VoluMill™

REFERENCE GUIDE

VERSION 6.1

EDUCATIONAL SERVICES

Developed by Celeritive™

VoluMill™ NEXION™ Reference Guide

VoluMill™ NEXION™ Reference Guide
Date: May 2014

U.S. Patent 8,295,972. Other patents pending.
Software: VoluMill™ NEXION™ version 6.1

IMPORTANT NOTICE!

The accompanying executable code version of VoluMill™ and related documentation (the "Product") is made available to you under the terms of this VOLUMILL™ END-USER SOFTWARE LICENSE AGREEMENT (THE "AGREEMENT"). BY CLICKING THE "ACCEPT" BUTTON, OR BY INSTALLING OR USING THE PRODUCT, YOU ARE CONSENTING TO BE BOUND BY THE AGREEMENT. IF YOU DO NOT AGREE TO THE TERMS AND CONDITIONS OF THIS AGREEMENT, DO NOT CLICK THE "ACCEPT" BUTTON, AND DO NOT INSTALL OR USE ANY PART OF THE PRODUCT.

LICENSE GRANT

Celeritive Technologies, Inc. ("Celeritive") grants you a non-exclusive license to use the executable code version of the Product. This Agreement will also govern any software upgrades provided by Celeritive that replace and/or supplement the original Product, unless such upgrades are accompanied by a separate license, in which case the terms of that license will govern.

TERMINATION

If you breach this Agreement your right to use the Product will terminate immediately and without notice, but all provisions of this Agreement except the License Grant (Paragraph 1) will survive termination and continue in effect. Upon termination, you must destroy all copies of the Product.

PROPRIETARY RIGHTS

Portions of the Product are available in source code form under the terms of the GNU Lesser General Public License, Apache Software License, and other open source licenses (collectively, "Open Source Licenses") at http://www.celeritive.com. Nothing in this Agreement will be construed to limit any rights granted under the Open Source Licenses. Subject to the foregoing, Celeritive Technologies, for itself and on behalf of its licensors, hereby reserves all intellectual property rights in the Product, except for the rights expressly granted in this Agreement. You may not remove or alter any trademark, logo, copyright or other proprietary notice in or on the Product. This license does not grant you any right to use the trademarks, service marks or logos of Celeritive Technologies or its licensors.

DISCLAIMER OF WARRANTY

THE PRODUCT IS PROVIDED "AS IS" WITH ALL FAULTS. TO THE EXTENT PERMITTED BY LAW, CELERITIVE TECHNOLOGIES AND CELERITIVE'S DISTRIBUTORS AND LICENSORS HEREBY DISCLAIM ALL WARRANTIES, WHETHER EXPRESS OR IMPLIED, INCLUDING WITHOUT LIMITATION WARRANTIES THAT THE PRODUCT IS FREE OF DEFECTS, MERCHANTABLE, FIT FOR A PARTICULAR PURPOSE AND NON-INFRINGING. YOU BEAR ENTIRE RISK AS TO SELECTING THE PRODUCT FOR YOUR PURPOSES AND AS TO THE QUALITY AND PERFORMANCE OF THE PRODUCT. THIS LIMITATION WILL APPLY NOTWITHSTANDING THE FAILURE OF ESSENTIAL PURPOSE OF ANY REMEDY. SOME JURISDICTIONS DO NOT ALLOW THE EXCLUSION OR LIMITATION OF IMPLIED WARRANTIES, SO THIS DISCLAIMER MAY NOT APPLY TO YOU.

LIMITATION OF LIABILITY

EXCEPT AS REQUIRED BY LAW, CELERITIVE TECHNOLOGIES AND ITS DISTRIBUTORS, DIRECTORS, LICENSORS, CONTRIBUTORS AND AGENTS (COLLECTIVELY, THE "CELERITIVE GROUP") WILL NOT BE LIABLE FOR ANY INDIRECT, SPECIAL, INCIDENTAL, CONSEQUENTIAL OR EXEMPLARY DAMAGES ARISING OUT OF OR IN ANY WAY RELATING TO THIS AGREEMENT OR THE USE OF OR INABILITY TO USE THE PRODUCT, INCLUDING WITHOUT LIMITATION DAMAGES FOR LOSS OF GOODWILL, WORK STOPPAGE, LOST PROFITS, LOSS OF DATA, AND COMPUTER FAILURE OR MALFUNCTION, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGES AND REGARDLESS OF THE THEORY (CONTRACT, TORT OR OTHERWISE) UPON WHICH SUCH CLAIM IS BASED. THE CELERITIVE GROUP'S COLLECTIVE LIABILITY UNDER THIS AGREEMENT WILL NOT EXCEED THE GREATER OF $500 (FIVE HUNDRED DOLLARS) AND THE FEES PAID BY YOU UNDER THIS LICENSE (IF ANY). SOME JURISDICTIONS DO NOT ALLOW THE EXCLUSION OR LIMITATION OF INCIDENTAL, CONSEQUENTIAL OR SPECIAL DAMAGES, SO THIS EXCLUSION AND LIMITATION MAY NOT APPLY TO YOU.

EXPORT CONTROLS

This license is subject to all applicable export restrictions. You must comply with all export and import laws and restrictions and regulations of any United States or foreign agency or authority relating to the Product and its use.

U.S. GOVERNMENT END-USERS

The Product is a "commercial item," as that term is defined in 48 C.F.R. 2.101, consisting of "commercial computer software" and "commercial computer software documentation," as such terms are used in 48 C.F.R. 12.212 (Sept. 1995) and 48 C.F.R. 227.7202 (June 1995). Consistent with 48 C.F.R. 12.212, 48 C.F.R. 27.405(b)(2) (June 1998) and 48 C.F.R. 227.7202, all U.S. Government End Users acquire the Product with only those rights as set forth herein.

MISCELLANEOUS

(a) This Agreement constitutes the entire agreement between Celeritive and you concerning the subject matter hereof, and it may only be modified by a written amendment signed by an authorized executive of Celeritive. (b) Except to the extent applicable law, if any, provides otherwise, this Agreement will be governed by the laws of the state of Arizona, U.S.A., excluding its conflict of law provisions. (c) This Agreement will not be governed by the United Nations Convention on Contracts for the International Sale of Goods. (d) If any part of this Agreement is held invalid or unenforceable, that part will be construed to reflect the parties' original intent, and the remaining portions will remain in full force and effect. (e) A waiver by either party of any term or condition of this Agreement or any breach thereof, in any one instance, will not waive such term or condition or any subsequent breach thereof. (f) Except as required by law, the controlling language of this Agreement is English. (g) You may assign your rights under this Agreement to any party that consents to, and agrees to be bound by, its terms; Celeritive Technologies, Inc. may assign its rights under this Agreement without condition. (h) This Agreement will be binding upon and will inure to the benefit of the parties, their successors and permitted assigns.

THIRD PARTY SOFTWARE

This Product includes software developed by the OpenSSL Project for use in the OpenSSL Toolkit (http://www.openssl.org/). This Product includes cryptographic software written by Eric Young (eay@cryptsoft.com). By using the Product, you agree to be bound by the licenses of these Third Party products. These Third Party licenses are attached here for your reference.

CONTENTS

Chapter 1

INTRODUCTION

This chapter covers the following topics:

- Installation
- Licensing
- Technical Support

Welcome to the VoluMill™ NEXION Reference Guide. This guide is designed to provide all the information possible in using VoluMill NEXION to complement your current CAD/CAM system. This guide can be used to jump right into using the software or it can be read cover-to-cover.

VoluMill NEXION is a stand-alone system capable of creating VoluMill toolpaths on wireframe geometry and surface models generated from your CAD/CAM system. VoluMill is an ultra high-performance technology developed by Celeritive Technologies, Inc. to be used in place of traditional roughing methods when reducing cycle times, extending tool life, and reducing the stress on machine tools is a priority. A VoluMill toolpath is designed to never exceed a defined Material Removal Rate during the entire program. See www.volumill.com for more information on the engine and its toolpath.

A license is required to use VoluMill. To purchase VoluMill NEXION, please contact your local authorized VoluMill reseller or Celeritive Technologies directly through our website, www.volumill.com, call us at (888) 253-6701, or email sales@celeritive.com.

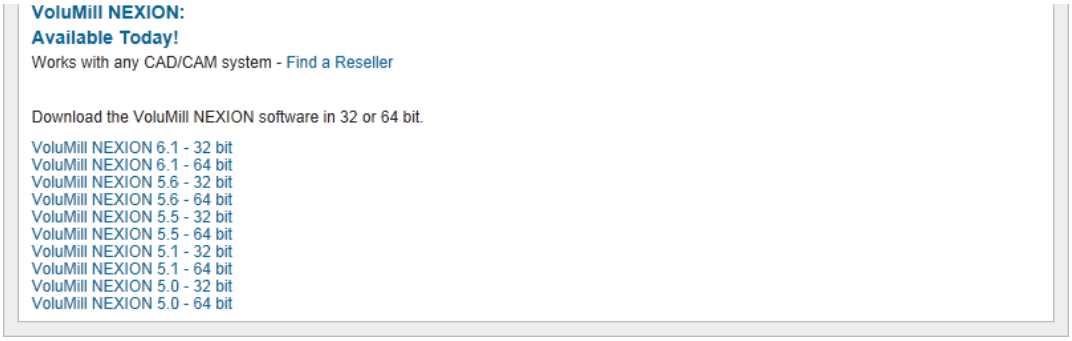
Installation

VoluMill NEXION is available as a download from the Celeritive Technologies website.

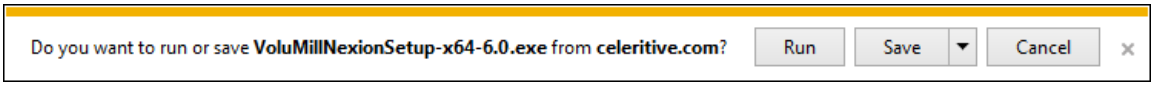
To download and install VoluMill NEXION:

1) Go to www.volumill.com, and then select Product Downloads from the Products menu:

2) Locate VoluMill NEXION, then select the appropriate link:

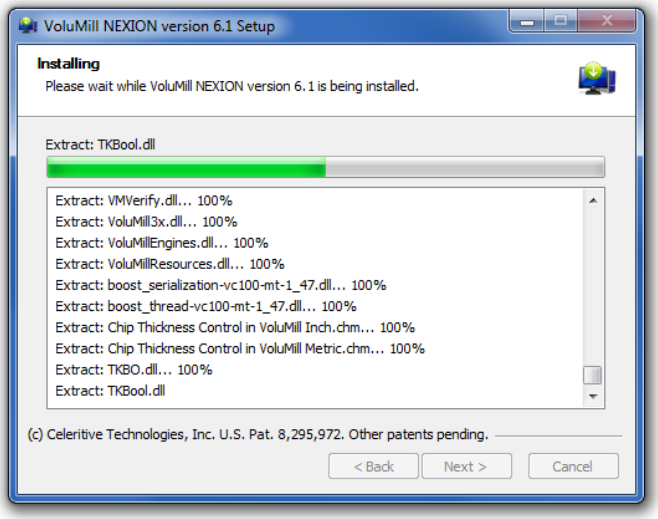


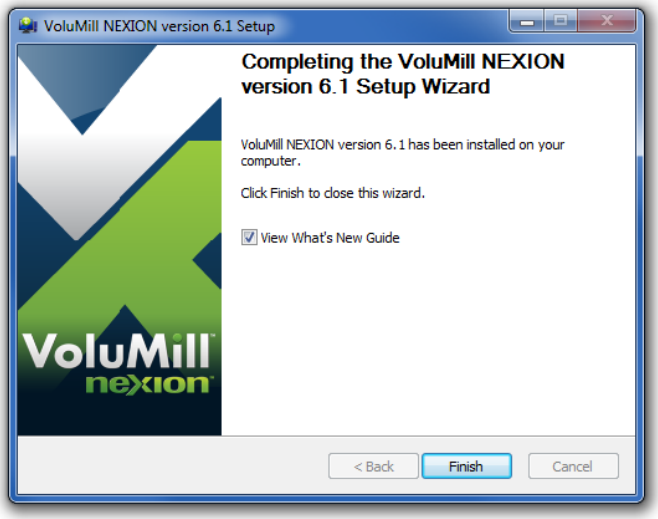
A dialog will appear asking you to either run or save the file. If you select Run, the software will be downloaded into your computer's temporary folder and the installation program will begin automatically.



If you select Save, you may be prompted for a location on your computer to save the software to. Once it's finished downloading, you'll need to navigate to the location the file was saved to and either double-click, or right-click on the file and select Run.

3) The installation Wizard will open and step you through the process.





Licensing

In order to use VoluMill you must have an activated license.

Please refer to the VoluMill Licensing Guide for information on how to install and activate licenses. It can be found in the VoluMill NEXION program group.

Technical Support

Our Technical Support department is available to answer your questions Monday through Friday, 8:00 AM to 5:00 PM. Technical Support is available to all users on maintenance.

Celeritive Technologies maintains a permanent presence on the World Wide Web:

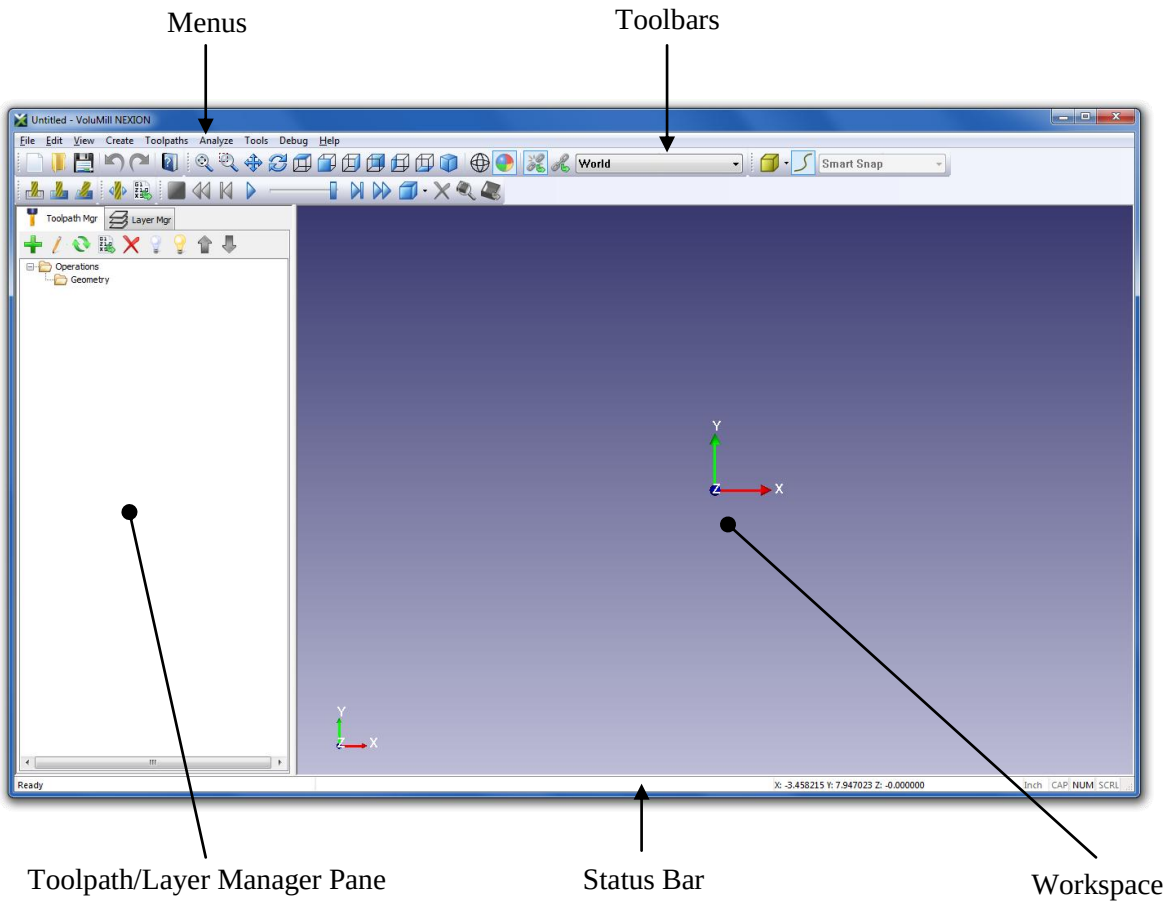
The Celeritive Technologies web site contains company news, product information, email links, user forums and much more. It is the preferred means of connecting to Celeritive Technologies electronically. The Celeritive Technologies website is located at www.celeritive.com or www.volumill.com.

The Celeritive website provides a support page to report any difficulties found while using VoluMill. To access the support section of the website, select Bug Reports from the Support menu: www.celeritive.com/bugreport.htm. Alternatively, you may email support directly. If you need to send a file, please include it as an attachment. When sending files, it is extremely helpful to include a contact name and phone number and a brief description of any issues. To send email to support, use: support@celeritive.com.

Chapter 2

User Interface

VoluMill NEXION uses an industry-standard style interface with common controls to make it easy to use.



Menus

File

New

This choice will close the current file and begin a new session. If the current file has any changes since it was last saved, a dialog will appear that provides an option to save the file.

Open…

This will display the File Open dialog to load a file. If the current file has any changes since it was last saved, a dialog will appear that provides an option to save the file.

VoluMill NEXION supports DXF, IGES, STEP, and STL files.

When saving a file, a ".vmuc" extension is used.

Merge…

Merge functions the same way Open does except that the current file is not closed.

Save

This will save the current changes made to the file. If the file has not yet been saved a dialog will appear to define the location and name of the file.

Save As…

This will display a dialog to save the file to a different name.

Recent Files

The last four files are listed for quick access.

Exit

This will close any open file and VoluMill NEXION

Edit

Undo

This choice reverses the last action.

Redo

This choice reverses the last Undo action.

Copy

This copies the current selection. This works within the Toolpath and Layer panes only.

Paste

This pastes the current selection. This works within the Toolpath and Layer panes only.

View

Fit

This choice adjusts the display scale so the geometry fits within the workspace.

Zoom Rectangle

This choice is used to zoom into a specific area within the workspace.

Pan

This choice is used to move the part within the workspace. Using this choice switches into a panning mode. To exit from this mode press the Esc key, or deselect the option.

Rotate

This choice is used to dynamically rotate the part within the workspace. Using this choice switches into a rotation mode. To exit from this mode press the Esc key, or deselect the option.

Top View through Isometric View

These choices display the part in its respective view.

Wireframe

This choice changes any body and back-plotting tool to a wireframe or polygon display mode.

Shaded

This choice changes any body and back-plotting tool to a rendered display mode.

Relative to World CS

This choice makes the available views relative to the world coordinate system.

Relative to Active CS

This choice makes the available views relative to the active coordinate system.

Show Active CS

This choice toggles the display of the current coordinate system's axes.

Create

Rectangular Body…

This choice creates a rectangular body that can be chosen as stock, check, etc.

Coordinates for the opposing corners may be entered or the Bounding box button may be used to automatically calculate the stock to contain the chosen body.

Cylindrical Body…

This choice creates a cylindrical body that can be chosen as stock, check, etc.

Values for the cylinder's center, length, diameter, and corners may be entered as well as the axis of orientation. The Bounding cylinder button may be used to automatically calculate the stock to contain the body.

Toolpaths

These choices are used to create VoluMill toolpaths for 2-, 3-, and 5-Axis applications.

Analyze

Measure

This opens the Measure dialog.

Tools

Options…

This opens the Options dialog.

Help

Help…

This choice opens the VoluMill NEXION Reference Guide (this document).

About…

This choice displays the current version and build number.

Toolbars

Main

New

This choice will close the current file and begin a new session. If the current file has any changes since it was last saved, a dialog will appear that provides an option to save the file.

Open

This will display the File Open dialog to load a file. If the current file has any changes since it was last saved, a dialog will appear that provides an option to save the file.

VoluMill NEXION supports DXF and STL files. DXF files are required for 2-Axis operations while STL files are required for 3-Axis operations.

When saving a file, a ".vmuc" extension is used.

Save

This will save the current changes made to the file. If the file has not yet been saved a dialog will appear to define the location and name of the file.

Undo

This choice reverses the last action.

Redo

This choice reverses the last Undo action.

About

Displays the version and build number.

View

Fit

This choice adjusts the display scale so the geometry fits within the workspace.

Zoom Rectangle

This choice is used to zoom into a specific area within the workspace.

Pan

This choice is used to move the part within the workspace. Using this choice switches into a panning mode. To exit from this mode press the Esc key.

Rotate

This choice is used to dynamically rotate the part within the workspace. Using this choice switches into a rotation mode. To exit from this mode press the Esc key.

Top View through Isometric View

These choices display the part in its respective view.

Shaded

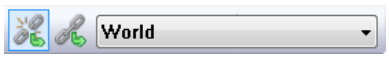
This choice changes any model and back-plotting tool to a rendered display mode.

Wireframe

This choice changes any model, and back-plotting tool to a wireframe or polygon display mode.

Relative to World or Active CS

These choices change how the standard views orient the model. Relative to World CS makes the views relative to the default coordinate system. Relative to Active CS makes the views relative to the active coordinate system displayed in the Coordinate System pull-down.



Active Coordinate System

This lists the currently available coordinate systems and provides a choice to create new ones. Whichever coordinate system is chosen will be used when creating new toolpaths.

The creation of coordinate systems is explained in detail later in this guide.

Selection

Solid/Face

This defines what can be selected. In most cases Solid can be used to select the part, stock, check, bodies. Face will most often be used to select wall geometry for 5-Axis toolpaths.

Curve

This is used to include the selection of curve/wireframe geometry.

Toolpath 

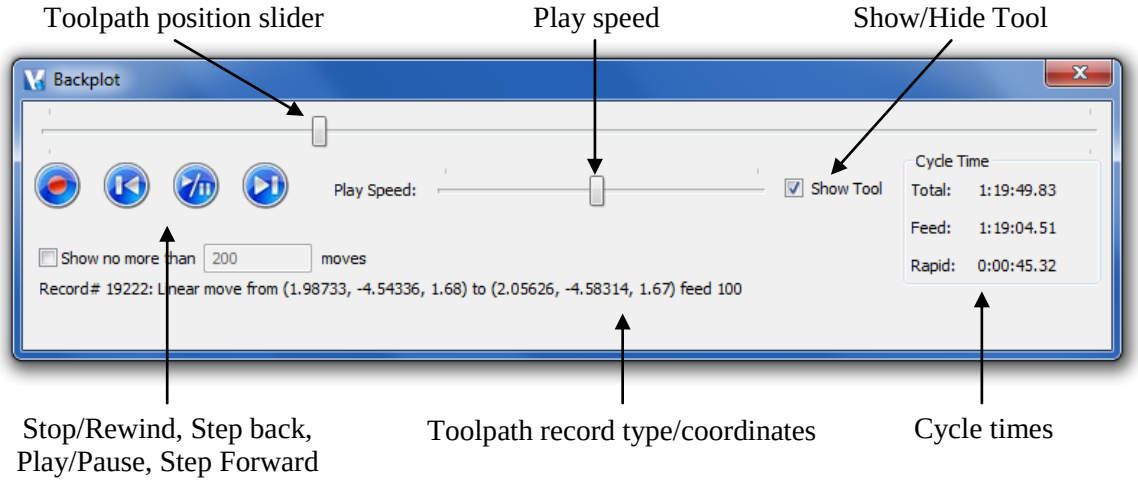
These choices are used to create VoluMill toolpaths for 2-, 3-, and 5-Axis applications.

Tools 

These choices are used to backplot and post process toolpaths

Backplot

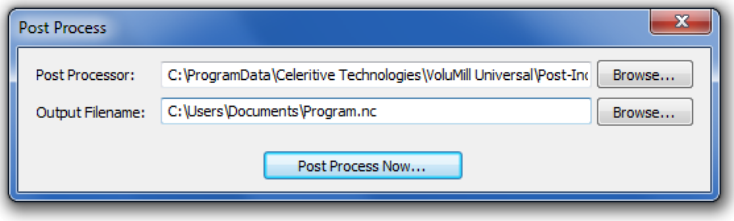
To animate the tool motion, highlight the desired toolpath, and choose the Backplot icon, or right-click and choose Backplot...



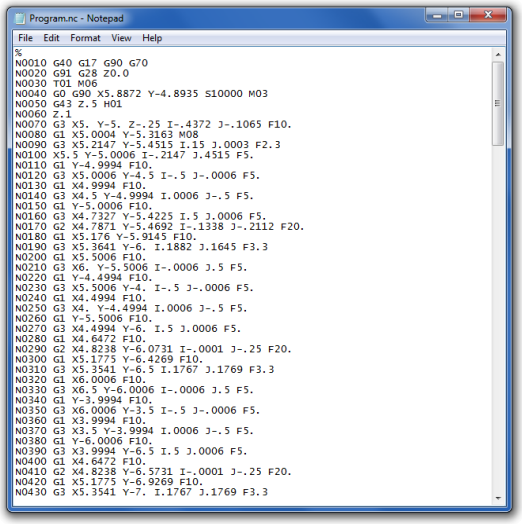
Arrow keys can be used to step forward or backward.

Post Process

This icon will post process the currently checked toolpaths.



Push the **Post Process Now…** button and the system will generate G-code for the VoluMill toolpath and automatically display the code.



Verification 

Stop

This exits the verification.

Rewind

This resets the verification to the beginning.

Step Backward

This moves the tool one step backward and replaces the material that was removed.

The left and down arrow keys may be used to step backward.

Play/Pause

This starts or pauses the verification.

The middle-mouse button may be used to pause/play during verification.

Speed Slider

This adjusts the play speed during verification.

Step Forward

This moves the tool one step forward.

The up and right arrow keys may be used to step forward.

Fast Forward/Pause

This plays/pauses the verification in turbo mode. The display is updated every 1,000 moves. This mode is different than regular verification and there's no speed control, step forward/backward, etc. The only way to get access to the regular mode is to Stop the verification and start again.

Display Mode

There are three display modes available when running verification in standard mode. Shaded, Shaded with Edges, and Wireframe.

Compare

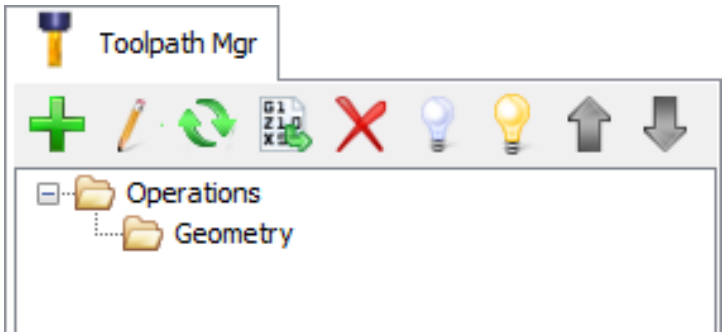
This compares the difference between the part and machined material. Colors displays are based on tolerances defined within the Verification section in the Options dialog.

In-Process Material

This creates an STL file of the current state of material removal. This is commonly used to import into another CAD/CAM system for finishing.

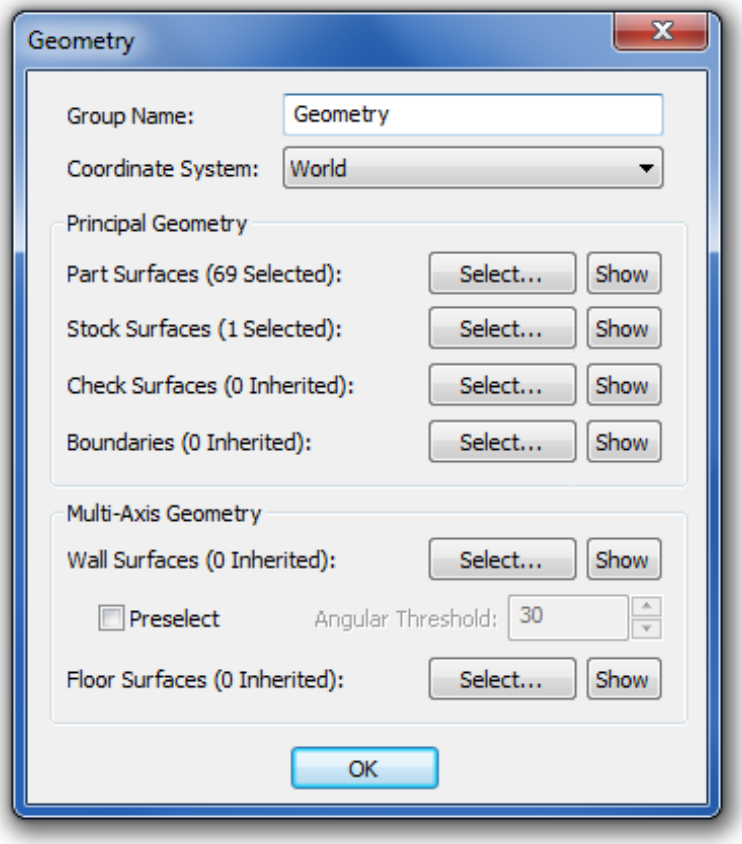
Toolpath Manager

The Toolpath Manager is used to define the geometry and toolpaths to machine the part. The desired geometry must be defined separately before a toolpath can be generated to machine it.



Geometry

To define the desired geometry to machine select "Geometry" in the tree and select the edit icon, or double-click, or right-click and select Edit...



Group Name

This is the name of the Geometry group.

Coordinate System

This provides a list of available coordinate systems defined using the Coordinate System list.

Part Surfaces

This is used to select surfaces that define the part to machine. These are used for VoluMill 3-/5-Axis toolpaths only. The number of surfaces have been selected are displayed as a reference.

Stock Surfaces

This is used to select surfaces that define the stock to machine from. These are used for VoluMill 3-/5-Axis Axis toolpaths only. The number of surfaces have been selected are displayed as a reference.

Check Surfaces

This is used to select surfaces that are not to be machined. These are used for VoluMill 3-/5-Axis toolpaths only. The number of surfaces have been selected are displayed as a reference.

Boundaries

This is used to select wireframe geometry.

For VoluMill 2-Axis toolpaths boundaries are actually used to define the part. For VoluMill 3-/5-Axis toolpaths they are used to constrain the toolpath.

The number of chains are displayed as a reference.

Wall Surfaces - Multi-Axis Geometry

This is used to select surfaces that define the walls of the part to machine. These are used for VoluMill 5-Axis toolpaths only. The number of surfaces have been selected are displayed as a reference.

Preselect - Multi-Axis Geometry

This is used in conjunction with Walls selection to select surfaces that define the walls of the part to machine. These are used for VoluMill 5-Axis toolpaths only. If checked, surfaces that are within the defined threshold of being perpendicular of the current XY Plane of the coordinate system will be automatically selected. This can save significant time programming.

It is important to note that using Preselect may or may not select the necessary surfaces. It's important to determine that walls that apply to the desired area to machine be selected. Also, Preselect requires Part surfaces to be already selected. If no Part surfaces are selected a message will appear indicating no Part surfaces have been selected.

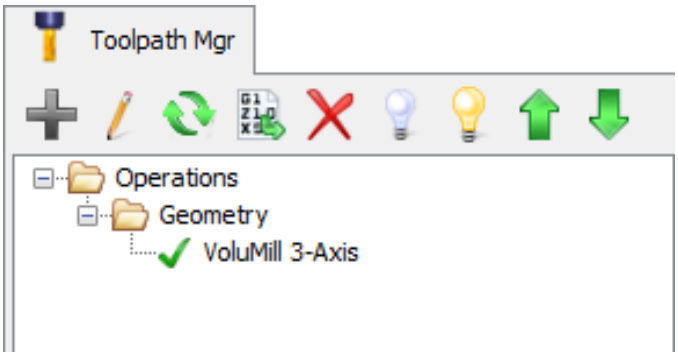
Floor Surfaces - Multi-Axis Geometry

This is used to select surfaces that define the floor of the part to machine. If the floors are parallel to the current XY plane then it's not necessary to select any surfaces. These are used for VoluMill 5-Axis toolpaths only. The number of surfaces have been selected are displayed as a reference.

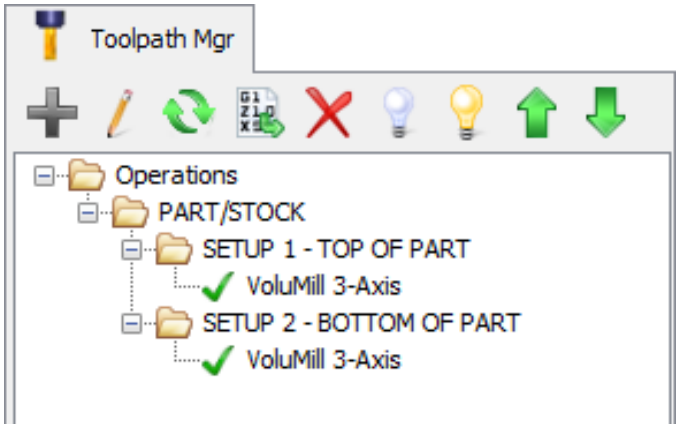
Parent/Child Structure

The ability to define geometry is very flexible and can be used to structure geometry to best fit the complexity of the job.

One example would be to define all the geometry at once. The geometry definition would include the part, stock, and check bodies in addition to any necessary boundaries. Using this structure the Toolpath Manager would appear as follows:

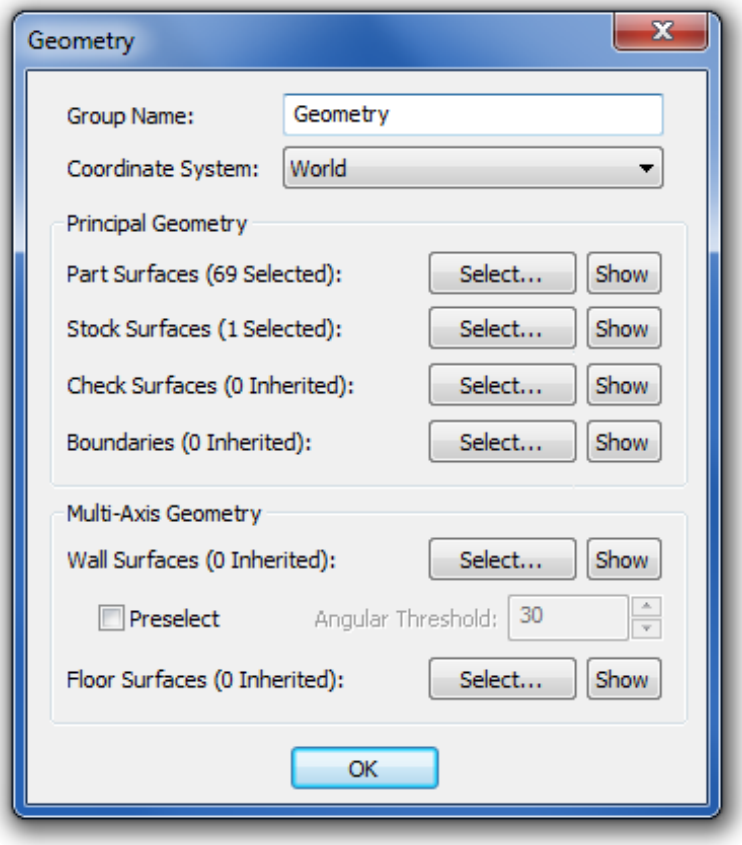


Another example would be to define the geometry separately. The geometry definition inherits the previous geometry definitions and this is indicated in the Geometry dialog. Using this structure the Toolpath Manager would appear as follows:



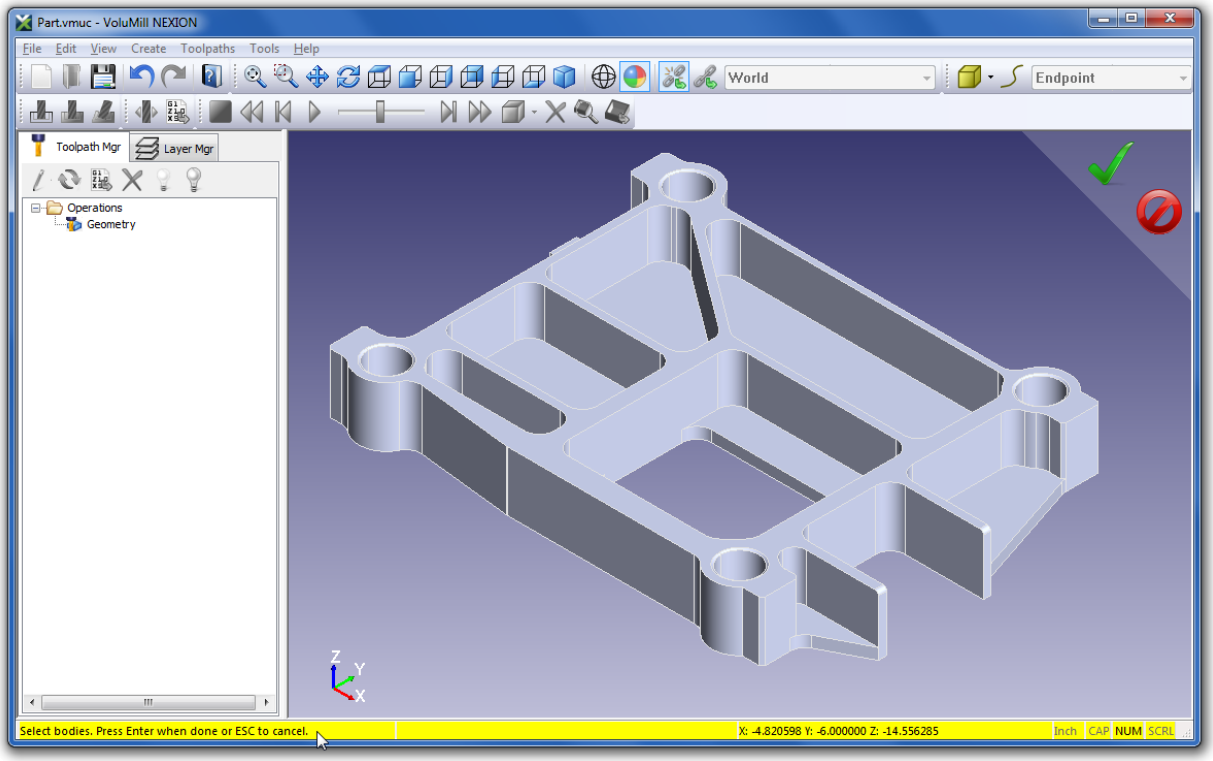
Selecting Geometry

When defining geometry to machine, there are different methods depending on the geometry type.



Surfaces

When the Select.. button is pushed the dialog disappears and the desired surfaces may be selected.



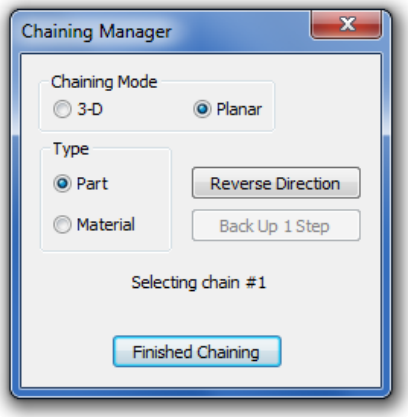
The status bar turns yellow to indicate there is a prompt to select the surfaces/bodies or hit the Esc key to exit.

The selection filter may be used to select the entire body or individual faces. When selecting surfaces, it will default to body.

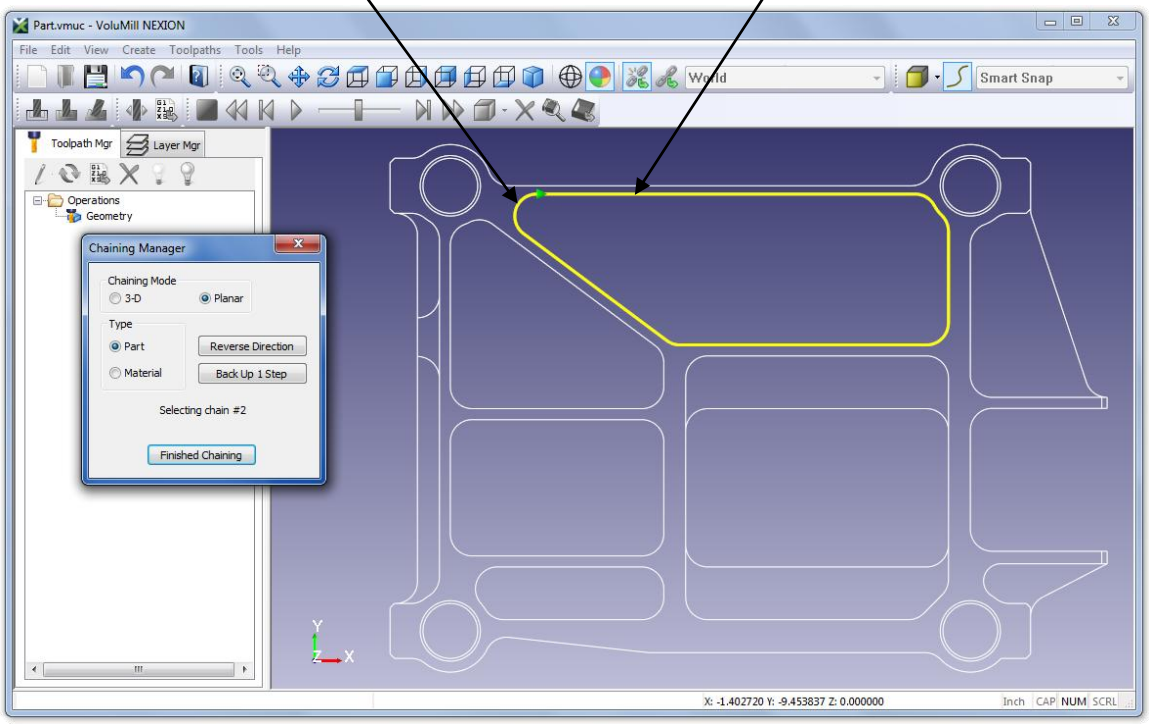
After the desired surfaces are selected, select the  icon to accept or the  icon to reject the current selections.

Boundaries

When the Select.. button is pushed the dialog disappears and the desired surfaces may be selected.



To chain a series of elements, select an element near the desired starting point, then select near the end of an element near the ending point.



To chain all connected elements in one step, double-click near the desired starting point.

3DThis option allows chaining in any direction.

Planar

This option constrains the chaining within a plane.

Part/Material

These options define the type of the next element in the chain.

Reverse Direction

The first element of a chain will display an arrow indicating the direction of the chain. Push this button to reverse the direction.

The direction of the chain has no effect on the toolpath.

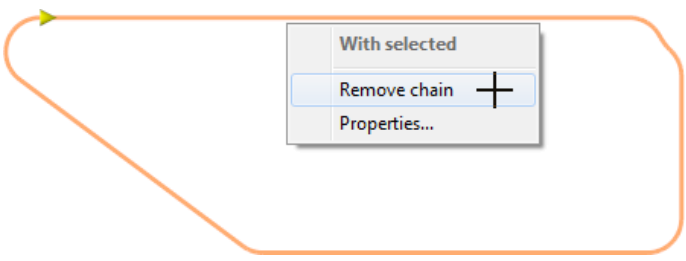
Back Up 1 Step

Pushing this button will step backwards along the chain one chain at a time.

Finished Chaining

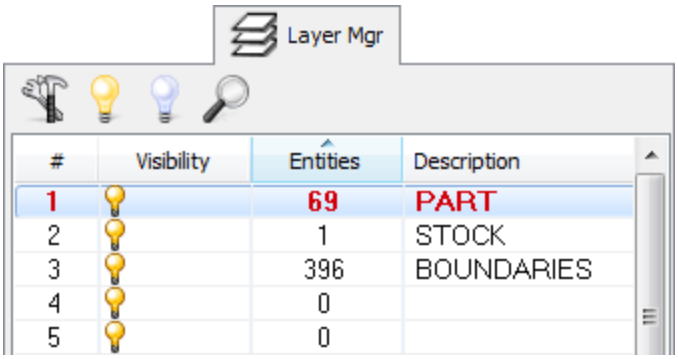
After all the profiles have been chained, push this button to proceed.

To remove a chain, right-click on the chain and select Remove chain.



Layer Manager

Layers are used to manage the display of individual or combinations of geometry. Selecting the Layer Mgr tab displays the list of layers.



Current Layer 

This icon sets the currently highlighted layer to be the working layer. Any geometry that is merged into the current file or rectangular/cylindrical bodies will be placed into this layer. Layer numbers within DXF files are supported.

To help identify which layer is the working layer, the text is red.

All Visible 

This icon makes all the layers visible. Individual layers can then be made invisible by clicking the light bulb in the individual column for the desired layer.

All Invisible 

This icon makes all the layers invisible. Individual layers can then be made visible by clicking the light bulb in the individual column for the desired layer.

Empty 

This icon toggles the display of layers that only have elements.

 # Column

This column lists the layer numbers. Clicking on the column header will sort the layers by their number.

Visibility Column

This column displays icons that can be clicked on to change the visibility. Clicking on the column header will sort the layers by their visibility.

Entities Column

This column displays the number of elements on each layer. Clicking on the column header will sort the layers by the number of entities.

Description Column

This column displays the description of each layer. To add/edit the description, click in the description field or hit F2. Clicking on the column header will sort the layers by the description.

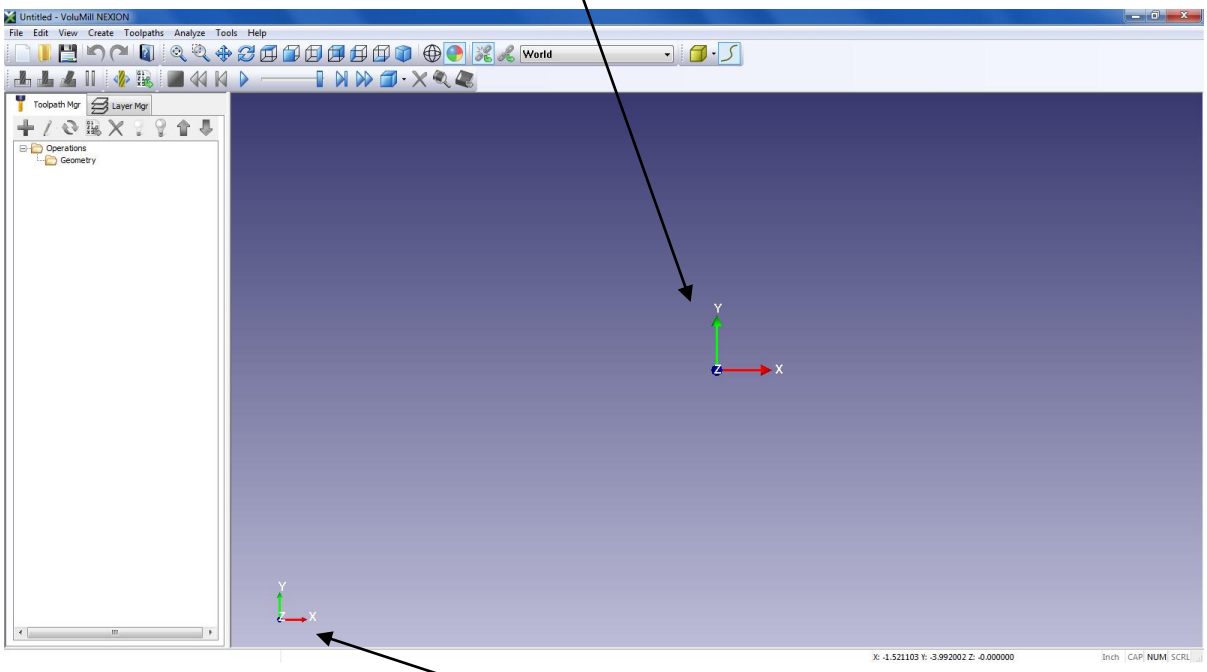
Coordinate Systems

Coordinate systems define the X, Y, Z axes used when creating toolpaths. There are two different types of coordinate systems in VoluMill NEXION, the World CS (coordinate system) or Current CS.

The World CS is the default coordinate system that all others are relative to. It is represented by the smaller X, Y, Z axes in the lower left corner of the workspace.

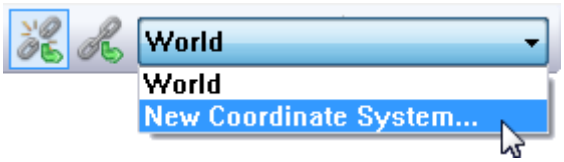
The Current CS is the coordinate system that is used when creating toolpaths. It is represented by the larger X, Y, Z axes in the center of the workspace.

Current Coordinate System Axes

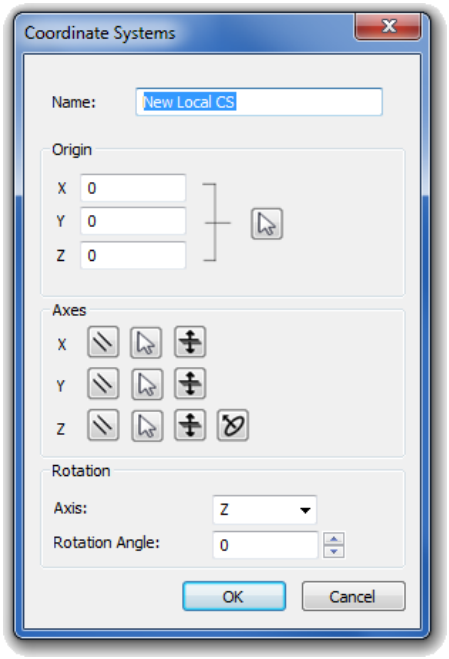


World Coordinate System Axes

To define a new CS, select New Coordinate System… from the Coordinate System list:



The Coordinate Systems dialog will appear:



Name

Enter a name for the new coordinate system.

Origin

This is the X, Y, Z origin of the new coordinate system. When post processing, all X, Y, Z values will be relative to this new origin. The values may be entered in the fields for each axis or the may be used to select a position on the part itself.

Axes

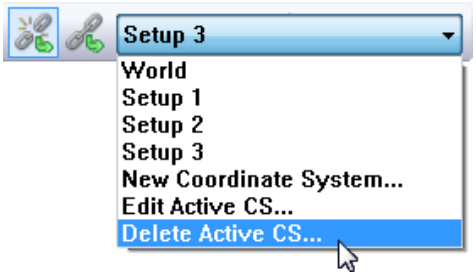
When creating a new CS, there are options to align a specific axis parallel to a line, through a point, and normal to a circle. There is also an option to invert an axis.

- Parallel – This aligns the axis to be parallel to a selected line
- Though Point – This aligns the axis to intersect a selected point
- Invert – This inverts, or flips, the axis
- Normal to Circle – This makes the Z axis normal to a circle

Plane Rotation

Select an axis to rotate around and enter a rotation angle.

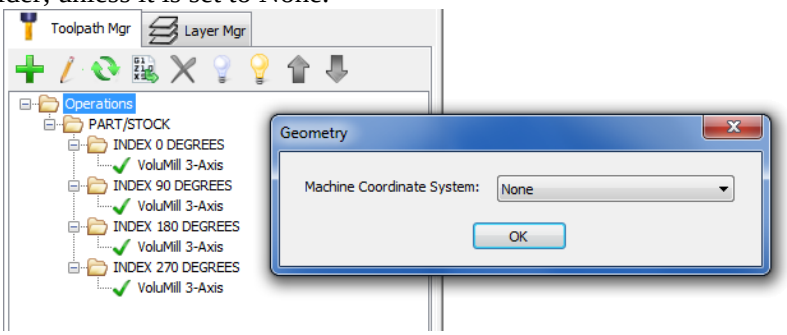
To edit or delete an existing coordinate system, select Edit Active CS… or Delete Active CS…. It's important to note that you must set the active CS before editing or deleting it.



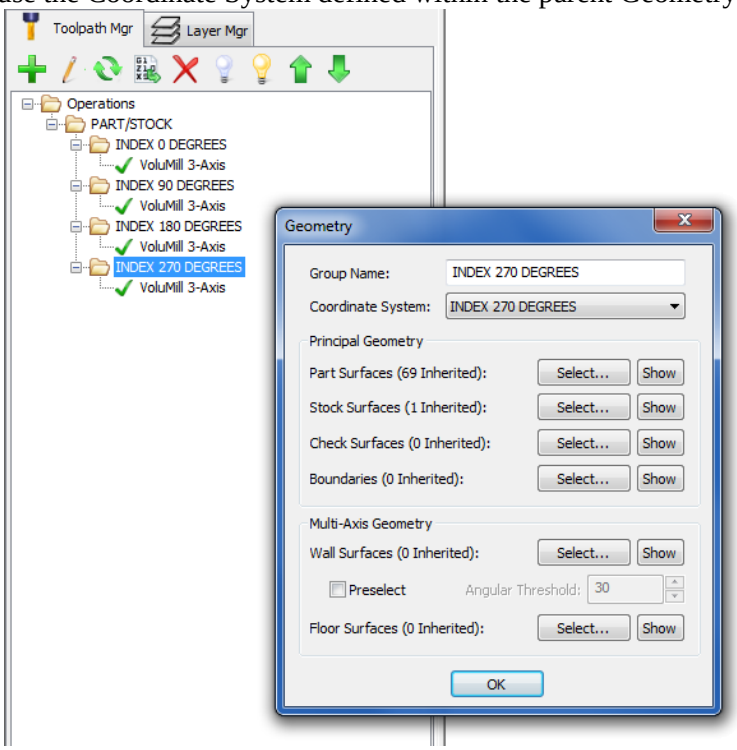
Any toolpaths that use an edited or deleted CS will be represented by a symbol and will require regeneration with a new CS.

Machine Coordinate Systems

When post processing, the output will be based using Machine Coordinate System defined within the base Operations folder, unless it is set to None.



Otherwise, is will use the Coordinate System defined within the parent Geometry folder.



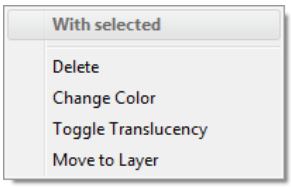
The benefits in using the Machine Coordinate System in the Operations folder are as follows:

- 4-Axis indexing support. Toolpaths using their own Coordinate System may created to machine on each face of a tombstone and then post processed through the Machine Coordinate System. NOTE: This is supported using the APT-CL based post processors only.
- Easier editing. Depending on how the Geometry groups are assembled it is easier to change the Coordinate System in one place.

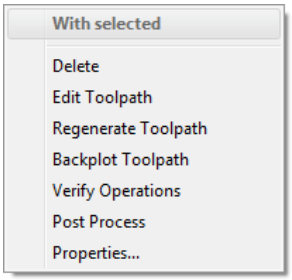
Workspace

Right-click

The right-click lists choices that are only available to the highlighted elements or toolpath.



right-click on geometry



right-click on toolpath

Delete

Deletes the highlighted toolpath and/or geometry.

Change Color

Displays a dialog to select a new color for the selected geometry.

Toggle Translucency

Change the translucency for the selected geometry.

Move to Layer

Displays the Layer Manager to select the desired layer to move the selected geometry to.

Edit Toolpath

This choice opens the dialog for the tool and machining parameters for the highlighted toolpath so they may be changed.

Regenerate

This choice reprocesses the selected toolpath.

Backplot Toolpath

This choice displays the backplot dialog for the selected toolpath.

Verify

This choice plays the verification for the selected toolpath.

Post Process

This choice displays the post-processing dialog for the highlighted toolpath.

Properties…

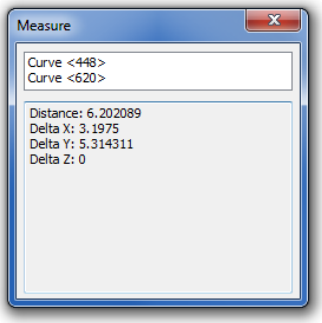
This choice displays the name of the coordinate system, object type, and number of records for the selected toolpath.

Export as STL…

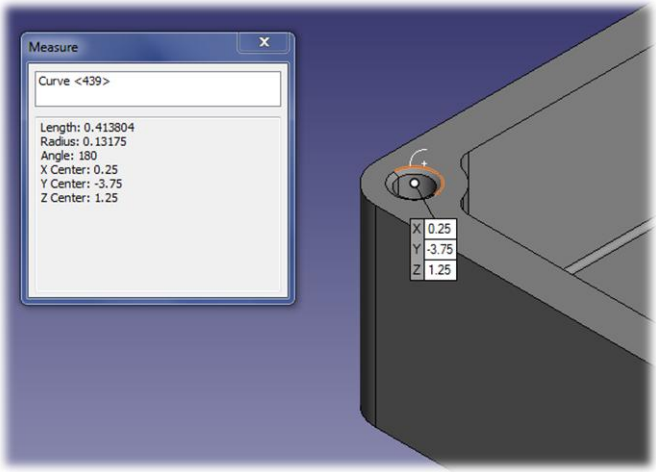
This choice opens the dialog to save the selected stock geometry as an STL file.

Analyze

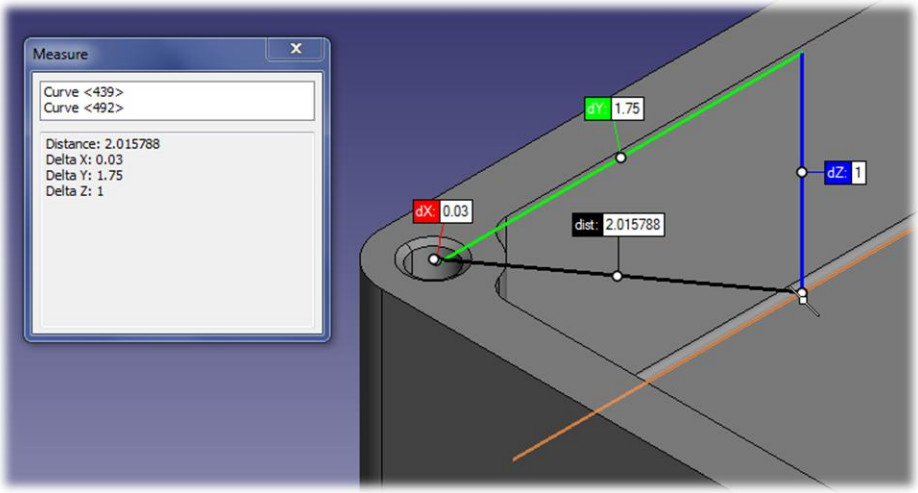
The Measure choice displays the following dialog enabling different types of measurements to be made.



Measuring can be used to analyze a number of different locations and distances. To measure a single coordinate simply select the desired location using the appropriate Snap Mode.



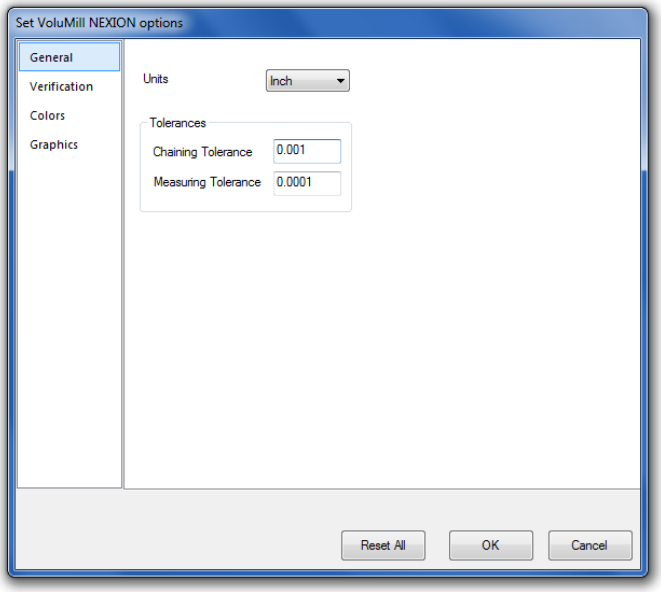
To measure the distance between two locations select a second location using the appropriate Snap Mode. The direct and delta distances are provided.



To reset the measurement click in the open workspace or select a third location.

Options

The Options dialog defines a number of options that are used when NEXION is started.



General

This page defines basic unit and tolerance options.

Units

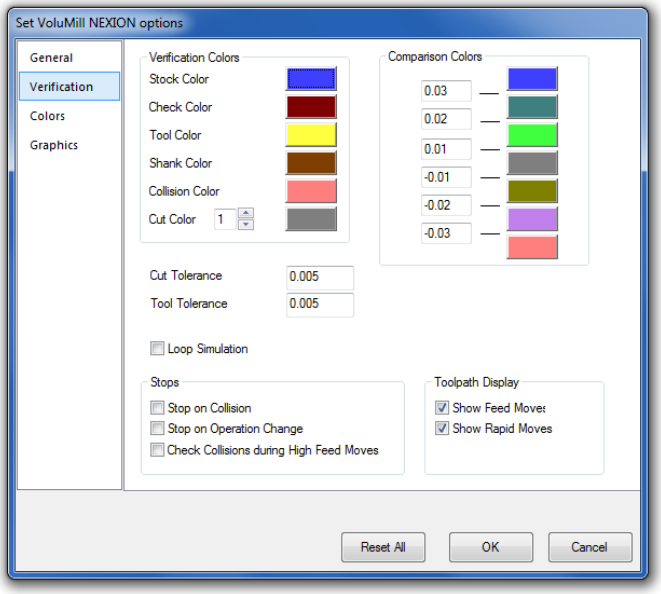
This option defines the startup unit system.

Chaining Tolerance

This value defines the tolerance used when chaining geometry.

Measuring Tolerance

This value defines the tolerance used when measuring geometry.



Verification

This page defines different verification options.

Verification Colors

These options define various colors used during verification.

Comparison Colors

These options define various colors used when the Compare function is used.

Cut/Tool Tolerances

These values define the tolerances used for the tool and part during verification. They are set to a low tolerance to allow adequate performance for large parts/toolpaths.

Loop Simulation

This option causes the verification to repeat the simulation.

Stop on Collision

This option causes the verification to pause when a collision is detected during a rapid move or when the non-cutting part of a tool touches material.

Stop on Operation

This option causes the verification to pause at the end of an operation.

Check Collisions during High Feed Moves

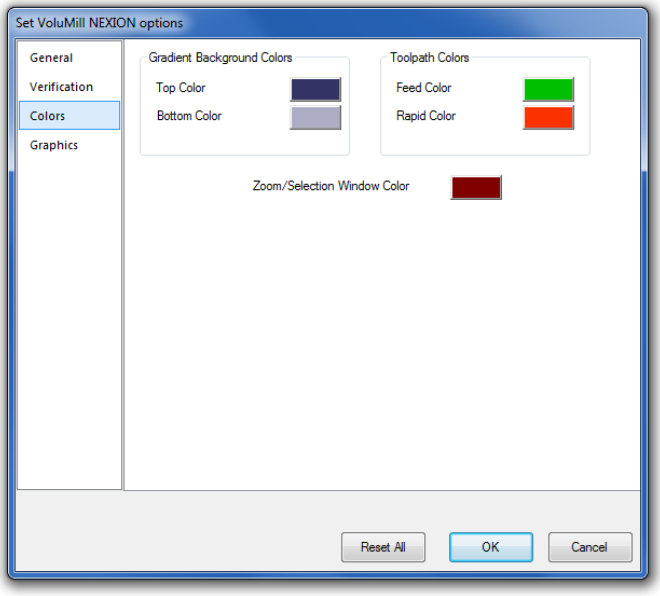
This option causes the verification to pause when a collision is detected during a high-feed move.

Show Feed Moves

This option hides the wireframe feed motion during animation.

Show Rapid Moves

This option hides the wireframe rapid motion during animation.



Colors

This page defines different color options.

Gradient Background Colors

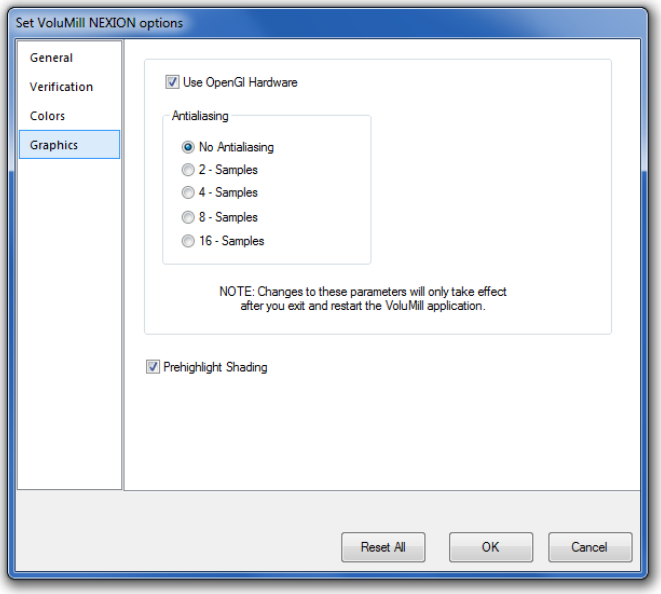
These options define the colors used for the workspace.

Toolpath Colors

These options define the colors used for the toolpath feed and rapid motion.

Zoom/Selection Window Color

This option defines the colors used when using the View/Zoom and Selection window.



Graphics

This page defines different graphics options.

Use OpenGL Hardware

If checked, this option uses the capabilities of your video card to maximize the graphics performance and quality in VoluMill NEXION. If you are having problems such as slow performance or unexplained graphics artifacts, try turning this option off. By default, hardware acceleration is on.

Antialiasing

These options only have an effect if "Use OpenGl Hardware" is checked. Curves and curved surfaces will generally look smoother with some level of antialiasing. Higher levels of antialiasing will appear smoother, but will slow down rendering and selection. If rendering or selecting appears slow, try using a lower setting or "No Antialiasing".

Prehighlight Shading

When hovering over surfaces both the surface and outline are highlighted. If unchecked, just the outline is highlighted.

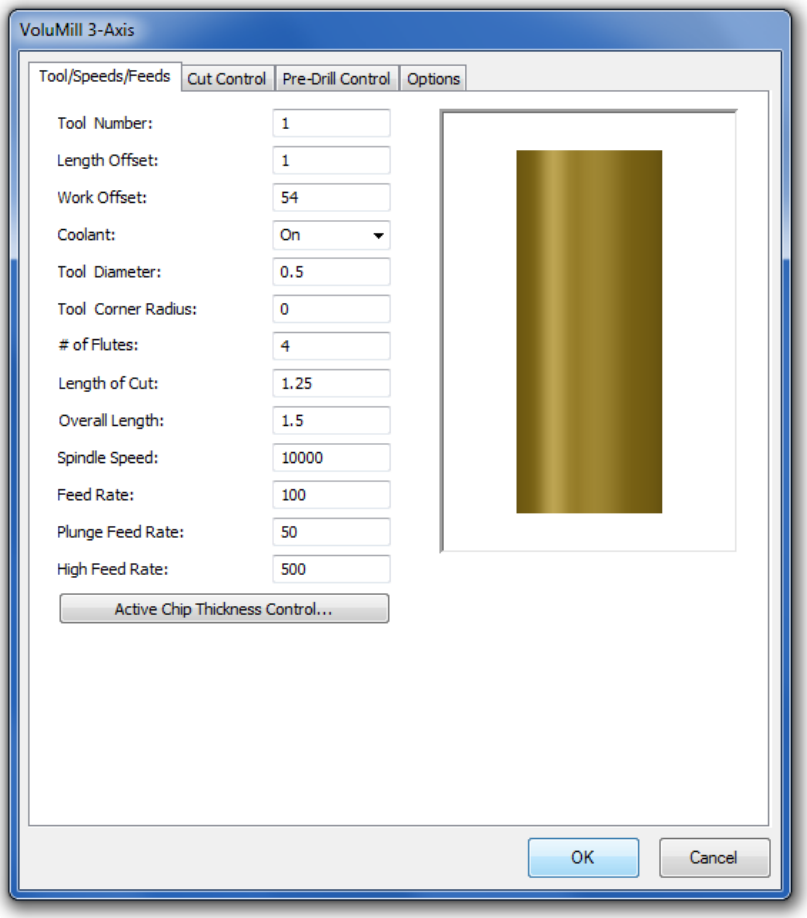
Chapter 3

MACHINING PARAMETERS

This chapter describes the machining parameters in detail and how they apply in creating VoluMill toolpaths using wireframe geometry.

Tool/Speeds/Feeds Tab

The Tool/Speeds/Feeds tab is identical for all the VoluMill toolpath types.



Tool Number

This value corresponds to the tool's position in the machine's carrousel, and most commonly the "T" number in the post-processed output.

Length Offset

This value corresponds to the tool length register on the control for the current tool.

Work Offset

This value corresponds to the work or fixture offset register on the control for the current tool.

Tool Diameter

The diameter of the tool.

Tool Corner Radius

The radius at the bottom corner of the tool for bull- and ball-mills.

# of Flutes

The number of flutes or cutting edges of the tool.

Length of Cut

This defines the length of the cutting edge along the tool. This is only used for verification and does not affect the toolpath.

Overall Length

This defines the overall length of the tool. This is only used for verification and does not affect the toolpath.

Spindle Speed

Determines the rotation speed of the spindle in revolutions per minute (RPM).

Feed Rate

This is the feed rate at which the tool will travel while engaged in the material. VoluMill automatically adjusts this feed rate downward in concave arcs in order to maintain a constant rate of material removal.

Because of this, a posted VoluMill toolpath will contain many feed rate changes. This is normal.

The concept of VoluMill is to generate toolpaths that never exceed a given rate of material removal when machining. It does this by dynamically adjusting the feed rates and depth of cuts. VoluMill will adjust the feed rate and/or depth of cut to maintain the optimal cutting conditions regardless of the shape of the part.

Plunge Feed Rate

This is the feed rate at which the tool will travel while plunging in a helical or ramping motion.

High Feed Rate

This is the feed rate at which repositioning moves within the toolpath occur, other than those that take place at Rapid above the part. Since VoluMill drives the tool in a manner that controls the material removal rate, it is typically necessary to reposition the tool from the end of one cut to the beginning of the next by moving it through an already-machined area. Setting this value higher than the programmed cutting feed rate helps minimize cycle time.

It is recommended that this value be set to the fastest non-Rapid feed rate at which the machine can interpolate linear and circular moves.

Active Chip Thickness Control

The Active Chip Thickness Control function in VoluMill is a calculator designed specifically to be used with VoluMill toolpaths. Since VoluMill toolpaths contain a far higher percentage of constant-width cuts than any other rough milling toolpath technologies, you can actively manipulate the maximum thickness of the chips to increase rough milling productivity even further. Cycle time reductions in the range of 35% beyond what you are currently achieving with VoluMill are now just a few clicks away.

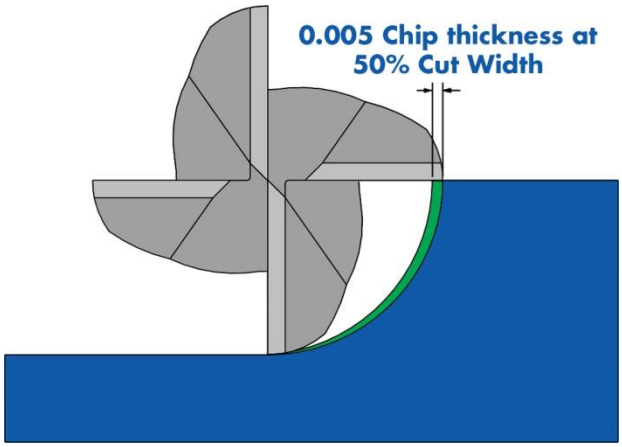
The following is an overview of this functionality, including examples and recommendations for its use.

Your cutting tool manufacturer recommends a Feed per Tooth (IPT) value for a specific cutting tool in a specific material. Though this value is often called a “chip load” or “chip load per tooth,” it really is not an indicator of the load on the tool.

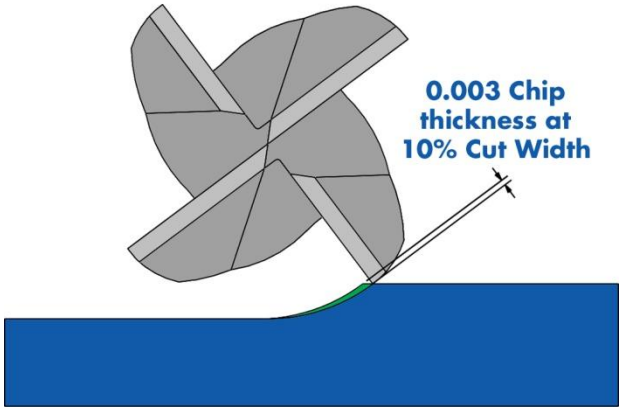
When climb milling, a chip is thicker towards its beginning than at its end. It has a maximum thickness at a single point, and thins down from there. This maximum chip thickness is a much

truer measure of the load on the tool than is the feed-per-tooth value. The thickness of a chip is a function of its length; the longer the chip, the thicker the chip. And the length of a chip is a function of the width of cut.

The Maximum Chip Thickness (MCT) is equal to the feed per tooth only if the cut width is greater than or equal to 50% of tool diameter:



If the cut width is less than 50% of tool diameter, the IPT is reduced. If the cut width is 10%, for example, the MCT is 60% of the feed per tooth:



Active Chip Thickness Control in VoluMill includes chip thickness in the parameter calculations. When the Active Chip Thickness Control dialog is displayed, the values for RPM, IPM, and Cut Width are inherited from the main VoluMill dialog, and values for SFM and IPT are shown as well. Note that the Chip Thickness is also displayed. This is the maximum thickness of the chips that will result, based on the current spindle speed, feed rate, and cut width, along with the tool diameter and number of flutes. This value is grayed out because Chip Thickness is the default option from the *Calculate the:* dropdown list at the top of the dialog.

From here any one of the parameters can be calculated based on changes to any of the other parameters. For example, a ½-inch diameter, 5-flute cutter, at 400 SFM (3,056 RPM), 100 IPM (.0065 IPT), at a 7% Cut Width (.035), yields a Chip Thickness of 0.0033. These are parameters commonly used to rough mill 6Al4V titanium with VoluMill. With a 1.000 axial depth of cut, these parameters produce a material removal rate of 3.5 cubic inches per minute while in the cut.

A good adjustment to make is to the Cut Width parameter. In this example, it is already known that the tool can clear the 0.0033 thick chip, so the following manipulation is available: with the Chip Thickness set to 0.0033, choose Feed from the *Calculate the:* dropdown list, and double the Cut Width value from .035 (7% of tool diameter) to .070 (14% of tool diameter). Pushing the *Calculate* button shows that the IPM must be reduced to 73.54 (0.0048 IPT) in order to maintain the chip thickness at 0.0033 with this larger cut width and longer chip. Yes, this is a feed rate reduction of 26.46%. But the doubling of the cut width effectively reduces the overall toolpath

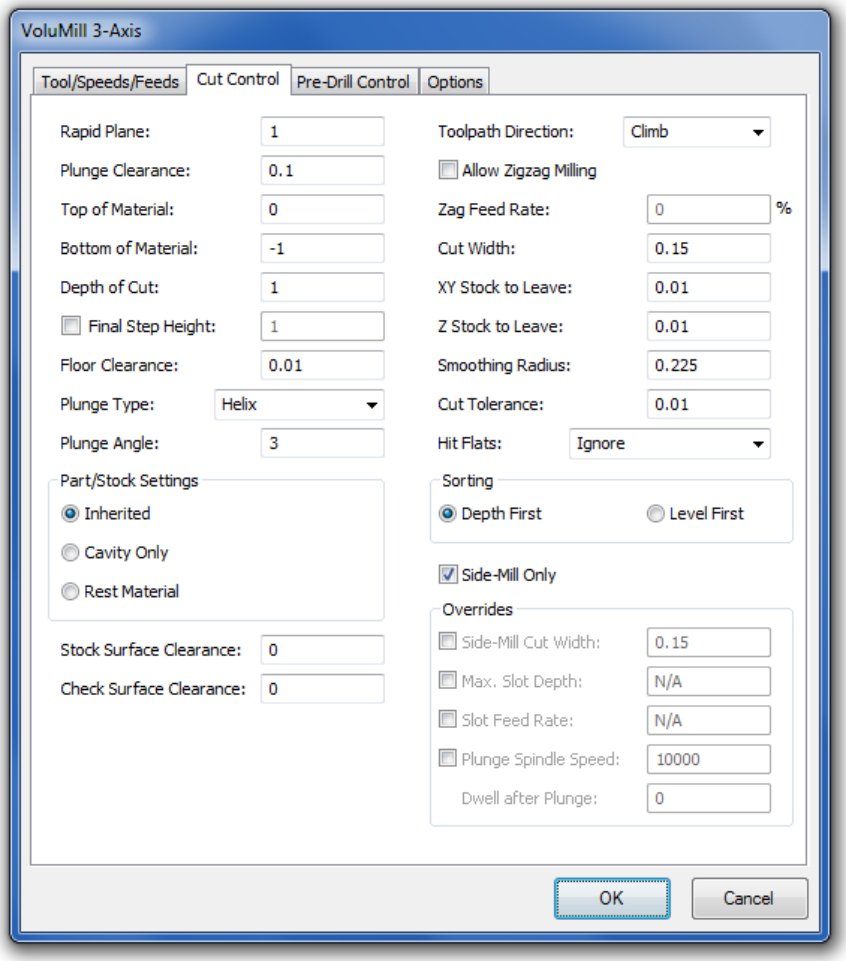
length by 50%, yielding a cycle time reduction of approximately 35%. Note that the calculated MRR increases to 5.15 cubic inches per minute. Pushing the Apply Parameters button will close the ACTC dialog and update the parameters on the VoluMill toolpath dialog.

The method described above is the recommended use of the ACTC function. If you choose to use it for other purposes, please remember that some laws of physics will always remain in effect. For example, you may choose to find out what happens to the Chip Thickness if you modify the spindle speed. From the initial parameters of 3,056 RPM, 100 IPM and resulting Chip Thickness of 0.0033, choosing Chip Thickness from the *Calculate the:* dropdown and increasing the RPM value to 4,000 updates the Chip Thickness to 0.0026. Also note, however, that the IPT value also updates, to 0.005. This is required due to the change in spindle speed. There are no new mathematics involved in the ACTC function, but you now have direct control of the actual thickness of the chips that your rough milling toolpaths will produce. The near constant, and always bounded, width of cut that VoluMill produces makes this possible.

As always, use your own machining experiences and knowledge of your machining environment as a guide. It may be tempting, for instance, to increase the Cut Width even more to further reduce cycle times, and this very well may be possible. But always remember that there will be physical limits to what your machining hardware can handle.

Cut Control Tab

Although the Cut Control tab shares many parameters for all the VoluMill toolpath types there are some minor differences which are explained below.



Rapid Plane

Sets the height at which the tool moves to and from the part.

Plunge Clearance

Sets the distance above the next feed move at which the tool moves to.

Top of Material (3-Axis/5-Axis Airframe Only)

Sets the starting height of the material in the Z axis. To define the value from a Z-axis coordinate of the part, type "z" in the field and the dialog will disappear allowing a position to be selected.

Depth (2-Axis Only)

Sets the overall depth of the area to be machined. To define the value from a Z-axis coordinate of the part, type "z" in the field and the dialog will disappear allowing a position to be selected.

Bottom of Material (3-Axis/5-Axis Airframe Only)

Sets the depth of the toolpath in the Z axis. To define the value from a Z-axis coordinate of the part, type "z" in the field and the dialog will disappear allowing a position to be selected.

Depth of Cut

Defines the maximum depth of cut.

For 2-Axis toolpaths, if the value does not divide equally into the total depth, VoluMill subdivides the number of cuts to create equal depths of cut.

For 3-Axis toolpaths, if the value does not divide equally into the total depth, VoluMill will not subdivide the number of cuts to create equal depths of cut.

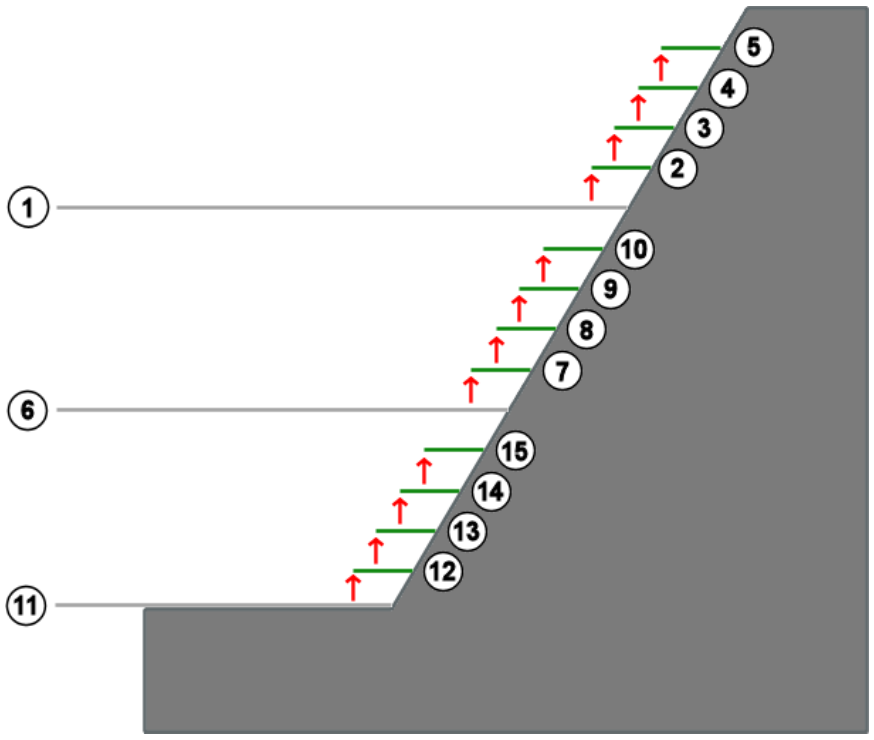
Floor Clearance

The value entered here establishes the Z-component of a helical move that is used when entering or exiting a cut. Only non-negative values are allowed. If a positive value is entered, repositioning moves between cuts will take place above the already-machined floor. If zero is entered, the tool will drag across the already-machined floor during these moves. In this case, set the High Feed Rate parameter to be no greater than the cutting feed rate to help ensure more consistent tool marks on the floor.

Final step height (3-Axis Only)

Use this option to control the height of the steps that will remain. Instead of making a shallow depth of cut across the entire part to leave smaller steps, VoluMill can efficiently first machine larger steps and then automatically re-machine to leave smaller steps. By doing this, the tool can remove the bulk of material most efficiently and still leave smaller steps for a semi-finish or finish toolpath.

The tool begins by machining the part using the Depth of Cut value. After the entire part is machined at each Depth of Cut, the tool then reduces the step using the Final Step Height value.

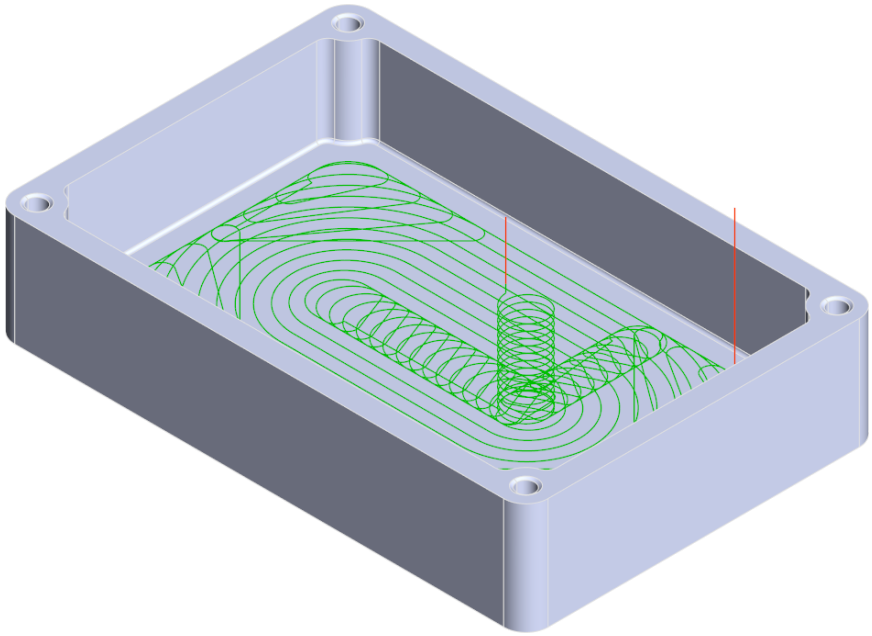


Plunge Type

This parameter defines the type of entry motion VoluMill uses to machine to the desired depth of cut.

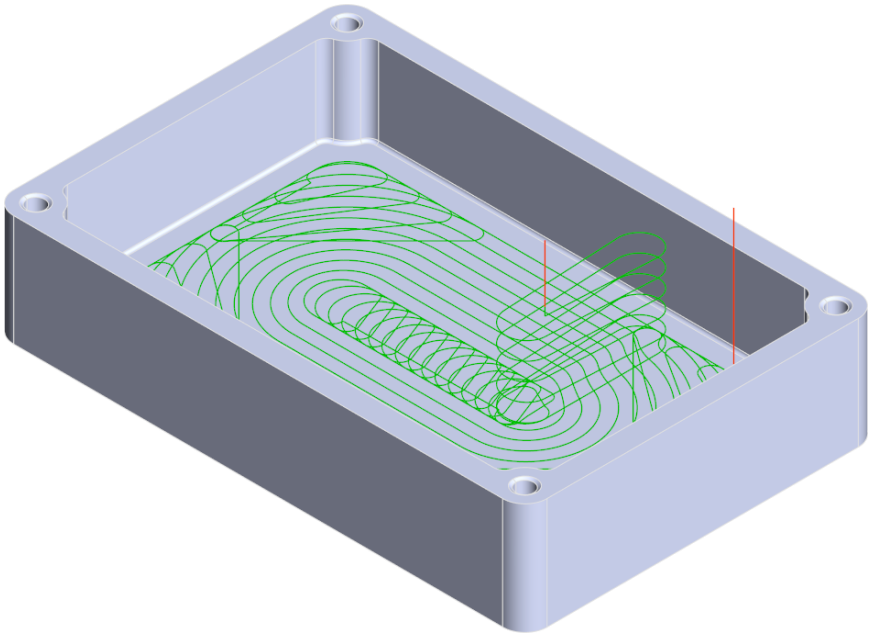
Helix

This Plunge type uses a helix to machine to the desired depth of cut. This is the default Plunge type and is recommended for harder materials.



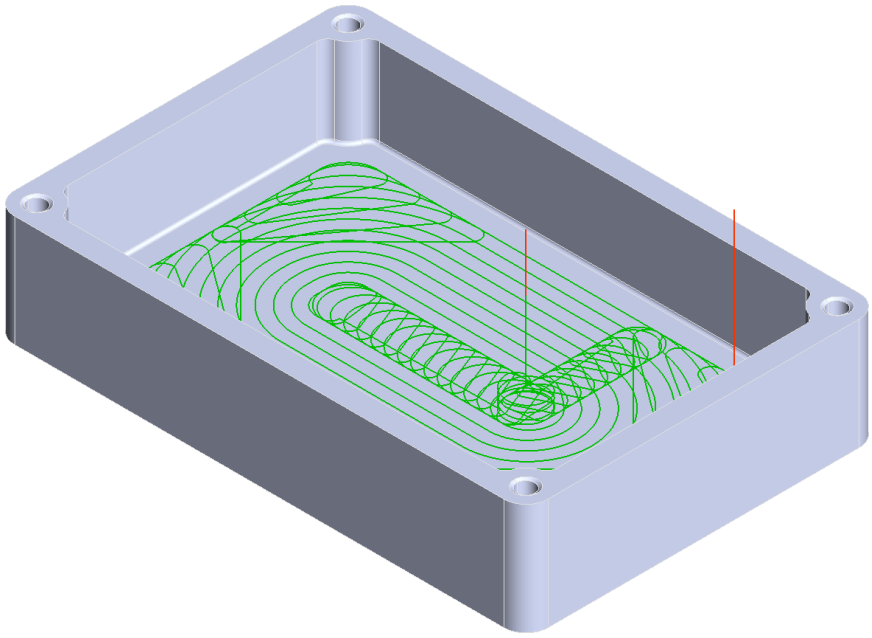
Ramp

This Plunge type uses a special ramping motion to machine to the desired depth of cut. VoluMill calculates the optimal position and shape of the ramp to create a transition area. This transition area is then used to connect from the end of one cut to the next while disengaged from the material at the High Feed Rate. This Plunge type is recommended for softer materials.



Pre-Drilled Hole(2-/3-Axis Only)

This Plunge type uses a combination of a straight plunge and helix where holes are expected to be drilled. This functionality is described in further detail in the Plunge Control tab.



Plunge Angle

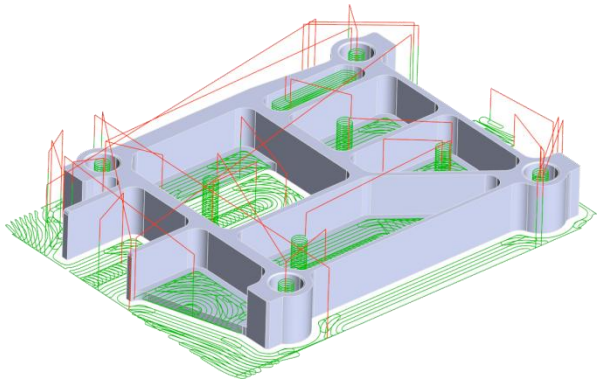
This parameter establishes the rate of descent, in degrees, at which the tool enters the material from the top, as is required when machining completely enclosed areas (pockets). VoluMill uses the entered value as a not-to-exceed value, meaning that the actual ramp angle may be adjusted downward from the entered value as needed to fill the ramping area. VoluMill automatically calculates the location, length, and orientation of the ramp based on the shape of the selected geometry. The feed rate for the plunge motion is determined by the Plunge Rate parameter.

Part/Stock Settings - Inherited

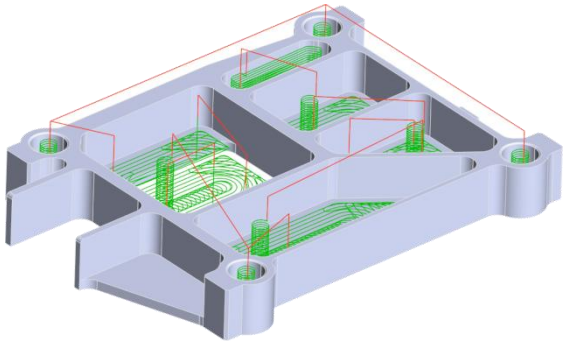
Stock used for generating toolpath is inherited from the Geometry definition.

Part/Stock Setting - Cavity only (3-Axis Only)

This option should be used when machining a cavity from material having a flat top surface. The benefit of using this option is that stock need not be defined separately.



Stock Setting - Inherited



Stock Setting - Cavity Only

If the Cavity machining only option is used on a core or a shape that doesn't have a flat top surface, the toolpath will begin machining at a level where it can create a closed area. This can create a situation where the first depth of cut is too deep. Make sure to use a stock definition appropriate for the part shape to avoid this.

Part/Stock Setting - Rest Material

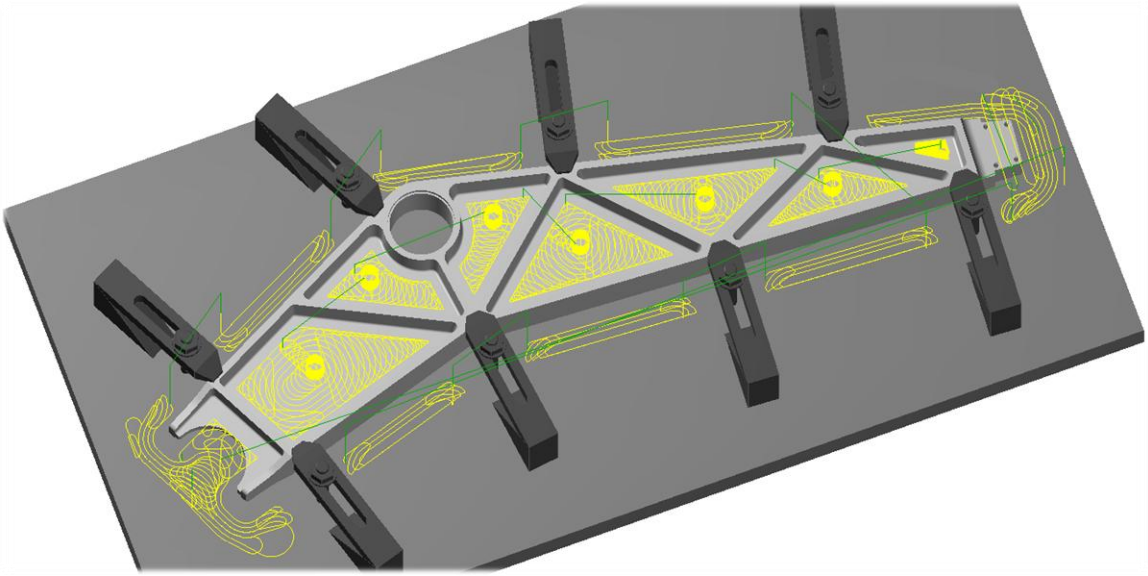
This option creates a toolpath that machines material remaining from previous operations. 2-Axis toolpaths requires the previous toolpath to be selected using the Select... button.

Stock Surf Clearance (3-/5-Axis Only)

This is an addition distance the tool will stay away from any surfaces or solids selected as stock.

Check Surf Clearance (3-/5-Axis Only)

This is the distance the tool will stay away from any surfaces or solids selected as check surfaces.



Toolpath Direction

Climb milling cuts the geometry with the tool rotating opposite the direction of travel along the cutting side of the tool. This type of machining generally produces a smoother surface finish than conventional milling.

Conventional milling cuts the geometry with the tool rotating in the same direction as the direction of travel along the cutting side of the tool.

Allow Zigzag Milling

Zigzag milling alternates between Climb and conventional cutting. In certain applications this can dramatically reduce the cycle time by reducing or eliminating the time spent repositioning for the next cut.

VoluMill takes into account the different feed rates to determine if it is faster to zigzag or reposition at rapid or high-feed.

The last cut is always made using the define Toolpath Direction.

Zag Feed Rate

This is the feed rate used when making the cut that is the opposite of the Toolpath Direction.

Cut Width

Commonly known as, stepover. With VoluMill, it's important to note that any Cut Width value that is less than the diameter of the flat portion of the tool can be used without fear of leaving uncut stands of material behind. For example, you can use up to a 100% Cut Width with a flat end mill.

XY Stock to Leave (2-/5-Axis Airframe Only)

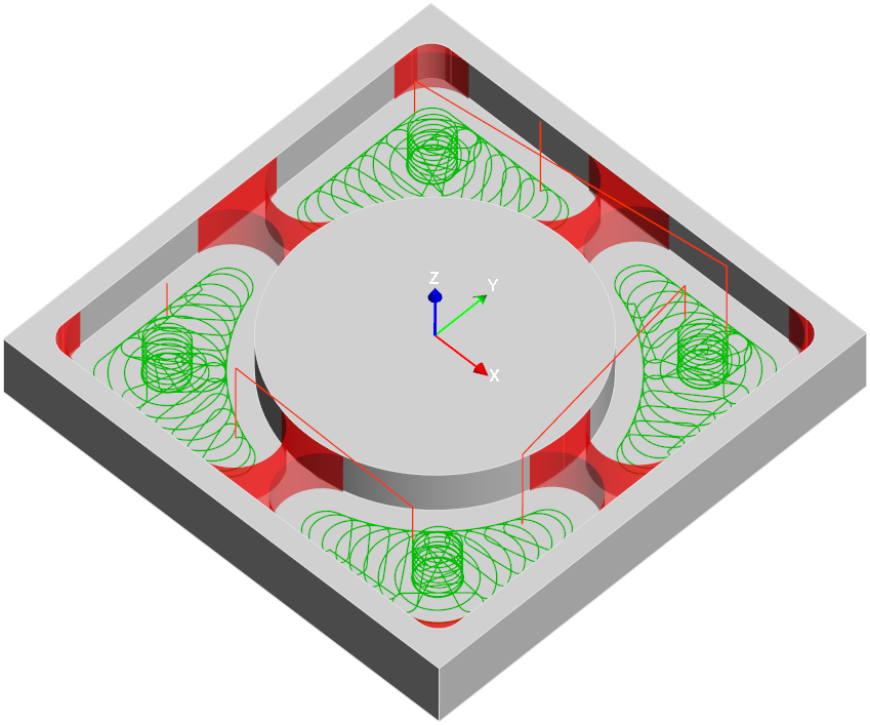
The amount of material that will remain on the vertical surfaces after the machining is complete.

Z Stock to Leave (2-/5-Axis Airframe Only)

The amount of material that will remain on the floor after the machining is complete.

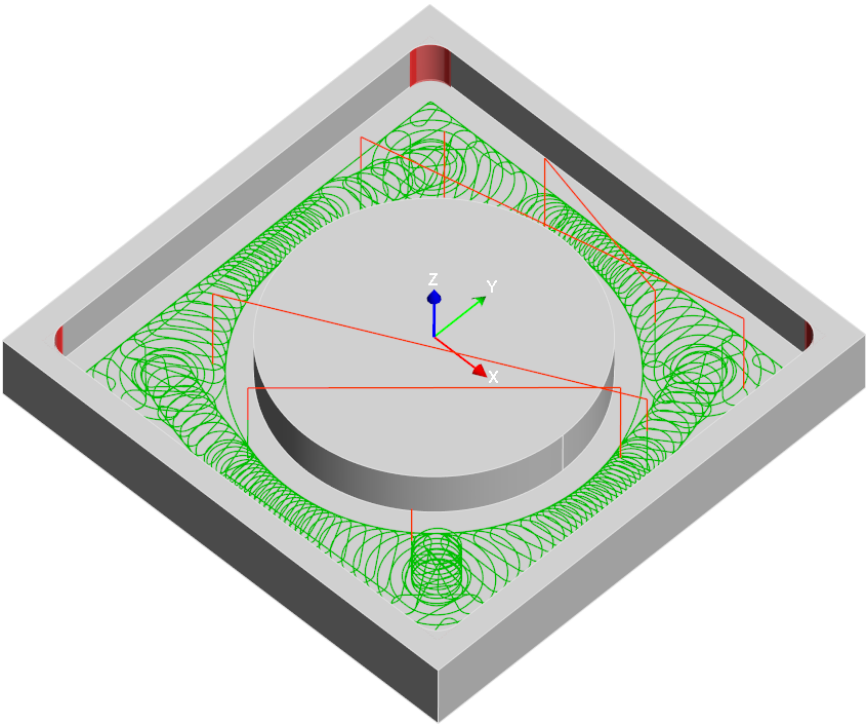
Smoothing Radius

This is the minimum radius the tool will transverse when cutting. To machine into sharp corners or tight areas, VoluMill has to make small moves that may be more effectively accomplished using a smaller tool in a clean-up operation. The optimal, and default, Smoothing Radius is 45% of the tool diameter. At this value VoluMill can reach velocities that can dramatically reduce cycle times. However, it may leave areas uncut.



Using a smaller value may enable the tool to machine more material. This, however, may be less efficient than switching to a smaller tool and using another VoluMill toolpath to machine the remaining material.

The smallest value allowed is 5% of the tool diameter. VoluMill is designed to never make sharp directional changes while in the cut. This means that if your part has a fillet in the corner that is equal to the tool radius, a small amount of material will be left in the corners, even if the Smoothing radius value is set to the minimum allowed.



VoluMill is a roughing technology and it is assumed that a finish pass will follow.

Cut Tolerance

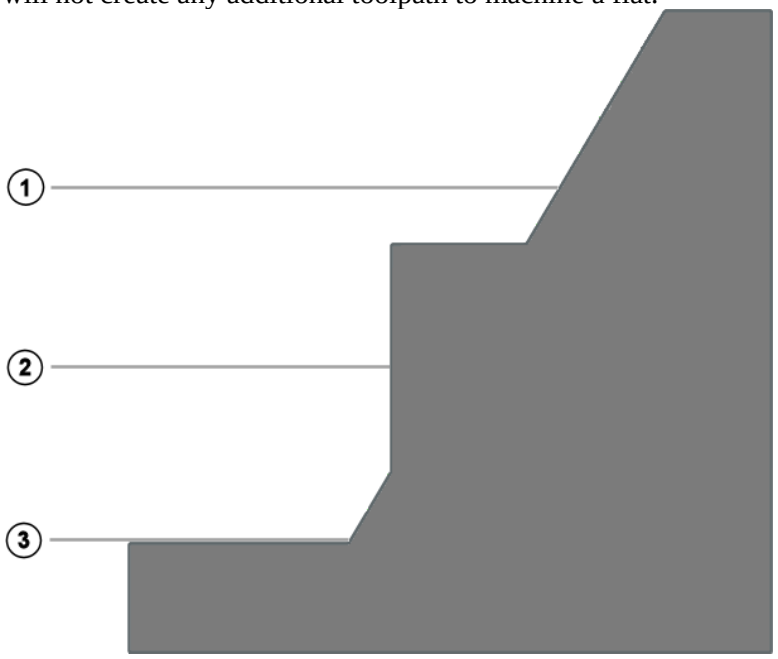
This value defines the chordal tolerance used when machining.

Hit Flats (3-Axis Only)

Use this option to control how flat surfaces, that are not coincident with any Depth of Cuts or Final Step Heights, are machined.

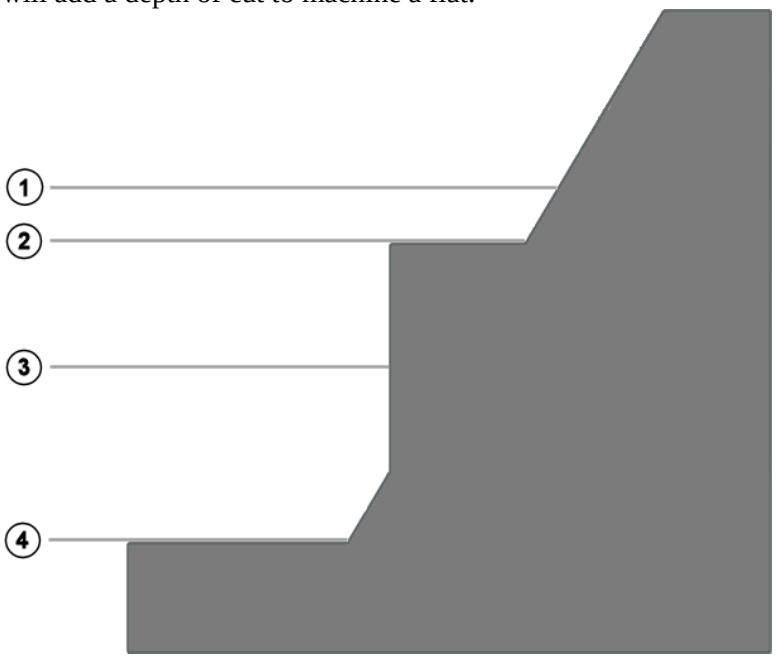
Ignore

This choice will not create any additional toolpath to machine a flat.



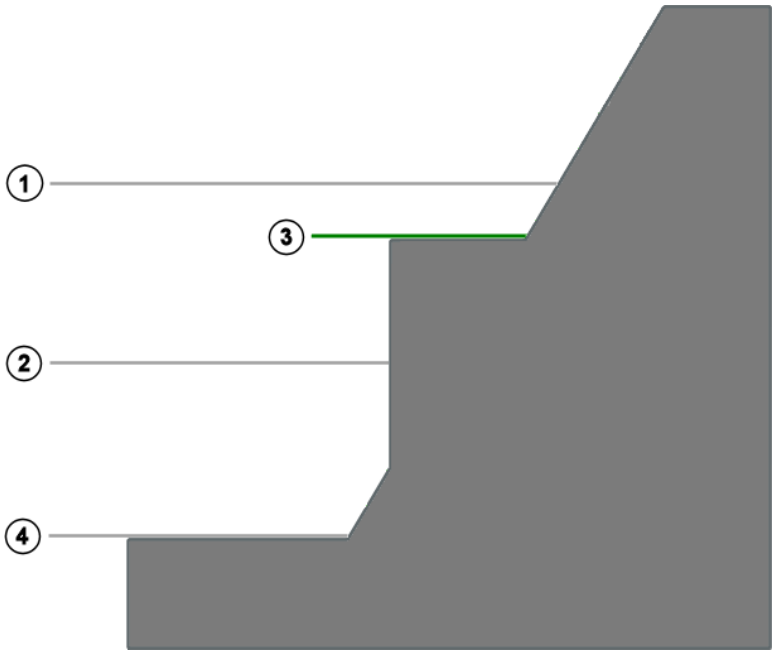
Add Depth of Cut

This choice will add a depth of cut to machine a flat.



Cut after Lower Depth

This choice will create toolpath to machine flat surfaces after they are exposed by a Depth of Cut.



Depth First

Each pocket will be machined to the total depth before the next one.

Level First

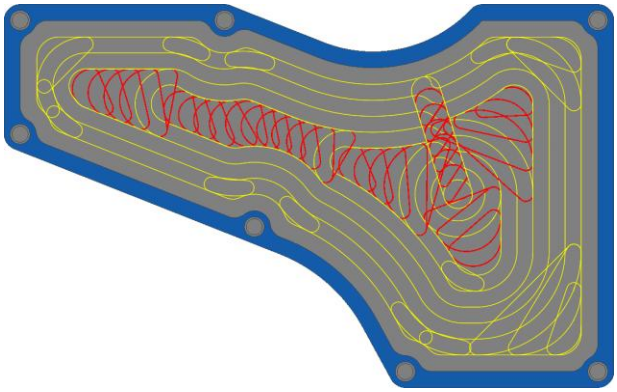
Each pocket will be machined to the Depth of Cut before the next one. This is helpful when machining pockets that have thin walls.

Side-Mill Only

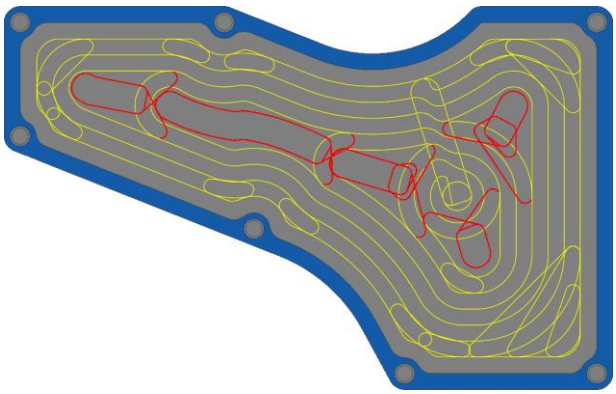
VoluMill achieves its superior cutting performance by striving to maintain a constant rate of material removal throughout the toolpath, regardless of the shape of the geometry.

Two strategies are considered when milling in confined areas: Side Milling and Slot Milling. If this checkbox is unchecked, VoluMill will automatically choose the strategy that produces the fastest cycle time using the current feed rate and distance traveled. Depending on the shape of the part, both methods may be used.

In some cases VoluMill will use a progressive milling method wherein the tool is never fully engaged in the material; this is referred to as Side Milling.



In some cases VoluMill will fully engage the cutting tool; this is referred to as Slot Milling. During the slotting cuts, VoluMill will reduce the feed rate and/or use multiple depths-of-cut so that the rate of material removal that is in effect during the rest of the toolpath is never exceeded.



To force the system to only use the Side Milling strategy, which is especially beneficial in harder materials, check this checkbox.

Side-Mill Cut Width

When the toolpath Side mills in tight areas, it uses the Cut Width distance value. To use a smaller Cut Width distance when side milling in tight areas, check this checkbox and enter the desired distance. The entered value must be less than or equal to the Cut Width distance value.

Max Slot Depth

This parameter determines the number and depth of any slotting cuts. VoluMill uses the entered value as a not-to-exceed value, meaning that the slotting depth-of-cut may be adjusted downward from the entered value as needed to ensure that each such cut removes the same amount of material. Since these slotting cuts are invariably at a greater effective Cut Width (radial depth-of-cut) than the non-slotting cuts of the toolpath, it is necessary to reduce the axial depth-of-cut, which this parameter controls, and/or reduce the slotting feed rate (see the Slot Feed Rate parameter below) to manage the material removal rate during these cuts.

If this checkbox is not checked, VoluMill will automatically set this parameter, in conjunction with the Slot Feed Rate parameter (see below) to establish a material removal rate for slotting that is less than that for the rest of the toolpath. The value that VoluMill calculates will be shown in gray in the input field.

This override is not available if the Side-mill only checkbox is checked.

Slot Feed Rate

This is the feed rate used for the slotting cuts. Since these slotting cuts are invariably at a greater effective Cut Width (radial depth-of-cut) than the non-slotting cuts of the toolpath, it is necessary to reduce the slotting feed rate, which this parameter controls, and/or reduce the axial depth-of-cut (see the Max Slot Depth parameter above) to manage the material removal rate during these cuts.

If this checkbox is not checked, VoluMill will automatically set this parameter, in conjunction with the Max. slot depth parameter (see above) to establish a material removal rate for slotting that is less than that for the rest of the toolpath. The value that VoluMill calculates will be shown in gray in the input field.

This override is not available if the Side-mill only checkbox is checked.

Plunge Spindle Speed

This is the spindle speed used for the Plunge motion.

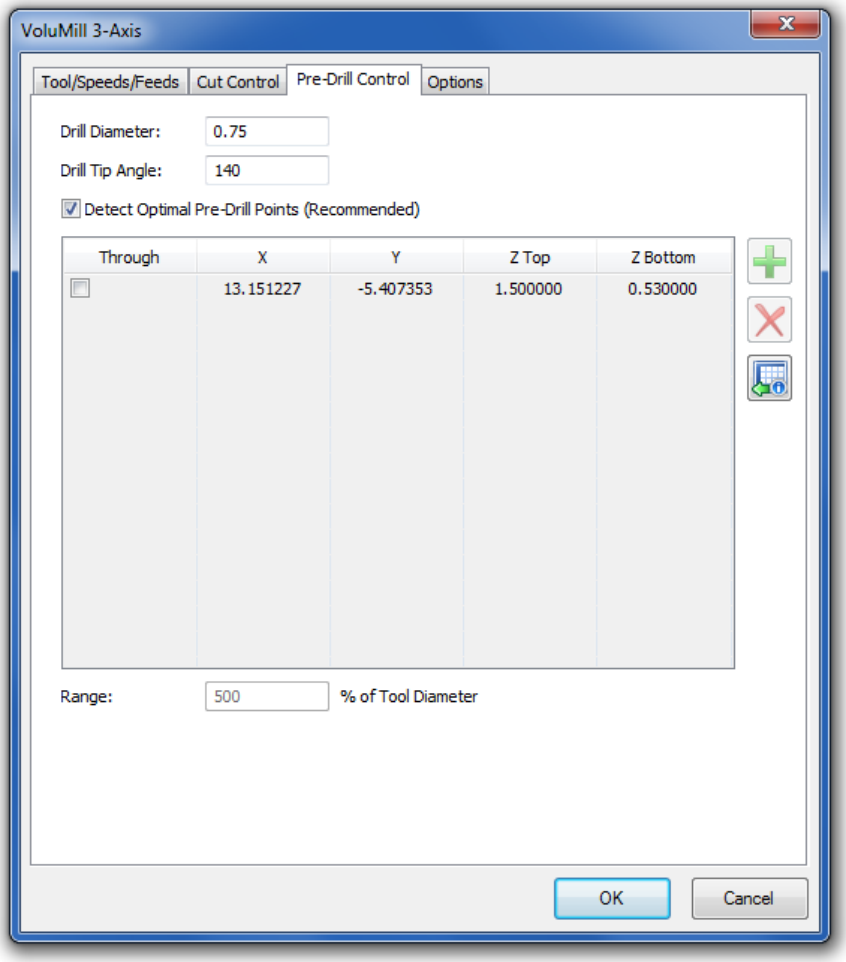
If this checkbox is not checked, VoluMill will use the Spindle speed defined on the toolpath parameters tab. In softer materials this may be acceptable. However, for harder materials it is recommended that the Plunge spindle speed be reduced to sync with the Plunge rate.

Dwell after Plunge

This is a value used to force a dwell after the tool has completed the plunge motion. Some machines will not pause to begin feeding until the programmed spindle speed is reached. For these machines the dwell will give the machine time to do so.

Pre-Drill Control Tab

The Pre-Drill Control tab applies to 2- & 3-Axis toolpaths.



Drill Diameter

The diameter of the drill.

NOTE: If a drill diameter is less than or equal to the tool diameter VoluMill will display an error saying the drill diameter must be larger than the tool diameter.

It is important to note that VoluMill never exceeds a defined rate of material removal. If the drill diameter is too close to the diameter of the tool VoluMill will reduce the feedrates to avoid exceeding the defined volume of material. This can, in some cases, extend the cycle time and reduce the advantage of using a pre-drilled hole. When possible, it is recommended that the drill be at least 25% larger than the tool diameter.

Drill Tip Angle

The included angle of the drill tip. For flat-bottomed drills enter 180.

When plunging into the hole the tool will stop above the shoulder of the drill and helix to depth.

Detect Optimal Pre-Drill Points

When checked, VoluMill will define pre-drill locations that are at the native start points for the toolpath.

When unchecked, VoluMill will use start points that are manually selected within the defined Range. If a point does not lie within the defined Range VoluMill will helix at the defined helix angle.

To add a point select the  button. To delete a point, select a row and then select the  button

Grid

The grid displays the X/Y location as well as the starting and ending Z depths. These values can be used in the primary CAM system to generate the drilling toolpaths. Each value can be highlighted then selected using Ctrl-C.

The column labeled Through contains checkboxes that can be used to define a through hole. To check a box, the Detect Optimal Pre-Drill Points checkbox must be unchecked. When a through hole box is checked, the tool will assume there's a hole drilled through the part and not helix to remove the material left by the drill tip.

Range

The distance used when determining whether to use a manually selected point to drill or not. The value entered is a percentage of the tool diameter.

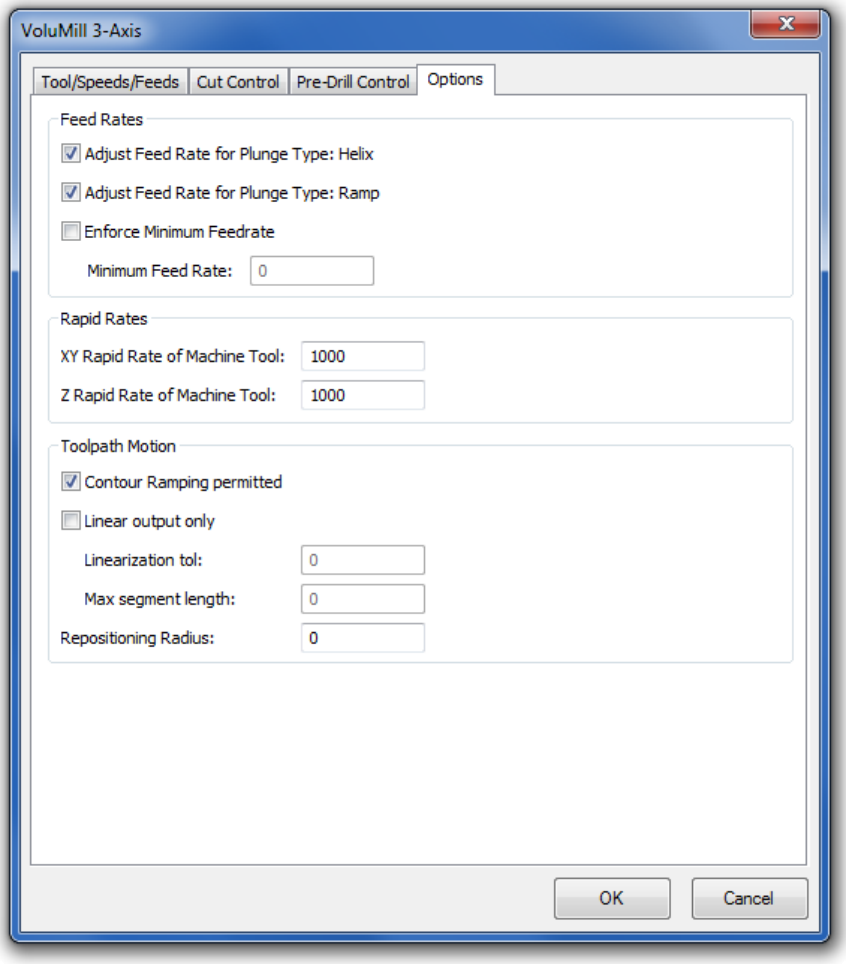
Through

To define a hole as a through hole; check the box preceding the hole location. This is only possible if the Detect Optimal Pre-Drill Points option is unchecked.

To achieve the best result using pre-drilling; generate a toolpath with the Detect Optimal Pre-Drill Points option checked. Then uncheck the Detect Optimal Pre-Drill Points option and check the box preceding the desired pre-drilled locations and regenerate.

Options Tab

The Options tab applies to 2-, 3-, 5-Axis toolpaths.



Adjust Feed Rate for Plunge Type: Helix

When the Plunge Type on the Cut Control tab is set to Helix, the feed rate at the tool center is reduced during material entry so that the periphery of the tool moves at the Plunge Feedrate. This feed rate reduction may not be desired and can be avoided by un-checking the option.

Adjust Feed Rate for Plunge Type: Ramp

When the Plunge Type on the Cut Control tab is set to Ramp, the feed rate at the tool center is reduced during material entry so that the periphery of the tool moves at the Plunge Feedrate. This feed rate reduction may not be desired and can be avoided by un-checking the option.

Enforce Minimum Feed Rate

This value defines the minimum feed rate used.

XY Rapid Feed Rate of machine tool

This value defines the rate at which the machine moves at rapid in the XY plane

Freeway Links are moves made from the end of a cut to the beginning of the next using the High Feed Rate. VoluMill compares the differences between the High Feed vs. the Rapid Rate and uses the fastest method to reduce the overall cycle time.

Z Rapid Feed Rate of machine tool

This value defines the rate at which the machine moves at along the Z-Axis

Freeway Links are moves made from the end of a cut to the beginning of the next using the High Feed Rate. VoluMill compares the differences between the High Feed vs. the Rapid Rate and uses the fastest method to reduce the overall cycle time.

Contour Ramping Permitted

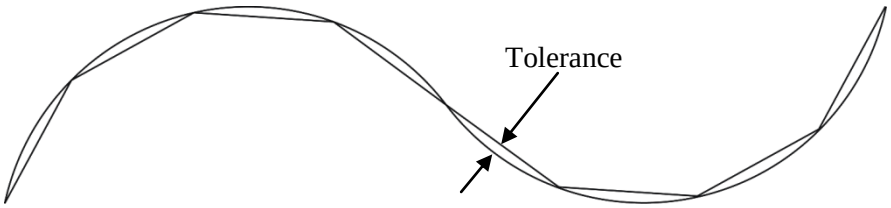
When the area to machine is too small for VoluMill to machine normally it will ramp along the profile at the Helix/Ramp Angle defined in the Cut Control tab. If this motion is not desired uncheck this option.

Linear Output only

Some machine tools benefit from having the toolpath broken up into point-to-point motion. Higher rates can be maintained therefore and reduce the cycle time.

Linearization tol:

This value is used when breaking down the toolpath motion into point-to-point motion.



Max segment length:

When toolpath motion is converted into linear (point-to-point motion) it uses the value entered for the Max Segment Length to limit the length of each move.

Repositioning Radius

A Repositioning radius changes all the sharp, non-cutting, directional moves to linear point-to-point motion representing a defined radius. This can assist a machine tool in maintaining a higher velocity.

Chapter 4

POST PROCESSING

VoluMill offers a post processor that is capable of formatting the output to wide range of formats. The following does not explain every detail of what the capabilities are, but should suffice for basic editing.

For further help or assistance, contact your local VoluMill distributor or contact Celeritive Technologies directly.

Post Processor Files

If the default installation path is accepted during installation, the sample post processors will be installed into:

C:\ProgramData\Celeritive Technologies\VoluMill NEXION\Post-Inch or \Post-MM

If a different path and directory is defined during installation, then they will be found in:

.\Post-Inch or .\Post-MM

To open a post processor, navigate to the ..\Post-Inch or ..\Post-MM directory and double-click on the desired post processor. It will open in Notepad. Any editor may be used to edit a post processor.

VoluMill post processors use *.vmpst extensions. Although any extension may be used, *vmpst is recommended.

Post Processor Formatting

VoluMill post processors are designed to make them easy to create and modify, while providing flexibility to support many different output formats.

Version

Each post should have the version defined on the first line. This comment should not be removed from the post processor.

// VoluMill Post Processor v1.0.0 !!Do not delete this line!!

Comments

Comments may be added to describe certain characteristics of particular formats within the post. They also help when editing a post after a period of time to describe specifics. Comments are solely used for descriptive text within the post processor and have no effect on the output.

To create a comment begin the line with "//". Example:

// *This is a comment in a post processor*

Comments must be on separate lines. Anything may follow the "//".

Sections

Sections are used to perform specific tasks, such as formatting the output of a tool change to defining what letter addresses are modal, etc.

There are specific sections available. The following describes the available sections:

section modal

This section defines what letter addresses are modal, or not repeated if the value is the same.

section format

This section defines the decimal formatting for the letter addresses.

section startup

This section defines the format for the start of the program.

section rapid

This section defines the format for the rapid motion of the toolpath, typically G0 blocks.

section linear

This section defines the format for the linear feed motion of the toolpath, typically G1 blocks.

section cw

This section defines the format for the clockwise circular motion of the toolpath, typically G2 blocks.

section ccw

This section defines the format for the counter clockwise circular motion of the toolpath, typically G3 blocks.

section spindlespeed

This section defines the format for the spindle speed changes.

section shutdown

This section defines the format for the end of the program.

Formatting within a section begins with "**{**" and ends with "**}**"

Letter Addresses

Letter addresses are not limited to a single character. Almost any combination of characters may be used, except " **[{ }]** " . Letter addresses must be followed by variables. Otherwise, the post processor treats the characters a straight text.

Variables

Variables are not case sensitive. The following variables are provided:

toolnumber

The current tool number.

tooldiameter

The current tool diameter.

toolcornerradius

The current tool corner radius.

topmaterialz

The absolute z-coordinate of the top of the stock.

rapidplanez

The absolute z-coordinate of the rapid plane.

startx

The first absolute x-coordinate of the toolpath.

starty

The first absolute y-coordinate of the toolpath.

startz
The first absolute z-coordinate of the toolpath.

endx
The last absolute x-coordinate of the toolpath.

endy
The last absolute y-coordinate of the toolpath.

endz
The last absolute z-coordinate of the toolpath.

ncfilename
The filename.

sequencenumber
The calculated sequence number.

sequenceincrement
The sequence number increment.

angle
The swept angle of the arc.

X
The current absolute x coordinate.

Y
The current absolute y coordinate.

Z
The current absolute z coordinate.

I
The current signed distance along the x axis from the end of an arc to its center.

J
The current signed distance along the y axis from the end of an arc to its center.

R
The current radius of an arc.

F
The current feed rate.

S
The current spindle speed.

prevx
The previous x coordinate.

prevy
The previous y coordinate.

prevz
The previous z coordinate.

centerx
The current absolute x axis coordinate of the center of an arc.

centery

The current absolute y axis coordinate of the center of an arc.

dirx

The X component of the tangent vector at the start of an arc.

diry

The Y component of the tangent vector at the start of an arc.

XD

The X component of the tool axis.

YD

The Y component of the tool axis.

ZD

The Z component of the tool axis.

linearizeHelix = 0.001

Breaks helixes into point-to-point moves. The chord is defined by the value.

linearizeArc = 0.001

Breaks arcs into point-to-point moves. The chord is defined by the value.

OutputDirectory = "C:\Users\User\Documents"

Defines the default output location. The output directory must be within quotes, as shown.

OutputName = "filename"

Defines the filename. The output directory must be within quotes, as shown. The variable [filename] may be used to use the same name as the file.

OutputExtension = "extension"

The extension must be within quotes, as shown.

Letter Address Formatting

Decimal formatting is accomplished as follows:

"variable" "number of digits left of decimal" "decimal character" "number of digits left of decimal"

The number of digits may be a range or forced. The following are example formats:

Format	Input value	Output
F 0-4 "." 0-3	1.25	F1.25
F 4 "."3	1.25	F0001.250
F 4 "" 3	1.25	F0001250
F 0-4 "." 3	1.25	F1.250
F 0-4 "." 1-3	1	F1.0
F 1-4 "." 1-3	0	0.0

VoluMill™

Developed by
Celeritive™