

CELERITIVE TECHNOLOGIES, INC.

# VOLUMILL™ UNIVERSAL REFERENCE GUIDE





# VoluMill Universal Reference Guide

8/30/2010

## VOLUMILL™ Universal Reference Guide

Date: August 2010

Copyright© 2010 Celeritive Technologies, Inc. All rights reserved.

Software: VoluMill Universal v3.5

### **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-INFRINGEMENT. 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 ([ey@cryptsoft.com](mailto:ey@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

Introduction.....	1
Installation.....	2
Licensing.....	4
Technical Support.....	4
Tutorials .....	5
Programming a 2-axis Part.....	5
Programming a 3-axis Part.....	11
User Interface.....	17
Menus.....	17
File .....	17
Edit.....	18
View.....	18
Create .....	18
Toolbars .....	19
Main .....	19
View.....	19
Toolpath .....	20
Tools .....	20
Select.....	21
Coordinate Systems .....	22
Toolpath Manager.....	24
Workspace.....	27
Right-click.....	27
Chaining.....	27
Units.....	28
2-axis Parameters .....	29
3-Axis Parameters.....	37
Post Processing .....	47
Post Processor Files .....	47
Post Processor Formatting .....	47
Troubleshooting .....	51

# Chapter 1

## INTRODUCTION

This chapter covers the following topics:

- Installation
- Licensing
- Technical Support

Welcome to the VoluMill™ Universal Reference Guide. This guide is designed to provide all the information possible in using VoluMill Universal 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 Universal 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](http://www.volumill.com) for more information on the engine and its toolpath.

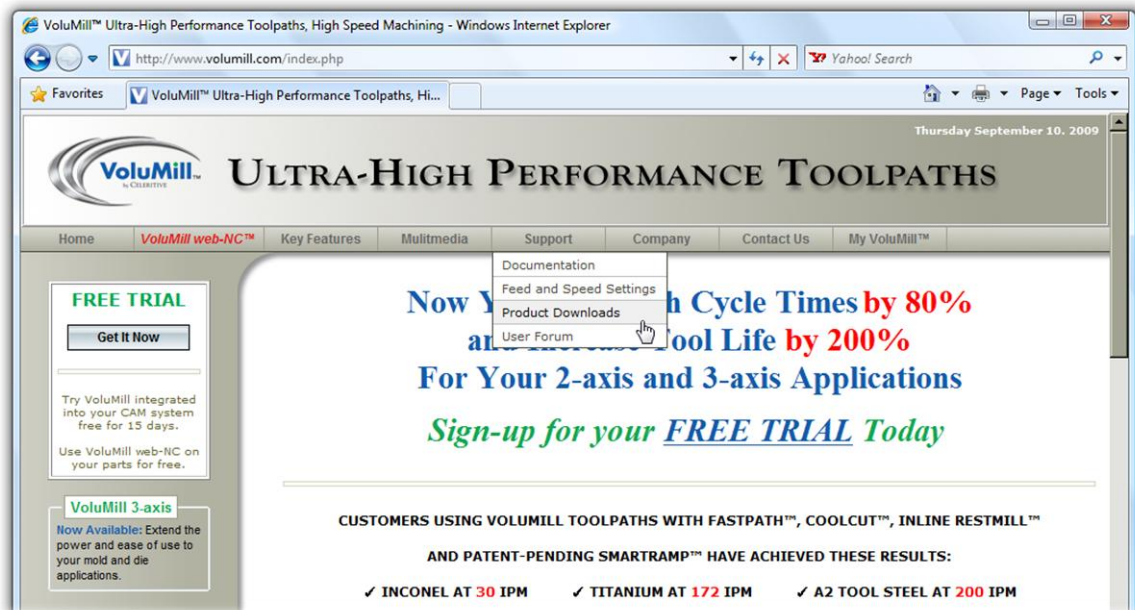
A license is required to use VoluMill. To purchase VoluMill Universal, please contact your local authorized VoluMill reseller or Celeritive Technologies directly through our website, [www.volumill.com](http://www.volumill.com), call us at (888) 253-6701, or email [sales@celeritive.com](mailto:sales@celeritive.com).

# Installation

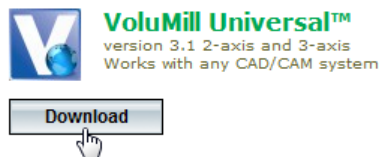
VoluMill is available as a download from the Celeritive Technologies website.

To download and install VoluMill:

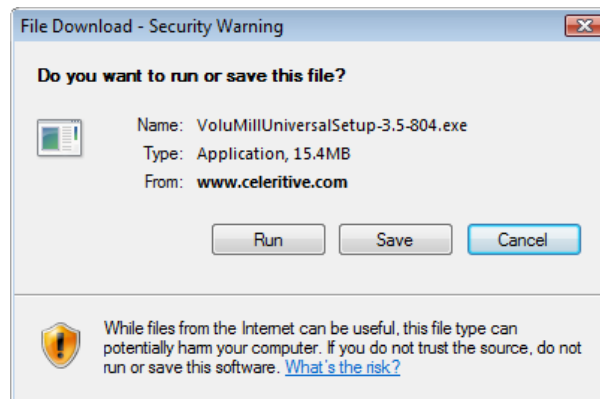
- 1) Go to [www.volumill.com](http://www.volumill.com), and then select Product Downloads from the Support menu:



- 2) Locate VoluMill Universal, then push the Download button:



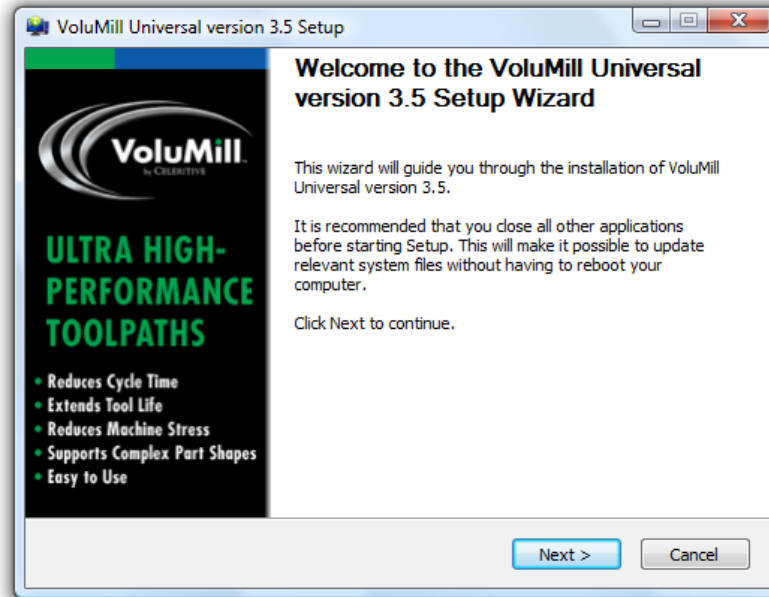
A dialog will appear asking you to either run or save the file. If you select Run, the software will be downloaded into your computer's temporary folder and the installation program will begin automatically.



If you select Save, you will 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 Universal program group.

## Technical Support

Our Technical Support department is available to answer your questions Monday through Friday, 8:00 AM to 5:00 PM, Pacific Standard Time. 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](http://www.celeritive.com) or [www.volumill.com](http://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](http://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](mailto:support@celeritive.com).

## Chapter 2

# TUTORIALS

The following tutorials provide step-by-step instructions on how to effectively use VoluMill Universal.

Example Feed Rates, Spindle Speeds, etc., are used as possible values. Your setup, tooling, material, machine tool, etc., will need to be considered when programming actual parts. Examples of actual values used in testing and from customer experiences are available from the VoluMill website: [www.celeritive.com](http://www.celeritive.com).

## Programming a 2-axis Part

What you will learn...

- How to create a 2-Axis VoluMill toolpath from a DXF file
- How to use the chaining options to define open edges of a pocket
- How spindle speeds and feed rates apply to VoluMill
- How to backplot a toolpath
- How to post process a toolpath

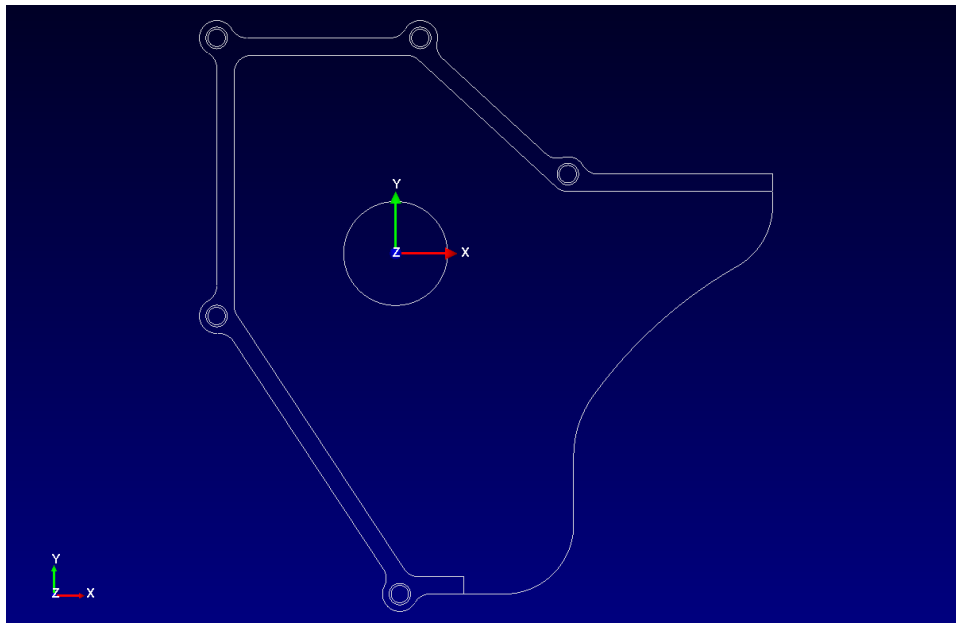


Launch VoluMill Universal and **open:**

C:\Program Files\Celeritive Technologies\VoluMill Universal\Samples\**VOLUMILL 2-AXIS.DXF**

*Note: Make sure you set your File Type to DXF when opening the file.*

Your workspace should appear as follows:

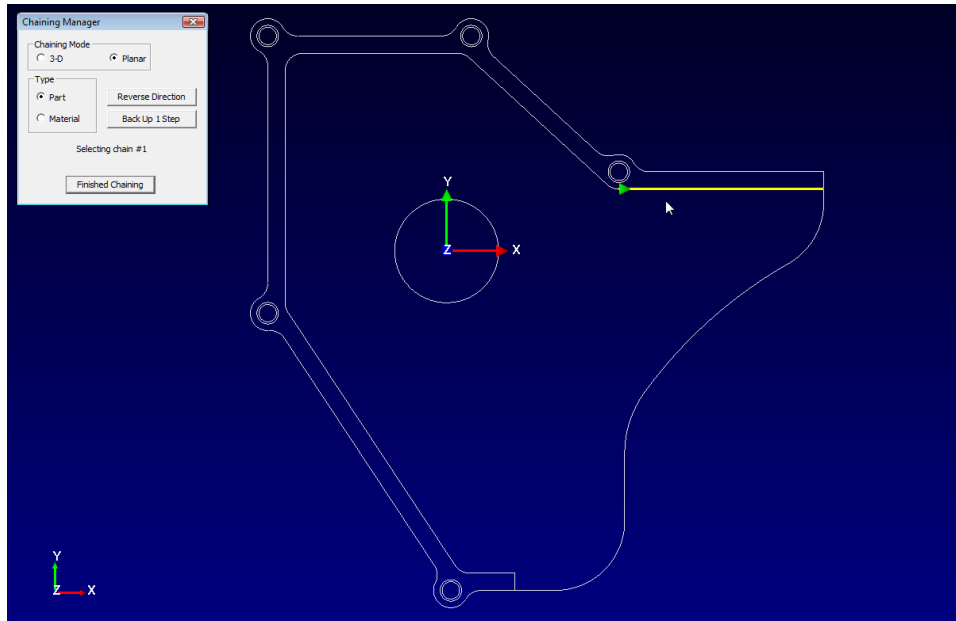




Select **VoluMill 2-Axis** from the Toolpaths menu. The Chaining Manager dialog will appear.

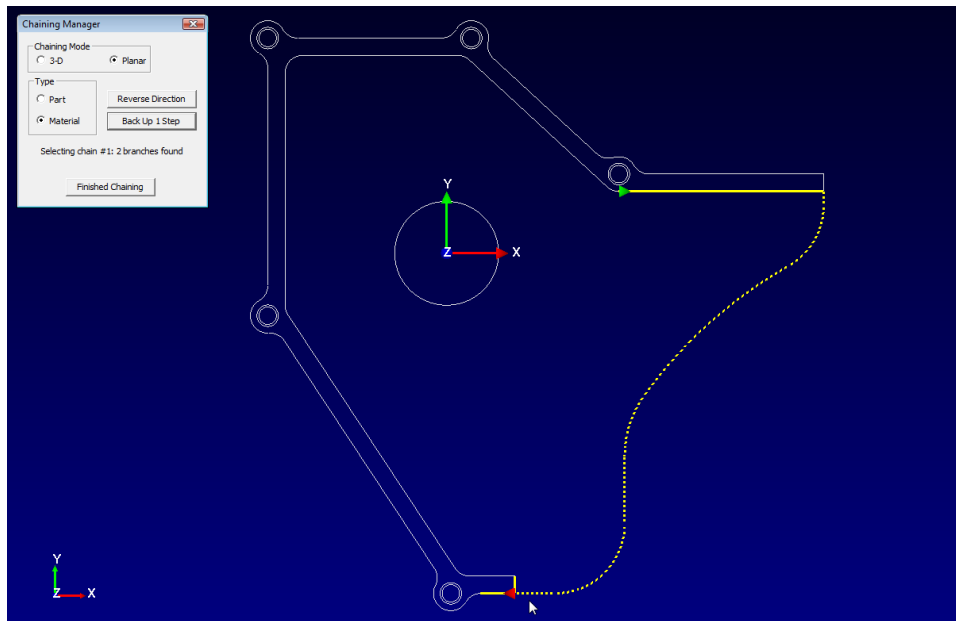
In looking at the image below, **select** the line that is shown in red. This shows the beginning of the chain and its direction. If the arrow is at the opposite end on the line and pointed to the left push the **Reverse Direction** button to switch the starting point of the chain.

At any point in the chaining process, pushing the Back Up 1 Step button will undo the last chain selection to correct for a mistake.



Change the Type in the Chaining Manager dialog to **Material**. **Click** the short vertical line below the red line to continue the chain. You'll notice the line type is now dashed. This indicates the chain is now defining the open part of the pocket.

**Click** the same line again and the chain will continue until it reaches a branch point.

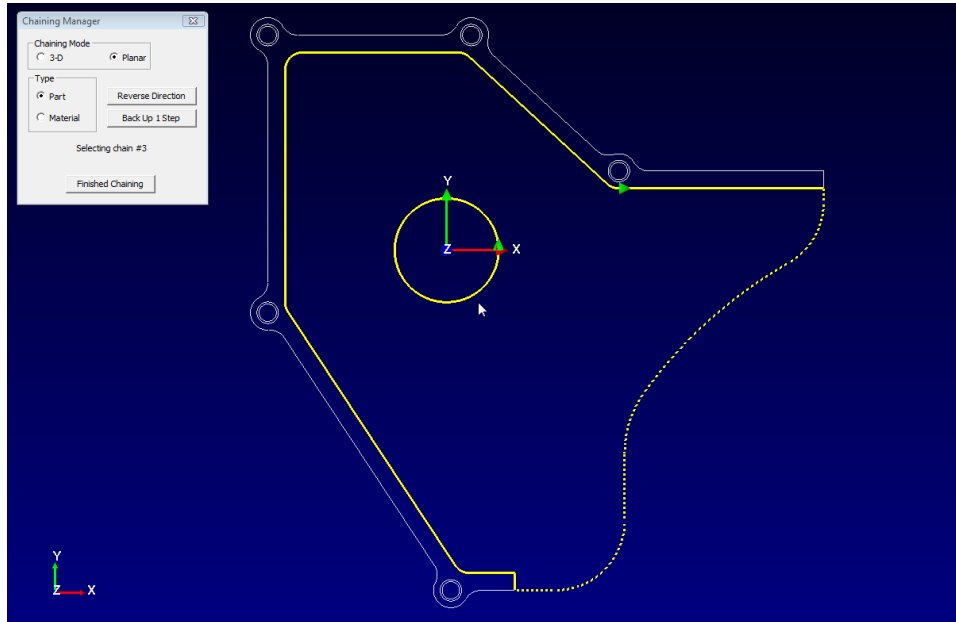




Now that the chain has reached a branch point it prompts for selection of the possible directions to continue. The available branches are highlighted in yellow. Change the Type back to **Part**. **Click** on the short vertical branch(line) to continue the chain. **Click** again to complete it.



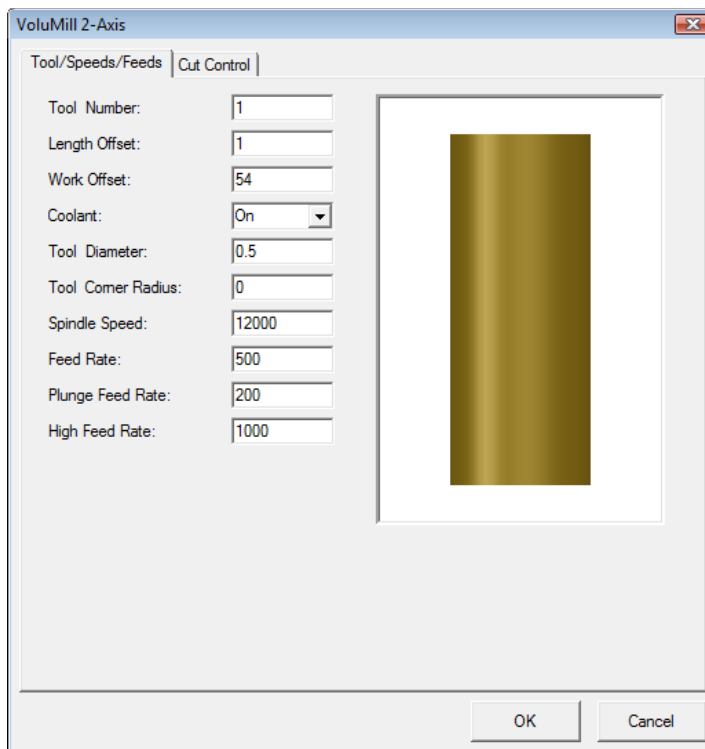
**Click** on the island to chain it and then push the **Finished Chaining** button.



When the VoluMill 2-axis dialog appears you'll see two tabs. The Tool/Speeds/Feeds tab lists all the parameters to define the tool, the spindle speeds, and the feed rates. The second tab lists the parameters associated with cutting.

Unlike traditional roughing, VoluMill is designed to run at much higher spindle speeds and feed rates. The spindle speeds and feed rates we will use are only an example of what is possible for machining a part like this in aluminum.

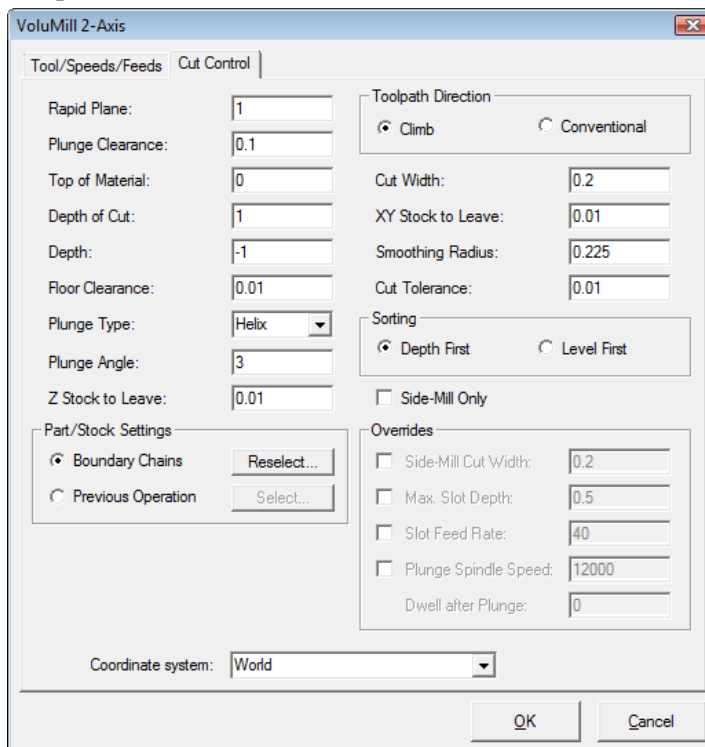
- ✓ Enter **0.5** for the **Tool Diameter**
- ✓ Enter **12000** for the **Spindle Speed**
- ✓ Enter **500.0** for the **Feed Rate**
- ✓ Enter **200.0** for the **Plunge Feed Rate**
- ✓ Enter **1000.0** for the **High Feed Rate**



Select the Cut Control tab.

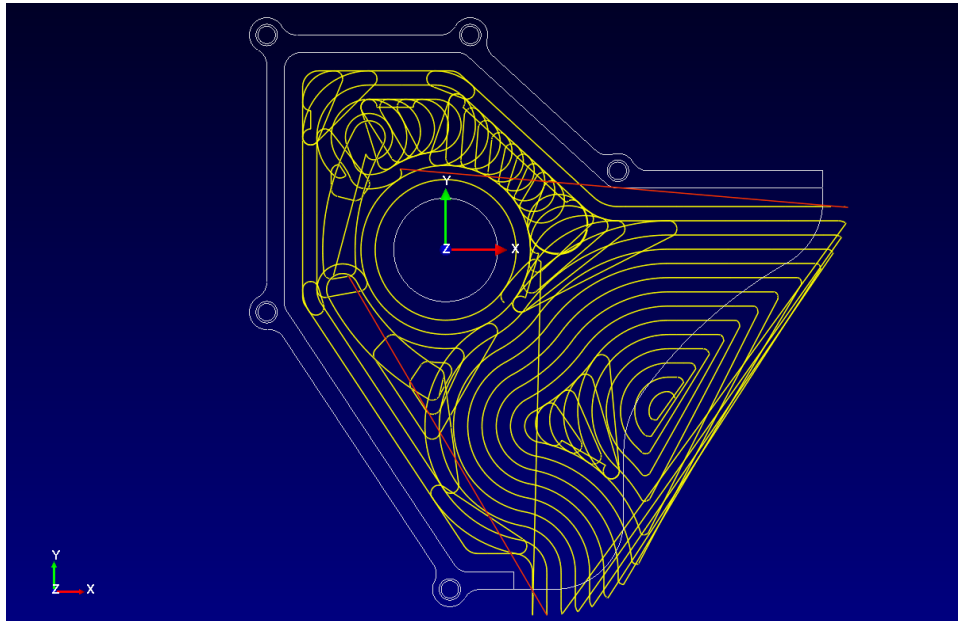
- ✓ Enter **1.0** for the **Depth of Cut**
- ✓ Enter **-1.0** for the **Depth**
- ✓ Enter **0.01** for the **Z Stock to Leave**
- ✓ Enter **0.2** for the **Cut Width**
- ✓ Enter **0.01** for the **XY Stock to Leave**

Set all other parameters as shown below.



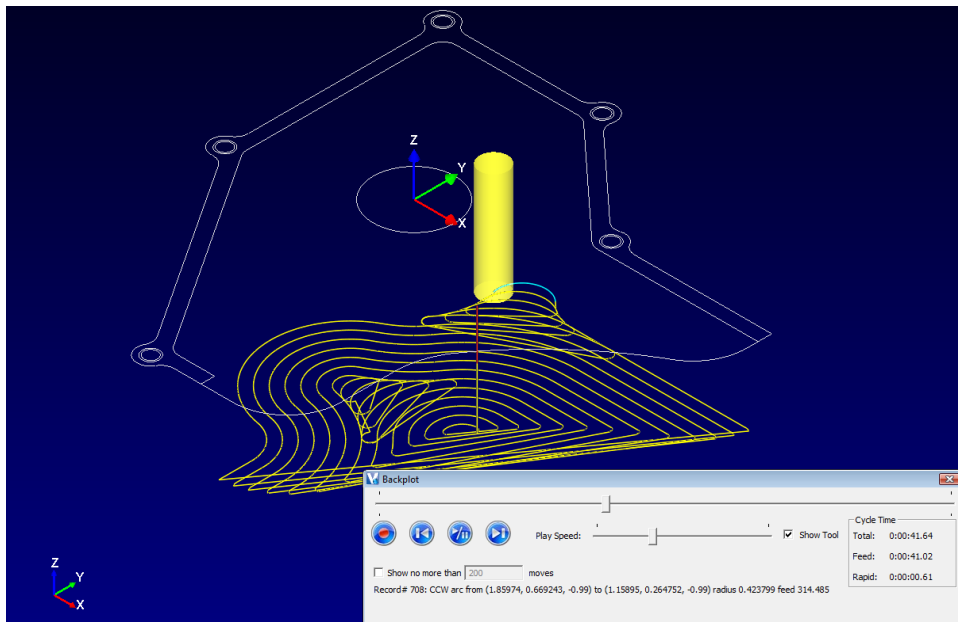


Press **OK** and the system will generate a VoluMill toolpath



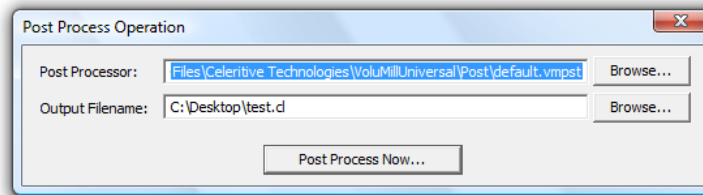
**Highlight** the toolpath by moving the mouse over it, **right-click**, and then select **Backplot Toolpath**.

Push the Play button to begin simulating the toolpath. Use the Play Speed to control how fast the tool moves. Try changing views to look at the toolpath from different angles.

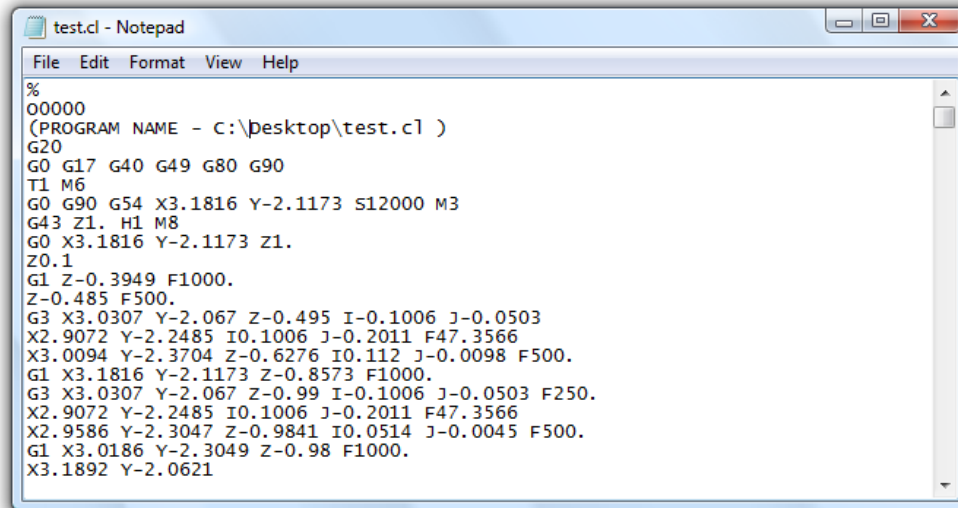


**Highlight** the toolpath by moving the mouse over it, **right-click**, and then select **Post Process**.

The Post Process Operation dialog will open. Push the **Browse...** button to navigate to a desired directory and provide an **Output Filename**.



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

A screenshot of a Notepad window titled 'test.cl - Notepad'. The window contains a menu bar (File, Edit, Format, View, Help) and a text area with the following G-code:

```
%
O0000
(PROGRAM NAME - C:\desktop\test.cl )
G20
G0 G17 G40 G49 G80 G90
T1 M6
G0 G90 G54 X3.1816 Y-2.1173 S12000 M3
G43 Z1. H1 M8
G0 X3.1816 Y-2.1173 Z1.
Z0.1
G1 Z-0.3949 F1000.
Z-0.485 F500.
G3 X3.0307 Y-2.067 Z-0.495 I-0.1006 J-0.0503
X2.9072 Y-2.2485 I0.1006 J-0.2011 F47.3566
X3.0094 Y-2.3704 Z-0.6276 I0.112 J-0.0098 F500.
G1 X3.1816 Y-2.1173 Z-0.8573 F1000.
G3 X3.0307 Y-2.067 Z-0.99 I-0.1006 J-0.0503 F250.
X2.9072 Y-2.2485 I0.1006 J-0.2011 F47.3566
X2.9586 Y-2.3047 Z-0.9841 I0.0514 J-0.0045 F500.
G1 X3.0186 Y-2.3049 Z-0.98 F1000.
X3.1892 Y-2.0621
```



# Programming a 3-axis Part

What you will learn...

- How to create a 3-axis VoluMill toolpath on a STL file
- How to define the stock to machine from
- How to backplot a toolpath
- How to post process a toolpath



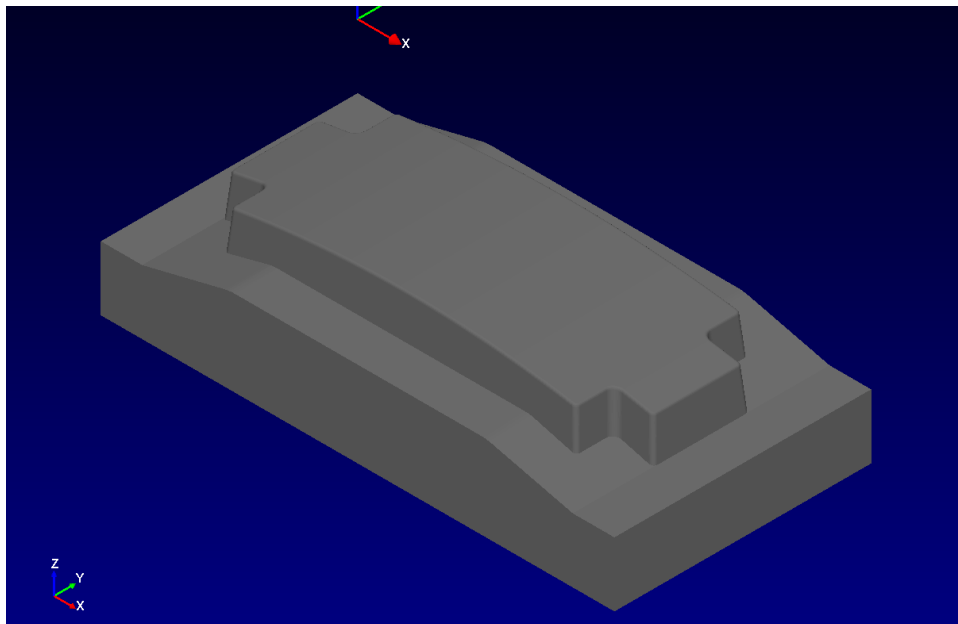
Launch VoluMill Universal and **open**:

...\VoluMill Universal\Samples\ **VOLUMILL 3-AXIS.STL**

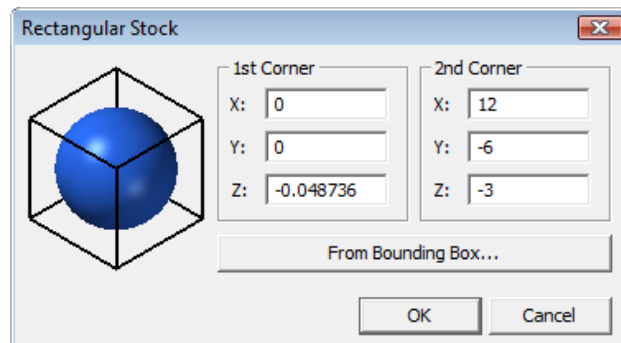
*Note: Make sure you set your File Type to STL when opening the file.*



Change to an **isometric** view and fit the part to the screen. Your workspace should appear as follows:

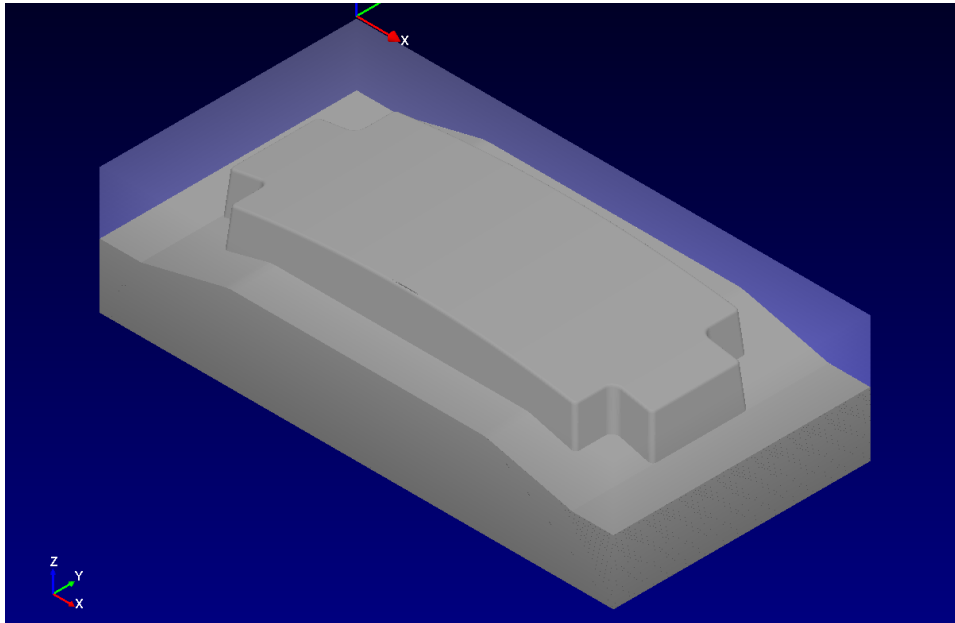


Select **Rectangular Stock** from the Create menu. The Rectangular stock dialog will appear.

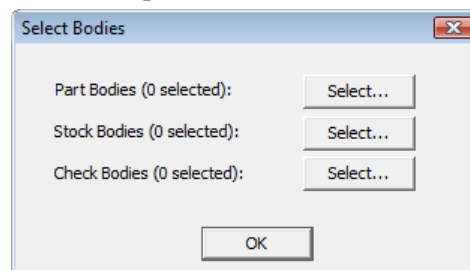


Select the **From Bounding Box...** button and select the part. The dialog will appear with the coordinates populated from the extents of the part.

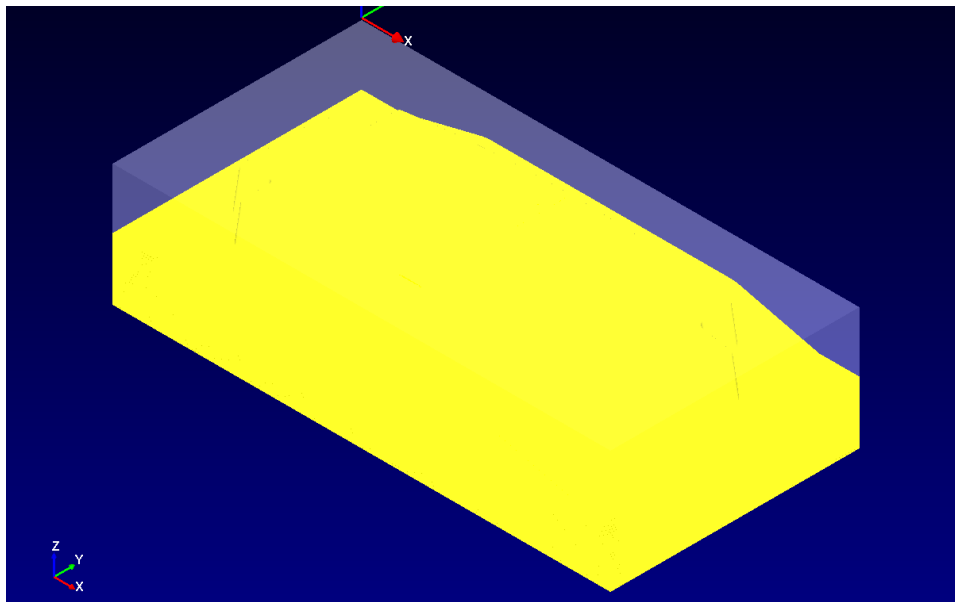
Select **OK** and the stock will appear.



Select **VoluMill 3-Axis** from the Toolpaths menu. The Select Bodies dialog will appear.



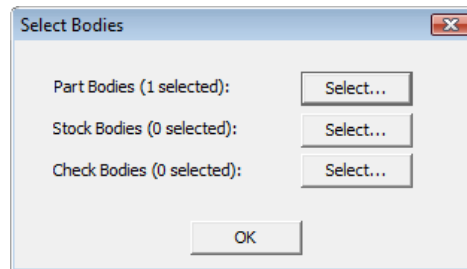
Select the **Select...** button next to **Part Bodies** and then select the part.



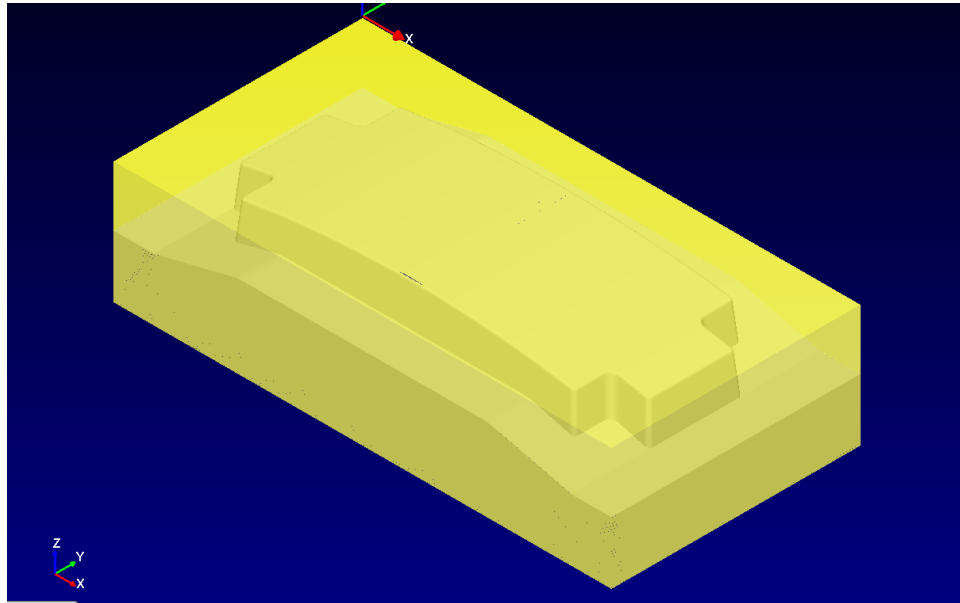
Select the **green check** icon or press **enter** to accept.



Select the **Select...** button next to **Stock Bodies**



Select the **stock**



Select the **green check** icon or press **enter** to accept. Select the **OK** button.



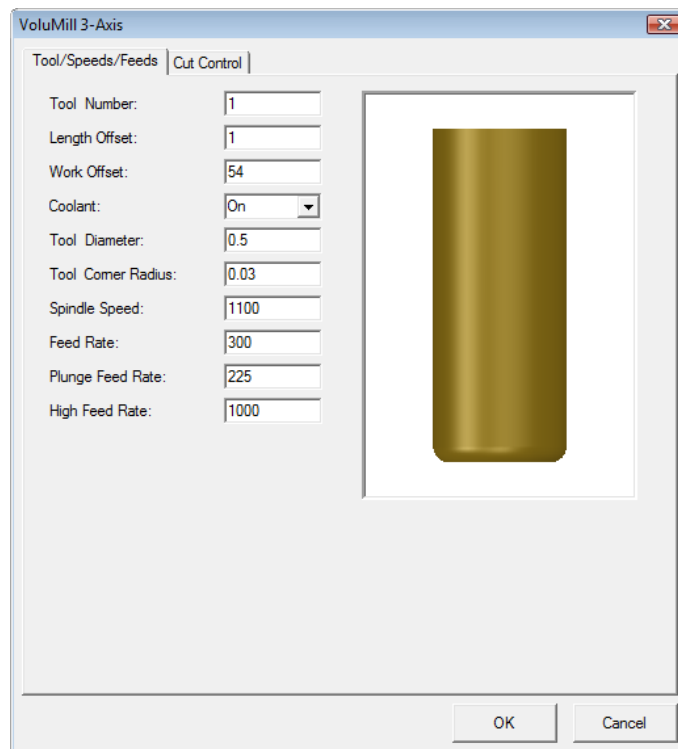
The 3-Axis dialog will appear. The Tool/Speeds/Feeds tab lists all the parameters to define the tool, the spindle speeds, and the feed rates. The second tab lists the parameters associated with cutting.

Unlike traditional roughing, VoluMill is designed to run at much higher spindle speeds and feed rates. The spindle speeds and feed rates we will use are only an example of what is possible for machining a part like this in 1045 steel.

- ✓ Enter **0.5** for the **Tool Diameter**
- ✓ Enter **0.030** for the **Tool Corner Radius**
- ✓ Enter **1100** for the **Spindle speed**
- ✓ Enter **300.0** for the **Feed rate**
- ✓ Enter **225.0** for the **Plunge rate**
- ✓ Enter **1000.0** for the **High Feed Rate**

Set all other parameters as follows

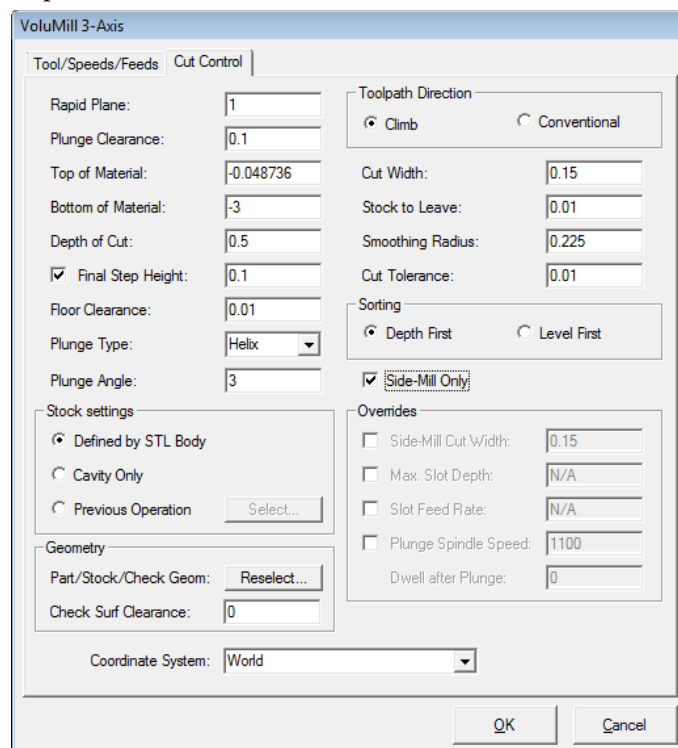
.



Select the Cut Control tab.

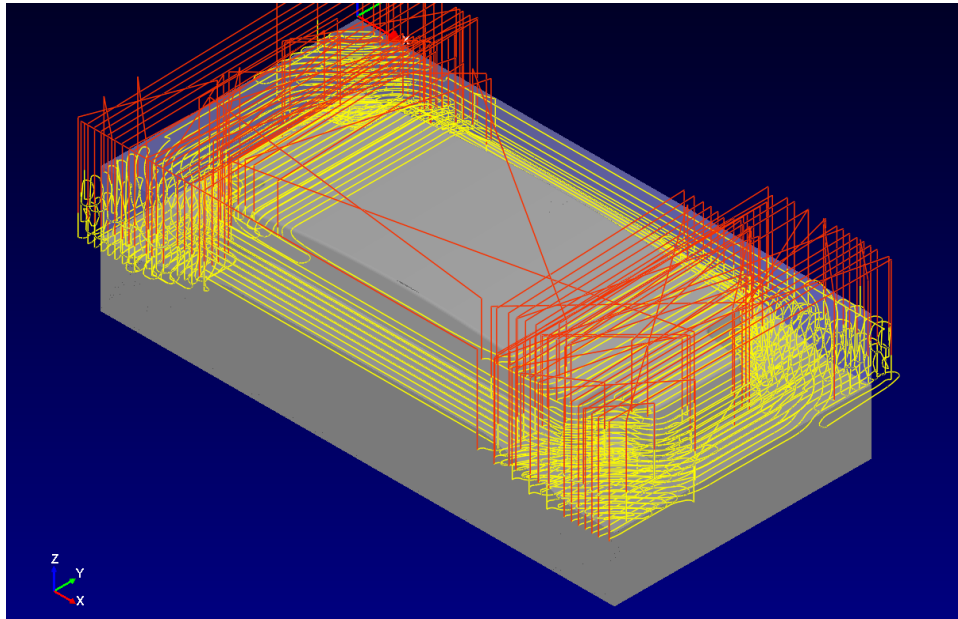
- ✓ Enter **0.5** for the **Depth of Cut**
- ✓ Check the **Final Step Height** option and enter **0.1**
- ✓ Enter **0.15** for the **Cut Width**
- ✓ Enter **0.01** for the **Stock to Leave**
- ✓ Check the **Side-Mill Only** option

Set all other parameters as shown below.



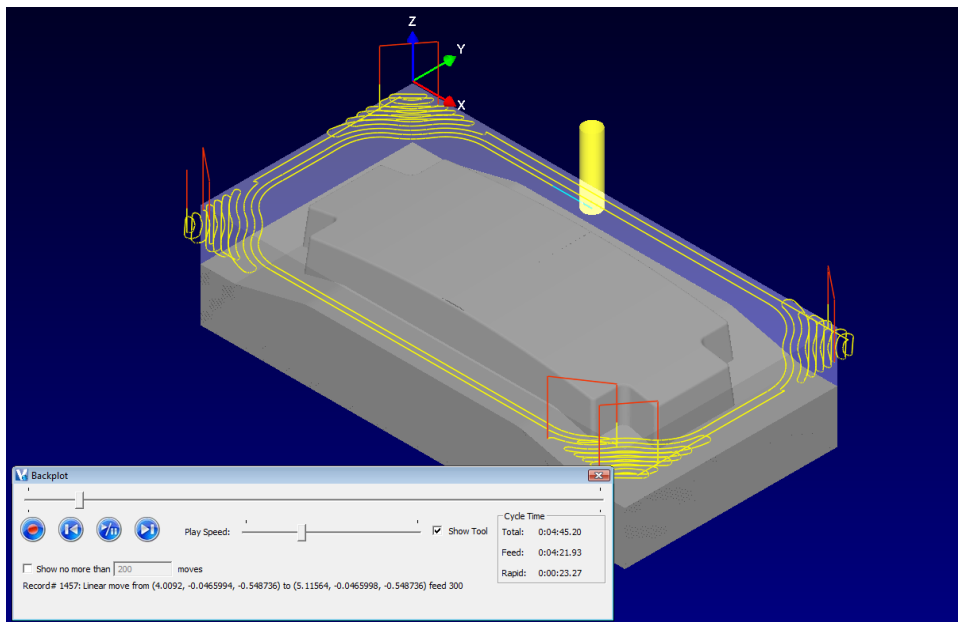


Press **OK** and the system will generate a VoluMill toolpath.



**Highlight** the toolpath by moving the mouse over it, **right-click**, and then select **Backplot Toolpath**.

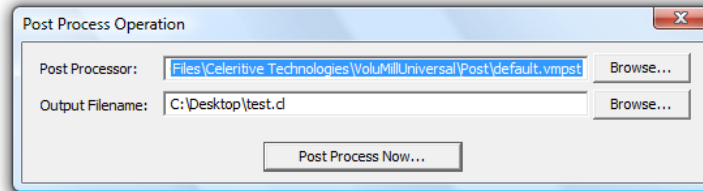
Push the Play button to begin simulating the toolpath. Use the play speed to control how fast the tool moves. Try changing views to look at the toolpath from different angles.



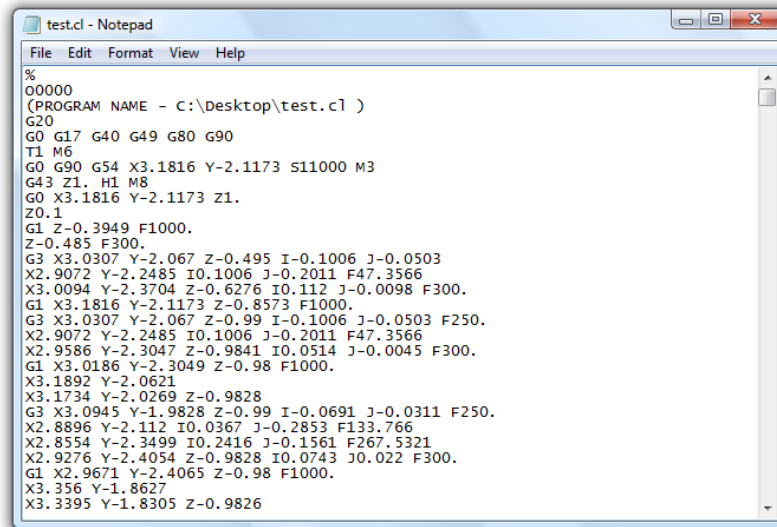


**Highlight** the toolpath by moving the mouse over it, **right-click**, and then select **Post Process**.

The Post Process Operation dialog will open. Push the **Browse...** button to navigate to a desired directory and provide an **Output Filename**.



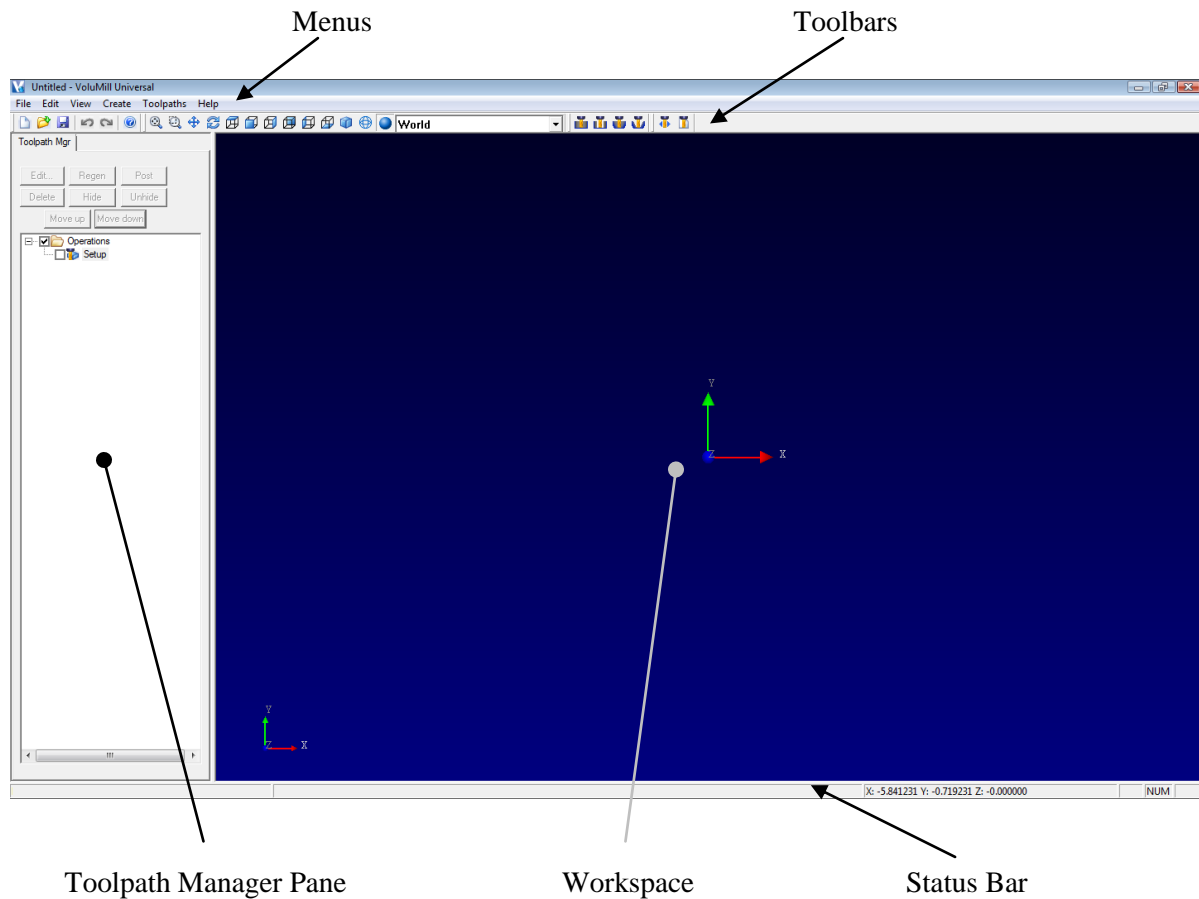
Push the **Post Process Now...** button and the system will generate G-code for the VoluMill toolpath.



## Chapter 3

# USER INTERFACE

VoluMill Universal 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 Universal 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.

## ***Merge***

Merge functions the same way Open does except that the current file is kept and the two combined into a single file.

## ***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.

## **Edit**

### ***Undo***

This choice reverses the last action.

### ***Redo***

This choice reverses the last Undo action.

## **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.

### ***Wireframe***

This choice changes the STL model, Stock, and Backplotting tool to a wireframe or polygon display mode.

### ***Shaded***

This choice changes the STL model, Stock, and Backplotting tool to a rendered display mode.

### ***Active CS***

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

## **Create**

### ***Rectangular Stock***

This choice creates a rectangular stock model that is used when machining STL files only.

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 STL model.



### ***Cylindrical Stock***

This choice creates a cylindrical stock model that is used when machining STL files only.

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 chosen STL model.

## **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 Universal 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 the STL model, Stock, and Backplotting tool to a rendered display mode.

### **Wireframe**

This choice changes the STL model, Stock, and Backplotting tool to a wireframe or polygon display mode.

### **Current 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.

## **Toolpath**

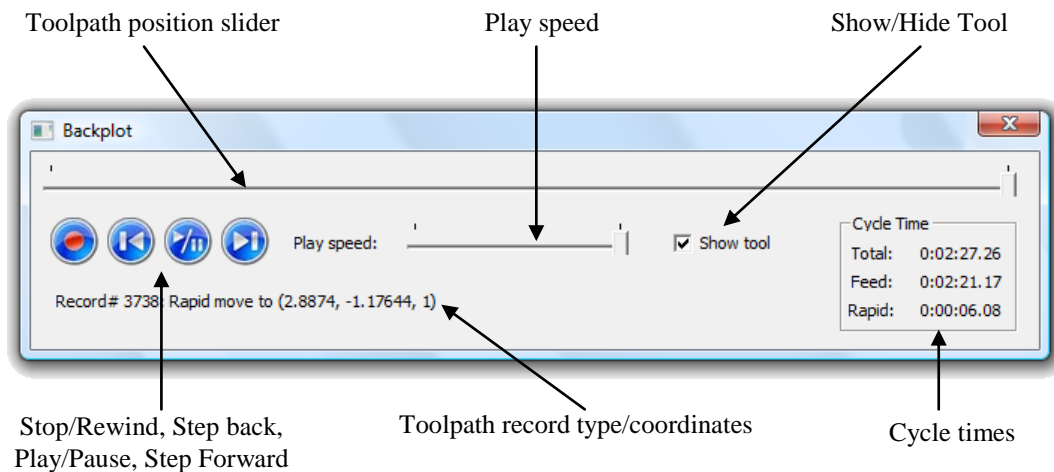
These choices are used to create VoluMill toolpaths for 2-axis, 2-axis Restmilling, 3-axis, and 3-axis Restmilling respectfully.

## **Tools**

These choices are used to backplot and post process toolpaths

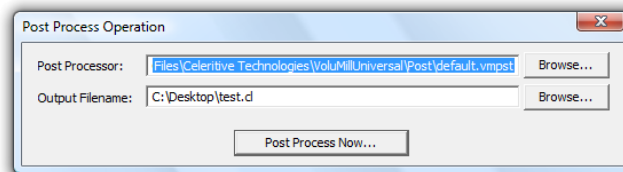
### **Backplot**

To animate the tool motion, highlight the desired toolpath, right-click and choose backplot.



### **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.

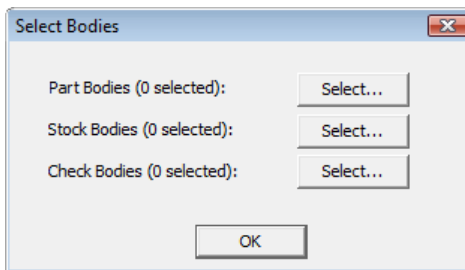
```

%
O0000
(PROGRAM NAME - C:\Desktop\test.cl )
G20
G0 G17 G40 G49 G80 G90
T1 M6
G0 G90 G54 X3.1816 Y-2.1173 S11000 M3
G43 Z1. H1 M8
G0 X3.1816 Y-2.1173 Z1.
Z0.1
G1 Z-0.3949 F1000.
Z-0.485 F300.
G3 X3.0307 Y-2.067 Z-0.495 I-0.1006 J-0.0503
X2.9072 Y-2.2485 I0.1006 J-0.2011 F47.3566
X3.0094 Y-2.3704 Z-0.6276 I0.112 J-0.0098 F300.
G1 X3.1816 Y-2.1173 Z-0.8573 F1000.
G3 X3.0307 Y-2.067 Z-0.99 I-0.1006 J-0.0503 F250.
X2.9072 Y-2.2485 I0.1006 J-0.2011 F47.3566
X2.9586 Y-2.3047 Z-0.9841 I0.0514 J-0.0045 F300.
G1 X3.0186 Y-2.3049 Z-0.98 F1000.
X3.1892 Y-2.0621
X3.1734 Y-2.0269 Z-0.9828
G3 X3.0945 Y-1.9828 Z-0.99 I-0.0691 J-0.0311 F250.
X2.8896 Y-2.112 I0.0367 J-0.2853 F133.766
X2.8554 Y-2.3499 I0.2416 J-0.1561 F267.5321
X2.9276 Y-2.4054 Z-0.9828 I0.0743 J0.022 F300.
G1 X2.9671 Y-2.4065 Z-0.98 F1000.
X3.356 Y-1.8627
X3.3395 Y-1.8305 Z-0.9826

```



This toolbar appears when applying VoluMill 3-Axis toolpaths. When a 3-Axis toolpath type is selected, the following dialog will appear.



VoluMill Universal creates a solid body, or “mesh”, from any solid or surface model that has been imported. These bodies can define the Part to machine, the Stock to machine from, or the Check bodies to avoid.

### Part Bodies

When the **Select...** button is selected, the chosen body defines the surfaces to machine.

### Stock Bodies

When the **Select...** button is selected, the chosen body defines the surfaces to machine away to the Part body(s).

### Check Bodies

When the **Select...** button is selected, the chosen body defines the surfaces to avoid.

When selecting any bodies, use the green check icon or hit enter to accept the current selections

When selecting any bodies, use the red cancel icon or hit Esc to reject the current selections

### Selection Cursor

When selecting bodies, there may be multiple possibilities depending on where the cursor is placed. When that happens, the cursor will change to a tab-select cursor:

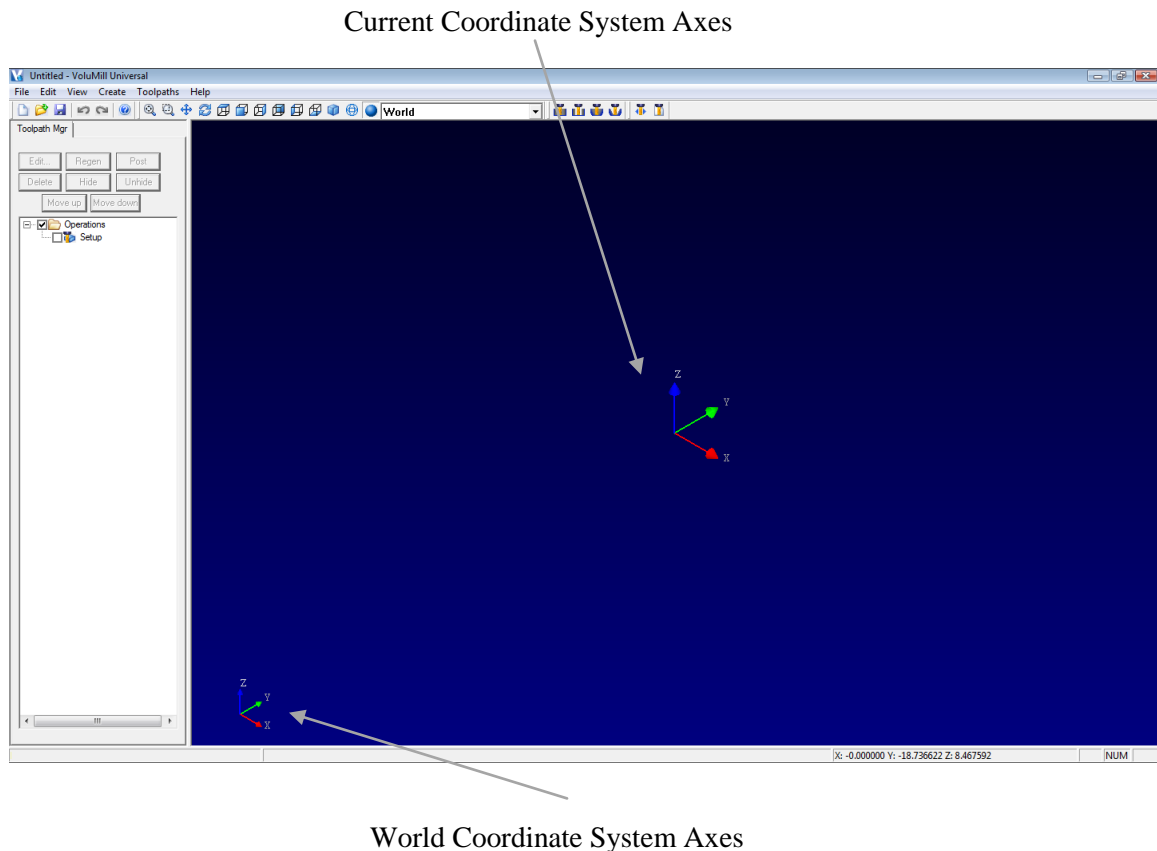
To toggle through the possibility selections, press the tab key.

# Coordinate Systems

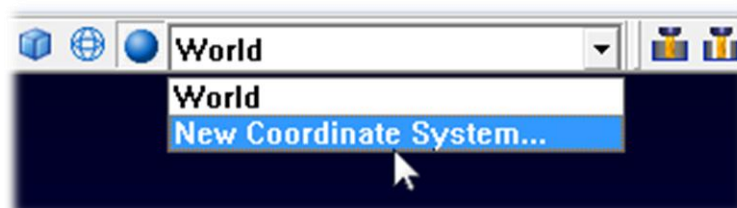
Coordinate systems define the X, Y, Z axes used when creating toolpaths. There are two different types of coordinate systems in VoluMill Universal, the World CS (coordinate system) or Current CS.

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.

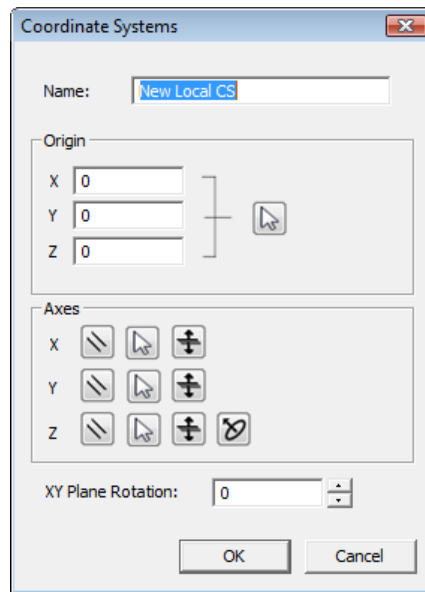
The Current CS is the coordinate system that is used when creating toolpaths. It is represented by the larger X, Y, Z axes in the center of the workspace.



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




The Coordinate Systems dialog will appear:



### **Name**





This is the name of 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**

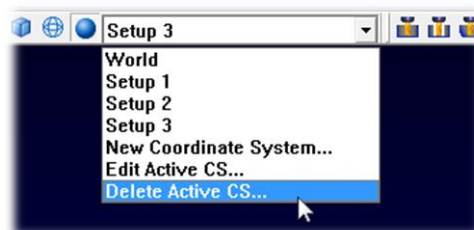
When creating a new CS there are options to align a specific axis parallel to a line, through a point, and normal to a circle. There is also an option to invert an axis.


-  Parallel – This aligns the axis to be parallel to a selected line
-  Through 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

### **XY Plane Rotation**

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.

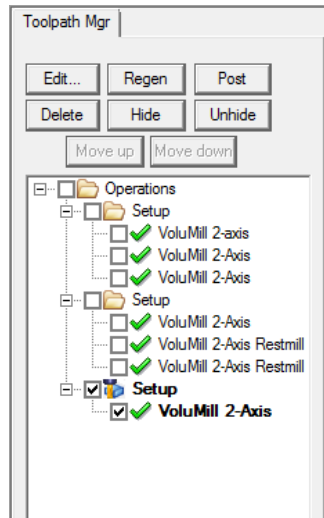
To edit or delete an existing coordinate system, select Edit Active CS... or Delete Active CS.... It's important to note that you must set the active CS before editing or deleting it.



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

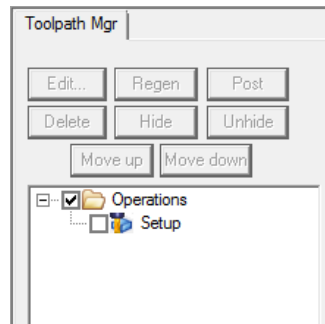
# Toolpath Manager

The Toolpath Manager is used to organize and manage toolpaths within individual setups.



Toolpaths may be managed in different ways to suit your process or method. However, the intended design is to have setups reflect different setups you would use when machining your part. It is common to have a separate setup to machine each side of a part, pause to change clamps, etc. When using setups a separate coordinate system and origin may be required as well.

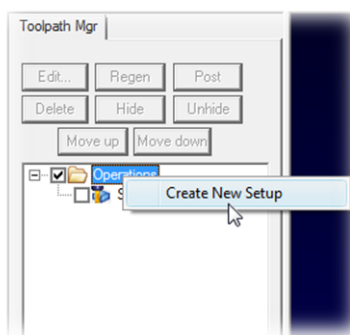
When VoluMill Universal first opens there is a single active setup:



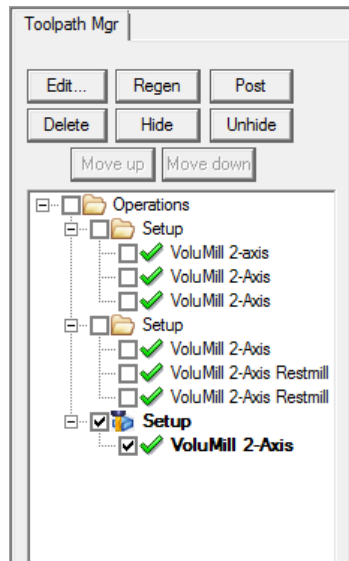
Setups are either active or inactive:



To create a setup, right-click on Operations and select Create New Setup:




As new toolpaths are created they are added to the active Setup:

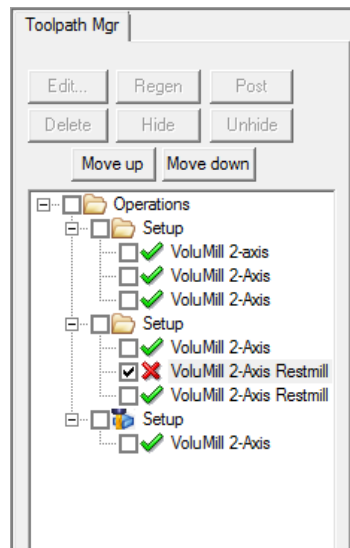


There are eight buttons at the top of the Toolpath Mgr. which buttons are available depend on what is checked.

### **Edit...**

This button opens the dialog for the tool and machining parameters for the checked toolpath so they may be changed. This is only available when a single toolpath is checked.

When a toolpath is edited it is represented by a 



### **Regen**

This button regenerates the checked toolpaths. This must be done to edited toolpaths to save the changes. There is no limit to the number of toolpaths that may be regenerated.

### **Post**

This button post processes the checked toolpaths. There is no limit to the number of toolpaths that may be post processed at once.

### **Delete**

This button post deletes the checked toolpaths. There is no limit to the number of toolpaths that may be post delete at once.

***Hide/Unhide***

These buttons hide or display the checked toolpath deletes the checked toolpaths. There is no limit to the number of toolpaths that may be hidden/shown at once.

***Naming***

To rename a setup or toolpath click once, then click again. The Operations cannot be renamed.

***Minimizing/Maximizing***

To expand or minimize setups click on the +/- or double-click on the individual step or Operations.

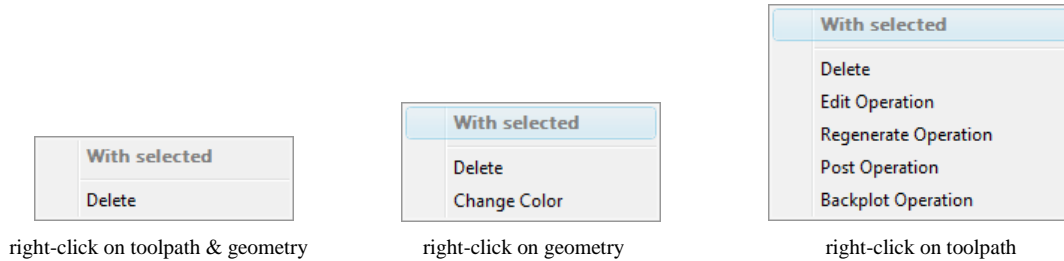
Also, the width to the Toolpath Manager pane may be resized by hovering over the right edge of the pane until the cursor changes and then holding down the left mouse button and dragging.



# Workspace

## Right-click

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



### Delete

Deletes the highlighted toolpath and/or geometry.

### Change Color

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

### Edit Operation

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

### Regenerate Operation

This choice reprocesses the highlighted toolpath.

### Post Operation

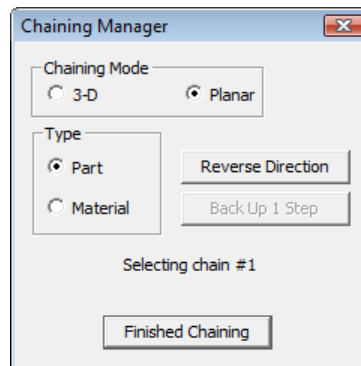
This choice displays the post-processing dialog for the highlighted toolpath.

### Backplot Operation

This choice displays the backplot dialog for the highlighted toolpath.

# Chaining

When creating a VoluMill 2-Axis toolpath, a dialog will appear offering different options



### Planar

This option allows chaining in any direction

### 3D

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***

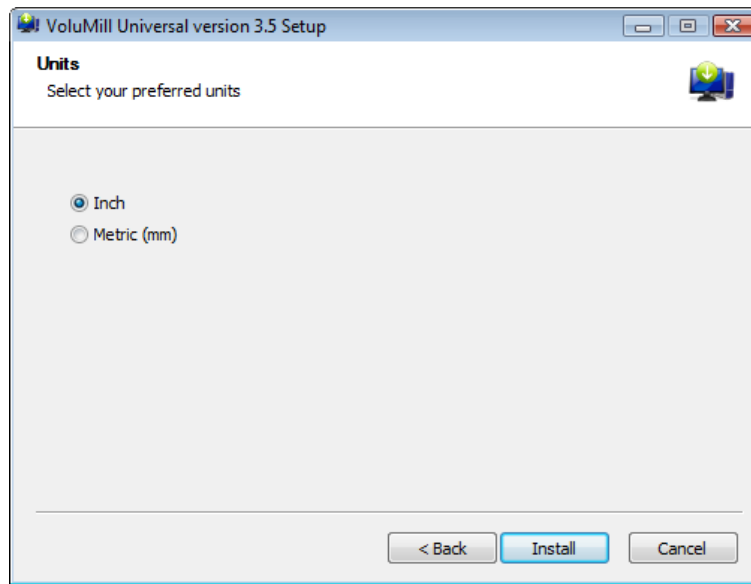
Pushing this button will step backwards along the chain one element at a time.

### ***Finished Chaining***

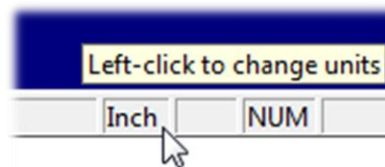
After all the profiles have been chained, push this button to proceed.

## Units

When installing VoluMill Universal the desired units are defined as either inch or Metric(mm):



To change the current units select the Inch or MM in the right corner of the status bar:

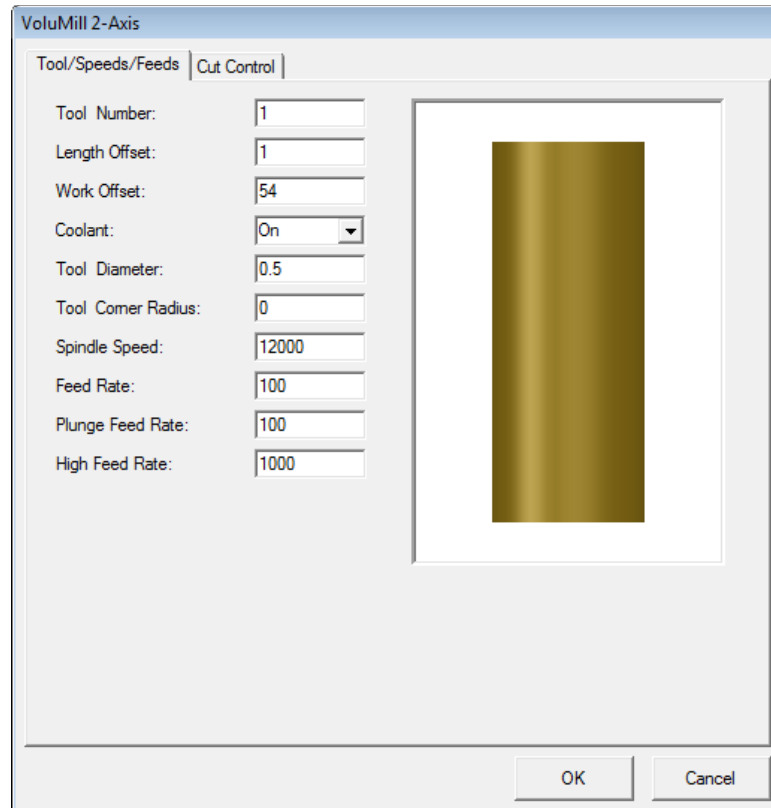


Changing the units defines the default startup state when VoluMill Universal is opened the next time.

## Chapter 4

# 2-AXIS PARAMETERS

This chapter describes the 2-axis parameters in detail and how they apply in creating VoluMill toolpaths using wireframe geometry.



The screenshot shows the 'VoluMill 2-Axis' dialog box with two tabs: 'Tool/Speeds/Feeds' and 'Cut Control'. The 'Tool/Speeds/Feeds' tab is active, displaying the following parameters:

Parameter	Value
Tool Number:	1
Length Offset:	1
Work Offset:	54
Coolant:	On
Tool Diameter:	0.5
Tool Corner Radius:	0
Spindle Speed:	12000
Feed Rate:	100
Plunge Feed Rate:	100
High Feed Rate:	1000

On the right side of the dialog, there is a preview window showing a 3D model of a cylindrical tool. At the bottom right, there are 'OK' and 'Cancel' buttons.

### ***Tool Number***

This value corresponds to the tool's position in the machine's carousel, 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.

### ***Too Diameter***

The diameter of the tool.

### ***Tool Corner Radius***

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

### ***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.

The screenshot shows the 'VoluMill 2-Axis' dialog box with the 'Tool/Speeds/Feeds' tab selected. The 'Cut Control' sub-tab is active. The settings are as follows:

Parameter	Value
Rapid Plane:	1
Plunge Clearance:	0.1
Top of Material:	0
Depth of Cut:	1
Depth:	-1
Floor Clearance:	0.01
Plunge Type:	Helix
Plunge Angle:	3
Z Stock to Leave:	0.01
Toolpath Direction:	Climb
Cut Width:	0.2
XY Stock to Leave:	0.01
Smoothing Radius:	0.225
Cut Tolerance:	0.01
Sorting:	Depth First
Side-Mill Only:	Unchecked
Part/Stock Settings:	Boundary Chains (Reselect...)
Overrides:	Side-Mill Cut Width: 0.2, Max. Slot Depth: 0.5, Slot Feed Rate: 40, Plunge Spindle Speed: 12000, Dwell after Plunge: 0
Coordinate system:	World

Buttons: OK, Cancel

## ***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 to which the tool positions.

***Top of Material***

Sets the height of the material in the Z axis.

***Depth of Cut***

Defines the maximum depth of cut. If the value does not divide equally into the total depth then VoluMill subdivides the number of cuts to create equal depths of cut.

***Depth***

Sets the overall depth of the area to be machined.

***Floor Clearance***

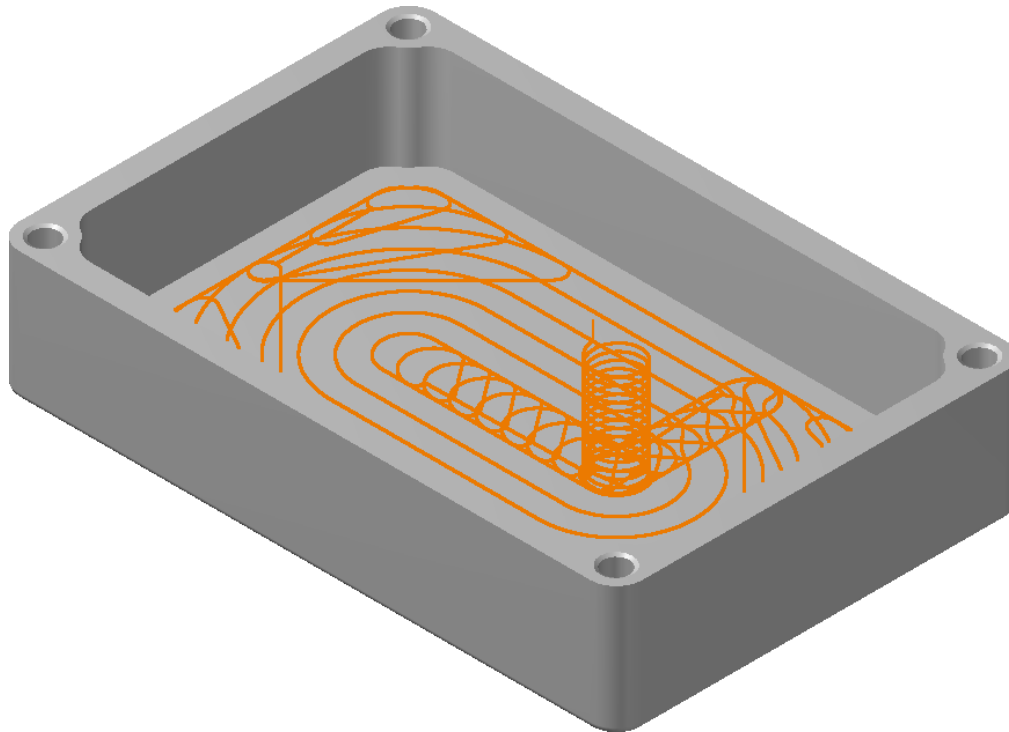
The value entered here establishes the Z-component of a helical move that is used when entering or exiting a cut. Only non-negative values are allowed. If a positive value is entered, repositioning moves between cuts will take place above the already-machined floor. If zero is entered, the tool will 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.

***Plunge Type***

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

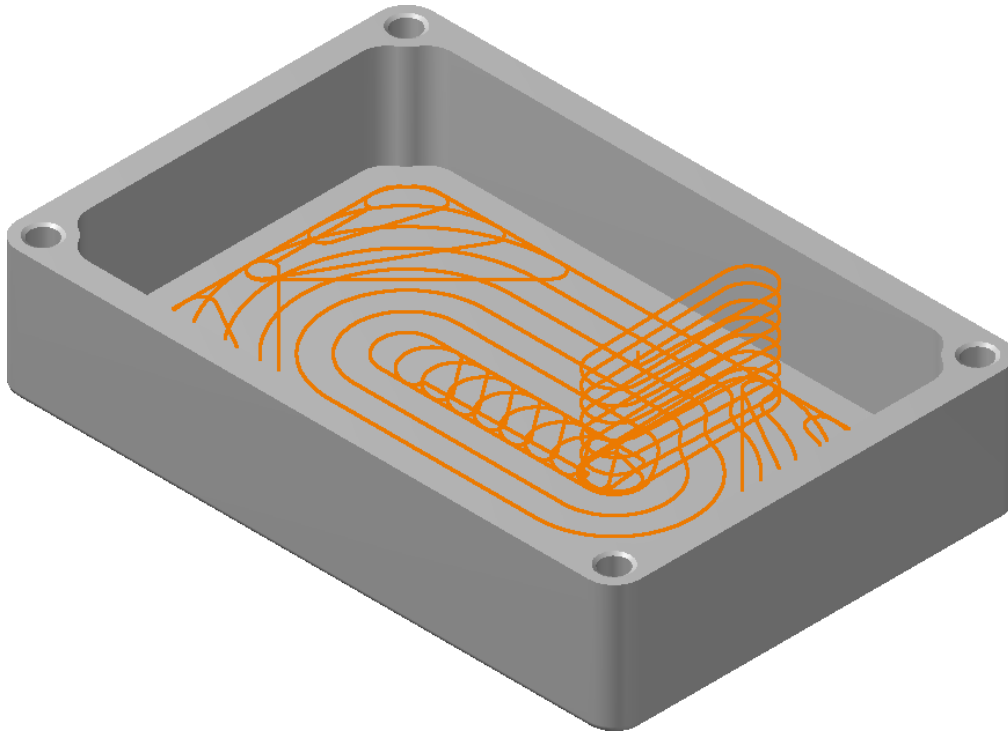
***Helix***

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.



### ***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.

### ***Z Stock to Leave***

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

### ***Boundary Chains***

When editing an existing toolpath this option prides the ability to add new chains or delete existing ones.

### ***Previous Operation***

When editing an existing toolpath this option prides the ability to define a different toolpath operation to use as a reference when rest-milling.

### ***Toolpath Direction***

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

### ***Cut Width***

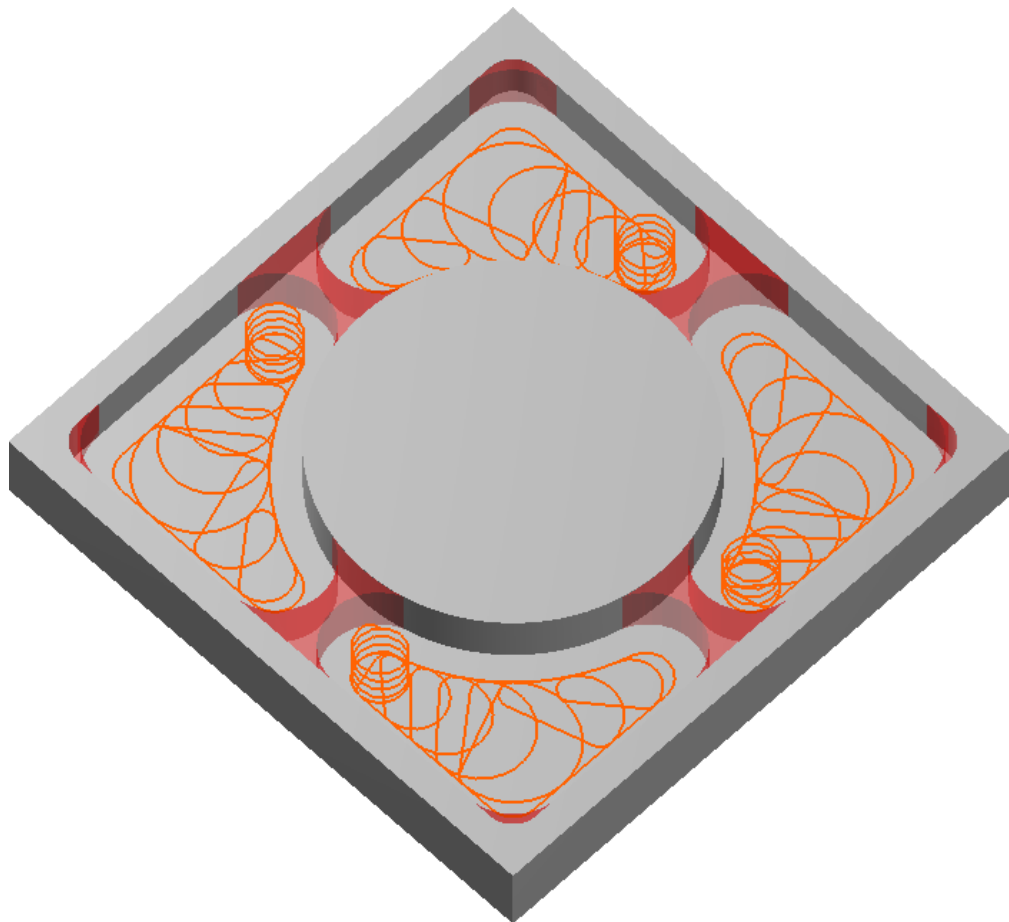
Commonly known as the stepover. With VoluMill, it's important to note that any Cut Width value that is less than the diameter of the flat portion of the tool can be used without fear of leaving uncut stands of material behind. For example, you can use up to a 100% Cut Width with a flat end mill.

### ***XY Stock to Leave***

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

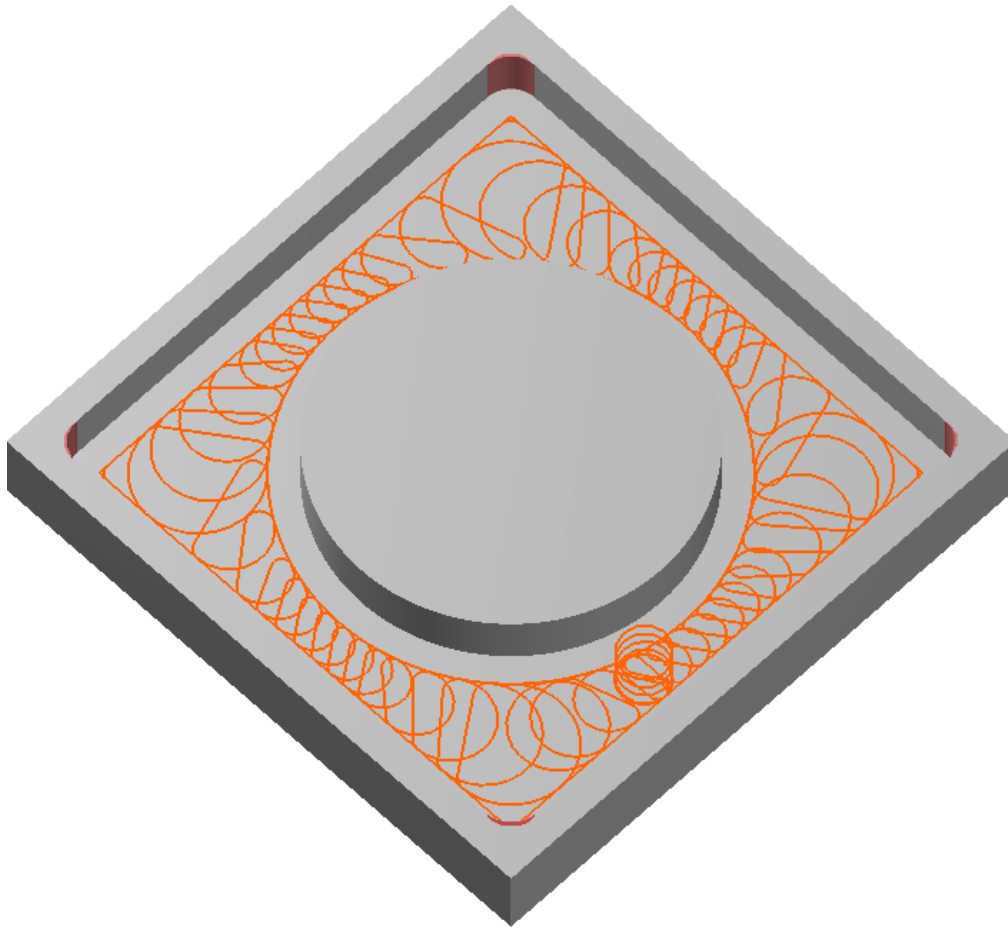
### ***Smoothing Radius***

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 tolerance used when machining profiles that include splines or polylines.

#### ***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.

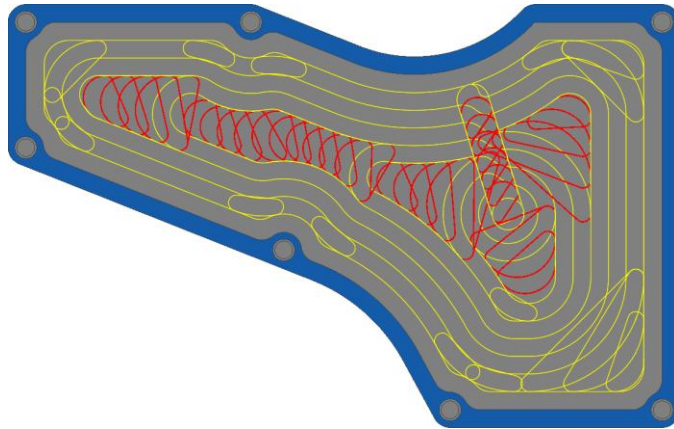
#### ***Side-Mill Only***

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.

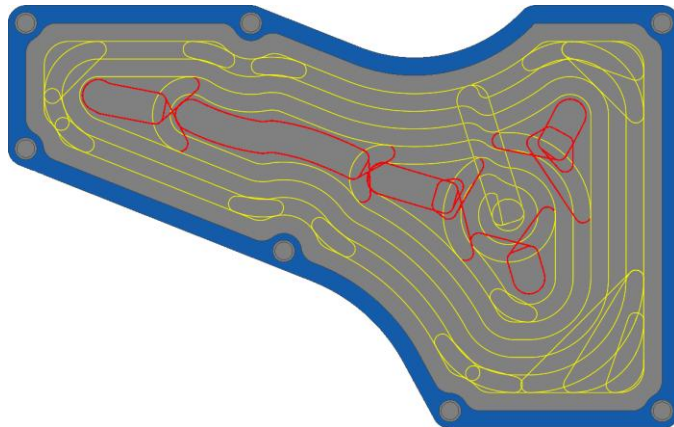
Two strategies are considered when milling in confined areas: Side Milling and Slot Milling. If this checkbox is unchecked, VoluMill will automatically choose the strategy that produces the fastest cycle time using the current feed rate and distance traveled. Depending on the shape of the part, both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

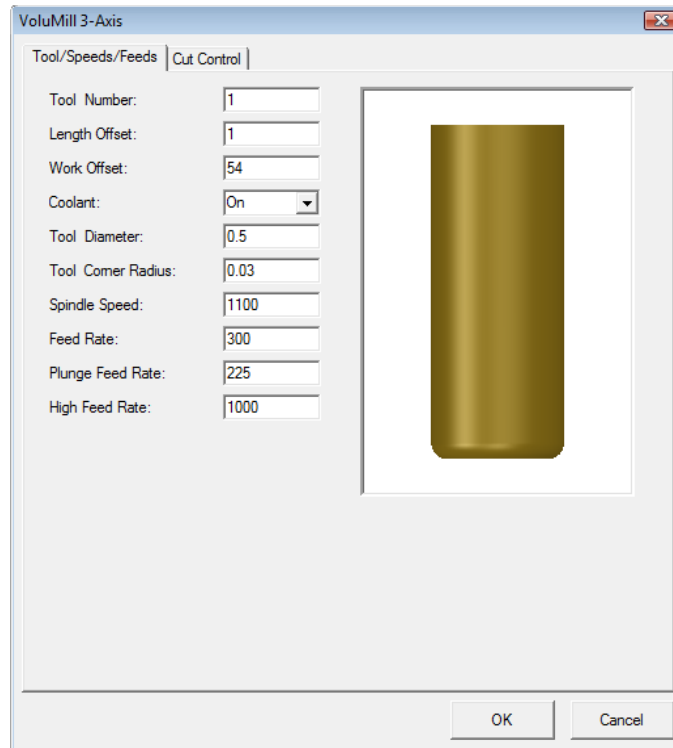
### ***Coordinate System:***

This lists the currently available coordinate systems. When creating a toolpath, VoluMill uses the current CS, or coordinate system defined at the top of the screen in the view toolbar. This list provides an opportunity to use a different CS if desired.

## Chapter 5

# 3-AXIS PARAMETERS

This chapter describes the 3-Axis parameters in detail and how they apply in creating VoluMill toolpaths using STL geometry.



### ***Tool Number***

This value corresponds to the tool's position in the machine's carousel, 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.

### ***Too Diameter***

The diameter of the tool.

### ***Tool Corner Radius***

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

### ***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.

The screenshot shows the 'VoluMill 3-Axis' software window with two tabs: 'Tool/Speeds/Feeds' and 'Cut Control'. The 'Cut Control' tab is active, displaying various parameters for toolpath generation. The parameters are organized into several sections: 'Rapid Plane' (set to 1), 'Plunge Clearance' (0.1), 'Top of Material' (0), 'Bottom of Material' (-3), 'Depth of Cut' (0.5), 'Final Step Height' (0.5), 'Floor Clearance' (0.01), 'Plunge Type' (Helix), and 'Plunge Angle' (3). The 'Toolpath Direction' section has 'Climb' selected. 'Cut Width' is 0.15, 'Stock to Leave' is 0.01, 'Smoothing Radius' is 0.225, and 'Cut Tolerance' is 0.01. The 'Sorting' section has 'Depth First' selected. The 'Side-Mill Only' checkbox is checked. The 'Overrides' section includes 'Side-Mill Cut Width' (0.15), 'Max. Slot Depth' (N/A), 'Slot Feed Rate' (N/A), 'Plunge Spindle Speed' (1100), and 'Dwell after Plunge' (0). The 'Stock settings' section has 'Defined by STL Body' selected. The 'Geometry' section has 'Part/Stock/Check Geom' (Reselect...) and 'Check Surf Clearance' (0). The 'Coordinate System' is set to 'World'. 'OK' and 'Cancel' buttons are at the bottom right.

## ***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***

Sets the starting height of the material in the Z axis.

### ***Bottom of Material***

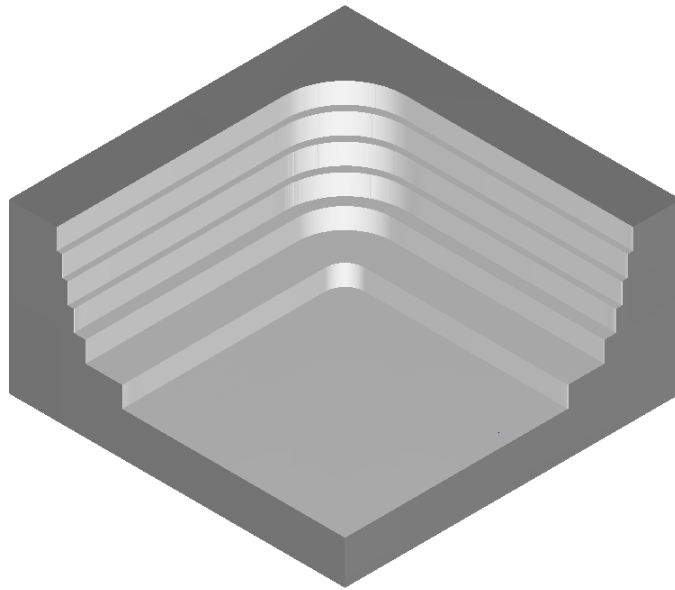
Sets the depth of the toolpath in the Z axis.

### ***Depth of Cut***

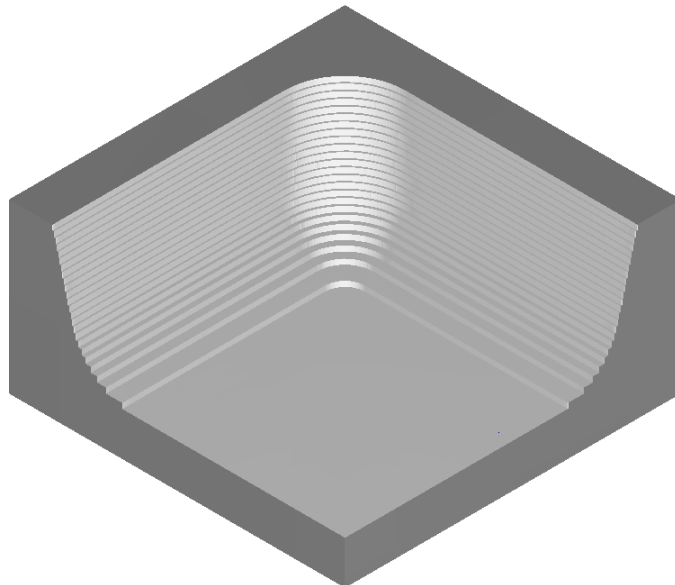
Defines the maximum depth of cut. If the value does not divide equally into the total depth, VoluMill subdivides the number of cuts to create equal depths of cut.

### ***Final step height***

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 remachine 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 machine at each Depth of Cut, the tool then reduces the step using the Final Step Height value.



### ***Floor Clearance***

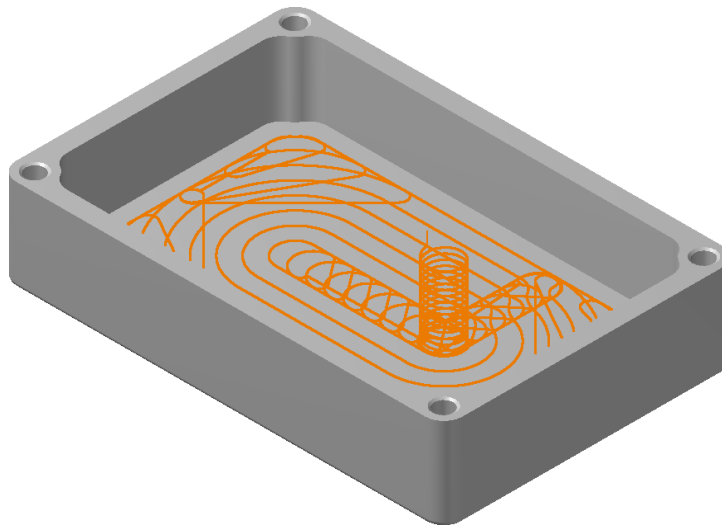
The value entered here establishes the Z-component of a helical move that is used when entering or exiting a cut. Only non-negative values are allowed. If a positive value is entered, repositioning moves between cuts will take place above the already-machined floor. If zero is entered, the tool will 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.

### ***Plunge Type***

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

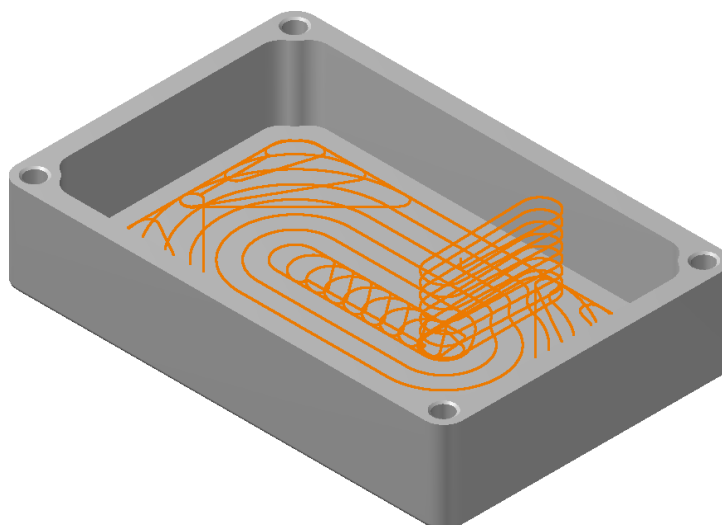
#### ***Helix***

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.



### ***Plunge Angle***

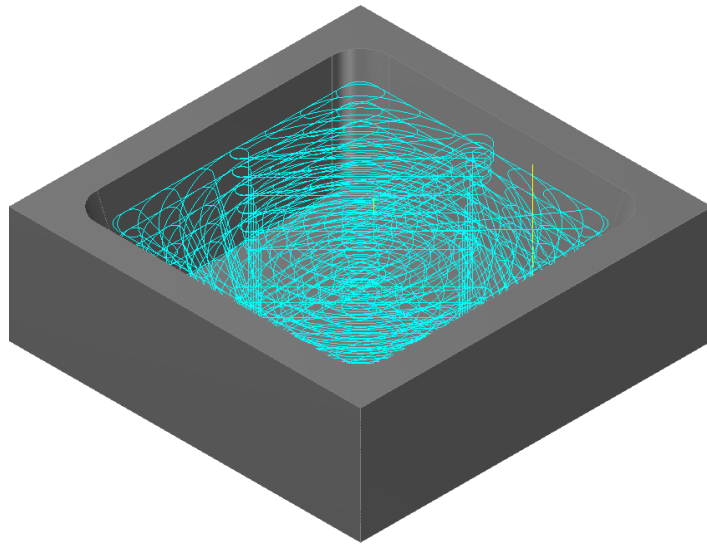
This parameter establishes the rate of descent, in degrees, at which the tool enters the material from the top, as is required when machining completely enclosed areas (pockets). VoluMill uses the entered value as a not-to-exceed value, meaning that the actual ramp angle may be adjusted downward from 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.

### ***Defined by STL Body***

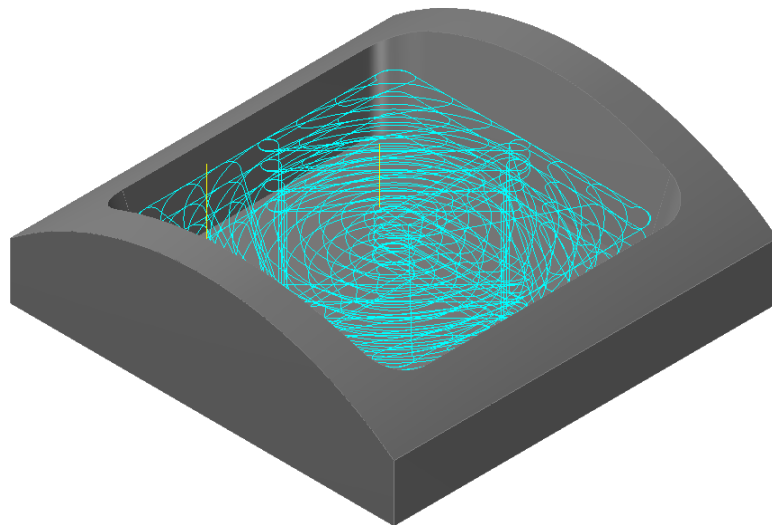
VoluMill can use an STL body as the stock to machine from. Use the Merge choice under the Files menu to add the STL file.

### ***Cavity 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.



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.



### ***Previous Operation***

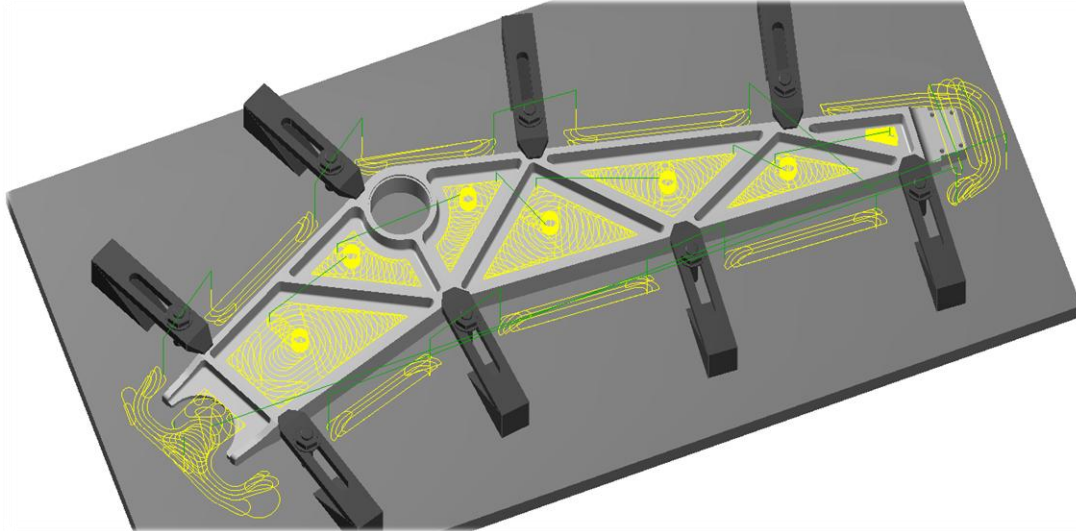
This option uses a chosen operation to determine the remaining material to machine.

### ***Part/Stock/Check Geom***

When editing an existing toolpath this option provides the ability to select a different part, stock, or check bodies to machine.

### ***Check Surf Clearance***

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



### ***Toolpath Direction***

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

### ***Cut Width***

Commonly known as the 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.

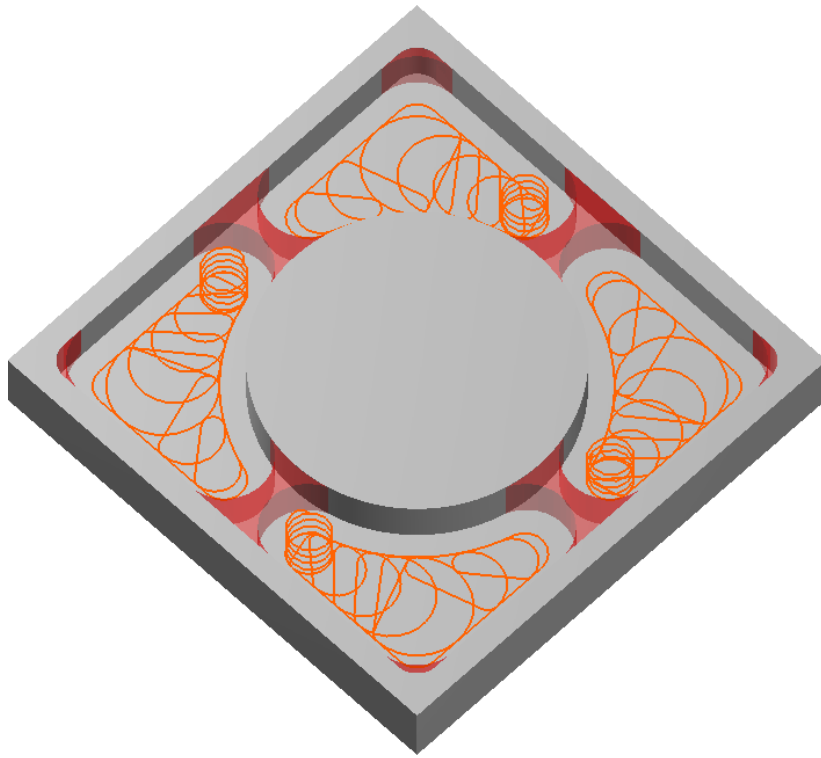
### ***Stock to leave***

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

### ***Smoothing Radius***

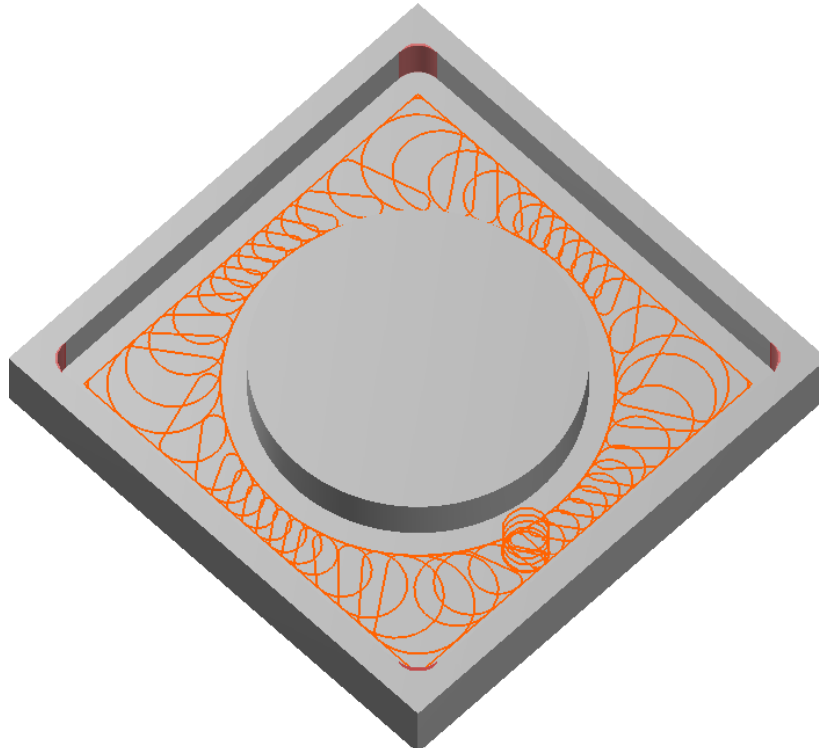
This is the minimum radius the tool will transverse when cutting. To machine into sharp corners or tight areas, VoluMill has to make small moves that may be more effectively accomplished using a smaller tool in a clean-up operation. The optimal, and default, Smoothing radius is 45% of the tool diameter. 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 tolerance used when machining surfaces.

### **Depth First**

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

### **Level First**

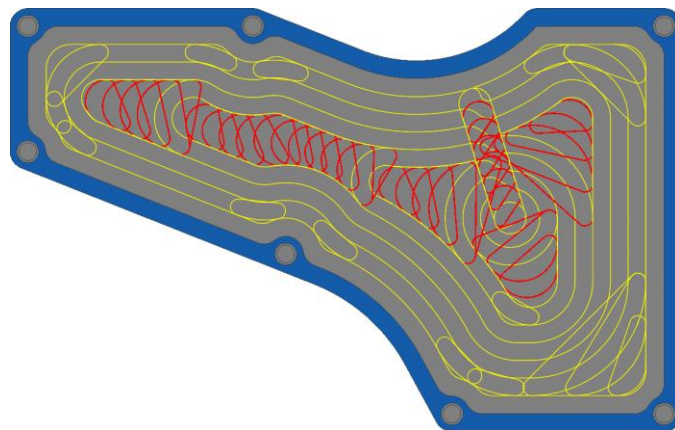
Each pocket will be machined to the Depth of Cut before the next one. This is helpful when machining pockets that have thin walls.

### **Side-Mill Only**

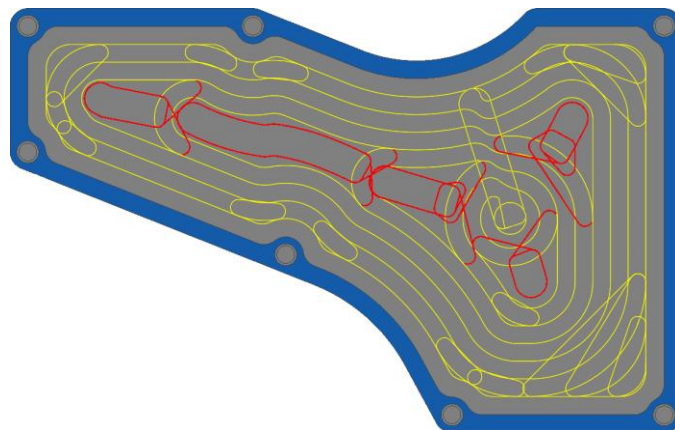
VoluMill achieves its superior cutting performance by striving to maintain a constant rate of material removal throughout the toolpath, regardless of the shape of the geometry.

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.

### ***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.

### ***Coordinate System:***

This lists the currently available coordinate systems. When creating a toolpath, VoluMill uses the current CS, or coordinate system defined at the top of the screen in the view toolbar. This list provides an opportunity to use a different CS if desired.



## Chapter 6

# 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:\Program Files\Celeritive Technologies\VoluMill Universal\Post*

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

*.\Post*

To open a post processor navigate to the *.\Post* 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 addressed 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 at 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.

**linearizeHelix = 0.001**

Breaks helices 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.

**lengthOffset**

The current tool length offset.

**workOffset**

The current tool length offset.

**dwll**

The current dwell value.

**tolerance**

The programmed tolerance.

**opName**

The name of the operation as listed in the Operation Manager.

**toolDescription**

The description of the tool. There are three descriptions:

- END\_MILL\_<tool diameter>
- BULL\_MILL\_<tool diameter>
- BALL\_MILL\_<tool diameter>

***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



## Chapter 7

# TROUBLESHOOTING

The following identifies solutions to problems you may encounter and attempts to answer frequently asked questions.

If your issue isn't addressed here, please refer to the Getting Help section in chapter 1, or contact [support@celeritive.com](mailto:support@celeritive.com).

- Once the VoluMill progress dialog is shown, it may take a few seconds to create the toolpath. Complex shapes may take a little longer. It is possible that the specified tool cannot fit into the selected geometry within VoluMill's motion requirements, which results in no toolpath being generated.
- Corners not completely machined? To avoid sharp directional changes and maintain smooth motion, VoluMill, by default, will not generate the sharp moves required to machine corners that are less than or equal to the tool radius. Reduce the Smoothing radius to make the tool machine farther into corners.
- Narrow areas uncut? To avoid sharp directional changes and maintain smooth motion, VoluMill uses a dynamically calculated minimum radius in the toolpath. This can result in material not being machined in areas where the tool can fit. Reduce the Smoothing radius to make the tool machine further into narrow areas. Please note, however, that if a smaller tool will subsequently be used, it may be more efficient to leave the Smoothing radius at a higher value and let the smaller tool remove the uncut material.

