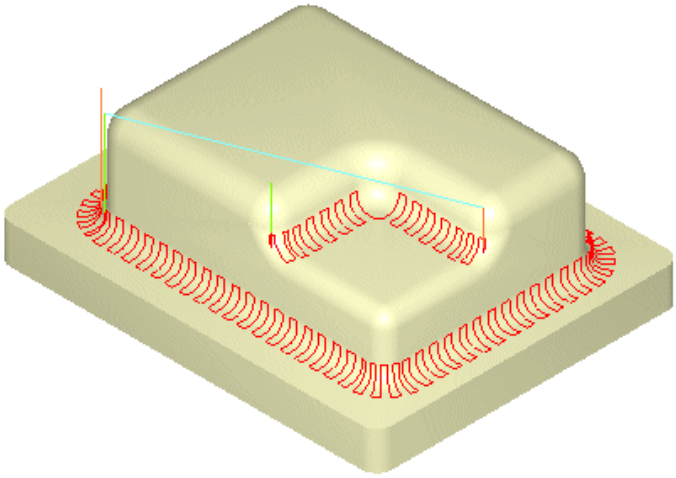
When used as a roughing operation, valleys and corners are relieves so that subsequent operations will not encounter large amounts of material in these regions, thereby reducing tool deflection and wear. When used as a cleanup operation, scallops that remain after finishing operations are removed.



Valley Remachining

Available in Alibre CAM Expert only.

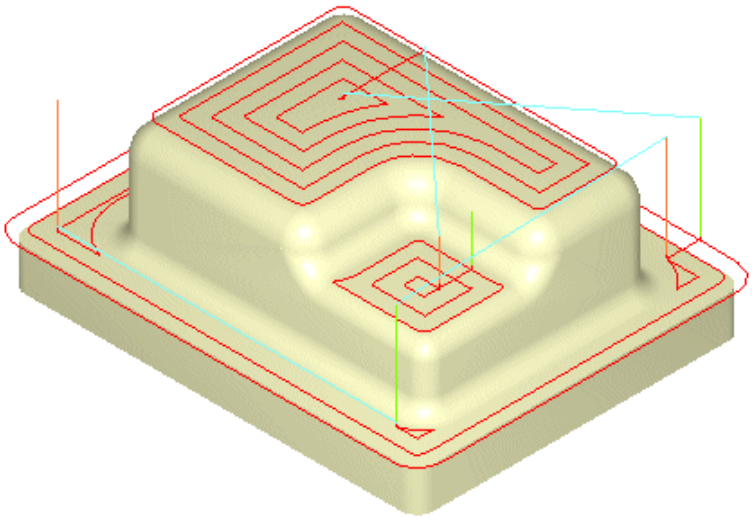
This is used to machine corners and valleys that were inaccessible in previous finishing operations.



Plateau Machining

Available in Alibre CAM Expert only.

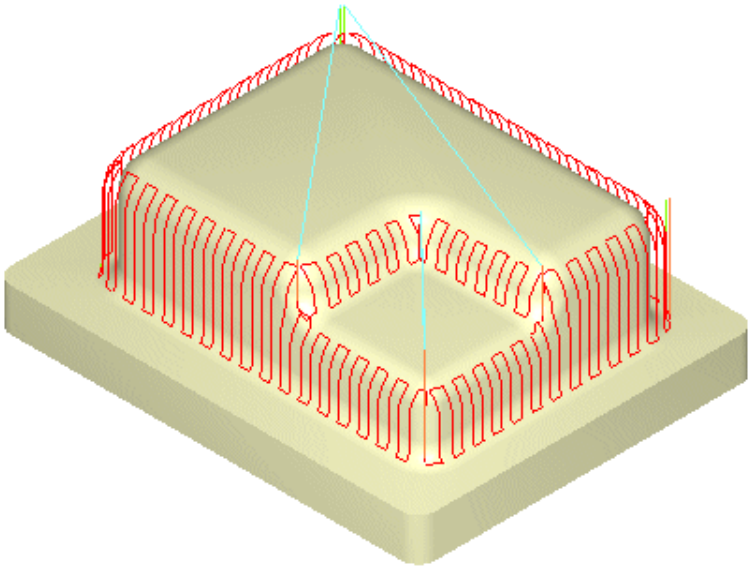
This method machines the tops of flat areas – areas that are within a specified angle from horizontal. This is typically used to re-machine areas that remain unmachined after a Horizontal Roughing or Horizontal Finishing toolpath.



Parallel Hill Machining

Available in Alibre CAM Expert only.

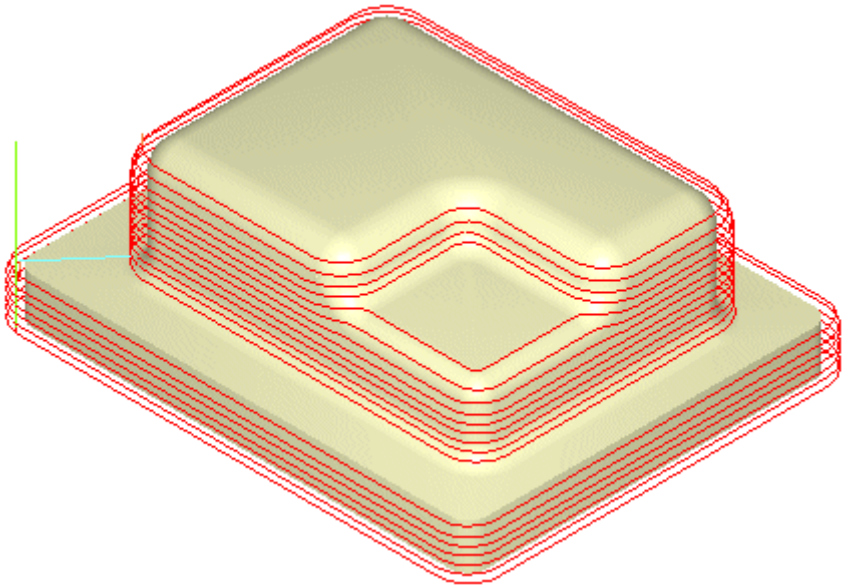
This method machines steep areas. These are areas that are within a specified angle from vertical. This method is typically used when a Parallel Finish toolpath leaves scallops on steep areas. The cut angle is adjusted so that machining is always normal to the steep areas, thereby leaving minimal scallops.



Horizontal Hill Machining

Available in Alibre CAM Expert only.

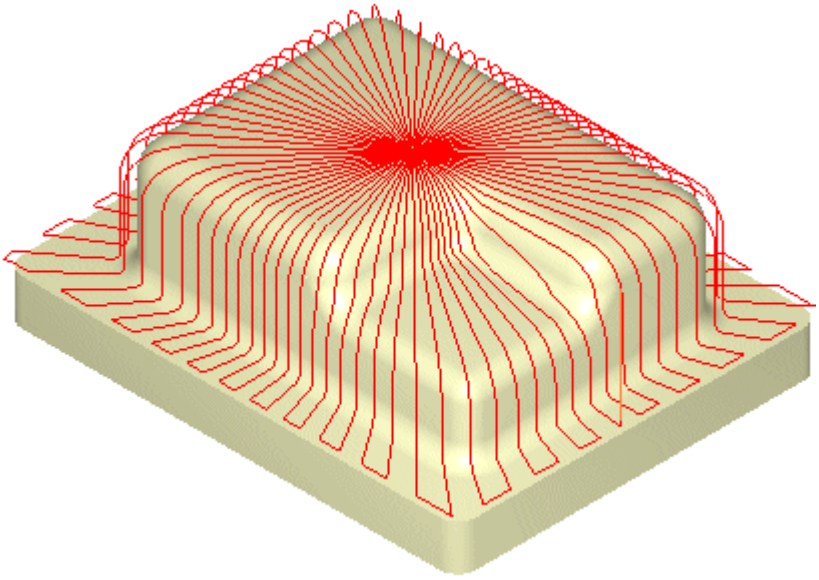
Similar to Horizontal Finishing, this method machines in constant Z levels. However, machining can be restricted only to areas in the part that are steeper than a user-defined steepness angle.



Radial Machining

Available in Alibre CAM Expert only.

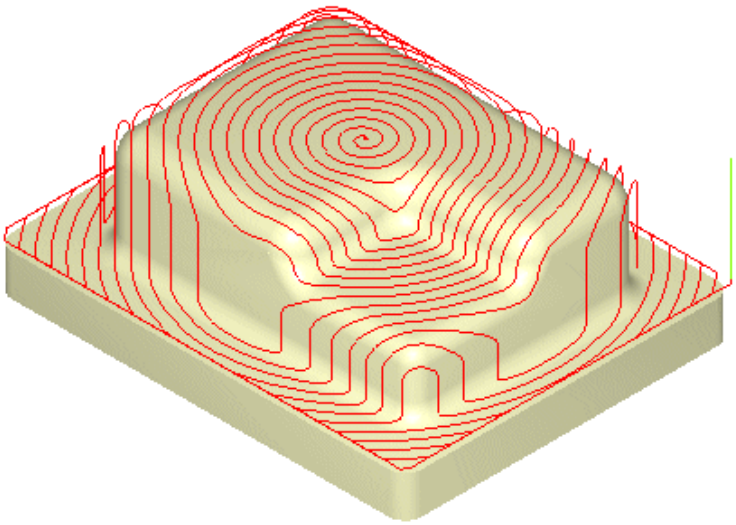
This method is used as a finishing operation for areas that have annular pockets. You must specify one or more machining regions; the tool moves radial from the centroid of these regions.



Spiral Machining

Available in Alibre CAM Expert only.

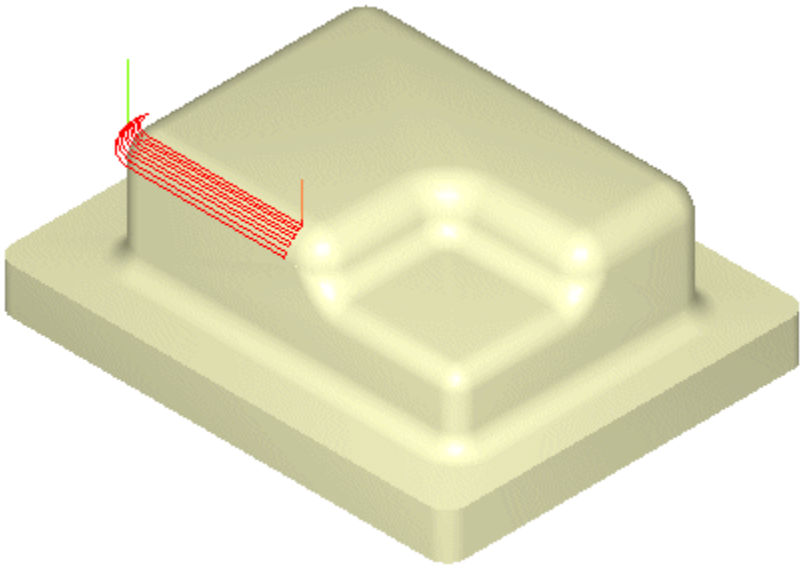
This method is used for finishing areas that have circular or near-circular characteristics, such as pocket bottoms. You must specify one or more machining regions; the tool moves in a spiral pattern based on the centroid of these regions.



Curve Machining

Available in Alibre CAM Expert only.

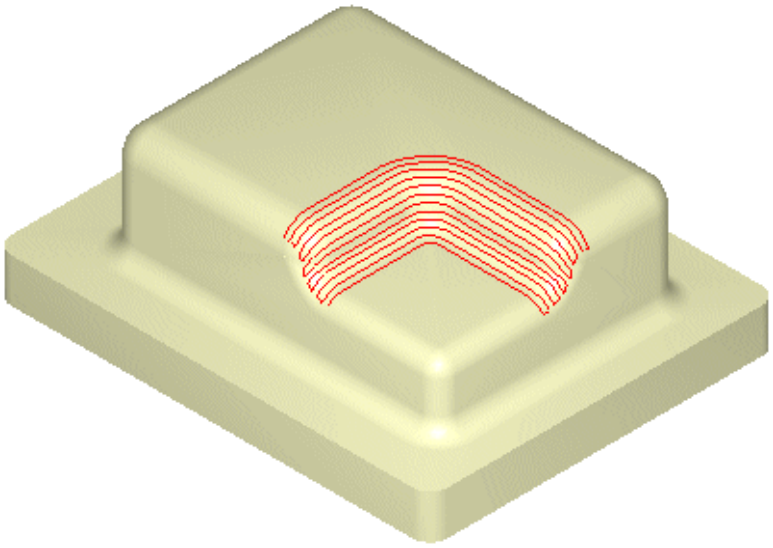
Suitable for machining isolated areas or shapes, this method machines along a curve. You must specify one or more machining regions, direction and cut pattern. The tool simultaneously follows the region and the contours of the part below.



Between 2 Curves Machining

Available in Alibre CAM Expert only.

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.



Reverse Post Milling

Available in Alibre CAM Expert only.

This feature loads toolpaths from APT CL files and ISO standard G Code files. You can use these toolpath motions to project the tool onto part surfaces. You can also load an existing toolpath in order to simulate it in Alibre CAM.

Four (4) Axis Operations

Note:

The 4th axis functionality is available with Alibre CAM Professional and Alibre CAM Expert.

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. In such cases, the object could be divided into top and bottom sections and machined separately. Alibre CAM'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.

This type of programming allows the machining of part geometry that have 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.

Indexed 4th Axis Machining Operations

Shown below is a typical part that would require index 4 axis milling operations to manufacture the part. Notice that the part has undercut geometry when looking down from the top. This means that there are areas of the part, which cannot be reached by a straight tool moving purely up and down. To successfully machine this part, it will have to be divided into two halves. The top half needs to be machined first. Then, this part will have to be flipped over by 180 degrees and the bottom part machined.

It should be noted that Alibre CAM allows for any user defined indexing angle and is not restricted to 180 degrees. This feature allows the user to machine multi-sided parts and not just two sided parts as the one shown above.

Specifying rotation axis and rotation center

Before creating 4th axis toolpaths, the user needs to specify to the system the rotation axis i.e. whether the rotation is about the X (A) or Y (B) axis. The user also can specify the center of rotation. Obviously the center of rotation is dictated by how the part is fixtured in the table. It is a common programming practice to specify the machine zero to lie on the rotation axis and use the machine zero as the rotation center. This can be done by selecting the Machine Tool button in the Setup window of the Browser.

Refer to Machine Setup for detailed description.

Rotate Table Operation

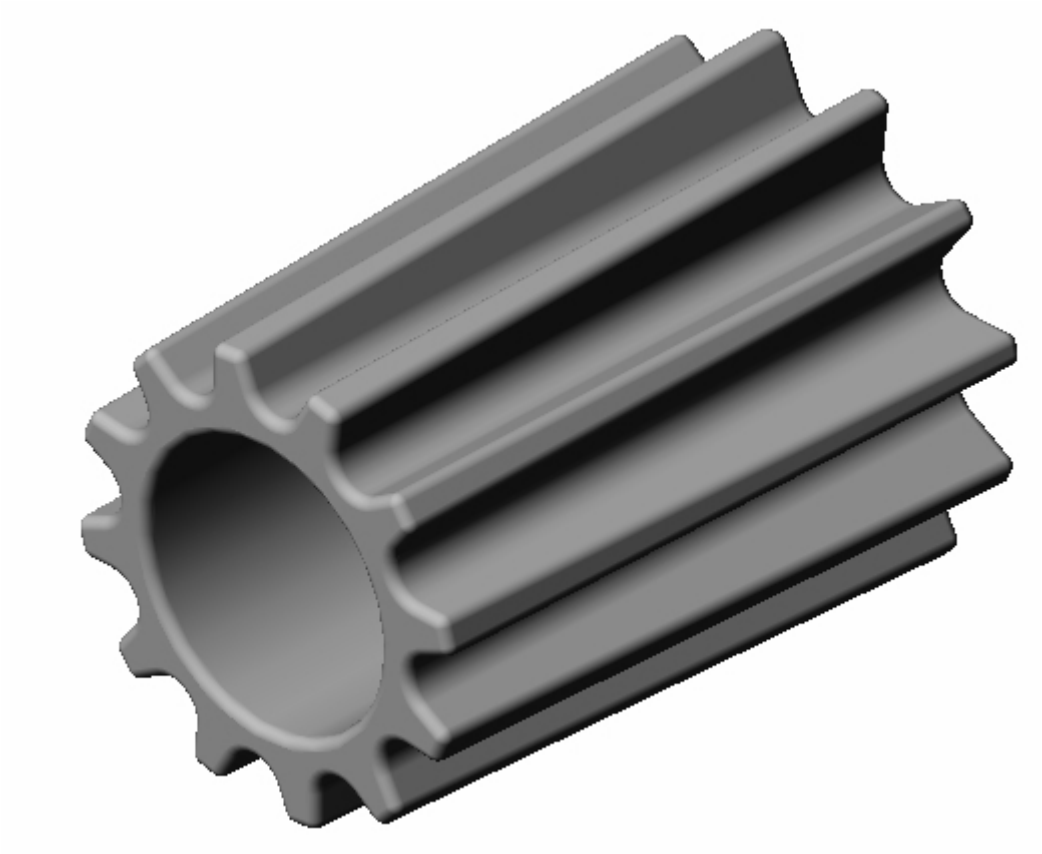
In order to perform an index 4 Axis Operation, the user first needs to use the Rotate Table Operation. Here the user can index the table to any specified angle and then lock the table in position. This can then be followed by any of the allowable 3 axis machining operations, thereby allowing access to the part from different orientations.

Alibre CAM allows the user to create 4th axis index toolpath operations. 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. Alibre CAM allows the user to program these part rotation motions as well to create standard 3 and 2 ½ axis machining operations in these new part orientations. Indexing is usually performed by fixturing the part in a rotary table.

Continuous 4th Axis Machining Operations

Available in Alibre CAM Professional and Alibre CAM Expert.

In addition to indexed machining of parts, 4 axis continuous machining can be used to machine parts such as a rings.



The part shown above is a design of a ring. It can be machined using Alibre CAM's continuous 4th axis machining mode. In the continuous mode, the tool axis is always pointed normal to the rotation axis of the machine tool. In this particular case, the rotary axis of the 4th axis table will be about the X axis. As the part is held such that it spins about the X axis the tool will move about the X and the Z axis simultaneously to perform the machining. This type of machining is called continuous 4th axis machining and can be employed to efficiently machine such parts.

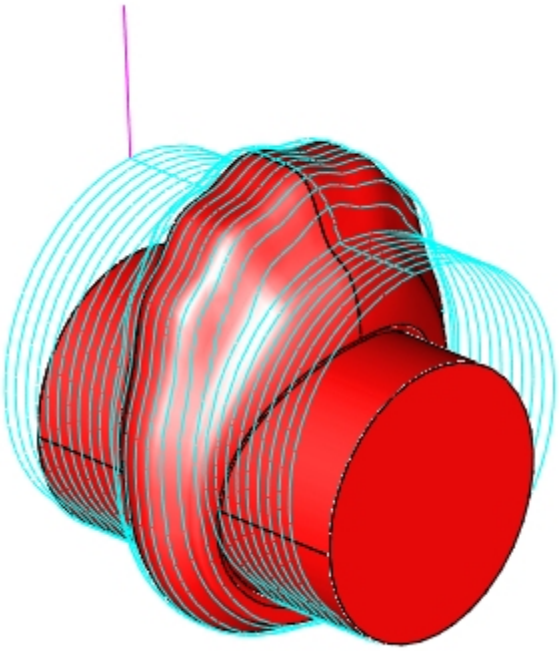
Alibre CAM allows the user to choose from a variety of rough, semi-finish and finish machining operations to satisfy various machining conditions and requirements. A list of the available types with a short description for each type is given below. For a more complete description follow the links.

Roughing

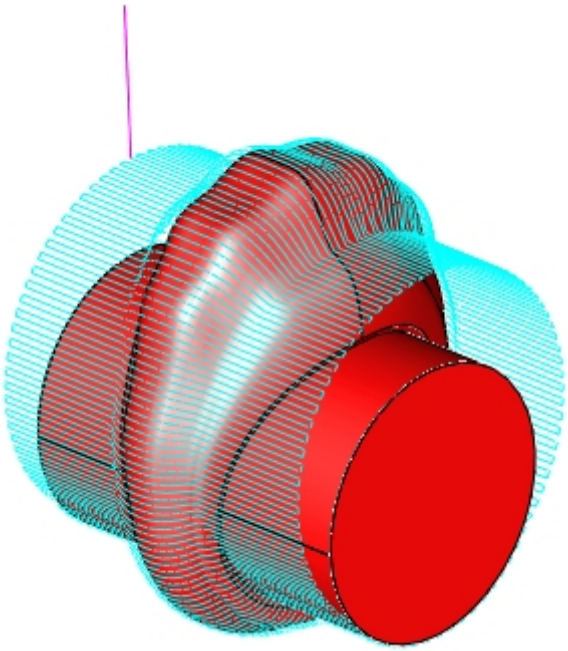
Available in Alibre CAM Professional and Alibre CAM Expert.

Here the user can perform roughing 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.

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.



4 Axis Roughing - Across Axis

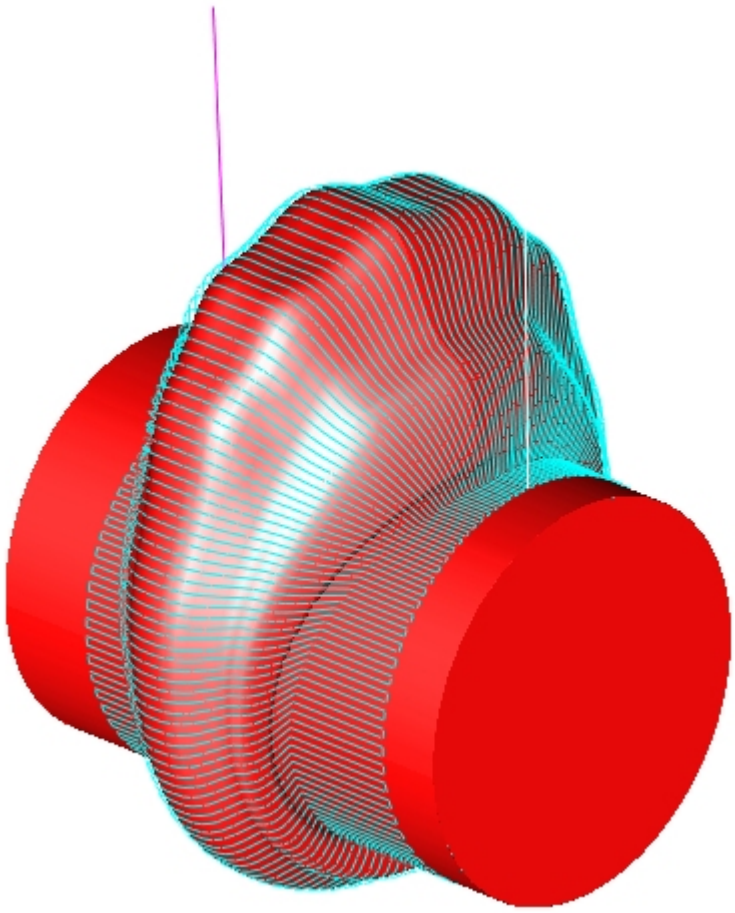


4 Axis Roughing - Along Axis

Finishing

Available in Alibre CAM Professional and Alibre CAM Expert.

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.

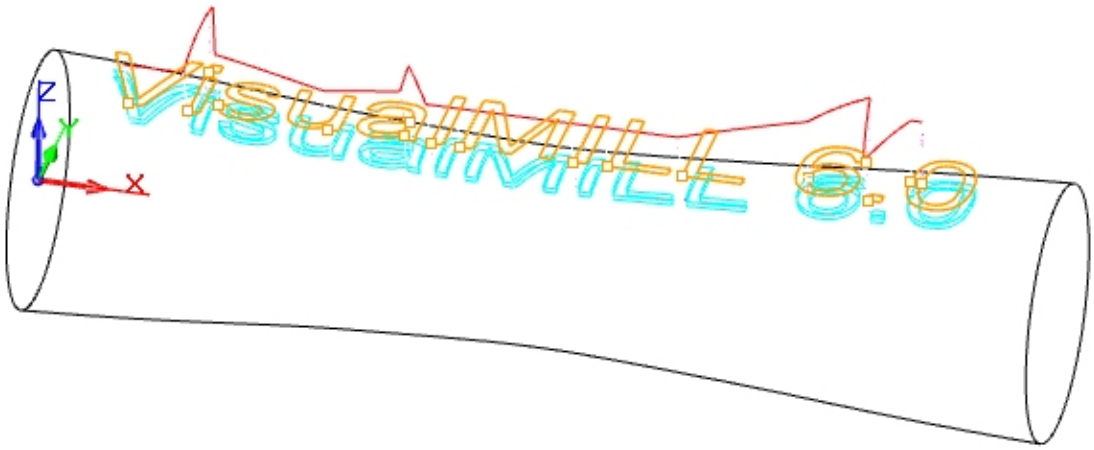


Engraving

Available in Alibre CAM Professional and Alibre CAM Expert.

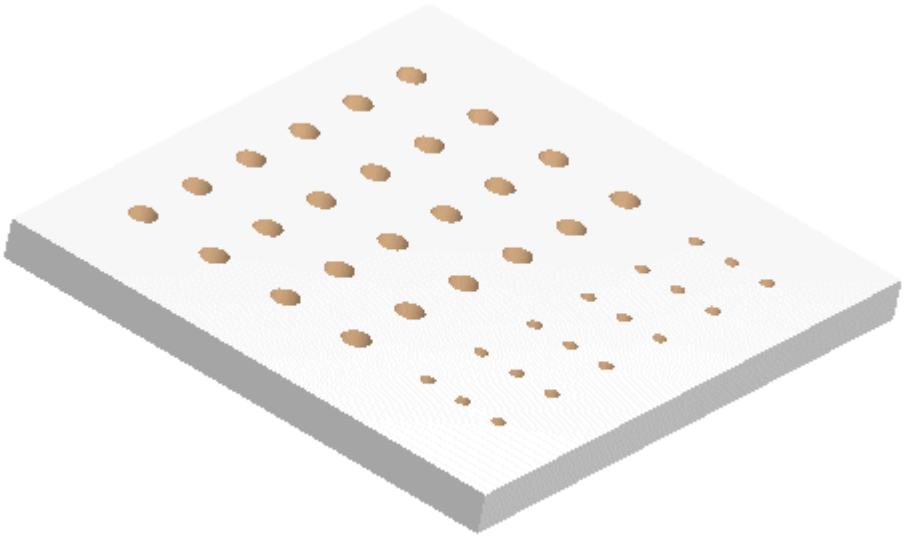
Curves can be selected and the tool follows the curve as the table rotates. The user can specify multiple depths of cut for the engraving option. Optionally the user can choose to project the curves to the surfaces below before machining. This method is useful in scribing text etc onto the part geometry in 4 axis mode.

Toolpath projected on to the model:



Drilling (Hole Making) Operations

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.



The following drilling operations are available:

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.

Tapping

Available in Alibre CAM Standard, Professional and Expert versions only

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

Boring

Available in Alibre CAM Standard, Professional and Expert versions only

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.

Reverse Boring

Available in Alibre CAM Standard, Professional and Expert versions only

This is simply 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.

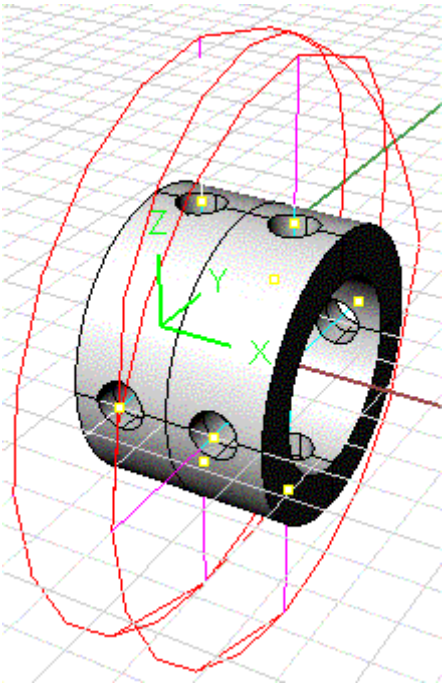
4th Axis Hole Making

Available in Alibre CAM Professional and Alibre CAM Expert

Description

All of the hole creating machining operations available in 3 axis are also available in 4 axis mode. 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.



4Axis 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.

4Axis Tapping

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

4Axis 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 simply 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.

Five (5) Axis Operations

5th axis functionality is available in Alibre CAM Expert product.

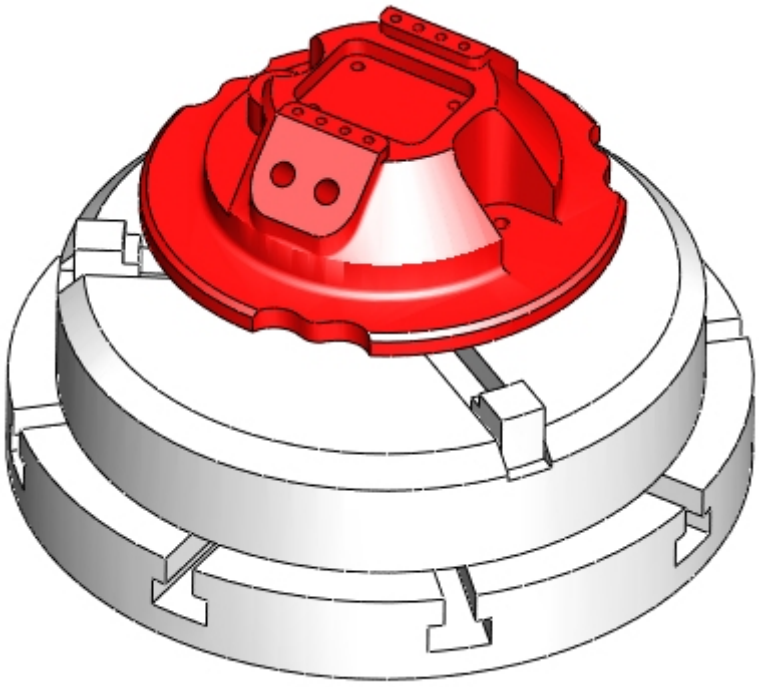
Fifth axis machining enables you to change tool direction relative to any set of axes. Fifth Axis operations are used to machine parts that cannot be machined with simple 2 ½ axis or 3 axis machining operations. Since the tool moves only up and down along the Z axis during 3-Axis milling, areas that cannot be seen from above cannot be cut. In such cases, the object could be divided into multiple sections and machined separately. Alibre CAM'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.

This type of programming allows the machining of part geometry that has more than one side that needs to be machined. Using an indexing head as the one shown will allow easy reorienting of machine tool. Note that in order to utilize 5th axis indexing you must have a CNC control that supports a fifth axis that can be controlled by the CNC machine.

Indexed 5th Axis Machining Operations

Shown below is a typical part that would require index 5 axis milling operations to manufacture the part. Notice that the part has undercut geometry when looking down from the top. This means

that there are areas of the part, which cannot be reached by a straight tool moving purely up and down. 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.



It should be noted that Alibre CAM allows for any user defined indexing angle and is not restricted to 180 degrees. This feature allows the user to machine multi-sided parts.

Specifying rotation axis and rotation center

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.

Refer to Machine Setup for detailed description.

Machining Operations

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. Re-machining operations may also be performed to machine areas that were not completely machined due to tool size and part configuration. These typically will be used to reduce the amount of handwork needed in producing the final finished part.

Alibre CAM allows the user to create machining operations to address the machining requirements in each stage of the manufacturing process. This section describes the functionality present in Alibre CAM to allow the user to create and manage these machining operations. Please follow these topics to get more information.

- Creating Machine Operations
- Editing Machine Operations
- Simulating Machine Operations
- Post Processing Machining Operations
- Archiving Machining Operations

Creating Machining Operations

Creating machining operations in Alibre CAM is a very simple process. The user loads the part, the stock geometry if necessary, selects a tool, the feeds & speeds to be used in the machining operation. The user can optionally select machining regions to restrict areas to be machined. The user then picks the type of machining operation required and sets the parameters for this operation and generates the toolpath. Generation of the toolpath begins once the user clicks on the "OK" button in the machining operation parameter dialog. Once the toolpath generation is complete the machining operation will be created and displayed in Alibre CAM Browser. The following sections describe each of the necessary and optional items that need to be selected or set before creating a machining operation.

- Geometry
- Tools
- Feeds and Speeds
- Feeds and Speeds
- Clearance Plane

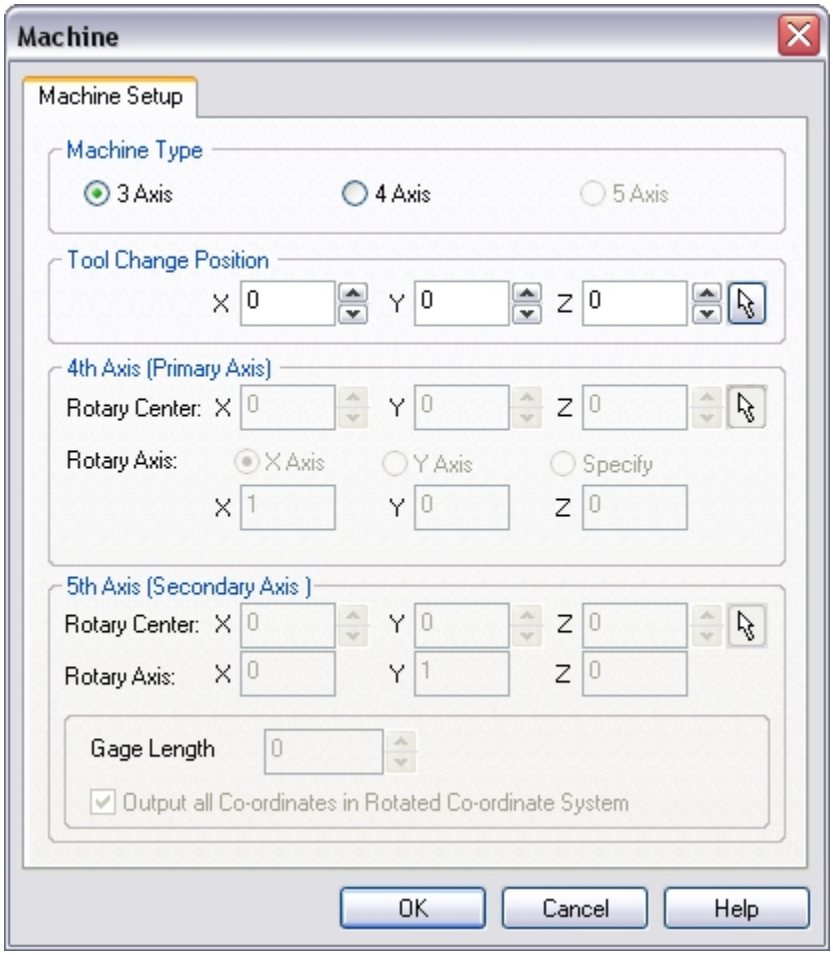
Alibre CAM allows the user to choose from a variety of rough, semi-finish and finish machining operations to satisfy various machining conditions and requirements. For a listing and description of these methods please jump to Machining Operations.

Machine Setup

Sets up the machine for the 3 axis, 4 Axis and 5 Axis operations. This is done by selecting "Machine" from the Setup tab under the Alibre CAM browser.

4 axis features are available with Alibre CAM Professional and Alibre CAM Expert.
5 axis features are available with Alibre CAM Expert Only.

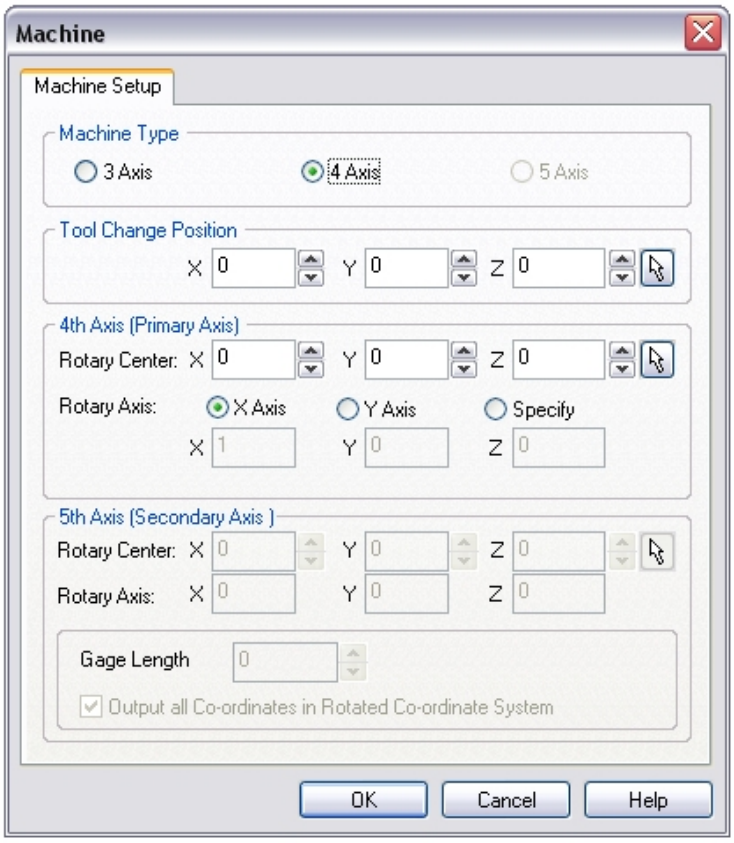
This will bring up the following dialog.

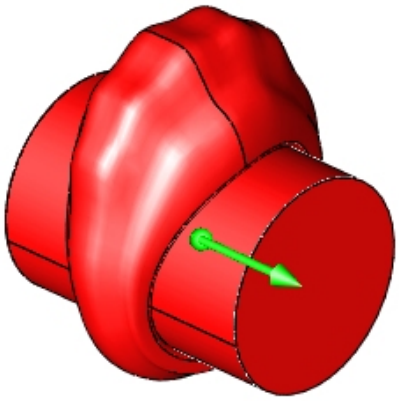


Machine Type: Allows the user to specify the machine type.

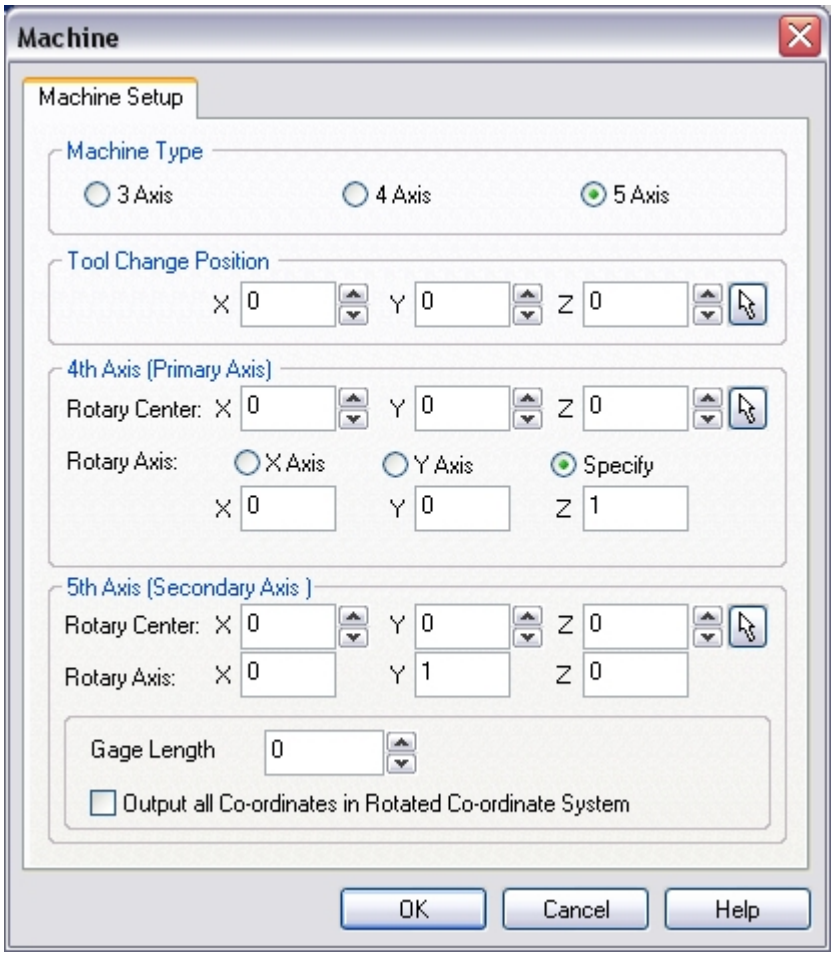
- Select 3 axis for 2 ½ and 3 axis Machining methods.
- Select 4 axis for index and continuous 4 axis machining methods.
    - Make sure to set the Rotary Axis to X if the part rotates about the X (A) axis or Y if the part is rotating about Y (B) axis.

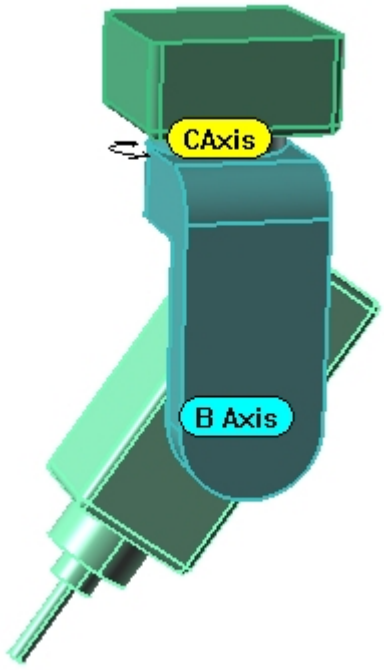
- Make sure to set the rotary center. The part geometry must pass through the rotary center. The rotary center is indicated with a Green arrow when the Machine Setup Dialog is selected and machine type is set to 4 Axis.





- 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).





5 axis Machine Setup for a C and B configuration

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

Geometry

The types of geometry that can be defined and used in Alibre CAM are described below:

Part Geometry

Part Geometry constitutes the end product of the manufacturing operation. This is also the design model. Design models in various data formats can be imported into Alibre Design. These design models can either be solid models or surface models.

Machining Regions

Machining regions are regions that are defined by sketches in Alibre CAM. Sketches can be used to contain the toolpath as done in 3 Axis machining operations or as drive geometry as done in 2-1/2 axis and drilling operations. Regions need to be closed curves if they are to be used as containment geometry. In other toolpath methods such as Profiling and Engraving, users can use open curves as drive geometry as well.

Refer to Machining Regions section for detailed description.

Stock Geometry

Represents the raw stock from which the design part needs to be manufactured. The methods that are currently available to the user are:

- Box Stock
- Cylinder Stock*****
- Part Box Stock
- Part Cylinder Stock*****
- Part Offset Stock******
- Regions Stock******
- Stock from Selection******
- Delete Stock – Deletes the created stock geometry
- Export Simulated Model to STL – Exports the simulated stock model as STL.

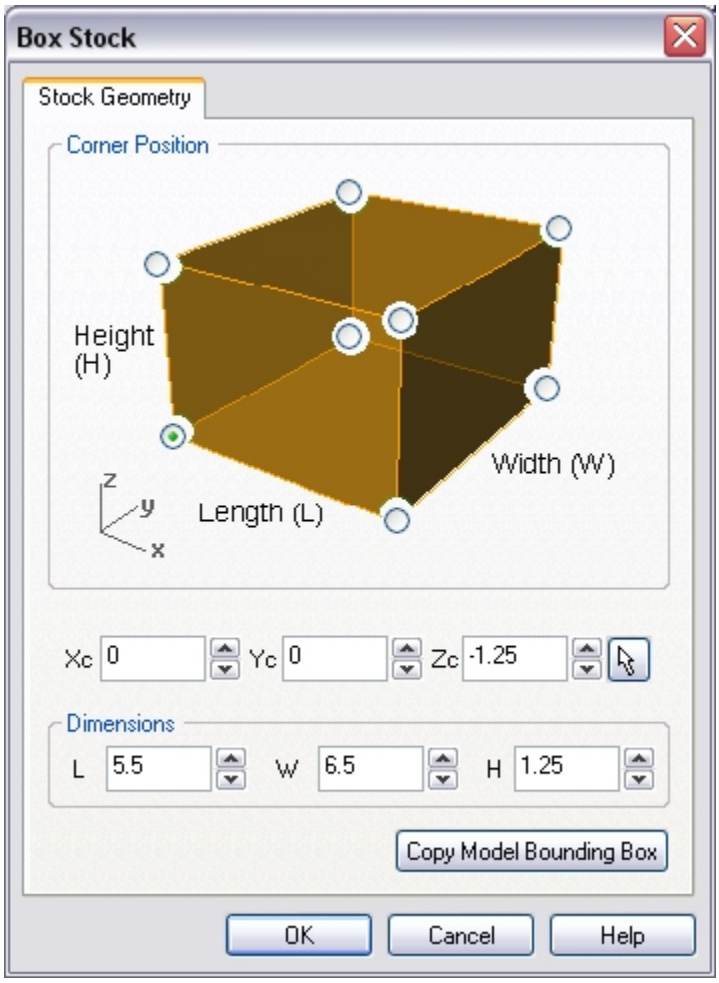
***** **-** Available in Alibre CAM Professional and Alibre CAM Expert

****** **-** Available in Alibre CAM Expert Only.

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 Alibre CAM Browser.

When the user selects this option, the following dialog will be invoked.

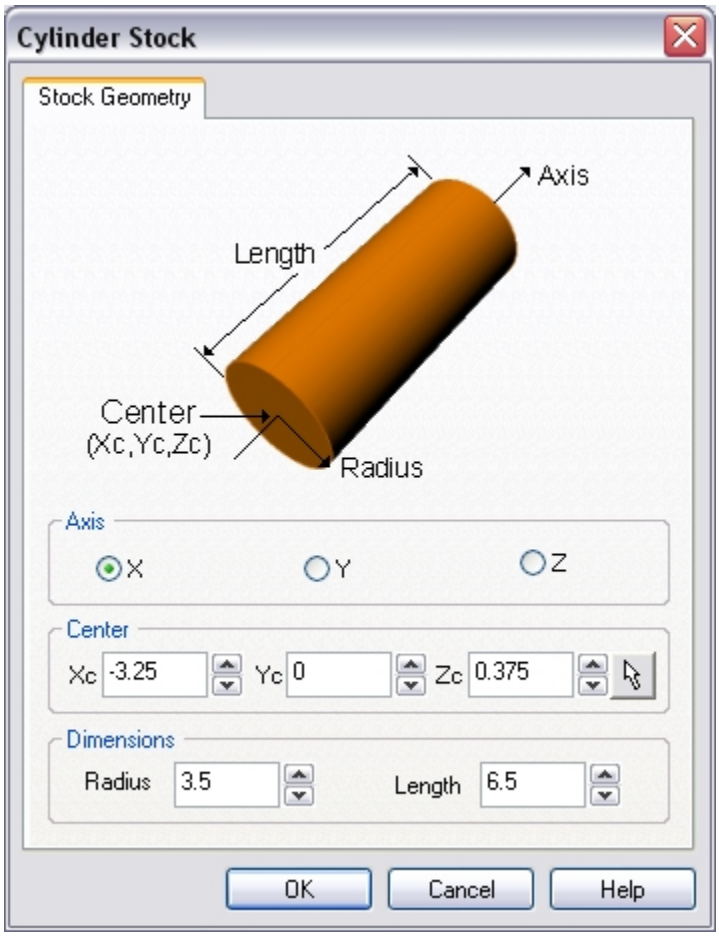


The user can define the box by simply typing in the length; width and height of the box in the corresponding edit boxes of the dialog. The corner position (origin) of the box can also be repositioned by picking the desired coordinate or by specifying the coordinate values in the corresponding edit boxes 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.

Cylinder Stock

Available in Alibre CAM Professional and Alibre CAM Expert

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

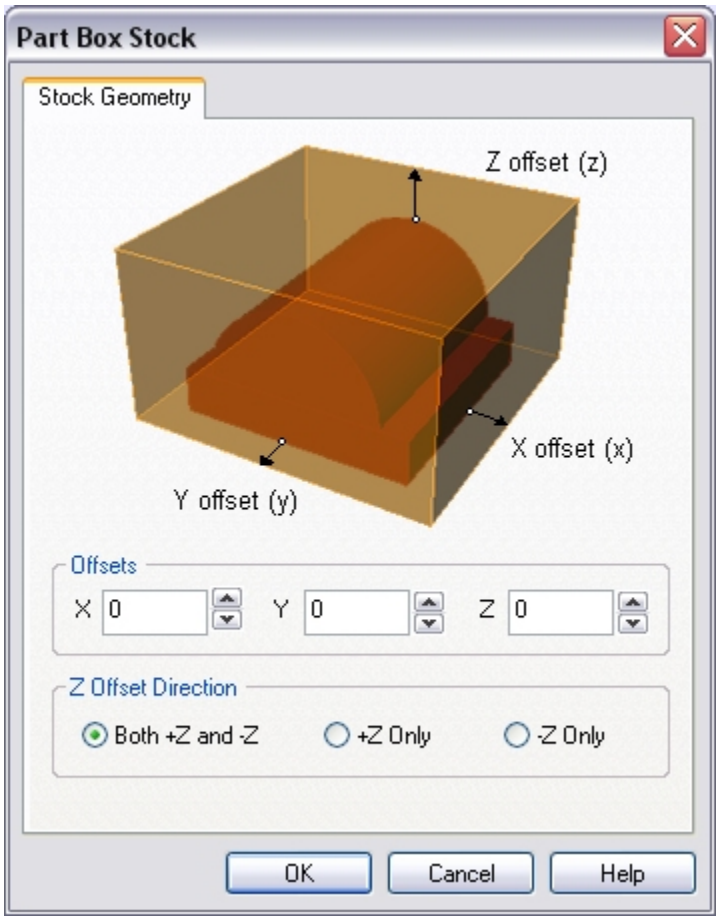


The user can define the cylinder by simply selecting the axis of the cylinder (either X Y or Z), specifying the center coordinates by typing in the values or graphically picking the center and then by specifying the radius and the length of the cylinder.

The Center (origin) of the stock can also be repositioned 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. The user can switch to the Stimulate tab of the browser window to display the stock model that was created.

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 Alibre CAM Browser.

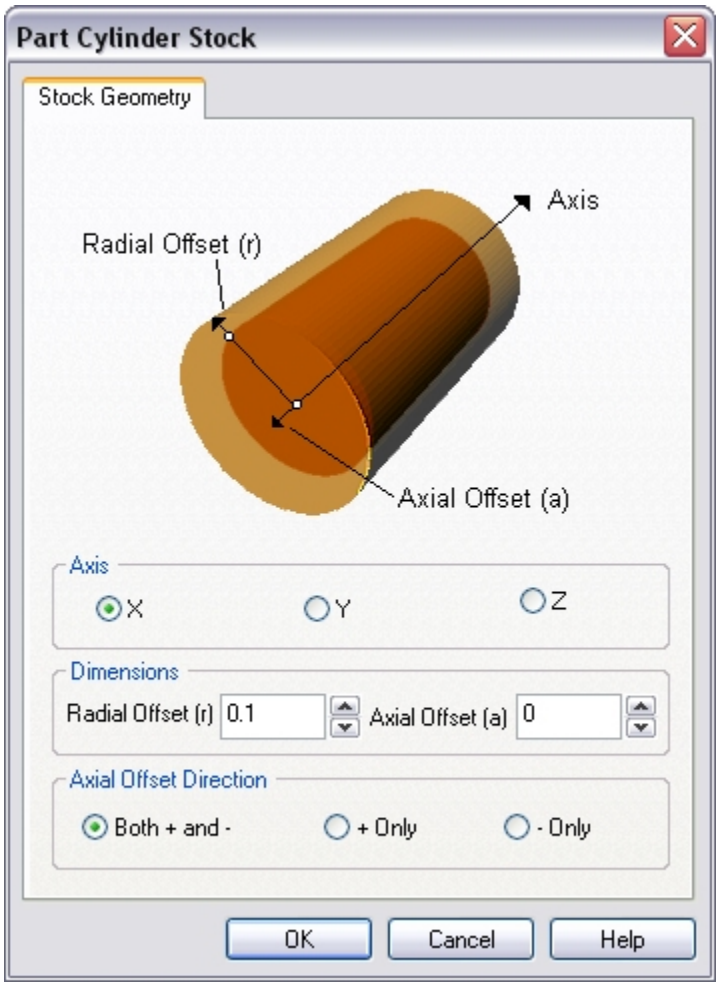


The system calculates the bounding box of the part model as the XYZ extents of geometry of the part model. The user can then define offsets in any of the three coordinate directions to apply to the computed bounding box. The system will expand the bounding box by the offset amount in each of the coordinate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

Part Cylinder Stock

Available in Alibre CAM Professional and Alibre CAM Expert

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



The user can define the cylinder by simply selecting the axis of the cylinder (either X, Y or Z) and defining the radial and the axial offset distances. The bounding cylinder will be calculated and these offset values will be used to expand the cylinder in both the radial and axial direction.

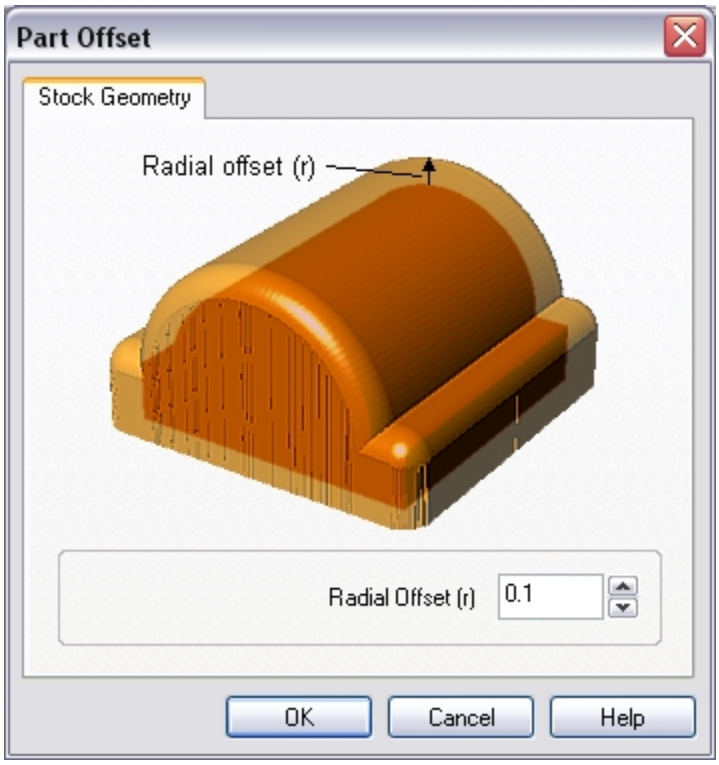
Additionally the user can specify the axial offset to be applied in both the positive and negative directions or only in the positive or only in the negative axial directions.

When the user clicks on the OK button, a stock model based on the part geometry and the user definition will be created. The user can switch to the Stimulate tab of the browser window to display the stock model that was created.

Part Offset Stock

Available in Alibre CAM Alibre CAM Expert Only

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



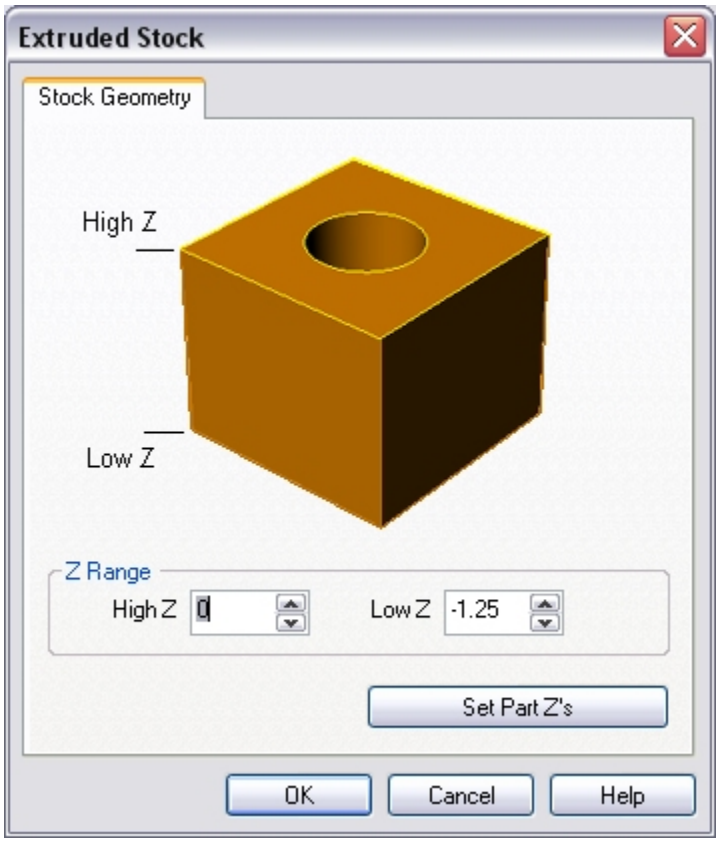
Regions Stock

Available in Alibre CAM Alibre CAM Expert Only

The user can define the raw stock model as a simple box by selecting the "Regions Stock" option from the "Create/Load Stock" under the Setup tab in Alibre CAM Browser

User selects regions for creating extrusions along the Z-axis.

The system creates a 3D stock model by extruding the currently active regions from the user defined upper Z-value to the user defined lower Z-value. These Z values can be defined in the dialog shown. Additionally these Z values can be automatically set to the part geometry maximum Z and minimum Z values by clicking on the Set Part Z's button shown in the dialog.



When the user clicks on the OK button, a stock model based on the selected regions and user definitions will be created. The user can switch to the Simulate tab of the browser window to display the stock model that was created.

Stock from Selection

Available in Alibre CAM Alibre CAM Expert Only

The user can define the raw stock model from the currently active selected 3-D geometry. The user can select 3D geometry and then select this command. No dialog will be invoked but the system will use the selected geometry and create a triangulated stock model. The user can switch to the Stock tab of the browser window to display the stock model that was created.

Tools

Alibre CAM allows the user to define, use and archive various types of milling tools. The tool types that are currently supported are ball, flat, corner-radius or bull, taper end mills*, Thread Mills*, Face Mills*, Dove Tail Mills**, Lollipop Mills**, User Defined Mills**, Drill tools, Center drills*, Reamer*, Tap*, Bore* and Reverse Bore*. Please note that some of these tools might not be supported in certain product configurations.

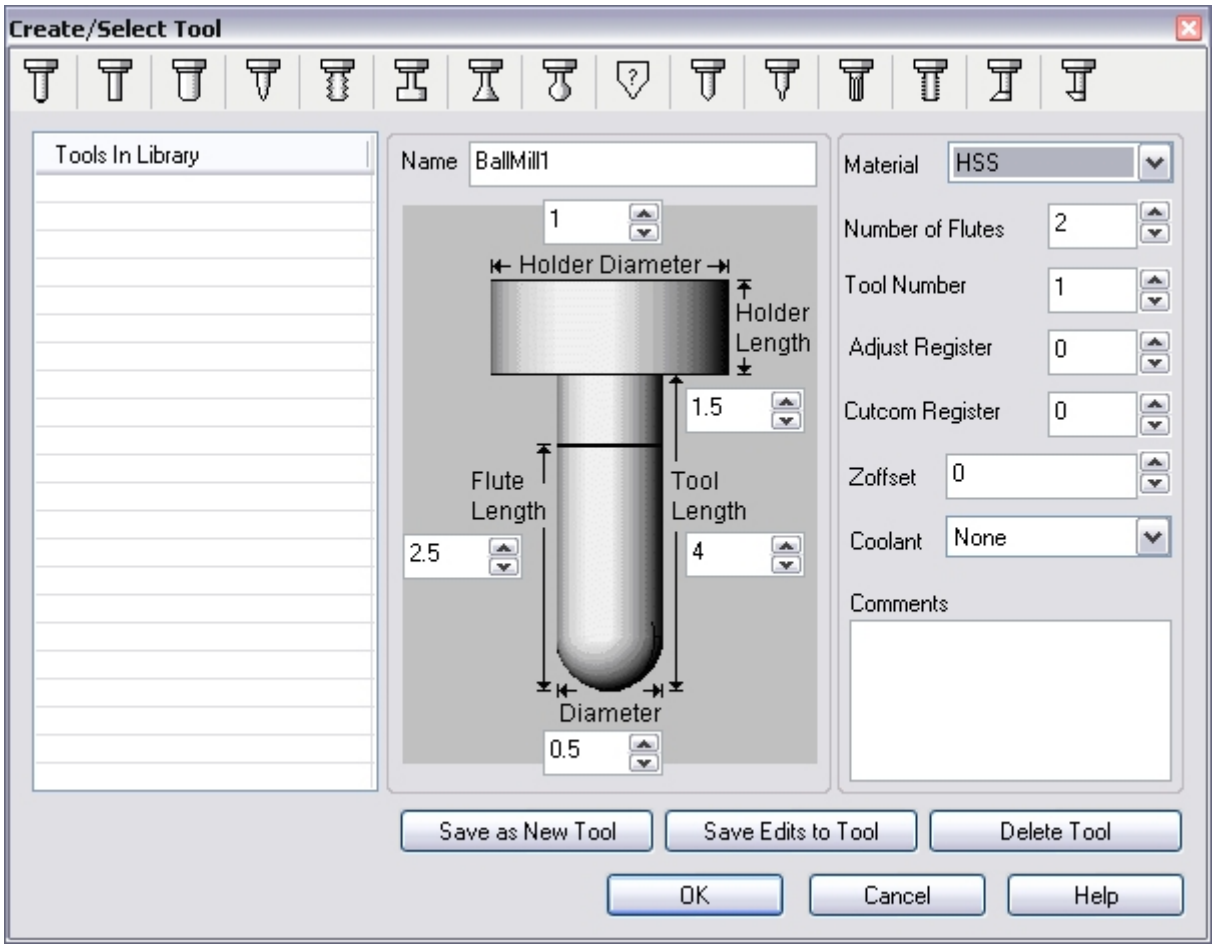
To create a tool select the "Create/Select Tool" button under the Tool tab in the Browser. This user selection brings up the following dialog box that the user can utilize to create and edit tool definitions.

* - Available in Alibre CAM Standard, Professional.

** - Available in Alibre CAM Expert Only.

Description:

The tool icon bar on the top of the dialog displays all of the types of tools available in Alibre CAM. Different tool types can be defined by selecting the desired icon in the dialog box.



The dialog box shows the tool name of the current selection if there is one selected in the list-box. If there is no selection then the tool name will be the name used for a new tool definition. The list box itself lists all of the tools of the corresponding type. The material selection box is used to select the tool material. The other fields beneath the tool list box display the number of flutes in the tool and the tool number. The number of flutes is used in feeds and speeds calculations. The tool number is used when post processing toolpaths.

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.

Create New Tool:

To create a new tool, select the tab to define the desired type of tool. Then change the parameters in the edit boxes of the dialog. Once the parameters of the new tool have been entered, click on the "Save as New Tool" button. Make sure that you specify a new tool name before selecting the "Save as New Tool" button. Otherwise the system will edit the existing tool of the same name with the modified parameters.

Edit Existing Tool:

To edit the parameters of an existing tool, select the tool in the list box. All of the fields in the dialog will be populated with the relevant information. Now change any of this information that is pertinent and then click on the "Save Edits to Tool" button to register these changes.

Make Active Tool:

The "OK" button will create (if not yet created) and save the highlighted tool. It will make it the current active tool in the system. The active tool can be seen displayed on the Status Bar.

Tool Library

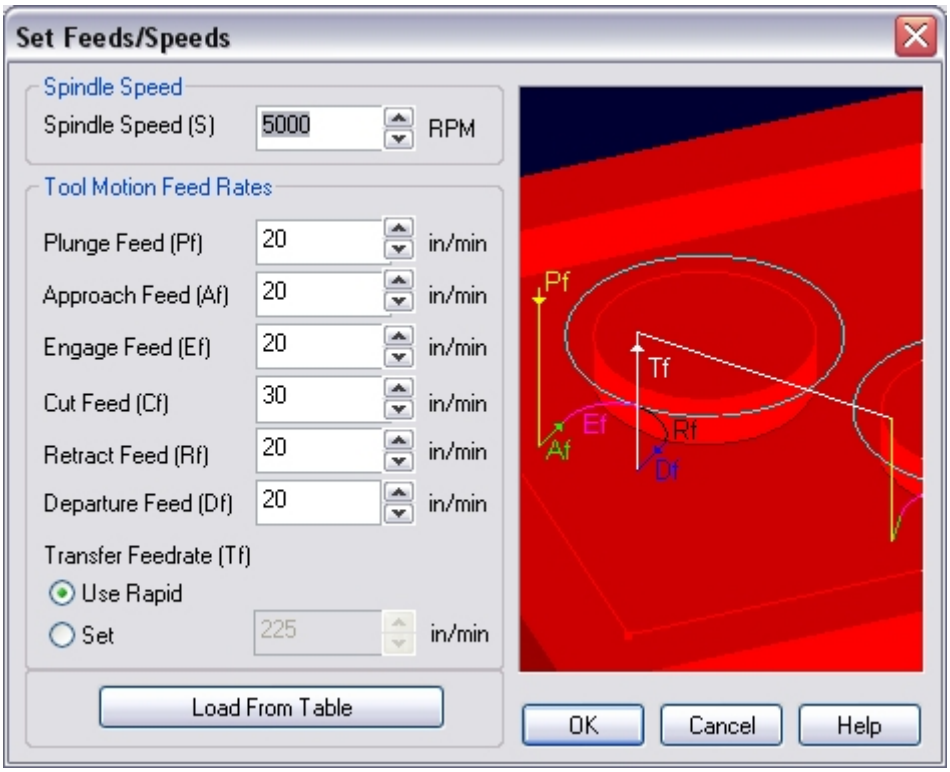
Alibre CAM allows the user to save tools in an external ASCII file. Such a file, called a tool library, can then be loaded into any part file to generate machining operations. This tool library is useful in shops where there is a standard set of tools that are used and these can be defined and stored in these library files. These tool library files can then always be loaded as a starting point for programming a new part. The tools can be saved as a Tool library (*.csv file) using the Save Tool Library option under the “Tools” menu bar option. When Alibre CAM is started the next time, the tool library can be loaded using the Load Tool Library option under the “Tools” menu bar option and selecting the previously saved tool library.

The list of defined tools can also be seen listed in the tools tab of the Browser.

Feeds and Speeds

Alibre CAM has powerful mechanisms that allow the user to not only set the feeds and speeds in a machining operation but also to customize these settings for later use. The following sections describe these mechanisms.

Alibre CAM allows setting of feeds and speeds to be used in the toolpaths via the Feeds/Speeds dialog. To bring up this dialog, click on the button in the machining bar. This brings up the dialog shown below:



Description:

This dialog gives the user the ability to set the following parameters:

- Spindle Speed – This is the rotational speed of the milling spindle expressed in RPM
- Plunge Feed - The Plunge speed rate is the speed before the tool starts to engage in material. This is always vertical.
- Approach Speed - The Approach speed is the speed used that prepares the cutter just before it starts engaging into material as it starts cutting. The approach motions are dependent on the method of machining.
- Engage Feed – The feed rate used when the milling tool is performing an engage move. Alibre CAM sets this value to be 75% of the cutting speed.
- Cut Feed – This is the feed rate used when the milling tool is cutting material
- Retract Feed – The feed rate used when the milling tool is performing a retract move away from material. Alibre CAM sets this also to also be 75% of the cutting speed.
- Departure Feed - The speed used to retract the tool from the material.

In addition to this the user can specify how transfer motions are to be handled by the post-processor. Transfer motions occur when the tool is moving in air to transfer from one cut location to another. The user has the choice to use the “Rapid” setting in the machine tool or to actually specify a transfer feed rate.

All of the entered parameters will be used by the post-processor during post-processing of toolpath files.

Load from Table

Alibre CAM allows the user to load feeds/speeds from an external table accessible through the Load From Table button. This will bring up the following dialog that will allow the user to load feeds and speeds by an externally customized table. An explanation of the Feeds and Speeds table is given below.

Feeds & Speeds Table

Alibre CAM allows the user to customize based on the stock material being machined, the material of the cutter used and also the operation type. This is done by archiving the users desired settings in an external man readable data file.

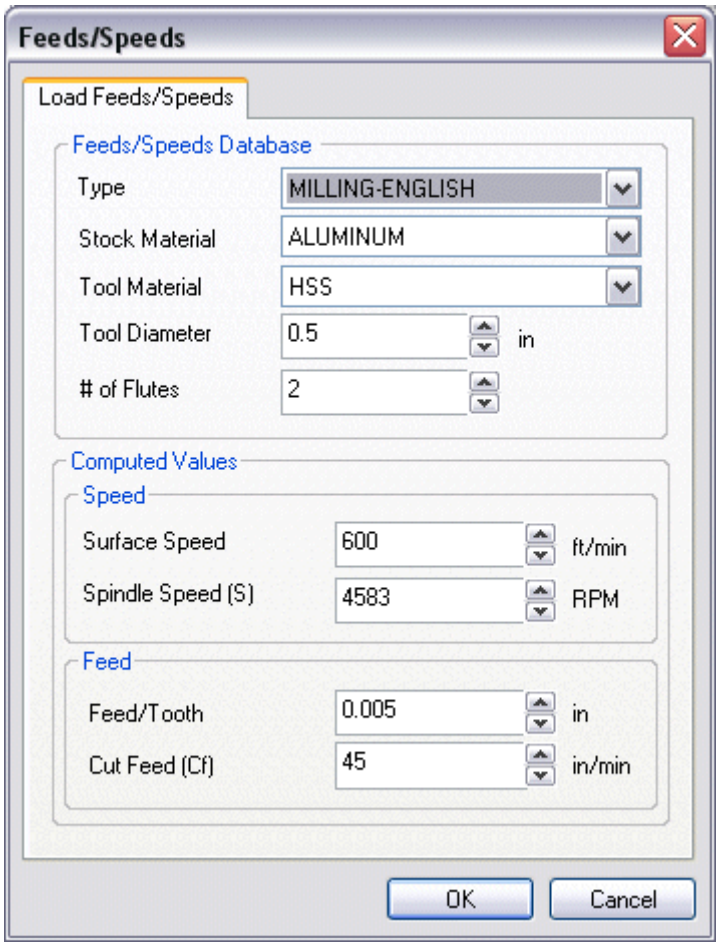
A default implementation of this table has been included with the Alibre CAM product and can be found in a folder called Data under the product installation directory. The name of this file is FEEDSSPEEDS without an extension. Users can edit this file with their favorite text editor and can customize the feeds/speeds settings based on user defined operation types, type of material being cut, tool material and geometry and then store these values in an ASCII external file. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program. The format for this file is shown below.

```
TYPE
{
STOCK_MATERIAL   TOOL_MATERIAL   SURFACE_SPEED   UNITS   FEED_PER_TOOTH   UNITS   MACHINABILITY   UNITS
STOCK_MATERIAL   TOOL_MATERIAL   SURFACE_SPEED   UNITS   FEED_PER_TOOTH   UNITS   MACHINABILITY   UNITS
.....
}
```

An example entry is shown below. For more information please look in the FEEDSSPEEDS file.

```
FaceMilling
{
aluminum-cast     carbide     900     fpm     0.0200     in     4.0     ci/min/hp
aluminum-cast     hss         650     fpm     0.0160     in     4.0     ci/min/hp
aluminum-plate    carbide     900     fpm     0.0200     in     4.0     ci/min/hp
aluminum-plate    hss         650     fpm     0.0160     in     4.0     ci/min/hp
}
```

Once the values are set in this file, they can be conveniently loaded by selecting the Load From Table button in the Feeds/Speeds dialog.



The top three edit boxes in the dialog shows the type, stock material and tool material. The values for these fields are populated from the external data file that the user has the ability to customize. Based on these choices Alibre CAM searches this data file and loads the corresponding Surface Speed and the Feed/Tooth values entered in the file. The Tool Diameter and the # of Flutes values are taken from the currently active tool (if any) and the corresponding Spindle Speed (S) and the Cut Feed (Cf) are computed. The user has the ability to change the diameter or the number of flutes values and see the corresponding feed and speed values being computed and updated. Thus the user can use this dialog as a feed rate and speed calculator if so desired.

When the user selects the OK button, the calculated cut feed will be transferred to the Cut Speed box in the Set Feeds/Speeds dialog.

Machining Regions

Regions are sketches that already exist in your model, or separate new sketches you create within Alibre Design that coexist with your part, but do not contribute to its geometry (the sketch is not used for a feature). Regions serve different purposes in 2½ and 3 axis milling.

In 3 axis milling, regions are used, when necessary, to define the machining boundary. The center of the tool remains inside the region while following the contours of the part. For example, if you define a region surrounding one surface, only that surface will be milled.

In 2½ axis milling, however, regions determine the entire area to be milled. Part and stock geometry are not considered, and the tool always moves at a fixed Z level. Therefore, regions act as the drive curves, and their geometry is projected onto the stock.

Regions must be selected before they can be used in an operation. Creating a region does not make it active; you must use one of the Select Regions tools before creating the toolpath.

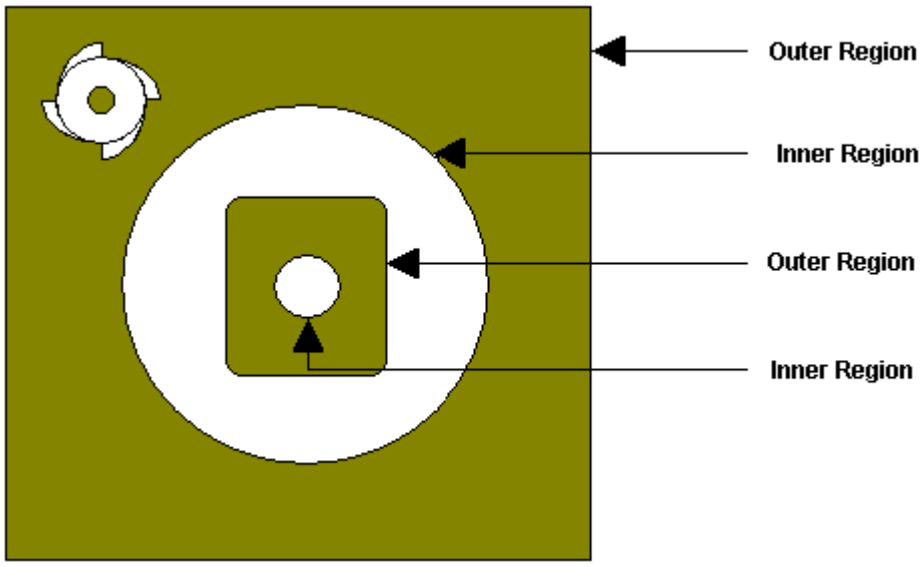
Note: *If a region is not defined for finishing operations, the part silhouette is automatically identified and used as a containment region for the tool. For roughing operations, the stock geometry determines the constraints on the tool location.*

Because 2½ axis milling does not use part geometry, milling regions must be created to define the areas to be machined.

There are four regions already defined for the part. You can also see the regions in the **Setup** tab of the Browser.

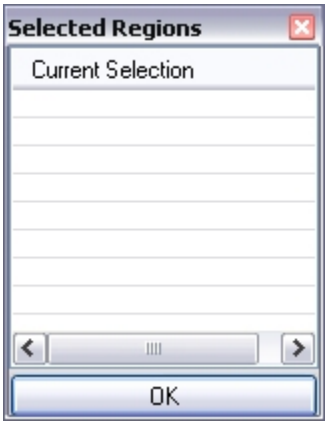
Multiple Regions

Multiple and nested regions can be selected, but not regions that intersect. Nested regions are handled according to the following rule: The tool will remain inside an outer region and outside an inner region. A region within an inner region is considered to be an outer region. In the following picture, the shaded areas are where the tool motions occur:



Selecting Regions in Machining Operations

Regions must be selected before they can be used in an operation. To select regions, you can use the standard Alibre Design selection methods to select curves. An optional way would be to click on the **Select Regions** button in the toolbar corresponding to the **MOps** tab. This will bring up the dialog shown below and allow the user to use single selection to select curves. As each sketch is selected by clicking on the graphics view-port, the selection will be indicated in the selector dialog.



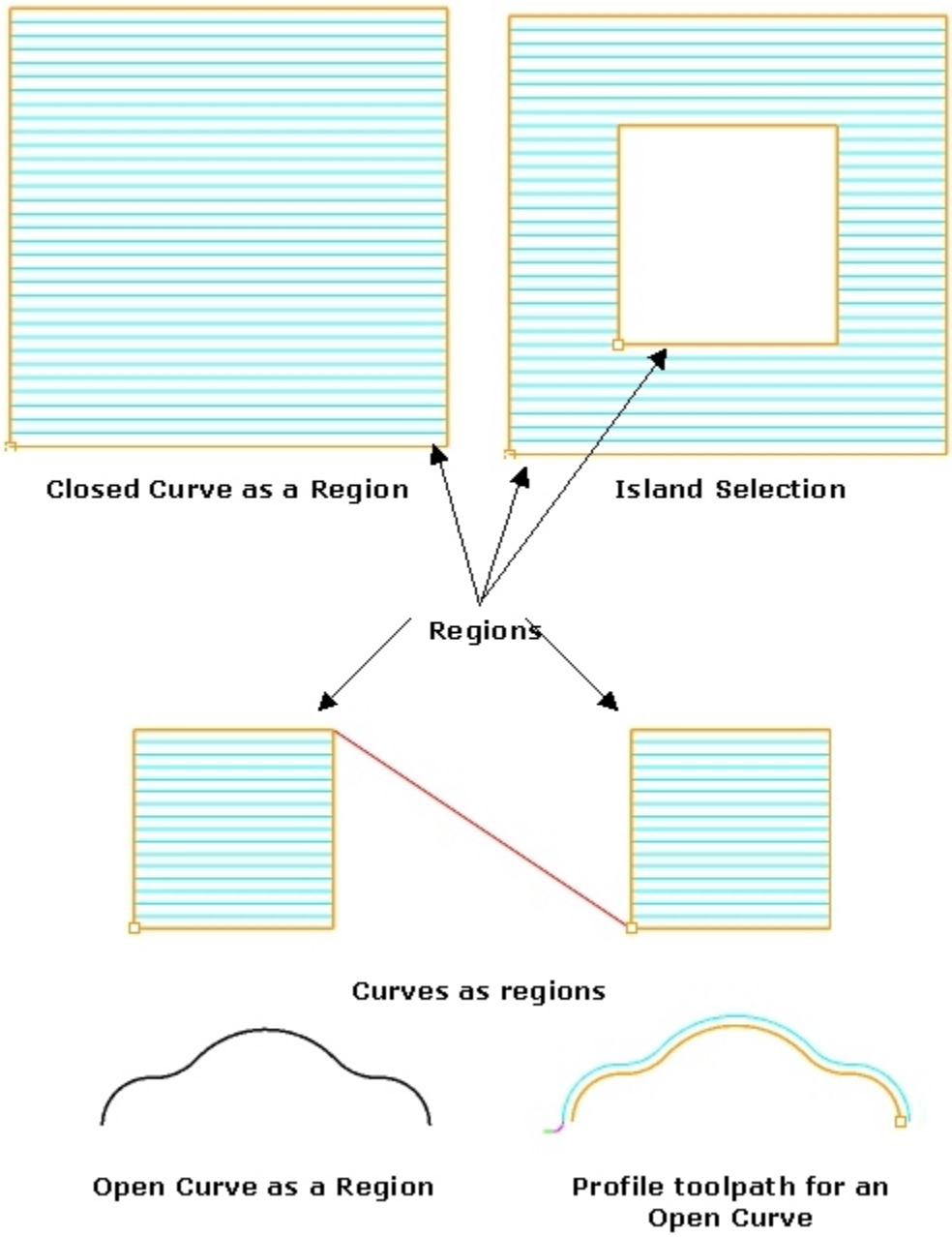
Once a machining operation is created, new regions can be selected or existing regions can be removed from the operation as well. Double clicking on the **Regions** icon under the **MOps** folder icon in the **Browser** window will bring up the same dialog shown above to allow the user to accomplish this.

Criteria for selecting Sketch(s) as regions

- Open and closed sketches (Lines, Polylines, Arcs, Circles, Polycurves) can be selected as regions for 2 ½ axis Profiling, Engraving operations.
- Only sketches can be selected as regions for 2 ½ Axis Facing, Pocketing, V-Cutting (Engraving TO condition), 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.
- 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)

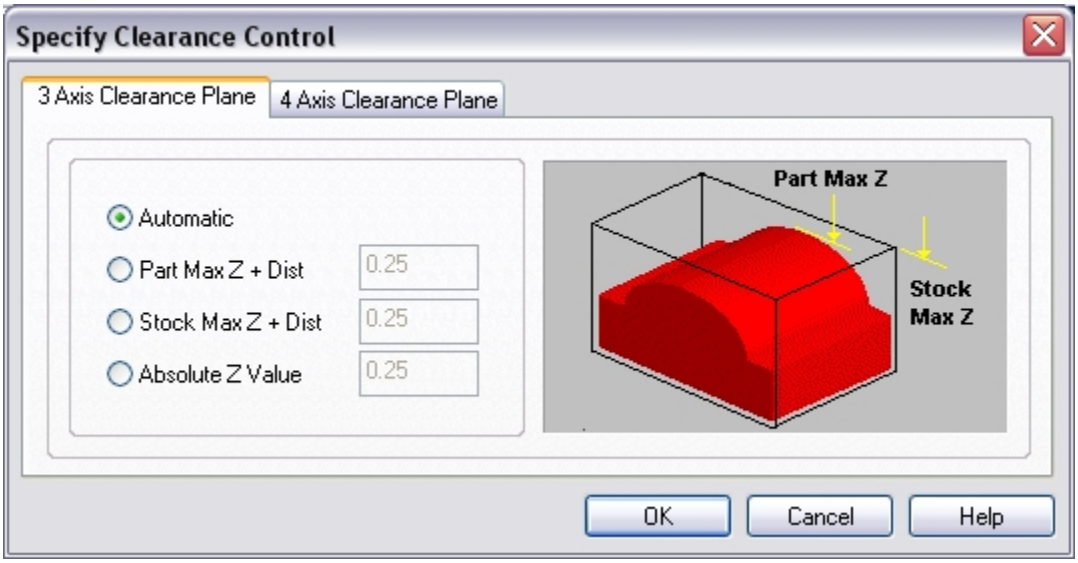


Clearance Plane

Alibre CAM allows setting of the default clearance plane for transfer motions in various ways. The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion

will take place. Typically the user would define this plane a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions because these motions usually use a very fast or rapid feed rate.

The default clearance plane can be defined by clicking on the **Clearance Plane** button in the toolbar of the **MOps** tab in the browser window. This will bring up the 3-axis clearance plane dialog shown below:

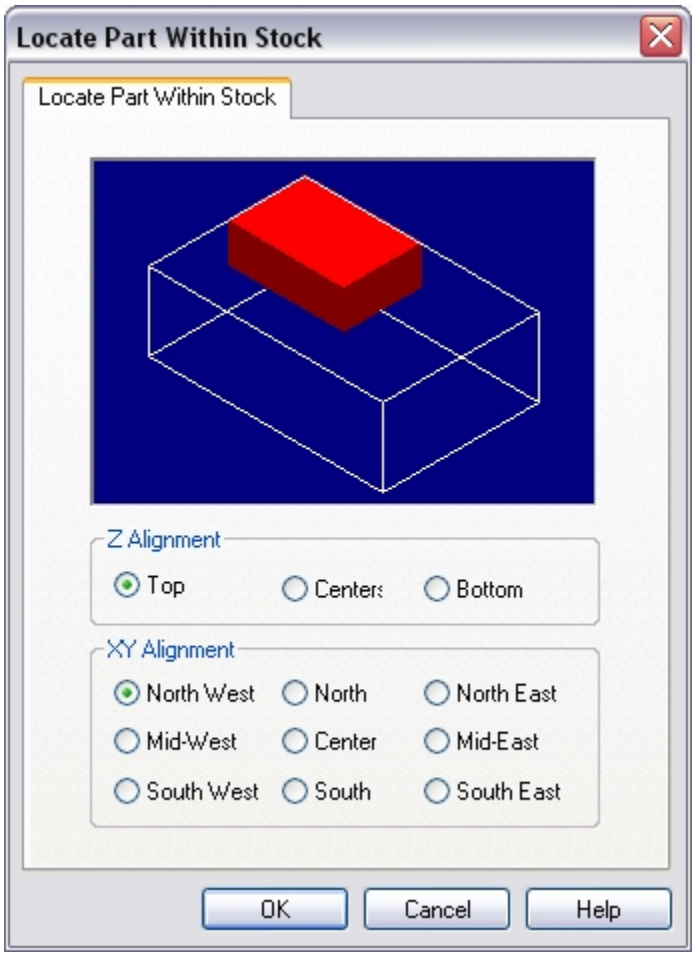


Description:

The user can define the clearance plane in four different ways as shown in the dialog radio box. The user selects one of the four choices to define the clearance plane. In the "Automatic" definition the system computes the clearance plane automatically by looking at the part and/or stock geometry and adding a safety distance to the highest point of the interrogated geometry. This safety distance is hard-coded currently to be the radius of the tool currently in use. In future releases, users will be able to customize this. If you desire to set the clearance plane as a specific distance above the maximum Z point of the part geometry use the second choice and key in the safety distance value. You can also set the clearance plane above the highest point in the stock geometry by selecting the third choice. The final choice allows entry of a specific Z value for the clearance plane.

Locate Part within Stock

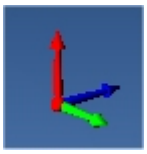
It is typical for users to need the ability to position stock geometry in some geometric relationship with the part geometry. A typical scenario is that the user has molded the part with a pre-determined origin. In such cases it would be desirable to locate the stock with respect to the already positioned part without having to go through actually calculating the transformation delta values.



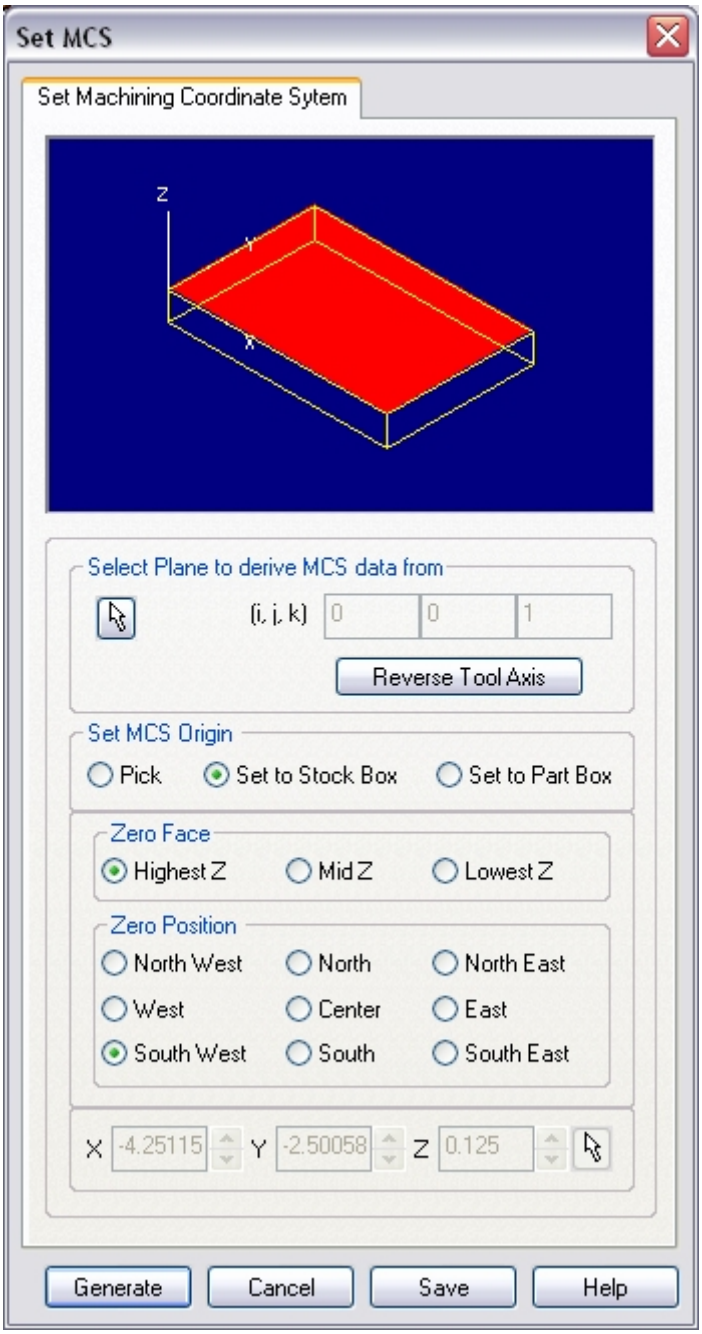
Z alignment of the stock can be set to the Top / Center / Bottom of the part geometry and the XY alignment can be set to one of the configurations shown in the dialog box above.

Set MCS

This dialog is used to set the Machine Coordinate System. The Machine Coordinate System is the coordinate system that defines the tool orientation as well as the tool zero position.



MCS is displayed as a Triad where the Red represents the Z-axis, Green represents X-axis and Blue represents Y-axis.



MCS can be set using one of the above methods.

1. Selecting a Plane (For example: XY, XZ or YZ). This feature is available only in Alibre CAM Pro & Expert versions.
2. Set to Stock box
3. Set to Part box
4. Using the Pick option and selecting an edge on the part.

Typically a user would create a stock geometry, locate the stock in relation to the part geometry and use the SetMCS to specify the Machine Zero Position.

Reverse Tool Axis inverts the Z-axis from the current Set MCS orientation. This feature is available only in Alibre CAM Pro & Expert versions.

Alibre CAM Pro & Expert configuration allows the user to create more than one Set MCS operation.

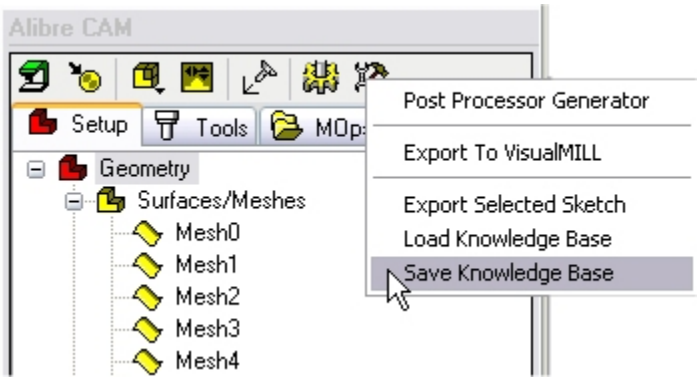
Knowledge Base

Available in Alibre CAM Expert only

Alibre CAM 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 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, pick the "Save to Knowledge Base" option located under the Setup Tab Utilities->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.

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.

Alibre CAM Preferences

Alibre CAM allows setting of various user preferences that will be saved even after the user exits the program. To access the functions to set Alibre CAM preferences, select the "Preferences" option under the Setup tab in Alibre CAM Browser.

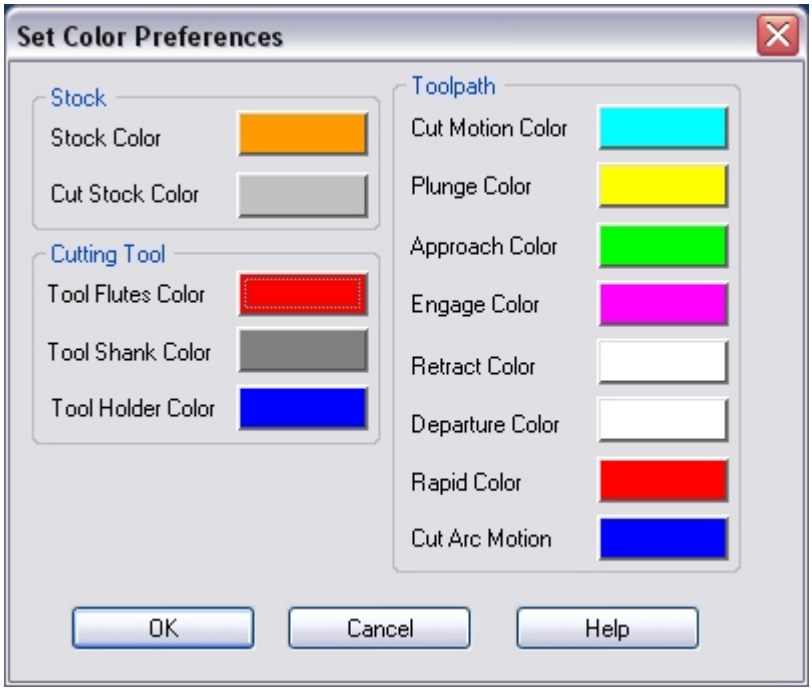
Color Preferences- Used to set the colors various objects in Alibre CAM

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

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

Color Preferences

Users can set the colors to display various objects in Alibre CAM 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.



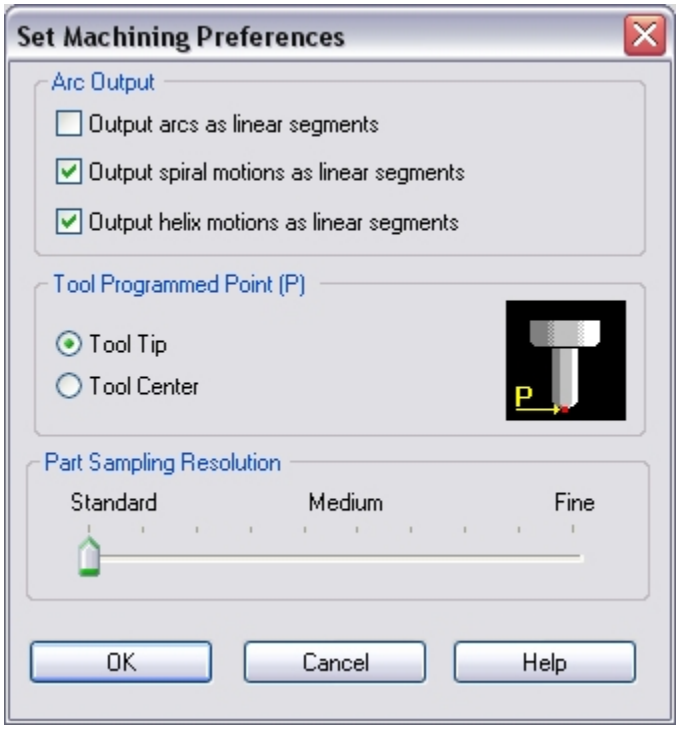
Cutting Tool Colors - Set the tool's flute (cutting portion) and the shank's (non-cutting portion) color in this section.

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

Users can set the machining preferences using this dialog.



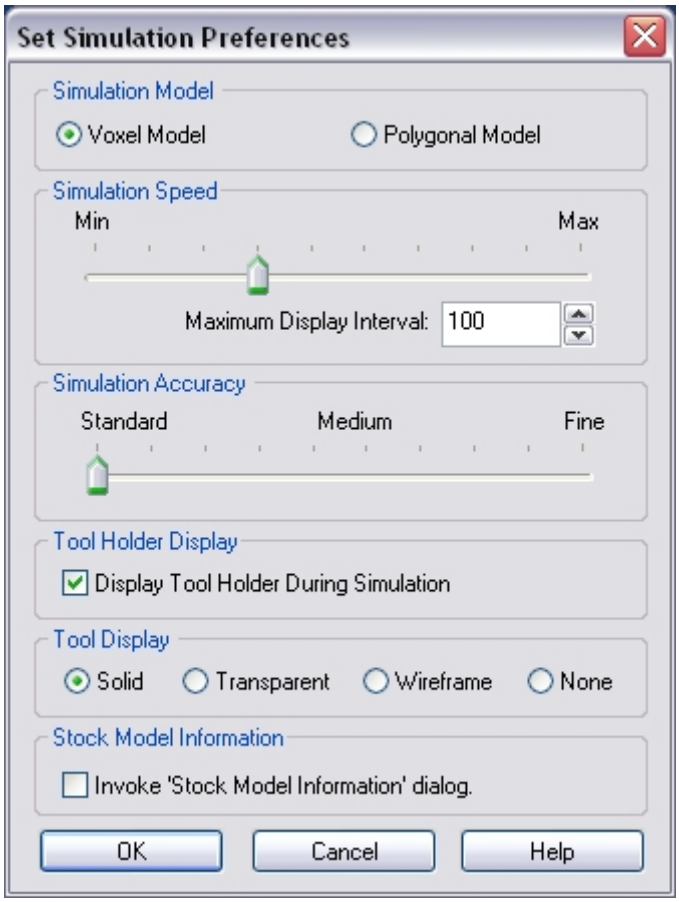
Arc Output: Some of the controllers do not have arc, spiral and helical output (for example G2, G3). For such type of controllers, the arcs that are generated in the Alibre CAM toolpath can be output as linear segments using this option.

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. Alibre CAM 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.

Simulation Preferences

Users can set the simulation preferences using this dialog.



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

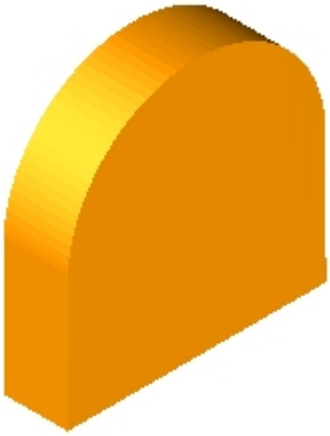
Note:

Voxel Model is available in Alibre CAM Standard, Expert and Professional versions only.

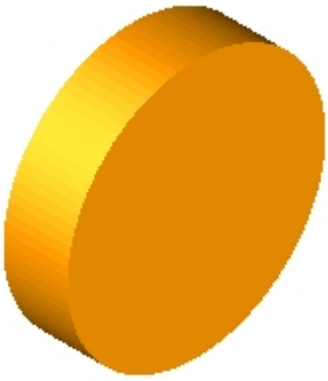
Polygonal model is only available in Alibre CAM Professional and Alibre CAM Expert.

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.

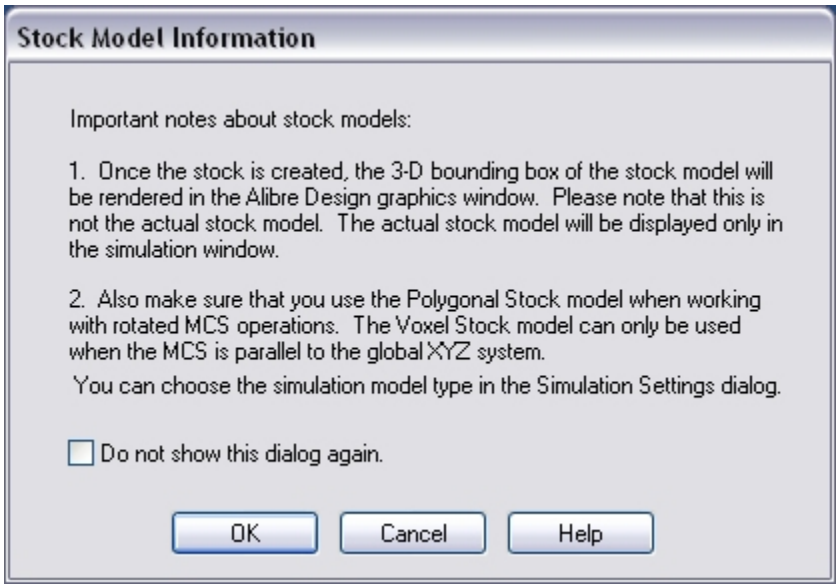
Tool holder Display: User can turn on /off the display of tool holder during simulation.

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

Invoke 'Stock Model Information' dialog: User can turn on/off the stock model information dialog.

Stock Model Information

The stock model information dialog is displayed when a stock geometry is created.



User can turn off this dialog by selecting Do not show this dialog again located on the bottom of the message window.

To display this dialog during stock creation, select Alibre CAM Preferences->Simulation Preferences and select Invoke 'Stock Model Information' dialog.

The dialog displays the following information

1. The stock model displayed in the Alibre Design graphics window represents the bounding box and not the actual stock model. The actual stock model will be displayed in the Alibre CAM Stock Simulation Window.
2. The stock model must be switched to Polygonal stock model when working with rotated MCS & rotate table operations. This can be set under the Simulation Preferences dialog.
3. The Voxel stock model can be used only when the MCS is parallel to the global XYZ system.

Utilities

Post Process Generator- Loads Post Processor Generator.

Export to VisualMILL – Exports the Part geometry, and machining operations to VisualMill 5.0

Export Selected Sketch – Exports selected sketch as a region file (*mrg)

Load Knowledge Base* – Loads a knowledge base.

Save Knowledge Base* – Save machining operations to a knowledge Base. Refer to Knowledge Base for detailed description.

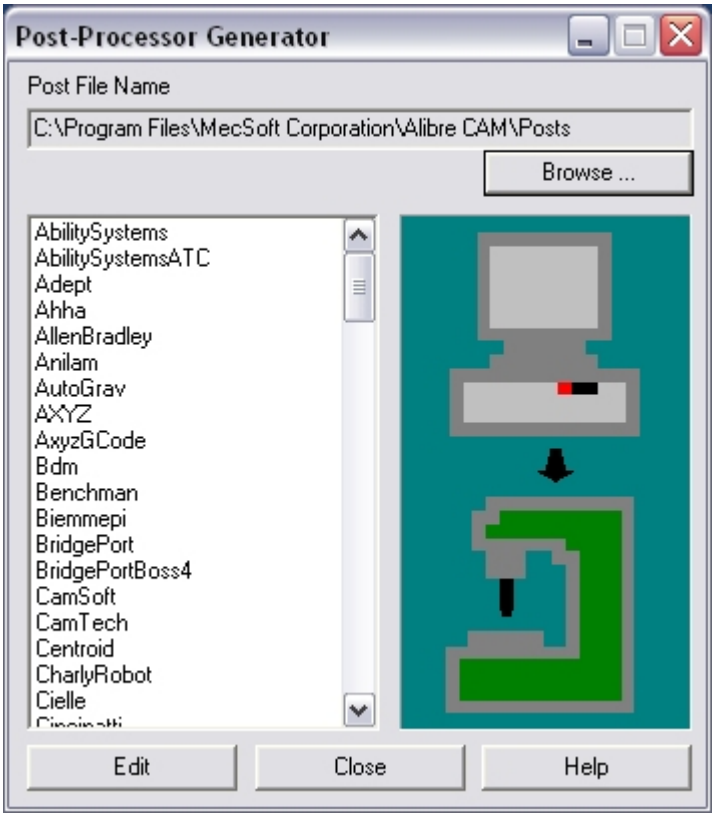
* - Available in Alibre CAM Expert Only.

Post Process Generator

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

Alibre CAM includes a utility called **Post Processor Generator** that you use to edit post processor parameters, and set up your own post-processors. You can access this by selecting the **Utilities** button in the **Setup** tab toolbar.

This will bring up a list of available post processors. Select a post processor to change or copy, and click Edit. The editable parameters can be found on the various tabs as shown below. Each tab can be selected to customize the various M & G codes for the selected post processor.



To create a new post-processor, define the parameters and click **Save As**. A new spm file will be created in the **Post** folder.

You can also output the toolpath in an APT standard Cutter Location (CL) file. APT is a widely accepted Numerical Control Machine standard. This CL file can then be used to create a machine specific post-processed output through any of the many commercially available APT post-processors.

Creating 2 ½ Axis Machining Operations

The section below details all of the 2 ½ axis machining operation types that can be created in Alibre CAM.

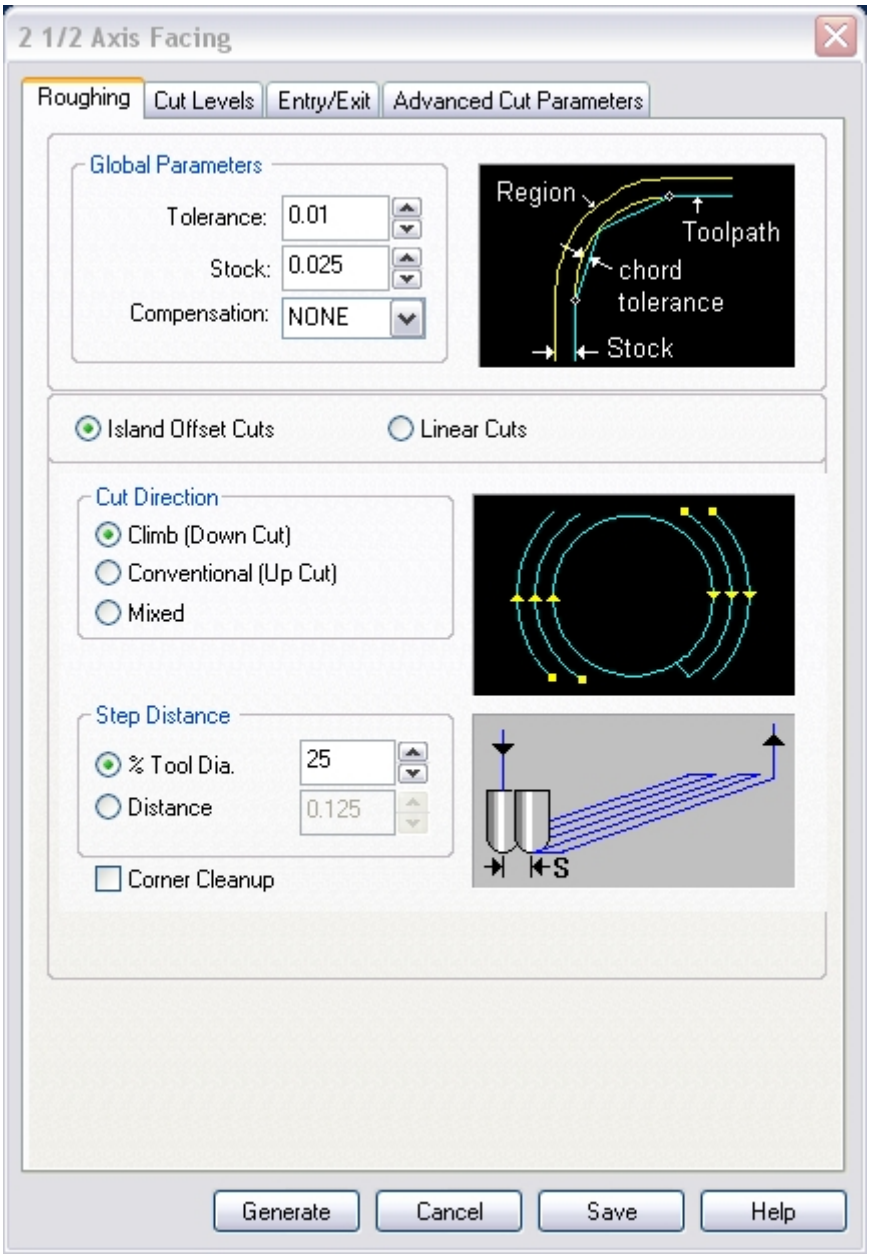
Facing

Available in Alibre CAM Standard, Expert and Professional versions only

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.

Description

The Facing toolpath is invoked by clicking on the  button in the MOps Browser and picking the **2½ Axis Milling** and **Facing** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



This dialog is tabbed with four tabs. Each tab defines a set of parameters that the user can specify. The sections below describe them in detail.

Roughing

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

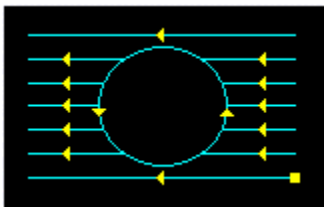
The Global Cut Parameters section allows the user to set the tolerance value to be used in machining. Also any uniform thickness or stock that needs to be left of the part also can be specified here.

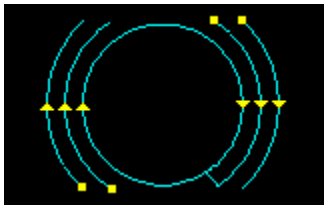
Stock The thickness of the layer that will remain on sides of the part after the toolpath is complete. Roughing operations generally leave a thin layer of stock, but for finishing operations this value is zero.

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

Compensation stands for cutter compensation. The user can turn this on by selecting from the drop down menu. The cutter compensation to the left or right is determined by the Climb or Conventional direction.

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 a linear cut pattern where the tool will traverse in a linear cuts always or an offset cut pattern where the tool will traverse in successive uniform offsets of the part shape.


Linear Cut Pattern


Island Offset Cut Pattern

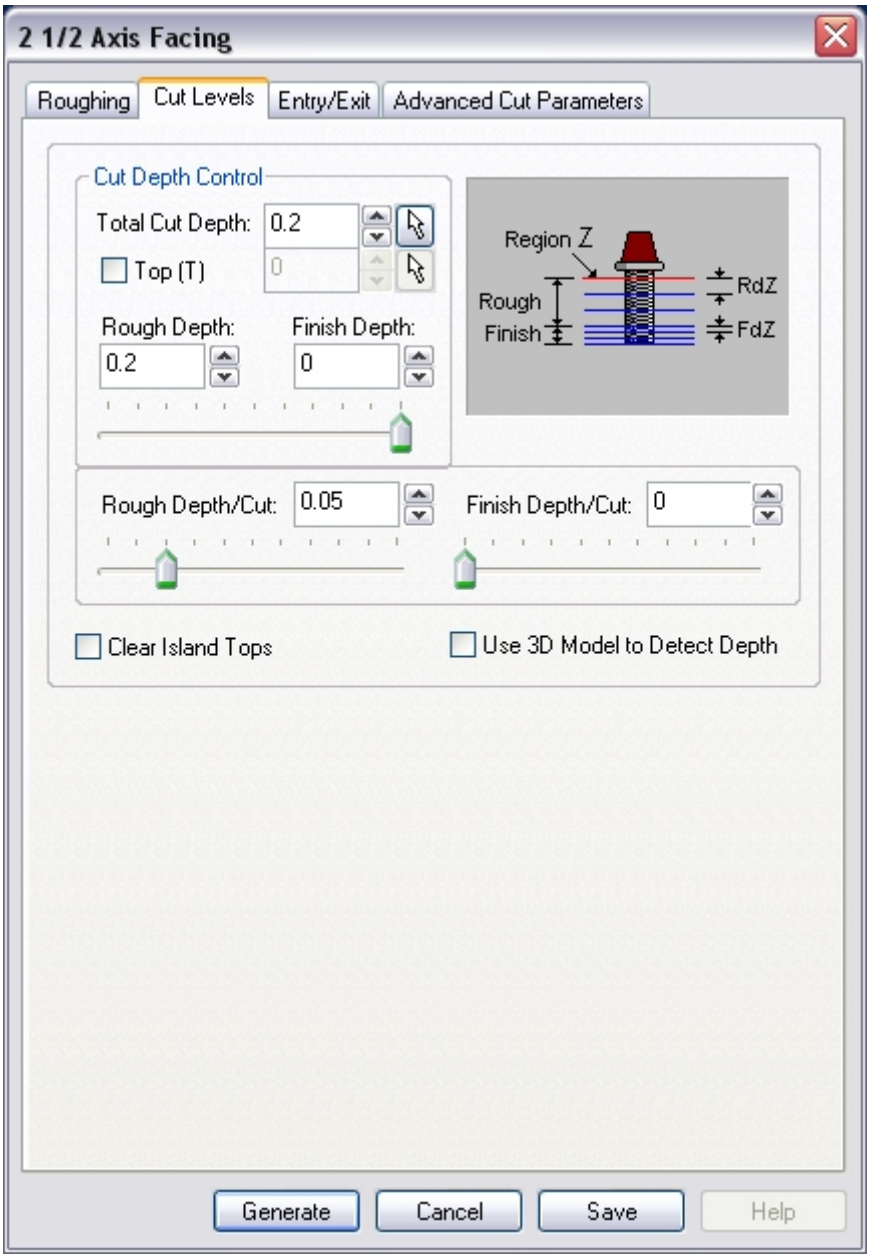
The **Cut Direction** can be specified as Climb, Conventional or Mixed.

The **Step-over 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

The corner cleanup will automatically detect all the corners where the tool could not get in, in between each pass and will add a cut based on the uncut area, either a linear cut in case of a smaller area or will travel along the shape of the uncut area, when the area is large.

Cut Levels

When the user selects this tab in the dialog the following property page is displayed.



The user can set the top Z level by enabling the check box titled Top and specifying the top level. If the top level is not specified than the top of the outer most region is selected as the top Z level. The cut depth can be specified either by typing in the value or by selecting the pick button and selecting the bottom of the pocket. A depth of 0.0 will give one pass at the top Z level. The cuts can be specified as rough cuts and finish cuts, by sliding the bar or by typing in the values. Similarly for the rough and finish depth of cuts.

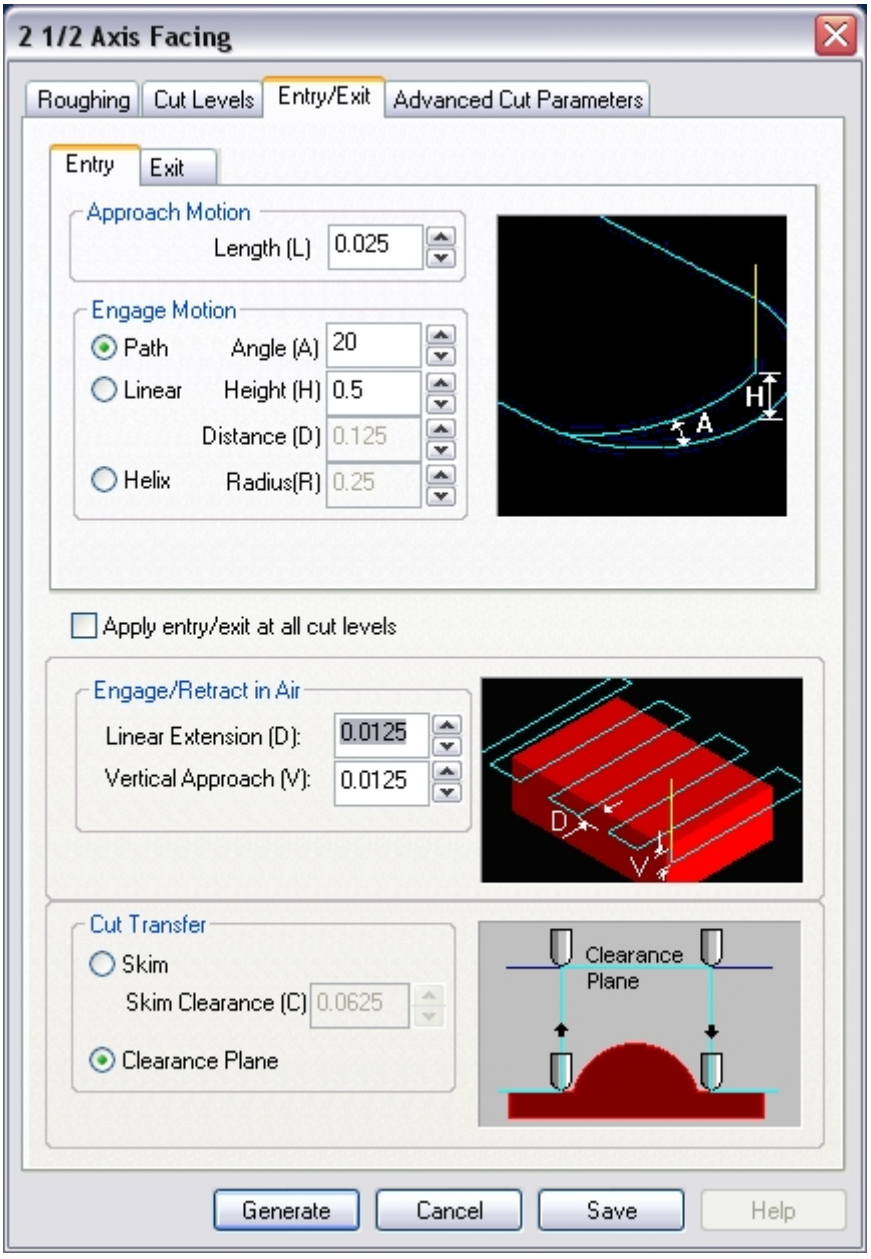
The clear islands will add a cut at the island levels (inner regions).

The use 3d Model to detect depth can be used when there is a 3d model available. This will detect the bottom most depth of the pocket (if there are multiple depths) and will add toolpath until that

level. The finish depth can be specified when this option is used. The roughing depth is determined by the difference of the total depth and the finishing depth.

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.



The Entry motion consists of Approach and Engage. 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.

In a facing operation, the outer most regions (the outer most region and its immediate inner regions) are cut in a facing approach and all the inner pockets are cut using the pocketing approach. The Engage in Material section is for the pocketing toolpath that is generated for the inner pockets. The Engage in Air section is for the outer most regions as described above

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.

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 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 form the outside, a certain distance away from the cut start point. Or optionally a straight vertical engage can be specified.

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.

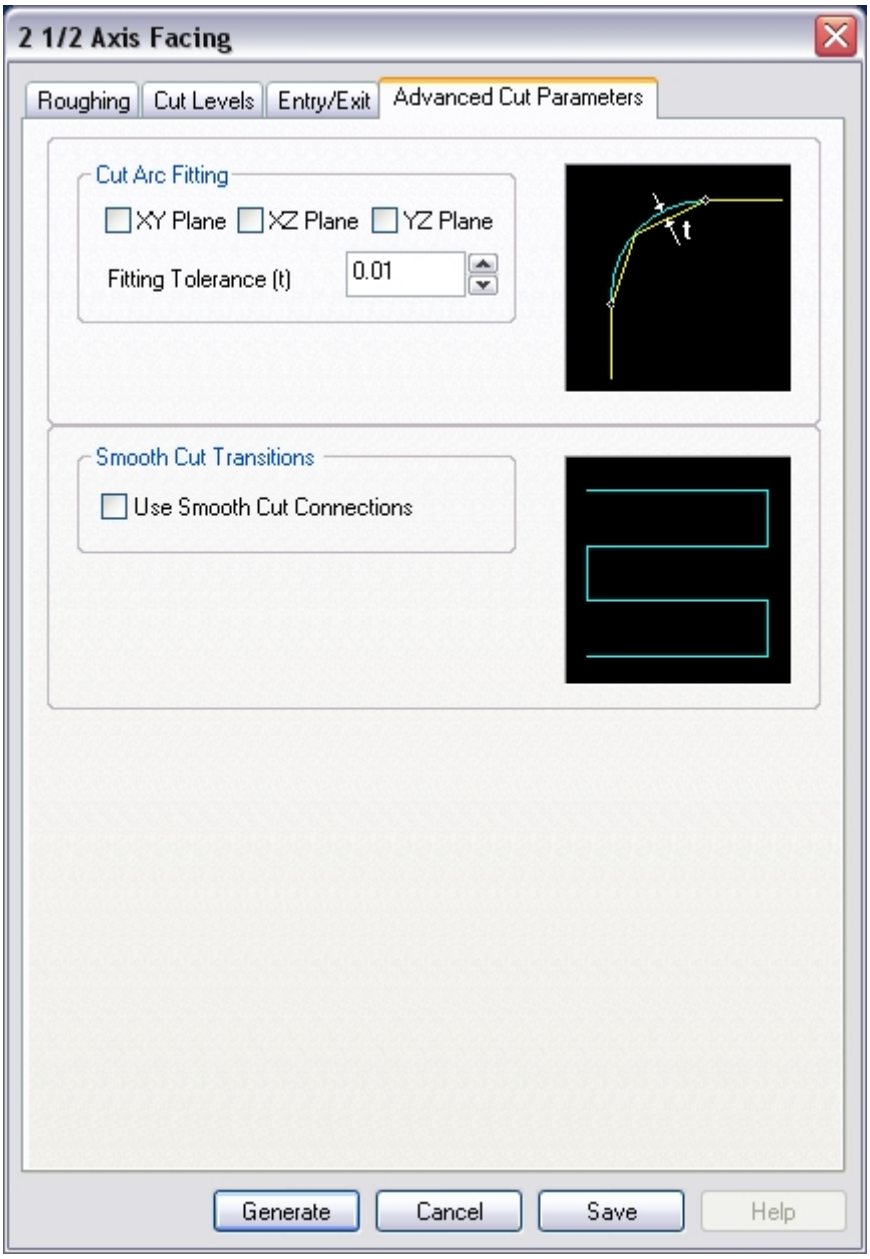
The user can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can either be instructed to move to the clearance plane and then perform the transfer motion to the next cut location or it could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

Advance Cut Parameters

The advanced cut parameters are 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 your machine during the cutting process. These parameters allow smoothing of the toolpaths by introduction of

arcs. You can use these parameters even if your controller does not support arcs. Make sure that your output is set to linear output. You can do this under the Machining Preferences dialog in the Preferences section of the Menu bar.

- **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 other in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.



Pocketing

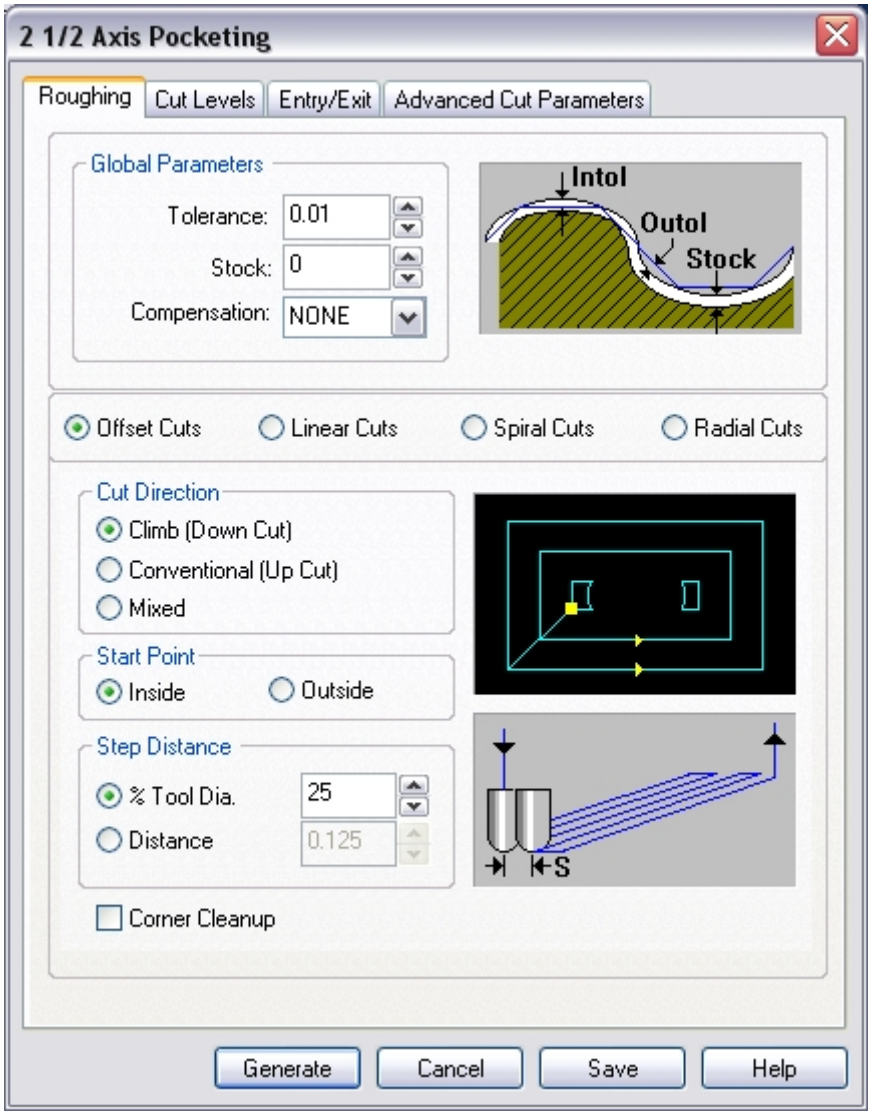
Available in Alibre CAM Standard, Expert and Professional versions only

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 a linear cut pattern or an offset cut pattern.

Description

The Pocket Machining toolpath is invoked by clicking on the button in the MOps Browser and picking the **2½ Axis Milling** and **Pocketing** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



This dialog is tabbed with four tabs. Each tab defines a set of parameters that the user can specify. The sections below describe them in detail.

Roughing

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

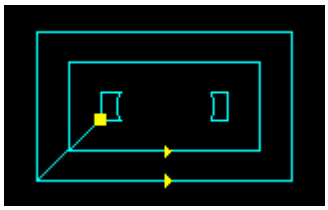
The Global Cut Parameters section allows the user to set the tolerance value to be used in machining. Also any uniform thickness or stock that needs to be left of the part also can be specified here.

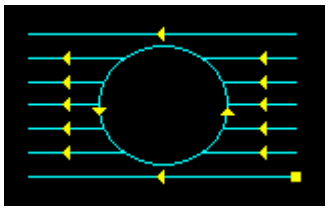
Stock The thickness of the layer that will remain on sides of the part after the toolpath is complete. Roughing operations generally leave a thin layer of stock, but for finishing operations this value is zero.

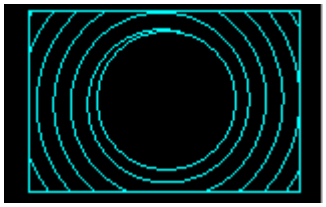
Tolerance is the allowable deviation from the actual part geometry plus the Stock layer (if any).

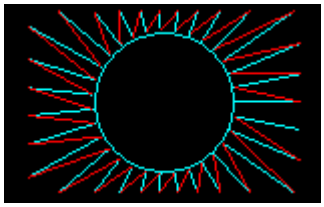
Compensation stands for cutter compensation. The user can turn this on by selecting from the drop down menu. The cutter compensation to the left or right is determined by the Climb or Conventional direction.

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 a linear cut pattern where the tool will traverse in a linear cuts always, or an offset cut pattern where the tool will traverse in successive uniform offsets of the part shape, or a spiral cut pattern which can go from inside out or outside in, or a radial cut pattern.


Offset


Linear


Spiral


Radial

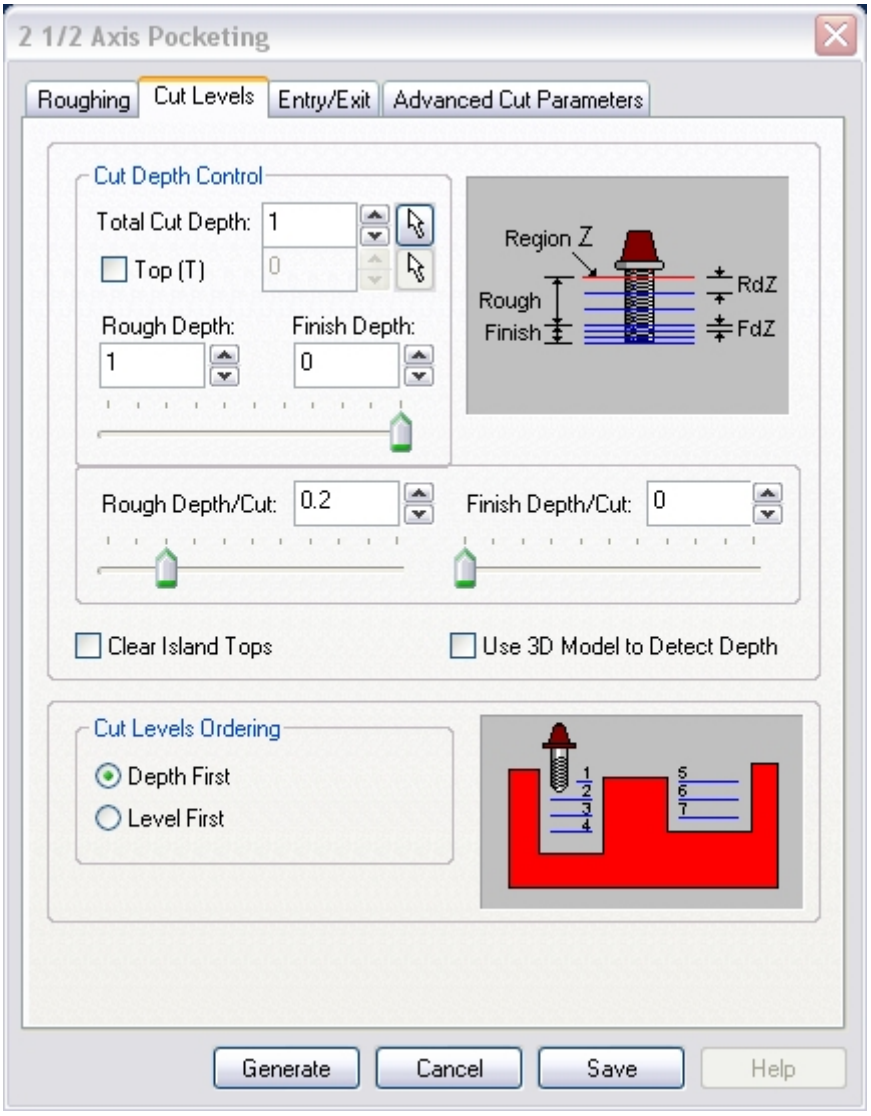
The **Cut Direction** can be specified as Climb, Conventional or Mixed.

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

The **corner cleanup** will automatically detect all the corners where the tool couldn't get in, in between each pass and will add a cut based on the uncut area, either a linear cut in case of a smaller area or will travel along the shape of the uncut area, when the area is large.

Cut Levels

When the user selects this tab in the dialog the following property page is displayed.



The user can set the top Z level by enabling the check box titled Top and specifying the top level. If the top level is not specified than the top of the outer most region is selected as the top Z level. The cut depth can be specified either by typing in the value or by selecting the pick button and selecting the bottom of the pocket. A depth of 0.0 will give one pass at the top Z level. The cuts can be specified as rough cuts and finish cuts, by sliding the bar or by typing in the values. Similarly for the rough and finish depth of cuts.

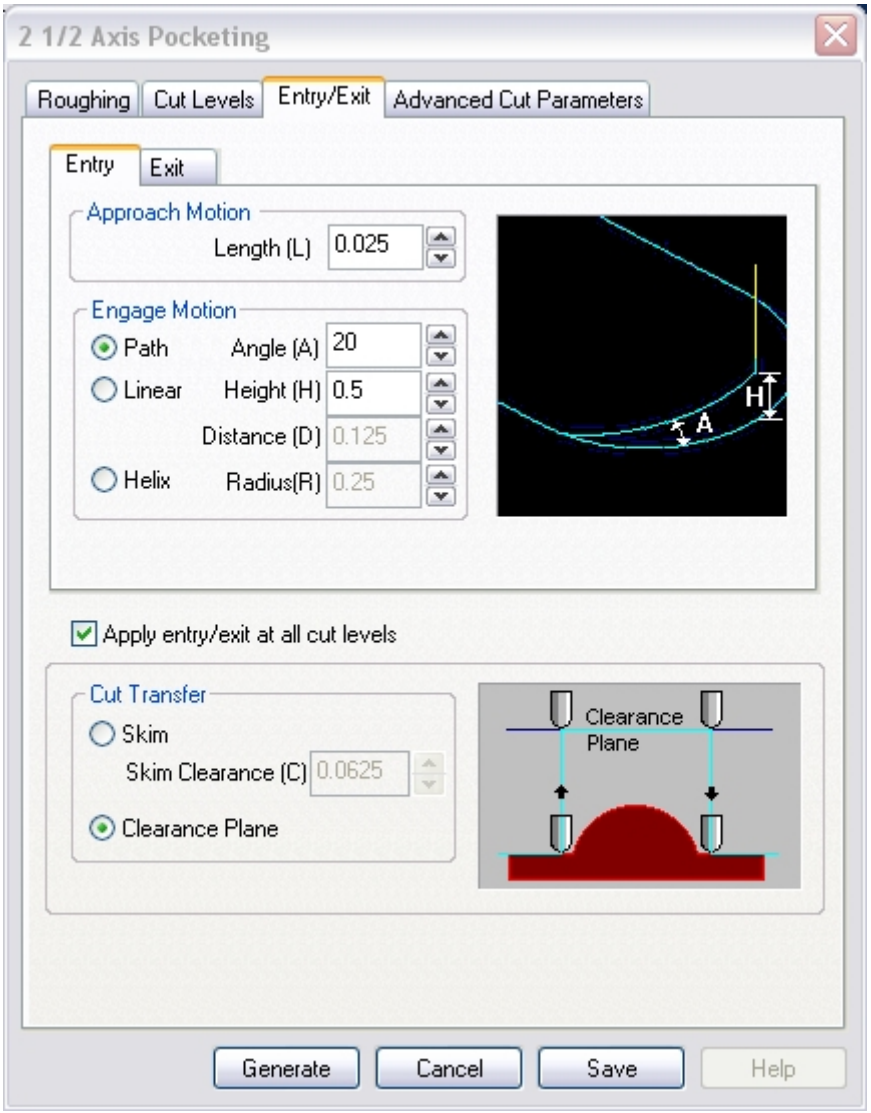
The clear islands will add a cut at the island levels (inner regions).

The use 3d Model to detect depth can be used when there is a 3d model available. This will detect the bottom most depth of the pocket (if there are multiple depths) and will add toolpath until that level. The finish depth can be specified when this option is used. The roughing depth is determined by the difference of the total depth and the finishing depth.

Cut levels can also be output as either a Depth first or as Level First.

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.



The Entry motion consists of Approach and Engage. 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 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.

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 he 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.

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.

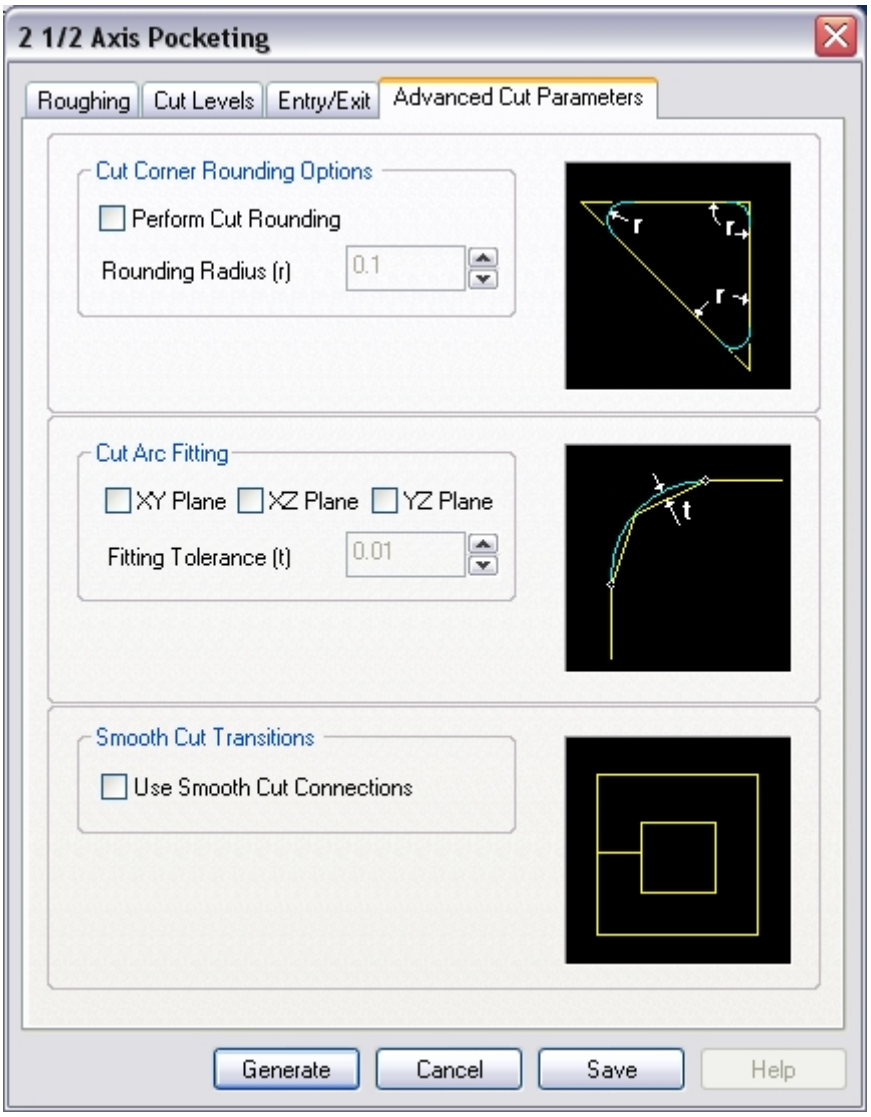
The user can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can either be instructed to move to the clearance plane and then perform the transfer motion to the next cut location or it could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

Advance Cut Parameters

The advanced cut parameters are 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 your machine during the cutting process. These parameters allow smoothing of the toolpaths by introduction of arcs. You can use these parameters even if your controller does not support arcs. Make sure that your output is set to linear output. You can do this under the Machining Preferences dialog in the Preferences section of the Menu bar.

- **Corner Rounding:** This option is used to round sharp corners in the toolpath. The user can specify a rounding radius and 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.

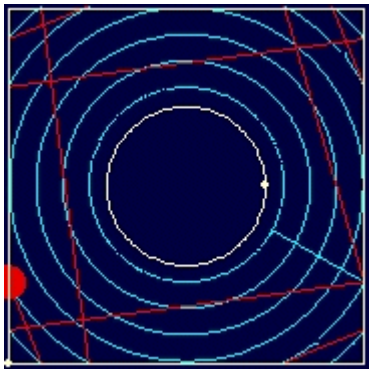
- **Smooth Cut Transitions:** This option can be used to introduce S shaped or C shaped cut transitions between two successive offset cuts. These transitions are introduced only in offsets that are generated in planes parallel to the XY plane. These transitions allow the cutter to transition from one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



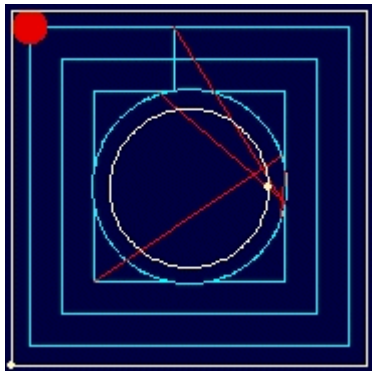
Differences between Facing and Pocket Machining

Facing and pocket machining are very similar methods of machining 2-D geometry. Both are area clear operations in that they both remove material over an area. However, the principal difference between these two operations is that the way the methods treat the outermost region of the selected region geometry. Facing considers all outermost regions as the boundaries of stock geometry, that is, the limits of the material to be machined away. All internal regions are considered as part geometry, that is, geometry that cannot be cut. Pocketing, on the other hand,

considers all regions as part geometry or geometry that cannot be cut. This difference in interpretation of the region geometry's purpose causes a significant difference in the tool's cut pattern during machining. The table below illustrates the cut pattern generated in both of these operations for identical region geometry.



Facing Toolpath



Pocketing Toolpath

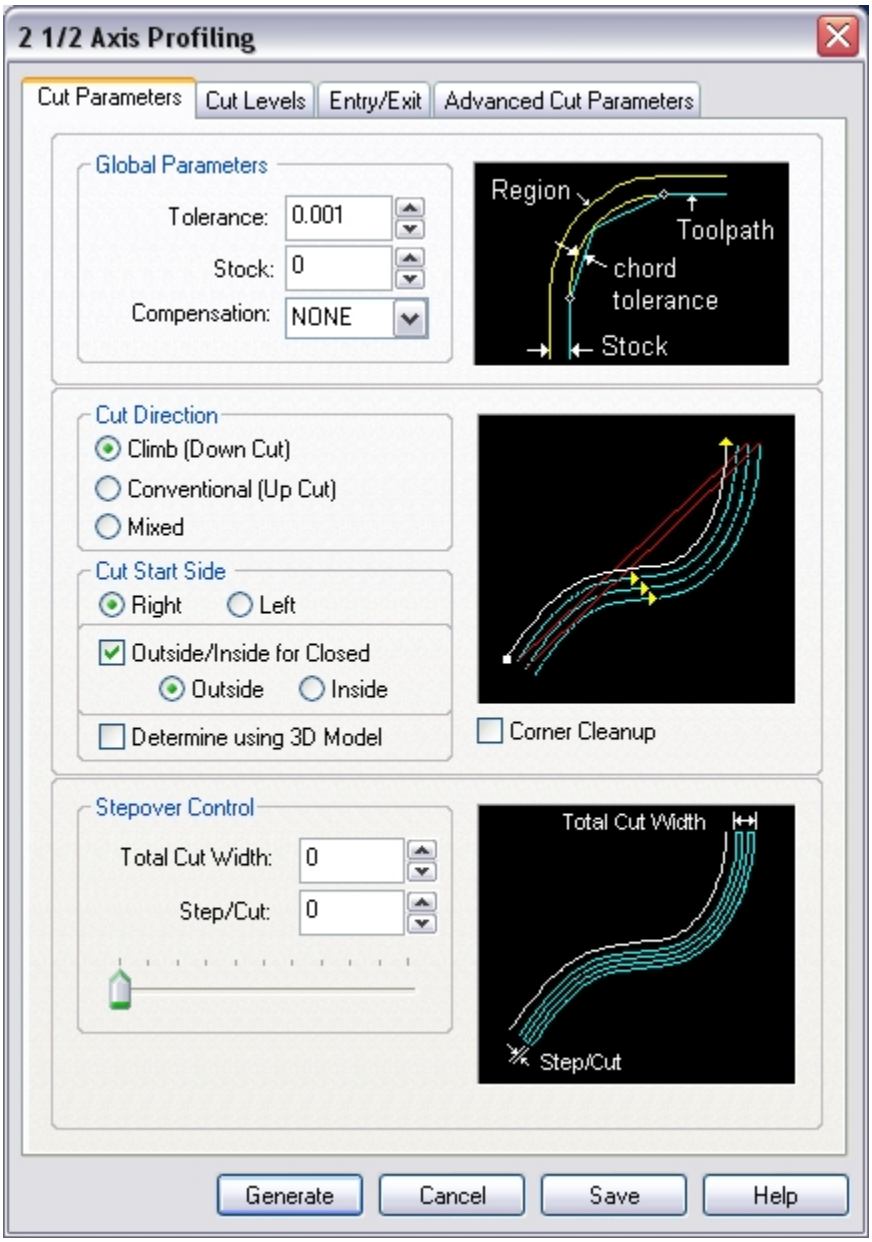
Notice that in the facing operation, the cutter (represented by the red filled circle) travels from the outside to the inside, treating the outside region as material to be cut away. In the pocketing operation, the cutter treats the outside region as a wall and stays tangent to it. Notice also the significant differences in the cut patterns for both of these operations.

Profiling

2½ Axis Profiling is a toolpath method, which employs regions and can be used either as a pre-finishing operation or as a finishing operation. 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. This method is similar to Horizontal Finishing except that the system does not consider the part geometry during computation. As in Horizontal Finishing, the cutter finishes in constant Z planes. 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 however, the direction of cutting is alternated between each parallel plane.

Description

The Pocket Machining toolpath is invoked by clicking on the  button in the MOps Browser and picking the **2½ Axis Milling** and **Profiling** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



This dialog is a tabbed dialog with four tabs. Clicking on each of the tabs allows the user to set the different parameters. The following sections describe each of the tabs in more detail.

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 l value to be used in machining. Also any uniform thickness or stock that needs to be left of the part also can be specified here.

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

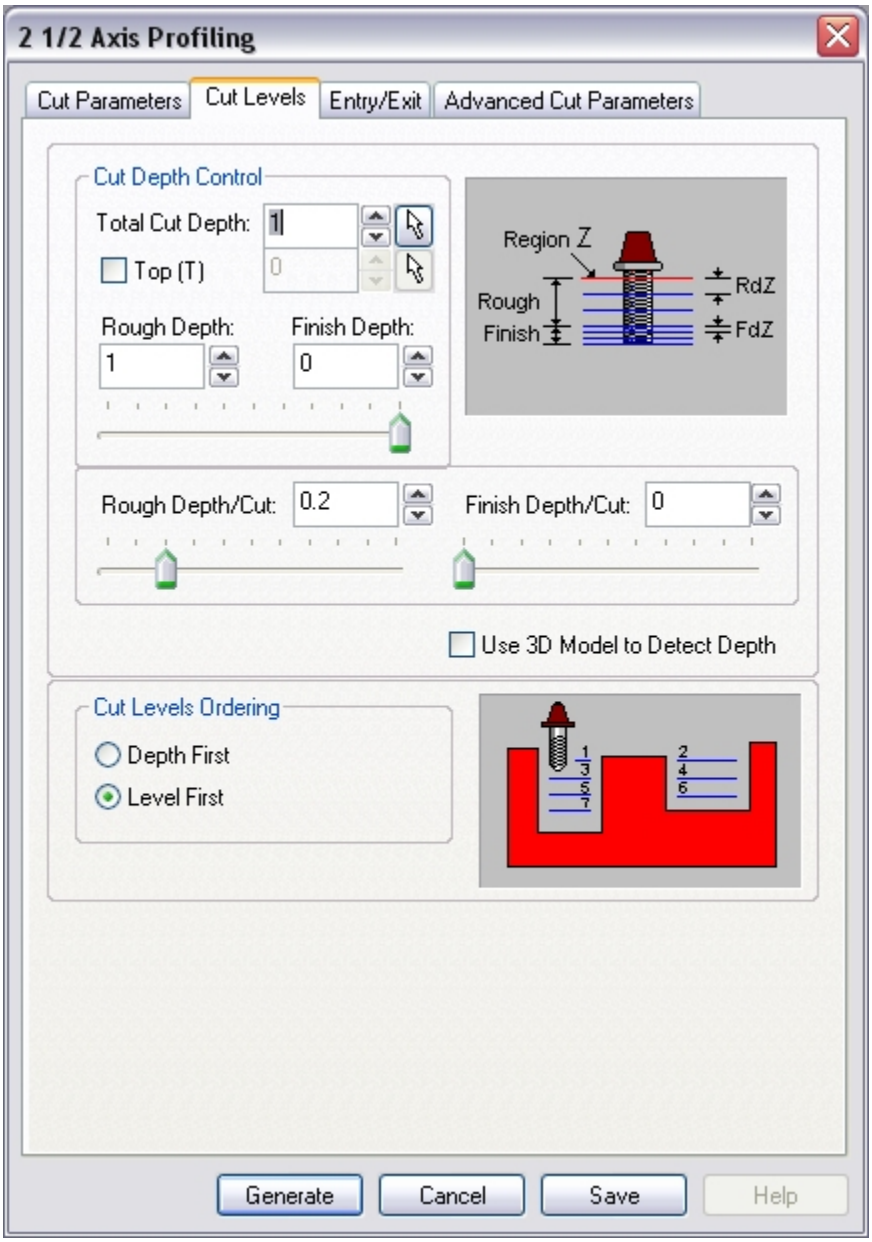
The **Cut Direction** allows the user to specify the Climb or Conventional or Mixed.

The **cut side** determines the side of the curve to be cut. For example, for a clockwise circle, the right will be inside the circle and the left will be outside the circle. For closed sketch(s) the user can set the cut start side to outside or inside. If there a 3d model available, the Determine using the 3d model can be used to automatically figure out the start side.

The **Stepover Control** 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 typing in the value or using the sliding bar.

Cut Levels

When the user selects this tab in the dialog the following property page is displayed. The user has the ability to set the Stepdown Control as well as specify the Cut Levels in this page.



The user can set the top Z level by enabling the check box titled Top and specifying the top level. If the top level is not specified than the top of the outer most region is selected as the top Z level. The cut depth can be specified either by typing in the value or by selecting the pick button and selecting the bottom of the pocket. A depth of 0.0 will give one pass at the top Z level. The cuts can be specified as rough cuts and finish cuts, by sliding the bar or by typing in the values. Similarly for the rough and finish depth of cuts.

The clear islands will add a cut at the island levels (inner regions).

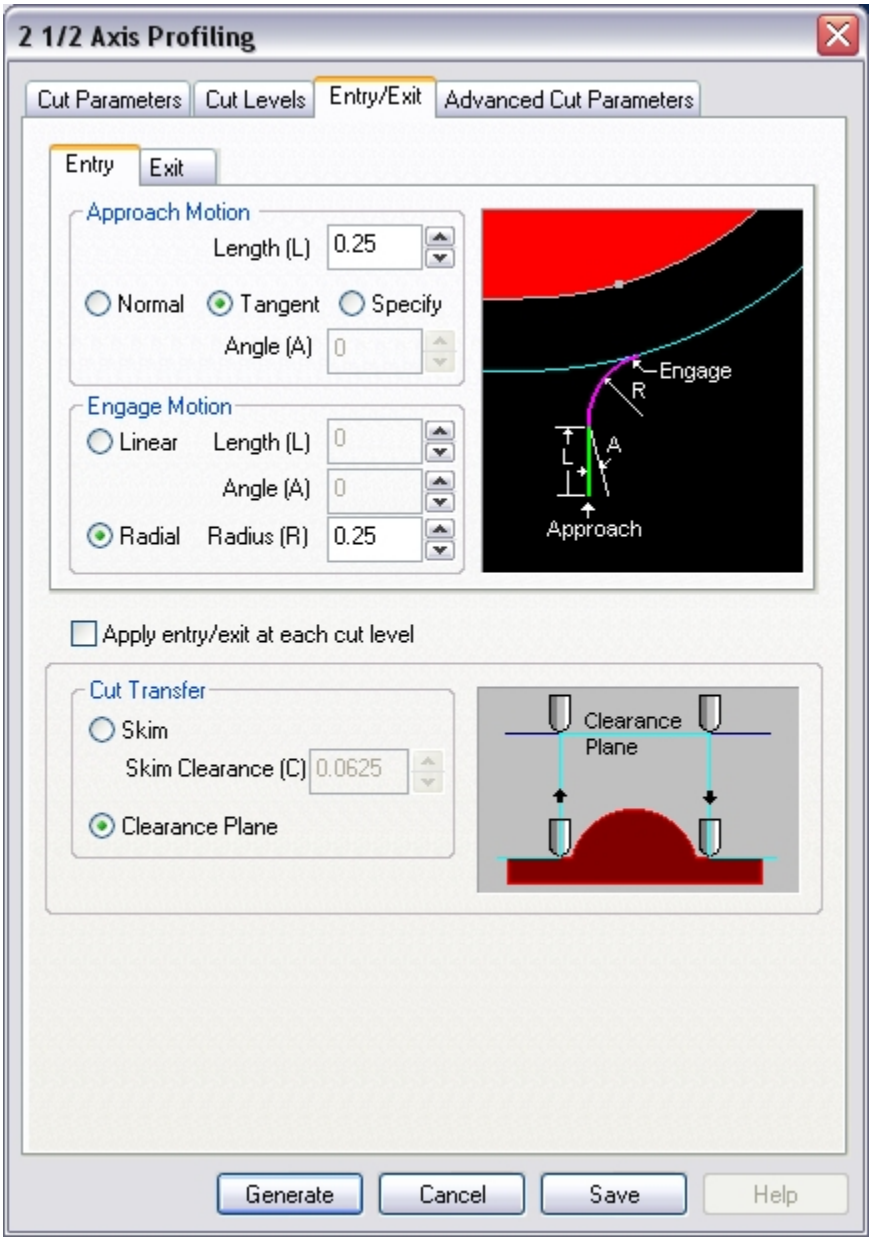
The use 3D Model to detect depth can be used when there is a 3D model available. This will detect the bottom most depth of the pocket (if there are multiple depths) and will add toolpath until that

level. The finish depth can be specified when this option is used. The roughing depth is determined by the difference of the total depth and the finishing depth.

Cut levels can also be output as either a Depth first or as Level First.

Entry / Exit

When the user selects the Entry/Exit tab the following property page is displayed. The user will be able to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut. The user can also specify the type of transfer motions to perform while cutting.



The Entry motion consists of Approach and Engage. 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 approach can be either Tangential or Normal or at any angle to the Engage motion. This is followed by the engage motion which can be a radial engage or a linear engage at an angle.

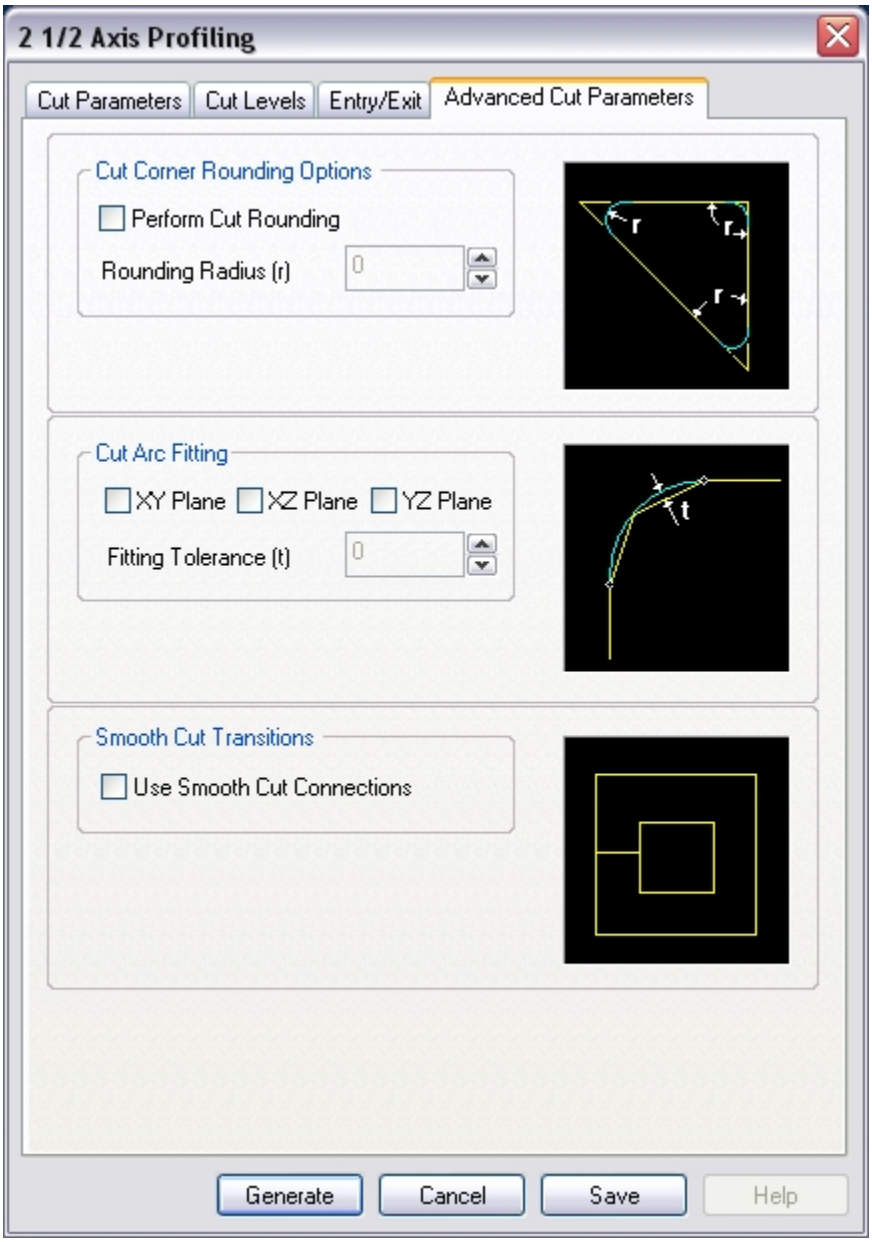
Similarly the Exit motion consists of a Retract motion followed by a departure motion. The retract motion can be either a radial (arc) motion or a linear motion at an angle. The departure motion can be either Tangential or Normal or at any angle to the Retract motion.

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 regions and using this Z value as the height to perform the transfer motions.

Advance Cut Parameters

The advanced cut parameters are 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 you machine during the cutting process. These parameters allow smoothing of the toolpaths by introduction of arcs. You can use these parameters even if your controller does not support arcs. Make sure that your output is set to linear output. You can do this under the Machining Preferences dialog in the Preferences section of the Menu bar.

- **Corner Rounding:** This option is used to round sharp corners in the toolpath. The user can specify a rounding radius and 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 other in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.



Hole Pocketing

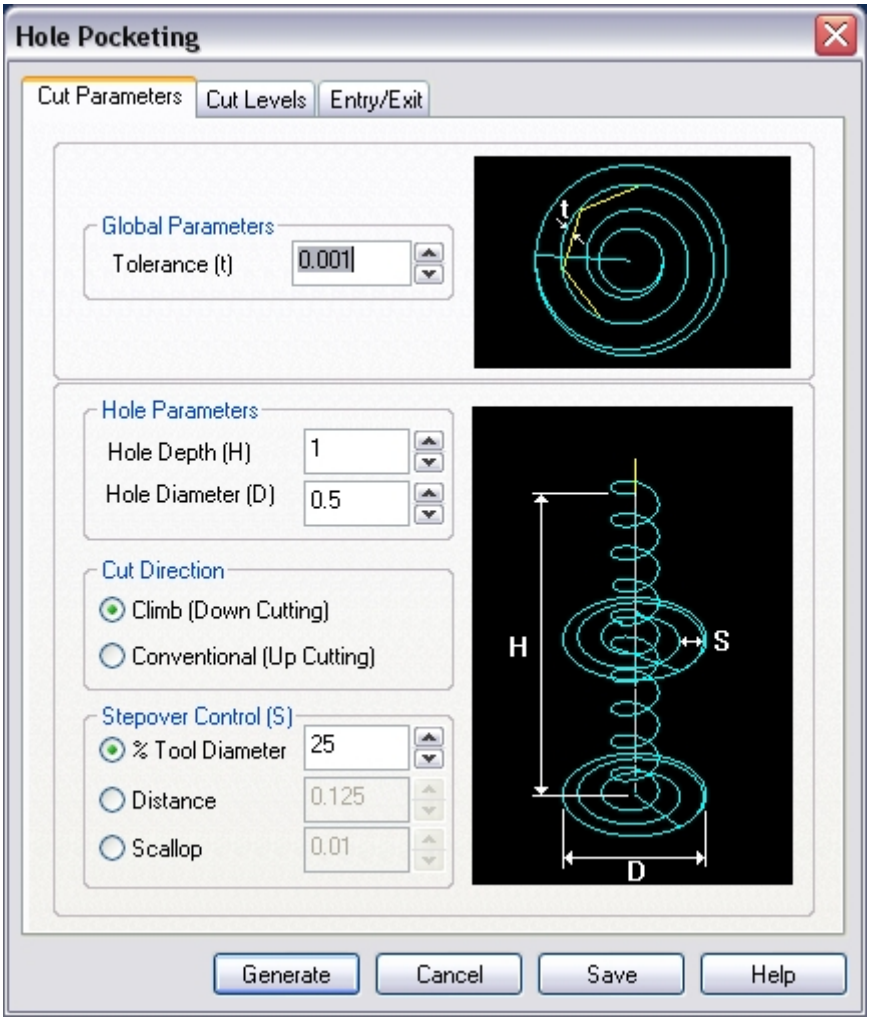
Available in Alibre CAM Standard, Expert and Professional versions only

This operation can be used to cut big holes as a milling operation instead of drilling. The hole radius and depth can be specified, along with the number of levels. The Engage can be specified as a helix with the height and angle or pitch. For machines capable of helix cycles, the output can be a helix cycle. For others, it can be a series of linear moves. The option to output either linear motions for the helix or a true helix cycle can be set in the Preferences->Machining Preferences. After the helix engage, the hole is cut until the outer diameter using a spiral motion, followed by a circular motion

to clean up the hole. The spiral output and the arc output can be controlled similar to the helix output, as cycles or linear movements, using Preferences->Machining Preferences

Description

The 21/2 Axis Hole Pocketing toolpath method is invoked by clicking on this button in the MOps Browser and picking the **2½ Axis Milling** and **Hole Pocketing** option. This section describes the various parameters that the user can set to execute this machining operation. The dialog that is invoked if the user chooses this toolpath method is shown below:



This dialog is a tabbed dialog with three tabs. Clicking on each of the tabs allows the user to set the different parameters. The following sections describe each of the tabs in more detail.

Cut Parameters

Tolerance is the allowable deviations from the actual part geometry

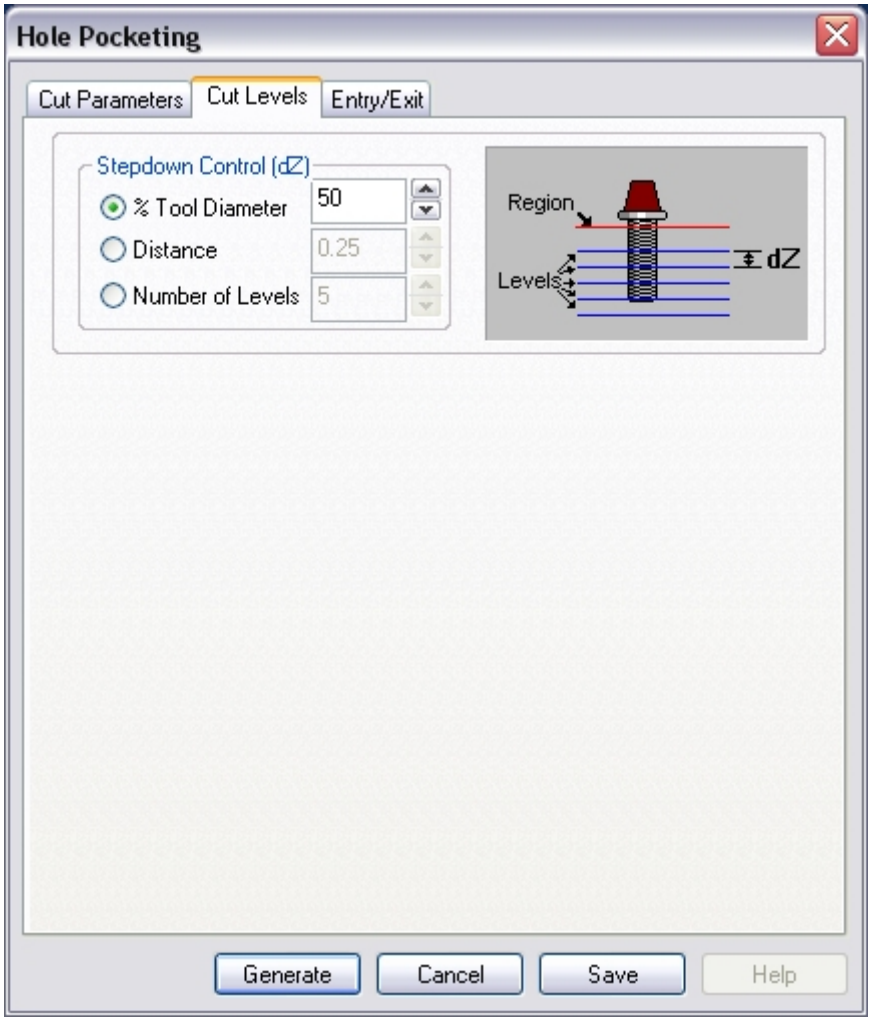
The Hole Depth and the Diameter can be specified for the Hole using the spaces for the input in the dialog box.

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

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 tool diameter or a 2d scallop height.

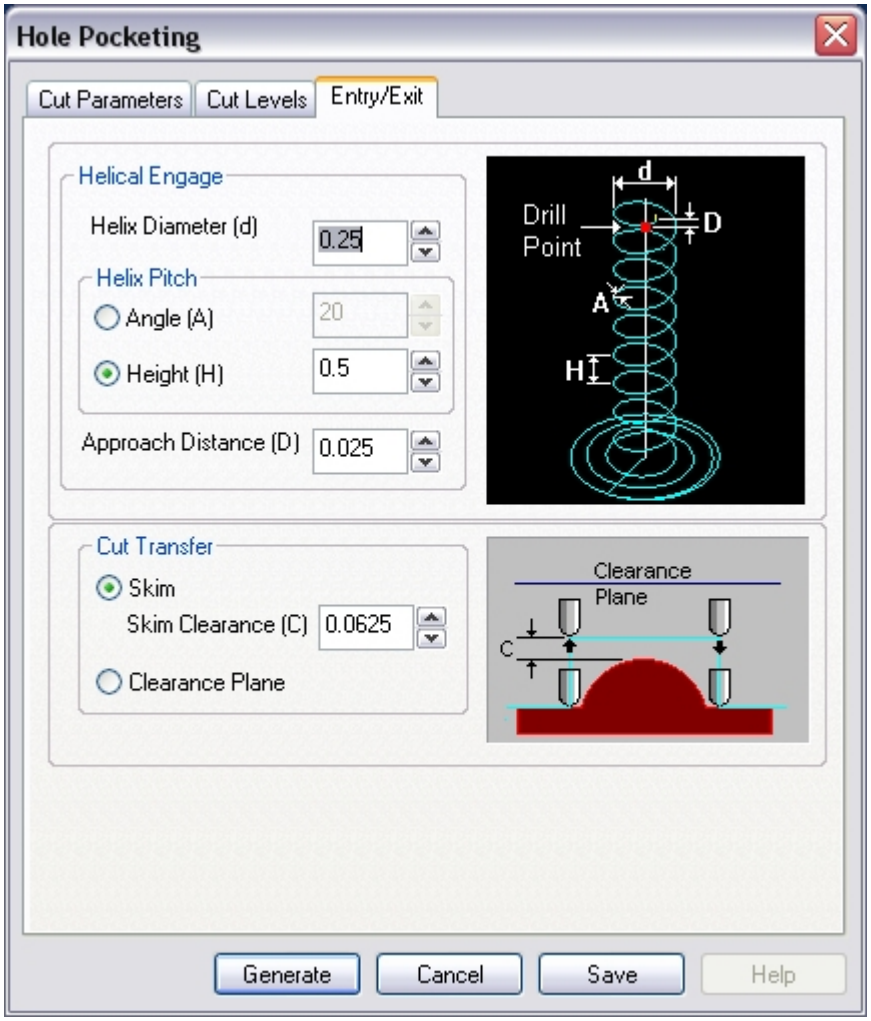
Cut Levels

When the user selects this tab in the dialog the following property page is displayed. The user has the ability to set the Stepdown Control as well as specify the Cut Levels in this page. The Step down control can be a percentage of the tool diameter or a distance or number of levels.



Entry/Exit

When the user selects the Entry/Exit tab the following property page is displayed. The user will be able to specify how the cutter approaches and engages during the cutting. The user can also specify the type of transfer motions to perform while cutting. 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.



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 regions and using this Z value as the height to perform the transfer motions.

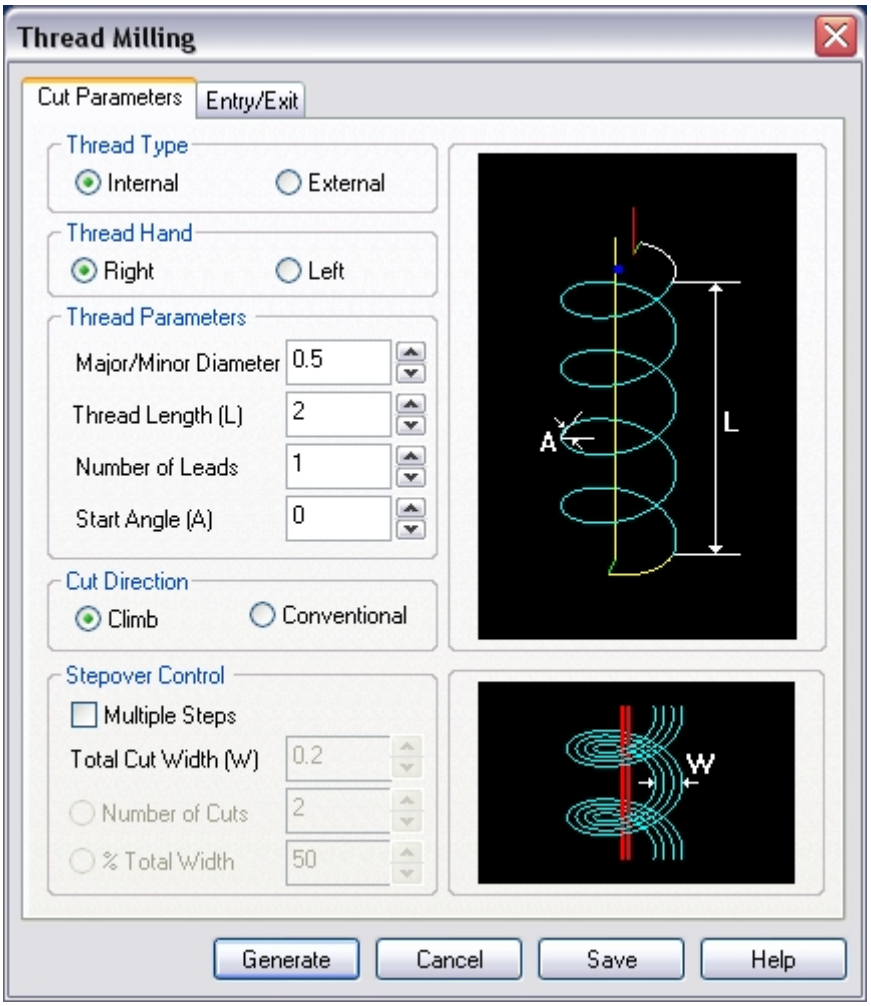
Thread Milling

Available in Alibre CAM Standard, Expert and Professional versions only

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

Description

The 21/2 Axis Thread Milling toolpath method is invoked by clicking on this  button in the MOps Browser and picking the **2½ Axis Milling** and **Thread Milling** option. This section describes the various parameters that the user can set to execute this machining operation. The dialog that is invoked if the user chooses this toolpath method is shown below:



This dialog is a tabbed dialog with two tabs. Clicking on each of the tabs allows the user to set the different parameters. The following sections describe each of the tabs in more detail.

Cut Parameters

The user has the ability to select either internal or external threads and also either left or right hand threads.

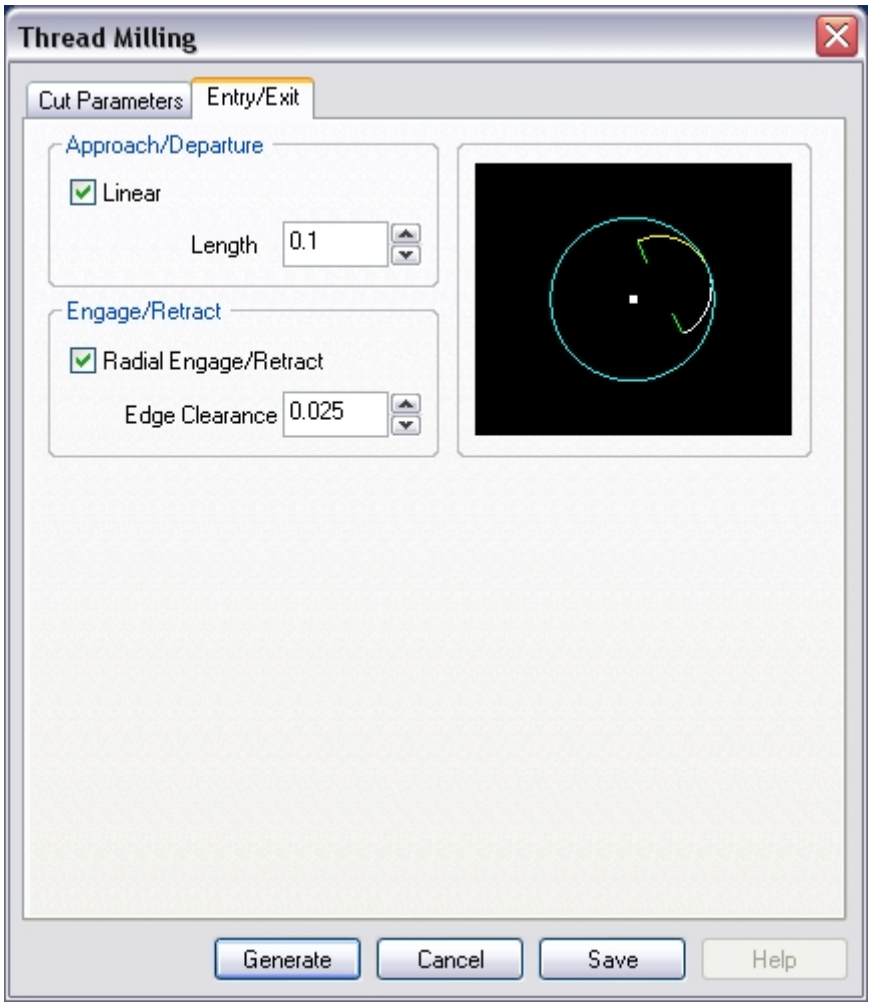
Using the Thread parameters section, the user can set the diameter of the thread, the thread length, number of lead and the start angle.

The cut direction can either be climb or conventional

The threads can be cut either in one pass or multiple passes. The Total cut width can be specified and it can be either a percentage of the total width or number of cuts.

Entry/Exit

When 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.



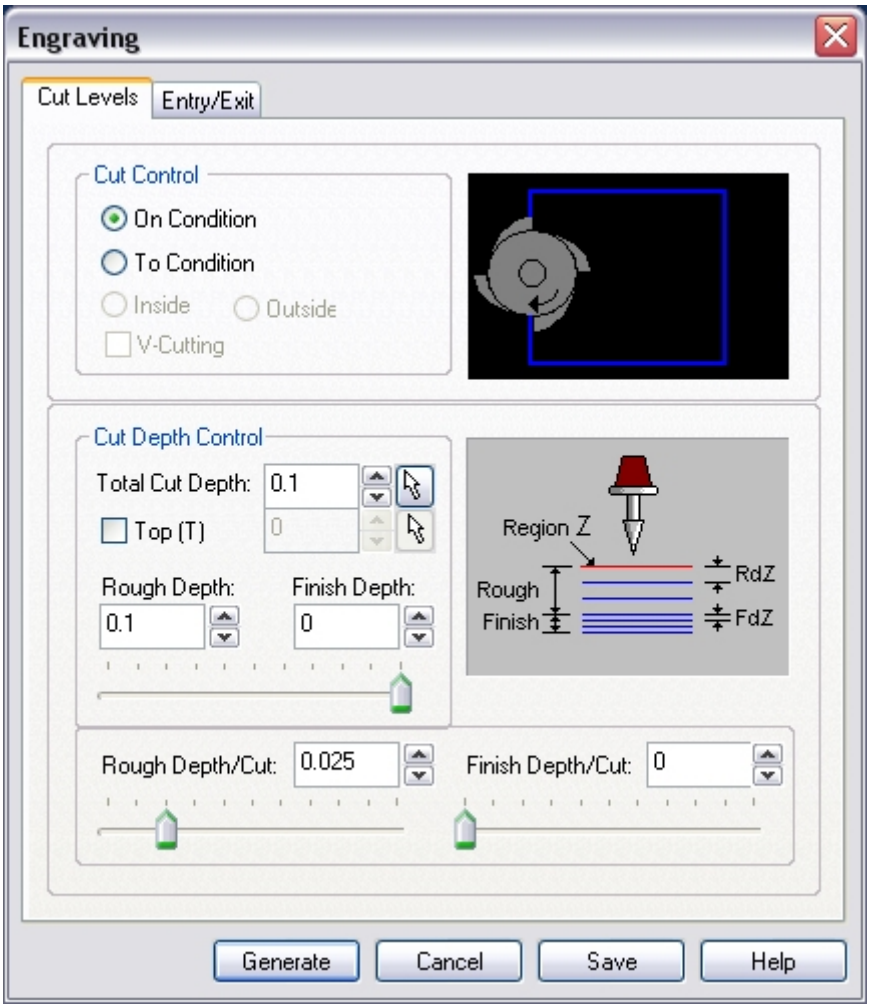
Engraving

Available in Alibre CAM Standard, Expert and Professional versions only

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. There are two ways to do the engraving operation. The on-condition engraving, in which the tool center follows the 3d curve exactly and the To condition in which the V-carving tool approaches from either left or right and to carve either inside or outside of the letters, logos etc. For the To-condition to work the regions must be closed regions. The regions are flattened to 2D regions and the toolpath is created. The corners of the regions between two passes are automatically figured out and are cleaned up using a 3D clean up pass.

Description

The Engraving toolpath is invoked by clicking on the button in the MOps Browser and picking the **2½ Axis Milling** and **Engraving** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the engraving toolpath operation. This dialog box is shown below.



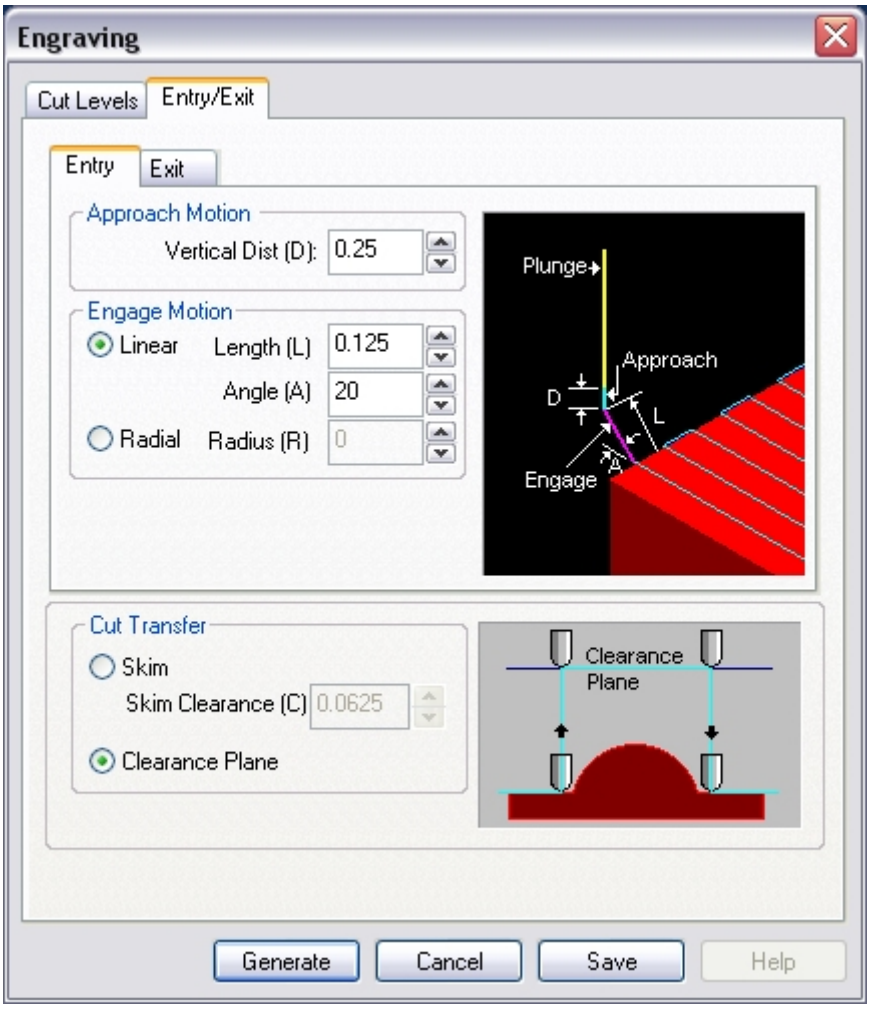
Cut Levels

The user has the ability to define multiple roughing passes as well as multiple finish passes. The On condition drives the center of the tool along the curve. The To condition is for V-carving where the tool can come from either to the left or right of the curve.

The user can set the top Z level by enabling the check box titled Top and specifying the top level. If the top level is not specified than the top of the outer most region is selected as the top Z level. The cut depth can be specified either by typing in the value or by selecting the pick button and selecting the bottom of the pocket. A depth of 0.0 will give one pass at the top Z level. The cuts can be specified as rough cuts and finish cuts, by sliding the bar or by typing in the values. Similarly for the rough and finish depth of cuts.

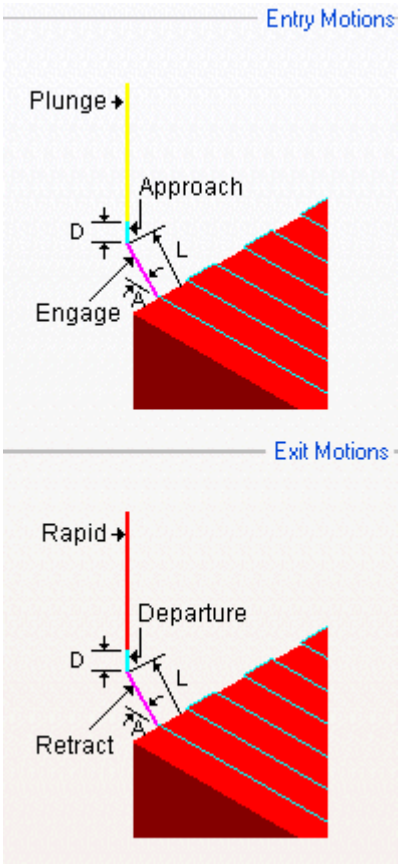
Entry / Exit

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.

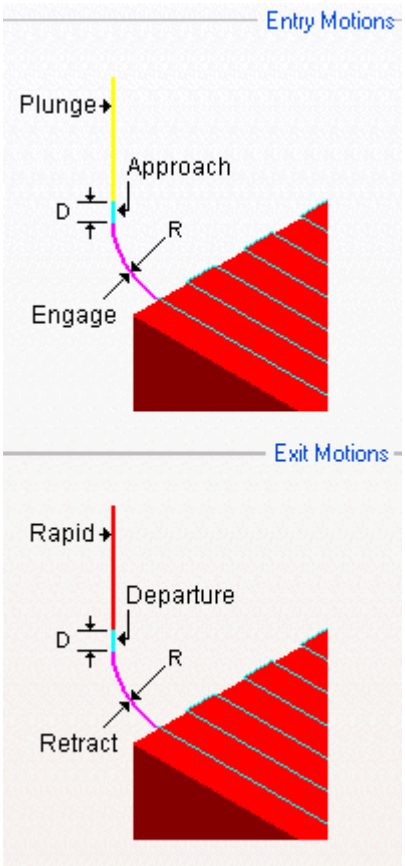


The Cut Entry section can be used to control how the cutter enters during the cutting process. The user can choose to approach vertically. The user can also specify the engaging as a ramp motion with the defined angle and distance away from the cut start point to engage or as a radial engage. The ramp angle is measured from the horizontal. In the radial engage 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. Similarly for the Retract and Depart motions.

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.



Linear Motion



Radial Motion

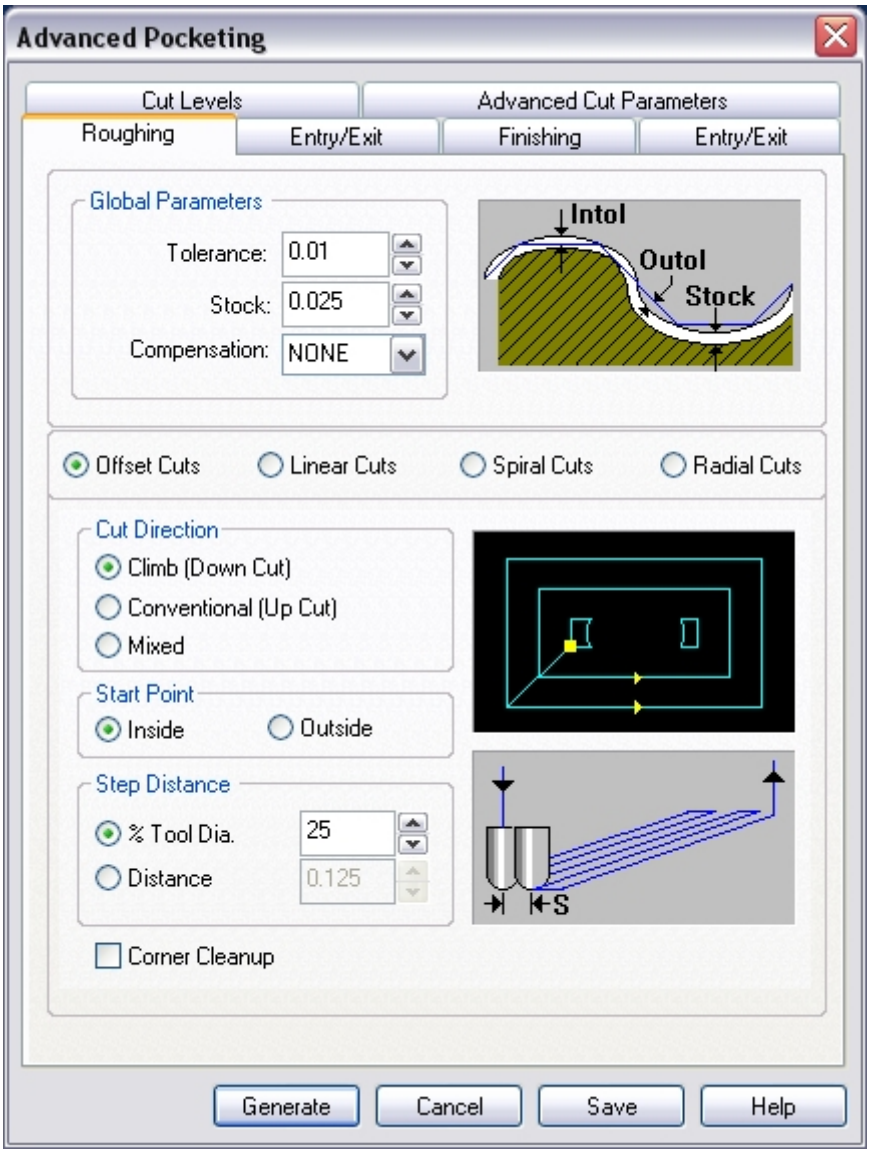
Advanced Pocketing

Available in Alibre CAM Expert only.

This machining operation encompasses the functionality of both pocketing and profiling operations. This allows the user to do roughing (pocketing) and finishing (profiling) in one operation and allows a greater degree of control. The stock left in the roughing can be cleaned up by the finishing operation, without the user having to know how much to cut in the finishing pass, since the stock to cut for finishing is figured out by the difference in the stock variable in roughing and the stock variable in finishing. Also, the finishing can be done after the roughing at each level, or they can be done after all the roughing levels are done. Additionally, finishing can be done only for the bottom most level only. The Engage/Retracts can be specified for roughing and finishing separately.

Description

The Advanced Pocket Machining toolpath is invoked by clicking on the button in the MOps Browser and picking the **2½ Axis Milling** and **Advanced Pocketing** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



This dialog is tabbed with six tabs. Each tab defines a set of parameters that the user can specify. The sections below describe them in detail.

Roughing

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

The Global Cut Parameters section allows the user to set the tolerance value to be used in machining. Also any uniform thickness or stock that needs to be left of the part also can be specified here.

Stock: The thickness of the layer that will remain on sides 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.

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

Compensation stands for cutter compensation. The user can turn this on by selecting from the drop down menu. The cutter compensation to the left or right is determined by the Climb or Conventional direction.

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 a linear cut pattern where the tool will traverse in a linear cuts always, or an offset cut pattern where the tool will traverse in successive uniform offsets of the part shape, or a spiral cut pattern which can go from inside out or outside in, or a radial cut pattern.

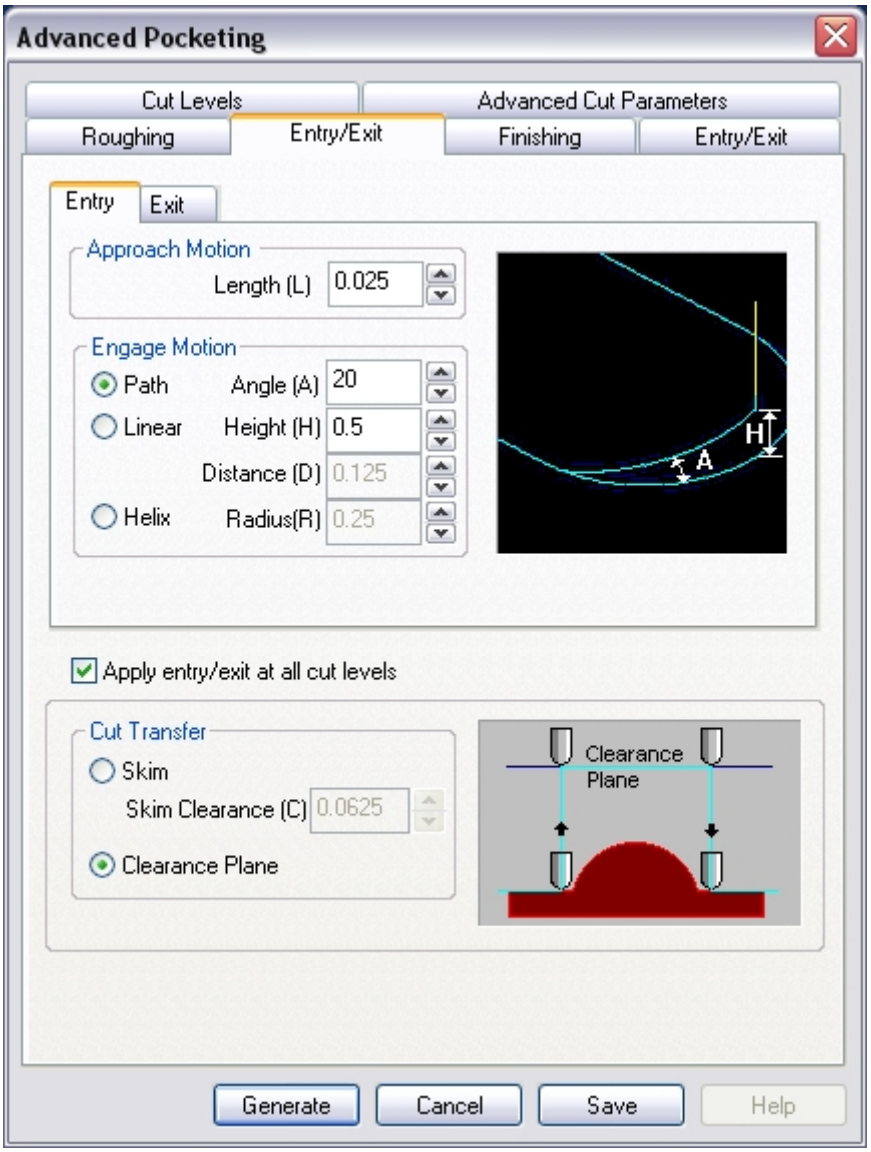
The **Cut Direction** can be specified as Climb, Conventional or Mixed.

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

The **corner cleanup** will automatically detect all the corners where the tool couldn't get in, in between each pass and will add a cut based on the uncut area, either linear cut in case of a smaller area or will travel along the shape of the uncut area, when the area is large.

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.



The Entry motion consists of Approach and Engage. 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 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.

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 he 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.

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 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 regions and using this Z value as the height to perform the transfer motions.

Finishing

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

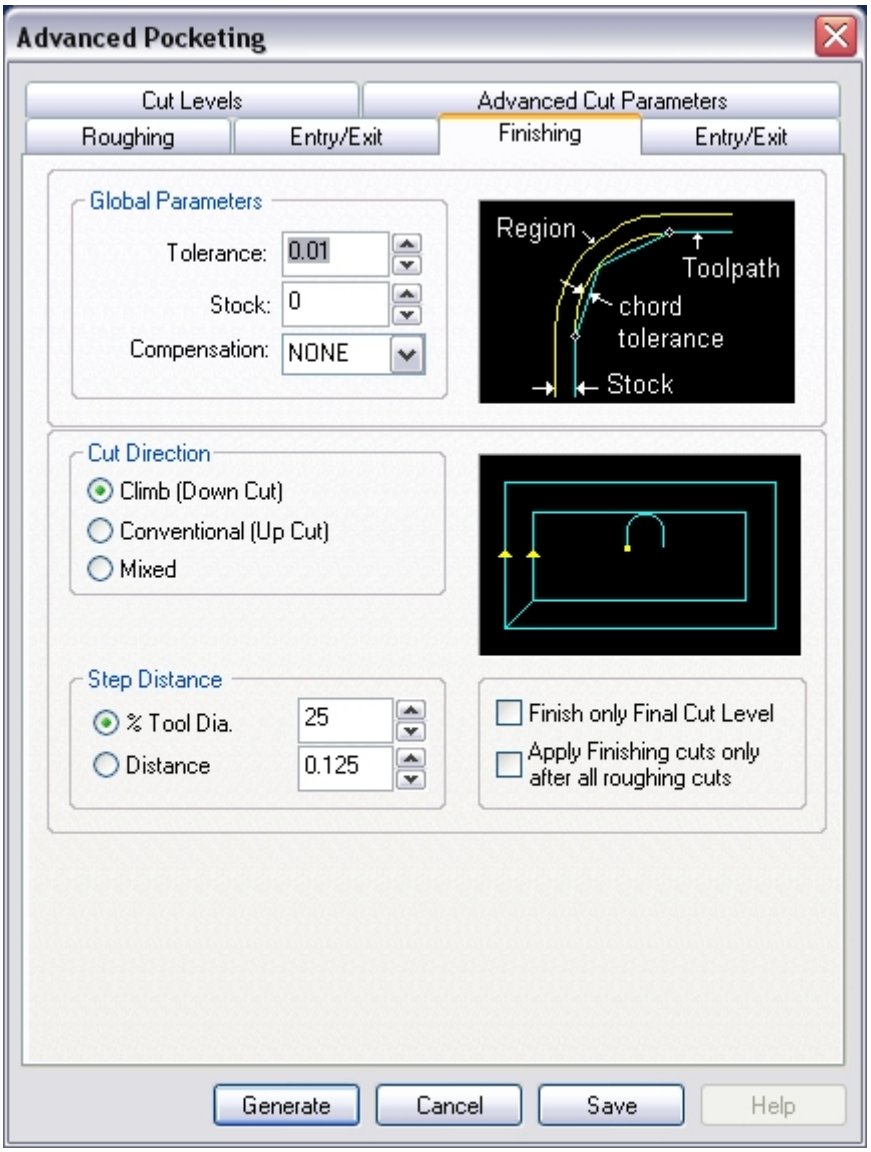
The Global Cut Parameters section allows the user to set the tolerance l value to be used in machining. Also any uniform thickness or stock that needs to be left of the part also can be specified here.

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

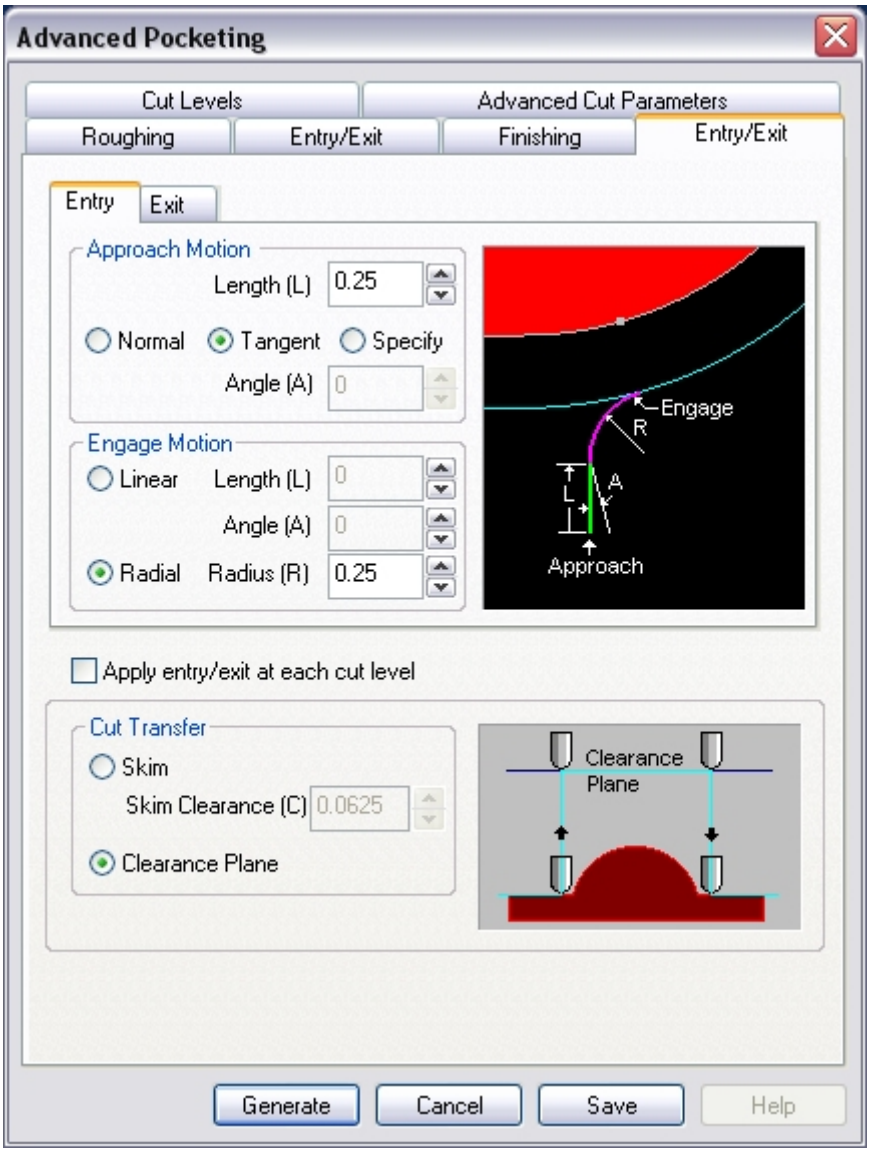
The cut side determines the side of the curve to be cut. For example, for a clockwise circle, the right will be inside the circle and the left will be outside the circle. If there a 3d model available, the Determine using the 3d model can be used to automatically figure out the start side.

These Finish levels can be applied only to the final cut level. Also, the finishing levels can be applied after all the roughing cut levels.



Finishing Entry/Exit

Then the user selects the Entry/Exit tab the following property page is displayed. The user will be able to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut. The user can also specify the type of transfer motions to perform while cutting.



The Entry motion consists of Approach and Engage. 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 approach can be either Tangential or Normal or at any angle to the Engage motion. This is followed by the engage motion which can be a radial engage or a linear engage at an angle.

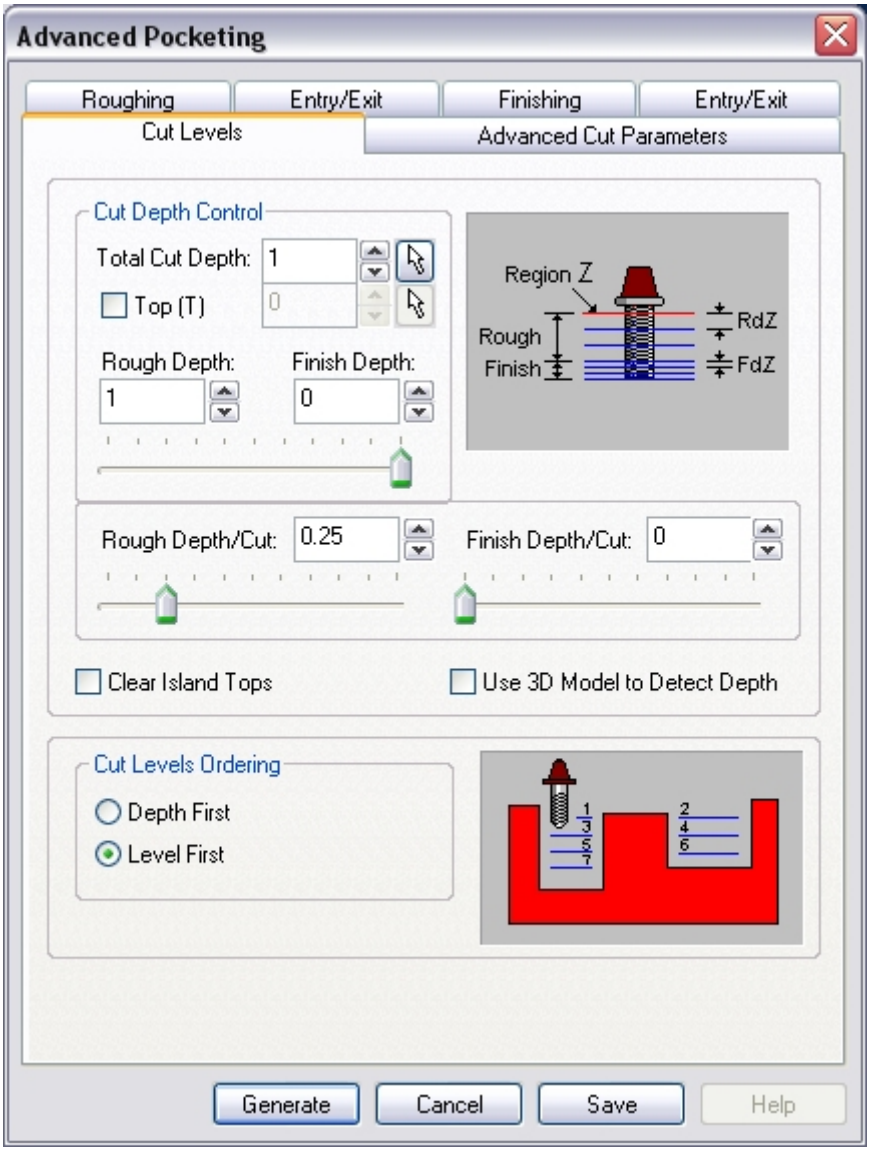
Similarly the Exit motion consists of a Retract motion followed by a departure motion. The retract motion can be either a radial (arc) motion or a linear motion at an angle. The departure motion can be either Tangential or Normal or at any angle to the Retract motion.

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 regions and using this Z value as the height to perform the transfer motions.

Cut Levels

When the user selects this tab in the dialog the following property page is displayed.



The user can set the top Z level by enabling the check box titled Top and specifying the top level. If the top level is not specified than the top of the outer most region is selected as the top Z level. The cut depth can be specified either by typing in the value or by selecting the pick button and selecting the bottom of the pocket. A depth of 0.0 will give one pass at the top Z level. The cuts can be specified as rough cuts and finish cuts, by sliding the bar or by typing in the values. Similarly for the rough and finish depth of cuts.

The clear islands will add a cut at the island levels (inner regions).

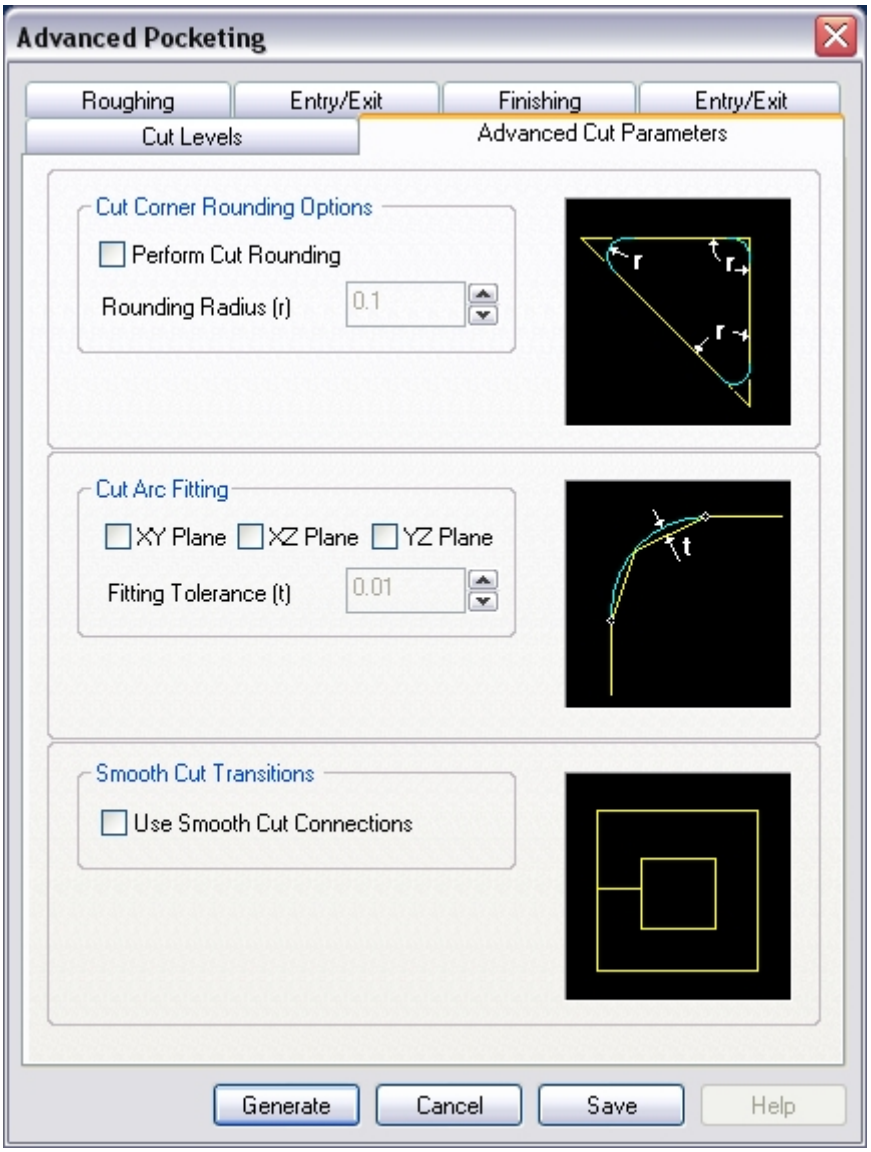
The use 3d Model to detect depth can be used when there is a 3d model available. This will detect the bottom most depth of the pocket (if there are multiple depths) and will add toolpath until that level. The finish depth can be specified when this option is used. The roughing depth is determined by the difference of the total depth and the finishing depth.

Cut levels can also be output as either a Depth first or as Level First.

Advanced Cut Parameters

The advanced cut parameters are 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 you machine during the cutting process. These parameters allow smoothing of the toolpaths by introduction of arcs. You can use these parameters even if your controller does not support arcs. Make sure that your output is set to linear output. You can do this under the Machining Preferences dialog in the Preferences section of the Menu bar.

- **Corner Rounding:** This option is used to round sharp corners in the toolpath. The user can specify a rounding radius and 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 other in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.



Advanced Profiling

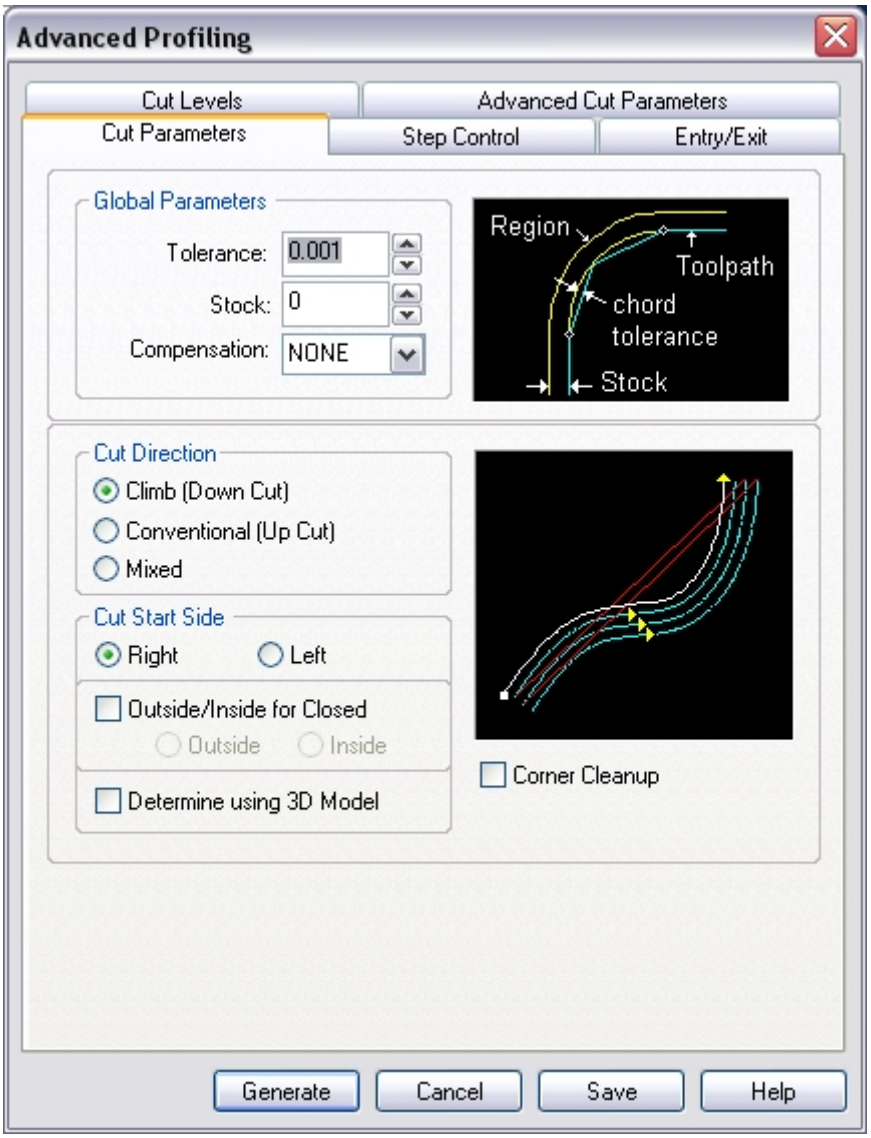
Available in Alibre CAM Expert only.

This operation can be used when multiple profiling passes are needed at varying widths with different step over distances. The width can be divided into roughing (larger step over) and finishing (smaller step over) passes. Similar to advanced pocketing, the finishing passes can be done after roughing at each level, or they can be done after all the roughing levels are done. Additionally, finishing can be done only for the bottom most level only.

Description

The Advanced Profile Machining toolpath is invoked by clicking on the button in the MOps Browser and picking the **2½ Axis Milling** and **Advanced Profile** option. The toolpath generated

depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



This dialog is a tabbed dialog with five tabs. Clicking on each of the tabs allows the user to set the different parameters. The following sections describe each of the tabs in more detail.

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 l value to be used in machining. Also any uniform thickness or stock that needs to be left of the part also can be specified here.

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

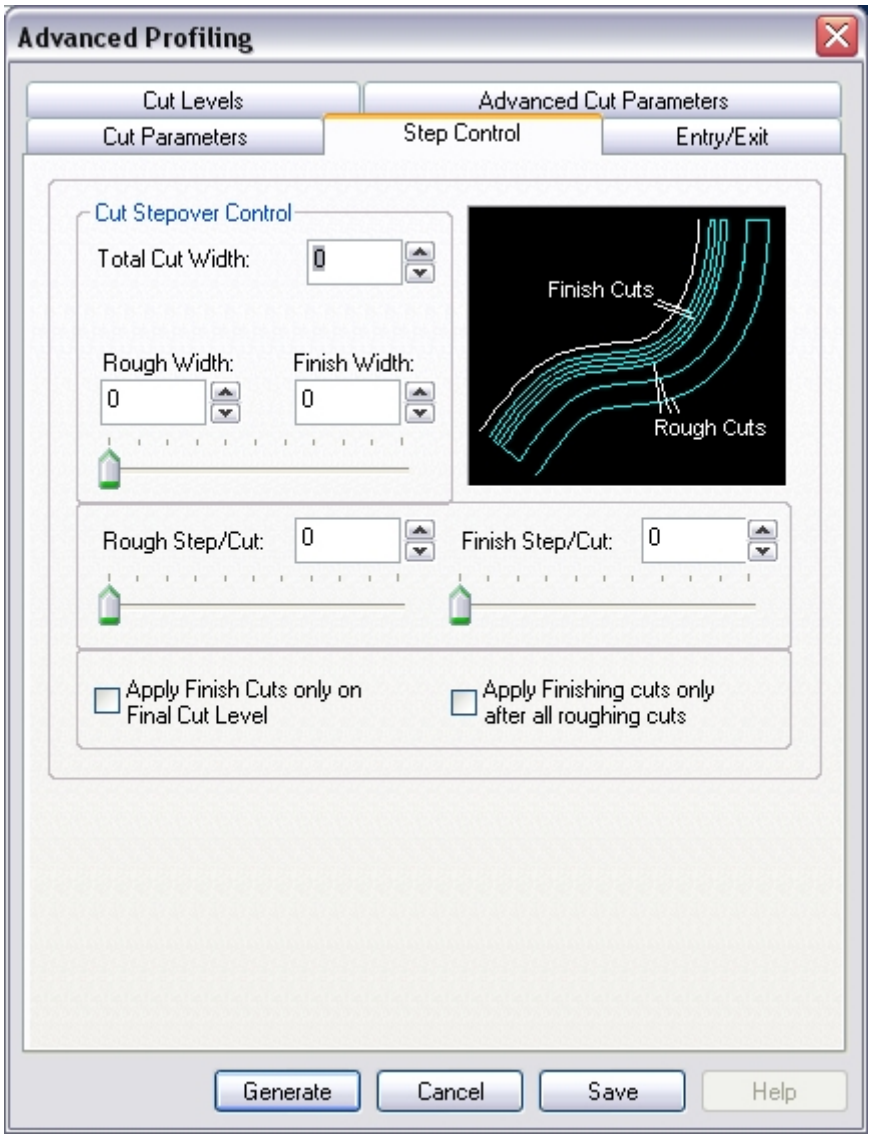
The cut side determines the side of the curve to be cut. For example, for a clockwise circle, the right will be inside the circle and the left will be outside the circle. If there a 3d model available, the Determine using the 3d model can be used to automatically figure out the start side. Alternatively to control the cut side for closed curves the button for Outside/Inside for Closed can be checked and the desired radio button for Outside or Inside can be chosen. This setting will then be used for all closed curves while the Right or Left settings will be used for all open ones.

Step Control

The Stepover Control 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 typing in the value or using the sliding bar. This can be divided into Rough cut and Finish cuts. The rough step per cut and the finish step per cut can be specified.

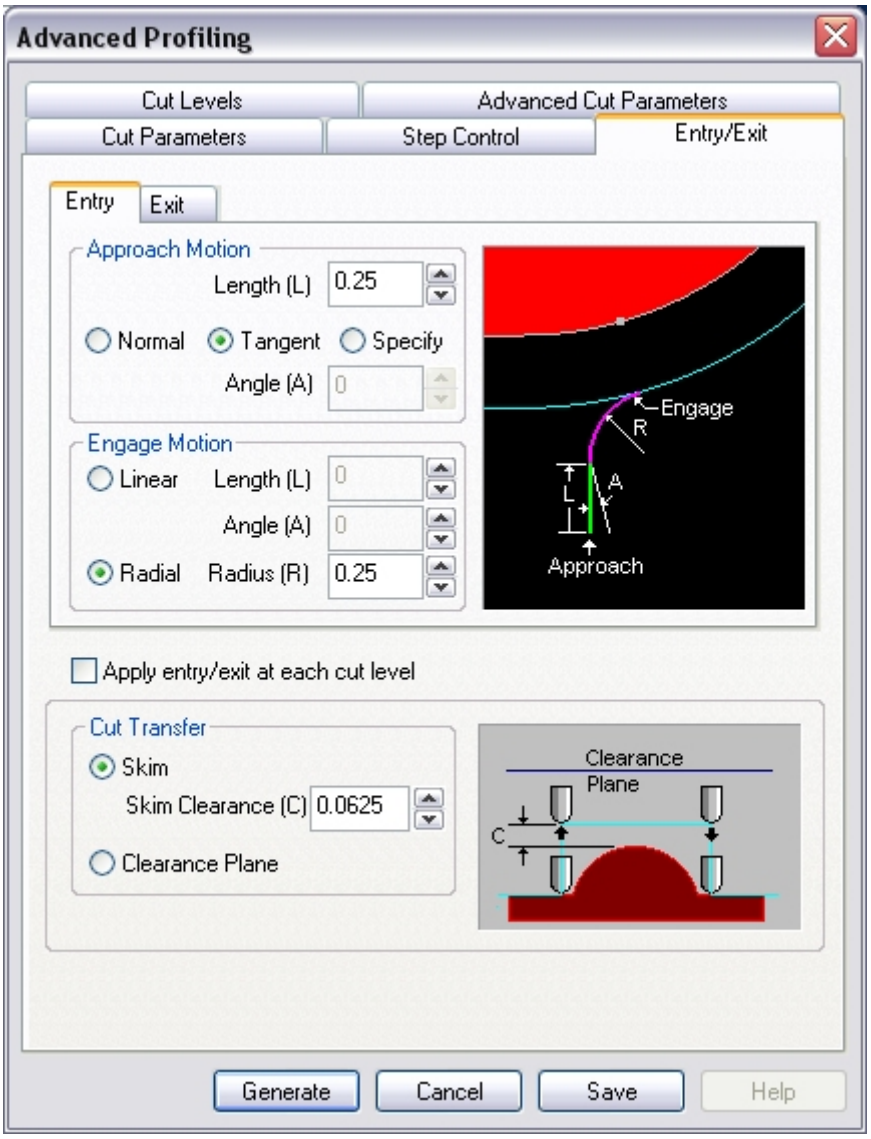
The finish cuts can be applied only at the final level by selecting the Apply Finish Cuts only on Final Cut Level

Additionally, the Finishing cuts can be applied after all roughing cuts by selecting the Apply Finish cuts after rough cuts.



Entry/Exit

Then the user selects the Entry/Exit tab the following property page is displayed. The user will be able to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut. The user can also specify the type of transfer motions to perform while cutting.



The Entry motion consists of Approach and Engage. 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 approach can be either Tangential or Normal or at any angle to the Engage motion. This is followed by the engage motion which can be a radial engage or a linear engage at an angle.

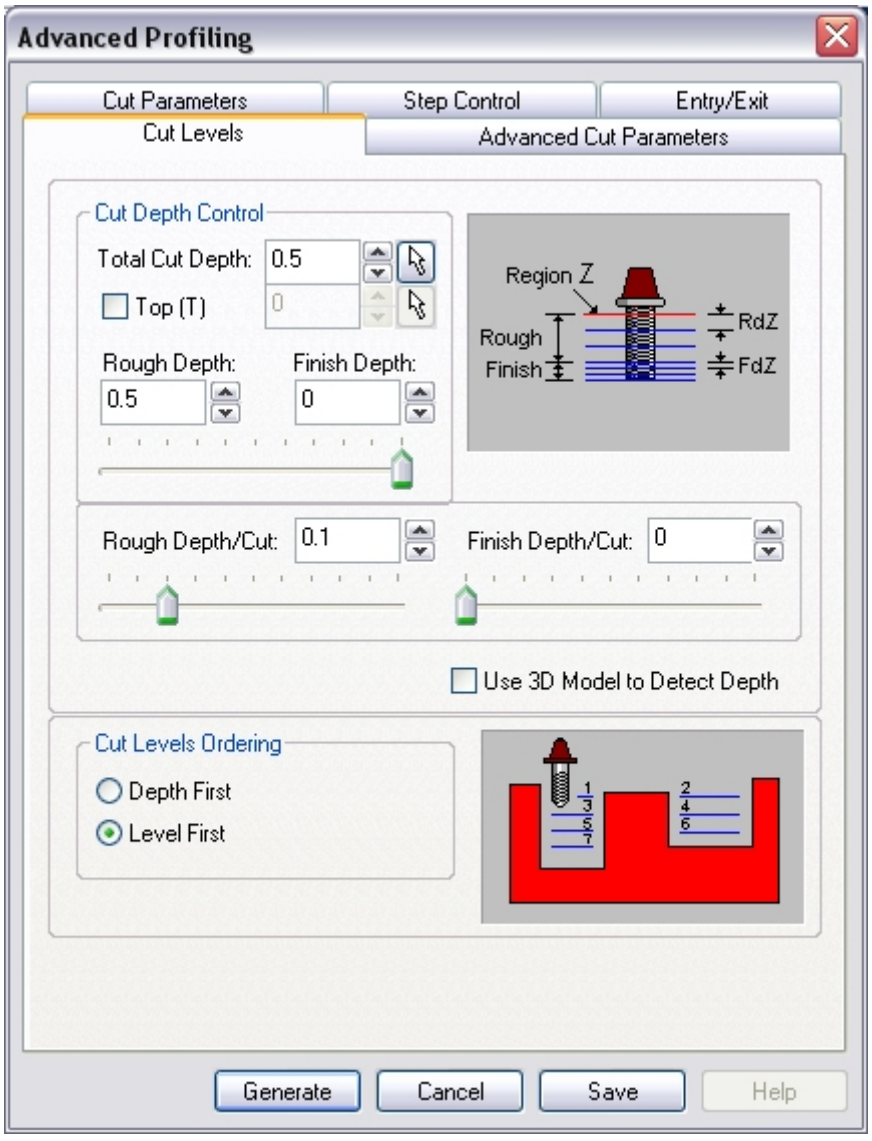
Similarly the Exit motion consists of a Retract motion followed by a departure motion. The retract motion can be either a radial (arc) motion or a linear motion at an angle. The departure motion can be either Tangential or Normal or at any angle to the Retract motion.

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 regions and using this Z value as the height to perform the transfer motions.

Cut Levels

Then the user selects this tab in the dialog the following property page is displayed. The user has the ability to set the Stepdown Control as well as specify the Cut Levels in this page.



The user can set the top Z level by enabling the check box titled Top and specifying the top level. If the top level is not specified than the top of the outer most region is selected as the top Z level. The cut depth can be specified either by typing in the value or by selecting the pick button and selecting the bottom of the pocket. A depth of 0.0 will give one pass at the top Z level. The cuts can be specified as rough cuts and finish cuts, by sliding the bar or by typing in the values. Similarly for the rough and finish depth of cuts.

The clear islands will add a cut at the island levels (inner regions).

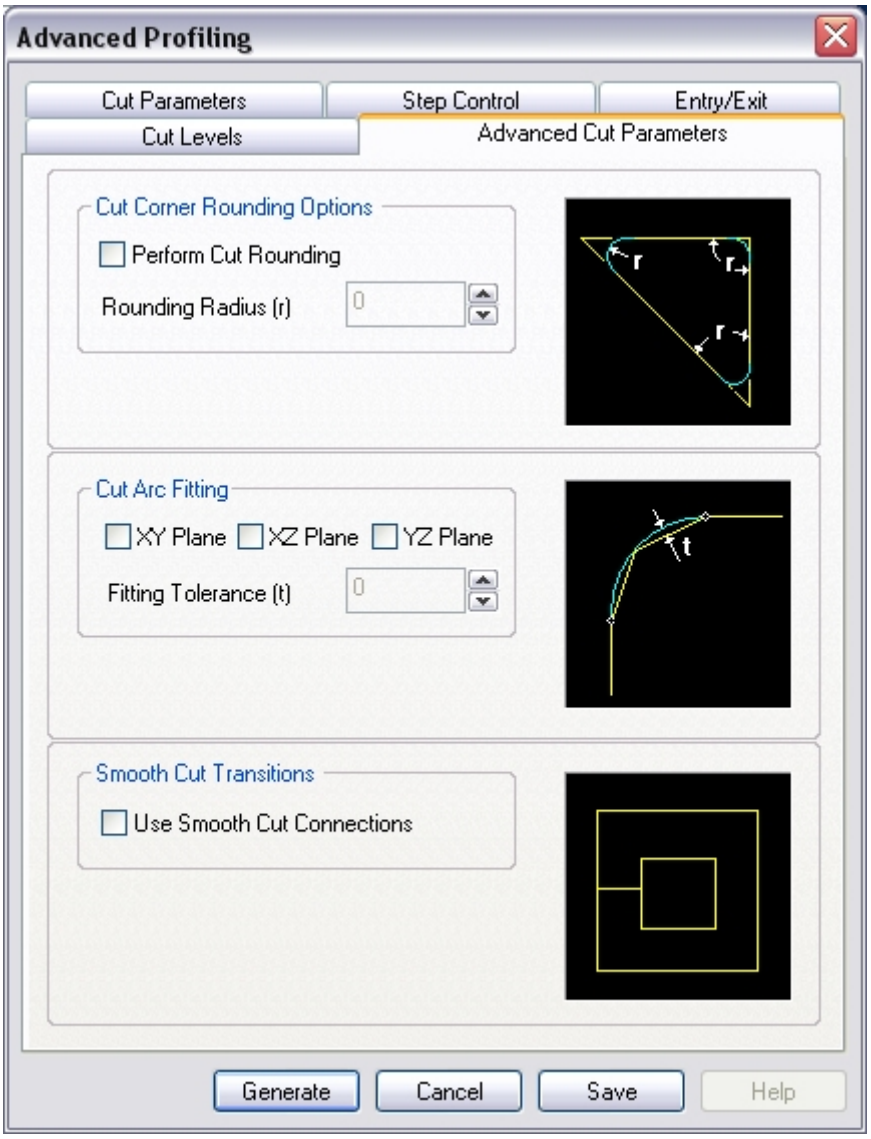
The use 3d Model to detect depth can be used when there is a 3d model available. This will detect the bottom most depth of the pocket (if there are multiple depths) and will add toolpath until that level. The finish depth can be specified when this option is used. The roughing depth is determined by the difference of the total depth and the finishing depth.

Cut levels can also be output as either a Depth first or as Level First.

Advanced Cut Parameters

The advanced cut parameters are 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 you machine during the cutting process. These parameters allow smoothing of the toolpaths by introduction of arcs. You can use these parameters even if your controller does not support arcs. Make sure that your output is set to linear output. You can do this under the Machining Preferences dialog in the Preferences section of the Menu bar.

- **Corner Rounding:** This option is used to round sharp corners in the toolpath. The user can specify a rounding radius and 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 other in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.



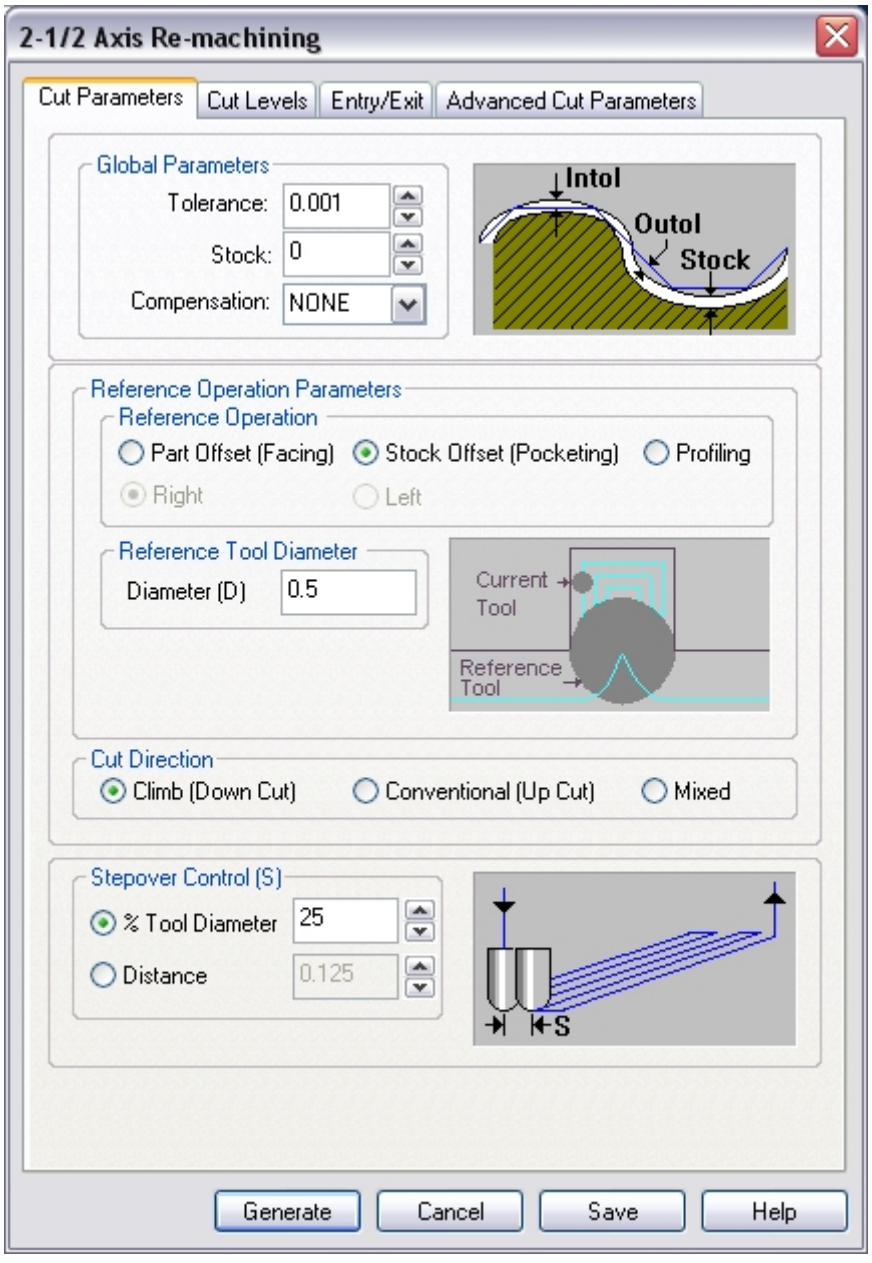
Re-Machining

Available in Alibre CAM Expert only.

This operation can be used to cut the uncut material from a previous operation using a larger tool. The previous operation and the previous tool diameter need to be specified for this operation. The previous operation can be either facing, pocketing or profiling. Given these parameters, all the uncut areas are automatically calculated. These uncut areas are processed similar to a facing operation.

Description

The Re-machining toolpath is invoked by clicking on the button in the MOps Browser and picking the **2½ Axis Milling** and **Re-machining** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



This dialog is tabbed with four tabs. Each tab defines a set of parameters that the user can specify. The sections below describe them in detail.

Cut Parameters

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

The Global Cut Parameters section allows the user to set the tolerance value to be used in machining. Also any uniform thickness or stock that needs to be left of the part also can be specified here.

Stock The thickness of the layer that will remain on sides 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.

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

Compensation stands for cutter compensation. The user can turn this on by selecting from the drop down menu. The cutter compensation to the left or right is determined by the Climb or Conventional direction.

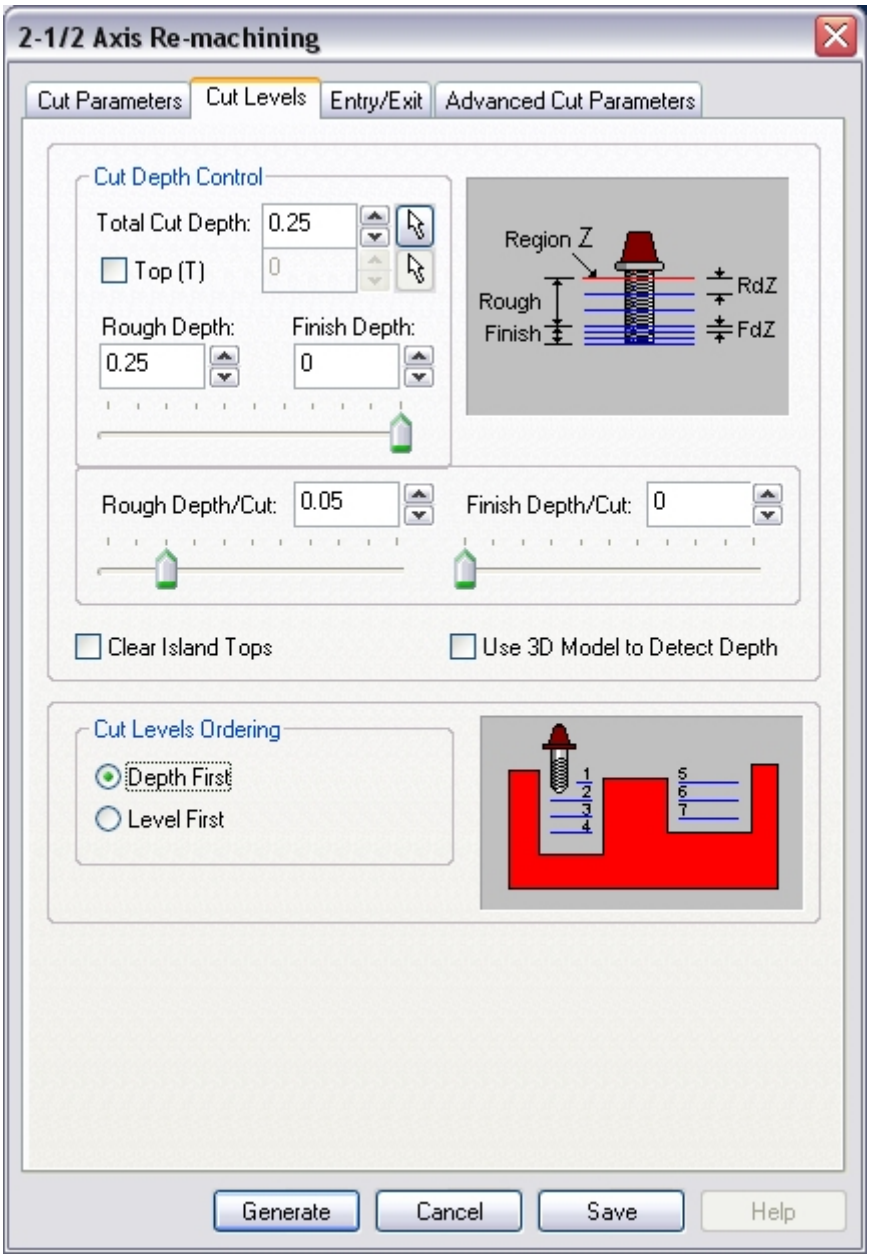
The Reference operation parameters section is used for determining the Uncut areas. The Reference operation, which can be either, a Facing, Pocketing or a Profiling operation needs to be specified along with the tool that was used for the reference operation. Once the Uncut areas are determined, they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

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

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

Cut Levels

When the user selects this tab in the dialog the following property page is displayed.



The user can set the top Z level by enabling the check box titled Top and specifying the top level. If the top level is not specified than the top of the outer most region is selected as the top Z level. The cut depth can be specified either by typing in the value or by selecting the pick button and selecting the bottom of the pocket. A depth of 0.0 will give one pass at the top Z level. The cuts can be specified as rough cuts and finish cuts, by sliding the bar or by typing in the values. Similarly for the rough and finish depth of cuts.

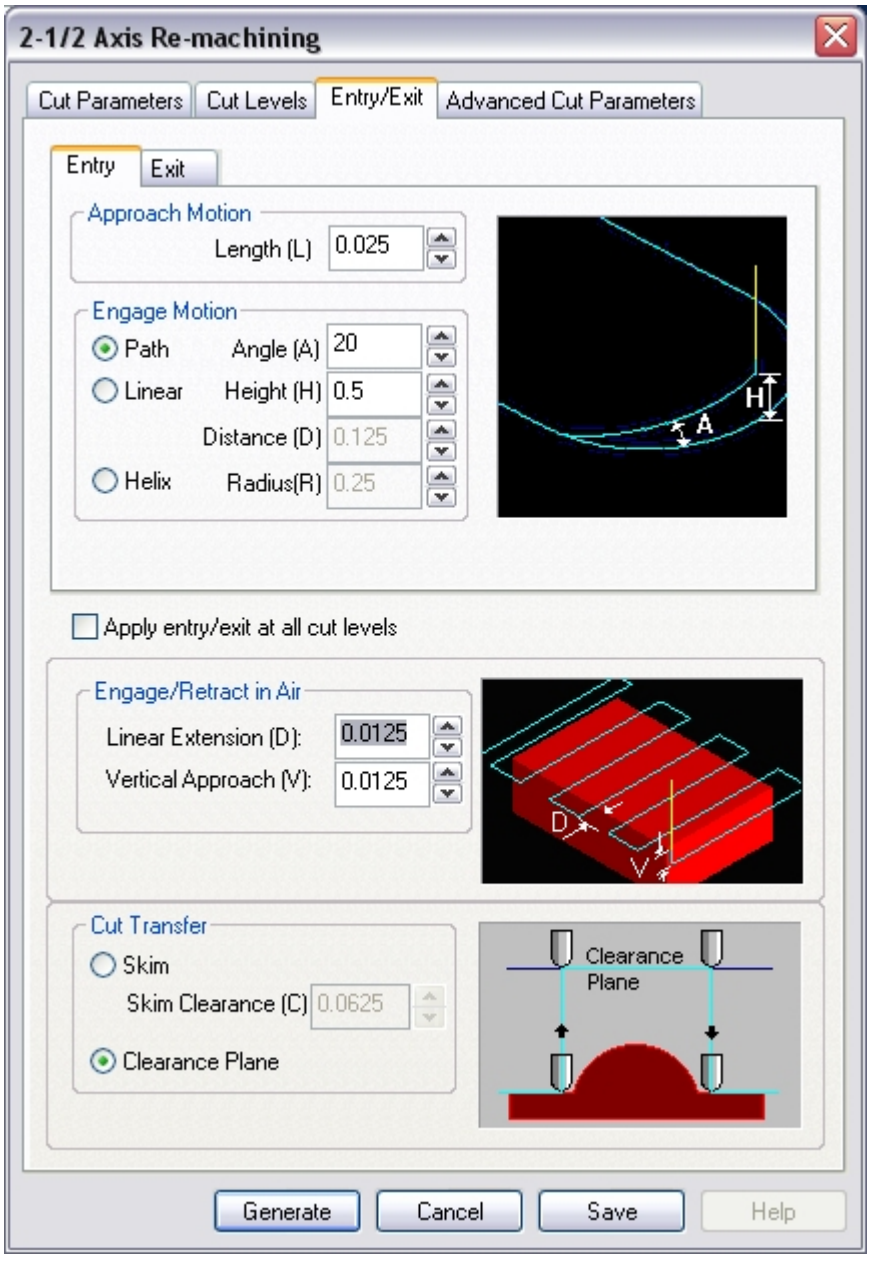
The clear islands will add a cut at the island levels (inner regions).

The use 3d Model to detect depth can be used when there is a 3d model available. This will detect the bottom most depth of the pocket (if there are multiple depths) and will add toolpath until that

level. The finish depth can be specified when this option is used. The roughing depth is determined by the difference of the total depth and the finishing depth.

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.



The Entry motion consists of Approach and Engage. 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.

In a facing operation, the outer most regions (the outer most region and its immediate inner regions) are cut in a facing approach and all the inner pockets are cut using the pocketing approach. The Engage in Material section is for the pocketing toolpath that is generated for the inner pockets. The Engage in Air section is for the outer most regions as described above

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.

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 he 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 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 form the outside, a certain distance away from the cut start point. Or optionally a straight vertical engage can be specified.

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.

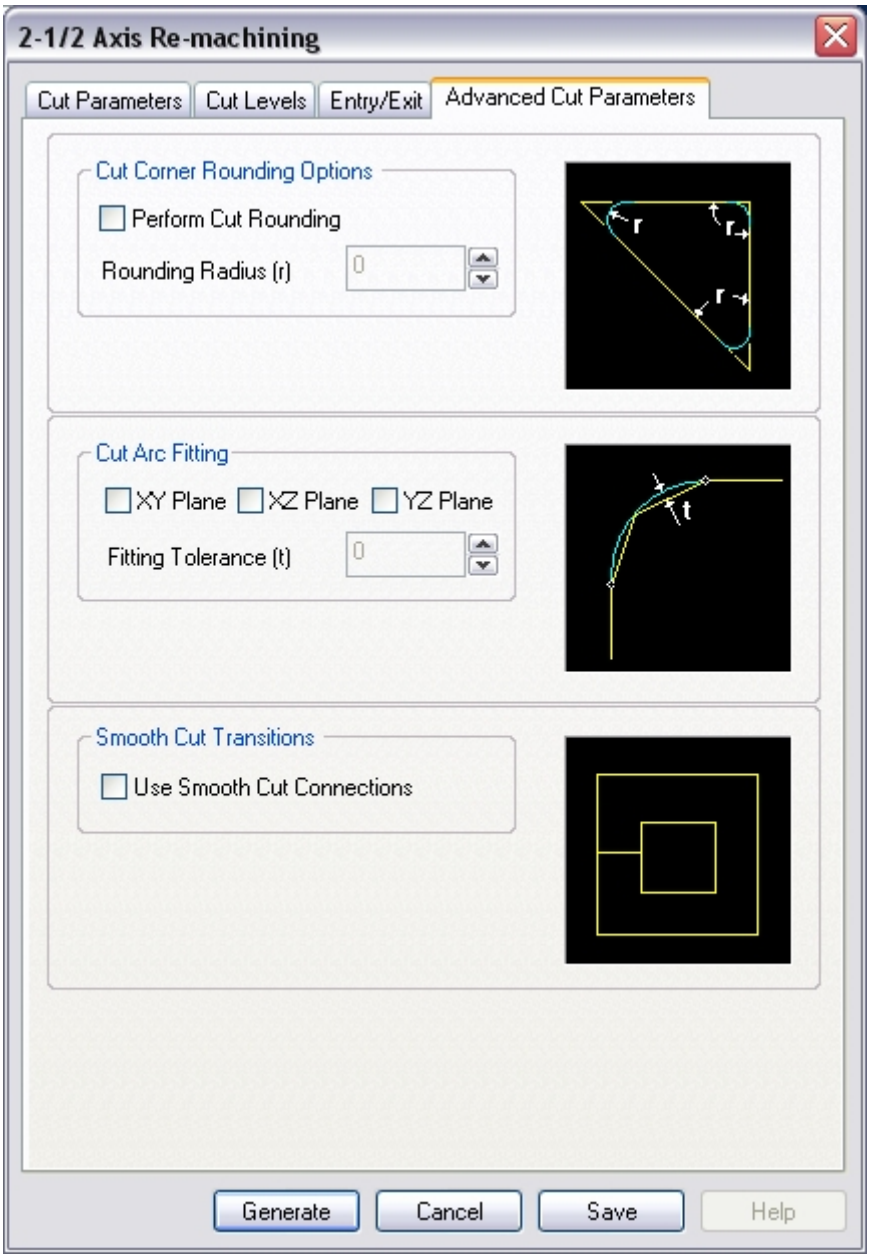
The user can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can either be instructed to move to the clearance plane and then perform the transfer motion to the next cut location or it could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

Advanced Cut Parameters

The advanced cut parameters are 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 you machine during the cutting process. These parameters allow smoothing of the toolpaths by introduction of arcs. You can use these parameters even if your controller does not support arcs. Make sure that

your output is set to linear output. You can do this under the Machining Preferences dialog in the Preferences section of the Menu bar.

- **Corner Rounding:** This option is used to round sharp corners in the toolpath. The user can specify a rounding radius and 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 other in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.



Creating 3 Axis Machining Operations

In this type of machining, the tool can move simultaneously in all three axes. This is appropriate for parts that have complex, curved, and non-vertical features.

A typical machining scenario would be to first use **Horizontal Roughing**, the pre-finishing using **Parallel Finishing**. Once the part is at near net shape then fine detailed finishing could be performed by selecting a smaller diameter tool and use a region to limit the toolpath.

Horizontal Roughing

This is Alibre CAM's principal method of roughing, also knows 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.

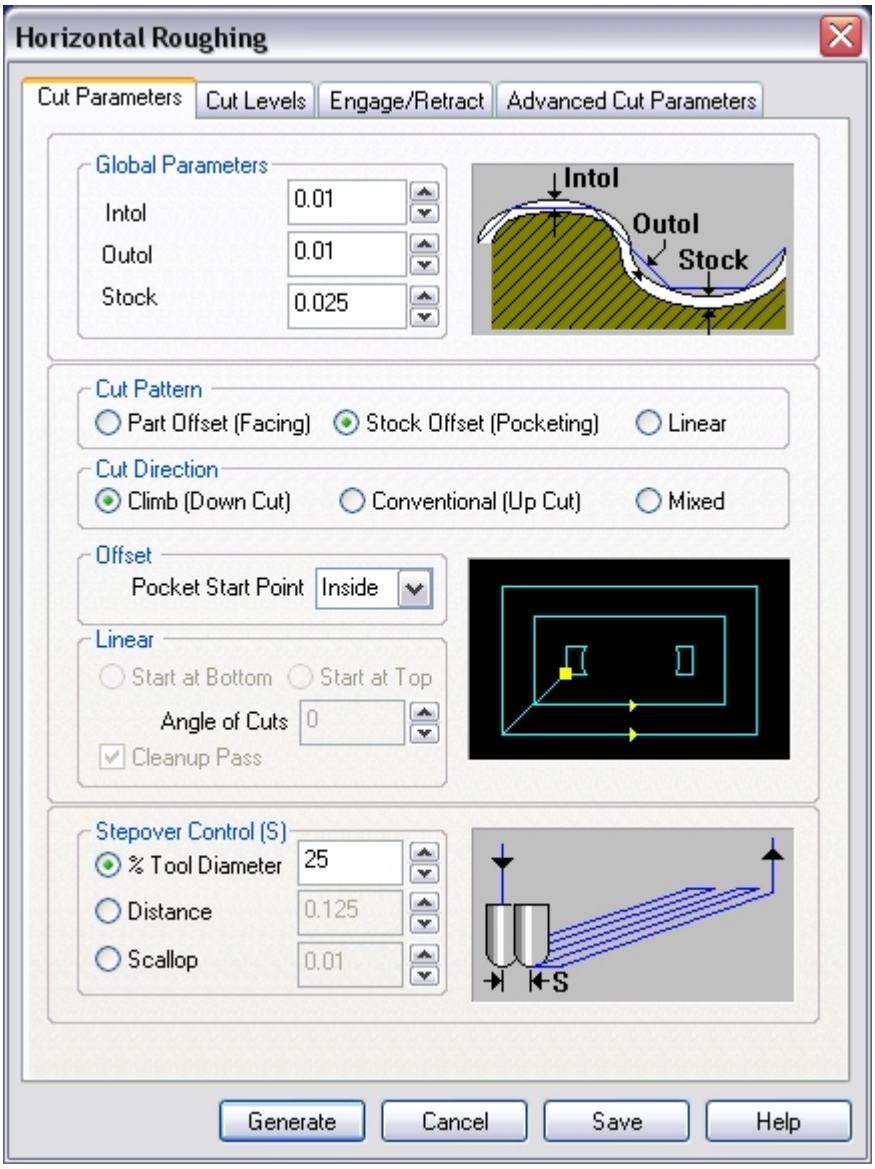
Rough machining where bulk material removal is accomplished from the raw stock model is done in Alibre CAM using the Horizontal Roughing machining operation. Here material is removed in layers or levels. Typical tools used for this kind of operation are flat end mills or corner radius end mills. The tool starts at the top of the stock model and removes material without changing its Z position and only moving in the XY plane. Once this level is completed, then the tool moves over the next successive (lower) Z level and removes material in this XY plane. This procedure is repeated until the bottom most Z level is reached. The spacing between the cut levels can be specified by the user. The user can also specify the Z values of the top and the bottom cut levels. The method in which the tool travels in each Z level to remove material can also be controlled by the user.

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).

Note: Horizontal Roughing needs to have a stock model defined.

Description

The Horizontal Roughing toolpath is invoked by clicking on the  button in the MOps Browser and picking the **3 Axis Milling** and **Horizontal Roughing** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



This dialog is a tabbed dialog with three tabs. Clicking on each of the tabs allows the user to set the different parameters. The following sections describe each of the tabs in more detail.

Cut Parameters

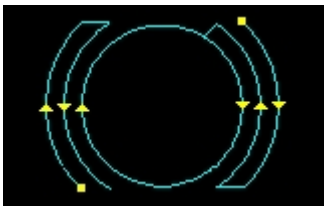
The user has the ability to set the Global Cut Parameters, the Cut Pattern, the Cut direction and the Step-over Control via this property page of the dialog.

- **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.

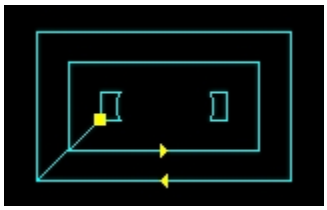
Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any).

- **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.

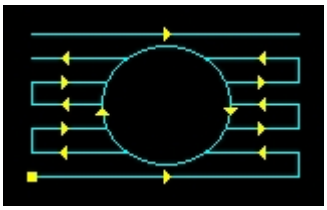
The Cut Pattern section allows the user to define the type of cut pattern that the tool will follow when it is at each Z level. There are three types of cuts that the user can choose. The first two are an offset cut type (stock offset or part offset) and the third is a linear cut type.



Part Offset

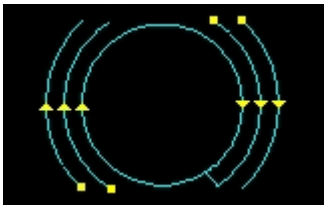


Stock Offset

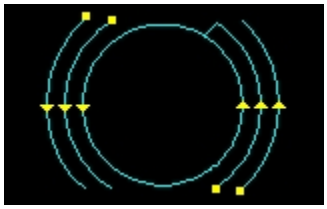


Linear

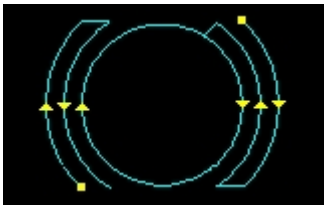
In the offset cut type, the cut regions are successively offset until there are nothing left to machine. This type of cutting is sometimes called spiral machining. The user can specify the cut start point to be either the inside of the cut regions or from the outside.



Climb



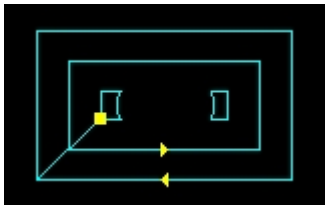
Conventional



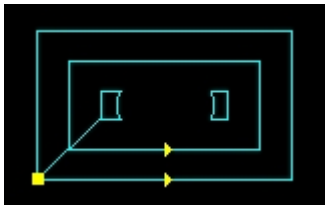
Mixed

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.



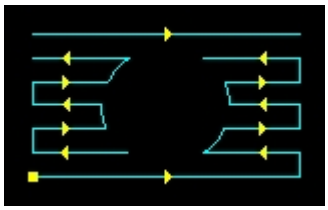
Inside



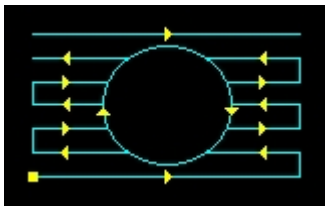
Outside

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.



Without Cleanup Pass

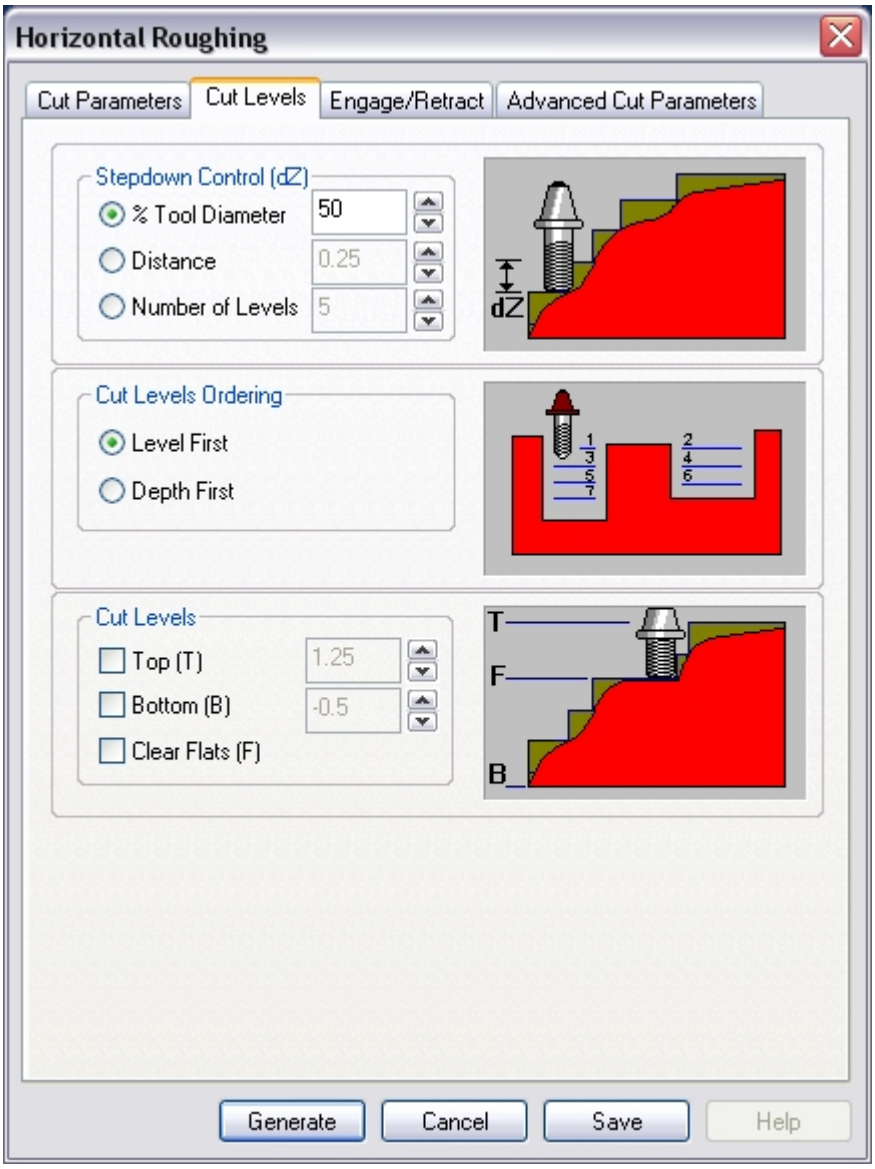


With Cleanup Pass

The Step-over 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.

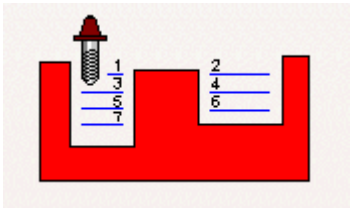
Cut Levels

When the user selects this tab in the dialog the following property page is displayed. The user has the ability to set the Stepdown Control as well as specify the Cut Levels in this page.

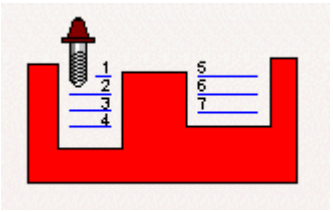


The Step-down Control section allows the user to define the spacing between the horizontal cut levels for the roughing operation. The spacing can be specified either as a percentage of tool diameter, a specific distance or as the total number of levels desired.

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.



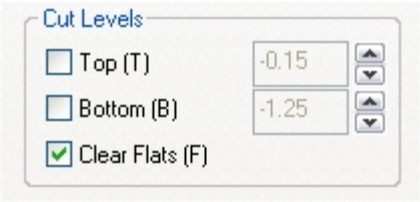
Level First



Depth First

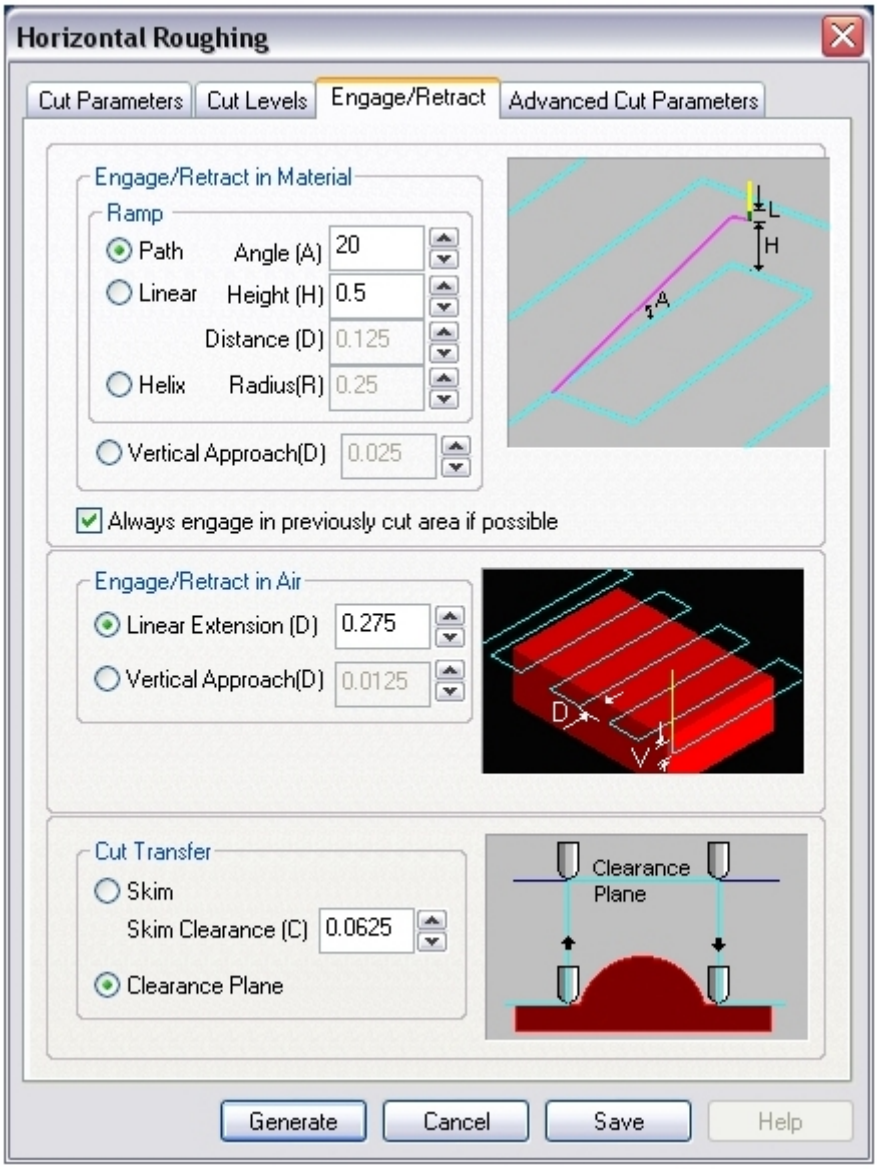
The Cut Levels section allows the user to optionally specify the top and bottom cut levels. If the top cut level is not specified, the maximum Z value of the stock model is used as the start Z value for computing the cut levels. If the bottom cut level is not specified then the minimum Z value of the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

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

When the user selects the Engage/Retract tab the following property page is displayed. 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.



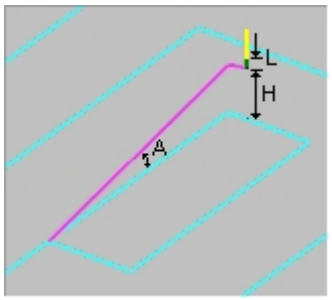
The Engage in Material section allows the user to define how the cutter would engage into material when forced into such a situation. This can happen when machining a cavity or pocket. The user has the option of Ramping the cutter or engaging vertically down. The ramp engage motion will typically be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

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

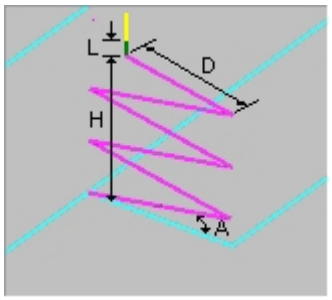
- **Path**: In this method the cutter follows the contour of the part in a ramping motion on he outside until it hits the cutting start point. The user can control the angle of descent and the length of this engage motion by specifying these parameters.
- **Linear**: Here the cutter follows a linear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

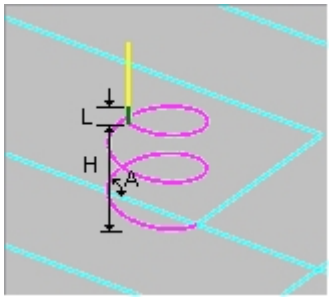
Path: In this method the cutter follows the contour of the part in a ramping motion on he outside until it hits the cutting start point. The user can control the angle of descent and the length of this engage motion by specifying these parameters.



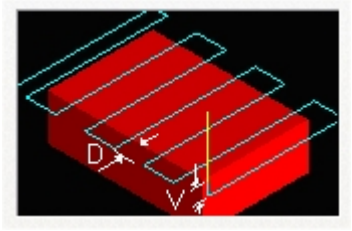
Linear: Here the cutter follows a linear ramp motion, ramping back and forth from a user specified height to the engage point. The length of this move, as well as the angle of this motion can be specified y the user.

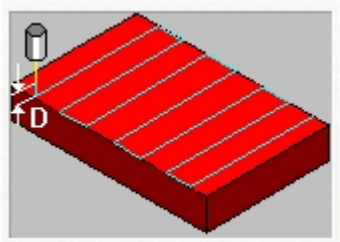


Helix: Here the cutter follows a helix as it descends from a user-defined height to the first cut point. The angle of the helix as well as the radius of the helix can be specified by the user.



The 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 form the outside, a certain distance away from the cut start point. Or optionally a straight vertical engage can be specified.



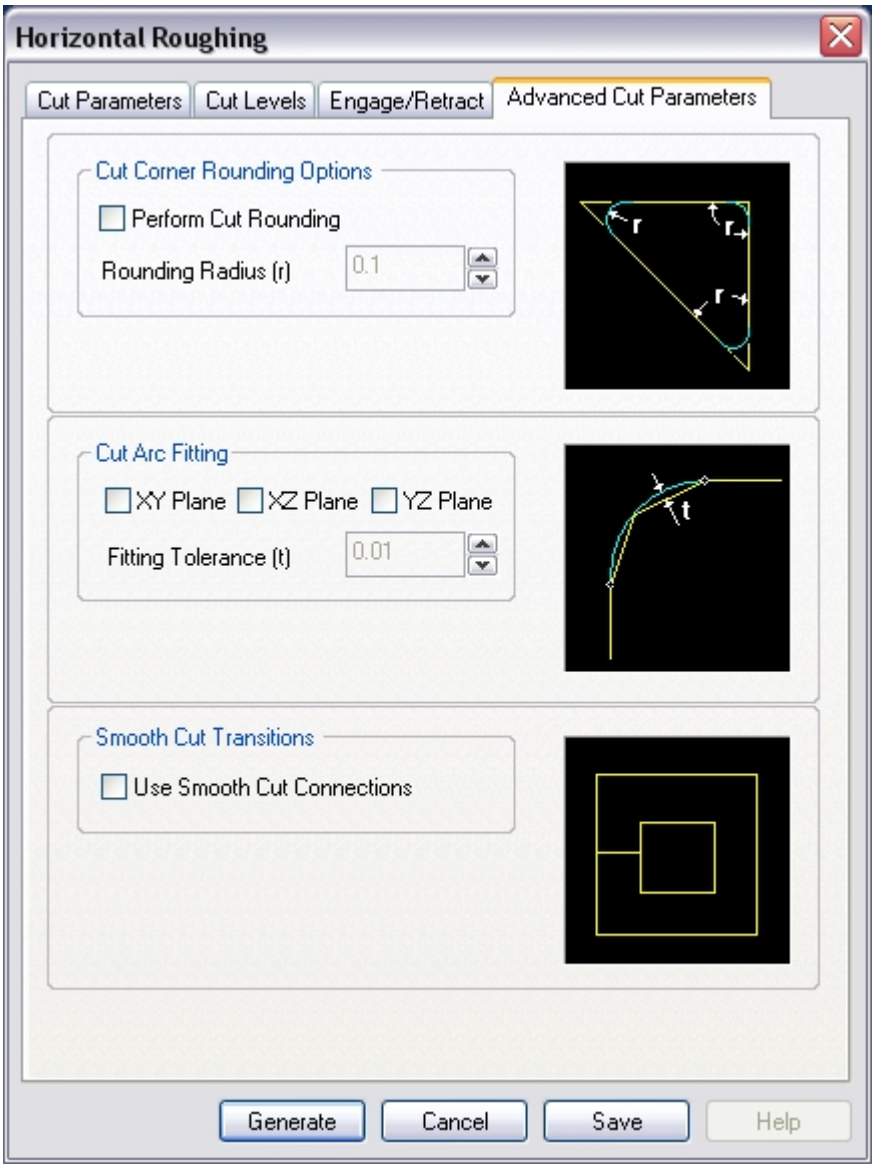


Linear Extension **Vertical Approach**

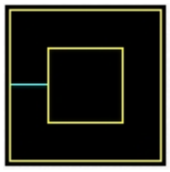
The user can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can either be instructed to move to the clearance plane and then perform the transfer motion to the next cut location or it could 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.

Advance Cut Parameters

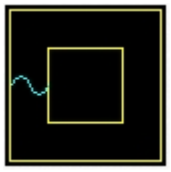
The advanced cut parameters are used to control the cuts for high speed machining. All of the options in this section are designed to reduce rapid acceleration and deceleration of you machine during the cutting process. These parameters allow smoothing of the toolpaths by introduction of arcs. You can use these parameters even if your controller does not support arcs. Make sure that your output is set to linear output. You can do this under the Machining Preferences dialog in the Preferences section of the Menu bar.



- **Corner Rounding:** This option is used to round sharp corners in the toolpath. The user can specify a rounding radius and 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 other in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.



Smooth Transition Off



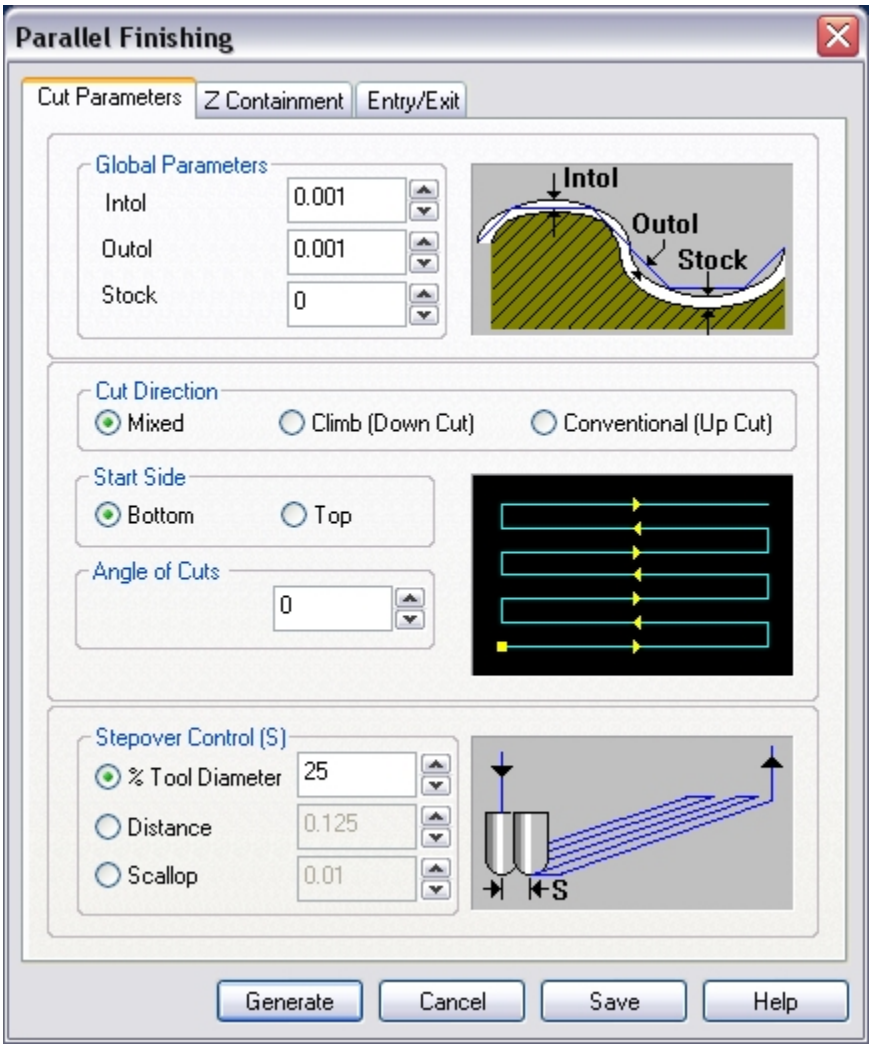
Smooth Transition On

Parallel Finishing

Parallel Finishing is a toolpath method, which can be used either as a pre-finishing operation or as a finishing operation. In this cut method, the cutter is restricted to follow the contours of a part in the Z direction while being locked to a series of parallel vertical planes. The orientation 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.

Description

The Parallel Finishing toolpath is invoked by clicking on the  button in the MOps Browser and picking the **3 Axis Milling** and **Parallel Finishing** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



This dialog is a tabbed dialog with two tabs. Clicking on each of the tabs allows the user to set the different parameters. The following sections describe each of the tabs in more detail.

Cut Parameters

The user has the ability to set the Global Cut Parameters, the Cut Direction, the Start Side, the Angle of Cuts and the Stepover Control via this property page of the dialog.

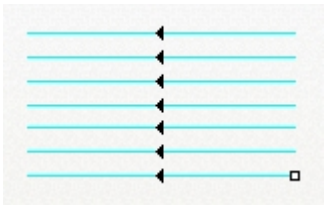
- Stock: The thickness of the layer that will remain on top of the part after the toolpath is complete. Roughing operations generally leave a thin layer of stock, but for finishing operations this value is zero.

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

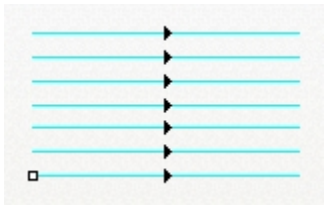
- Intol: Inward tolerance - the maximum thickness of material that can be removed from the Stock layer.

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

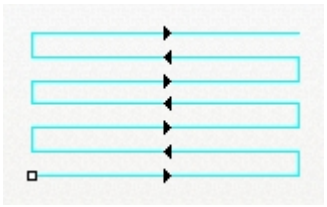
The cut direction can be controlled by specifying either climb or conventional or mixed.



Climb

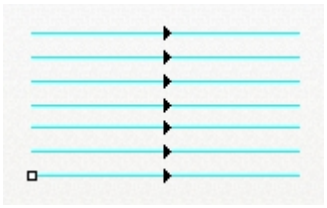


Conventional

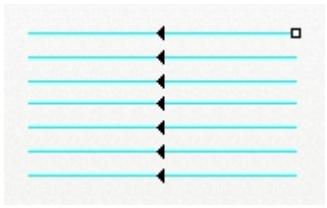


Mixed

The start side can be specified as from Top or Bottom



Start at Bottom

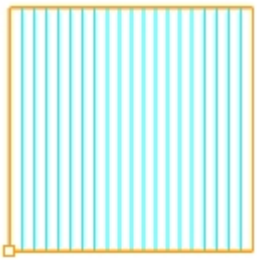


Start at Top

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

Angle = 0

Angle = 45

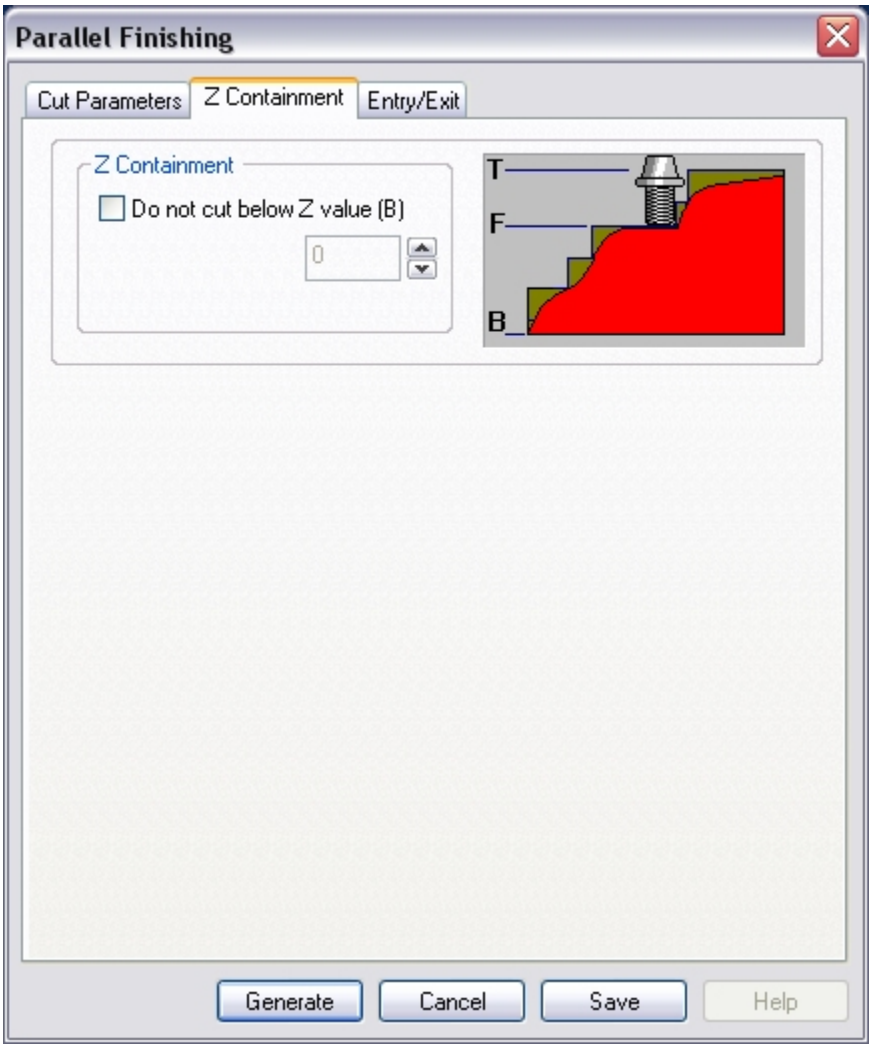


Angle = 90

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.

Z Containment

Then the user selects this tab in the dialog the following property page is displayed. The user has the ability to set the Z Containment through this dialog.



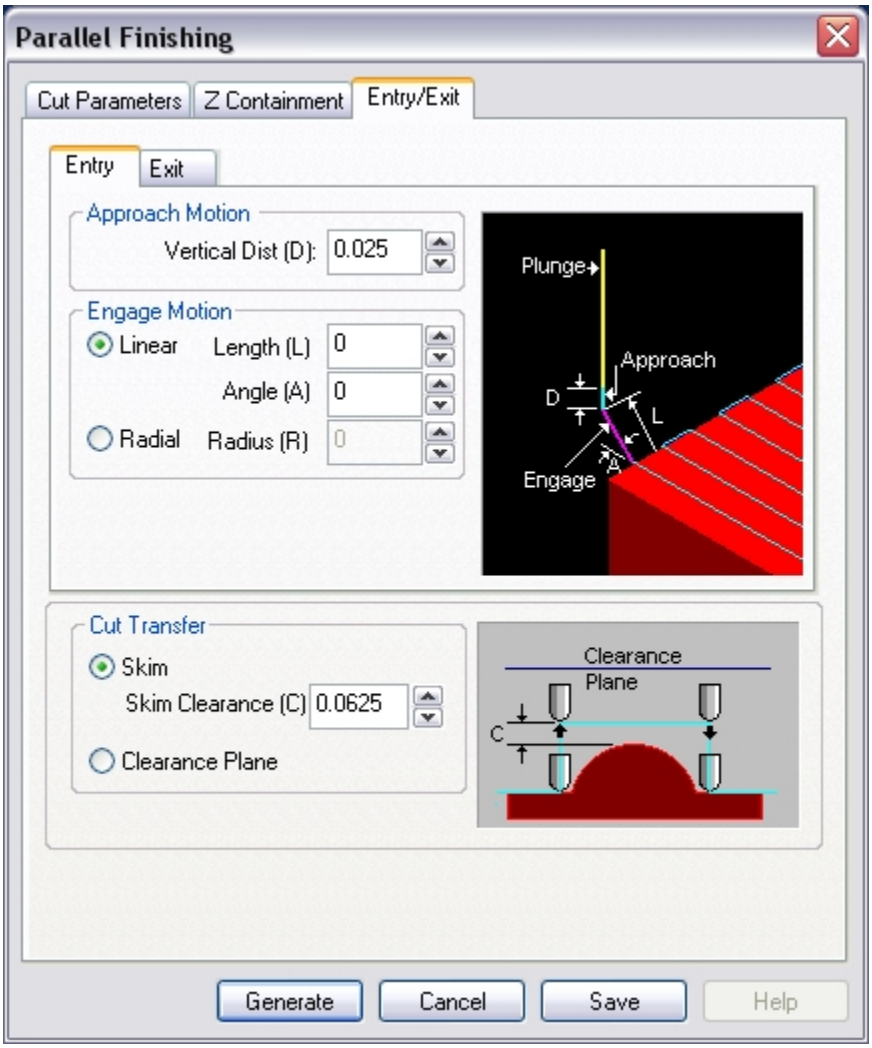
Entry / Exit

This entry/exit page is invoked in various 3 axis toolpath methods. This page controls the entry and exit motions in a vertical plane.

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

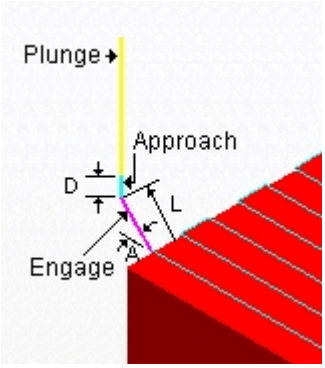
Entry
The user can choose to enter the material to be cut with a combination of two motions. These motions are Approach and Engage. The Approach motion is the motion as the cutter traverses from the transfer or plunge motion and meets the Engage motion. The Engage motion is the last motion before the cutter begins to cut material. The user has the ability to define the Approach motion as a vertical distance above the start of the Engage motion. The user can define the Engage

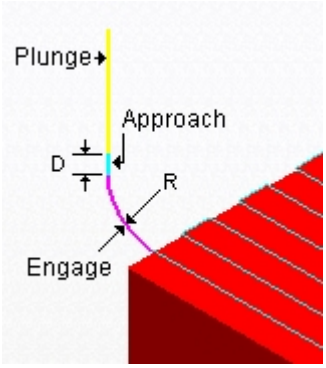
motion either as a Linear ramp or Radial ramp motion as the graphic in the dialog indicates.



Exit

The user can choose to exit the cutting also as a combination of two motions. These motions are Retract and Departure. The Retract motion is the first motion after the cutter loses contact with the material to be cut. The Departure motion is the motion that bridges the Retract motion and the transfer motion. The Retract motion can be defined either as a Linear ramp or Radial ramp motion as the graphic in the dialog indicates. The user has the ability to define the Departure motion as a vertical distance above the end of the Retract motion.





Linear Entry **Radial Entry**

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.

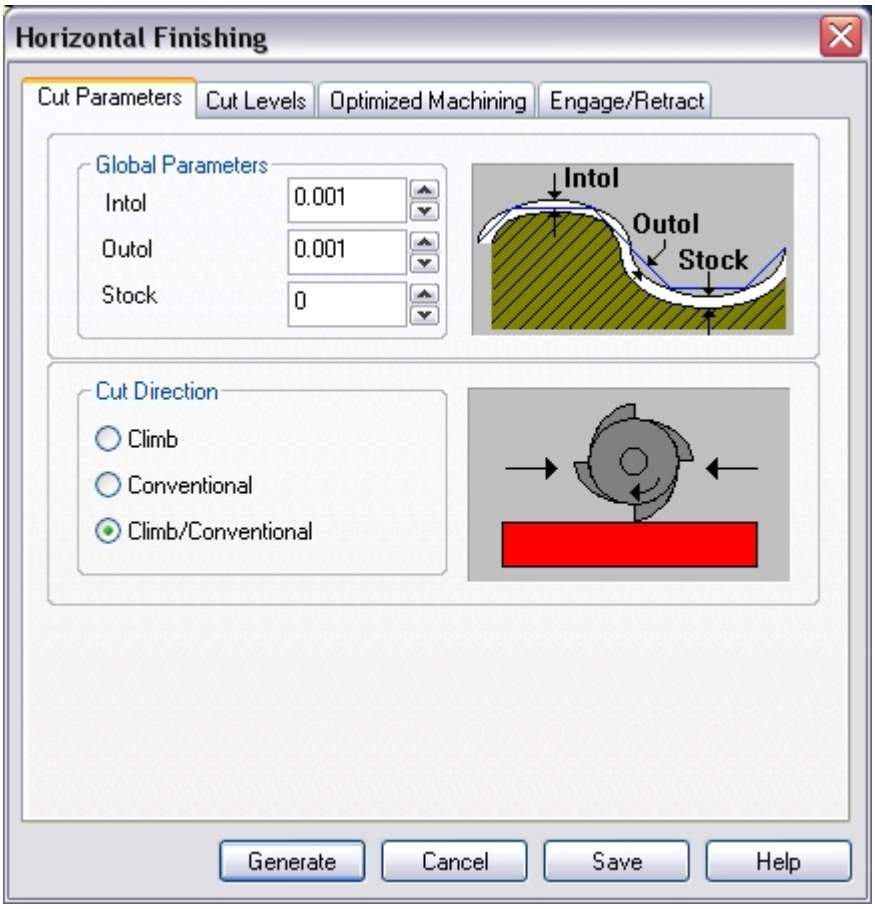
Horizontal Finishing

Available in Alibre CAM Standard, Expert and Professional versions only

Horizontal Finishing is a toolpath method, which can be used either as a pre-finishing operation or as a finishing operation. This method is similar to Parallel 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 however, the direction of cutting is alternated between each parallel plane, similar to the ZigZag machining in Contour Machining.

Description

The Horizontal Finishing toolpath is invoked by clicking on the button in the MOps Browser and picking the **3 Axis Milling** and **Horizontal Finishing** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



This dialog is a tabbed dialog with three tabs. Clicking on each of the tabs allows the user to set the different parameters. The following sections describe each of the tabs in more detail.

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 intol and the outol values to be used in machining. Also any uniform thickness or stock that needs to be left of the part also can be specified here.

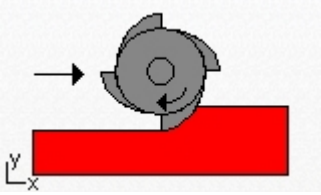
- Stock: The thickness of the layer that will remain on top of the part after the toolpath is complete. Roughing operations generally leave a thin layer of stock, but for finishing operations this value is zero.

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

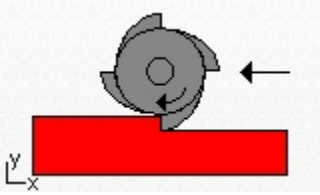
- Intol: Inward tolerance - the maximum thickness of material that can be removed from the Stock layer.

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

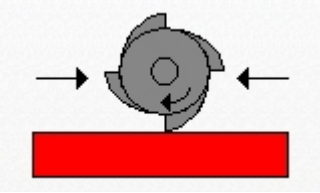
The Cut Direction of the toolpath can be specified either as Climb, Conventional or Climb/Conventional by selecting the appropriate radio buttons. As mentioned before, in climb or conventional, the direction of cutting is maintained so as the corresponding cutting condition is maintained on the part.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



Climb Cut



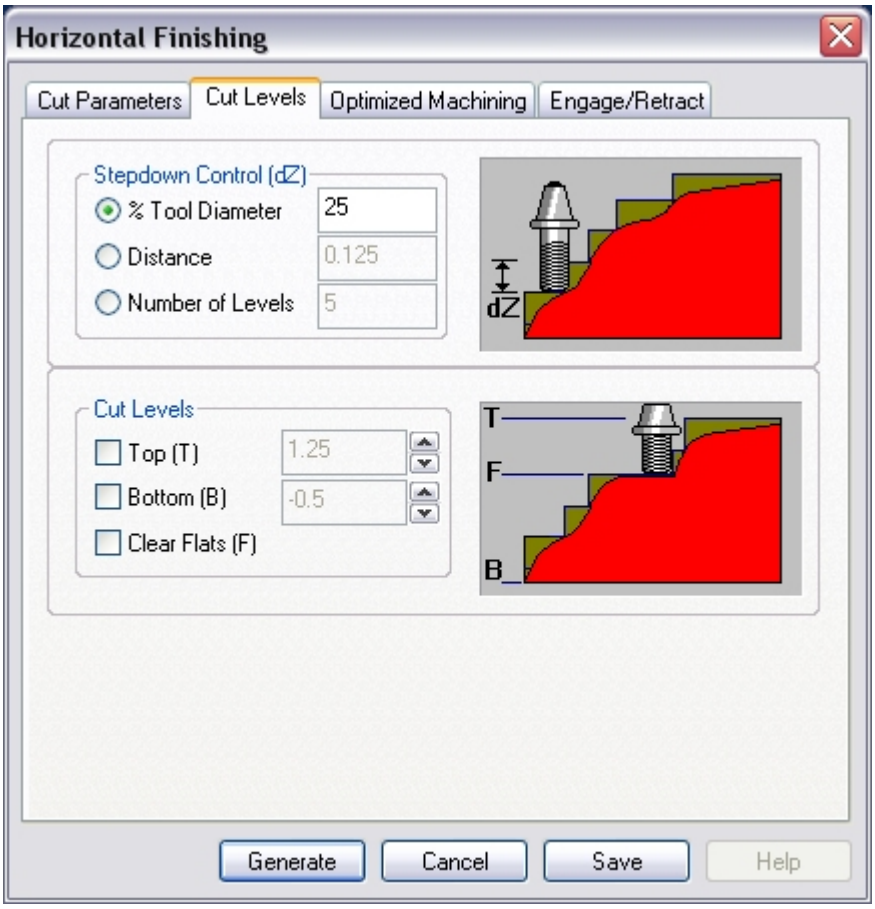
Conventional Cut



Climb/Conventional Cut

Cut Levels

When the user selects this tab in the dialog the following property page is displayed. The user has the ability to set the Stepdown Control as well as specify the Cut Levels in this page.



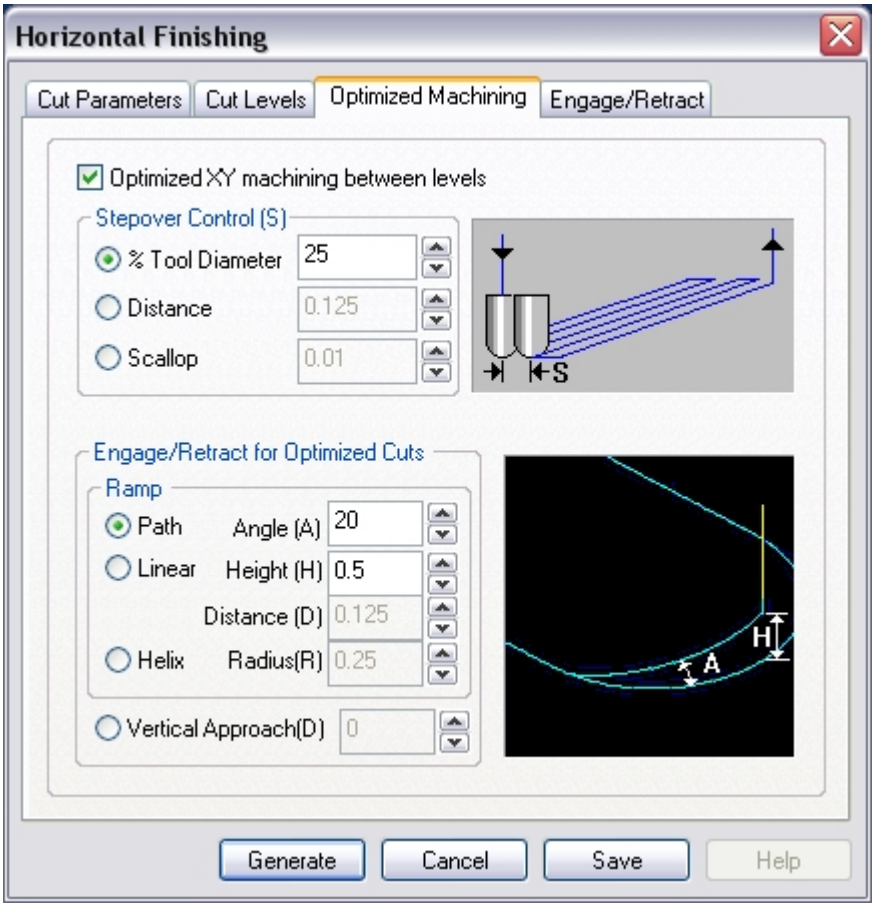
The Stepdown Control section allows the user to define the spacing between the horizontal cut levels for the roughing operation. The spacing can be specified either as a percentage of tool diameter, a specific distance or as the total number of levels desired.

The Cut Levels 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. 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.

Optimized XY Machining

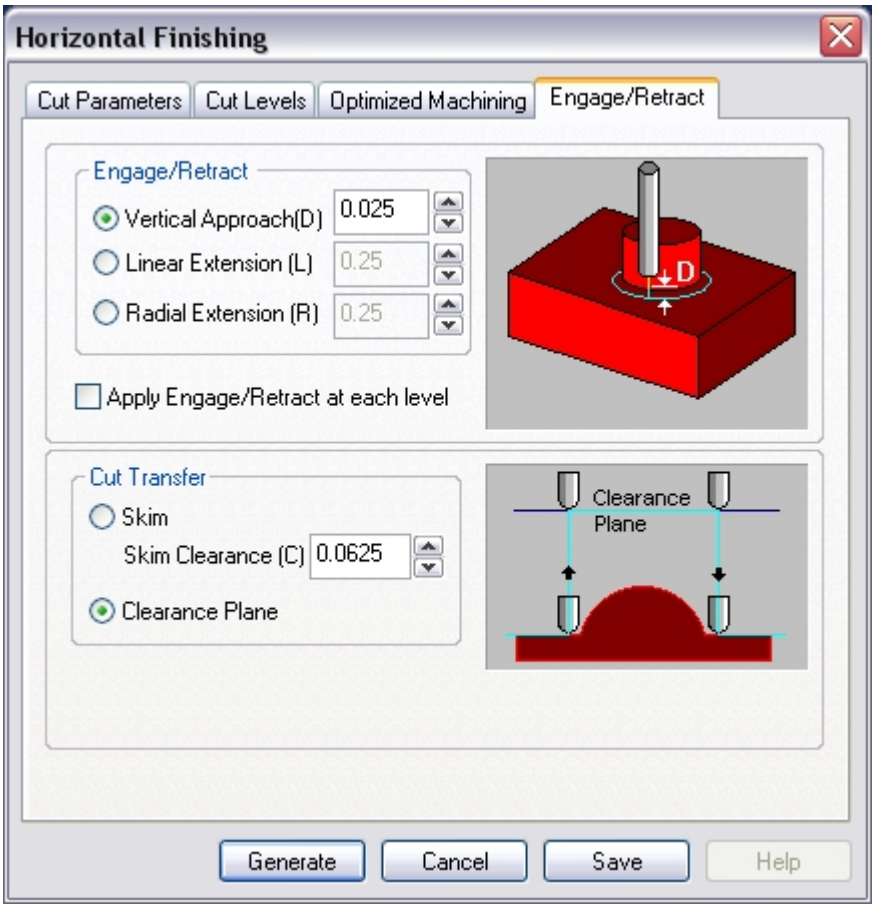
Optimized machining is used when the user wants to maintain almost constant scallop height during machining with the Horizontal Finishing method. In this method, machining takes place in constant Z planes spaced a constant Z distance from each other. Due to this constant Z spacing, machining relatively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

retracts for these pocket toolpaths. It should be noted that optimized machining will work only between closed cuts. The uncut material between open cuts will be ignored.



Engage / Retract

When the user selects the Engage/Retract tab the following property page is displayed. The user will be able to specify how the cutter engages and retracts when starting and stopping a cut. The user can also specify the type of transfer motions to perform while cutting.



The user can optionally pick a vertical engage/retract in which case the tool engages into the first cut start point vertically from above and switches feed-rate to cut feed rate the specified distance above the first cut point. In the case of a linear extension, the cuts are linearly extended by the specified distance at an angle of 45 degrees to the tangent at the start and end of cut. In the case of radial extensions, the cuts are radially extended by a quarter circle of specified radius from the cut start and end points.

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.

Plunge Roughing

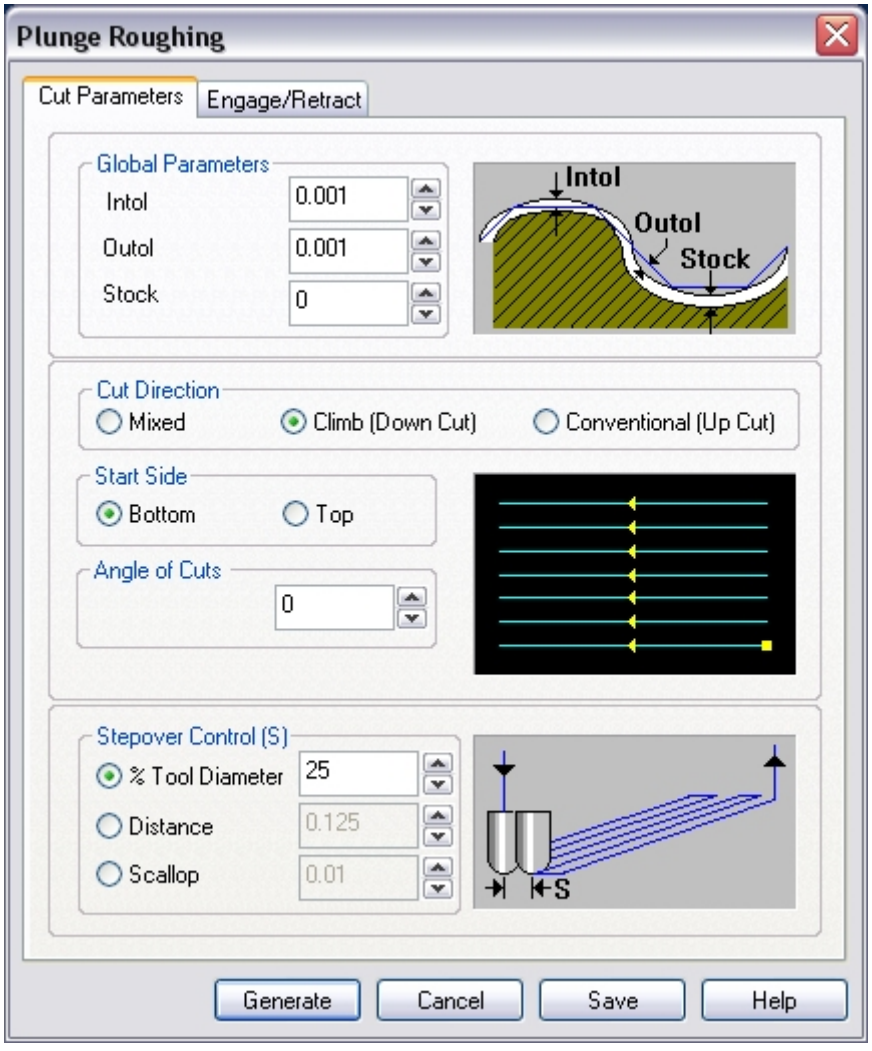
Available in Alibre CAM Expert only.

Alibre CAM'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. Plunge roughing is sometimes called Drill Roughing.

Description

The Plunge Roughing toolpath is invoked by clicking on the button in the MOps Browser and picking the **3 Axis Milling** and **Plunge Roughing** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



This dialog is a tabbed dialog with two tabs. Clicking on each of the tabs allows the user to set the different parameters. The following sections describe each of the tabs in more detail.

Cut Parameters

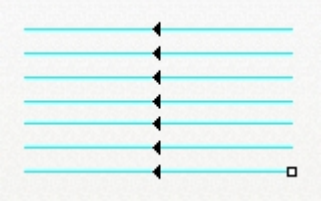
The user has the ability to set the Global Cut Parameters, the Cut Pattern, the Cut direction and the Stepover Control via this property page of the dialog.

- 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.

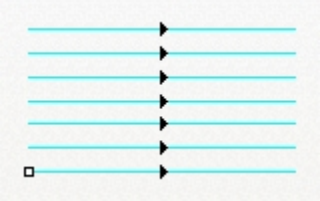
Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any).

- 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.

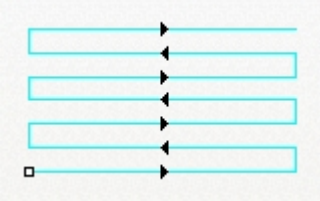
The cut direction can be controlled by specifying either climb or conventional or mixed.



Climb

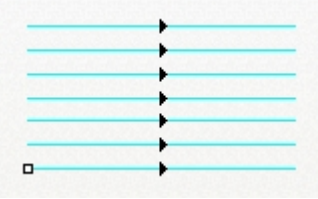


Conventional

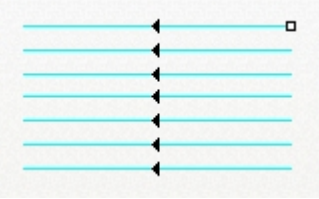


Mixed

The user can specify the cut start side to be either the bottom or top and the angle of cuts can be specified.



Start at Bottom

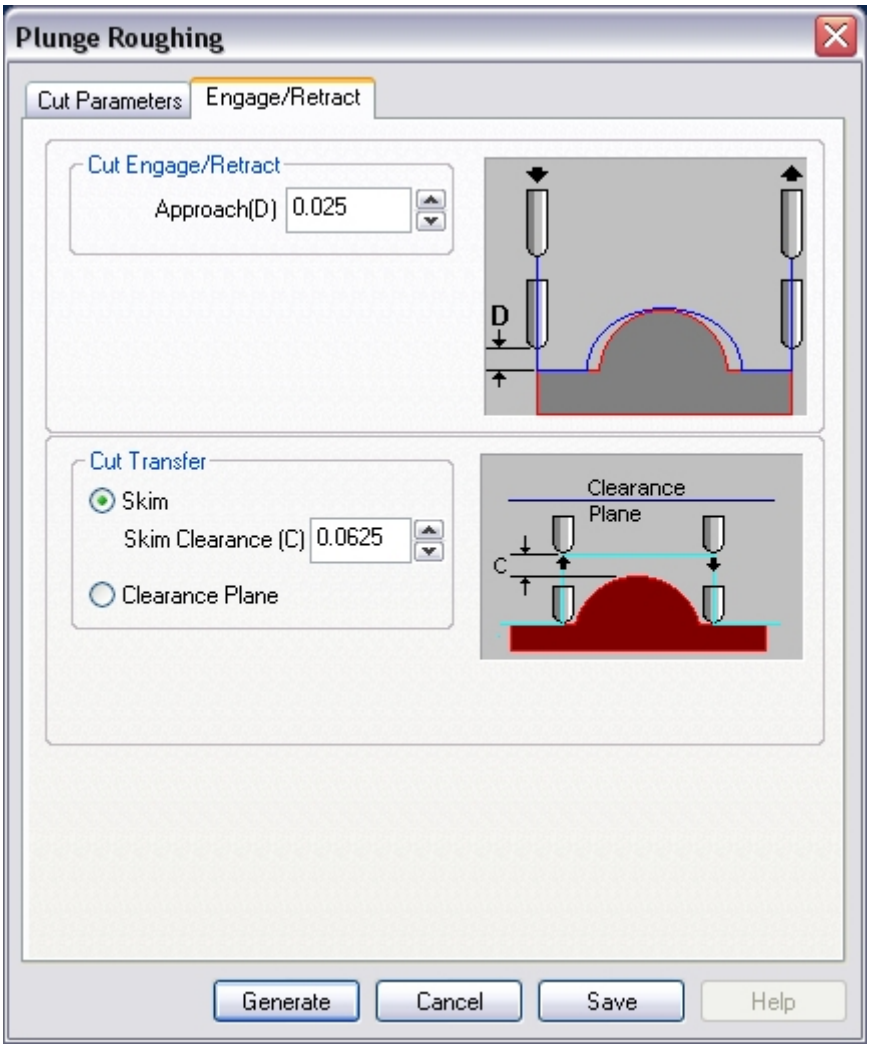


Start at Top

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

Then the user selects this tab in the dialog the following property page is displayed. The user has the ability to set the Engage/Retract parameters via this dialog.



The approach distance for Engage/Retract can be specified. 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.

Horizontal Re-Roughing

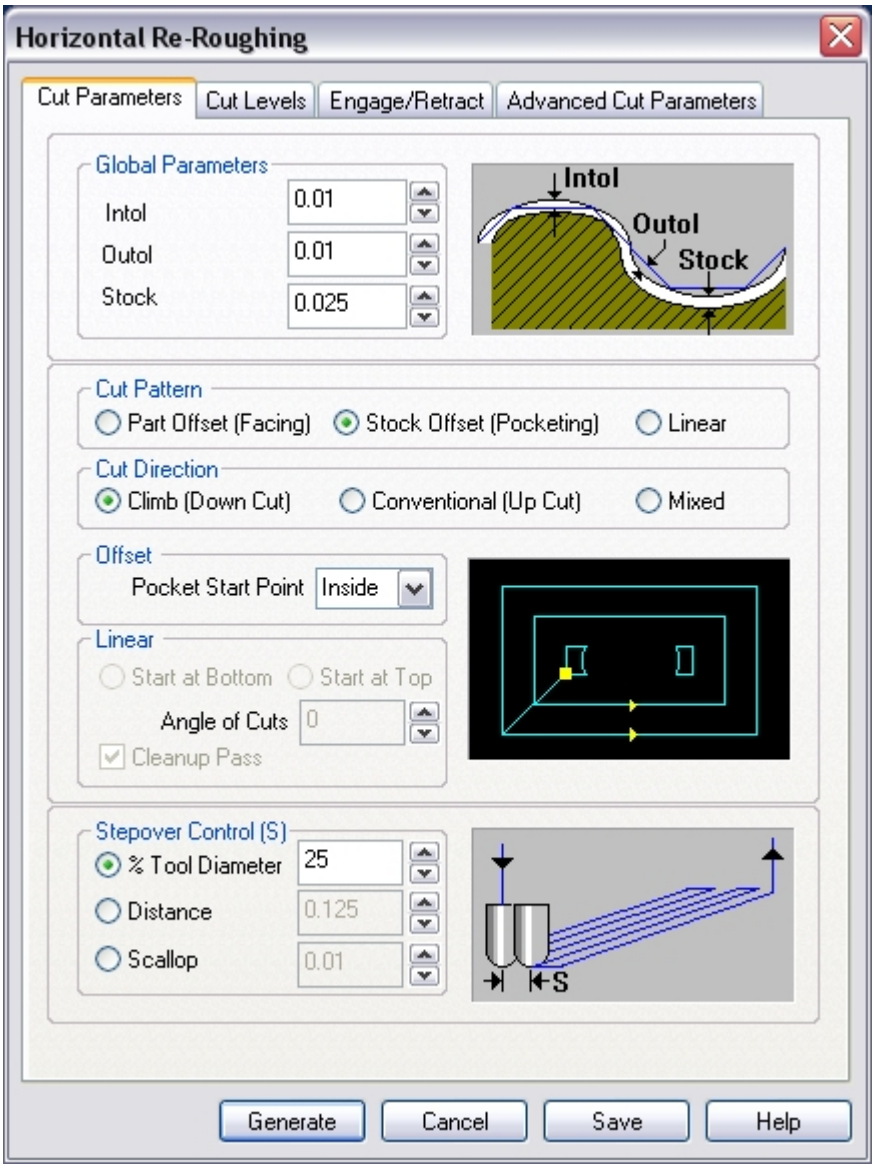
Available in Alibre CAM Expert only.

Alibre CAM's horizontal re-roughing algorithms can be used to create toolpaths only in areas that were not machined by a previous operation. 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 Alibre CAM 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.

Description

The Horizontal Re-Roughing toolpath is invoked by clicking on the  button in the MOps Browser and picking the **3 Axis Milling** and **Horizontal Re-Roughing** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



This dialog is a tabbed dialog with three tabs. Clicking on each of the tabs allows the user to set the different parameters. The following sections describe each of the tabs in more detail.

Cut Parameters

The user has the ability to set the Global Cut Parameters, the Cut Pattern, the Cut direction and the Stepover Control via this property page of the dialog.

- 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 are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any).

- 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.

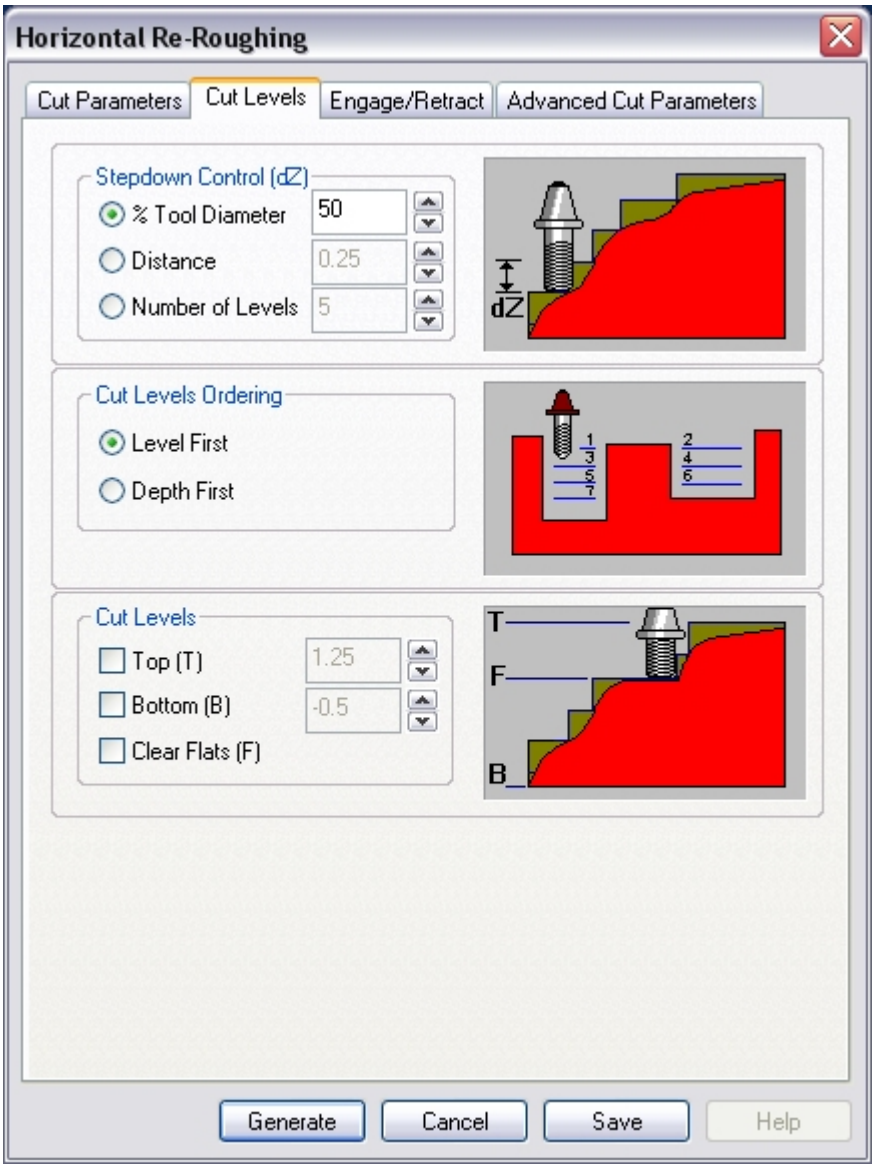
The Cut Pattern section allows the user to define the type of cut pattern that the tool will follow when it is at each Z level. There are three types of cuts that the user can choose. The first two are an offset cut type (stock offset or part offset) and the third is a linear cut type.

In the offset cut type, the cut regions are successively offset until there are 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. Additionally the cut direction can be controlled by specifying either climb or conventional or mixed.

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

Then the user selects this tab in the dialog the following property page is displayed. The user has the ability to set the Stepdown Control as well as specify the Cut Levels in this page.



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.

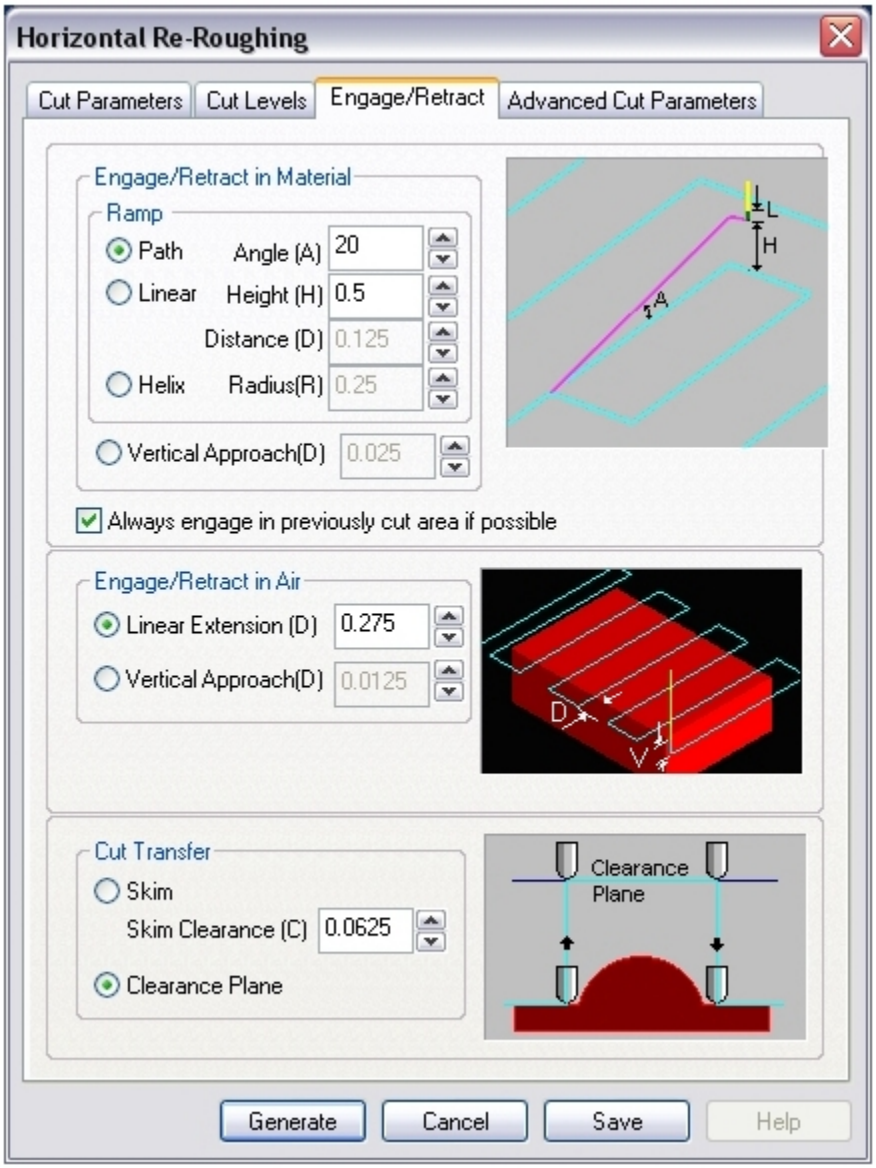
The Cut Levels Ordering section allows the user to order the cut regions. The order of cutting can either be specified as level first or depth first. In the level first option cut regions are ordered such that all cut regions in a single Z level or machined first before the next cut level is machined.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

Then the user selects the Engage/Retract tab the following property page is displayed. 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.

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 he 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.

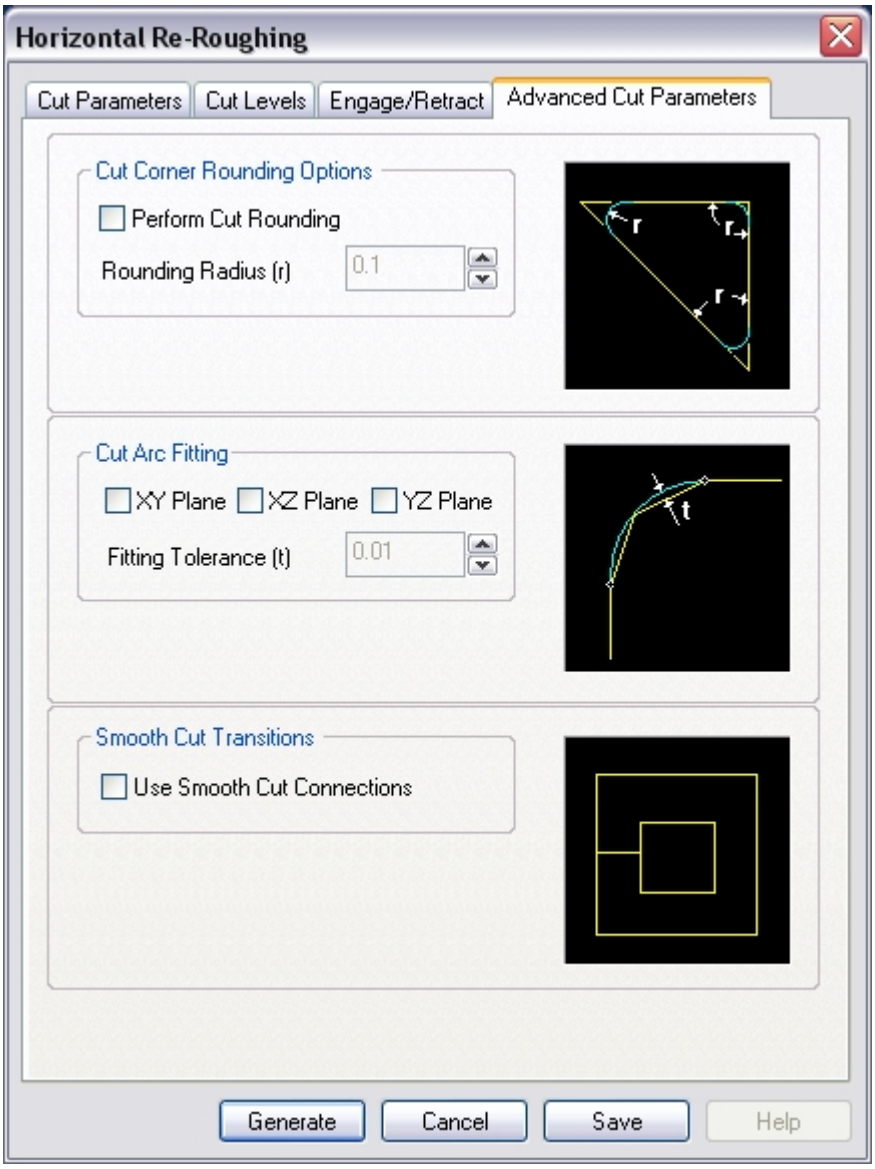
The Engage in Air section allows the user to define how the cutter would engage to start cutting when starting from outside of the part. This can happen when machining a core or performing a facing operation. In such cases the user has the option of specifying the cutter to start form the outside, a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

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.

Advance Cut Parameters

The advanced cut parameters are 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 you machine during the cutting process. These parameters allow smoothing of the toolpaths by introduction of arcs. You can use these parameters even if your controller does not support arcs. Make sure that your output is set to linear output. You can do this under the Machining Preferences dialog in the Preferences section of the Menu bar.

- **Corner Rounding:** This option is used to round sharp corners in the toolpath. The user can specify a rounding radius and 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 other in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.



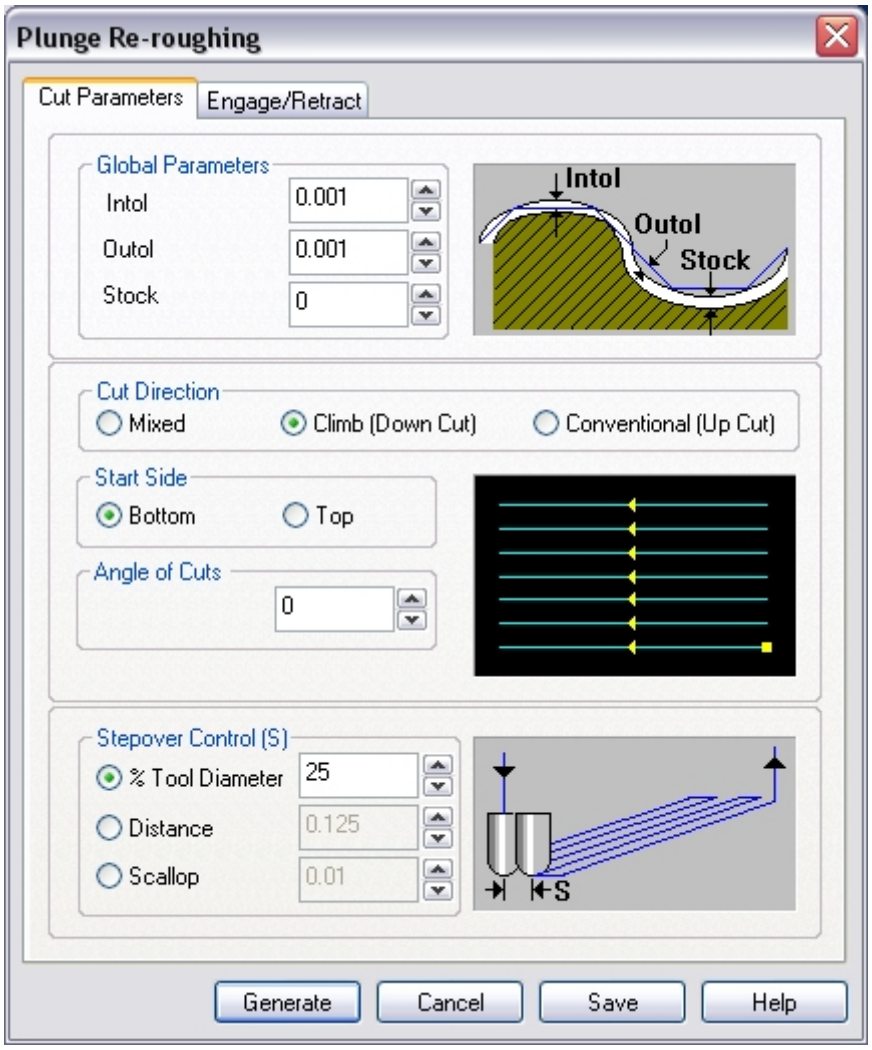
Plunge Re Roughing

Available in Alibre CAM Expert only.

As in Horizontal Re-roughing, Alibre CAM 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.

Description

The Plunge Re-Roughing toolpath is invoked by clicking on the button in the MOps Browser and picking the **3 Axis Milling** and **Plunge Re-Roughing** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



This dialog is a tabbed dialog with two tabs. Clicking on each of the tabs allows the user to set the different parameters. The following sections describe each of the tabs in more detail.

Cut Parameters

The user has the ability to set the Global Cut Parameters, the Cut Pattern, the Cut direction and the Stepover Control via this property page of the dialog.

- 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 are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any).

- 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.

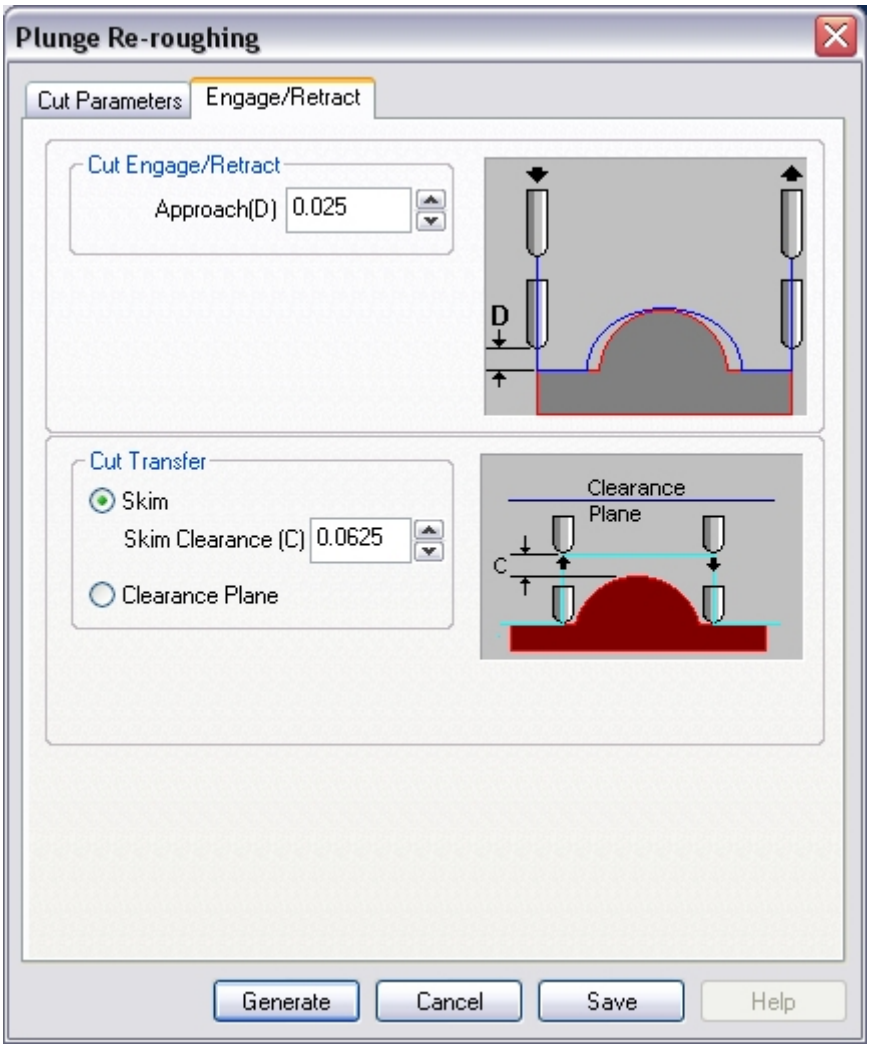
The cut direction can be controlled by specifying either climb or conventional or mixed.

The user can specify the cut start side to be either the bottom or top and the angle of cuts can be specified.

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

Then the user selects this tab in the dialog the following property page is displayed. The user has the ability to set the Engage/Retract parameters via this dialog.



The approach distance for Engage/Retract can be specified. 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.

Three (3) Axis Pocketing

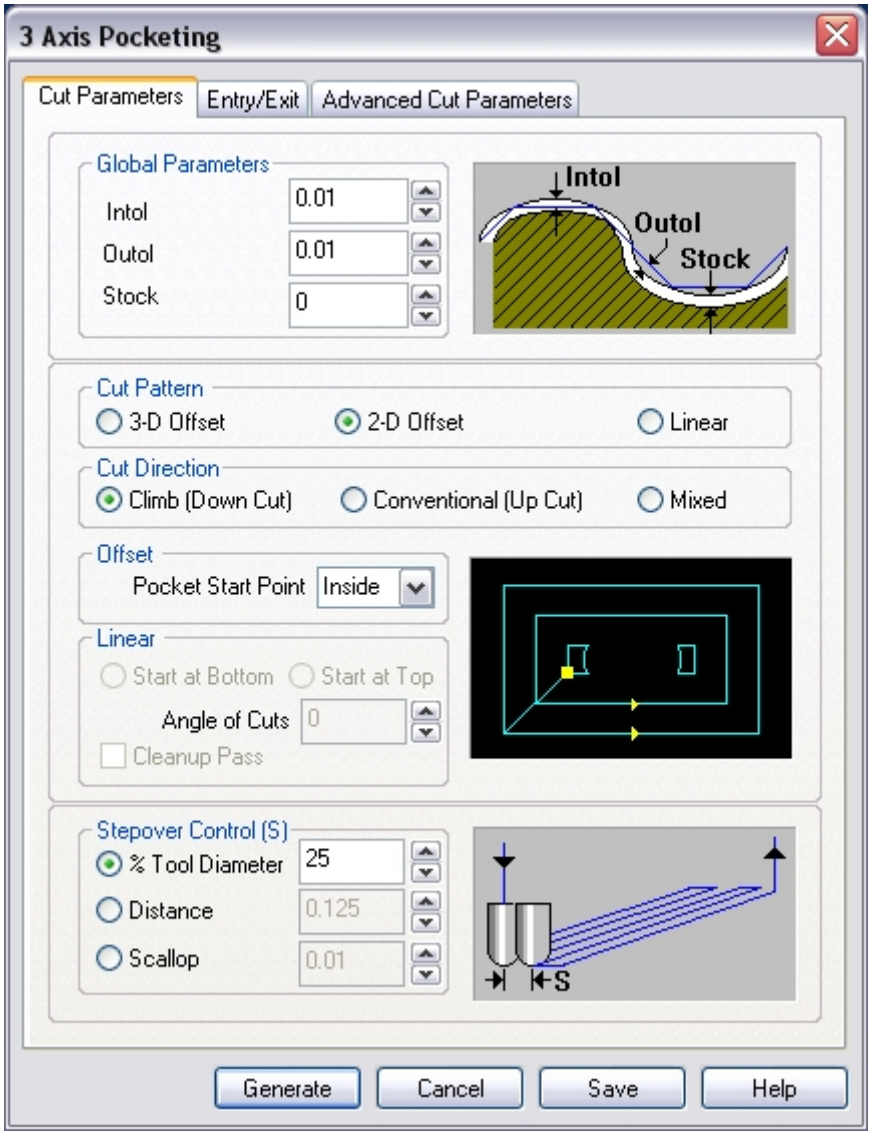
Available in Alibre CAM Expert only.

The 3 Axis Pocket Machining operation can be used for 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 nestings 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.

Description

The 3 axis Pocket Machining toolpath is invoked by clicking on the button in the MOps Browser and picking the **3 Axis Milling** and **3 Axis Pocketing** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



This dialog is a tabbed dialog with two tabs. Clicking on each of the tabs allows the user to set the different parameters. The following sections describe each of the tabs in more detail.

Cut Parameters

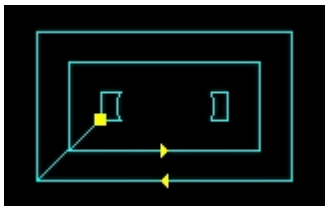
The user has the ability to set the Global Cut Parameters, the Cut Pattern, the Cut direction and the Stepover Control via this property page of the dialog.

- 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.

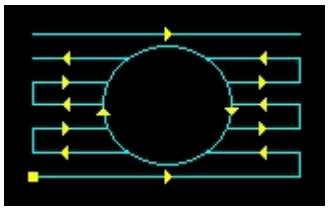
Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any).

- 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.

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(part offset) and the second one is a linear cut type.



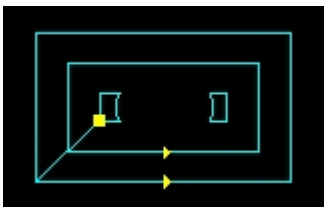
Part Offset



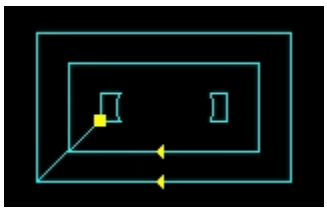
Linear

In the offset cut type, the cut regions are successively offset until there are 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.

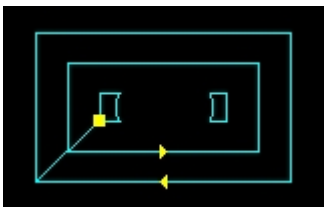
Cut direction can be controlled by specifying either climb or conventional or mixed.



Climb

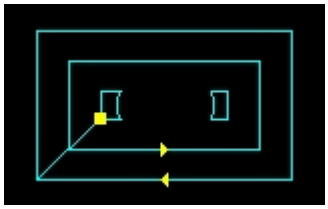


Conventional

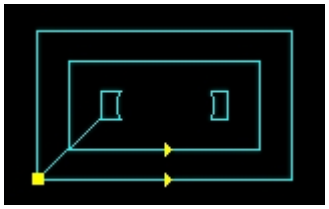


Mixed

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

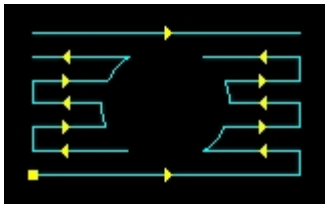


Inside

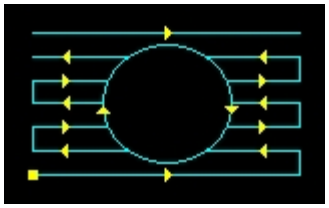


Outside

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.



Without Cleanup Pass



With Cleanup Pass

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

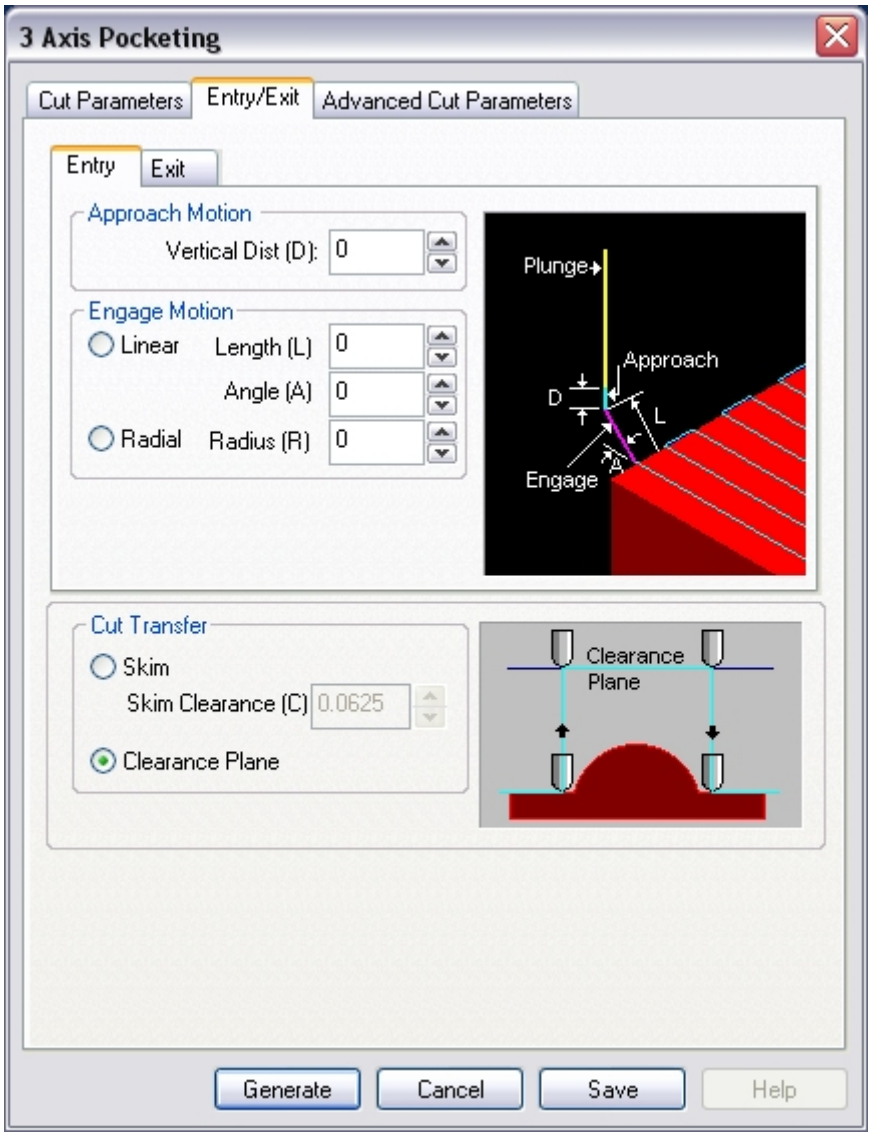
This entry/exit page is invoked in various 3 axis toolpath methods. This page controls the entry and exit motions in a vertical plane.

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

Entry
The user can choose to enter the material to be cut with a combination of two motions. These motions are Approach and Engage. The Approach motion is the motion as the cutter traverses from the transfer or plunge motion and meets the Engage motion. The Engage motion is the last motion before the cutter begins to cut material. The user has the ability to define the Approach motion as a vertical distance above the start of the Engage motion. The user can define the Engage motion either as a Linear ramp or Radial ramp motion as the graphic in the dialog indicates.

Exit
The user can choose to exit the cutting also as a combination of two motions. These motions are Retract and Departure. The Retract motion is the first motion after the

cutter loses contact with the material to be cut. The Departure motion is the motion that bridges the Retract motion and the transfer motion. The Retract motion can be defined either as a Linear ramp or Radial ramp motion as the graphic in the dialog indicates. The user has the ability to define the Departure motion as a vertical distance above the end of the Retract motion.

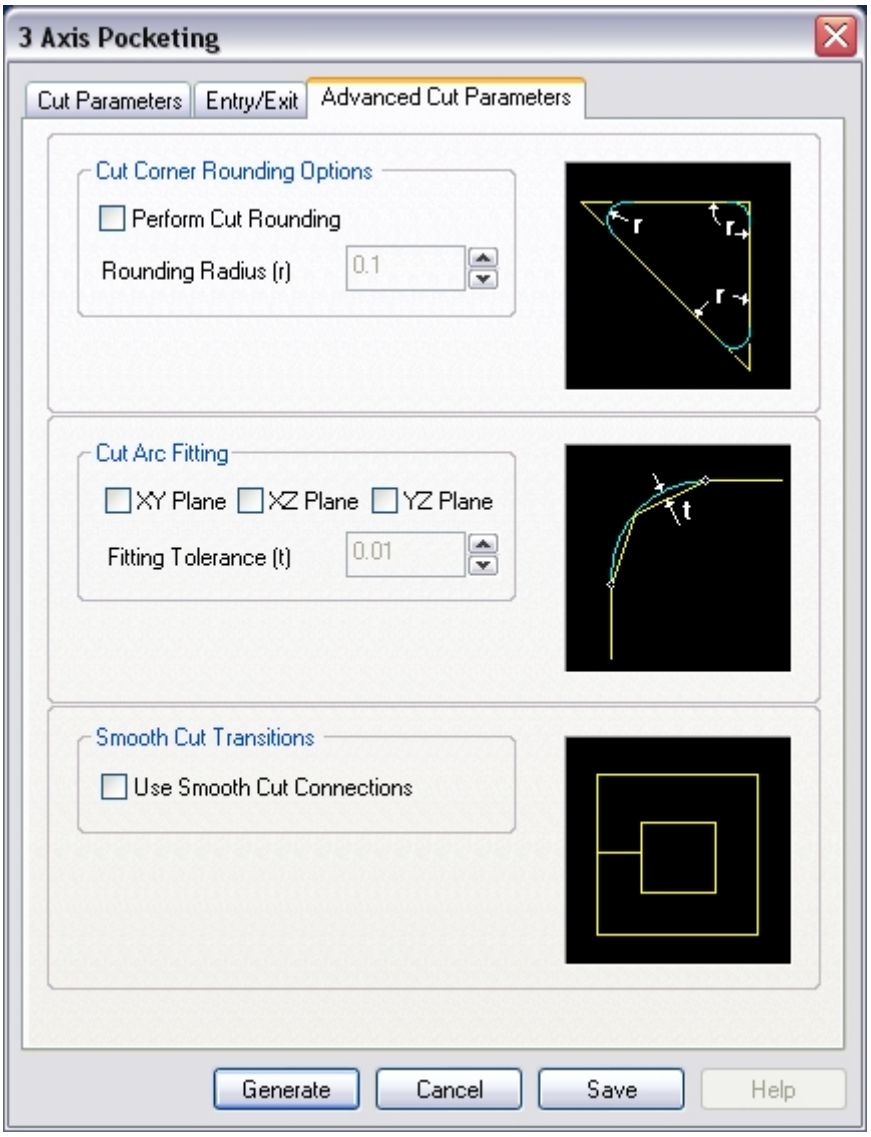


The user can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can either be instructed to move to the clearance plane and then perform the transfer motion to the next cut location or it could 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.

Advance Cut Parameters

The advanced cut parameters are 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 you machine during the cutting process. These parameters allow smoothing of the toolpaths by introduction of arcs. You can use these parameters even if your controller does not support arcs. Make sure that your output is set to linear output. You can do this under the Machining Preferences dialog in the Preferences section of the Menu bar.

- **Corner Rounding:** This option is used to round sharp corners in the toolpath. The user can specify a rounding radius and 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 other in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.



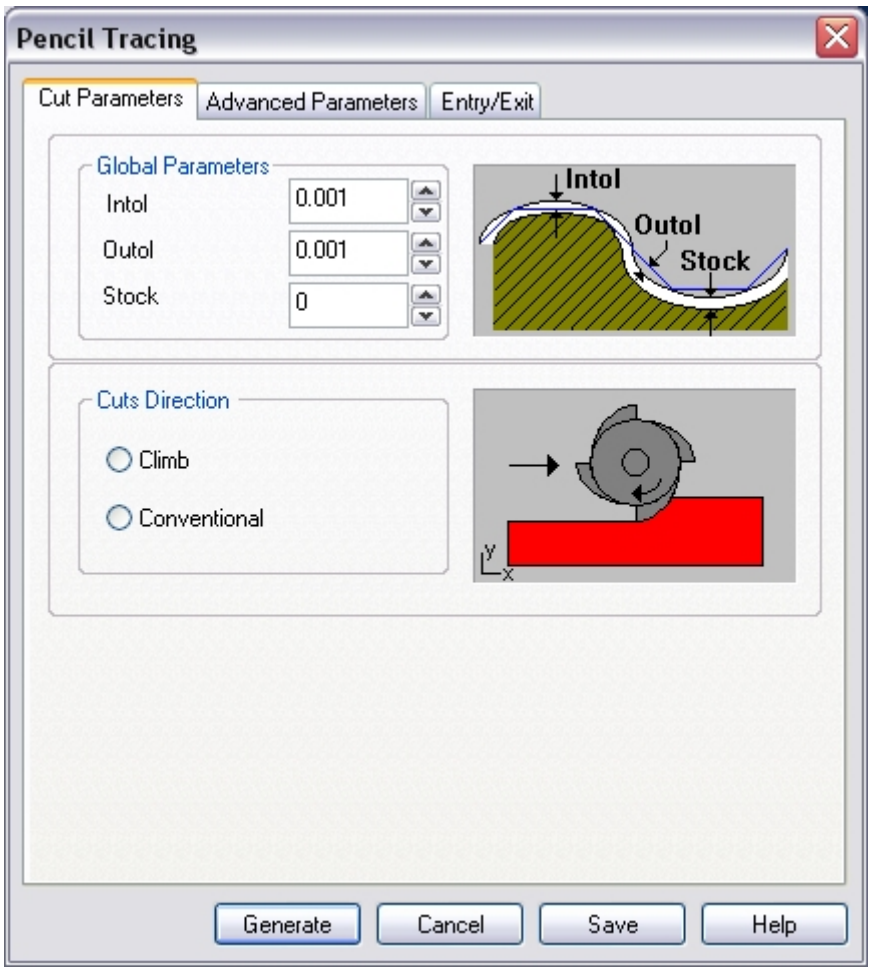
Pencil Tracing

Available in Alibre CAM Expert only.

Pencil Trace machining is a toolpath method which can be used either as a pre-finishing operation or as a re-finishing operation. In the cut method, a ball end cutter is typically used and the cutter is restricted to follow a path where two or more sides of the cutter are in simultaneous contact with the part. Such a path typically follows the valleys and the corners of the part. Thus, for re-finishing, this method can be used to cleanup the scallops left in a corner after a contour finishing operation. It can also be used as a pre-finishing operation to remove material from the valleys and corners before running a finish operation. This will prevent the cutter from running into material found in these valleys during the finish operation, thereby extending cutter life.

Description

The Pencil Tracing toolpath is invoked by clicking on the button in the MOps Browser and picking the **3 Axis Milling** and **Pencil Tracing** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



This dialog is a tabbed dialog with three tabs. Clicking on each of the tabs allows the user to set the different parameters. The following sections describe each of the tabs in more detail.

Cut Parameters

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

The Global Cut Parameters section allows the user to set the intol and the outol values to be used in machining. Also any uniform thickness or stock that needs to be left of the part also can be specified here.

- 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.

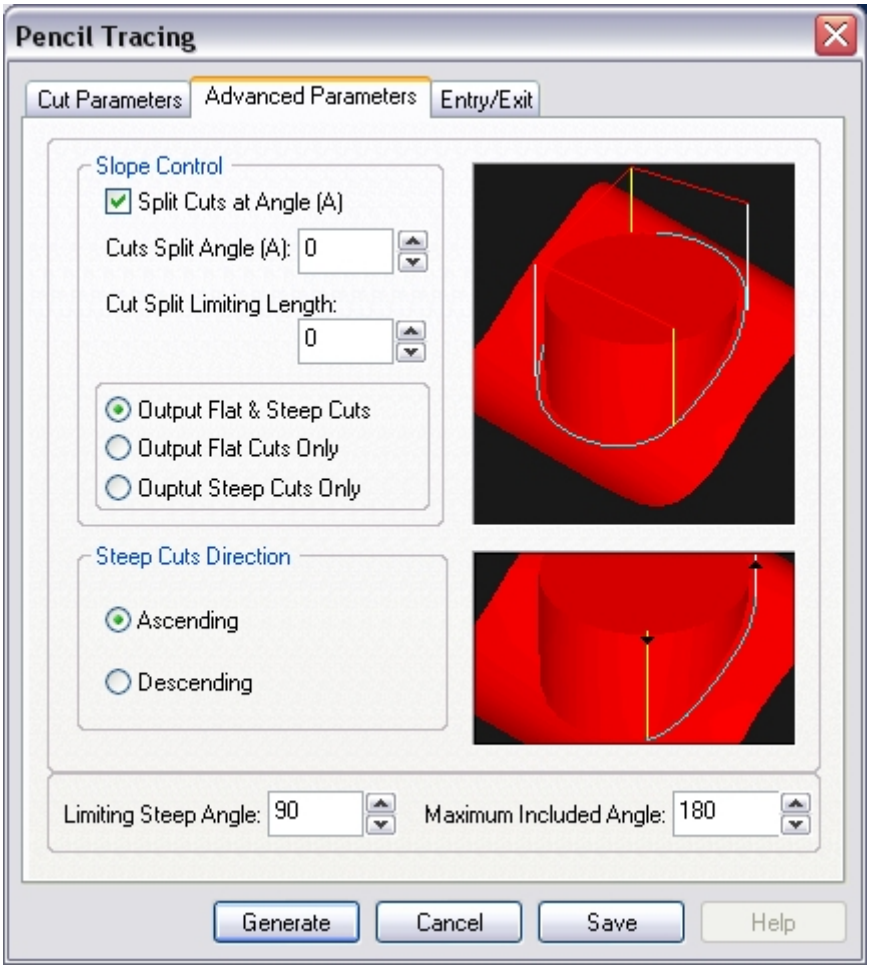
Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any).

- 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.

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

Advanced Cut Parameters

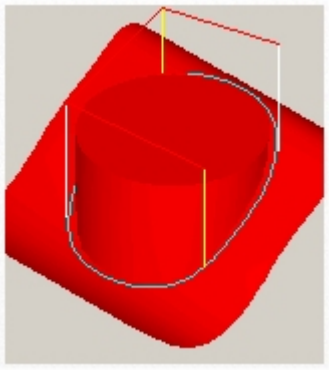
Clicking on the Advanced Cut Parameters will display the property page of the dialog.



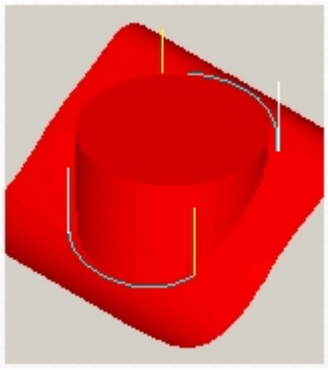
This section of the dialog is used to control the cuts based on the slope of the cuts.

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

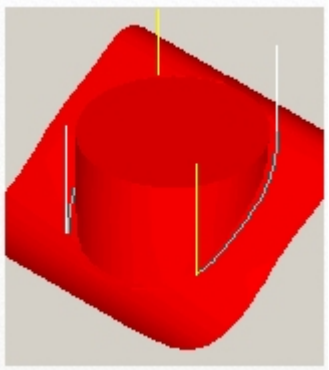
- Cuts Split Angle - The cuts will be examined and split at this specified angle. This angle is computed with respect to the XY plane
- Cut Split Limiting Length - In some cases the split angle could cause small cut segments to be output. This can happen when the pencil trace cuts trajectories bounce 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
- 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

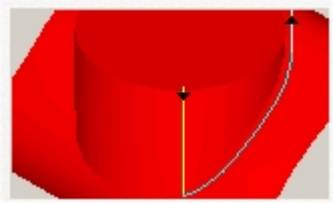


Flat Cuts Only

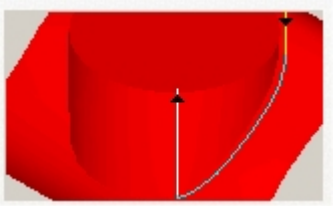


Steep Cuts Only

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.



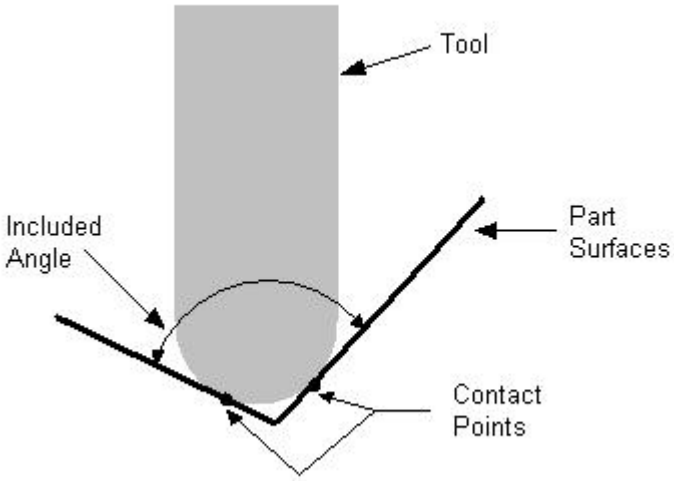
Ascending



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

This entry/exit page is invoked in various 3 axis toolpath methods. This page controls the entry and exit motions in a vertical plane.

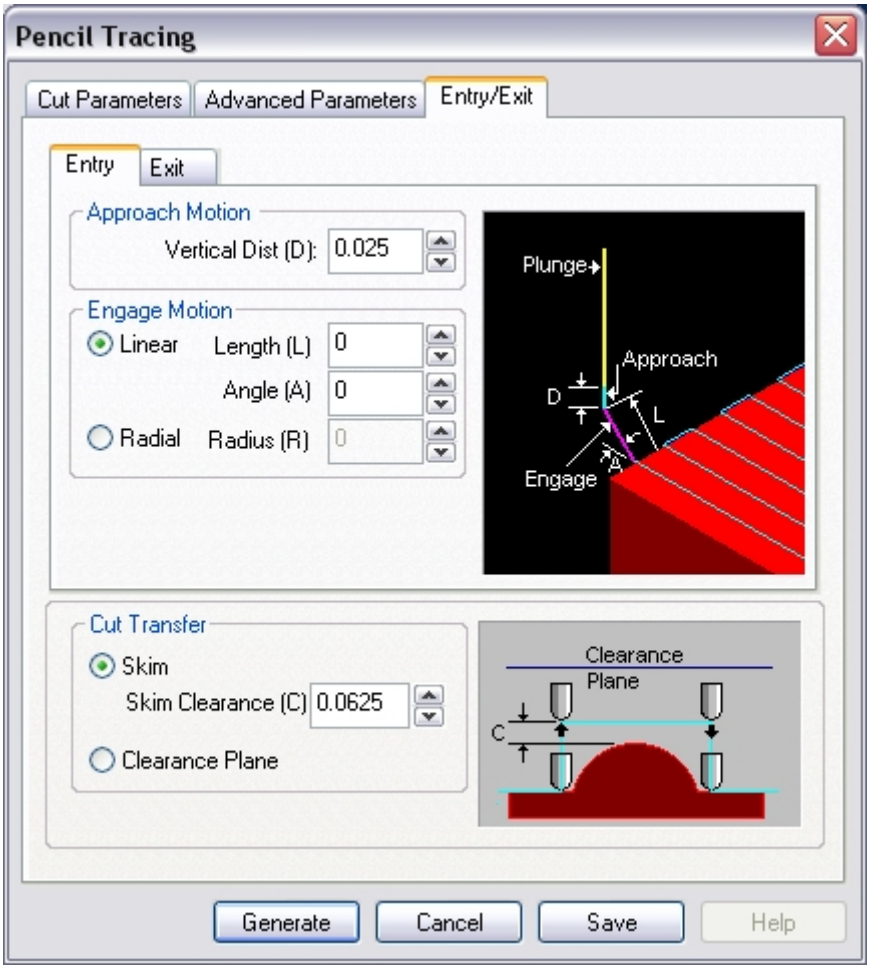
The Cut Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

Entry

The user can choose to enter the material to be cut with a combination of two motions. These motions are Approach and Engage. The Approach motion is the motion as the cutter traverses from the transfer or plunge motion and meets the Engage motion. The Engage motion is the last motion before the cutter begins to cut material. The user has the ability to define the Approach motion as a vertical distance above the start of the Engage motion. The user can define the Engage motion either as a Linear ramp or Radial ramp motion as the graphic in the dialog indicates.

Exit

The user can choose to exit the cutting also as a combination of two motions. These motions are Retract and Departure. The Retract motion is the first motion after the cutter loses contact with the material to be cut. The Departure motion is the motion that bridges the Retract motion and the transfer motion. The Retract motion can be defined either as a Linear ramp or Radial ramp motion as the graphic in the dialog indicates. The user has the ability to define the Departure motion as a vertical distance above the end of the Retract motion.



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.

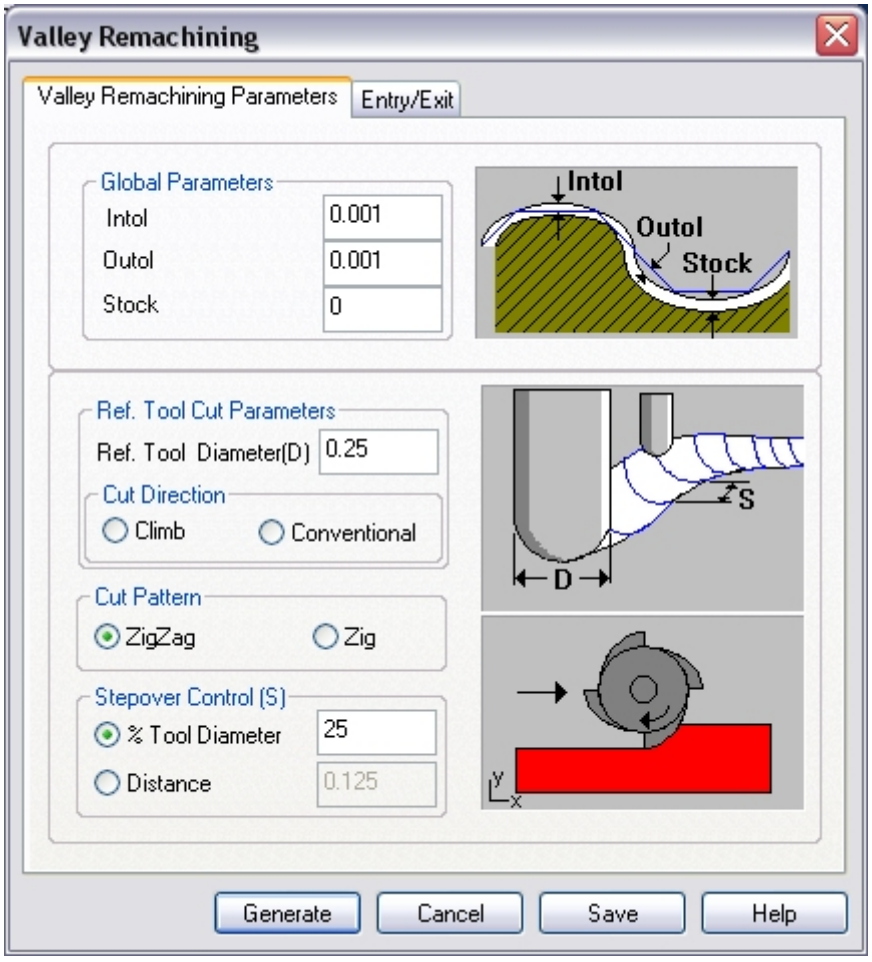
Valley Re-machining

Available in Alibre CAM Expert only.

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.

Description

The Valley Re-Machining toolpath is invoked by clicking on the button in the MOps Browser and picking the **3 Axis Milling** and **Valley Re-Machining** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



This dialog is a tabbed dialog with two tabs. Clicking on each of the tabs allows the user to set the different parameters. The following sections describe each of the tabs in more detail.

Cut Parameters

The user has the ability to set the Global Cut Parameters, the Reference Tool Cut Parameters, Cut Pattern and Stepover Control via this property page of the dialog.

The Global Cut Parameters section allows the user to set the intol and the outol values to be used in machining. Also any uniform thickness or stock that needs to be left of the part also can be specified here.

- 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 are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any).

- 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.

The reference tool section allows the user to specify the reference tool diameter. This diameter must be larger than the current tool diameter. Also the cutting direction for the reference tool is specified as Climb/Conventional.

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 one end of the region to the other in a linear fashion. In the Zig cut pattern the cutter traverses from one end of the region to the other in a linear pattern also, but once reaching the end of the cut picks up and returns to the starting point of the next cut.

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 or a specific distance.

Entry/Exit

This entry/exit page is invoked in various 3 axis toolpath methods. This page controls the entry and exit motions in a vertical plane.

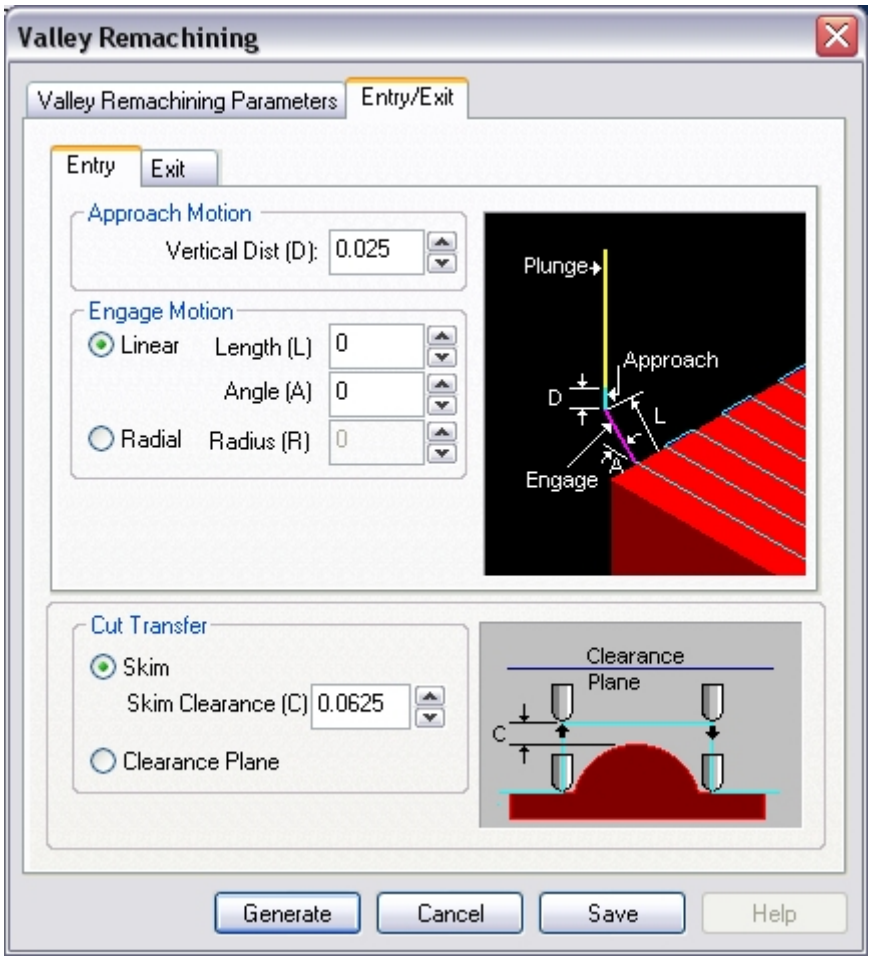
The Cut Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

Entry
The user can choose to enter the material to be cut with a combination of two motions. These motions are Approach and Engage. The Approach motion is the

motion as the cutter traverses from the transfer or plunge motion and meets the Engage motion. The Engage motion is the last motion before the cutter begins to cut material. The user has the ability to define the Approach motion as a vertical distance above the start of the Engage motion. The user can define the Engage motion either as a Linear ramp or Radial ramp motion as the graphic in the dialog indicates.

Exit

The user can choose to exit the cutting also as a combination of two motions. These motions are Retract and Departure. The Retract motion is the first motion after the cutter loses contact with the material to be cut. The Departure motion is the motion that bridges the Retract motion and the transfer motion. The Retract motion can be defined either as a Linear ramp or Radial ramp motion as the graphic in the dialog indicates. The user has the ability to define the Departure motion as a vertical distance above the end of the Retract motion.



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.

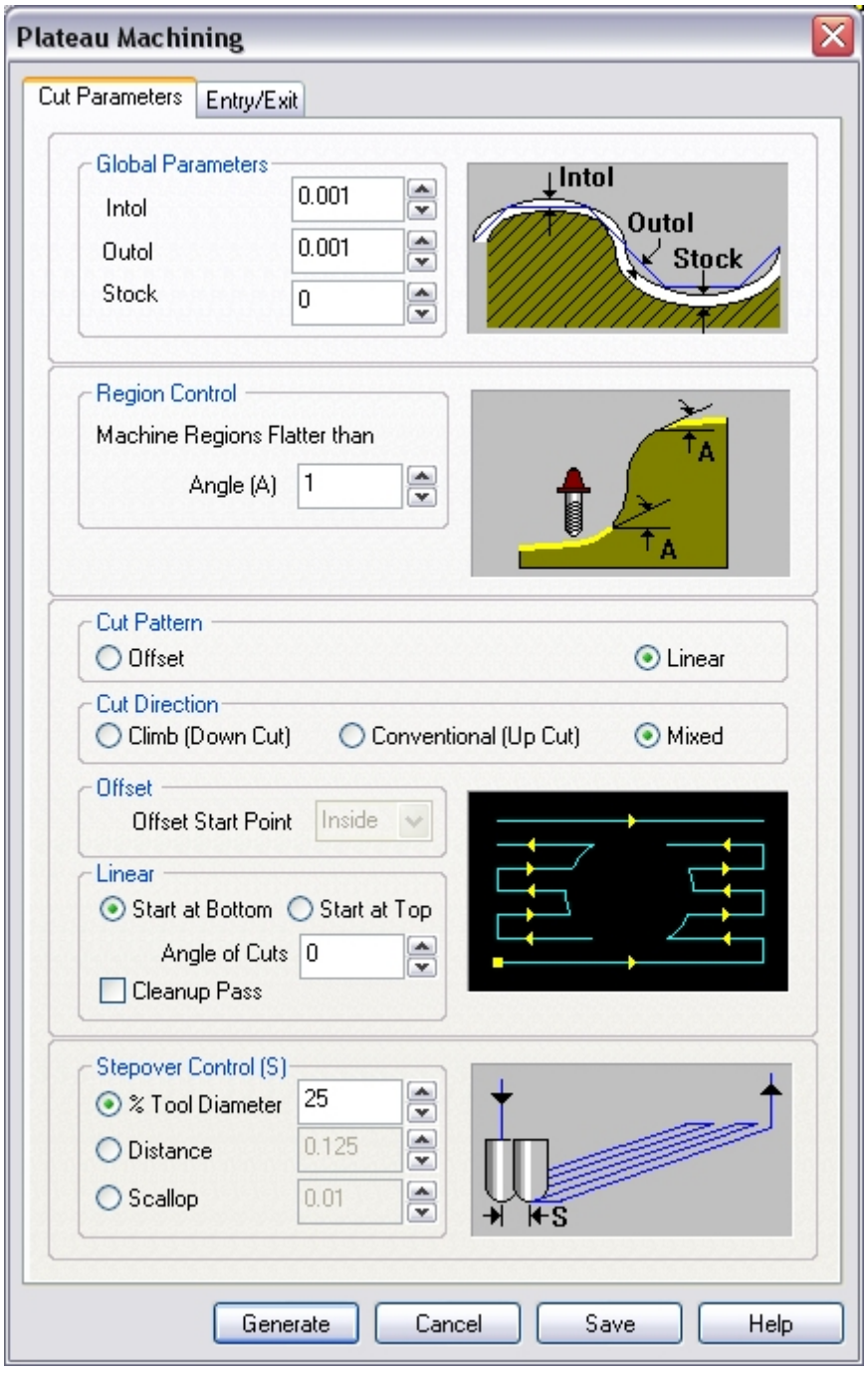
Plateau Machining

Available in Alibre CAM Expert only.

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.

Description

The Plateau Machining toolpath is invoked by clicking on the  button in the MOps Browser and picking the **3 Axis Milling** and **Plateau Machining** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



This dialog is a tabbed dialog with two tabs. Clicking on each of the tabs allows the user to set the different parameters. The following sections describe each of the tabs in more detail.

Cut Parameters

The user has the ability to set the Global Cut Parameters, the Region Control, the Cut Pattern and Stepover Control via this property page.

The Global Cut Parameters section allows the user to set the intol and the outol values to be used in machining. Also any uniform thickness or stock that needs to be left of the part also can be specified here.

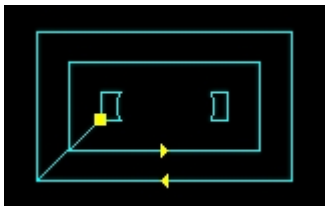
- Stock: The thickness of the layer that will remain on top of the part after the toolpath is complete. Roughing operations generally leave a thin layer of stock, but for finishing operations this value is zero.

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

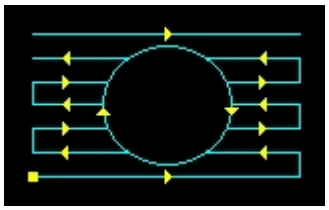
- Intol: Inward tolerance - the maximum thickness of material that can be removed from the Stock layer.
- Outol: Outward tolerance - the maximum thickness of material that can remain above the Stock layer.

The Region Control section 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 (part offset) and the second one is a linear cut type.

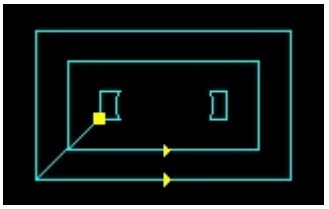


Part Offset

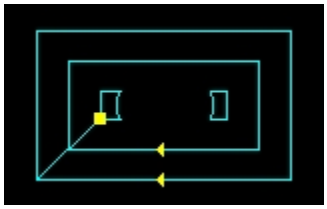


Linear

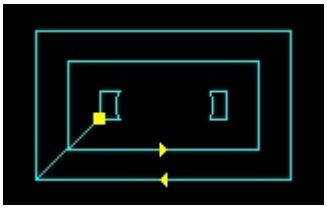
In the offset cut type, the cut regions are successively offset until there are nothing left to machine. This type of cutting is sometimes called spiral machining. The user can specify the cut start point to be either the inside of the cut regions or from the outside.



Climb



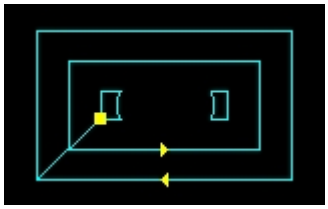
Conventional



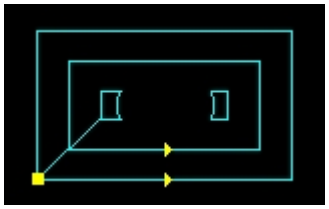
Mixed

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.

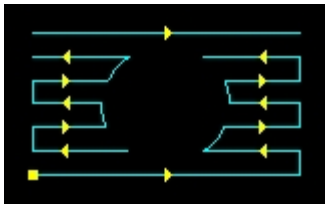


Inside

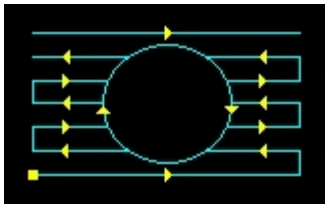


Outside

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.



Without Cleanup Pass



With Cleanup Pass

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

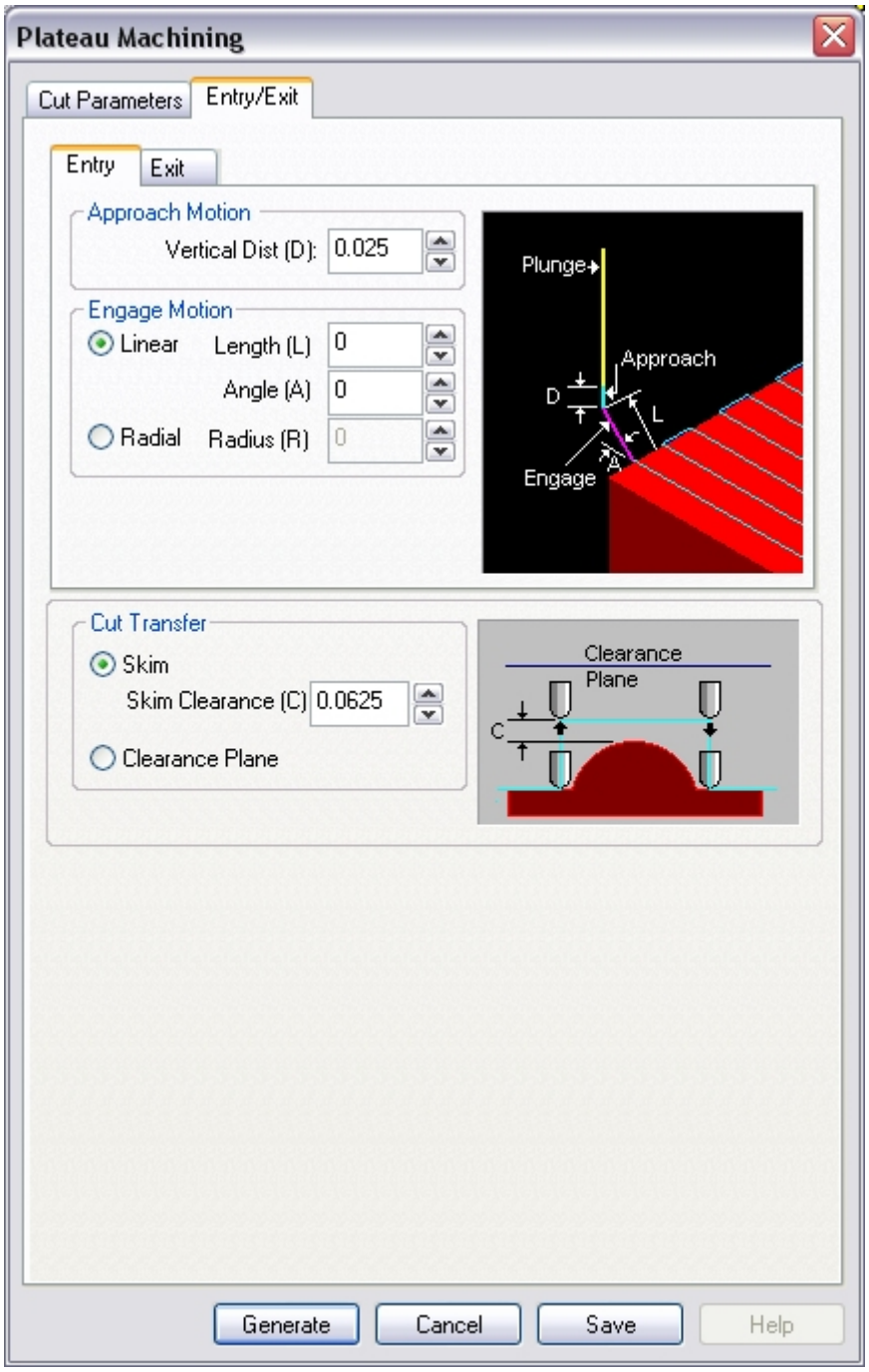
This entry/exit page is invoked in various 3 axis toolpath methods. This page controls the entry and exit motions in a vertical plane.

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

Entry
The user can choose to enter the material to be cut with a combination of two motions. These motions are Approach and Engage. The Approach motion is the motion as the cutter traverses from the transfer or plunge motion and meets the Engage motion. The Engage motion is the last motion before the cutter begins to cut material. The user has the ability to define the Approach motion as a vertical distance above the start of the Engage motion. The user can define the Engage motion either as a Linear ramp or Radial ramp motion as the graphic in the dialog indicates.

Exit
The user can choose to exit the cutting also as a combination of two motions. These motions are Retract and Departure. The Retract motion is the first motion after the

cutter loses contact with the material to be cut. The Departure motion is the motion that bridges the Retract motion and the transfer motion. The Retract motion can be defined either as a Linear ramp or Radial ramp motion as the graphic in the dialog indicates. The user has the ability to define the Departure motion as a vertical distance above the end of the Retract motion.



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.

Parallel Hill Machining

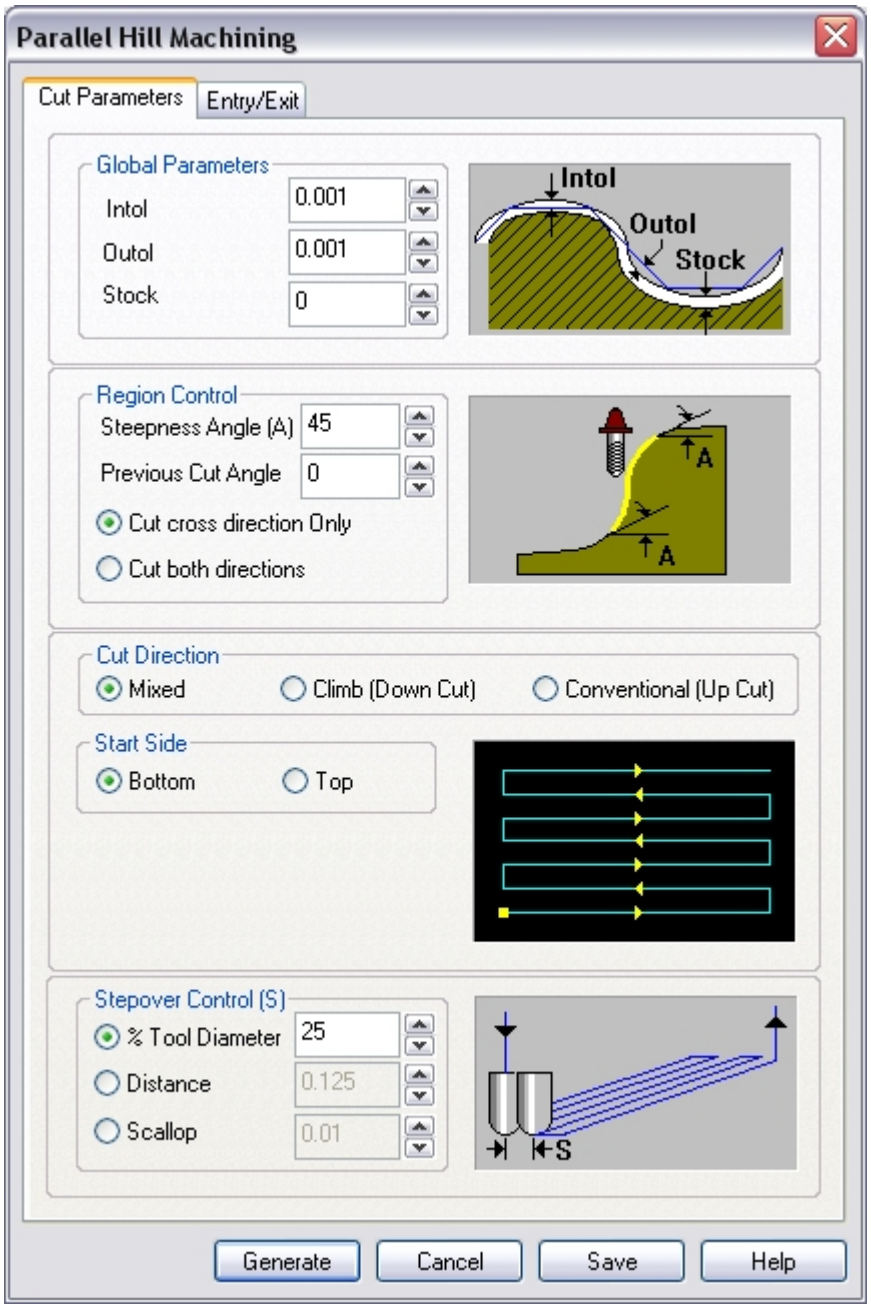
Available in Alibre CAM Expert only.

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.

Description

The Parallel Hill Machining toolpath is invoked by clicking on the  button in the MOps Browser and picking the **3 Axis Milling** and **Parallel Hill Machining** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



This dialog is a tabbed dialog with two tabs. Clicking on each of the tabs allows the user to set the different parameters. The following sections describe each of the tabs in more detail.

Cut Parameters

The user has the ability to set the Global Cut Parameters, the Region Control, the Cut Pattern and Stepover Control via this property page.

The Global Cut Parameters section allows the user to set the intol and the outol values to be used in machining. Also any uniform thickness or stock that needs to be left of the part also can be specified here.

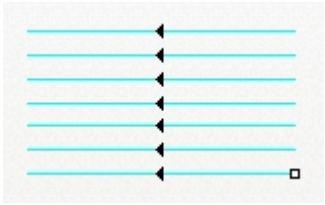
- Stock: The thickness of the layer that will remain on top of the part after the toolpath is complete. Roughing operations generally leave a thin layer of stock, but for finishing operations this value is zero.

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

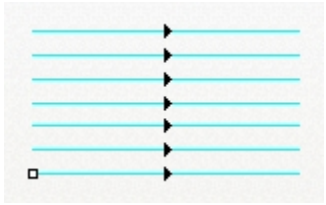
- Intol: Inward tolerance - the maximum thickness of material that can be removed from the Stock layer.
- Outol: Outward tolerance - the maximum thickness of material that can remain above the Stock layer.

The Region 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 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.

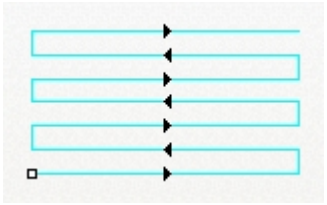
The cut direction can be controlled by specifying either climb or conventional or mixed.



Climb

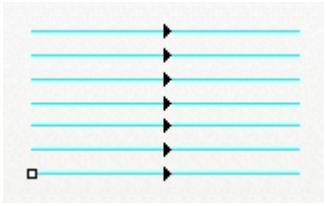


Conventional

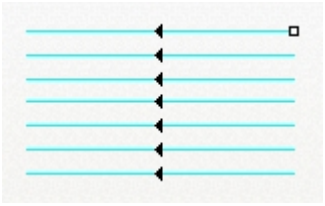


Mixed

The start side can be specified as top or bottom.



Start at Bottom



Start at Top

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.

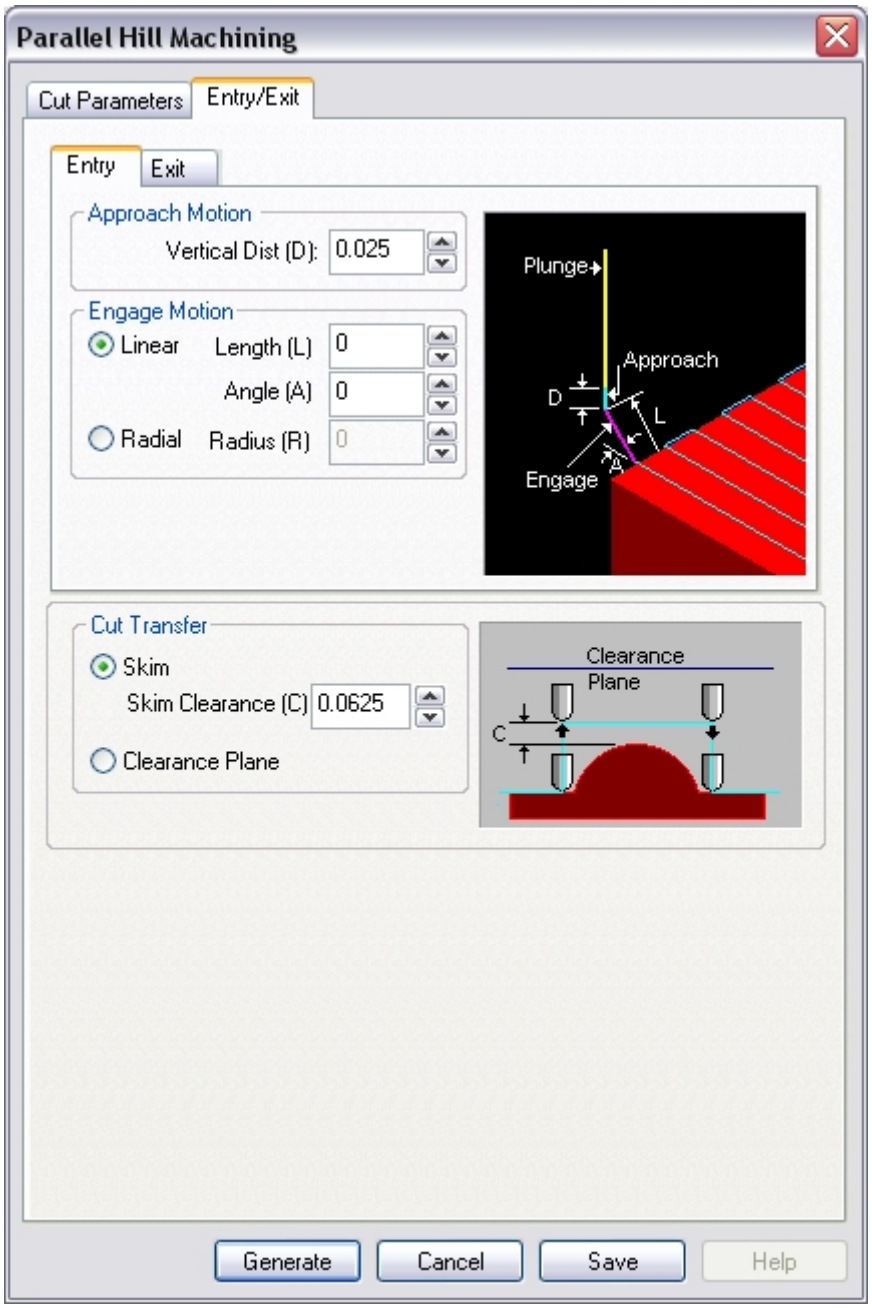
Entry / Exit

This entry/exit page is invoked in various 3 axis toolpath methods. This page controls the entry and exit motions in a vertical plane.

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

Entry
The user can choose to enter the material to be cut with a combination of two motions. These motions are Approach and Engage. The Approach motion is the motion as the cutter traverses from the transfer or plunge motion and meets the Engage motion. The Engage motion is the last motion before the cutter begins to cut material. The user has the ability to define the Approach motion as a vertical distance above the start of the Engage motion. The user can define the Engage motion either as a Linear ramp or Radial ramp motion as the graphic in the dialog indicates.

Exit
The user can choose to exit the cutting also as a combination of two motions. These motions are Retract and Departure. The Retract motion is the first motion after the cutter loses contact with the material to be cut. The Departure motion is the motion that bridges the Retract motion and the transfer motion. The Retract motion can be defined either as a Linear ramp or Radial ramp motion as the graphic in the dialog indicates. The user has the ability to define the Departure motion as a vertical distance above the end of the Retract motion.



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.

Horizontal Hill Machining

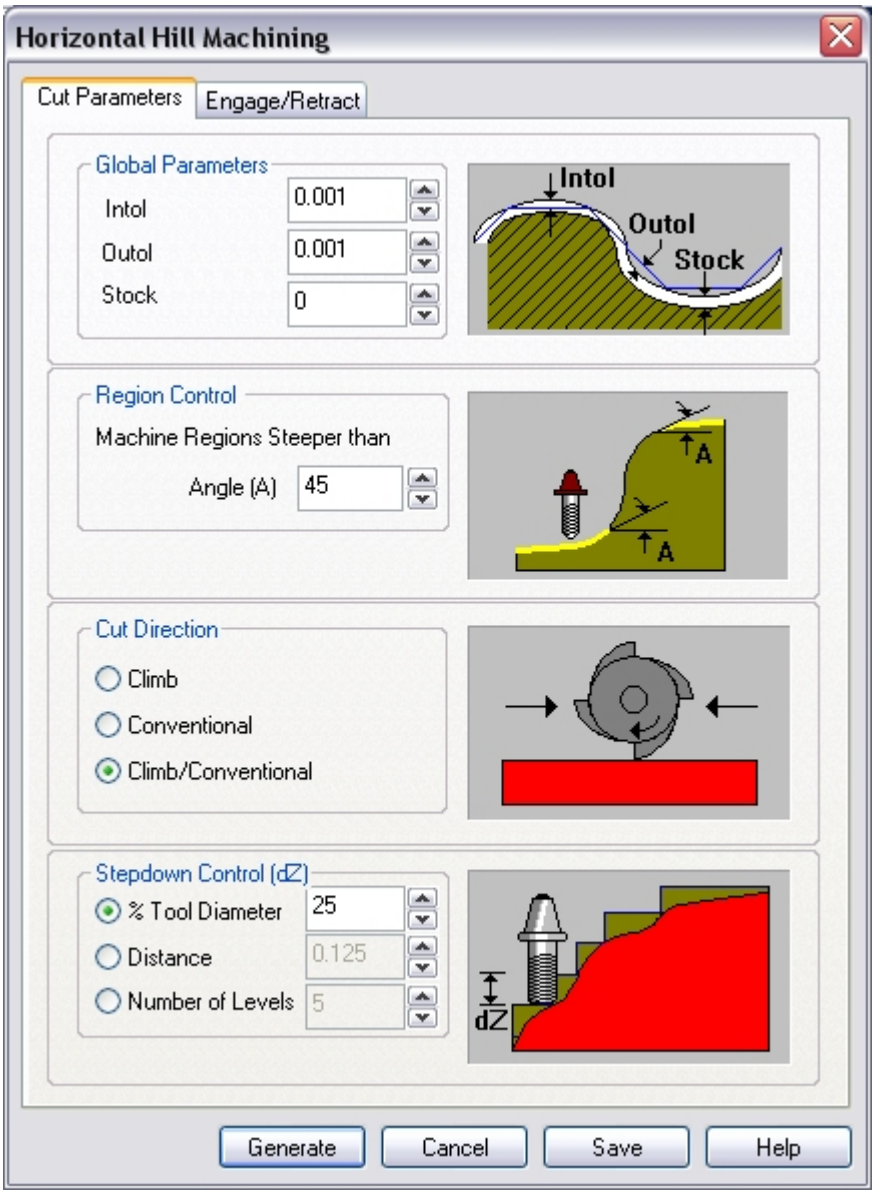
Available in Alibre CAM Expert only.

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.

Description

The Horizontal Hill Machining toolpath is invoked by clicking on the  button in the MOps Browser and picking the **3 Axis Milling** and **Horizontal Hill Machining** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



Cut Parameters

The user has the ability to set the Global Cut Parameters, the Region Control, the Cut Direction and Stepdown Control via this property page.

The Global Cut Parameters section allows the user to set the intol and the outol values to be used in machining. Also any uniform thickness or stock that needs to be left of the part also can be specified here.

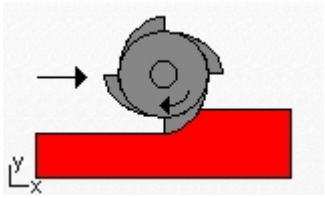
- Stock: The thickness of the layer that will remain on top of the part after the toolpath is complete. Roughing operations generally leave a thin layer of stock, but for finishing operations this value is zero.

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

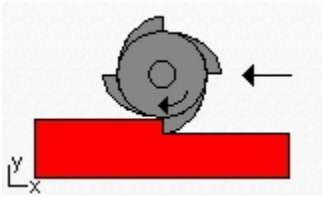
- Intol: Inward tolerance - the maximum thickness of material that can be removed from the Stock layer.
- Outol: Outward tolerance - the maximum thickness of material that can remain above the Stock layer.

The Region 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 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.

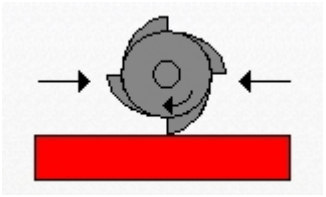
The Cut Direction of the toolpath can be specified either as Climb, Conventional or Climb/Conventional by selecting the appropriate radio buttons. As mentioned before, in climb or conventional, the direction of cutting is maintained so as the corresponding cutting condition is maintained on the part.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



Climb Cut



Conventional Cut

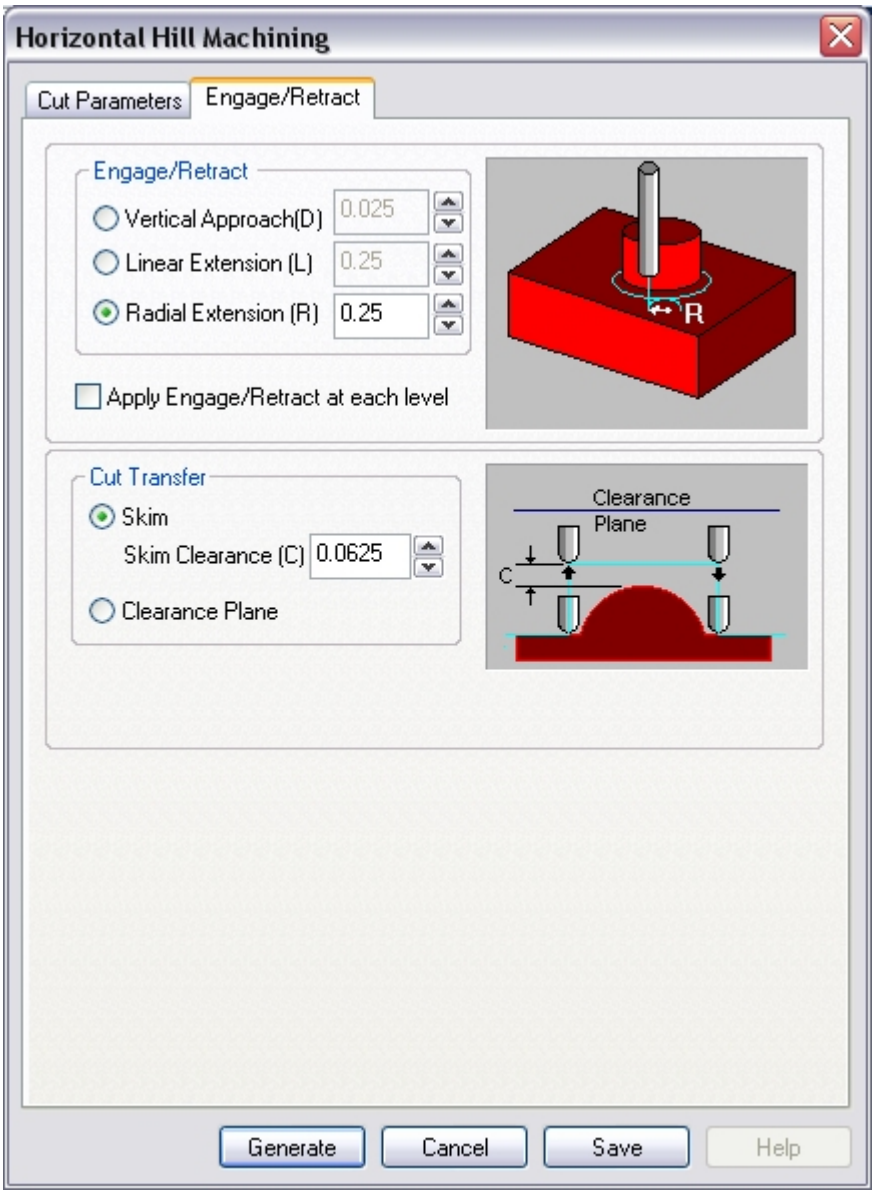


Climb/Conventional Cut

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.

Engage/Retract

Then the user selects the Engage/Retract tab the following property page is displayed. The user will be able to specify how the cutter engages and retracts when starting and stopping a cut. The user can also specify the type of transfer motions to perform while cutting.



The user can optionally pick a vertical engage/retract in which case the tool engages into the first cut start point vertically from above and switches feedrate to cut feed rate the specified distance above the first cut point. In the case of a linear extension, the cuts are linearly extended by the specified distance at an angle of 45 degrees to the tangent at the start and end of cut. In the case of radial extensions, the cuts are radially extended by a quarter circle of specified radius from the cut start and end points.

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.

Radial Machining

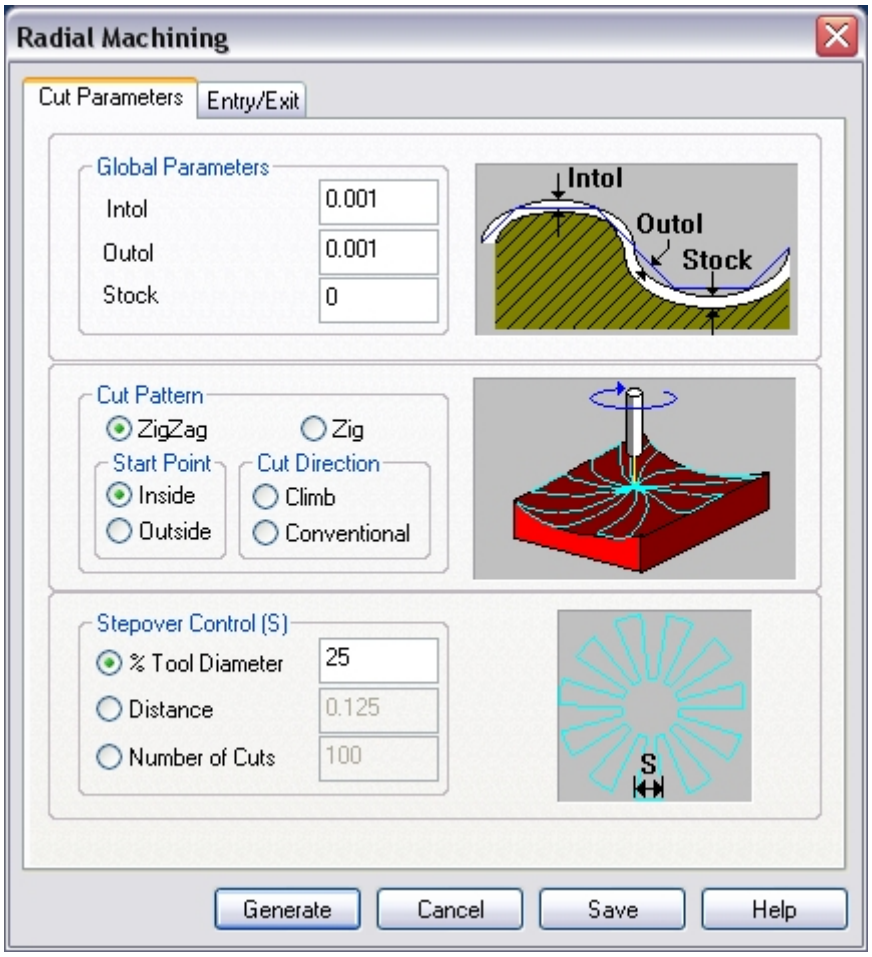
Available in Alibre CAM Expert only.

Radial Finishing is a toolpath method, which 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 system calculated optimum 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 nestings 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 start point of the cuts can also be changed from outside to inside or inside to outside.

Description

The Radial Machining toolpath is invoked by clicking on the  button in the MOps Browser and picking the **3 Axis Milling** and **Radial Machining** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



This dialog is a tabbed dialog with two tabs. Clicking on each of the tabs allows the user to set the different parameters. The following sections describe each of the tabs in more detail.

Cut Parameters

The cut parameters are set in the property page shown above. The user has the ability to set the Global Cut Parameters, the Cut Directional via this property page of the dialog.

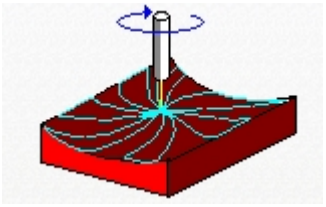
The Global Cut Parameters section allows the user to set the intol and the outol values to be used in machining. Also any uniform thickness or stock that needs to be left of the part also can be specified here.

- 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.

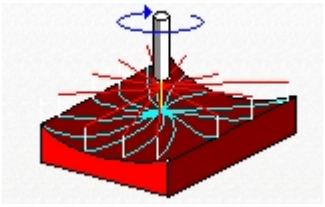
Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any).

- 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.

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.

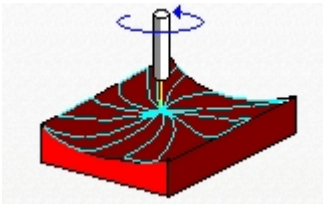


ZigZag

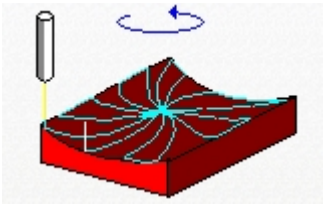


Zig

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.

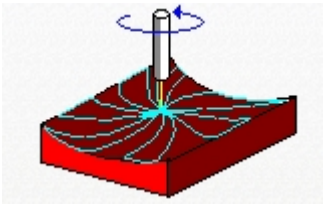


Start - Inside

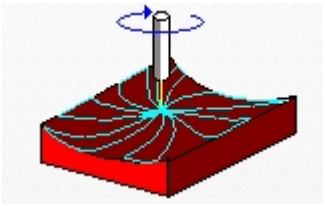


Start - Outside

The user can control the **cut direction** as either climb or conventional.



Climb



Conventional

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.

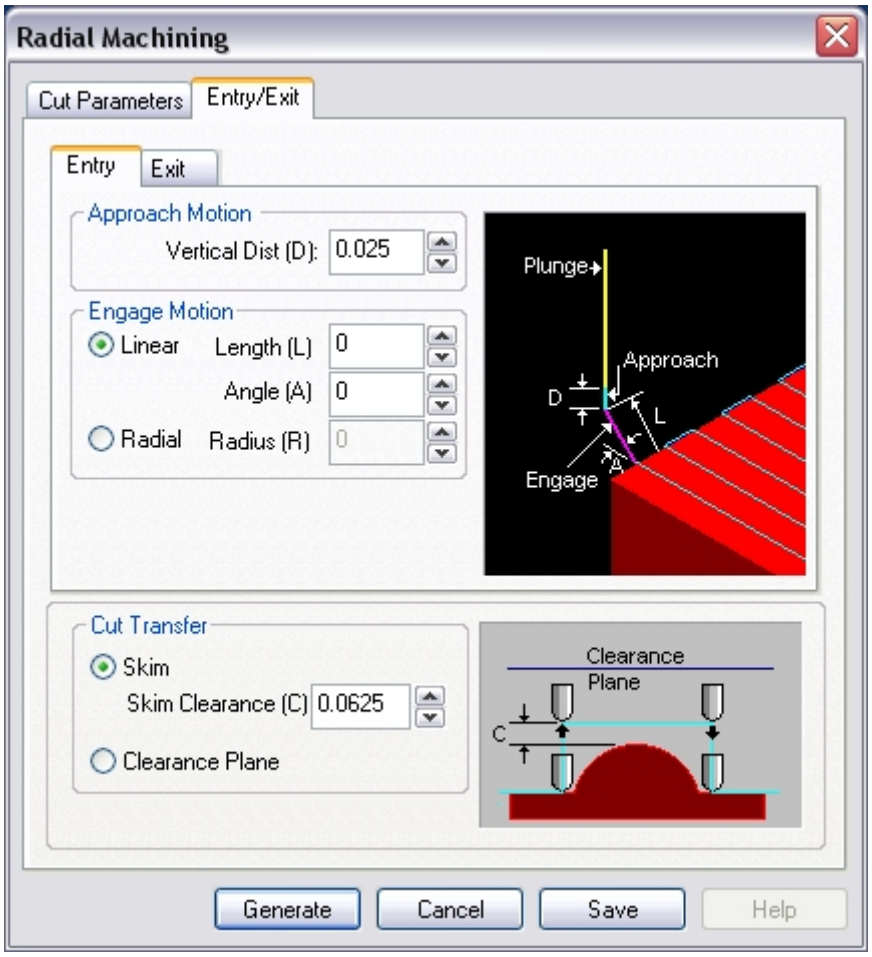
Entry / Exit

This entry/exit page is invoked in various 3 axis toolpath methods. This page controls the entry and exit motions in a vertical plane.

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

Entry
The user can choose to enter the material to be cut with a combination of two motions. These motions are Approach and Engage. The Approach motion is the motion as the cutter traverses from the transfer or plunge motion and meets the Engage motion. The Engage motion is the last motion before the cutter begins to cut material. The user has the ability to define the Approach motion as a vertical distance above the start of the Engage motion. The user can define the Engage motion either as a Linear ramp or Radial ramp motion as the graphic in the dialog indicates.

Exit
The user can choose to exit the cutting also as a combination of two motions. These motions are Retract and Departure. The Retract motion is the first motion after the cutter loses contact with the material to be cut. The Departure motion is the motion that bridges the Retract motion and the transfer motion. The Retract motion can be defined either as a Linear ramp or Radial ramp motion as the graphic in the dialog indicates. The user has the ability to define the Departure motion as a vertical distance above the end of the Retract motion.



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.

Spiral Machining

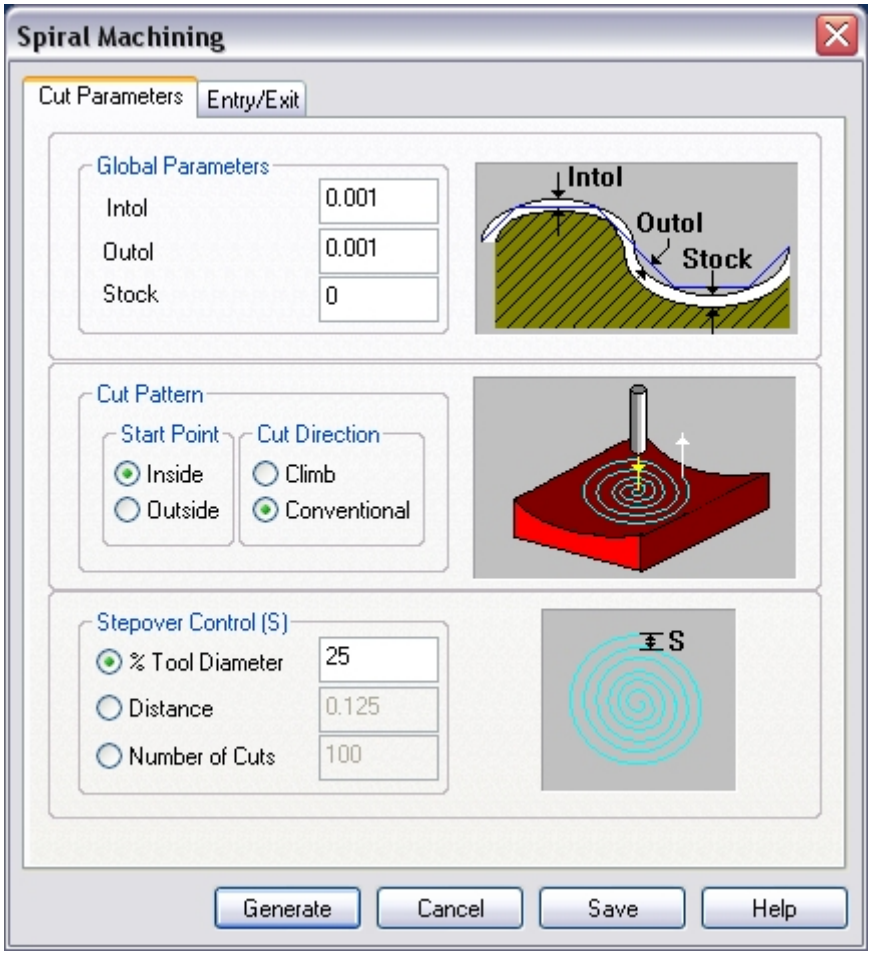
Available in Alibre CAM Expert only.

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.

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.

Description

The Spiral Machining toolpath is invoked by clicking on the button in the MOps Browser and picking the **3 Axis Milling** and **Spiral Machining** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



This dialog is tabbed with two tabs. Each tab defines a set a parameters that the user can specify. The sections below describe them in detail.

Cut Parameters

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

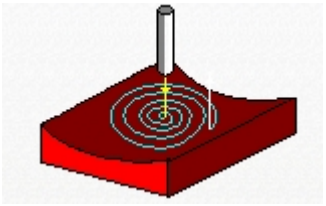
The Global Cut Parameters section allows the user to set the intol and the outol values to be used in machining. Also any uniform thickness or stock that needs to be left of the part also can be specified here.

- 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.

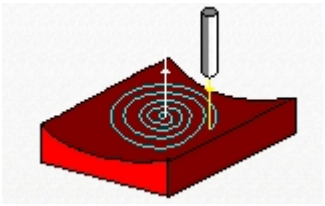
Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any).

- 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.

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.

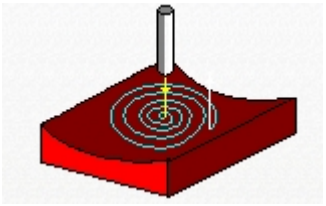


Start - Inside

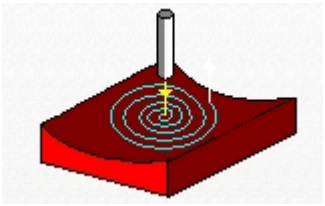


Start - Outside

The direction of the cutter can be set to either Climb or Conventional. If the user chooses the Climb operation the toolpath is generated in the counter-clockwise direction. If the conventional mode is chosen, machining will start at the same point but in the clockwise direction.



Climb



Conventional

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.

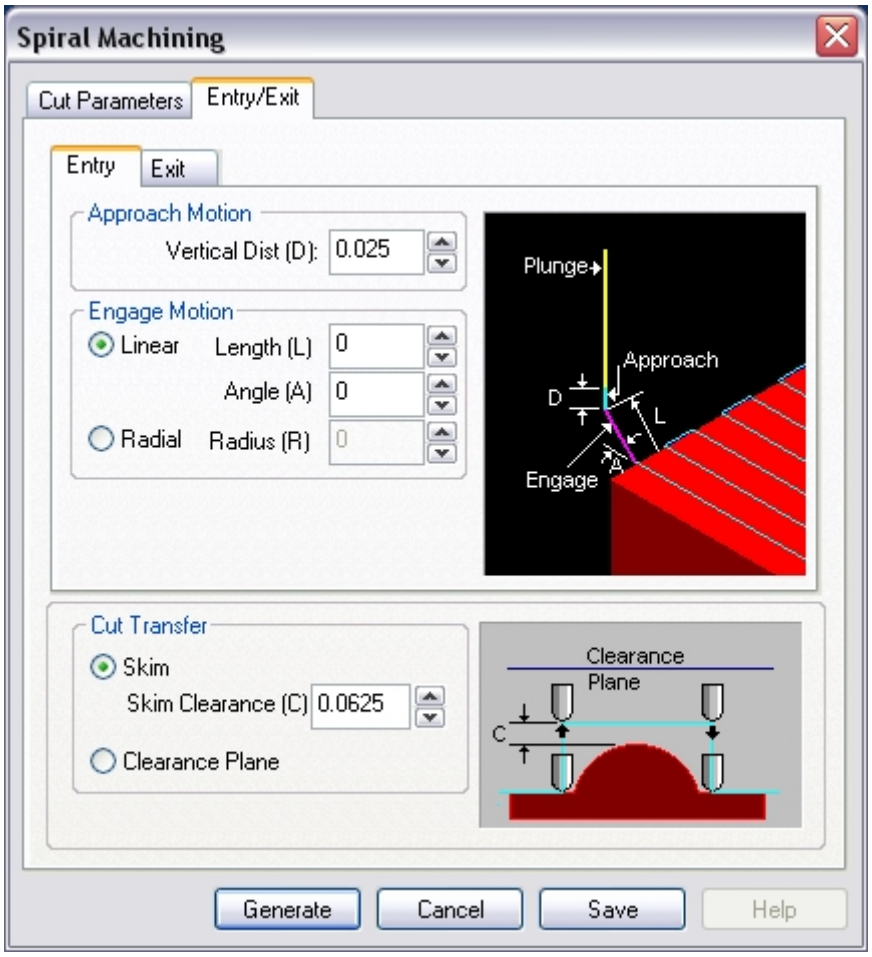
Entry / Exit

This entry/exit page is invoked in various 3 axis toolpath methods. This page controls the entry and exit motions in a vertical plane.

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

Entry
The user can choose to enter the material to be cut with a combination of two motions. These motions are Approach and Engage. The Approach motion is the motion as the cutter traverses from the transfer or plunge motion and meets the Engage motion. The Engage motion is the last motion before the cutter begins to cut material. The user has the ability to define the Approach motion as a vertical distance above the start of the Engage motion. The user can define the Engage motion either as a Linear ramp or Radial ramp motion as the graphic in the dialog indicates.

Exit
The user can choose to exit the cutting also as a combination of two motions. These motions are Retract and Departure. The Retract motion is the first motion after the cutter loses contact with the material to be cut. The Departure motion is the motion that bridges the Retract motion and the transfer motion. The Retract motion can be defined either as a Linear ramp or Radial ramp motion as the graphic in the dialog indicates. The user has the ability to define the Departure motion as a vertical distance above the end of the Retract motion.



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.

Curve Machining

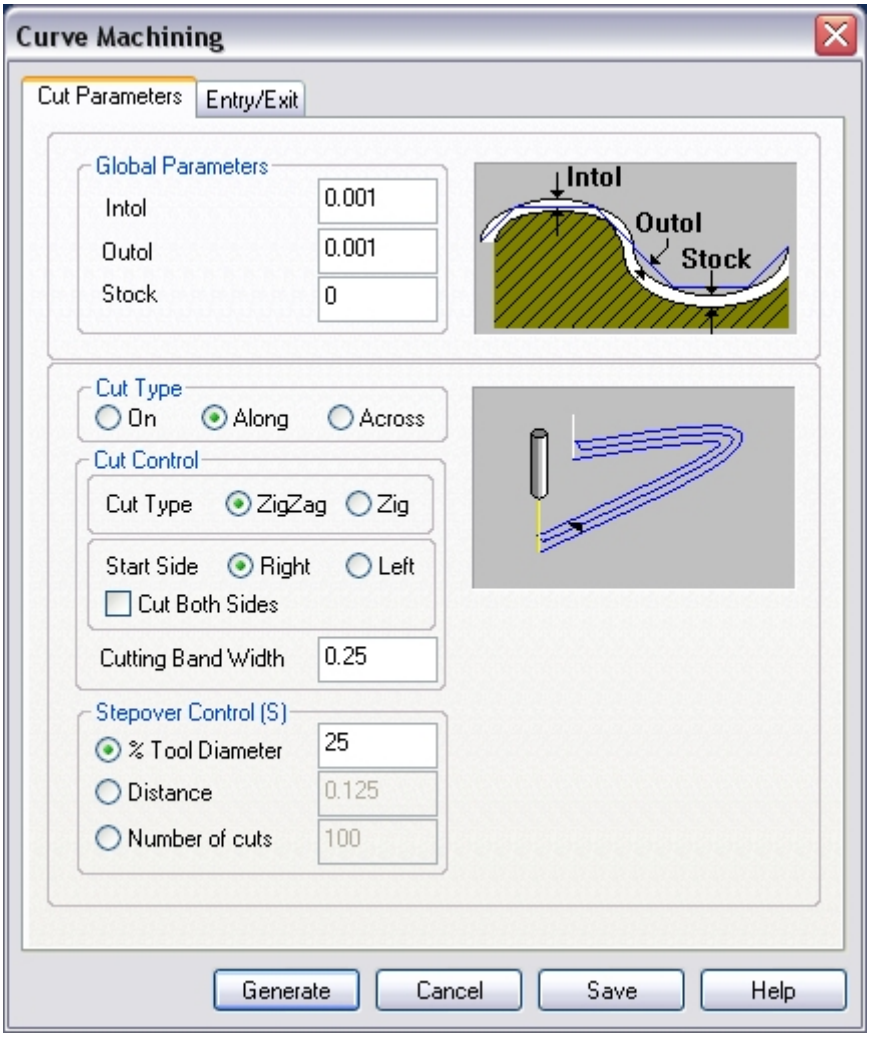
Available in Alibre CAM Expert only.

The Curve Machining operation can be used for fine finishing in closed regions. 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 nestings 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.

Description

The Curve Machining toolpath is invoked by clicking on the button in the MOps Browser and picking the **3 Axis Milling** and **Curve Machining** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



This dialog is a tabbed dialog with two tabs. Clicking on each of the tabs allows the user to set the different parameters. The following sections describe each of the tabs in more detail.

Cut Parameters

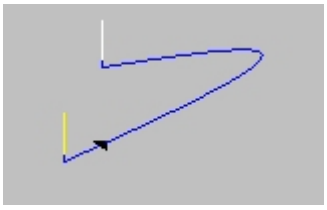
The Global Cut Parameters section allows the user to set the intol and the outol values to be used in machining. Also any uniform thickness or stock that needs to be left of the part also can be specified here.

- 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.

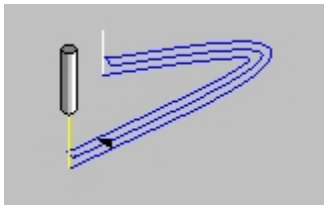
Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any).

- 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.

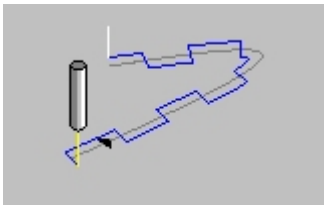
The Cut Type can be specified to be one of the following three types. It can be exactly on the curve; or it could be along the curve within specified band width either to the left, right or on both sides of the curve. It can also be set to move across the curve within specified band width either to the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



On



Along



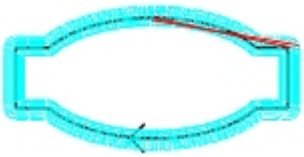
Across

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. 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 total number of linear cuts desired. The stepover is computed by the cutting band-width. This guarantees that the desired stepover will never be exceeded at any time during the cut operation.

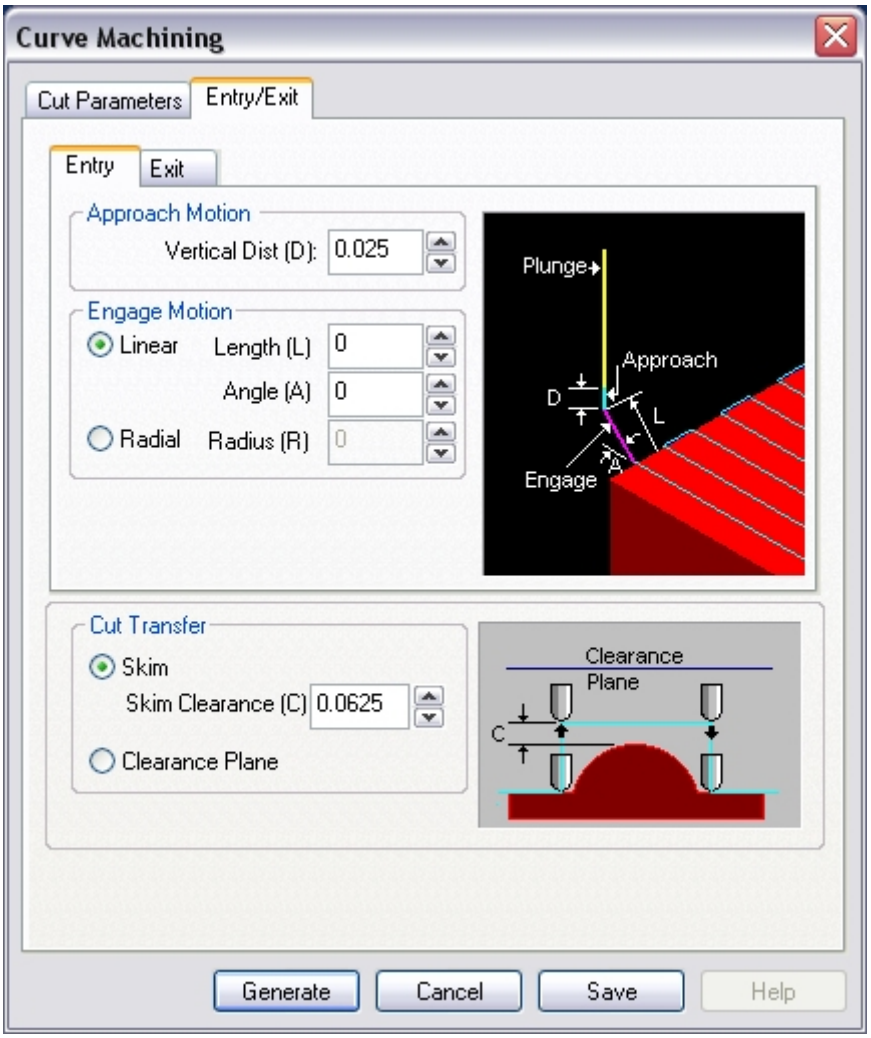
Entry / Exit

This entry/exit page is invoked in various 3 axis toolpath methods. This page controls the entry and exit motions in a vertical plane.

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

Entry
The user can choose to enter the material to be cut with a combination of two motions. These motions are Approach and Engage. The Approach motion is the motion as the cutter traverses from the transfer or plunge motion and meets the Engage motion. The Engage motion is the last motion before the cutter begins to cut material. The user has the ability to define the Approach motion as a vertical distance above the start of the Engage motion. The user can define the Engage motion either as a Linear ramp or Radial ramp motion as the graphic in the dialog indicates.

Exit
The user can choose to exit the cutting also as a combination of two motions. These motions are Retract and Departure. The Retract motion is the first motion after the cutter loses contact with the material to be cut. The Departure motion is the motion that bridges the Retract motion and the transfer motion. The Retract motion can be defined either as a Linear ramp or Radial ramp motion as the graphic in the dialog indicates. The user has the ability to define the Departure motion as a vertical distance above the end of the Retract motion.



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.

Between 2 Curves Machining

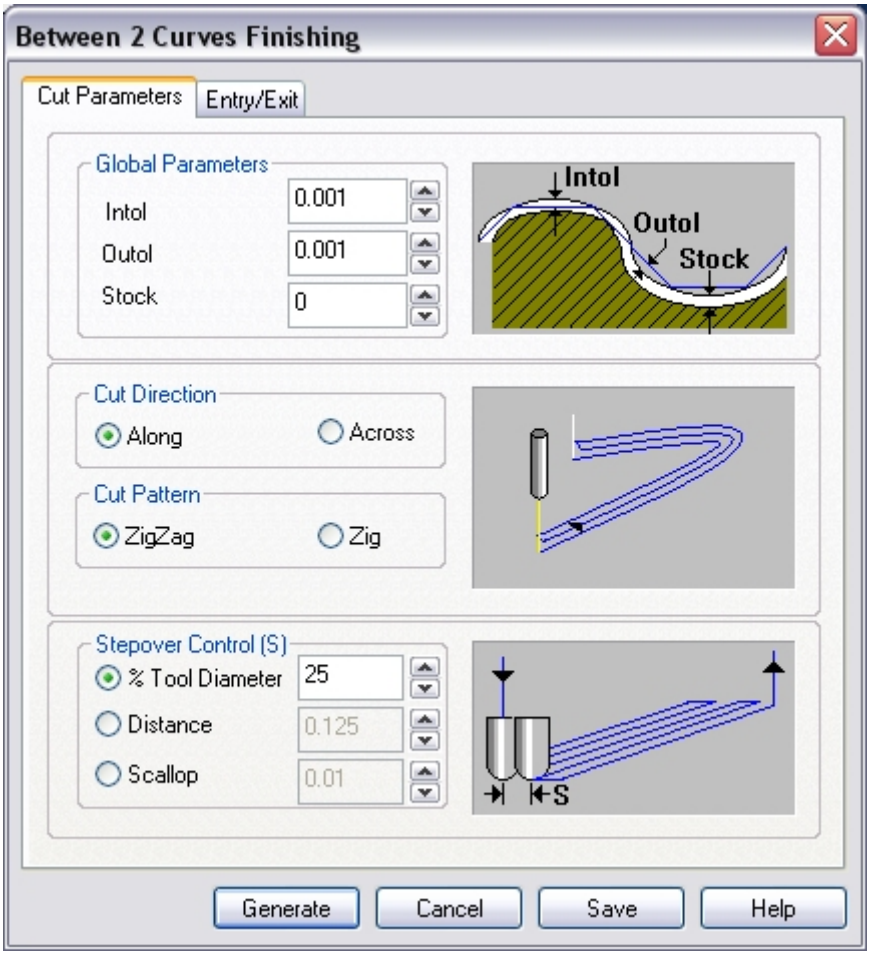
Available in Alibre CAM Expert only.

This toolpath method allows machining between two user-defined curves. These can be open or closed curves. The user can select machining to be performed either parallel or normal to these curves. The created toolpath will make a gradual transition from one curve to the other depending

on the geometric form of the two curves. This creates a blended toolpath that can be used to efficiently finish complex shapes. This type of machining is sometimes called Flowline machining.

Description

The Between 2 Curves Machining toolpath is invoked by clicking on the  button in the MOps Browser and picking the **3 Axis Milling** and **Between 2 Curves Machining** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



Cut Parameters

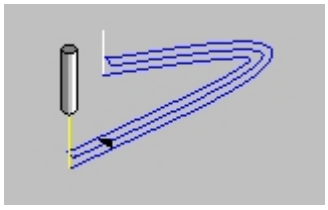
The Global Cut Parameters section allows the user to set the intol and the outol values to be used in machining. Also any uniform thickness or stock that needs to be left of the part also can be specified here.

- 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.

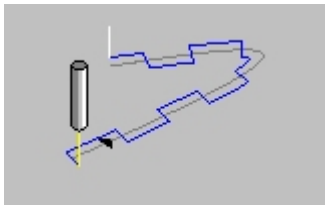
Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any).

- 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.

The Cut Direction can be along or across the two curves.

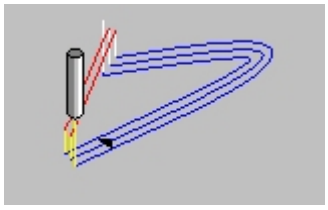


Along

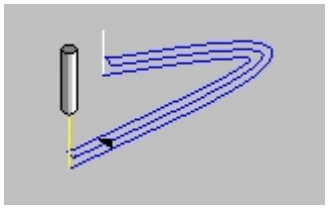


Across

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.

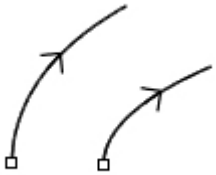
The machining regions for Between 2 Curves machining need to be 2 curves only.

The 2 sketches (curves)

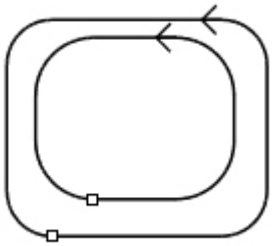
- Need to be either open or closed.
- User cannot select an open curve and the other as closed.
- Need to be in the same direction

- Start point needs to be at the same location on both the curves.

Following types of sketches **can** be used as Machining Regions

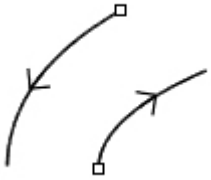


Open sketches- same direction

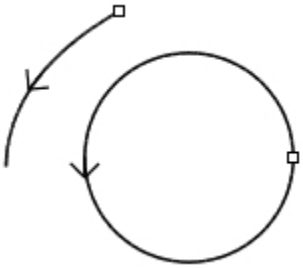


Closed sketches same direction and start points near each other

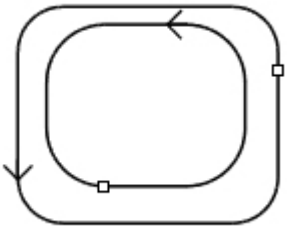
The following types of selections **cannot** be used as Machining Regions



Open sketch – opposite directions



Open and closed sketch



Closed sketch start points not near

Entry / Exit

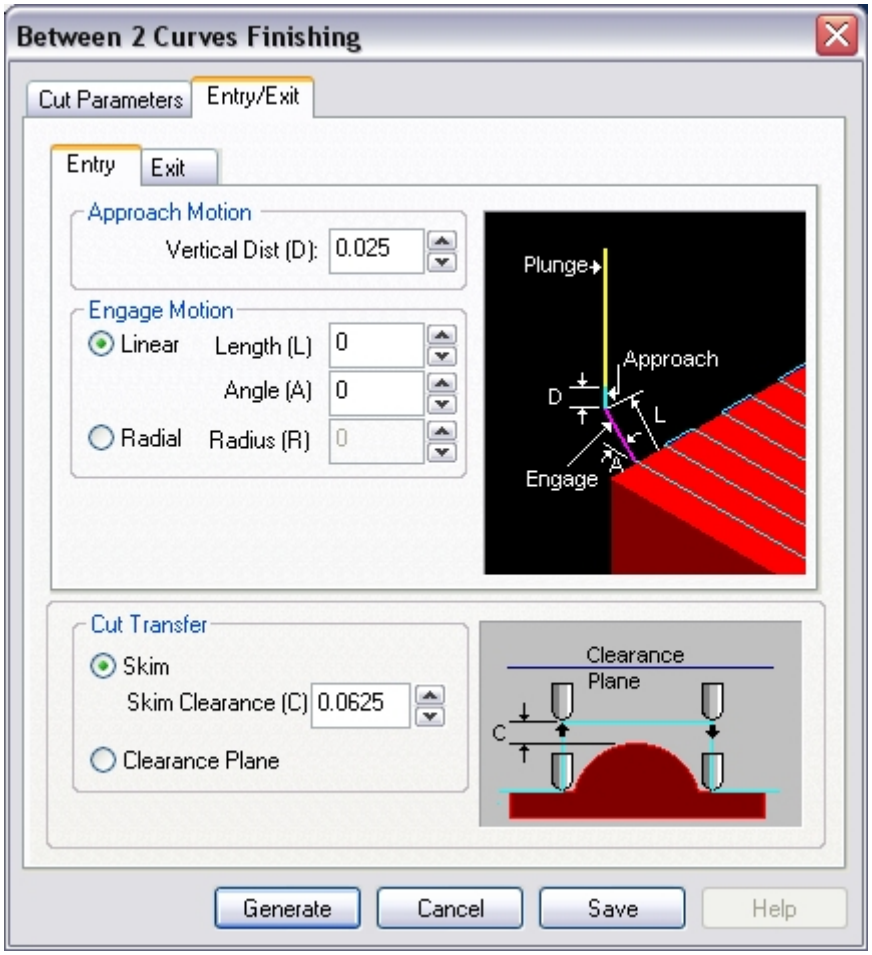
This entry/exit page is invoked in various 3 axis toolpath methods. This page controls the entry and exit motions in a vertical plane.

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

Entry
The user can choose to enter the material to be cut with a combination of two motions. These motions are Approach and Engage. The Approach motion is the motion as the cutter traverses from the transfer or plunge motion and meets the Engage motion. The Engage motion is the last motion before the cutter begins to cut material. The user has the ability to define the Approach motion as a vertical distance above the start of the Engage motion. The user can define the Engage motion either as a Linear ramp or Radial ramp motion as the graphic in the dialog indicates.

Exit

The user can choose to exit the cutting also as a combination of two motions. These motions are Retract and Departure. The Retract motion is the first motion after the cutter loses contact with the material to be cut. The Departure motion is the motion that bridges the Retract motion and the transfer motion. The Retract motion can be defined either as a Linear ramp or Radial ramp motion as the graphic in the dialog indicates. The user has the ability to define the Departure motion as a vertical distance above the end of the Retract motion.



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.

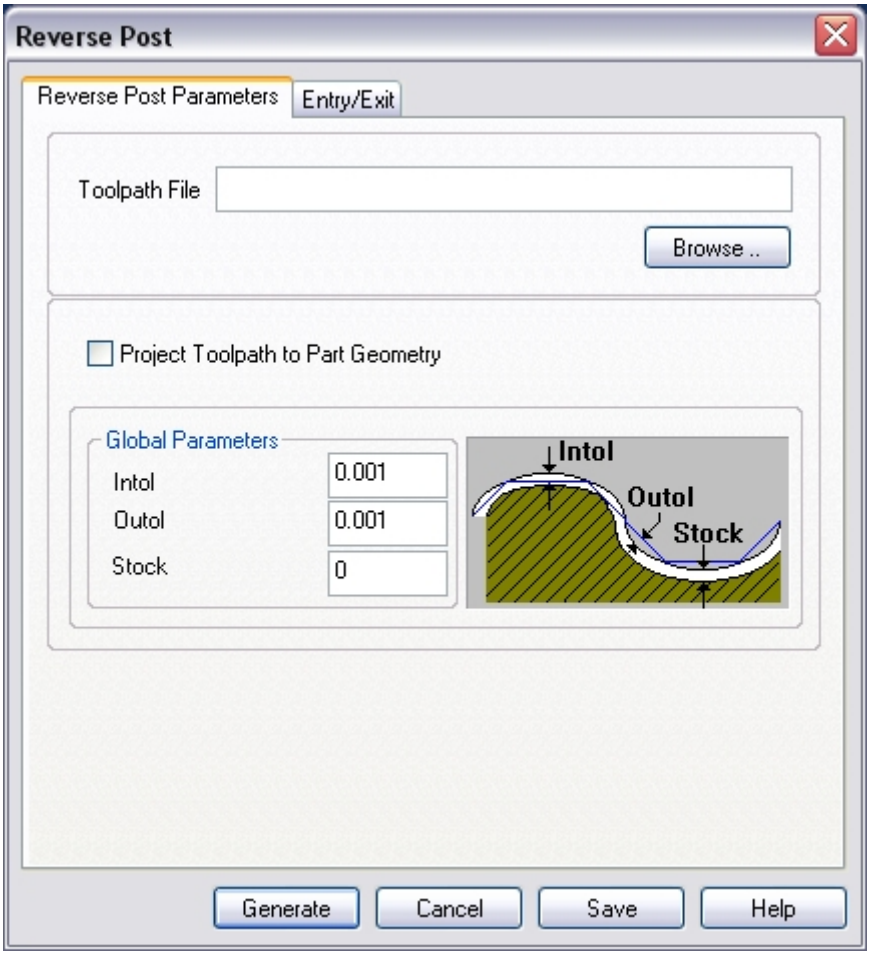
Reverse Post Milling

Available in Alibre CAM Expert only.

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.

Description

The Reverse Post Machining toolpath is invoked by clicking on the button in the MOps Browser and picking the **3 Axis Milling** and **Reverse Post Machining** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



This dialog is a tabbed dialog with two tabs. Clicking on each of the tabs allows the user to set the different parameters. The following sections describe each of the tabs in more detail.

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. The Global Cut Parameters section allows the user to set the intol and the outol values to be used in machining. Also any uniform thickness or stock that needs to be left of the part also can be specified here.

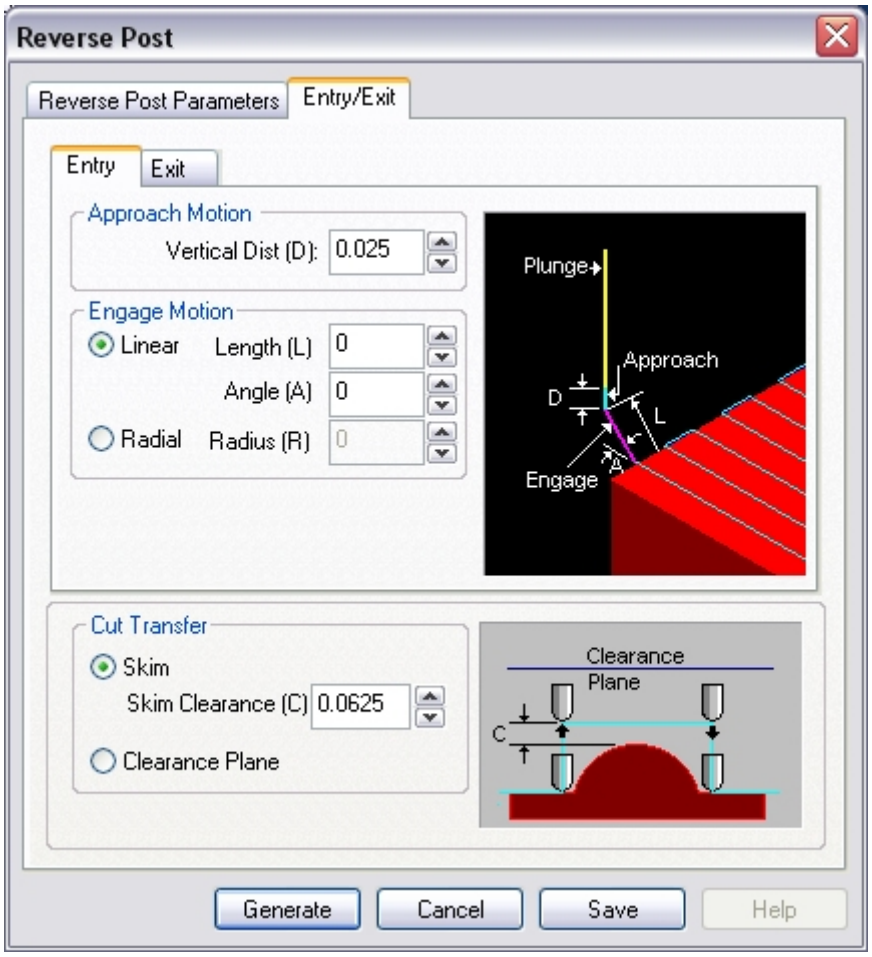
Entry/Exit

This entry/exit page is invoked in various 3 axis toolpath methods. This page controls the entry and exit motions in a vertical plane.

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

Entry
The user can choose to enter the material to be cut with a combination of two motions. These motions are Approach and Engage. The Approach motion is the motion as the cutter traverses from the transfer or plunge motion and meets the Engage motion. The Engage motion is the last motion before the cutter begins to cut material. The user has the ability to define the Approach motion as a vertical distance above the start of the Engage motion. The user can define the Engage motion either as a Linear ramp or Radial ramp motion as the graphic in the dialog indicates.

Exit
The user can choose to exit the cutting also as a combination of two motions. These motions are Retract and Departure. The Retract motion is the first motion after the cutter loses contact with the material to be cut. The Departure motion is the motion that bridges the Retract motion and the transfer motion. The Retract motion can be defined either as a Linear ramp or Radial ramp motion as the graphic in the dialog indicates. The user has the ability to define the Departure motion as a vertical distance above the end of the Retract motion.



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.

Hole Making Operations

These operations are used to create holes in the part, including drill holes, counter sunk holes and through holes.

The following **Drilling** operations 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.

Drilling

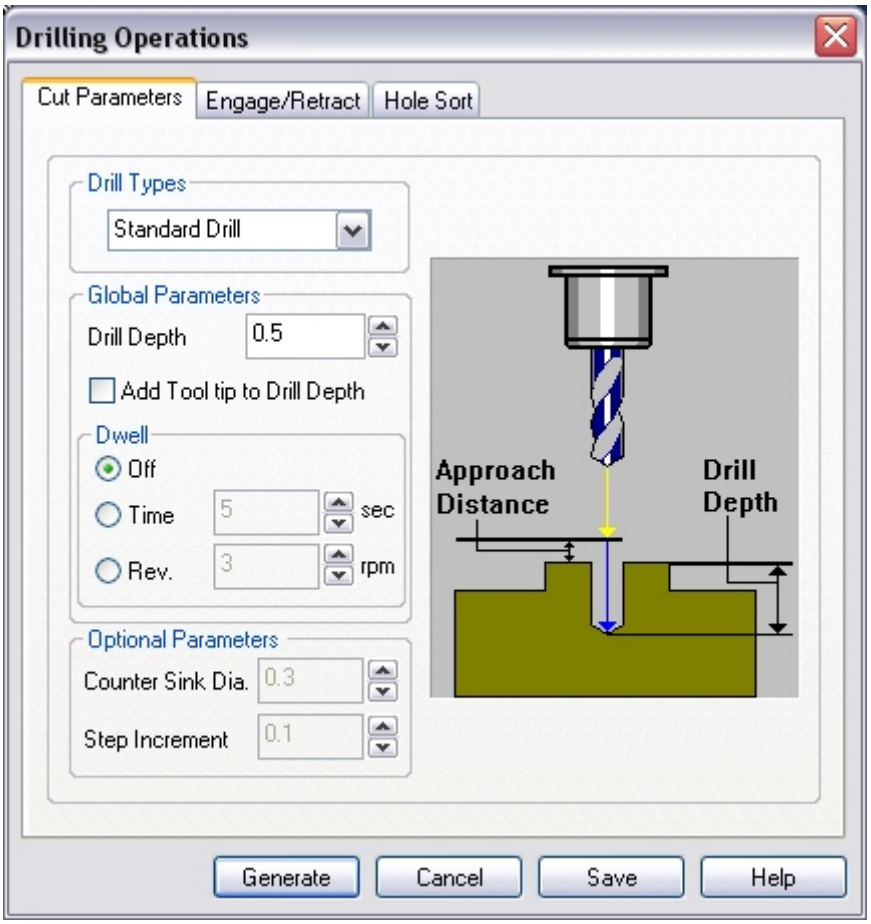
The Drill cycle is used to cut holes in the part. The drill cycles supported are:

- Standard
- Deep
- Break-chip
- Countersink

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.

Description

The Drilling toolpath is invoked by clicking on the  button in the MOps Browser and picking the **3 Axis Hole Machining** and **Drilling** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.

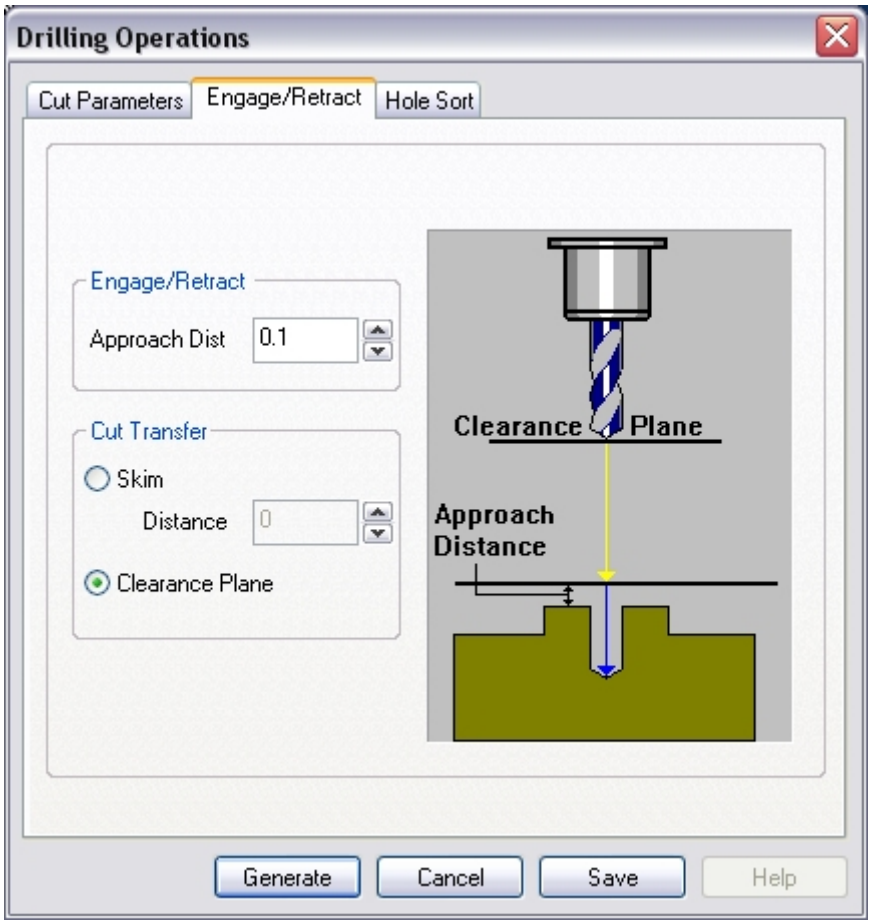


Cut Parameters

The user has the ability to choose the type of drilling operation required. “Drill depth” specifies the hole depth. It needs to be specified for the Standard, Deep and Breakchip drilling. “Dwell” is an optional parameter that allows a machine delay of either ‘t’ seconds or ‘n’ revolutions of the spindle. The Counter-Sink Diameter is required only for the Counter-Sink Drill type. Alibre CAM will automatically calculate the drill depth. “Step Increment” specifies the tool retract distance. It needs to be specified for Deep and Breakchip drilling.

Engage / Retract

The Engage and Retract distances can be set using the following dialog.

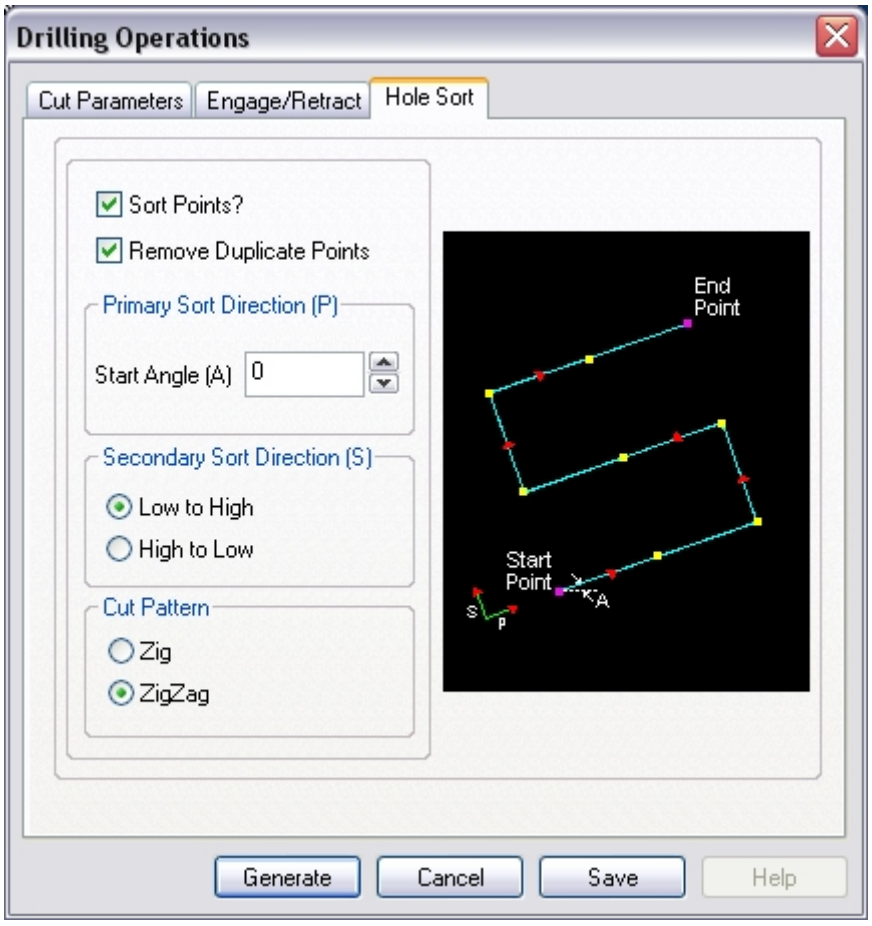


Hole Sorting

Hole Sorting is done for optimization of the cutter paths during the various hole-making operations. Note that the hole sorting options are available both in the 3 Axis as well as the 4 Axis hole making operations.

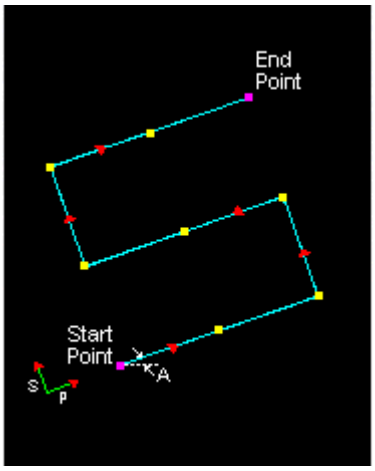
Hole sorting is performed according to the **Primary** and **Secondary** sort directions. The **Primary Sort** direction is defined by the **Sort Angle**. The sort angle is computed from the X-axis of the World Coordinate System. The **Secondary Sort Direction** is normal to this direction. This sort direction can either be defined as going from **Low to High**, or from **High to Low**. The traversal of the holes for the hole making operation, once the holes are sorted can either be in a **zig-zag** or a uni-directional **zig** fashion.

The bitmaps seen on the dialog box clearly display the sorting selected for the various options.

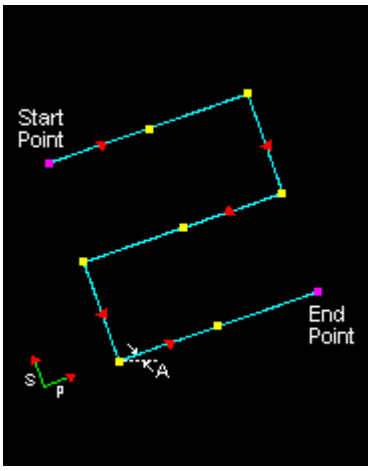


Sort Direction

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 way). The bitmaps seen on the dialog box clearly display the sorting selected for the various options.

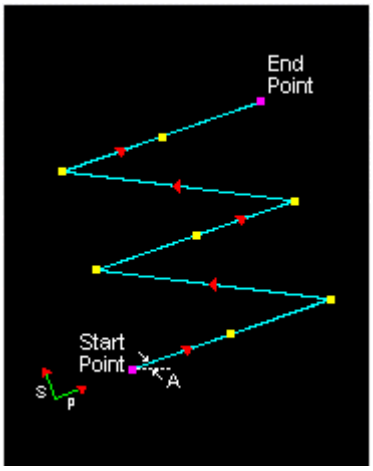


Low to High

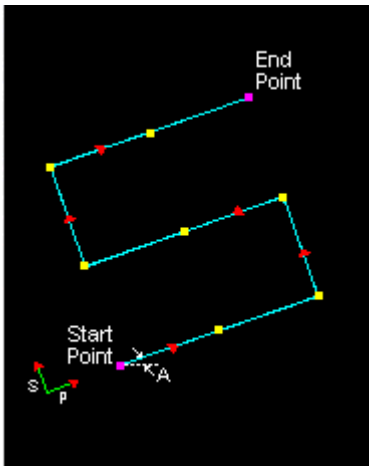


High to Low

Cut Pattern



Zig Pattern

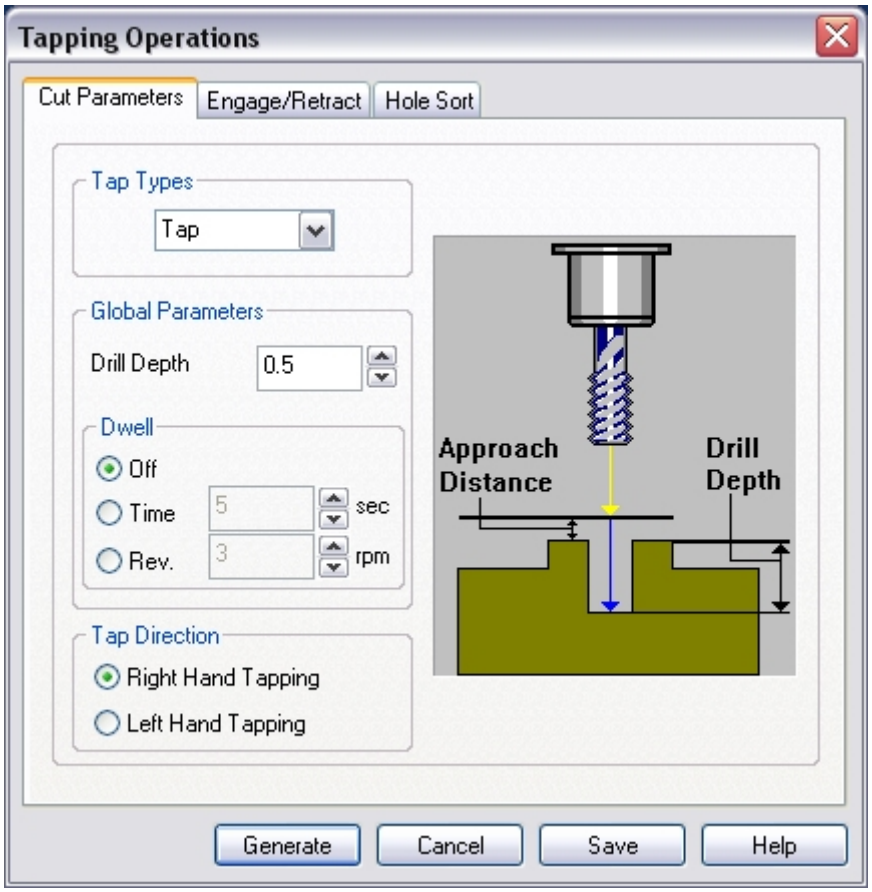


ZigZag Pattern

Tapping

Available in Alibre CAM Standard, Expert and Professional versions only

The Tapping toolpath is invoked by clicking on the button in the MOps Browser and picking the **3 Axis Hole Machining** and **Tapping** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.

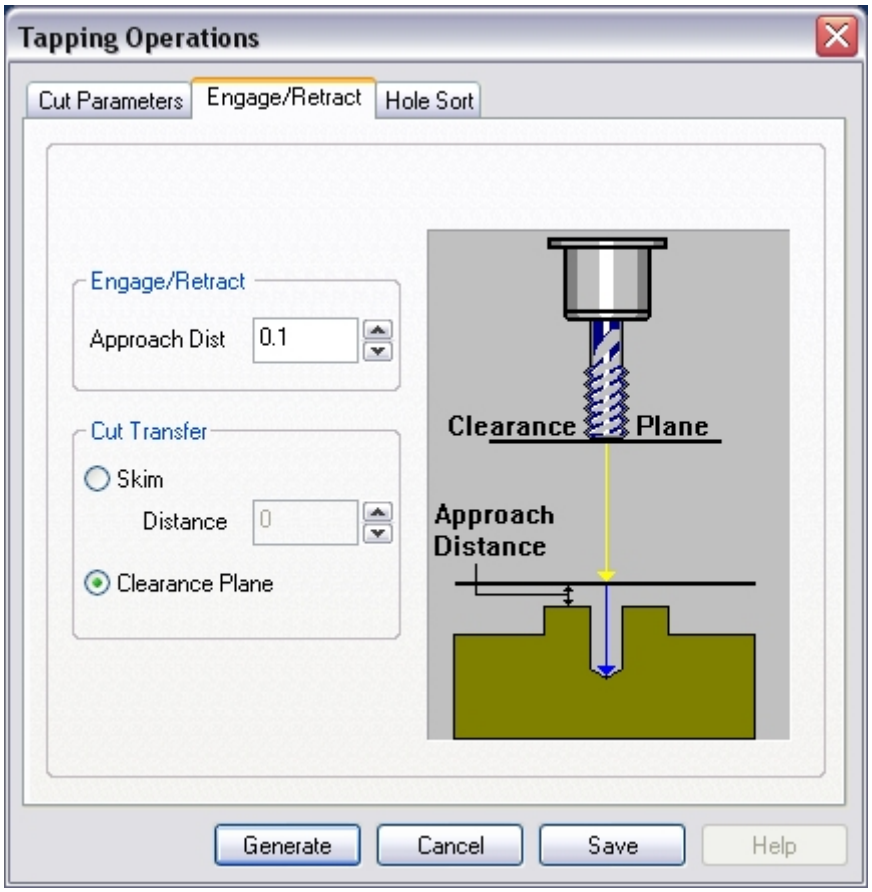


Cut Parameters

The user needs to specify the tapping depth need for the operation. The parameter “dwell” is usually toggled OFF for tapping. When turned ON, it causes the spindle to stop and dwell for a specified time before retracting. The threads could be in the Clockwise or the Counter-clockwise directions, thus resulting in the user choices of Right handed or Left handed tapping.

Engage / Retract

The Engage and Retract distances can be set using the following dialog.

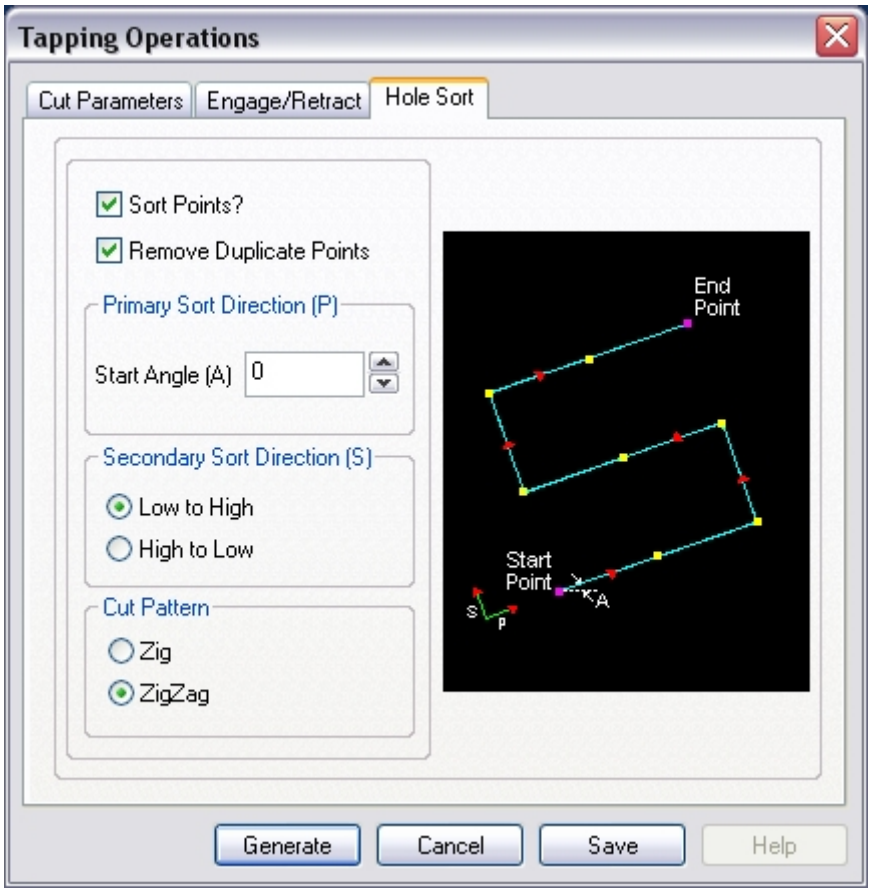


Hole Sorting

Hole Sorting is done for optimization of the cutter paths during the various hole-making operations. Note that the hole sorting options are available both in the 3 Axis as well as the 4 Axis hole making operations.

Hole sorting is performed according to the **Primary** and **Secondary** sort directions. The **Primary Sort** direction is defined by the **Sort Angle**. The sort angle is computed from the X axis of the World Coordinate System. The **Secondary Sort Direction** is normal to this direction. This sort direction can either be defined as going from **Low to High**, or from **High to Low**. The traversal of the holes for the hole making operation, once the holes are sorted can either be in a **zig-zag** or a uni-directional **zig** fashion.

The bitmaps seen on the dialog box clearly display the sorting selected for the various options.



Boring

Available in Alibre CAM Standard, Expert and Professional versions only

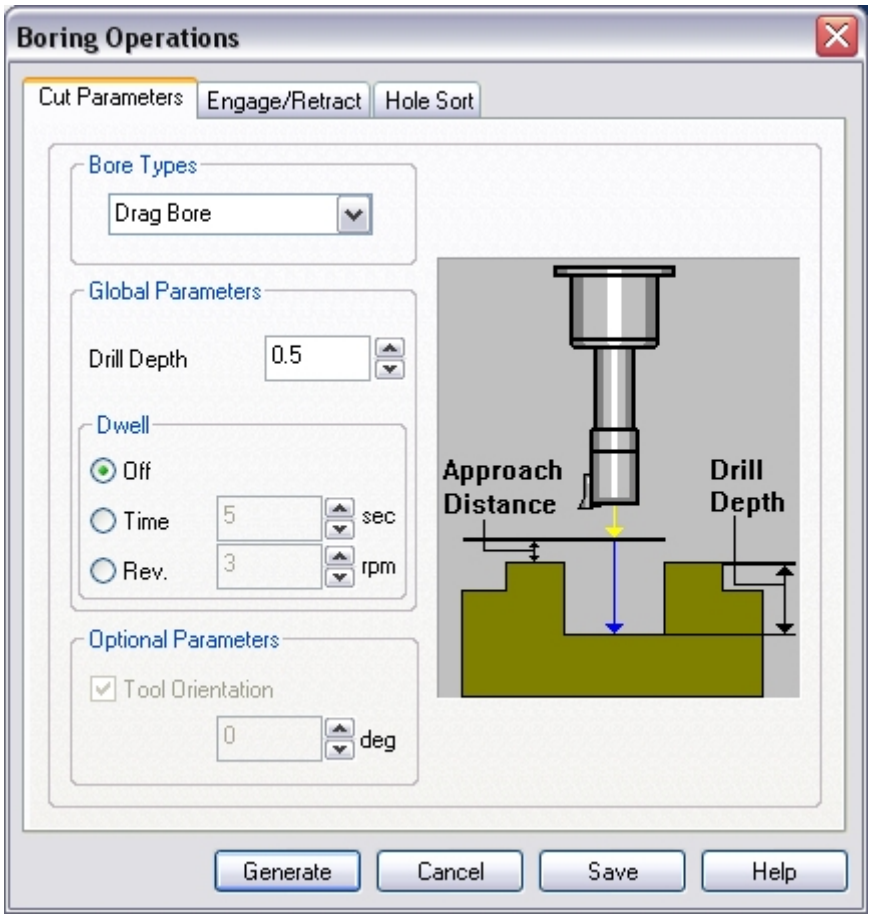
The Bore cycle is used to form shapes inside of a hole. The Bore cycle supported are:

- Drag
- No Drag
- Manual

In the Drag type, the tool is fed to specified depth at the controlled feed rate; the spindle stopped and then a rapid retract is done. In case of the No Drag type, 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 finally retracted. In case of Manual boring, the tool traverses to the programmed point; feeds to the specified depth at the controlled feed rate; and then stops motion for a manual retract.

Description

The Boring toolpath is invoked by clicking on the button in the MOps Browser and picking the **3 Axis Hole Machining** and **Boring** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.

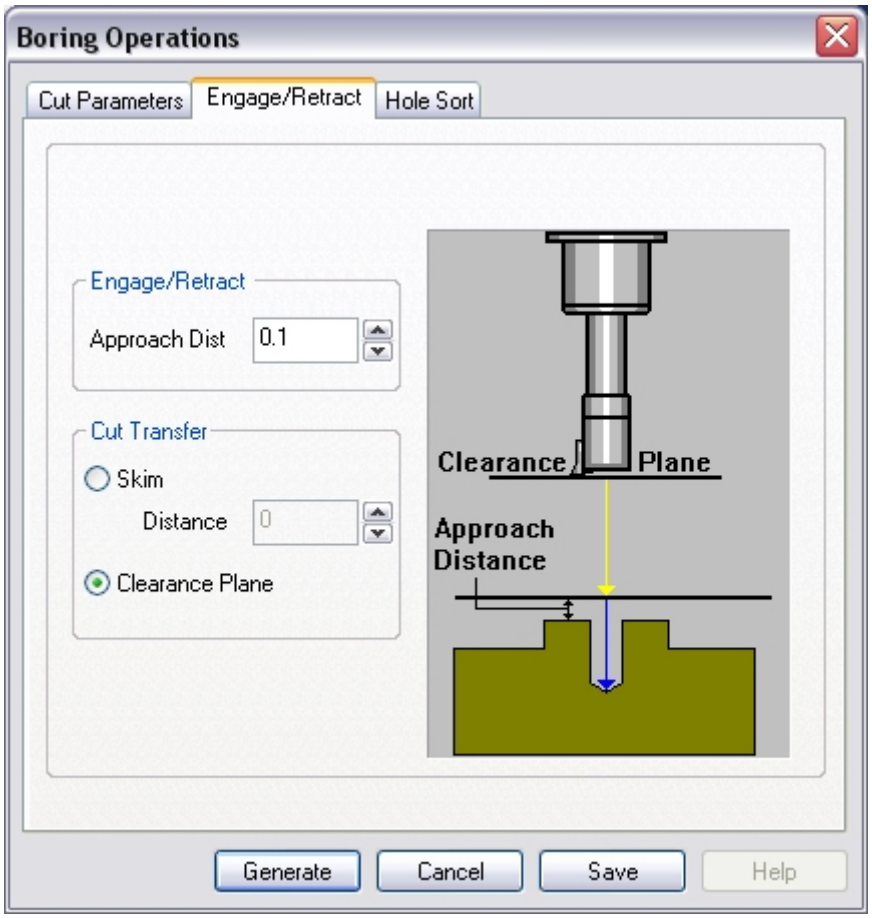


Cut Parameters

"Drill depth" specifies the hole depth. "Dwell" is an optional parameter that allows a machine delay of either 't' seconds or 'n' revolutions of the spindle. The "No Drag" boring can specify the angle at which the tool needs to be oriented before commencing the cycle. Tool orientation is not used in "drag" and "Manual" boring.

Engage / Retract

The Engage and Retract distances can be set using the following dialog.

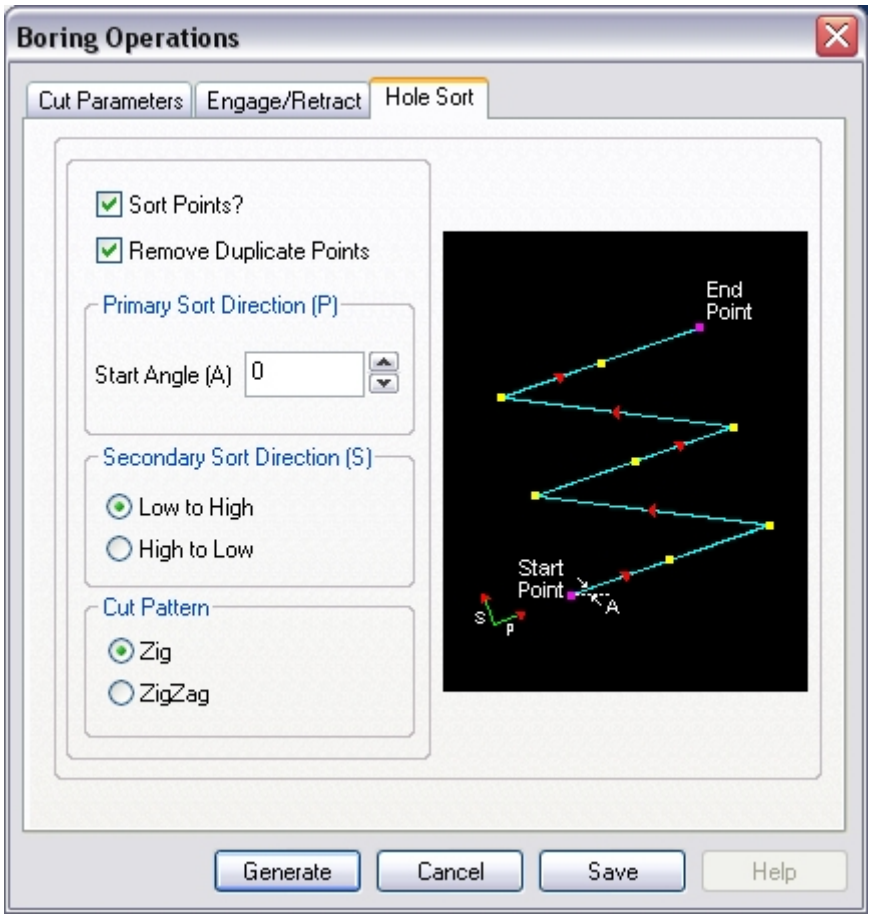


Hole Sorting

Hole Sorting is done for optimization of the cutter paths during the various hole making operations. Note that the hole sorting options are available both in the 3 Axis as well as the 4 Axis hole making operations.

Hole sorting is performed according to the **Primary** and **Secondary** sort directions. The **Primary Sort** direction is defined by the **Sort Angle**. The sort angle is computed from the X axis of the World Coordinate System. The **Secondary Sort Direction** is normal to this direction. This sort direction can either be defined as going from **Low to High**, or from **High to Low**. The traversal of the holes for the hole making operation, once the holes are sorted can either be in a **zig-zag** or a uni-directional **zig** fashion.

The bitmaps seen on the dialog box clearly display the sorting selected for the various options.



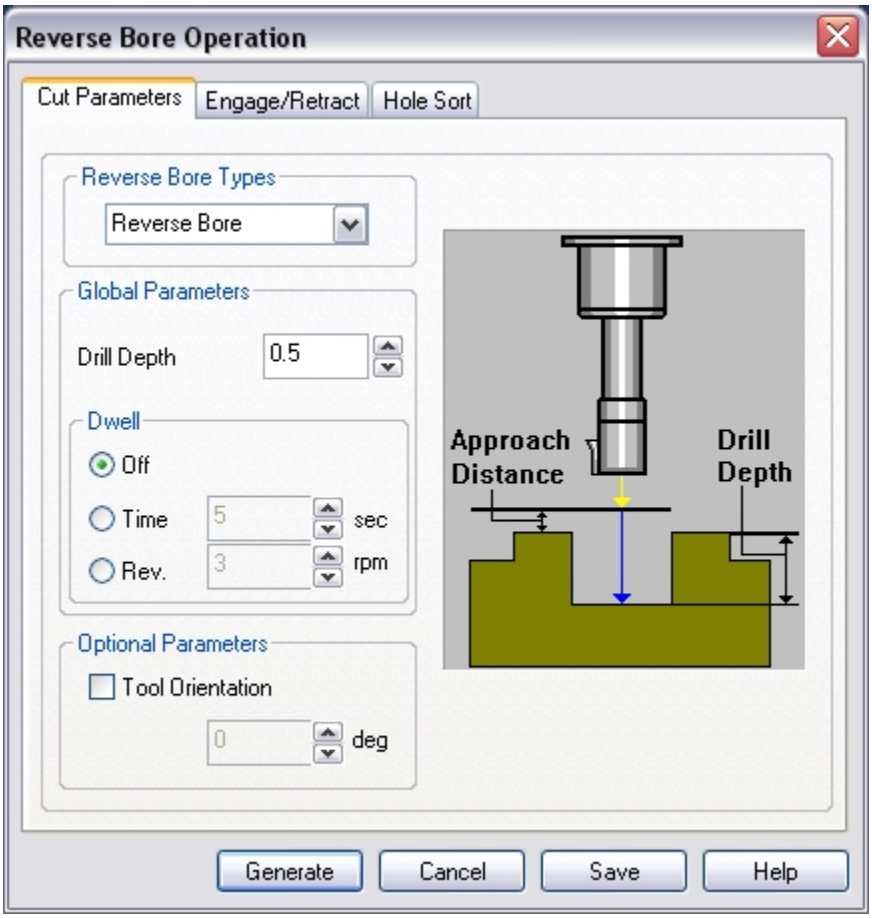
Reverse Boring

Available in Alibre CAM Standard, Expert and Professional versions only

The Reverse bore cycle is nothing but the bore cycle in the opposite direction. The tool spindle is oriented at the specified angle. It then rapids to the feed depth and moved to the work piece. The spindle is turned on and the cycle is started.

Description

The Reverse Boring toolpath is invoked by clicking on the button in the MOps Browser and picking the **3 Axis Hole Machining** and **Reverse Boring** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.

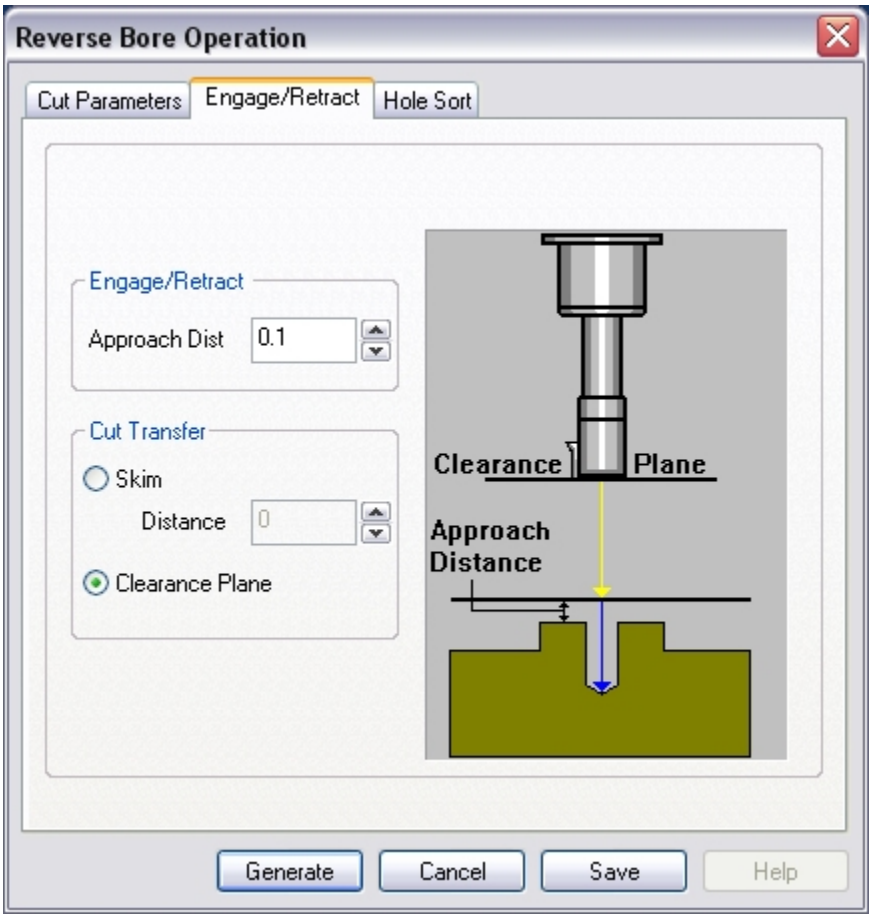


Cut Parameters

“Drill depth” specifies the hole depth. “Dwell” is an optional parameter that allows a machine delay of either ‘t’ seconds or ‘n’ revolutions of the spindle. “Tool orientation” is the angle at which the spindle needs to be oriented before the reverse bore operation is stared.

Engage / Retract

The Engage and Retract distances can be set using the following dialog.

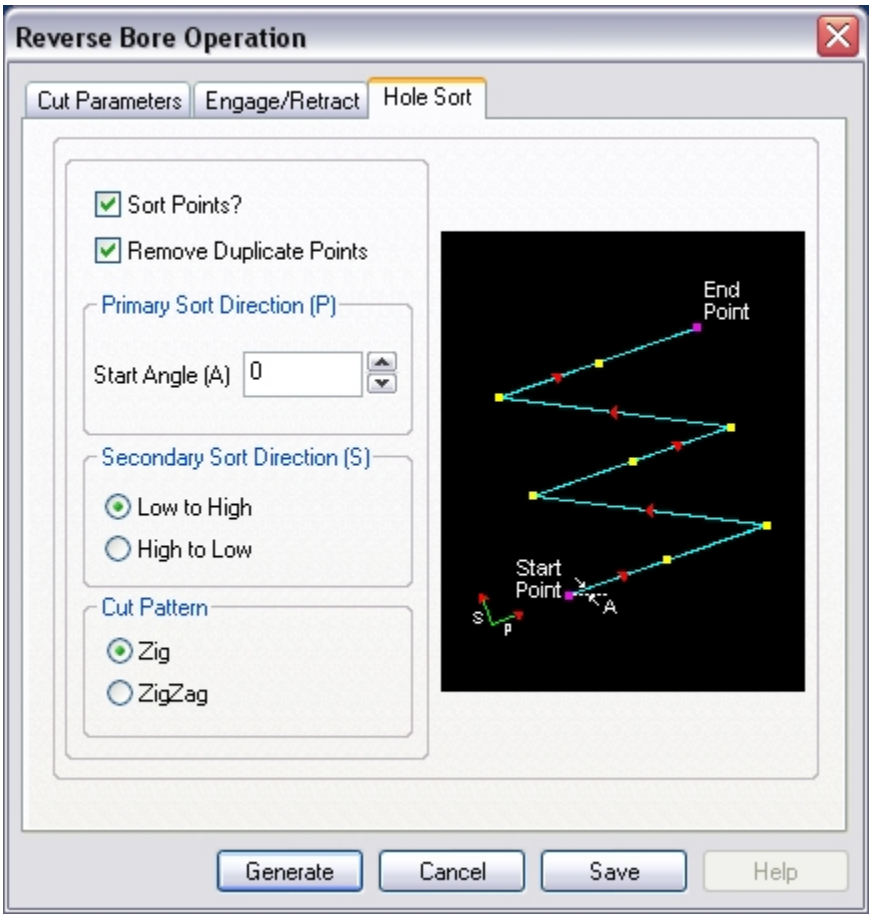


Hole Sorting

Hole Sorting is done for optimization of the cutter paths during the various hole making operations. Note that the hole sorting options are available both in the 3 Axis as well as the 4 Axis hole making operations.

Hole sorting is performed according to the **Primary** and **Secondary** sort directions. The **Primary Sort** direction is defined by the **Sort Angle**. The sort angle is computed from the X axis of the World Coordinate System. The **Secondary Sort Direction** is normal to this direction. This sort direction can either be defined as going from **Low to High**, or from **High to Low**. The traversal of the holes for the hole making operation, once the holes are sorted can either be in a **zig-zag** or a uni-directional **zig** fashion.

The bitmaps seen on the dialog box clearly display the sorting selected for the various options.



Create 4th Axis Machining Operations

Note: The 4th axis functionality is available with Alibre CAM Professional and Alibre CAM Expert.

Fourth Axis operations are used to machine parts that cannot be machined with simple 21/2 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 top and bottom sections and machined separately. Alibre CAM'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.

4th Axis Rotate Table

This dialog allows you to create a 4th axis rotation table machining operation.

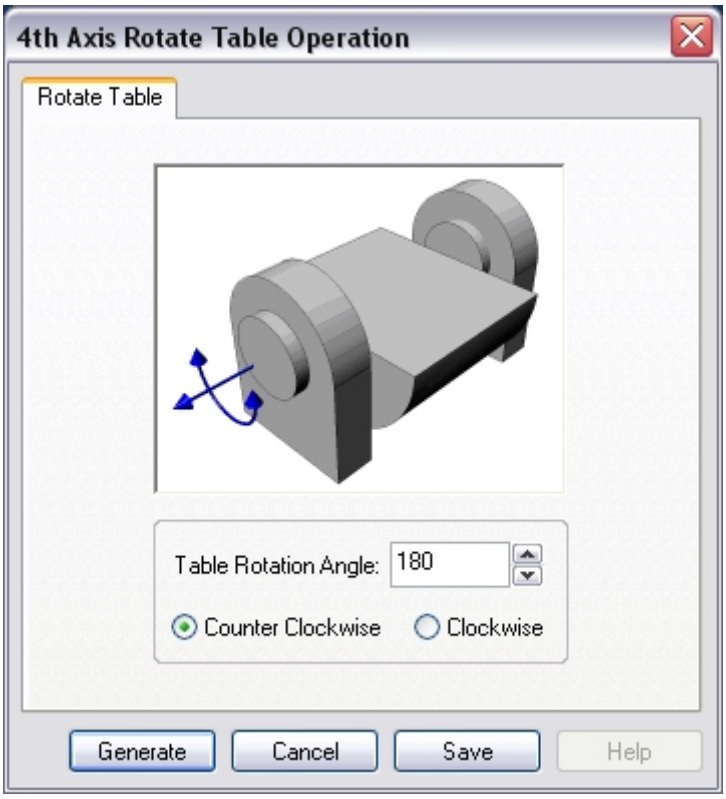
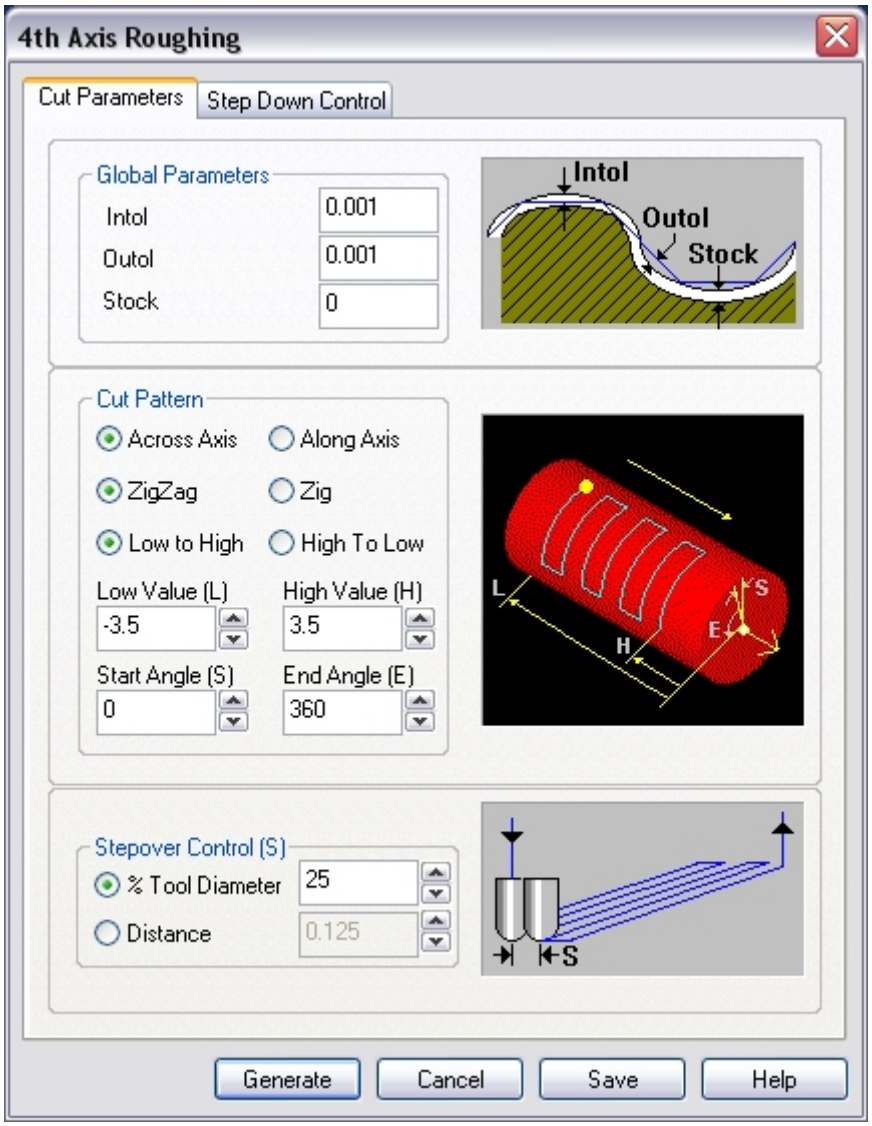


Table Rotation Angle - Specify the rotation angle value in degrees. When the user selects the Generate button, a Rotate Table machining operation will be created and displayed in the browser. When this operation is post processed and sent to the machine tool, a table rotation of the specified angle will be performed.

4th Axis Roughing

Available with Alibre CAM Professional and Alibre CAM Expert

The 4th Axis Roughing toolpath is invoked by clicking on the  button in the MOps Browser and picking the **4th Axis Milling** and **4th Axis Roughing** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



This dialog is tabbed with two tabs. Each tab defines a set of parameters that the user can specify. The sections below describe them in detail.

Cut Parameters

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

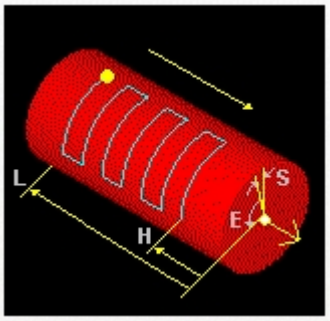
The Global Cut Parameters section allows the user to set the intol and the outol values to be used in machining. Also any uniform thickness or stock that needs to be left of the part also can be specified here.

- 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.

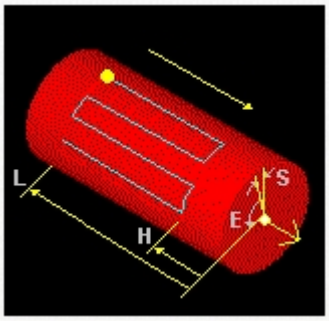
Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any).

- 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.

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.



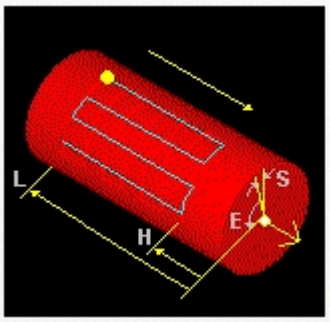
Across Axis



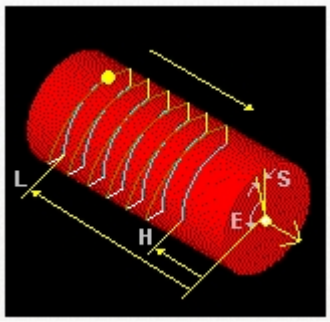
Along Axis

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.

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.

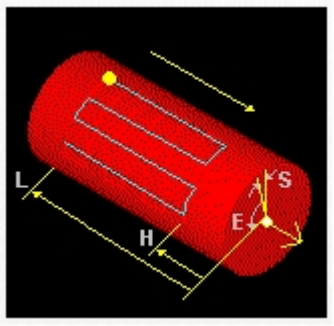


Zig Zag

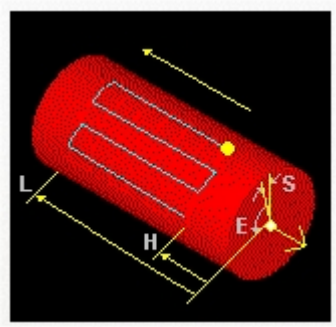


Zig

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



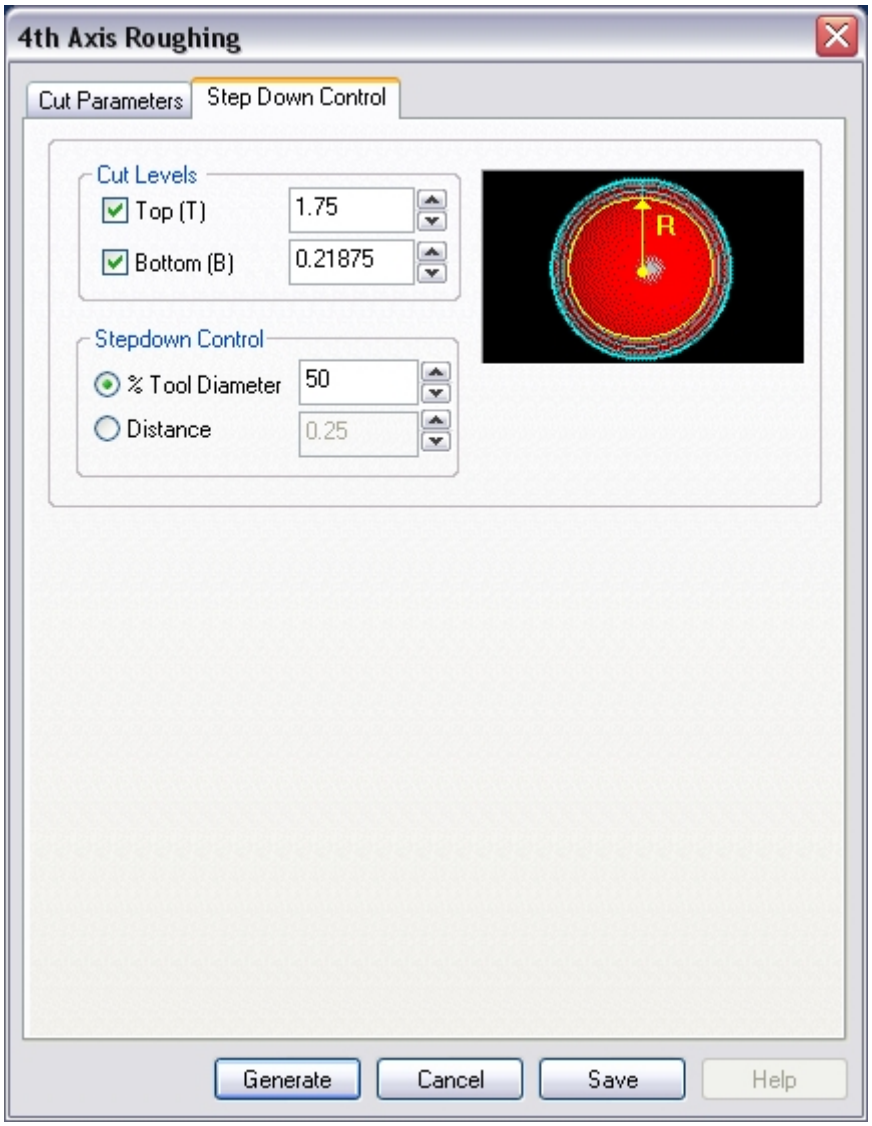
High to Low

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.

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

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



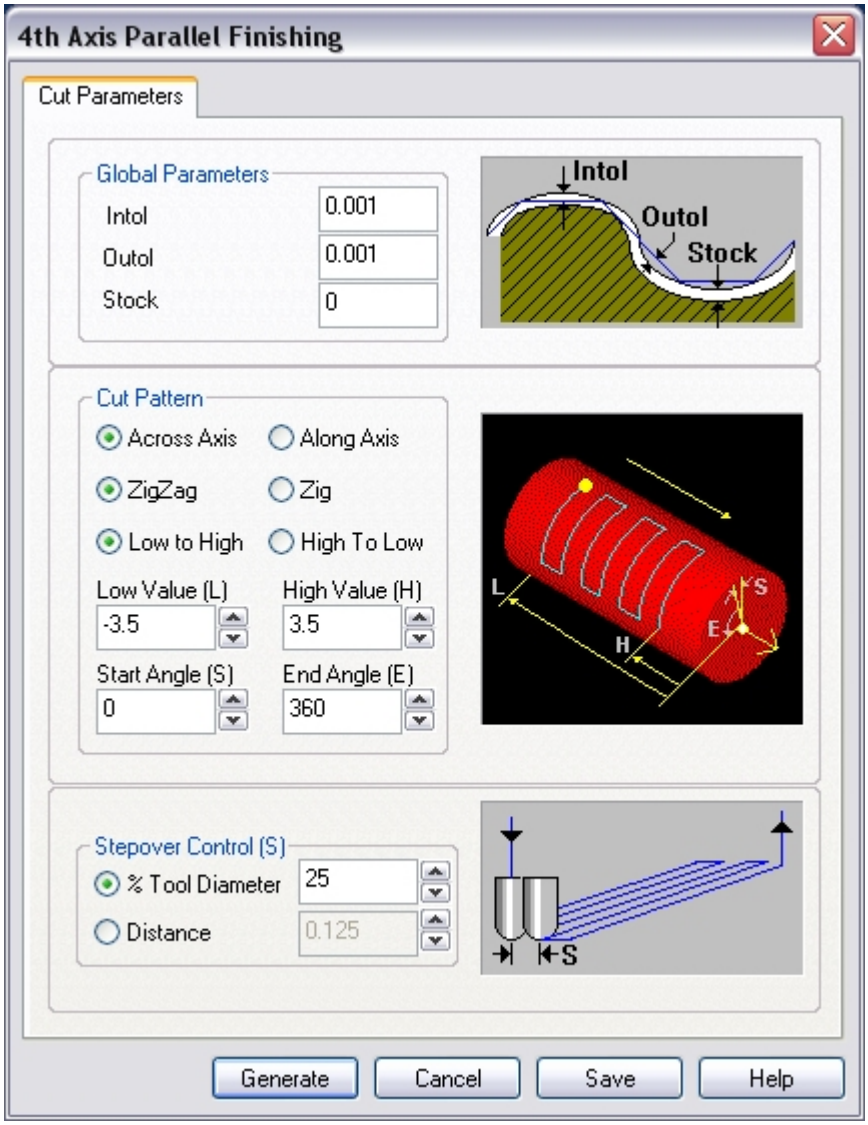
Selecting the Top check box will allow the user to specify the radial distance of the top most level from the rotary axis. If this is not selected, the system will use the largest radial distance based on the part geometry. Selecting the Bottom check box will allow the user the specify the radial distance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

4th Axis Finishing

Available with Alibre CAM Professional and Alibre CAM Expert

The 4th Axis Finishing toolpath is invoked by clicking on the button in the MOps Browser and picking the **4th Axis Milling** and **4th Axis Finishing** option. The toolpath generated depends on

the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.



Cut Parameters

The user can set the Global Cut Parameters and the Cut Pattern via this property page.

The Global Cut Parameters section allows the user to set the intol and the outol values to be used in machining. Also any uniform thickness or stock that needs to be left of the part also can be specified here.

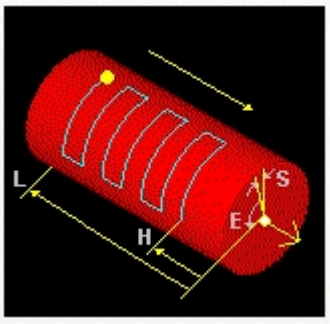
Stock: The thickness of the layer that will remain on top of the part after the toolpath is complete. Roughing operations generally leave a thin layer of stock, but for finishing operations this value is zero.

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

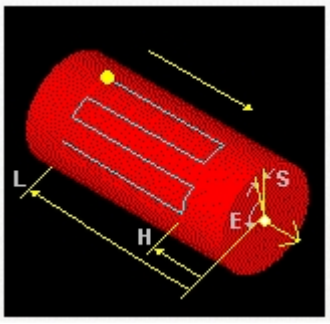
Intol: Inward tolerance - the maximum thickness of material that can be removed from the Stock layer.

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

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.



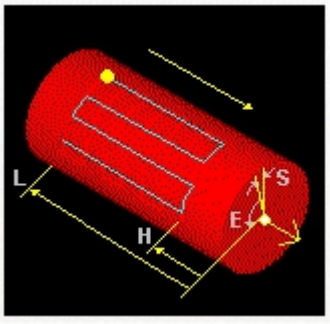
Across Axis

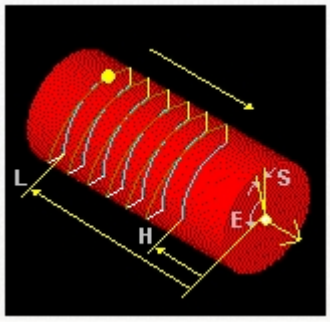


Along Axis

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.

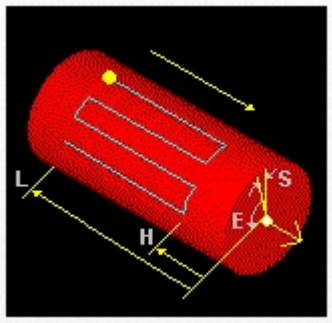
Selecting ZigZag will allow the toolpath to traverse back and forth while the Zig option will force the toolpath to be in one direction only.

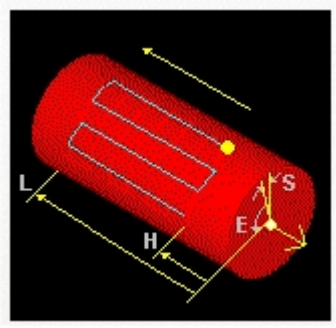




Zig Zag **Zig**

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.

Stepover Control

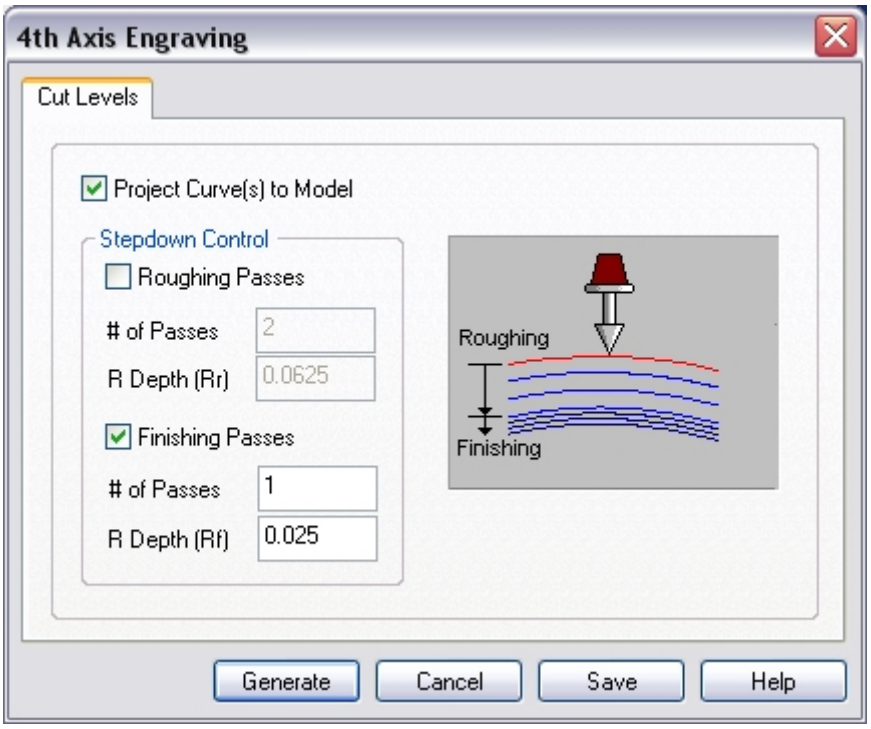
The Stepover Control section allows the user to define the spacing between the cuts. The spacing can be specified either as a percentage of tool diameter or a specific distance.

4th Axis Engraving

Available with Alibre CAM Professional and Alibre CAM Expert

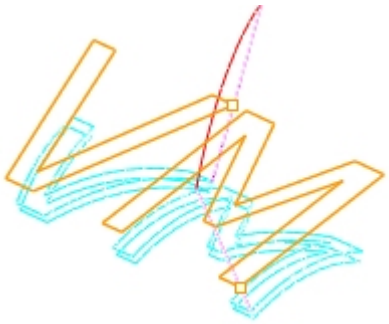
Description

The 4th Axis Engraving toolpath is invoked by clicking on the button in the MOps Browser and picking the **4th Axis Milling** and **4th Axis Engraving** option. The toolpath generated depends on the user-defined parameters. The various parameters that the user can set can be seen in the dialog box that is invoked when the user chooses the spiral toolpath operation. This dialog box is shown below.

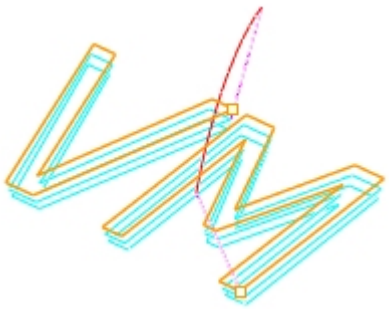


Cut Levels

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

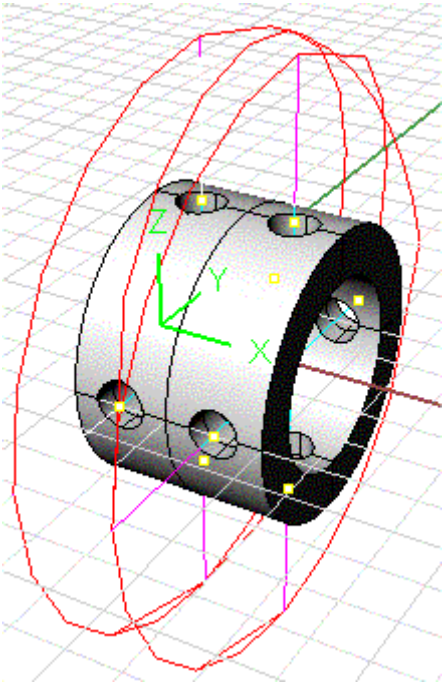
The **Step Down Control** section allows the user to select the Roughing and Finishing passes and the depth of each pass. The user can optionally specify both the Roughing passes and/or the Finishing passes. If the user does not specify either of them the cutter will simply follow the select region. 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.

4th Axis Hole Making

Available with Alibre CAM Professional and Alibre CAM Expert

Description

All of the hole creating machining operations available in 3 axis are also available in 4 axis mode. 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.



4th Axis Drilling

The 4th Axis Drilling Method is used to make holes on the part. The Drilling toolpath is invoked by clicking on the button in the MOps Browser and picking the **4th Axis Hole Machining** and **Drilling** option.

Refer to the Drilling section for detailed description.

4th Axis Tapping

The Tapping toolpath is invoked by clicking on the button in the MOps Browser and picking the **4th Axis Hole Machining** and **Tapping** option.

Refer to the Tapping section for detailed description.

4th Axis Boring

The Boring toolpath is invoked by clicking on the  button in the MOps Browser and picking the **4th Axis Hole Machining** and **Boring** option.

Refer to the Boring section for detailed description.

4th Axis Reverse Boring

The Reverse Boring toolpath is invoked by clicking on the  button in the MOps Browser and picking the **4th Axis Hole Machining** and **Reverse Boring** option.

Refer to the Reverse Boring section for detailed description.

Editing Machining Operations

Once a machining operation has been created it 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 tool, feeds/speeds, regions and machining parameters and the toolpath regenerated. 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. The second way of editing the operation is actually editing the toolpath associated with the operation. This type of editing is called non-associative editing because any changes made to the toolpath, though saved with the operation, would be lost if the user regenerated the machining operation.

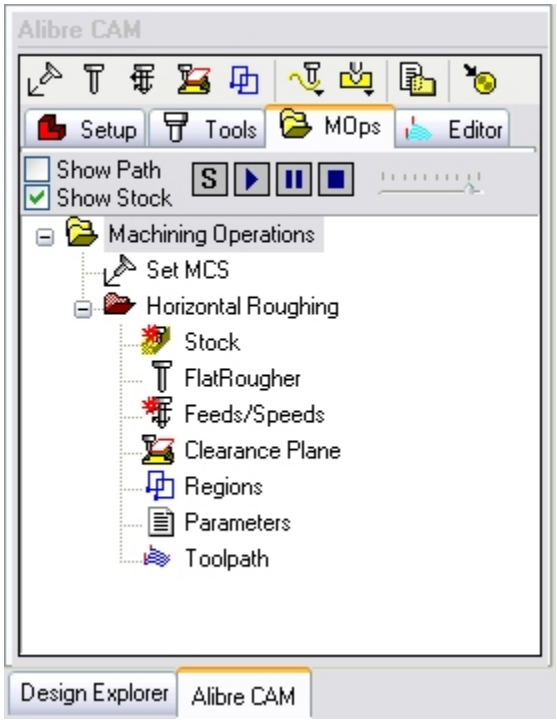
Editing Operations Associatively

Machining Operations can be edited by using the Mops Browser. Each machining operation is represented as a folder in the browser. In the expanded state of this folder icon, seven icons representing six different objects that make up the operation are displayed. These are the tool, feeds/speeds, clearance plane, machining regions, parameters and the toolpath objects. Double clicking on any of these icons gives the user an opportunity to edit the object. The same dialog that was used in the creation of the object being edited would be invoked and editing is exactly similar to creation. For example, double clicking on 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. Alibre CAM indicates such a condition by coloring the operation icon red. The object that necessitated this re-computation is also displayed with a red marker. An example of this is shown below.

Notice that in the browser window, the Horizontal Roughing operation is colored red. This indicates that one of the objects constituting the operation has been edited since the operation was generated. In this case the tool used in the operation was edited after the machining operation

was created and so is shown with a red star before it. Also notice that the Stock icon also has a red star before it. This means that the stock model for simulation has not yet been created yet.

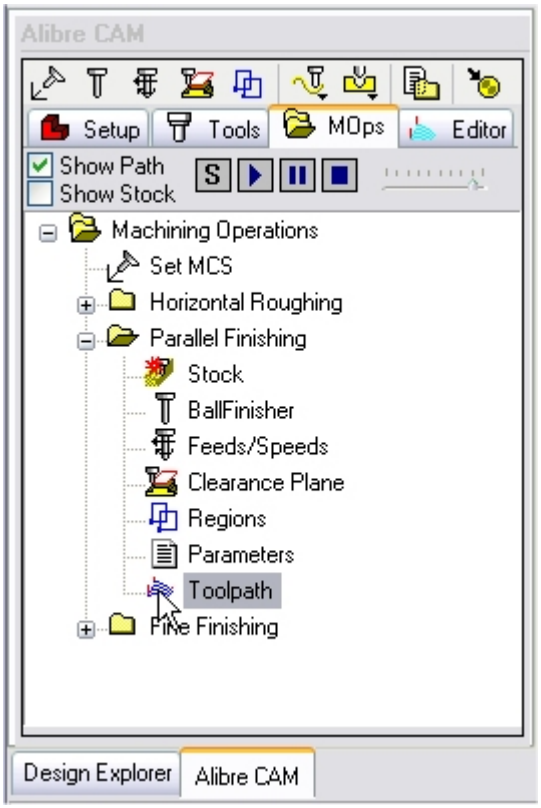


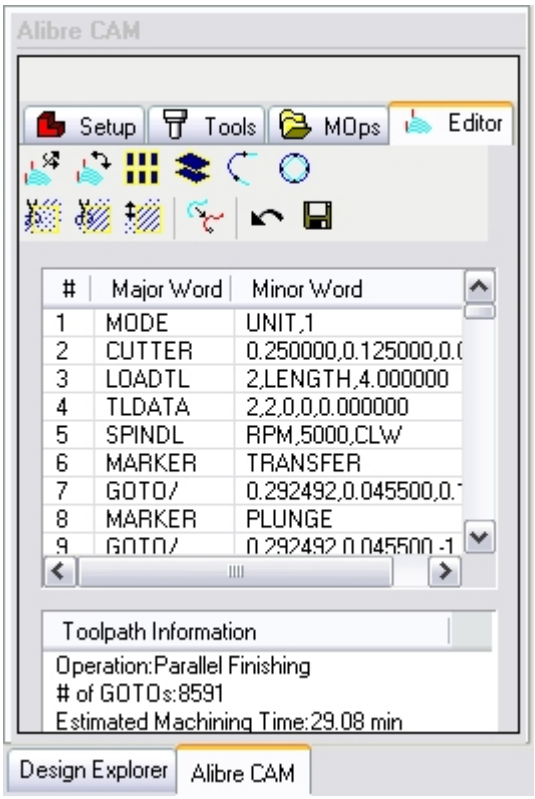
Editing Toolpaths (Toolpath Editor)

Available in Alibre CAM Expert only

Once a machining operation is created, the toolpath can be edited to add post-processor commands. To bring up the toolpath editor, the user switches to the toolpath editor tab located on the Alibre CAM browser. Alternatively user can click on the toolpath icon of the operation in the Browser.

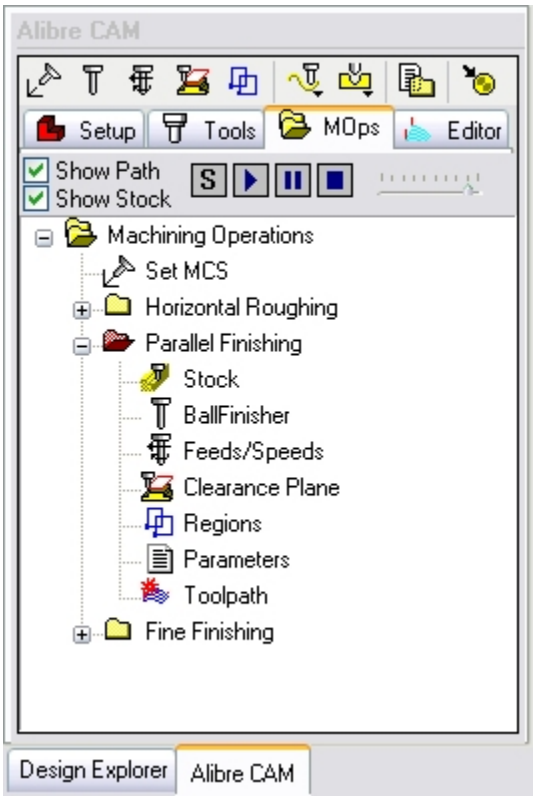
This is shown below.





The dialog shows the 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.



Transform Toolpath: Clicking on this button will bring up a Transform Toolpath dialog that allows the transformation of the toolpath. Toolpaths can be moved, rotated, scaled and mirrored.

Reverse Toolpath: Clicking on this button allows the reversal of the direction of the toolpath. All engage motions are changed to retract motions and all retract motions are converted to engage motions.

Instance Toolpath: Once a toolpath is generated, the user can create multiple instances of the toolpath in XY by clicking on this button and bringing up the following Instance Toolpath dialog.

Z instance Toolpath: Once a toolpath is generated, the user can create multiple instances of the toolpath in Z by clicking on this button and bringing up the following Z Instance Toolpath dialog.

Fit Arcs to Toolpath: Clicking on this button will bring up the Fit Arcs to Toolpath dialog that allows fitting of circular motions to point-to-point toolpath motions.

Linearize Toolpath: Clicking on this button will bring up the Linearize Toolpath dialog. 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.

 **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:** Clicking on this button will bring up the Move Selection in Z dialog to move in Z all tool motions that are currently selected. As with the cut and isolate actions, tool motions that intersect with the selection regions will be clipped and moved in Z.

 **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.

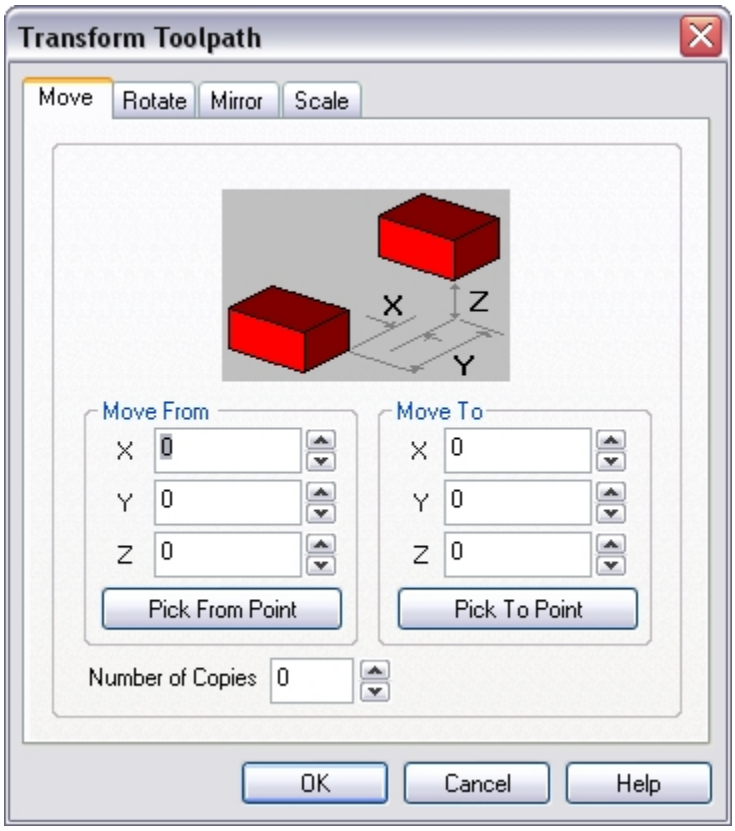
 **Save Toolpath Edits:** Saves the edited toolpaths.

Transform Toolpath

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

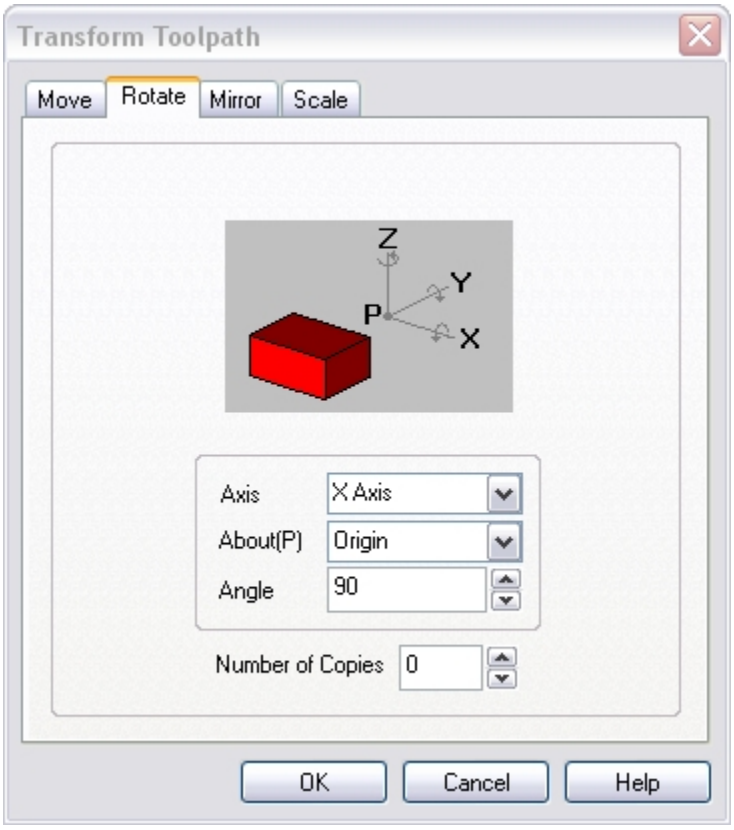
Move Toolpath

Allows the user to translate the toolpath by specifying the from and the 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



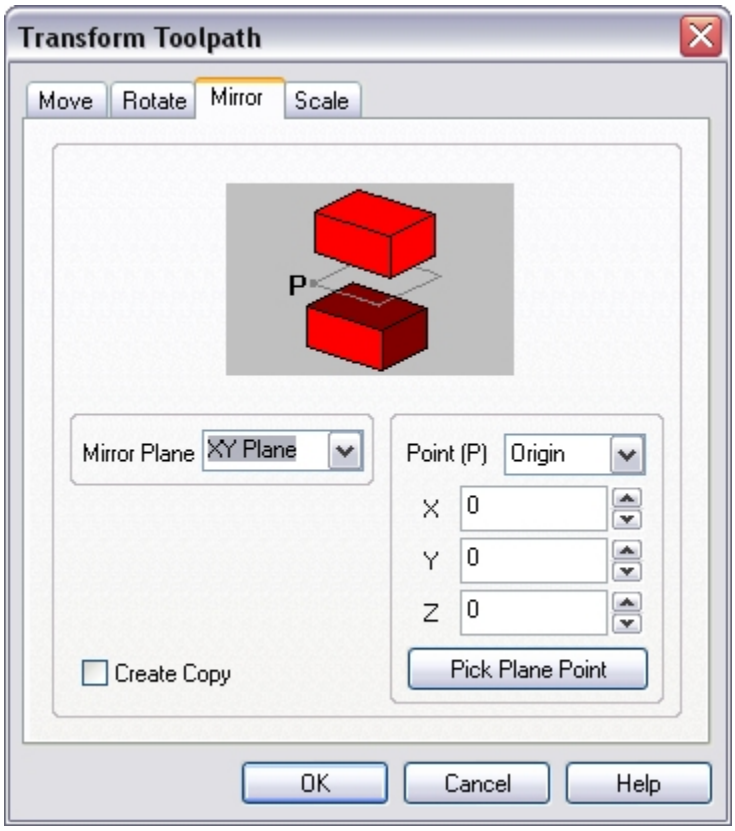
Rotate Toolpath

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.



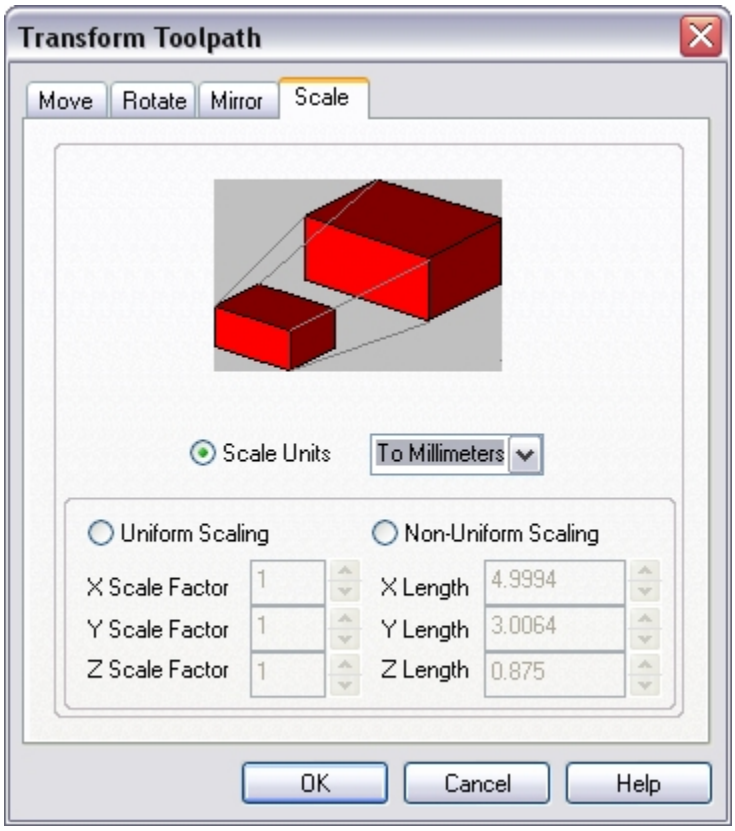
Mirror Toolapth

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



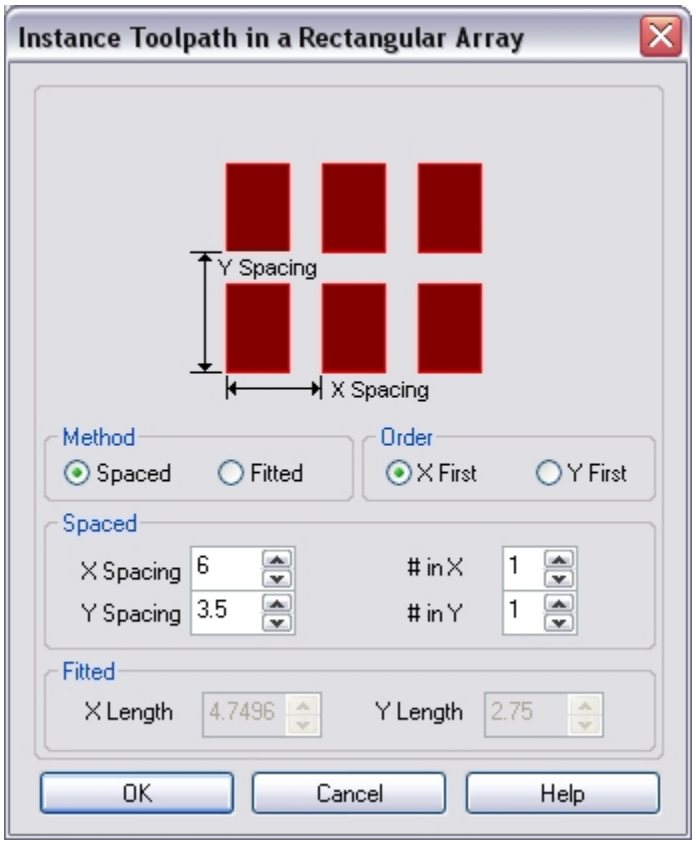
Scale Toolpath

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 a mm to an inch or vise 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.

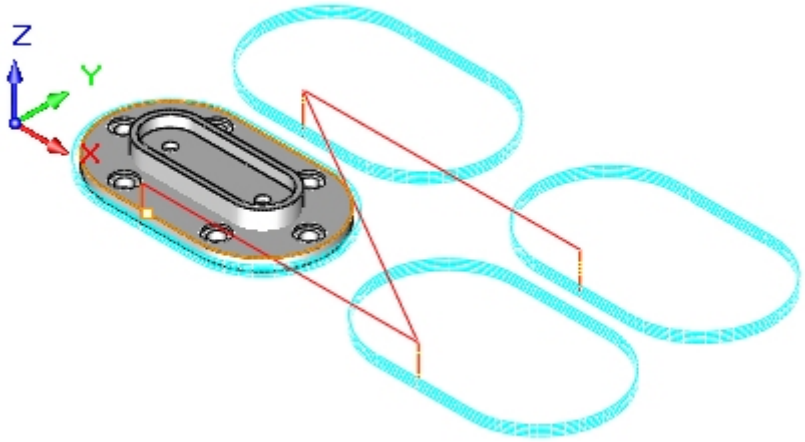


Instance Toolpath

Once a toolpath is generated, the 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.

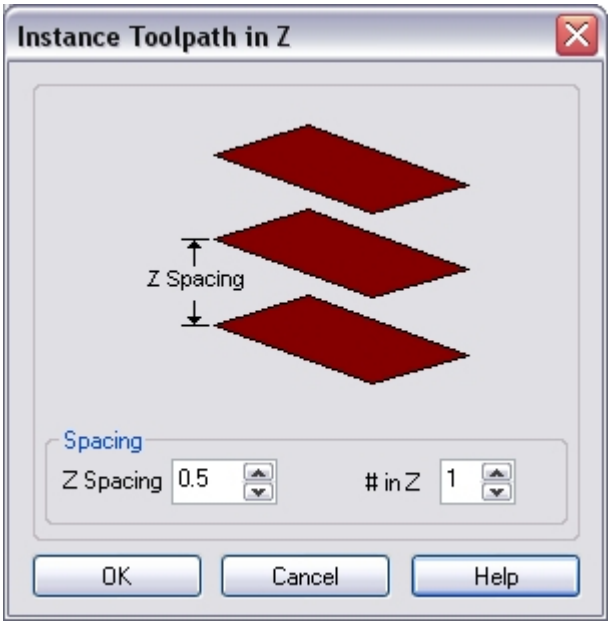


The user can specify multiple instances in the X and Y directions in two different ways. The first method, or the spaced method, the user specifies the independent X and Y spacing between the toolpaths. The spacing is then used and applied to the bounding box of the part geometry as shown in the picture 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.

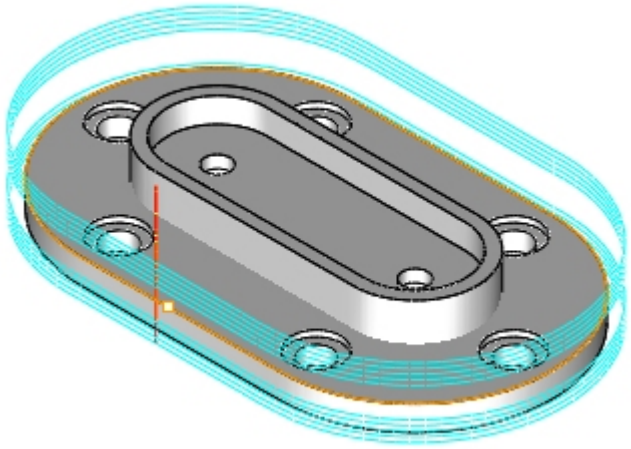


Z Instance Toolpath

Once a toolpath is generated, the 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.

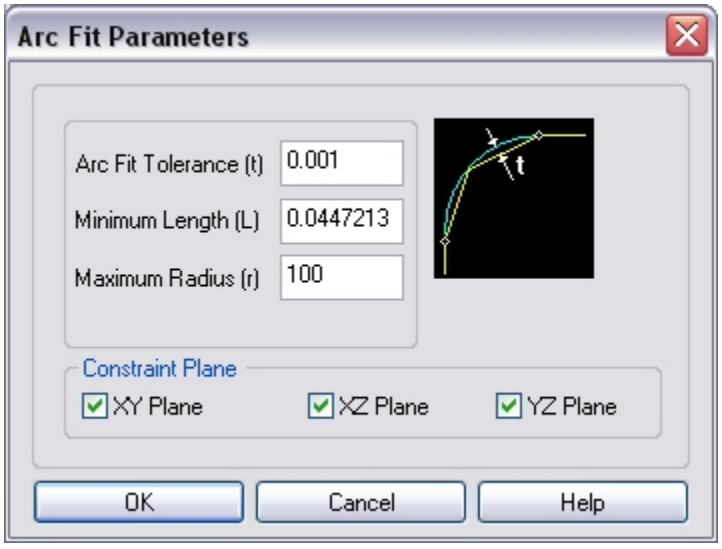


The user can specify the number of instances in Z as well as the spacing between the Z instances. The toolpath will be copied multiple times and the tool initial engage will start from the top-most instance and work its way to the bottom most instance. Negative values can be specified in the Z spacing.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

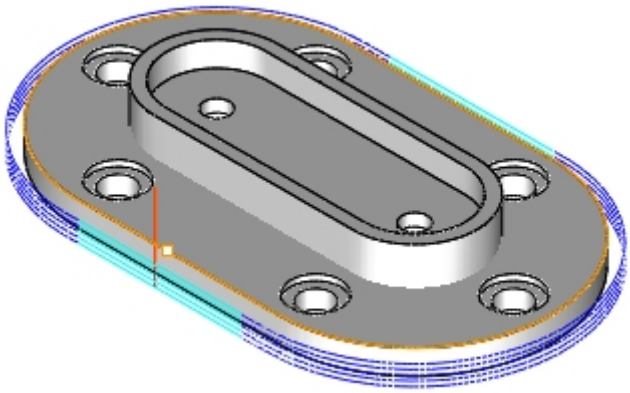


Fit Arcs to Toolpath

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.

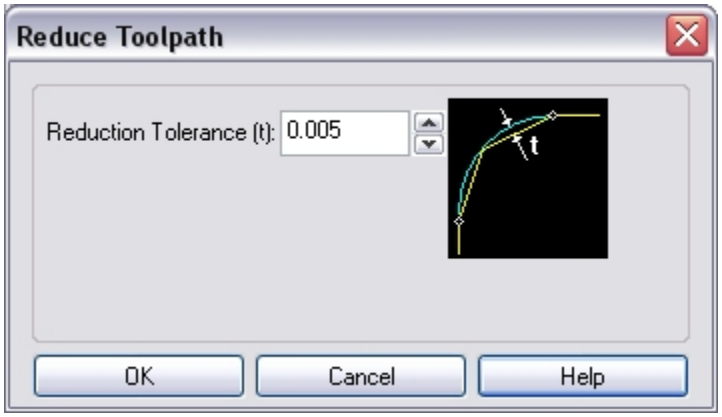


The user specifies an arc fitting tolerance, a limiting arc length as well as the smallest radius of the arc allowed. The arc motions can further be restricted to be created in any or all of the principal planes.



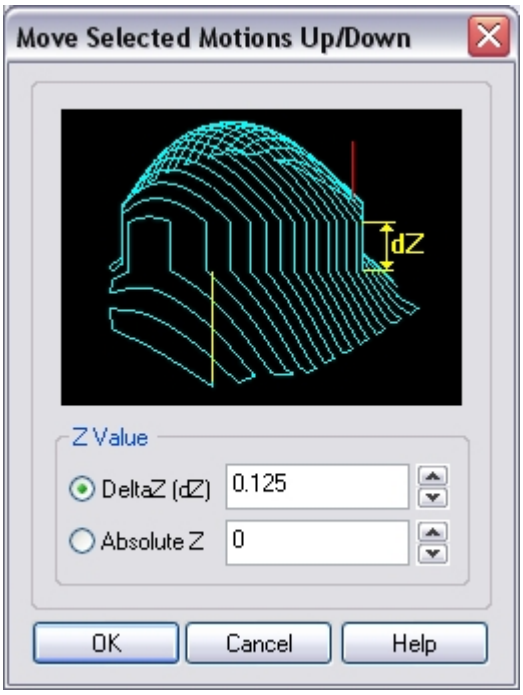
Linearize Toolpath

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.



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.



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.

Simulating Machining Operations

Cut material simulation available in Alibre CAM Standard, Expert and Professional versions only

Alibre CAM 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 Alibre CAM.

Toolpath simulation  or tool animation  can be performed on the selected or active operation. The active operation is the one that is selected and shown highlighted in the Machining Operations Browser. To simulate any other operation, simply select the operation in the browser and click on the simulate button on the dialog as shown below.

There are two kinds of toolpath simulation available in Alibre CAM. These are

Tool Animation

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 MOps tab of the browser or by using **right click** and simulate.

Tool Animation

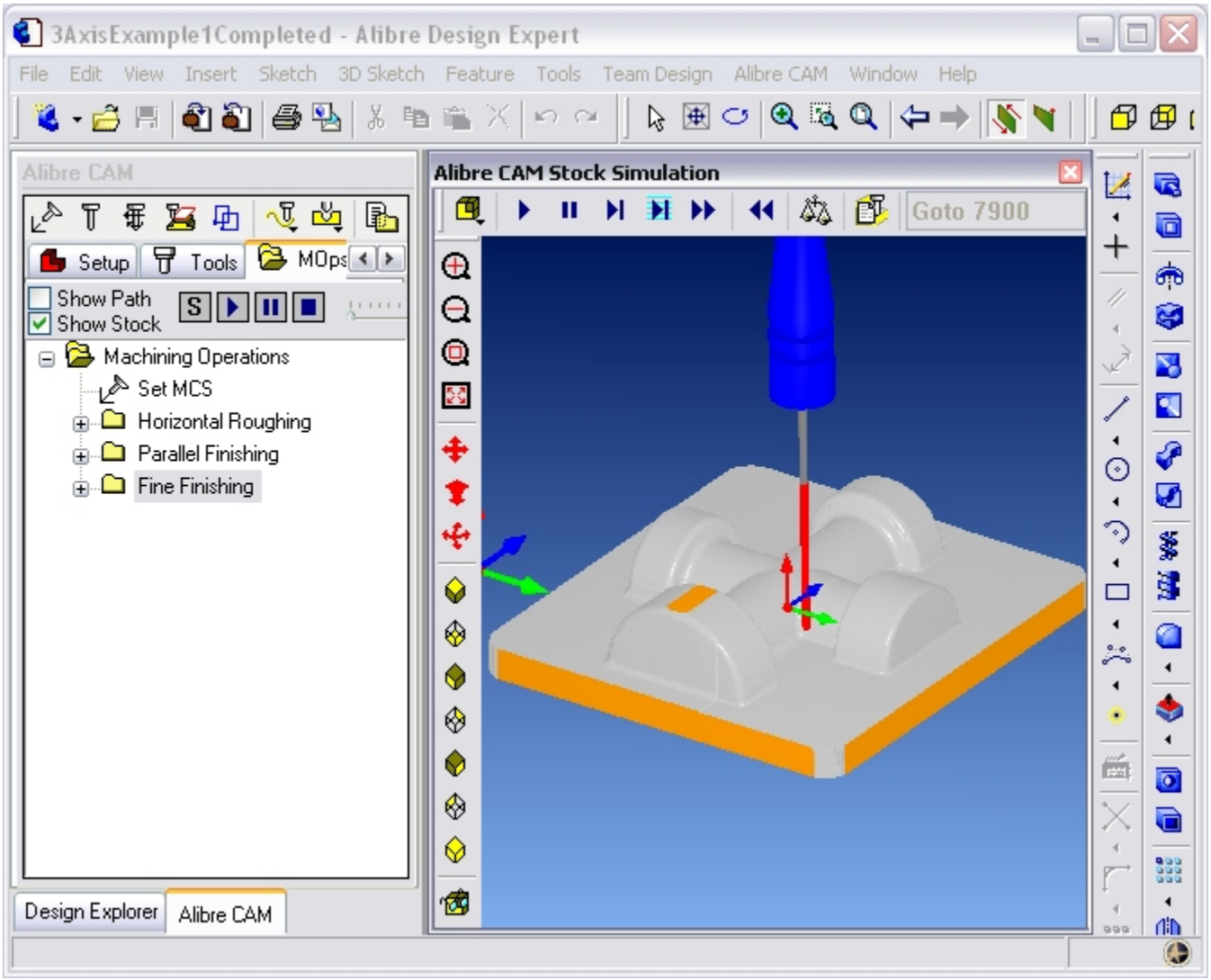
Simple tool animation can be carried out in Alibre CAM by using the controls on the MOps Browser. The tool can be animated to follow the toolpath by setting the step increment to the desired value and clicking on the  Animate button on the MOps tab of the browser or by selecting an operation and choosing right click to animate.

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

Cut Material Simulation

Available in Alibre CAM Standard, Expert and Professional versions only

As mentioned earlier, Alibre CAM offers very 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. Selecting Simulate  on the MOps tab launches the Alibre CAM Stock Simulation window. Using the controls in the stock simulation window the user can perform the cutting simulation.



Description of the Stock Simulation Window Controls

Create/Load Stock: This tool is used to create various types of stock geometry.

Simulate: The simulation will be run for the entire toolpath, and the end result of the material removal will be displayed.

Pauses: Pauses/Stops the simulation.

Step: The simulation will be performed for a specified number of toolpath motions. To set the step value, open the **Simulation Preferences** (the last icon on the toolbar), and adjust the **Maximum Display Interval**.

 **Step Z Levels:** Shows the resultant stock after each Z level. This is applicable only for machining methods that use multiple Z levels for machining, such as **Horizontal Roughing** and **Pocketing**.

 **Simulate to End:** Jumps to the end of the simulation.

 **Rewind:** Jumps to the start of the simulation.

 **Compare Part/Stock:** Performs a visual comparison of the stock model against the part model. You can color-code areas based on the amount of material remaining or overcut.

 **Simulation Settings:** Opens the **Simulation Preferences**, in which you can set various properties of the simulation and display.

Goto: Indicates the progression of toolpath simulation ("Goto"), displaying the number of the motion being simulated. When the simulation is complete, the last motion number will be displayed.

 **Zoom In:** Doubles the displayed size.

 **Zoom Out:** Halves the displayed size.

 **Zoom Box:** Zooms in on an area you specify by defining a rubber-banded rectangle.

 **Fit View:** Fits the entire part into display extents.

 **Dynamic Pan View:** Pans the view by holding and dragging the mouse.

 **Dynamic Zoom View:** Zooms the view by holding and dragging the mouse. Move the mouse up to zoom in, move the mouse down to zoom out.

 **Dynamic Rotate View:** Rotates the view by holding and dragging the mouse. The rotation follows the mouse movements as if there were an imaginary trackball at the center of the view.

 **Top View:** Displays the top view - the XY plane.

 **Bottom View:** Displays the bottom view

 **Front View:** Displays the front view - the YZ plane.

Back View: Displays the back view.

Right View: Displays the right view - the XZ plane.

Left View: Displays the left view.

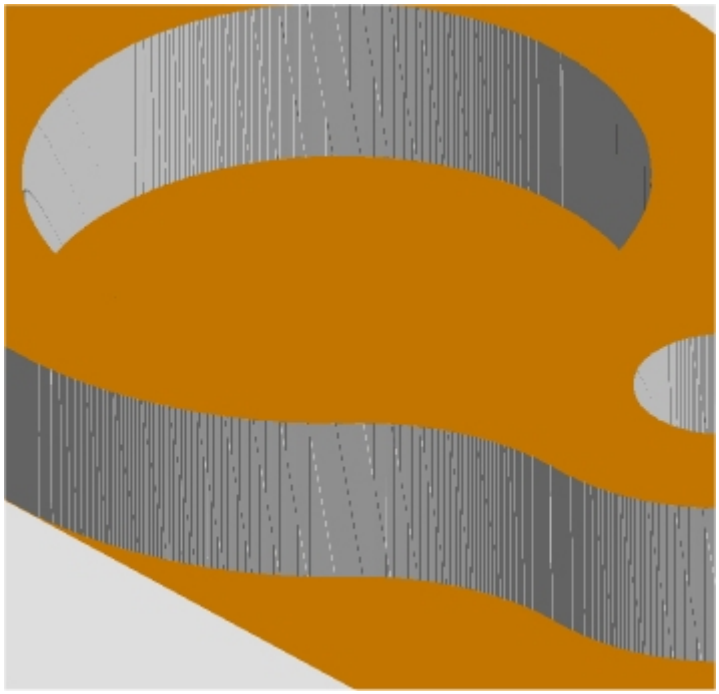
Iso View: Displays the model in isometric projection.

Hide Stock: Toggles the display of the stock geometry.

Hide Toolpath: Toggles the display of the toolpath associated with the current machining operation.

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 below.



Advanced Simulation Engine

The Alibre CAM Professional and Alibre CAM Expert products have two material removal simulation modes (or models) available for use. The Voxel model can only be used for 2½ and 3 axis toolpaths. The main advantage of this model is very rapid processing times. The Polygonal model is necessary for simulating all 4th axis toolpaths. This model is more comprehensive and results in better display quality, however at the expense of speed.

Note: If you are simulating 4th axis toolpaths make sure that you are using the Polygonal model.

To change the simulation module in Alibre CAM select Simulation Preferences located on the Alibre CAM Stock Simulation window or switch to the Setup tab in the browser and select Alibre CAM Preferences-> Simulation Settings button and select the appropriate model for simulation.

Refer to Simulation Preferences for detailed description.

Part/Stock Compare

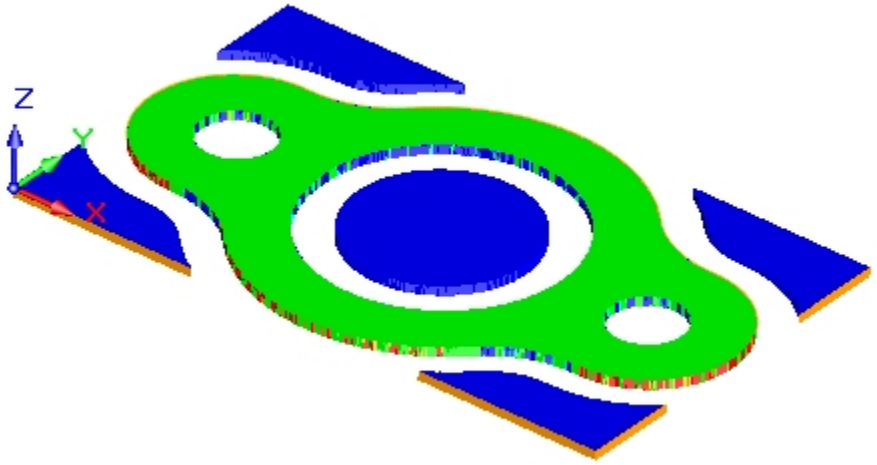
Available in Alibre CAM Expert only

This feature graphically compares the differences between the part and the stock geometry. The part geometry must contain solid/surface geometry.

Run the simulation from the Alibre CAM Stock Simulation window and select Part/Stock compare.

The following dialog is displayed and the user can specify the tolerance for part/stock comparison.

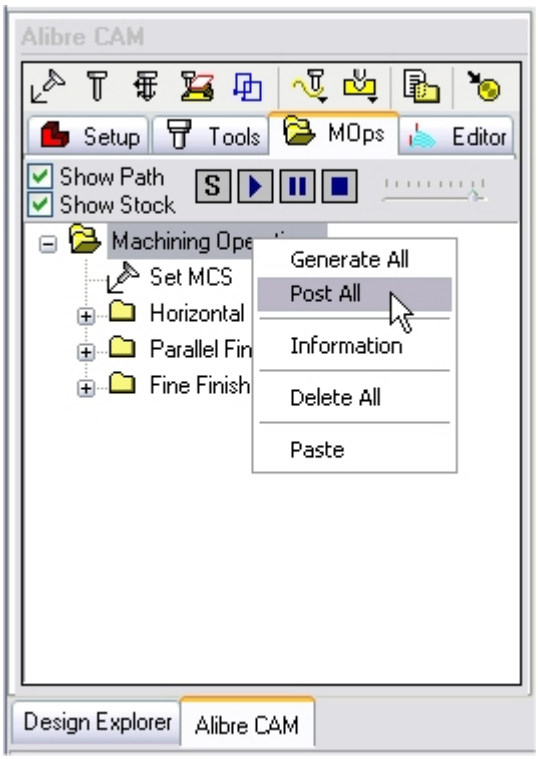
A visual comparison of the stock model against the part model is displayed. You can color-code areas based on the amount of material remaining or overcut.



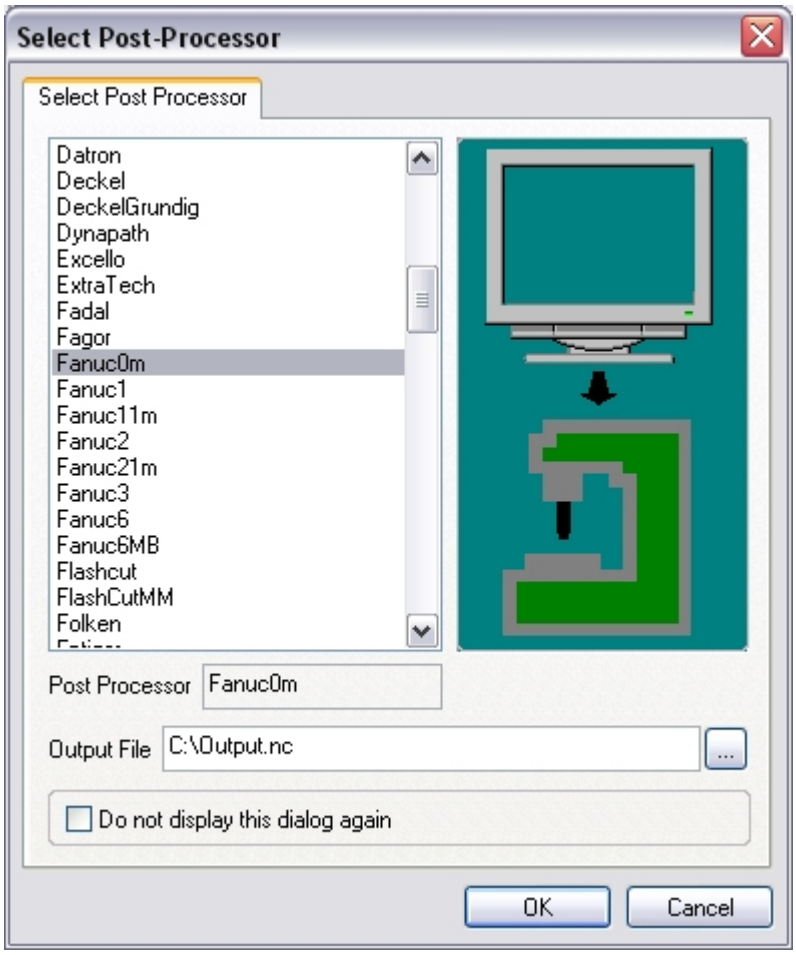
Post Processing Machining Operations

Once the toolpath has been generated, it can be post-processed to a specific machine controller. Alibre CAM Standard comes with a set of post-processors to choose from. Each post-processor is represented by an ***.spm** file, all of which are located in the **Posts** folder under the Alibre CAM installation folder.

You can post-process an individual toolpath, or all toolpaths at once. For post processing an individual toolpath, right-click on its name in the **Mops** tab of the Browser and select **Post**. You can Right-Click in the **Mops** tab on any machine operations. You will see the options shown in the picture below. Select **Post**.



This will bring up the following dialog.

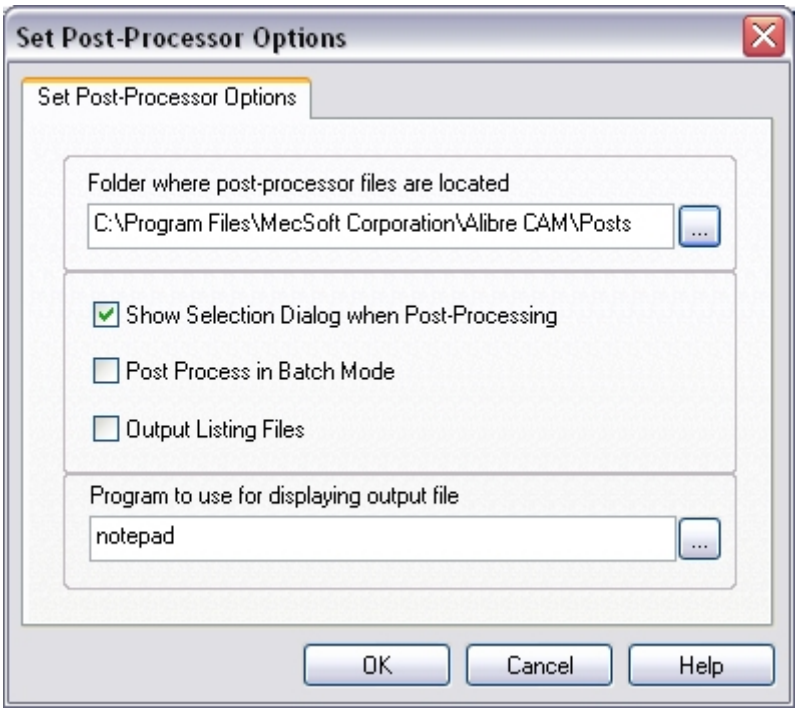


The standard Alibre CAM product comes with a set of over 100 postprocessors to choose from. The user can specify the folder to find the Post-Processor macro files by selecting the "Browser for Folder" button in the dialog. Alibre CAM uses macro files with a .spm extension to handle post-processing to different controllers. These files are typically located in the "Data" directory under the Alibre CAM installation folder. Alibre CAM by default looks in this directory to build the list of available post-processors shown in the "Post Process" dialog. However, the user has the ability to point to a different folder to select these macro files from by using this button.

To post process the chosen toolpath, the user simply picks the desired controller to post process the toolpath to from the list of postprocessors in the list box and assigns an output file where the post processed output needs to go to. Once the user clicks on the "Ok" button the dialog will be cancelled and the post processing will begin.

Set Post Options

The user can specify certain post-processor options for the Alibre CAM post-processor. This is done by selecting the **Set Post Options** from the toolbar in the Setup tab of the browser. This will bring up the following dialog.

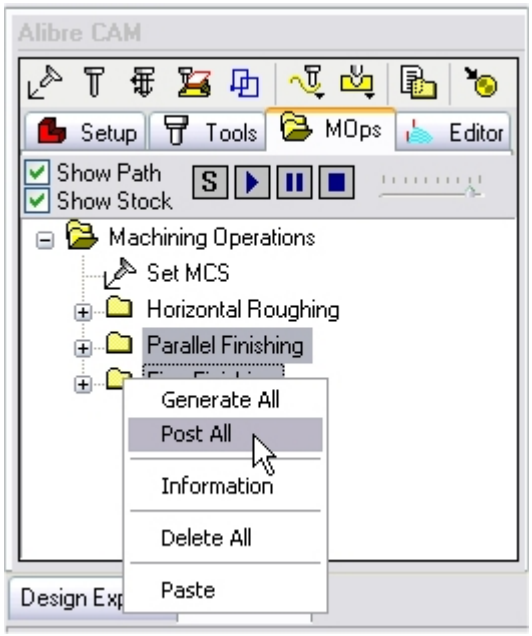


Alibre CAM by default performs interactive post-processing. That is, when the user selects a toolpath for post-processing, Alibre CAM launches the post-processor and waits for the post-processor to complete. The user can specify Alibre CAM not to wait for the completion of the post-processor. This is done by checking on the Batch Mode check box in this dialog. The user also has the option to output the list files and display the selection dialog for each post processing operation.

During interactive post-processing, Alibre CAM 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.

Post-processing Multiple Machining Operations

The user has the ability to select multiple operations or the entire set of machining operations and post process all of them with a single button click. To do this the simply select multiple operations in the browser (or the root level icon for post-processing the entire set) and click on the right mouse button. This will bring up an pop-up menu shown below. You now have the option of performing the post-processing operation of the entire set of operations by selecting the **Post All** option shown in the pop-up menu.



Archiving Machining Operations

Once machining operations are created they can be archived along with the Alibre Design part file. This can be accomplished by simply saving the part file. When the part file is retrieved, all archived operations will be loaded along with the Alibre CAM part file.

Please note that you need to be running a fully licensed copy of Alibre CAM before you are able to store Alibre CAM data in the Alibre Design part file.

Appendix I: Trouble shooting Alibre CAM Installation

If you have followed the installation steps outlined in the installation section correctly and are unable to load and run Alibre CAM correctly follow these troubleshooting steps to correct the problem.

Troubleshooting the Software Installation

Make sure that the software was correctly installed. To do this you can browse to the installation folder of Alibre CAM and make sure that the file **Alibre CAM.dll** is present. Also make sure that all the folders described in the following section are correctly installed. If you detect an incorrect installation, un-install the software completely and re-install the software. You can un-install the software by selecting the **Add or Remove Programs** option under the **Control Panel** settings of your computer.

Alibre CAM Installation Folder

Alibre CAM installation creates a main installation folder whose name and location you can specify during the installation process. If you accept the default location during the installation process the default location would be found under C:\Program Files\MecSoft Corporation\Alibre CAM. This

folder contains the Alibre CAM executable and library files. There are also several subfolders in the installation directory:

Data: Contains tool library files - **DefaultEnglishTools.csv** and **DefaultMetricTools.csv**. These files can be used as they are, or you can use them as templates and customize them with your own data. You will also find a speeds/feeds & material library file called **FEEDSSPEEDS**. For more information on how to modify these tool library files, please refer to Alibre CAM's online help.

Help: Contains the online help files used with Alibre CAM. You can open these files directly from this folder, or access them within Alibre CAM.

Posts: Contains the standard set of post-processor (*.spm) files. Additional post-processor files can be obtained from MecSoft Corporation. If you receive additional *.spm files, be sure to place them in this folder, so that Alibre CAM will recognize them.

Tutorials: Contains a tutorial and several part files to help first-time users get familiar with Alibre CAM. These tutorials can be located under C:\Program Files\MecSoft Corporation\Alibre CAM\Tutorials folder. Documentation for these tutorials can be launched by selecting the Alibre CAM menu entry in Alibre Design and select Getting Started-> Getting Started Tutorial.

If the Alibre CAM is installed and it does not load

Load Alibre Design and select **New Part**.

Go to **Tools** from the Menu and Select **Add-on Manager…**

Check the box next to Alibre CAM and click OK. This should now load the Alibre CAM entry on the menu bar. Now Select Alibre CAM and Browser from the main menu bar after exiting the add-on manager.

If you still are not able to load Alibre CAM contact us at support@mecsoft.com

Appendix II: Multiple Face Machining

Performing Machining Operations on Multiple Faces

You can perform machining operations on multiple faces of a part by using Alibre Design's Part Boolean function. Combined with configurations, you can create G Code for multiple machining operations on multiple faces within the same part workspace. Though this is not the typical use for a Part Boolean operation, it can successfully be used for this purpose. This portion of the Help assumes you are familiar with Part Booleans, have a general knowledge of assembly constraints, and have a basic knowledge of Configurations. If you need more detail on any of those items, please see the Alibre Design Help for clarification.

Workflow

The workflow you will use when trying to create multiple machining operations on multiple faces detailed below.

Orienting the Part Correctly

1. Create the part in Alibre Design and save it. Close the part.
2. Open a new part workspace.
3. Create a Boolean Unite with only your original part.
    a. Select the Boolean Unite tool  from the modeling toolbar.
    b. You are then prompted to select a part to insert.
    c. Locate your saved part from step 1 and click OK. Click once to place your part, then click Finish in the dialog.
4. Orient your part correctly within the Boolean workspace.
    a. To machine on a particular face, you need to orient the part in a certain way. There are two main concepts you must use.
        1. The face you are machining on must be parallel to the XY plane. You need to create a mate or align constraint between the XY plane and the planar face you are machining on.
        2. The face must be "facing" the positive Z direction. This will determine whether you use a mate or align constraint.
5. Exit the Boolean workspace by accepting the changes using the  button. You have just reoriented the part so that the tool, coming from the positive Z direction towards the negative Z direction, can machine on this face. Now you will need to repeat the process for each face you would like to machine on. However, you can accomplish this all within the same workspace with configurations, detailed below.

Repeating the Process with Configurations

6. Once you are done orienting your part, create a new Configuration for the next orientation. In the New Configuration dialog, select the Lock All button and the Active checkbox at the bottom. Click OK.
7. Now you are active in Config <2>. Right click Config <1> from the Design Explorer and select Lock. Now, changes we make in Config <2> will not affect Config <1>.
8. Suppress the first Boolean Unite feature in the Design Explorer. Now, insert a new Boolean Unite feature in the same fashion you did in the above section. This time, however, the face you will orient with the XY plane will be the next face you would like to generate G Code for.
9. Repeat this process for all the faces you need to machine on. For example, if you needed to machine details on all 6 sides of a cube, you would need 6 configurations. When you switch between the configurations, you should see the same part basically appear to rotate by 90 degrees, each time orienting the face to be machined on towards the positive Z direction.

Creating Machining Operations with Alibre CAM

Now that you have all your configurations set up for all the different orientations you will require, you can create machining operations for each different configuration. Switch to the Alibre CAM browser.

10. In Alibre CAM, create the machining operations for Config<1>.
11. Switch to the Design Explorer using the tab below the Alibre CAM browser. In Alibre Design, activate the next Configuration, in this case Config <2>.
12. Switch back to the Alibre CAM browser. In Alibre CAM, create the machining operations for Config<2>.
13. Repeat as necessary.

Hint: You can name each machining operation you create it. It may be useful to name the operations after the Configuration the operation is using. That way, you know that the Config <4> configuration goes with the Config <4> machining operation, for example.

Appendix III: Where to go for more help

In addition to the features described in this guide, Alibre CAM has many more features designed to make it easier for you to create toolpaths and G-code. Alibre CAM's complete on-line help provides reference information for each of Alibre CAM's features and functions.

If you need additional help, or if you have any questions regarding Alibre CAM, first try the FAQ section on our web site, www.alibre.com. Most of the common issues that users face are cataloged here. If you still have additional questions, visit our Users Forum at our web site to learn from other Alibre CAM users. You can also contact us via e-mail at support@alibre.com

Lista Studio srl
www.lista.it
Borgo Belvigo 33
36016 Thiene Vi ITALY
Tel. 0445,382056
info@lista.it