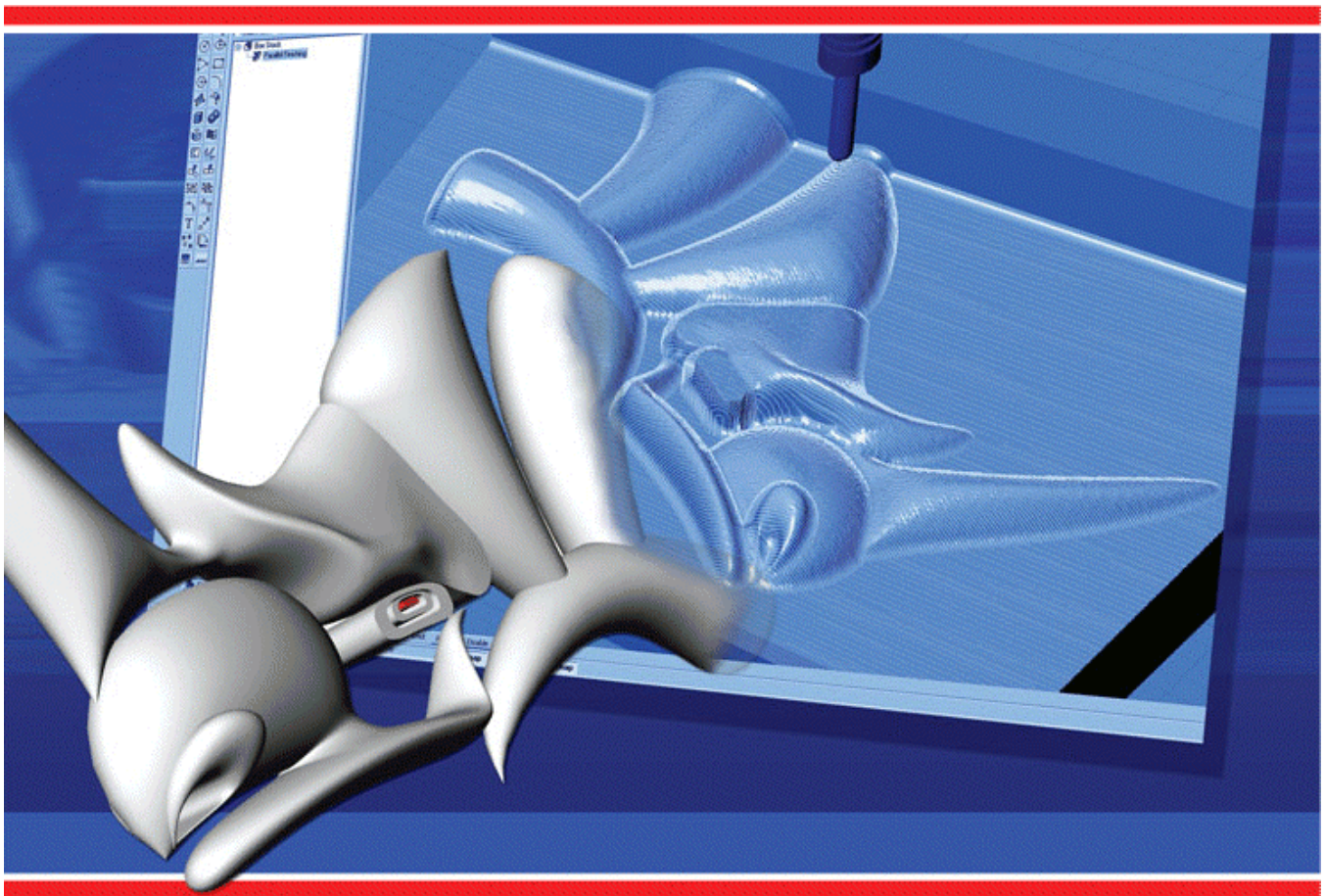


RhinoCAM®



Cutting edge manufacturing technology in Rhino



MecSoft Corporation

End-User Software License Agreement

This is the MecSoft Corporation's RhinoCAM End User Software License Agreement that accompanies the RhinoCAM® software product ("Software") and related documentation ("Documentation"). The term "Software" shall also include any upgrades, modified versions or updates of the Software licensed to you by MecSoft.

MecSoft Corporation grants to you a nonexclusive license to use the Software and Documentation, provided that you agree to the following:

1. USE OF THE SOFTWARE.

You may install the software on multiple computers. You may not have more than the number of legally purchased licenses of the software running concurrently at one time.

2. COPYRIGHT.

The Software is owned by MecSoft Corporation and its' suppliers. The Software's structure, organization and code are the valuable trade secrets of MecSoft Corporation and its suppliers. The Software is also protected by the United States Copyright Law and International Treaty provisions. You must treat the Software just as you would any other copyrighted material, such as a book. You may not copy the Software or the Documentation, except as set forth in the "Use of the Software" section. Any copies that you are permitted to make pursuant to this Agreement must contain the same copyright and other proprietary notices that appear on or in the Software. You agree not to modify, adapt, translate, reverse engineer, de-compile, disassemble or otherwise attempt to discover the source code of the Software. Trademarks shall be used in accordance with accepted trademark practice, including identification of trademark owner's name.

Trademarks can only be used to identify printed output produced by the Software. Such use of any trademark does not give you any rights of ownership in that trademark. Except as stated above, this Agreement does not grant you any intellectual property rights in the Software.

3. TRANSFER.

You may not rent, lease, sublicense or lend the Software or Documentation.

4. LIMITED WARRANTY.

MecSoft Corporation warrants to you that the Software will perform substantially in accordance with the Documentation for the thirty (30) day period following your receipt of the Software. To make a warranty claim, you must notify MecSoft Corporation within such thirty (30) day period. If the Software does not perform substantially in accordance with the Documentation, the entire and exclusive liability and remedy shall be limited to either the replacement of the Software or the refund of the license fee you paid for the Software.

MECSOFT CORPORATION AND ITS SUPPLIERS DO NOT AND CANNOT WARRANT THE PERFORMANCE OR RESULTS YOU MAY OBTAIN BY USING THE SOFTWARE. THE FOREGOING STATES THE SOLE AND EXCLUSIVE REMEDIES FOR MECOFT CORPORATION'S OR ITS SUPPLIERS' BREACH OF WARRANTY. EXCEPT FOR THE FOREGOING LIMITED WARRANTY, MECOFT CORPORATION AND ITS SUPPLIERS MAKE NO WARRANTIES, EXPRESS OR IMPLIED, AS TO THE NON-INFRINGEMENT OF THIRD PARTY RIGHTS, MERCHANTABILITY, OR FITNESS FOR ANY PARTICULAR PURPOSE. IN NO EVENT WILL MECOFT CORPORATION OR ITS SUPPLIERS BE LIABLE TO YOU FOR ANY CONSEQUENTIAL, INCIDENTAL OR SPECIAL DAMAGES, INCLUDING ANY LOST PROFITS OR LOST SAVINGS, EVEN IF A MECOFT CORPORATION REPRESENTATIVE

Getting Started with RhinoCAM

HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES OR FOR ANY CLAIM BY ANY THIRD PARTY.

Some states or jurisdictions do not allow the exclusion or limitation of incidental, consequential or special damages, or the exclusion of implied warranties or limitations on how long an implied warranty may last, so the above limitations may not apply to you. To the extent permissible, any implied warranties are limited to thirty (30) days. This warranty gives you specific legal rights. You may have other rights which vary from state to state or jurisdiction to jurisdiction. For further warranty information, please contact MecSoft Corporation's Customer Support.

5. GOVERNING LAW AND GOVERNING PROVISIONS.

This Agreement will be governed by the laws in force in the State of California excluding the application of its conflicts of law rules. This Agreement will not be governed by the United Nations Convention on Contracts for the International Sale of Goods, the application of which is expressly excluded. If any part of this Agreement is found void and unenforceable, it will not affect the validity of the balance of the Agreement, which shall remain valid and enforceable according to its terms. You agree that the Software will not be shipped, transferred or exported into any country or used in any manner prohibited by the United States Export Administration Act or any other export laws, restrictions or regulations. This Agreement shall automatically terminate upon failure by you to comply with its terms. This Agreement may only be modified in writing signed by an authorized officer of MecSoft Corporation.

6. U.S. GOVERNMENT RESTRICTED RIGHTS

Use, duplication, or disclosure by the government is subject to restrictions as set forth in subparagraph (c) (1) (ii) of The Rights in Technical Data and Computer Software clause at DFARS 252.227-7013 or subparagraphs (c) (1) and (2) of Commercial Computer Software – Restricted Rights at 48 CFR 52.227-19, as applicable. Manufacturer is: MecSoft Corporation, 18019 Sky Park Circle, Suite K, Irvine, CA – 92614-6386, USA.

Unpublished - rights reserved under the copyright laws of the United States.

MecSoft Corporation
18019 Sky Park Circle, Suite KL
Irvine, CA 92614-6386

RhinoCAM is a registered trademark of MecSoft Corporation
© 1998-2005, MecSoft Corporation

Trademark credits

Windows is a registered trademark of Microsoft Corporation
Pentium is a registered trademark of Intel Corporation
Parasolid is a registered trademark of Unigraphics Solutions
SolidWorks is a registered trademark of SolidWorks Corporation
Solid Edge is a registered trademark of Unigraphics Solutions
RhinoCAM is a registered trademark of MecSoft Corporation
Rhino is a registered trademark of McNeel & Associates.

Table of Contents

Welcome to RhinoCAM	6
About This Guide.....	6
RhinoCAM Configurations.....	6
System Requirements.....	6
RhinoCAM Setup.....	7
Step 1: Install Hardware Drivers (Required).....	8
Step 2: Install RhinoCAM (Required)	8
Step 3: Install Other Products (Optional).....	8
Step 4: Registering RhinoCAM (Required).....	9
Troubleshooting RhinoCAM	10
Troubleshooting the Software Installation	10
RhinoCAM Installation Folder.....	10
Troubleshooting the Hardware Security Key.....	11
Network Installation of RhinoCAM	12
RhinoCAM User Interface.....	13
RhinoCAM Menu Bar Item	14
RhinoCAM Browser Window	14
RhinoCAM Geometry Toolbar	15
RhinoCAM View Toolbar	15
Typical Scenario	17
Programming Workflow	17
Post-Processing	18
Machining Methods	19
2½ Axis Operations.....	19
3 Axis Operations.....	27
4th Axis Operations	38
3 Axis Drilling (Hole Making) Operations	41
Reverse Boring.....	42
4 Axis Drilling (Hole Making) Operations	42
5 th Axis Machining.....	43

Getting Started with RhinoCAM

Tutorial 1: 3 Axis Milling	44
Loading a Part Model.....	44
Browser.....	46
Setup Tab.....	46
Tools Tab.....	46
MOps Tab.....	47
Stock Tab.....	48
Creating the Stock.....	48
Creating Tools.....	51
Setting Feeds and Speeds.....	55
Creating the Horizontal Roughing Toolpath.....	57
Simulation.....	60
Simulating the Horizontal Roughing Toolpath.....	62
Creating the Pre-Finish Toolpath.....	63
Creating the Pencil Tracing Toolpath.....	66
Creating the Finish Toolpath.....	68
Post-Processing.....	70
Set Post Options.....	71
Post Processor Generator.....	72
Post-Processor Problems.....	75
Tutorial 2: 2½ Axis Milling Operations	76
Creating Tools and Stock.....	77
Regions.....	79
Multiple Regions.....	80
Creating Toolpaths.....	81
Clearance Plane.....	88
Tutorial 3: 3 Axis Drilling Operations	97
Regions in Drilling.....	98
Creating Tools.....	98
Creating Toolpaths.....	99
4th Axis Machining Operations	107
Simulation during 4Axis Machining.....	108

Tutorial 4: 4th Axis Roughing	110
Tutorial 5: 4th Axis Finishing Operation	115
Cutting Strategy	116
Tutorial 6 – 4th Axis Engraving	125
Tutorial 7 – 4th Axis Hole making	130
Tutorial 8: 5th Axis Machining Operations	136
Setting the Construction Plane and MCS.....	136
Machining the Top Face	139
Machining the Side Face.....	144
Machining the Front Face	148
Where to go for more help	151

Welcome to RhinoCAM

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

RhinoCAM is a unique CAM product plugin that runs inside of Rhinoceros 3.0. This fully integrated product seamlessly integrates Rhino's CAD functionality with toolpath generation and cutting simulation/verification, in one package that is both easy and fun to use. RhinoCAM's machining technology capabilities enable you to produce toolpaths that you can send to the machine with utmost confidence.

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

About This Guide

This guide is designed to introduce first-time users to RhinoCAM. The first part describes aspects of the user interface, machining strategies, and post processing. This is followed by several tutorials designed to familiarize you with the main features of RhinoCAM. All the models used in these tutorials can be located in the **Tutorials** folder under the RhinoCAM installation folder.

In addition to the information provided in this guide, see the context-sensitive online help for more comprehensive explanations.

RhinoCAM Configurations

RhinoCAM 1.0 comes in two configurations – RhinoCAM Basic and RhinoCAM Pro.

RhinoCAM Basic is a general-purpose machining program targeted at the typical machinist. It is ideal for the rapid-prototyping, general machining, hobby and educational markets. The RhinoCAM Pro product is an advanced version, suitable for mold, die and tool making, woodworking, and other complex applications. This configuration is suitable for machinists with sophisticated manufacturing requirements. Both configurations support 2½ and 3 axis milling, as well as drilling. The Pro version additionally supports 4th and 5th axis milling.

System Requirements

This section describes the minimum system requirements necessary for the proper functioning of RhinoCAM.

- Intel Pentium compatible computer
- Rhinoceros Version 3.0 Service Release 3.0 and above

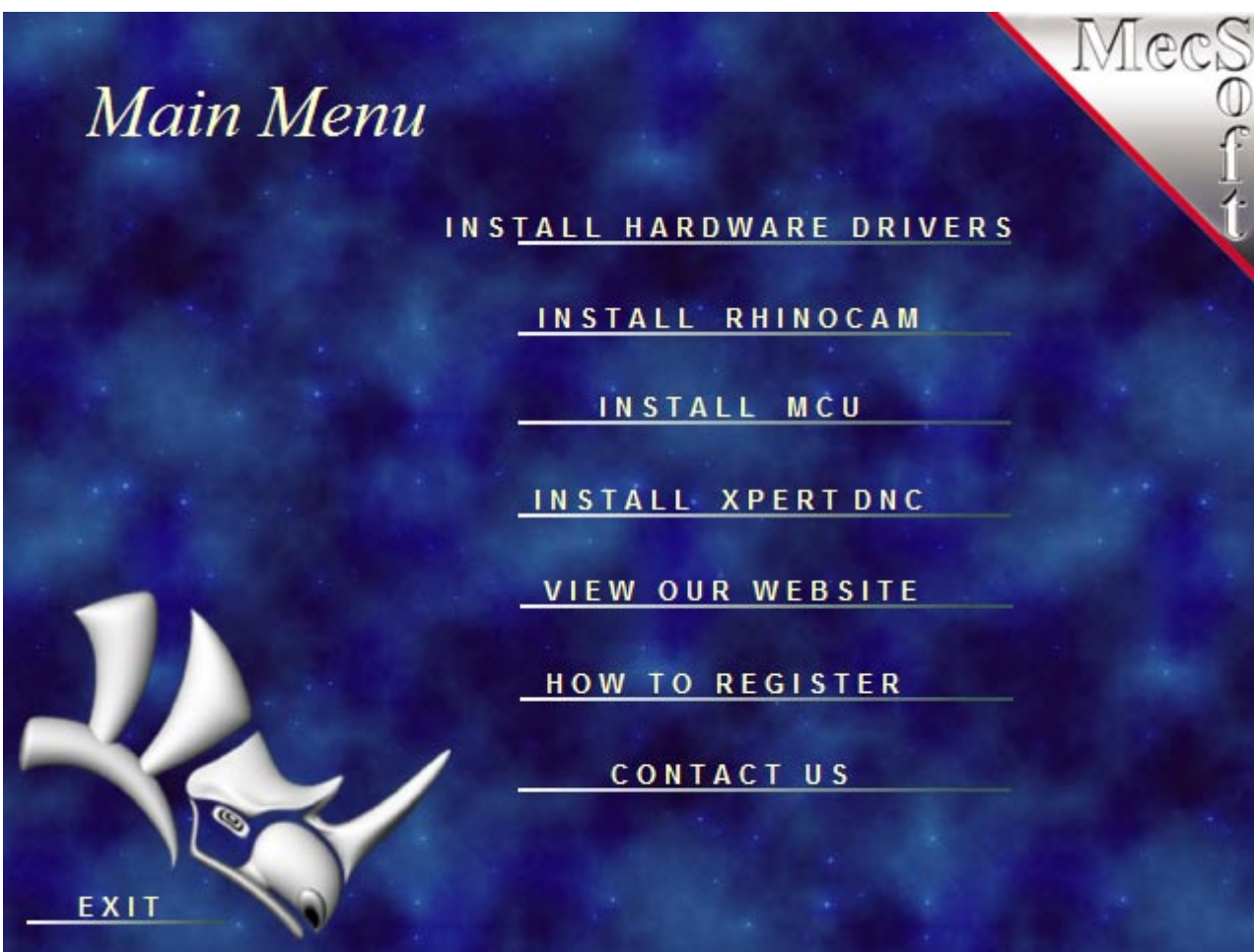
- Windows NT, 95, 98, 2000, ME, or XP with at least 64 MB RAM.
- OpenGL-compatible graphics card, displaying at least 64,000 colors
- Approximately 95 MB of hard disk space.

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

RhinoCAM Setup

To install RhinoCAM software, follow these instructions:

1. Insert the CD-ROM Disk into your computer's CD ROM drive.
2. The setup program will automatically launch once the computer detects the CD.
3. If the program is not automatically launched, browse the CD using the Windows Explorer program and double click on the **Launch** program found in the CD. This will launch the screen shown below:



Getting Started with RhinoCAM

Step 1: Install Hardware Drivers (Required)

RhinoCAM ships with a hardware security device called the security key (or “dongle”). This is either a 25-pin connector that connects to the parallel port of your computer, or a USB key that plugs into any USB port on your computer. You will have to install the drivers to allow RhinoCAM to communicate with this security device as the first step. Click on the **Install Hardware Drivers** button on the installation screen and follow instructions to install the drivers.

Note: Plug the hardware key into your computer only after you install the drivers for it.

Once you have installed the drivers you can attach the key to your computer.

- If you have a parallel port security key and if you have any other device, such as a printer, connected through the parallel port, disconnect the device(s) and connect the RhinoCAM security key to the port. Then reattach the connector of the original device(s) on top of the security key; the device(s) will continue to operate as before.
- If you have a USB port key, attach the key to any free USB port on your system

Make sure that the RhinoCAM hardware key is connected to the computer. RhinoCAM will not operate correctly if the security key is not connected to the computer!

Step 2: Install RhinoCAM (Required)

Once you have installed the hardware key drivers and attached the key to your computer, you can install the RhinoCAM product by clicking on the Install RhinoCAM button on the main installation screen. Follow the instructions to complete the installation. The install program will not only install all the files necessary for the proper functioning of RhinoCAM but also will make registry modifications on your computer so that RhinoCAM will be launched automatically when Rhino is invoked.

Note: Make sure you have privileges to modify the system registry before you install RhinoCAM.

Step 3: Install Other Products (Optional)

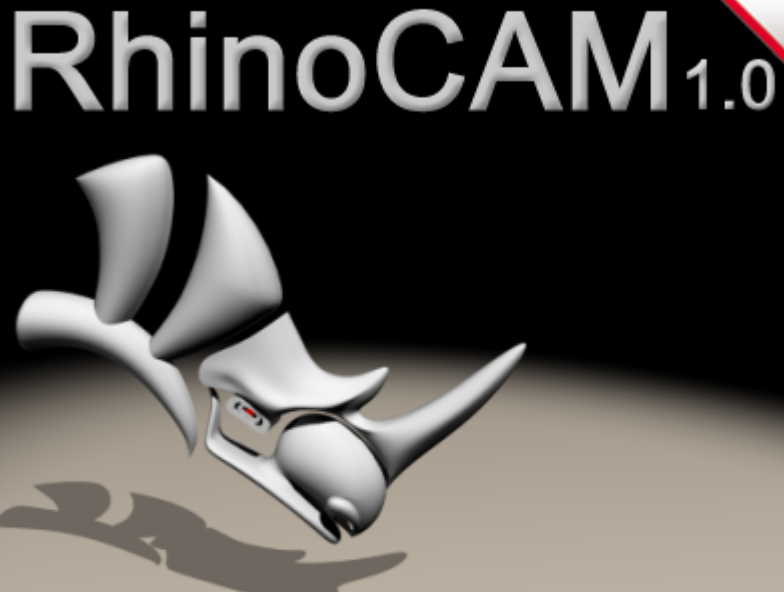
Once you have installed RhinoCAM you can optionally install MCU and/or Xpert DNC. These are two third party products that are included with RhinoCAM.

- The MCU or Meta Cut Utilities product is a back-plot viewer that allows the user of RhinoCAM to view the generated G-code graphically. This can be useful in making sure the posted output is correct before sending it to the machine tool.
- The Xpert DNC product is a single port DNC product is a communication program that allows you to send G-code files via DNC or Direct Numerical Control from your computer to the controller of the machine tool.

Step 4: Registering RhinoCAM (Required)

Upon successful installation, you can run the full RhinoCAM version 50 times or for 30 days without registering the product. After this period, RhinoCAM will not operate anymore. RhinoCAM needs to be registered with MecSoft and valid license codes obtained before it can become operable again.

To register RhinoCAM, launch **Rhinoceros 3.0**. To launch Rhino, click on the Windows **Start** button and select **Programs**. Point to the program group containing **Rhinoceros 3.0**. The name of this program group will usually be called **Rhinoceros 3.0**, unless you specified otherwise during setup. Once you locate the program group, select it and then select **Rhinoceros and launch Rhino**. Once Rhino comes up correctly, it will automatically load the RhinoCAM plug-in and display the RhinoCAM browser window as well as the RhinoCAM menu bar entry in Rhinoceros. Once RhinoCAM is loaded and ready, you will see the **Enter License Codes** dialog shown below. The **Tries Left** field indicates the number of times you can run RhinoCAM before it starts operating in demo mode.

Enter License Codes	
	
License Type: <input type="text" value="Registered Copy"/>	Dongle ID: <input type="text" value="1067886367-1050627819001757-R"/>
Tries Left: <input type="text" value="0"/>	Code 0: <input type="text" value="15531482"/>
Days Left: <input type="text" value="0"/>	Code 1: <input type="text" value="1018441993"/>
Network Licenses Left: <input type="text"/>	
<input type="button" value="Request License Codes .."/> <input type="button" value="Register"/> <input type="button" value="Cancel"/> <input type="button" value="Help"/>	

Note: This registration dialog can also be invoked from the RhnoCAM item in the Rhino menu bar.

Getting Started with RhinoCAM

To obtain license codes you must register the product using the Web form available at www.rhinocam.com. You can automatically launch this web form by selecting the **Request License Codes..** button in the dialog. If you have purchased the product directly from MecSoft Corporation, you will have to provide the purchase invoice number before you can be licensed. If you have purchased the product through an authorized RhinoCAM reseller, please obtain the license codes from your reseller. In addition to this information make sure you also provide the Dongle ID that is shown on the registration screen.

Troubleshooting RhinoCAM

If you have followed the installation steps outlined above correctly and are unable to load and run RhinoCAM correctly follow these troubleshooting steps to correct the problem.

Troubleshooting the Software Installation

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

RhinoCAM Installation Folder

RhinoCAM installation creates a main installation folder whose name and location you can specify during the installation process. If you accept the default location during the installation process the default location would be found under the **Plugins** folder under the Rhinoceros 3.0 main installation folder, typically **C:\Program Files\Rhinoceros 3.0\Plug-ins**. This folder contains the RhinoCAM executable and library files. There are also several subfolders in the installation directory:

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

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

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

Tutorials: Contains a tutorial and several part files to help first-time users get familiar with RhinoCAM. These are similar to the tutorials presented in this guide, in onscreen format. To launch these tutorials, open the **RhinoCAMTutorials.chm** file, and use the table of contents or arrows to browse through the steps.

WIN_9x and WIN_NT: Contains the necessary files for the proper functioning of the hardware security key (dongle).

MCU: Contains the necessary files for the proper functioning of a third party G-code analyzer tool, called **Meta Cut Utilities**, that is bundled with RhinoCAM.

Troubleshooting the Hardware Security Key

If you have installed the dongle but RhinoCAM is not running properly, try restarting your computer. If you are still having problems launching RhinoCAM it is probably due to the hardware drivers not being correctly installed to allow RhinoCAM to communicate with the security device.

1. For Windows NT / 2000 / XP, double-click on the **setupx86.exe** icon found in the **Win_NT** folder in the RhinoCAM installation folder (see the section below that describes the installation folder). For Windows 95 / 98 / ME, double-click on the **sentw9x.exe** icon in the **Win_9x** folder.
2. Select **Functions / Remove Sentinel** driver and click OK.
3. Double-click on the **RainbowInstaller.exe** icon in the RhinoCAM folder, and then click OK.
4. Restart your computer and launch Rhinoceros 3.0

If you are still having trouble launching RhinoCAM, you can run the program called the **Combo Installer** from the installation CD. Alternatively you can also download the **Combo Installer** from the Rainbow Corporation website (<http://www.safenet-inc.com/support/tech/sentinel.asp>) and run the installation program. This will automatically install the drivers necessary for the proper operation of your security key.

If the above method does not work, download the **Sentinel Medic** from the Rainbow website. Install it and go to Start->Programs->Rainbow Technologies->Sentinel Medic->Sentinel Medic. Click **Find SuperPro** and send the following information that appears on the screen to support@mecsoft.com, so that we can locate and fix your specific problem:

1. System Driver Info
2. Status
3. Description
4. Medic Says

Network Installation of RhinoCAM

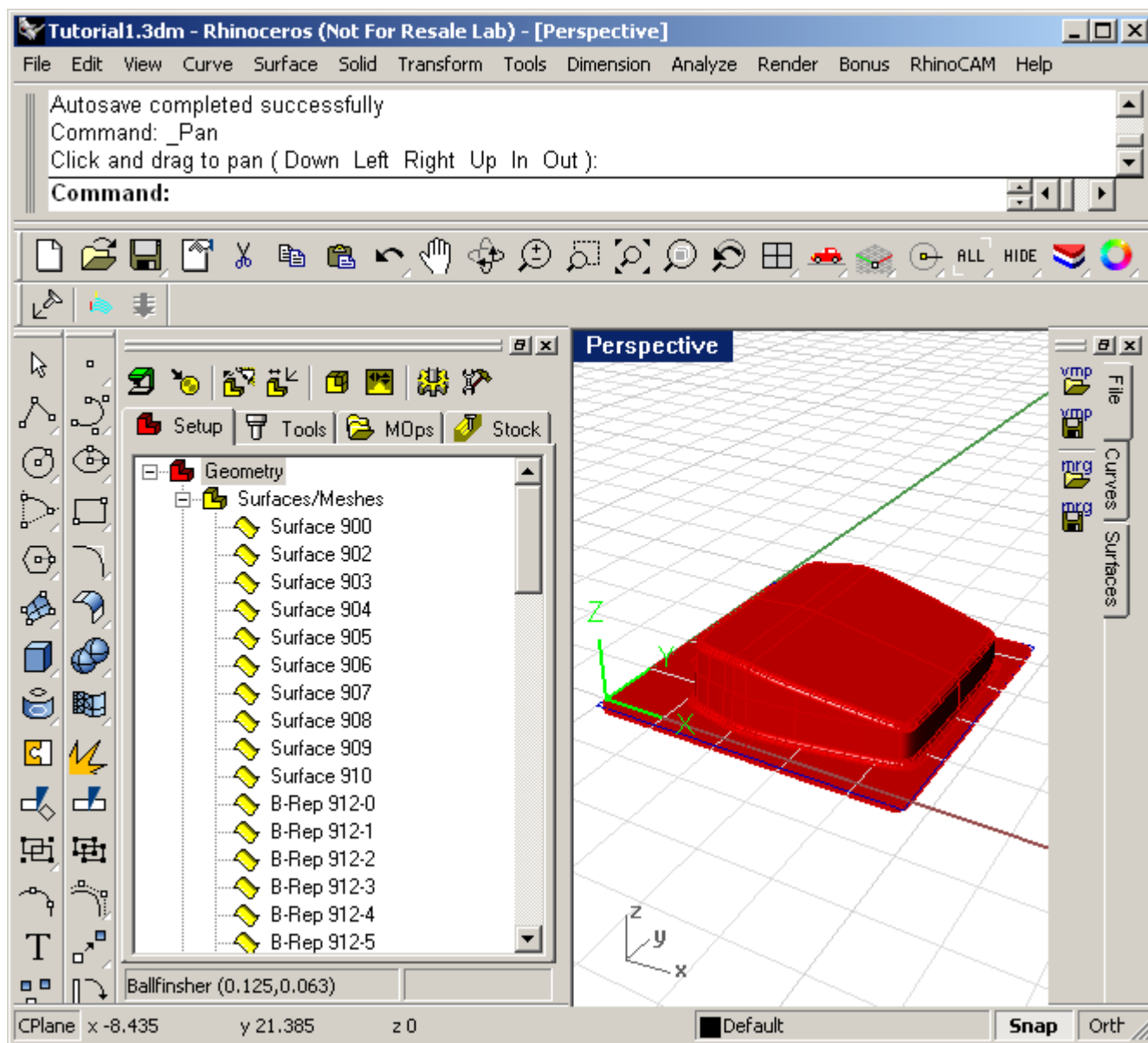
If you have purchased a multiple seat license of RhinoCAM your product will ship with a network enabled key that will allow you to run multiple concurrent licenses of the software using a single network key. To install the network key please follow these instructions:

- 1) Install the **RhinoCAM software** on the server machine as well as all the client machines connected to this server.
- 2) Install the **Rainbow hardware key drivers** on the server as well as all the client machines connected to this server.
- 3) Install the Key server installation program called **RainbowServerInstaller.exe** on the server. You will find this program in the RhinoCAM install folder, typically: C:\Program Files\Rhinoceros 3.0\Plug-ins\rhinoCAM 1.0
- 4) Set an environment variable, VMILL_LICENSE_HOST on each of the client machines to the servers' IP Address. This can be done as follows:
 - a. Go to **Start->Control Panel->System**
 - b. From the System Properties dialog box that pops up select the **Advanced** tab.
 - c. Click on the **Environment Variables** button at the bottom.
 - d. In the Environment Variables dialog box that pops up, click on the **New** button under **System variables**
 - e. In the New System Variable dialog box that pops up, define
Variable Name = VMILL_LICENSE_HOST
Variable Value = IP Address of the server machine (your system administrator should be able to provide you with this)
- 5) Now plug in the hardware key (parallel/USB) to the port.
- 6) To work across different subnets, please do the following in addition to the above instructions: Open the UDP port 6001 in any router installed on the network, this will allow the communication to go across to any subnets on the network.

Important: *Install the hardware key on your machine only after fully completing installation of all software.*

RhinoCAM User Interface

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



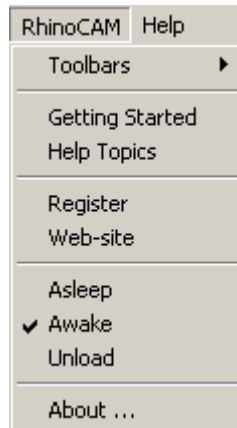
Getting Started with RhinoCAM

There are four main user interface objects that are created when RhinoCAM is loaded. These are the

1. RhinoCAM menu bar item in the Rhino menu-bar
2. RhinoCAM browser window
3. RhinoCAM Geometry Toolbar
4. RhinoCAM View Toolbar

RhinoCAM Menu Bar Item

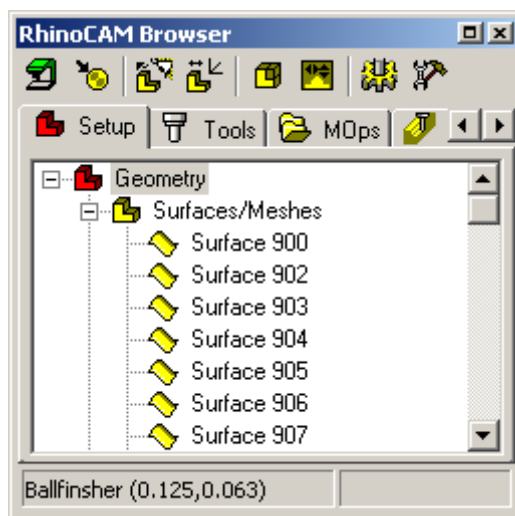
When RhinoCAM is invoked it will add a menu bar item, titled **RhinoCAM** to the main Rhino menu bar. Selecting this menu bar item will create a drop down menu as shown below.



RhinoCAM Browser Window

The Browser is a dock-able window that allows management of various entities or objects that can be created in RhinoCAM. By default, this window will appear docked on the left hand side of the Rhino display when RhinoCAM first comes up. This window can be undocked and move to different locations on the main screen.

This window has four main modes of operation represented by tabs at the top of the window. These are **Setup, Tool, MOps** and **Stock**. Selecting each of these tabs allows different views of objects in the RhinoCAM database. In addition each tabbed view also incorporates a context sensitive toolbar at the top. These toolbars are groups of functions that are associated with the type of object(s) in the tab.



RhinoCAM Geometry Toolbar

The Geometry toolbar is a dock-able toolbar that initially appears on the right hand side of the screen as shown in the screenshot picture. This toolbar is a tabbed toolbar that allows the selection of different tabs. Each tab selection will load and display a separate toolbar that can be used to create RhinoCAM related geometry such as parting planes and containment regions.



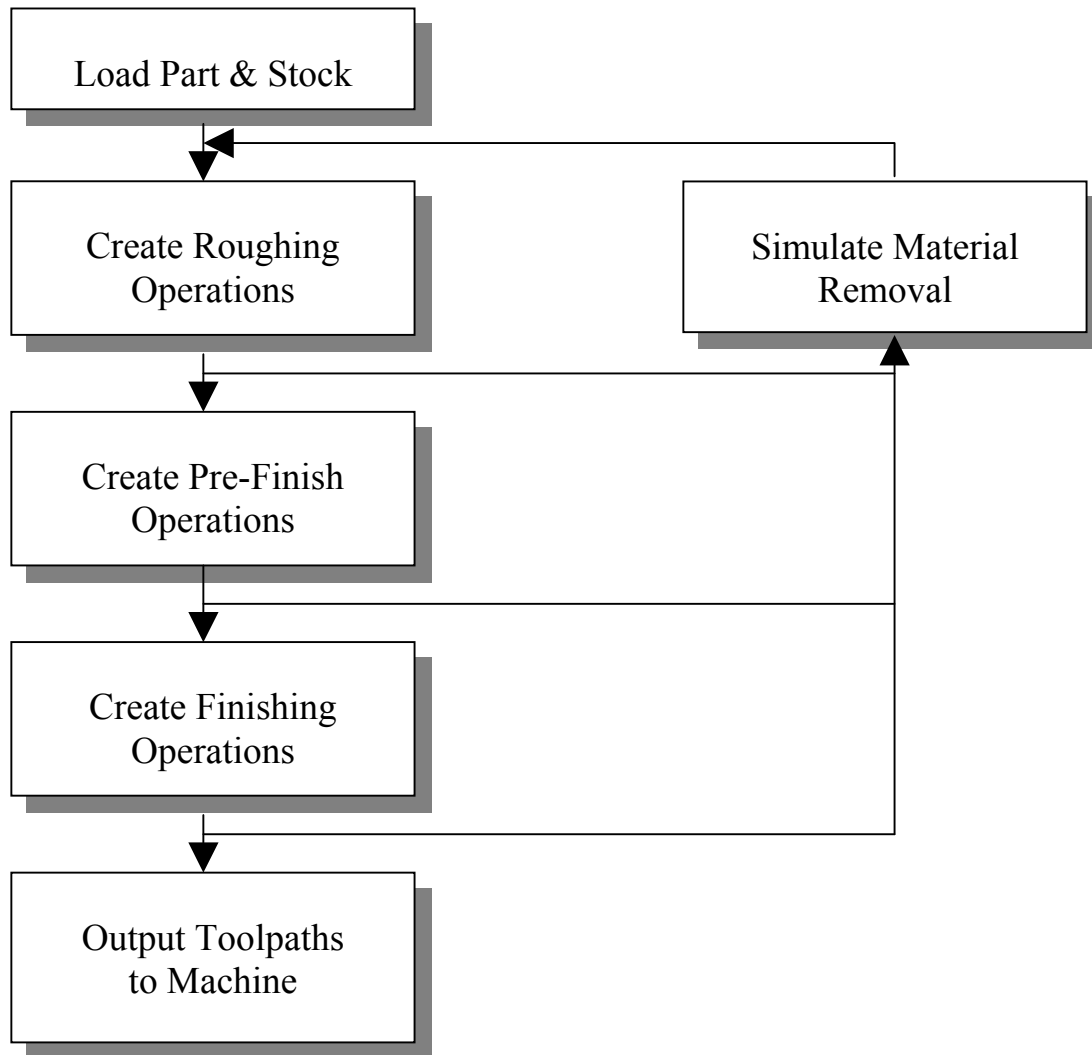
RhinoCAM View Toolbar

The View toolbar is a dock-able toolbar that is shown below. This toolbar allows the user to select RhinoCAM related viewing functions such as toggling the display of **the Machine Coordinate System** and the **Toolpath**.



RhinoCAM Workflow

The manufacturing process aims to successively reduce material from the stock model until it reaches the final shape of the designed part. To accomplish this, the typical machining strategy is to first use large tools to perform bulk removal from the stock (roughing operations), then use progressively smaller tools to remove smaller amounts of material (pre-finish operations). When the part has a uniform amount of stock remaining, a small tool is used to remove this uniform stock layer (finish operations).



This machining strategy is what you program using RhinoCAM. You can also simulate material removal to visualize how the stock model will look at any time during the process. This provides valuable feedback that can help you choose the most appropriate machining strategy.

Typical Scenario

Rough machining can be done by **Horizontal Roughing** (3 axis) or **Pocketing or Facing** (2½ axis) operations, using a relatively large flat end mill or an end mill with a corner radius. These rough operations can be followed by subsequent roughing operations, either using the same tool or a smaller tool.

The part can then be pre-finished by using **Parallel Finishing** or **Horizontal Finishing** (3 axis) or **Profiling** (2½ axis). Pre-finishing and finishing operations typically use ball end mills, with or without a side angle.

For complex 3D parts, additional finish operations may be necessary. For example, **Parallel Finishing** operation can be defined using a small ball tool with a fine step-over value. Additionally certain areas of the part geometry may need some special machining requirements. For such conditions, containment regions can be used to restrict the area of milling.

Once all of the operations are completed, you can go back and review the operation sequence, re-order and/or change operations if desired, simulate the material removal, and post-process the toolpaths. The Browser can be used to manage these operations.

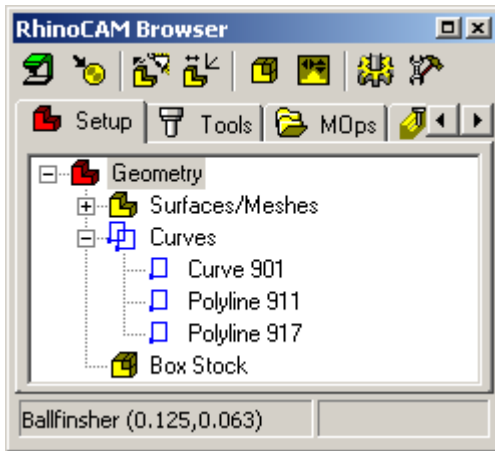
Programming Workflow

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

Thus the user typically would start with the **Setup** tab and access each of the buttons, optionally, in the toolbar that appears when this tab is selected in sequence from left to right. Once the setup functions are completed, the user will then proceed to the **Tools** tab to create, select and save tools to be used in the machining. After this the user will proceed to the **MOPs** or **Machining Operations** tab and commence programming the part. Once a program is completed the user can switch to the **Stock** tab to perform the material removal simulation and/or the tool animation to preview the toolpath before sending it to the machine tool.

Getting Started with RhinoCAM

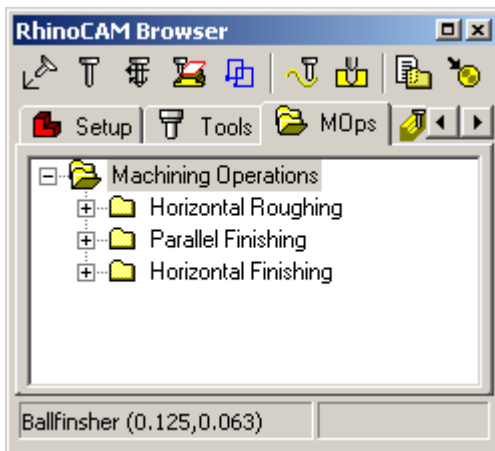
Step 1: Setup before programming



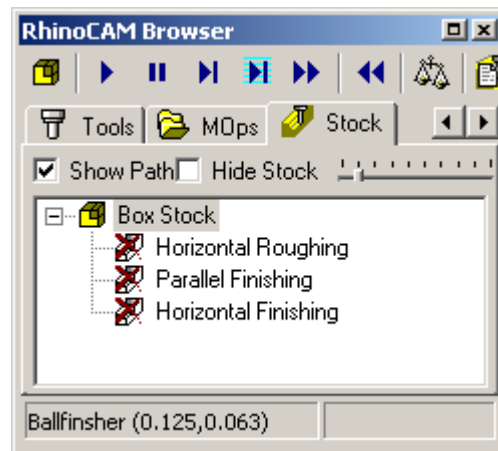
Step 2: Create, select and save tools



Step 3: Create machining operations



Step 4: Simulate machining operations



Post-Processing

Once the machining operations have been created and verified, they can be post processed to create G-code files. These G-code files can then be sent to the controller of the machine tool to drive the actual machine tool. RhinoCAM comes with a set of over one hundred post-processors to choose from.

Machining Methods

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

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

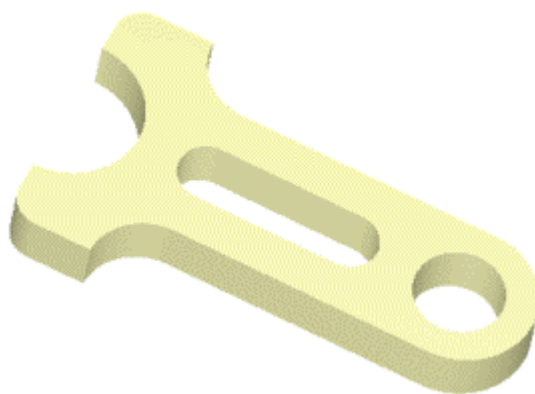
- **2½ Axis:** The tool can move in X and Y directions, while the Z level is fixed at set locations.
- **3 Axis:** The tool can move simultaneously in all three directions.
- **4th Axis:** Rotates the machine table in order to machine parts that cannot be machined with simple 2½ axis or 3 axis operations.
- **5th Axis:** The tool can rotate in all directions, so that areas of any orientation can be milled.

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

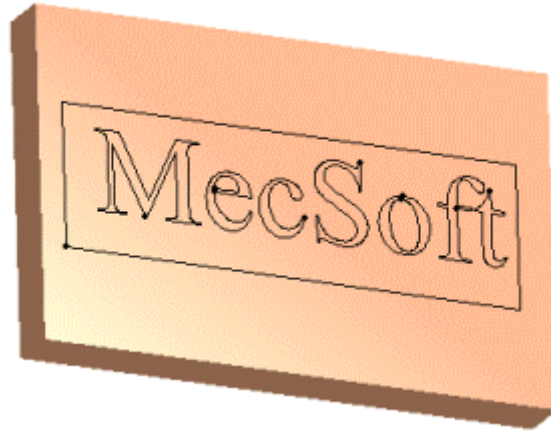
2½ Axis Operations

In 2½ axis toolpath, the tool can move in X and Y directions, but Z movements are limited to set levels. Because 2½ toolpaths do not relate to either part or stock geometry, machining regions must be selected; these define the boundaries of tool motions.

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



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



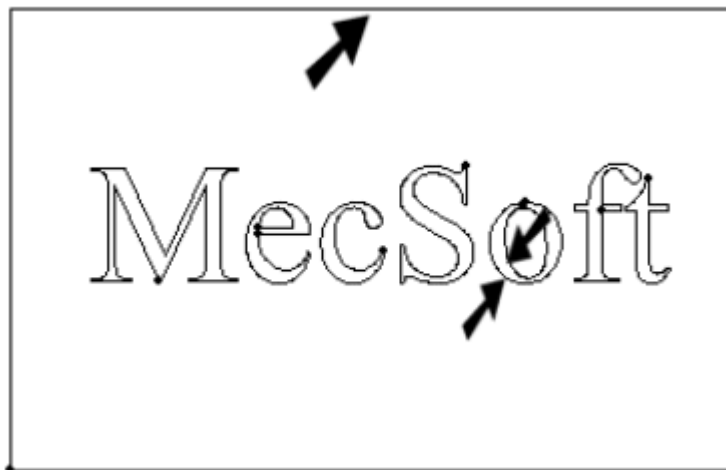
As with 3 axis operations, a typical machining operation would involve roughing then finishing. Unlike the more complex 3 axis parts, detailed finishing is typically not necessary for prismatic parts. Roughing is typically performed using a combination of **Facing** and **Pocketing** operations, and **Profiling** is used for finishing.

The available 2½ axis operations are shown and described below.

Facing

This method machines closed regions as if they were completely enclosing material to be removed. This means that the tool can approach the material from outside the outer regions, creating reverse pockets.

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

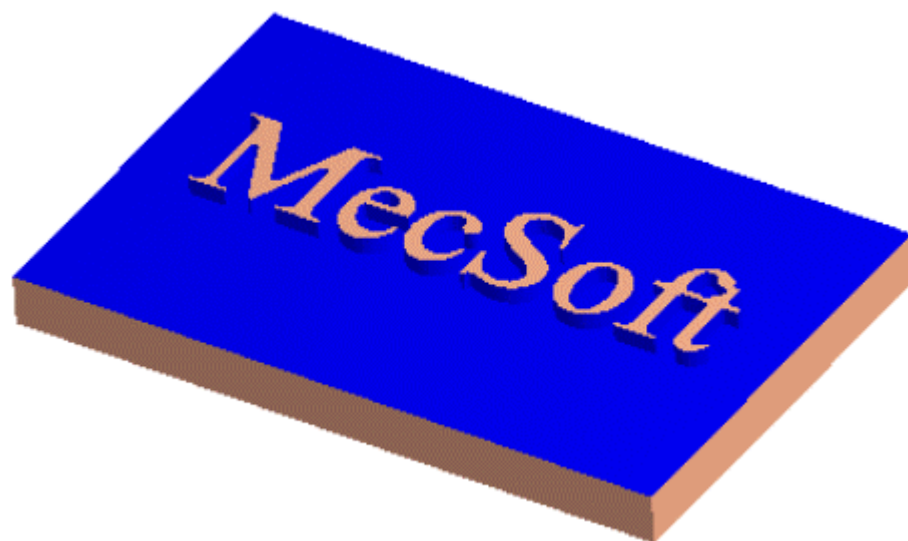


***Note:** The outer region should encompass the stock. You can easily create this type of region by selecting **Rectangle** command from the **Geometry** bar.*

The toolpath looks like the following:



The stock simulation:



Pocketing

This method machines closed regions as if they were pockets - completely enclosed by inner and outer regions. The tool cannot go beyond the outer region, and cannot go within inner regions. This is unlike **Facing**, in which the outermost region is considered to enclose material to be removed.

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



The toolpath looks like the following:



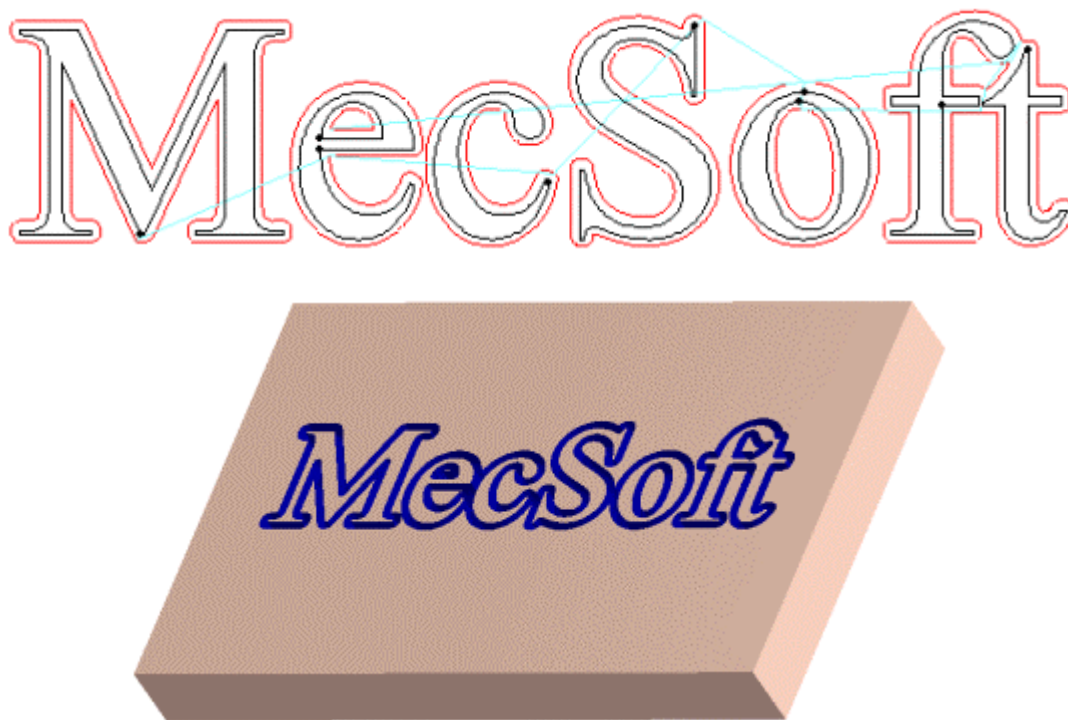
The stock simulation:



Profiling

This method machines open and closed regions by tracing along one side of their contours. You can define offsets so that the tool makes multiple passes relative to the regions.

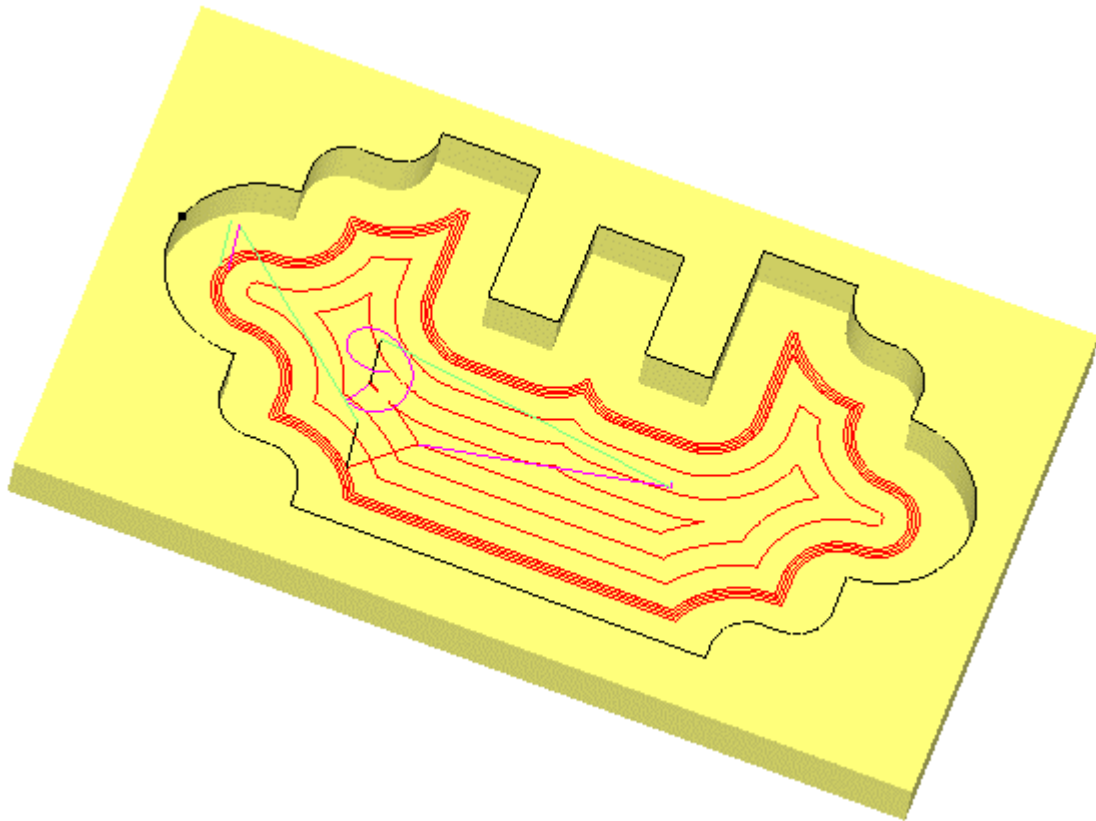
Profiling can be used as a finishing operation after a **Pocketing** or **Facing** toolpath, or it can be used alone.



Advanced Pocketing

Not available in RhinoCAM Basic

This method encompasses the functionality of both pocketing and profiling operations. You can rough (pocket) and finish (profile) in one operation. The stock left after roughing can be cleaned up by finishing, without having to input how much to cut while finishing.



Advanced Profiling

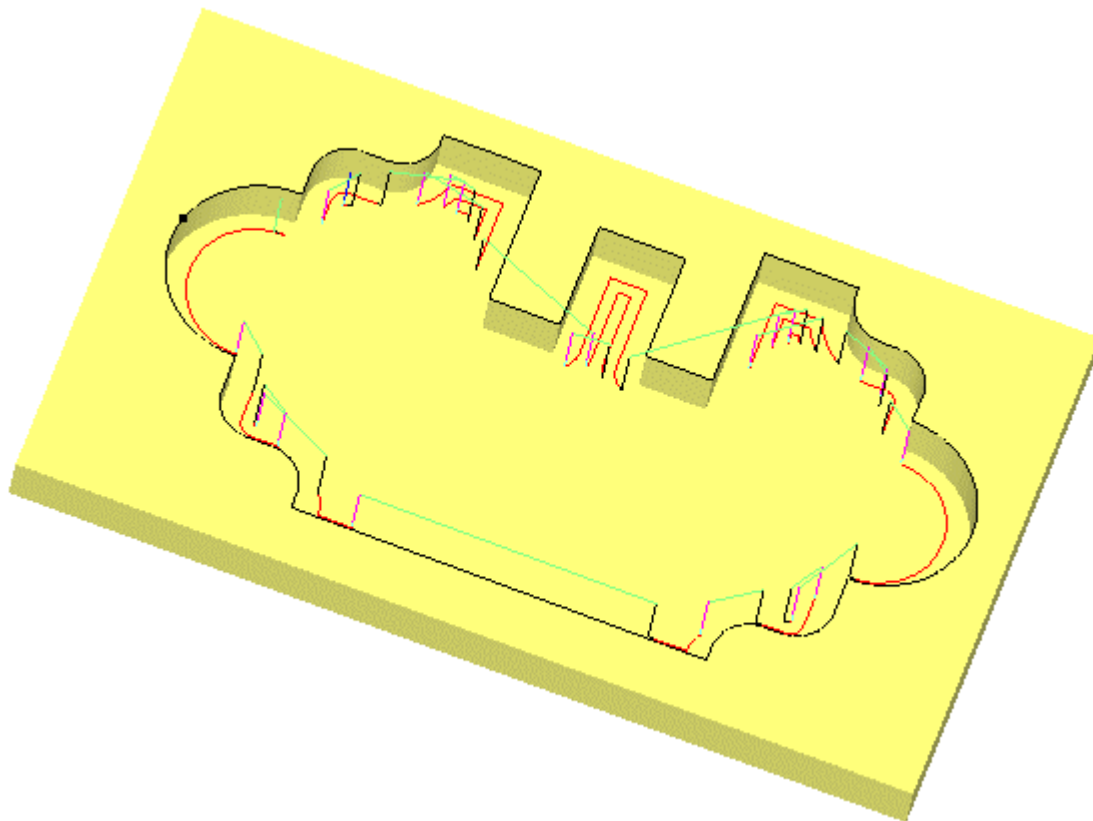
Not available in RhinoCAM Basic

This method is used when multiple profiling passes are needed at varying widths with different step over distances. The width can be divided into roughing (larger step over) and finishing (smaller step over) passes.

Re-machining

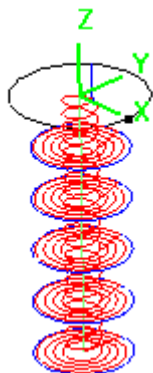
Not available in RhinoCAM Basic

This method uses a smaller tool to remove uncut material left after a previous operation.



Hole Pocketing

This method is used to cut large holes as a milling operation, rather than drilling. Engage can be specified as a helix with height and angle or pitch. After the helix engage, the hole is cut to the outer diameter using a spiral motion, followed by a circular motion to clean up the hole.



Thread Milling

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



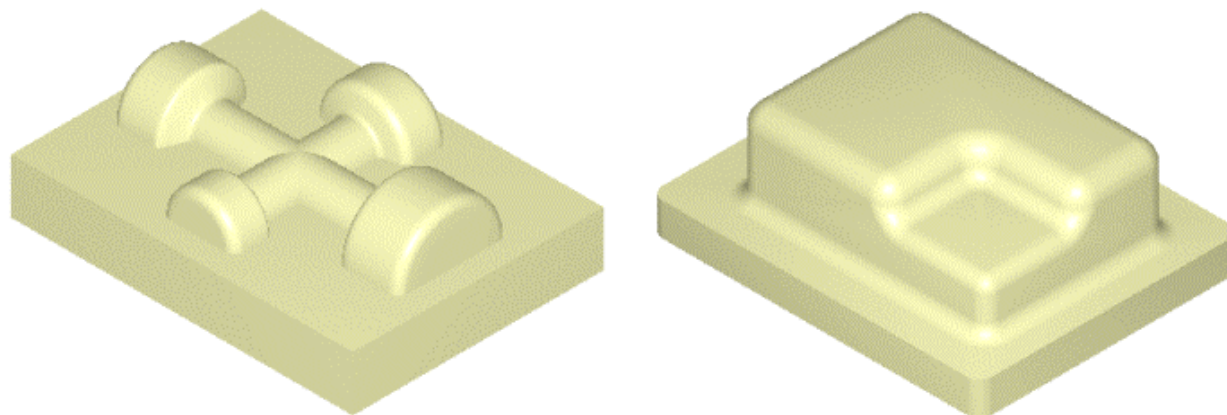
Engraving

Typically used for engraving text or logos on a finished model, this method machines open or closed, 2D or 3D regions by tracing along the contours. This method is similar to **Curve Machining**, in which the tool is projected onto the part surfaces below the regions being followed. A conical tool is used for engraving.



3 Axis Operations

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



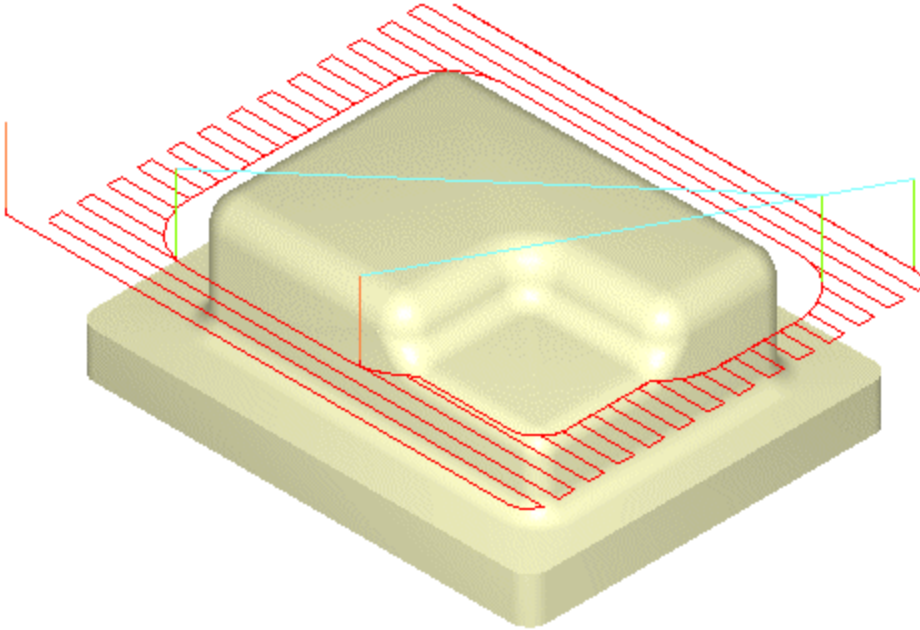
A typical machining scenario would be to first use **Horizontal Roughing**, the pre-finishing using **Parallel Finishing** and/or **Horizontal Finishing**. Once the part is at near net shape then fine detailed finishing could be performed by any of the re-machining or the region-based projection methods.

*Note: Only **Horizontal Roughing**, **Parallel Finishing** and **Horizontal Finishing** are available in RhinoCAM Basic.*

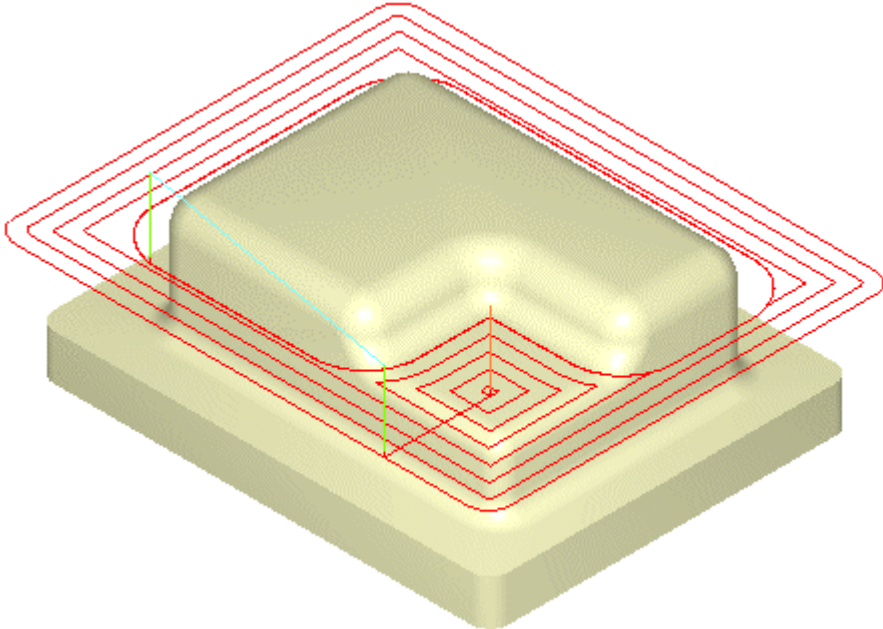
Horizontal Roughing

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

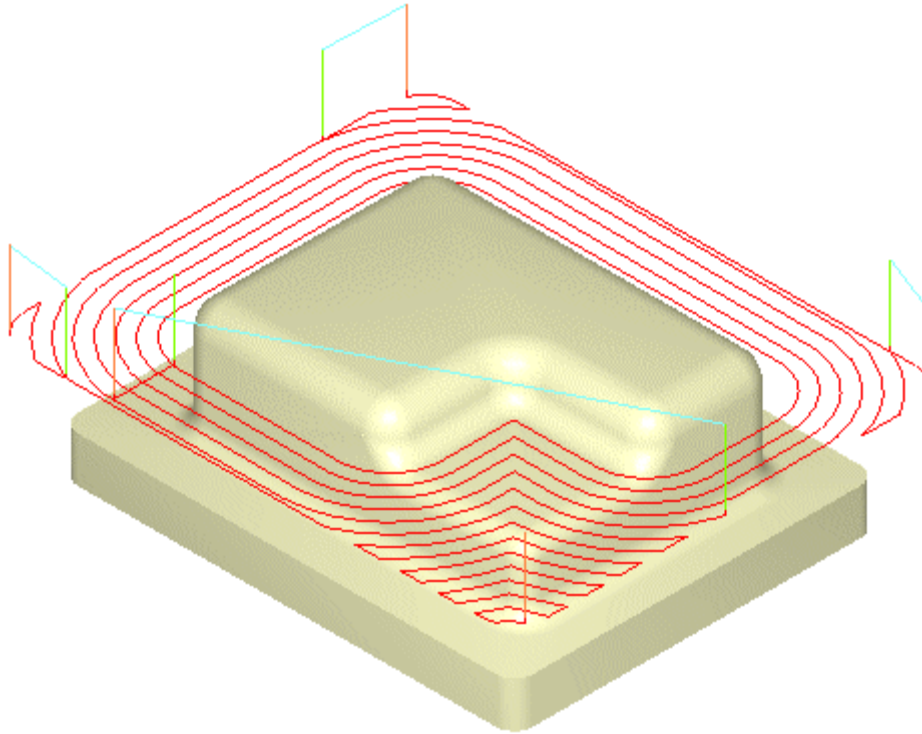
Both part and stock geometry are used to determine the regions that can be safely machined. Three types of cutting patterns are available: **Linear** (parallel, zigzag lines), **Stock Offset** (spiral pattern within stock and part), and **Part Offset** (spiral pattern outside the stock and outside the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



Horizontal Roughing – Linear



Horizontal Roughing – Stock Offset (Pocketing)

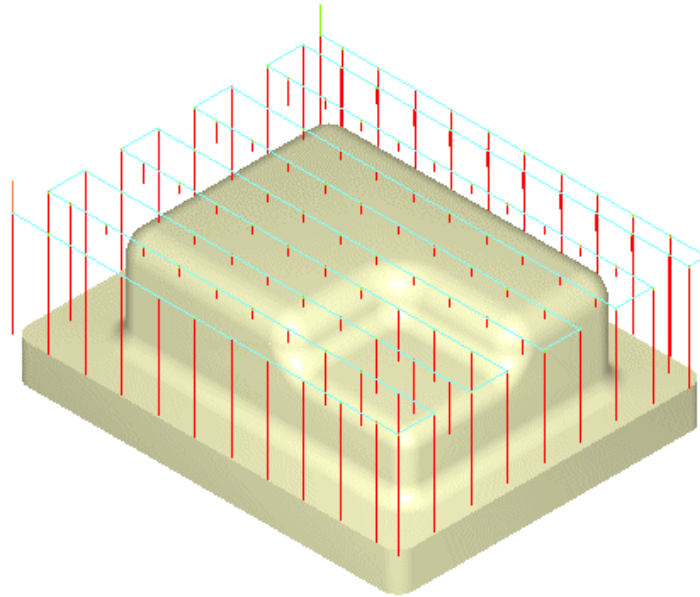


Horizontal Roughing – Part Offset (Facing)

Plunge Roughing

Not available in RhinoCAM Basic

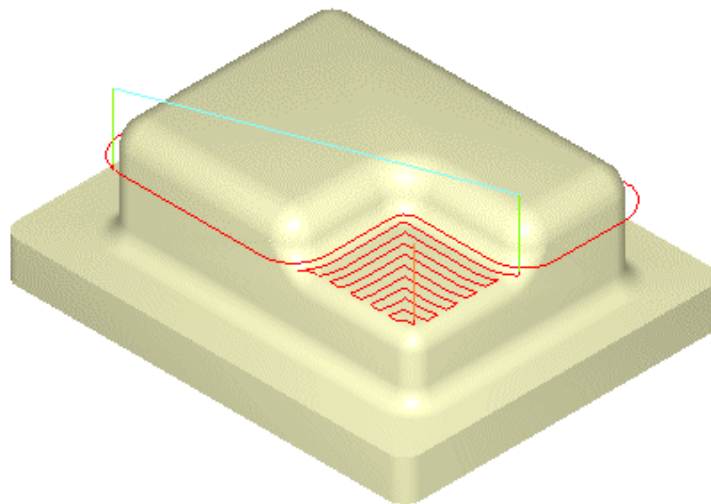
Sometimes called drill roughing, the tool can cut in the Z direction only, not in X and Y. The tool makes a series of overlapping plunges to remove cylindrical plugs of material.



Horizontal Re-roughing

Not available in RhinoCAM Basic

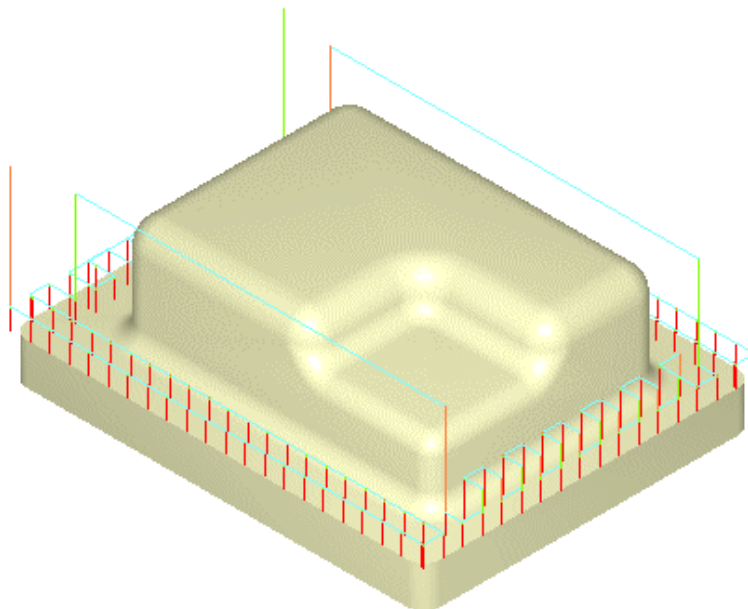
This is used to create toolpaths in areas that were not machined by previous operations. Unmachined areas are determined by comparing the part to the stock remaining after the previous operation. Machining is performed in constant Z levels, one of which is shown below:



Plunge Re-roughing

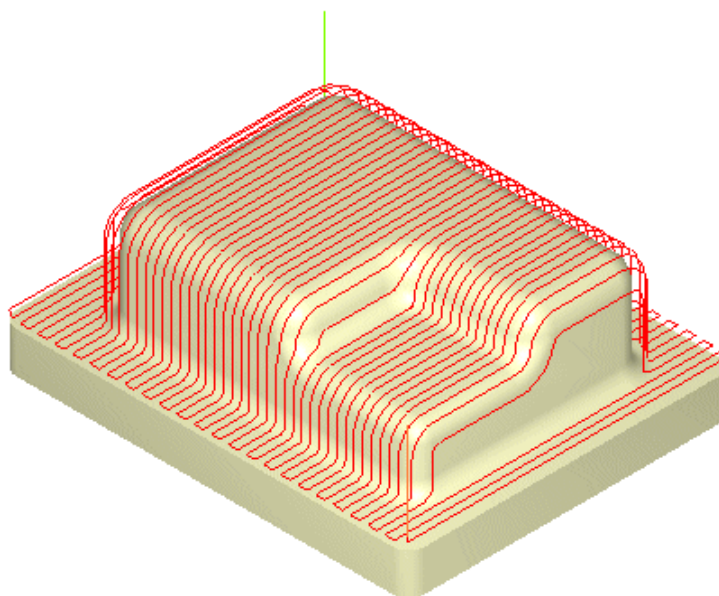
Not available in RhinoCAM Basic

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



Parallel Finishing

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

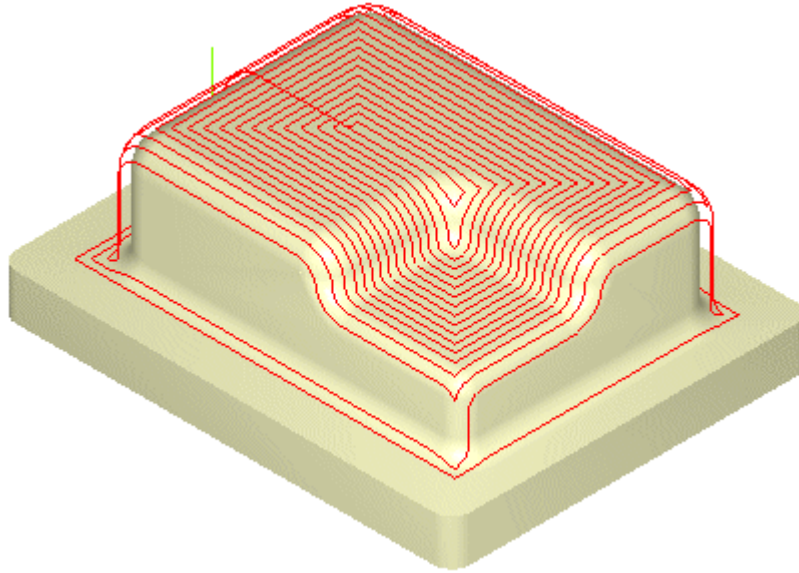


Getting Started with RhinoCAM

Pocket Finishing

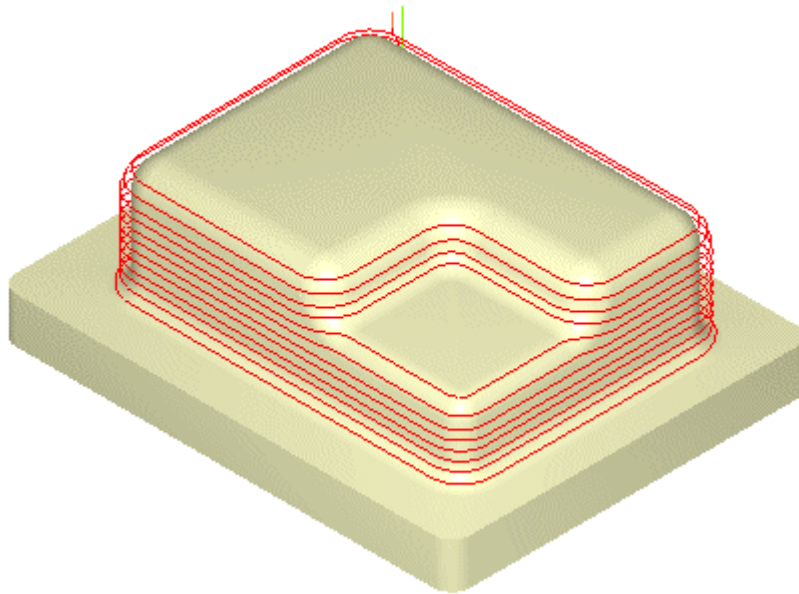
Not available in RhinoCAM Basic

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



Horizontal Finishing

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

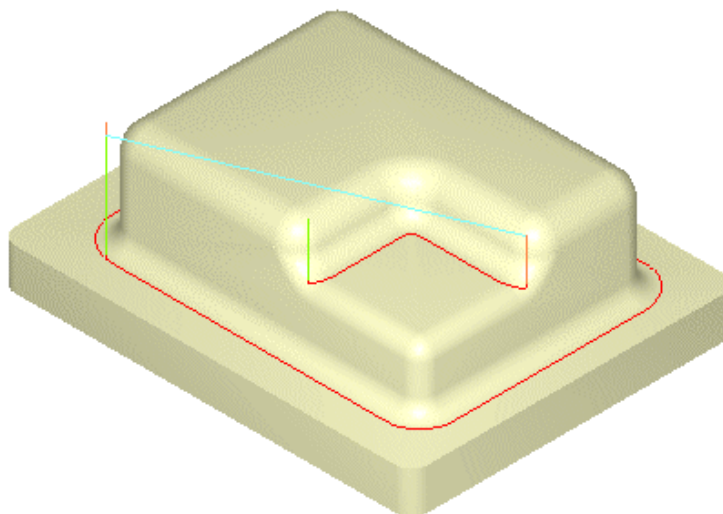


Pencil Tracing

Not available in RhinoCAM Basic

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

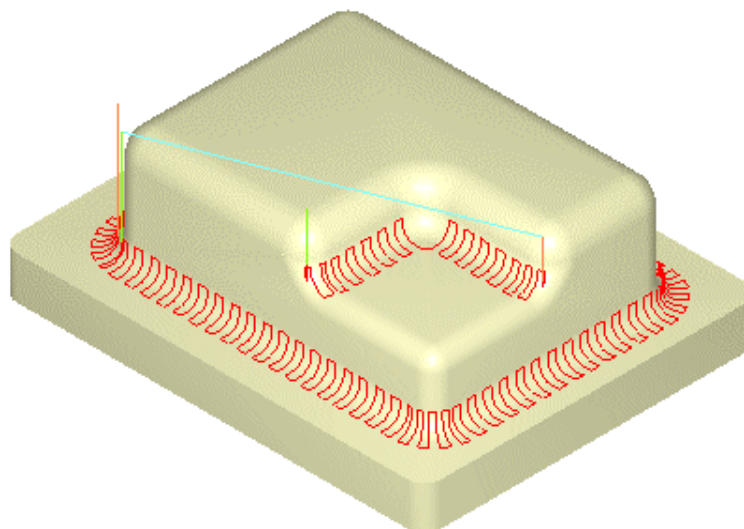
When used as a roughing operation, valleys and corners are relieved so that subsequent operations will not encounter large amounts of material in these regions, thereby reducing tool deflection and wear. When used as a cleanup operation, scallops that remain after finishing operations are removed.



Valley Re-machining

Not available in RhinoCAM Basic

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

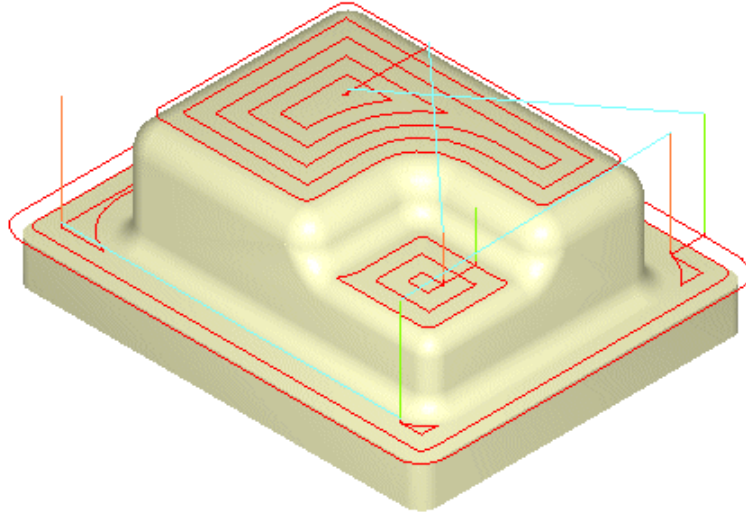


Getting Started with RhinoCAM

Plateau Machining

Not available in RhinoCAM Basic

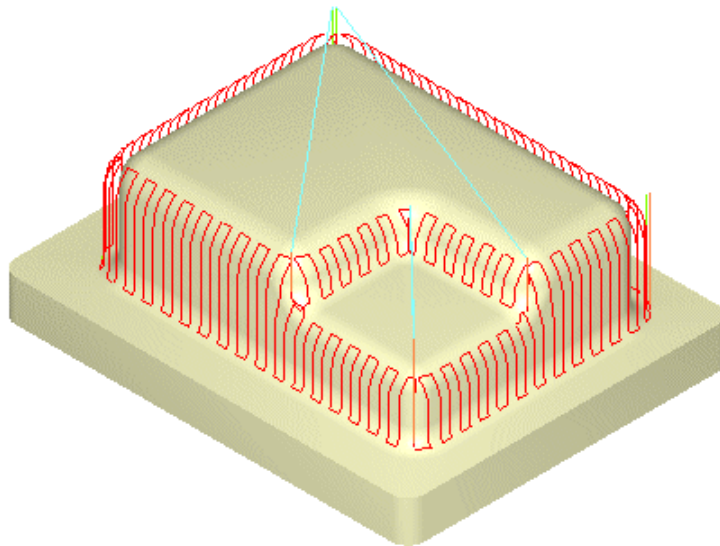
This method machines the tops of flat areas – areas that are within a specified angle from horizontal. This is typically used to re-machine areas that remain un-machined after a **Horizontal Roughing** or **Horizontal Finishing** toolpath.



Parallel Hill Machining

Not available in RhinoCAM Basic

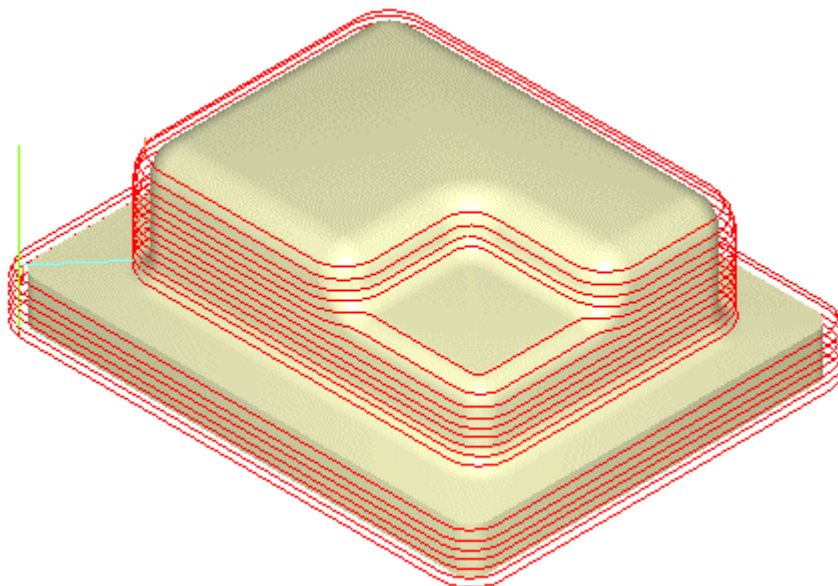
This method is used to machine steep areas. These are areas that are within a specified angle from vertical. This method is typically used when a **Parallel Finish** toolpath leaves scallops on steep areas. The cut angle is adjusted so that machining is always normal to the steep areas, thereby leaving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



Horizontal Hill Machining

Not available in RhinoCAM Basic

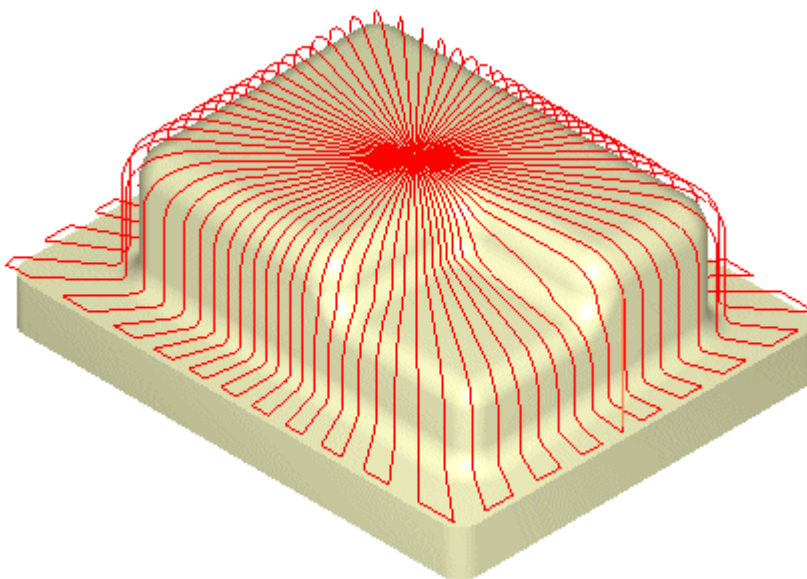
Similar to **Horizontal Finishing**, this method machines in constant Z levels. However, machining can be restricted only to areas in the part that are steeper than a user-defined steepness angle.



Radial Machining

Not available in RhinoCAM Basic

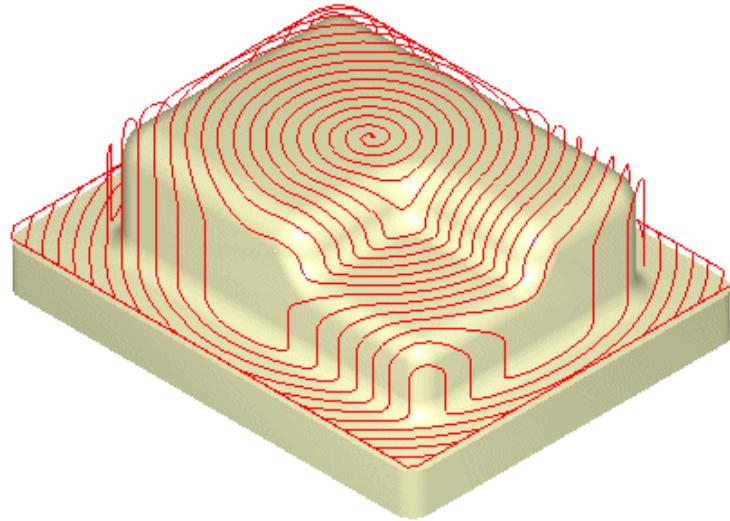
This method is used as a finishing operation for areas that have annular pockets. You must specify one or more machining regions; the tool moves radially from the centroid of these regions.



Spiral Machining

Not available in RhinoCAM Basic

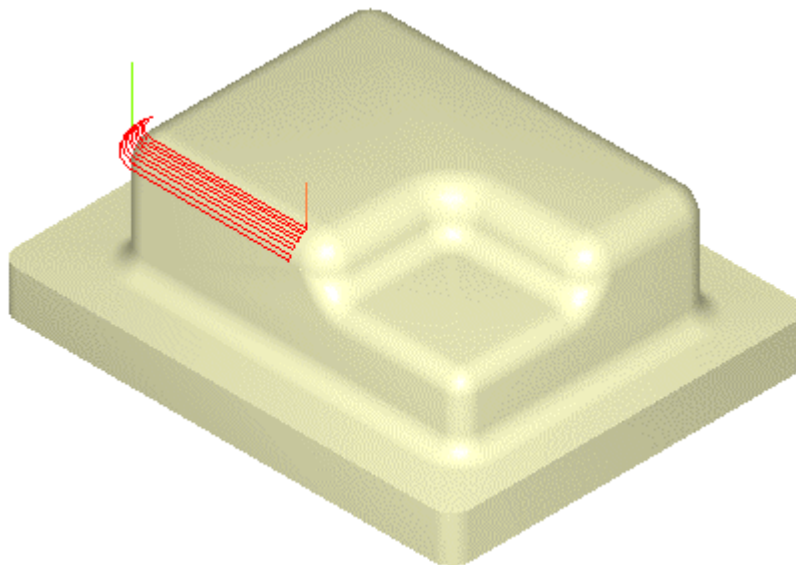
This method is used for finishing areas that have circular or near-circular characteristics, such as pocket bottoms. You must specify one or more machining regions; the tool moves in a spiral pattern based on the centroid of these regions.



Curve Machining

Not available in RhinoCAM Basic

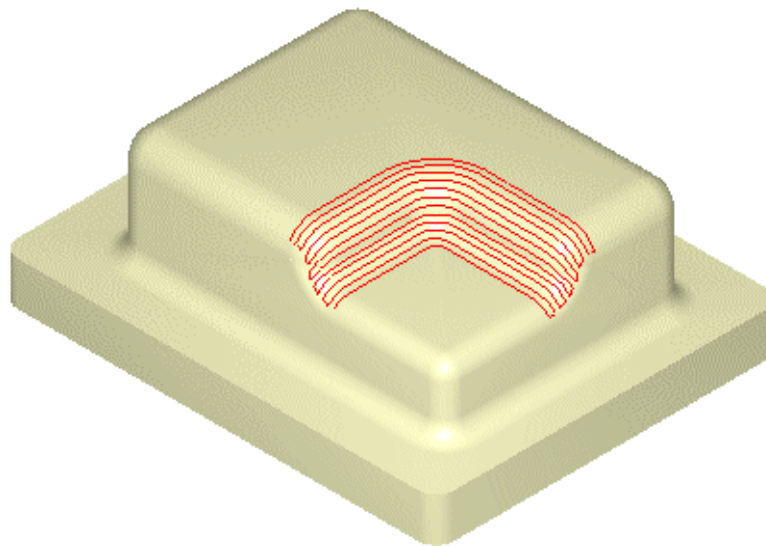
Suitable for machining isolated areas or shapes, this method machines along a curve. You must specify one or more machining regions, direction and cut pattern. The tool simultaneously follows the region and the contours of the part below.



Between 2 Curves Machining

Not available in RhinoCAM Basic

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



Reverse Post Milling

Not available in RhinoCAM Basic

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

4th Axis Operations

Not available in RhinoCAM Basic

4th Axis operations are used to machine parts that cannot be machined with simple 2½ axis or 3 axis operations. During 2½ and 3 axis milling, the tool remains vertical and the table is fixed. This means that areas of the part that cannot be accessed from above cannot be cut.

There are generally two ways to use 4th axis operations – indexed or continuous mode. The **Set Table Rotation Params** command enables you to set the rotation axis and origin for the table.

- **Indexed mode:** Use 2-1/2 or 3 axis operations to mill one half of the part, flip the table using a **Rotate Table** command, and then mill the other half. You are not restricted to 180-degree rotations.
- **Continuous mode:** The part is rotated continuously so that the tool is always normal to the axis of rotation. This is useful for machining multi-sided parts.

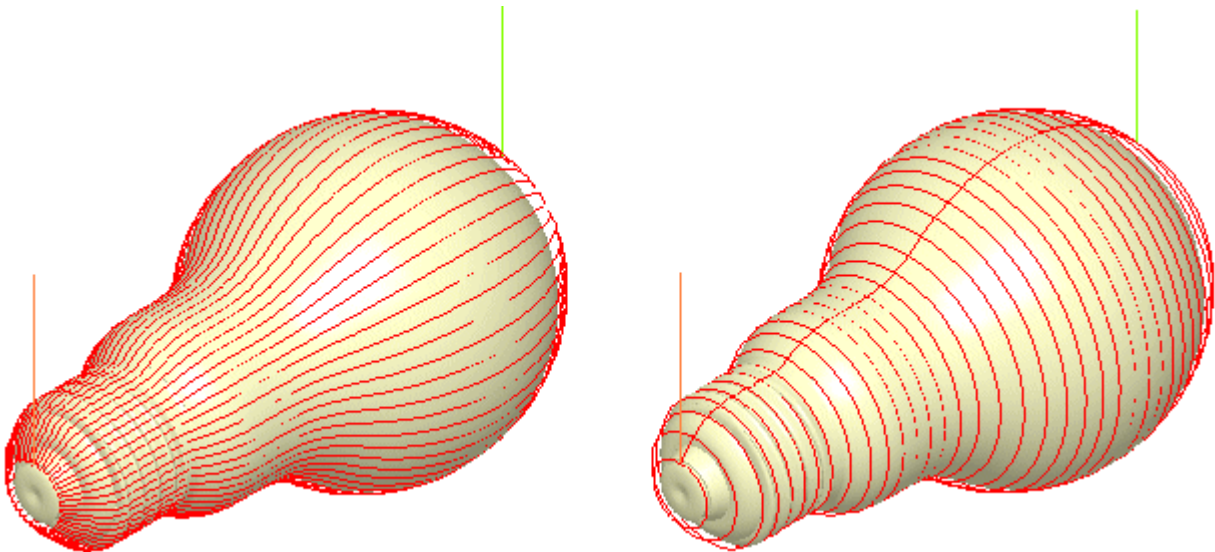
The available 4th axis operations are shown and described below.

Rotate Table

Rotates the table so that different sections of the part can be machined.

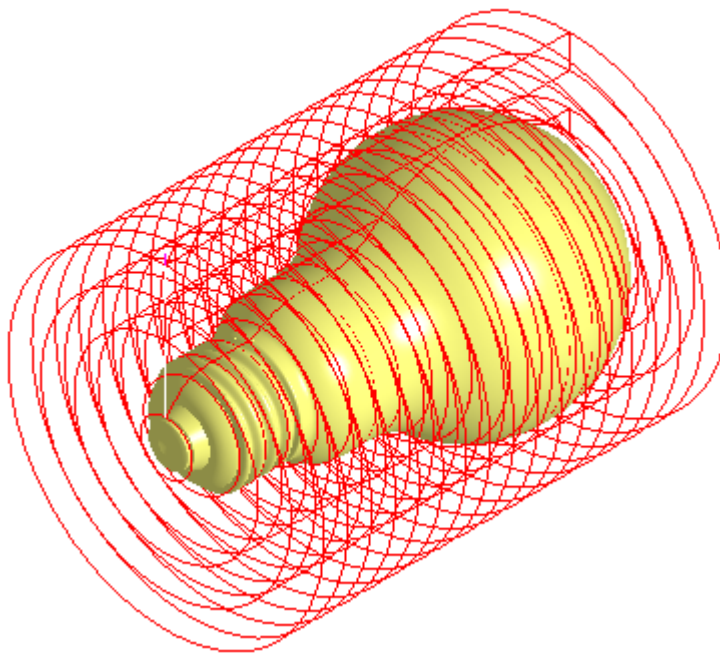
4th Axis Parallel Finishing

In this method, the tool is always normal to the axis of table rotation (continuous mode). The tool motions can be parallel to or normal to the rotation axis.



4th Axis Parallel Roughing

In this method, the finishing path is computed first and then successively offset about the axis of rotation to create multiple passes around the part. As with the finishing path the tool motions are always normal to the rotary axis and the tool motions can be either parallel or normal to the rotation axis.



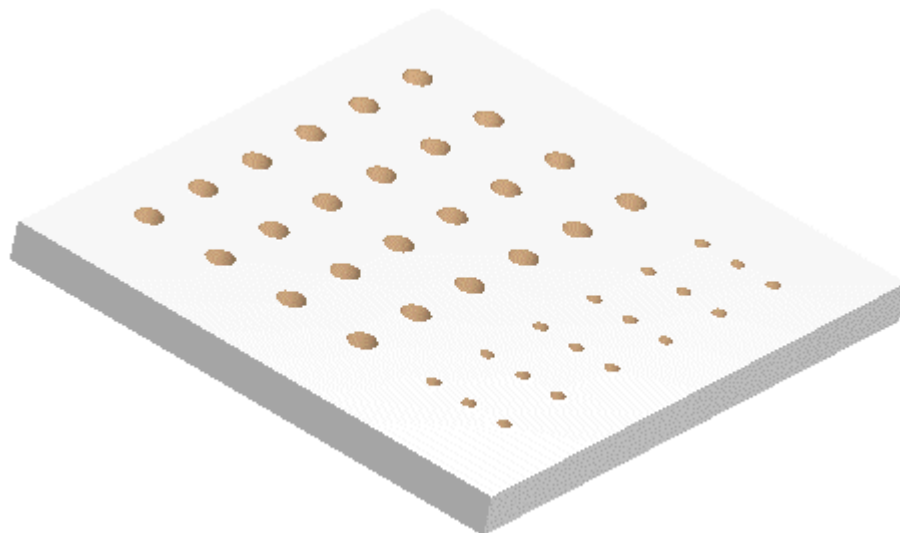
4th Axis Engraving

Similar to 3 axis engraving, this method machines text or logos by following the contours of the selected regions. Continuous mode is also used here.



3 Axis Drilling (Hole Making) Operations

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



The following drilling operations are available:

Drilling

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

Tapping

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

Boring

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

- **Drag:** The tool is fed to the specified depth at the controlled feed rate. Then the spindle is stopped and the tool retracts rapidly.

Getting Started with RhinoCAM

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

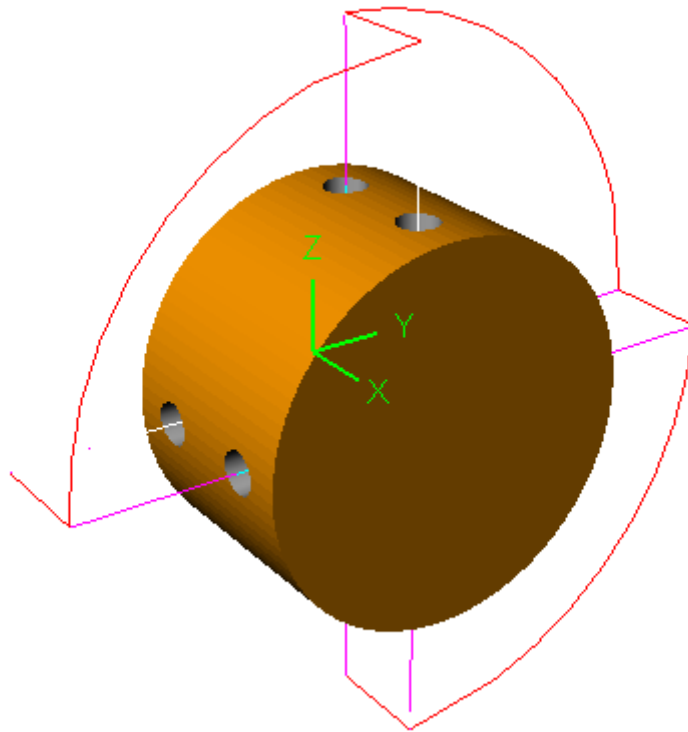
Reverse Boring

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

4 Axis Drilling (Hole Making) Operations

Not available in RhinoCAM Basic

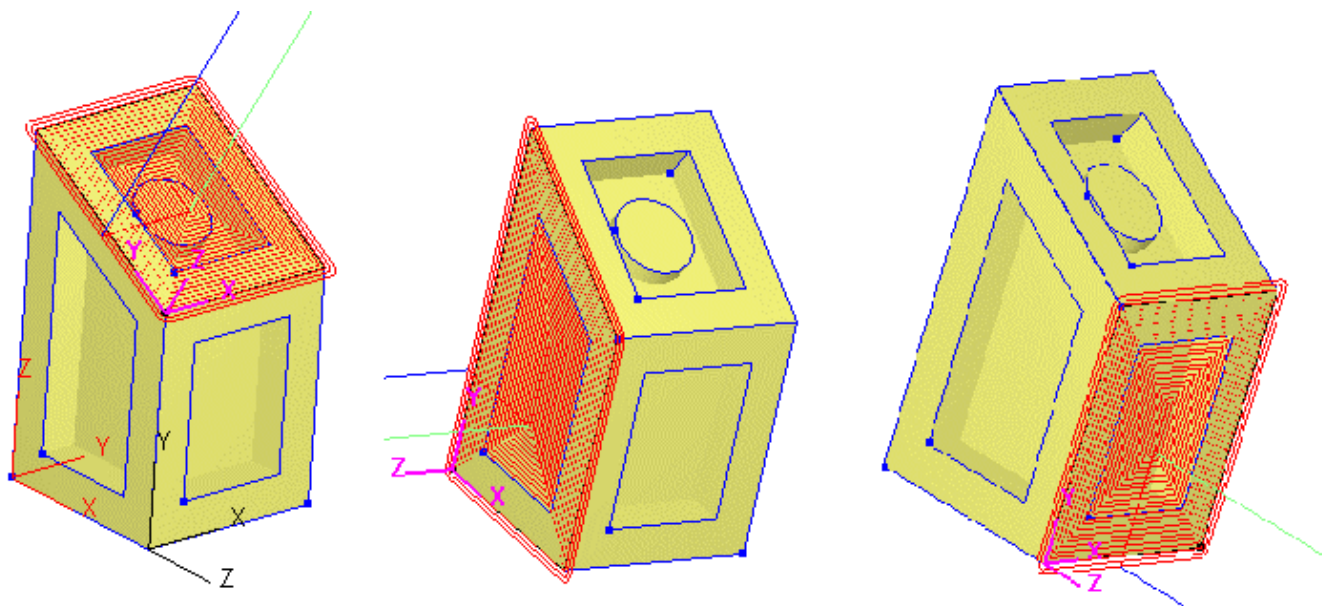
All of the above mentioned 3 axis drilling operations can be used in 4 axis mode as well. As with any 4th axis operation the tool motions are always normal to the rotary axis.



5th Axis Machining

Not available in RhinoCAM Basic

In 5th axis machining, the tool can rotate in all directions, so that areas of any orientation can be milled.



Tutorial 1: 3 Axis Milling


In this tutorial, you will create 3-axis milling toolpaths to manufacture a designed part.


The stepped instructions are accompanied by explanatory and introductory text. Reading this text will help you understand the tutorial methodology and provide information about additional options available. However, if you prefer to work straight through the steps without any additional reading, look for the following symbol:




Don't forget to save your work periodically! You may want to save the file under a different name so that the original file will be preserved.

Use the Rhinoceros menu bar or the Standard toolbar buttons to create, load and save part geometry.

 **New:** Creates a new file. The user can create a file that is in inches or millimeters units only. RhinoCAM will not be able to operate if a file of different units is created.

 **Open:** Loads part geometry into RhinoCAM. This geometry is typically imported from other CAD formats, but can be created from within RhinoCAM as well.

 **Save:** Saves the current file as a 3DM file. We recommend saving your work periodically and avoid losing data.

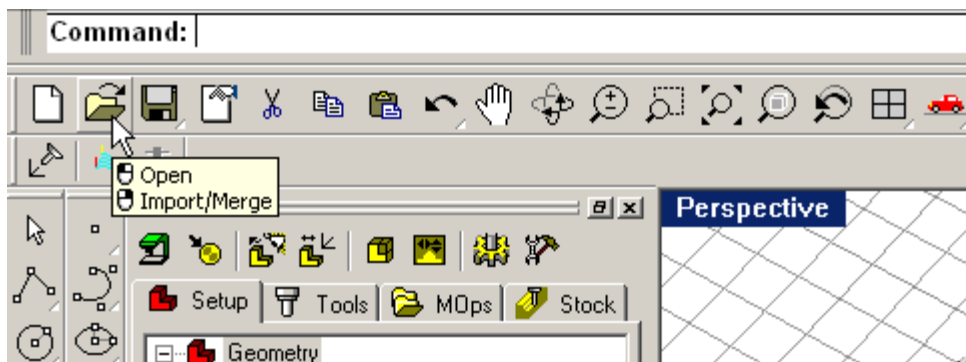
Loading a Part Model

“Part” refers to the geometry that represents the final manufactured product. You can create parts within Rhinoceros, but it is also possible to import geometry created in another CAD system.



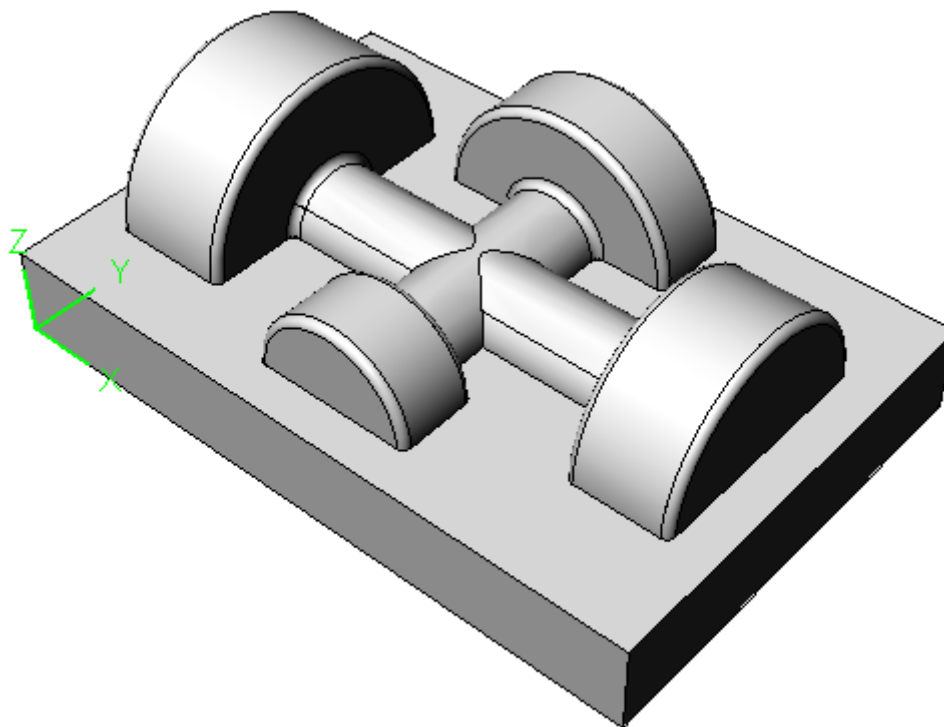
To load a part:

1. Select **File / Open**, or click the **Open Part File** icon from the Rhino standard toolbar.



2. From the **Open** dialog box, select the **Dumbbells.3dm** file from the **Tutorials** folder in the RhinoCAM installation folder.

The imported part appears as shown below.



Note: You must work in shaded mode in order for you to be able to visualize toolpaths and stock models created in RhinoCAM. It is suggested for best visual performance with RhinoCAM to work with only one view port open and the view port operating in shaded mode.

Getting Started with RhinoCAM

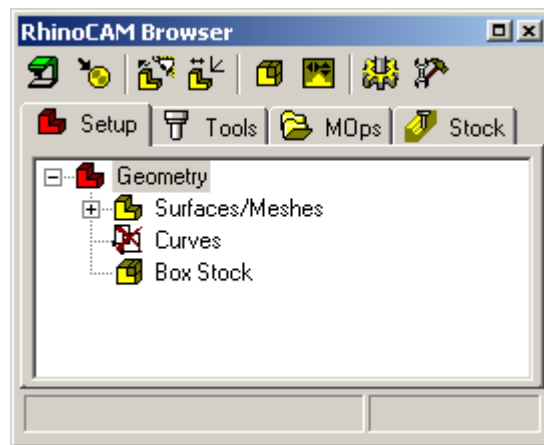
Browser

The Browser appears to the left of the screen by default. It can be hidden by clicking on the close button on the top right of the browser. To re-display the hidden Browser window, the user can access the **Show Browser** from the RhinoCAM menu entry. Like all toolbars, you can drag it to another location by clicking and dragging it by one of its corners. You can also resize it by dragging one of its sides.

The Browser contains the following four tabs:

Setup Tab

Selecting this tab displays the setup toolbar at the top of the Browser window. Additionally it displays the three types of geometry that are relevant to RhinoCAM: **Surfaces/Meshes**, **Curves** and **Stock**.



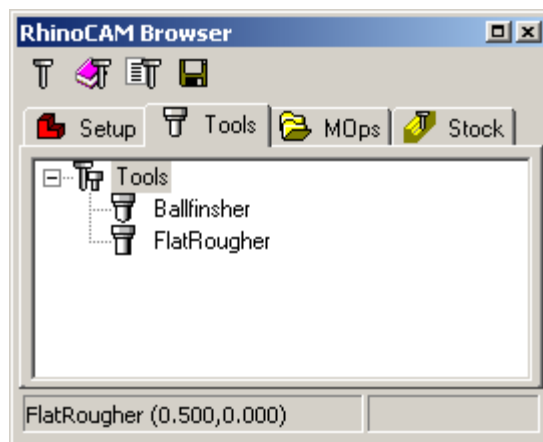
The first icon represents the **Surfaces/Meshes**. For an imported part, the full path is indicated. If the part consists of surfaces, each surface is represented as a **Mesh** in RhinoCAM. It is this mesh data that is used for creating toolpaths. You can also use the browser view of the CAD data to perform selections. By clicking on a mesh icon it will be selected and displayed as selected in the graphics window.

Curves in the model are regions used to define machining boundaries or regions. You will work with regions in the next tutorial.

Lastly, the **Stock** icon indicates the type of stock. You can double-click on this icon to create different types of stock models.

Tools Tab

This tab lists all tools currently defined in the file.

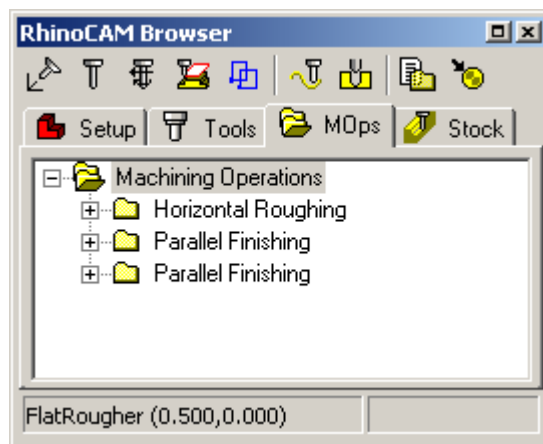


If you have created machining operations, the toolpath will appear in the **Tools** tab underneath the tool it uses. You can rename and delete tools, but you cannot delete a tool that is used in a toolpath. Double-click a tool icon to edit its parameters.

You can also save the tools created to a tool library by selecting the **Save Tool Library** button in the toolbar. Conversely you can also load tool libraries to be used in the RhinoCAM session.

MOps Tab

“MOps” stands for Machining Operations. All machining operations you create are listed here, in order of creation.



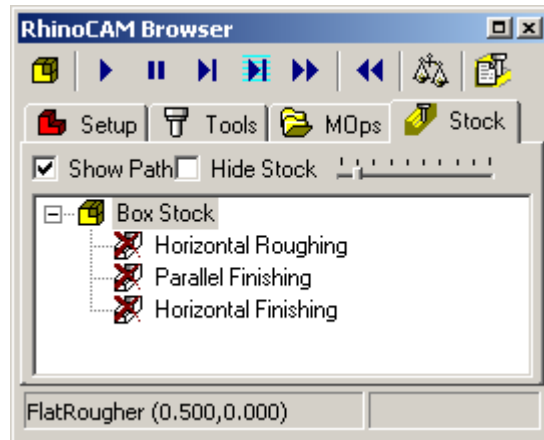
Within each Mop folder you can edit its various components, such as tool, regions, or cut parameters, by double-clicking the relevant icon. Right-clicking on a toolpath name provides several options, including simulation, generation, and post-processing.

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

Getting Started with RhinoCAM

Stock Tab

This tab lists the stock model corresponding to each machining operation in the part. The root level icon lists the original stock model. As with the MOPs tab, MOPs are listed in the order of creation. Selecting the icon corresponding to each operation will display the corresponding in-process stock model associated with this operation.



The buttons on toolbar corresponding to this tab are used for toolpath simulation.

Creating the Stock

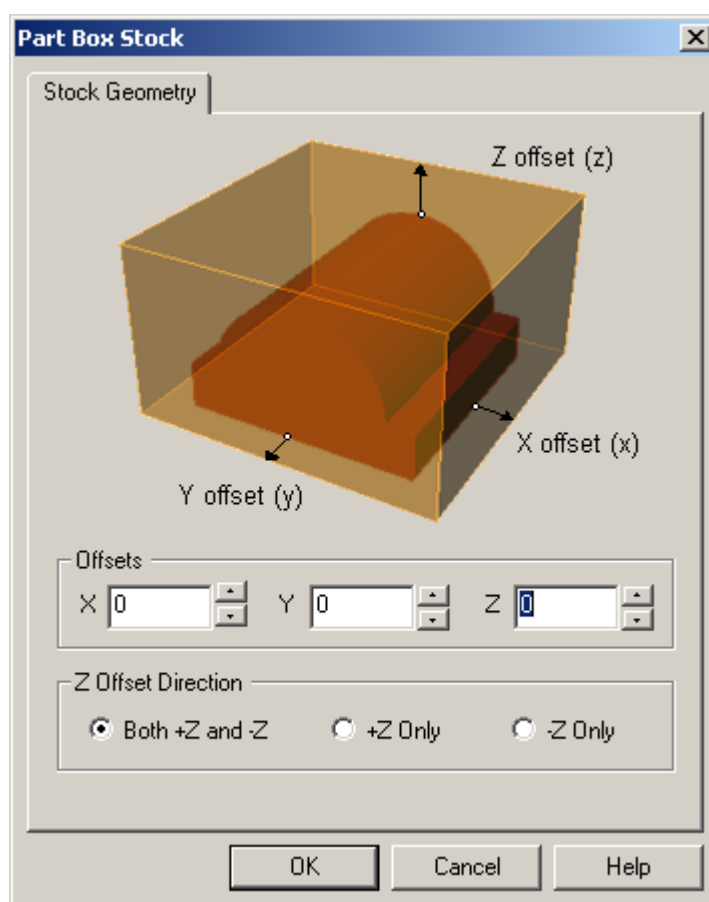
“Stock” represents the raw stock from which the part will be manufactured. In RhinoCAM stock can be created by entering coordinates of a box or as the bounding box of the part geometry. This stock model will be used for creating the simulation model for each of the operation. In addition to this some of the machining methods use the stock model for computations.

Note: You must define a stock model before creating **Horizontal Roughing** operations. All other operations can be created without first creating a stock model.



To create the stock:

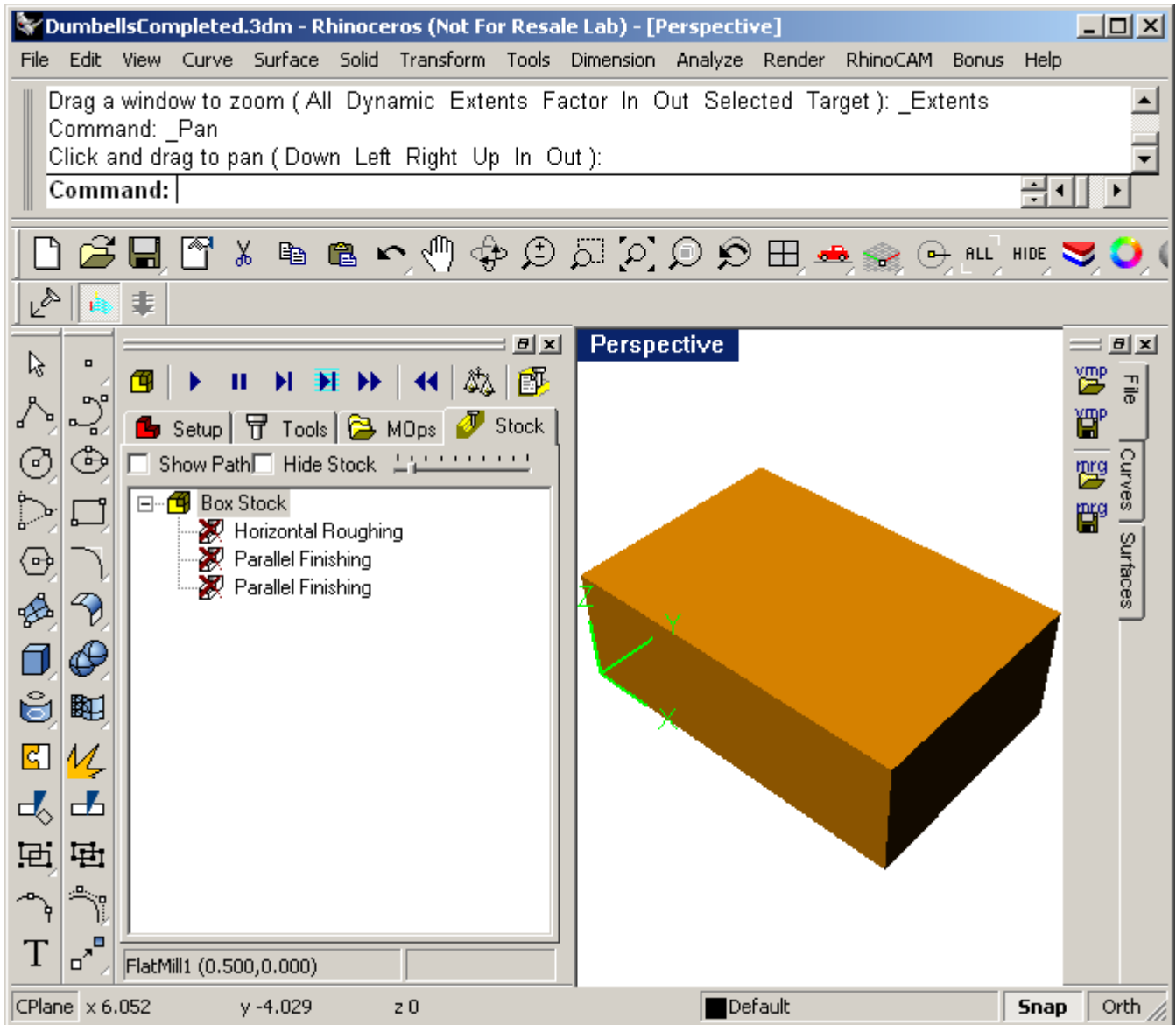
1. Select **Create/Load Stock** from the **Setup** tab of the Browser and select **Part Box Stock**. (This button is also available in the **Stock** tab toolbar of the Browser.) This option enables you to create a stock box that encloses all of the part geometry (bounding box).
2. In the Part Bounding Stock Box window, you can expand the bounding box by entering offset values. For the purposes of this exercise, do not enter any offsets. Click OK.



3. The stock model is created. To display the stock, click the **Stock** tab of the Browser. The stock is displayed as a box over the part. Its color can be set in the Color Preferences.

Note: Stock is used for simulation, and its display involves data-intensive rendering. This can slow down RhinoCAM's performance. Therefore, we recommend turning off the stock display when not needed. Additionally for performance reasons, RhinoCAM renders the stock model only in the active view port. So it is recommended that you work with a single view port rather than multiple view ports.

Getting Started with RhinoCAM



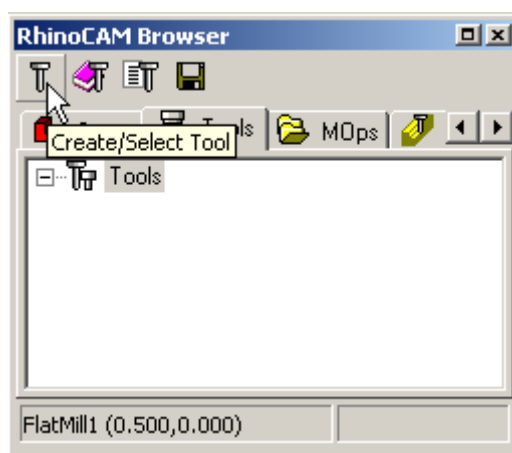
Creating Tools

RhinoCAM supports numerous types of milling and drilling tools. For each tool you can specify standard APT parameters: diameter, corner radius, taper angle, flute length and tool length. Flute length is the cutting length, and Tool Length is the total length of the tool up to the tool holder. All defined tools can be viewed in the **Tools** tab of the Browser. You can also save a set of tools to an external folder that can be loaded in other files.

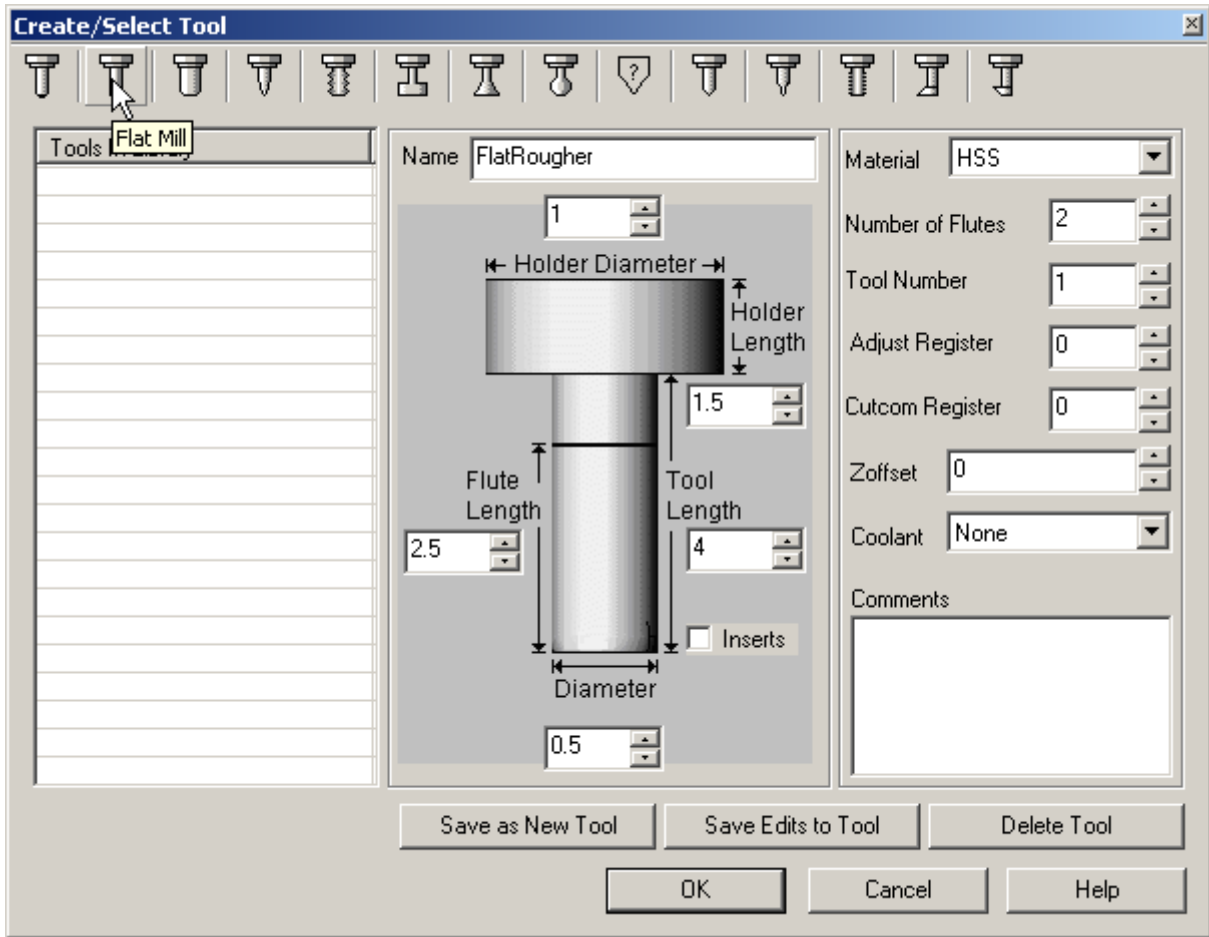


To create the roughing tool:


1. Select the **Create/Select Tool** icon on the **Tools** tab of the Browser.




2. This will bring up the Select/Create Tool dialog. Click the **Flat Mill** button from the tool types toolbar at the top of the dialog.
3. Name the tool as “**FlatRougher**” and leave the other values as default values.
4. Click on the **Save as New Tool** button in the dialog.



The tool will be created and its name **FlatRougher** will be displayed in the tool list.

 **To create the pre-finishing tool:**

1. In the **Select/Create Tools** dialog, click the **Ball Mill** button.
2. Create a **“Ballfinisher”** tool with 0.25 inch diameter and select the **Save as New Tool** button.
3. Click **OK** to close the window.

 **To create the finishing tool:**

1. In the **Select/Create Tools** dialog, click the **Ball Mill** button.
2. Create a **“Finefinisher”** tool with 0.125 inch diameter and select the **Save as New Tool** button.

3. Click **OK** to close the window.

Now note that all tools have been created and listed in **Tools** tab of the Browser.

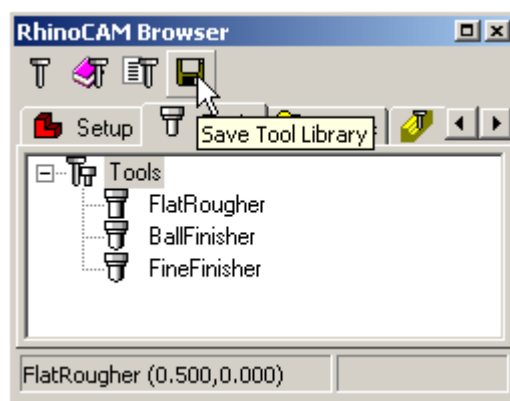
***Note:** You can double-click on any tool icon in the **Tools** tab to open its definition window. This is an easy way to make changes, if needed.*



To create the tool library:

You can save the tools in the list to a library, which can be accessed during future sessions.

1. Now the lists contains only the three tools you have defined.
2. A group of tools can be saved to a library file for future use. Select the **Save Tool Library** button from the toolbar in the **Tools** tab of the Browser.

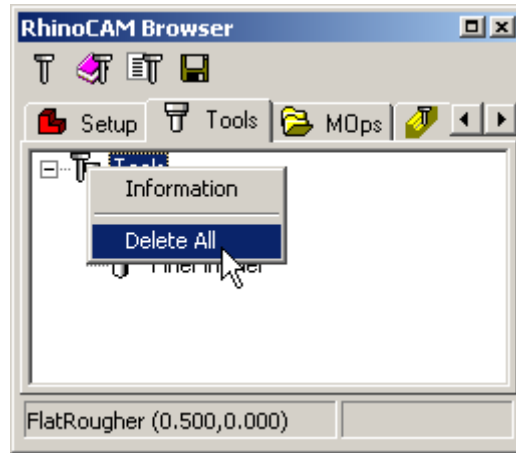


3. In the default folder (should be Tutorials, which contains the part file), assign the name **Dumbells**. The default extension is csv (comma-separated values), which means the file can be opened and edited as a spreadsheet or in a text editor. Click **Save**.

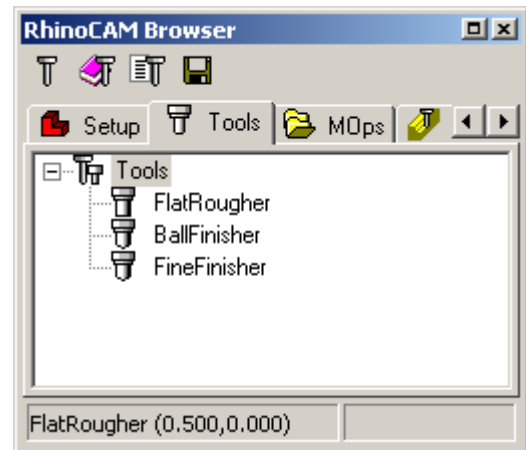
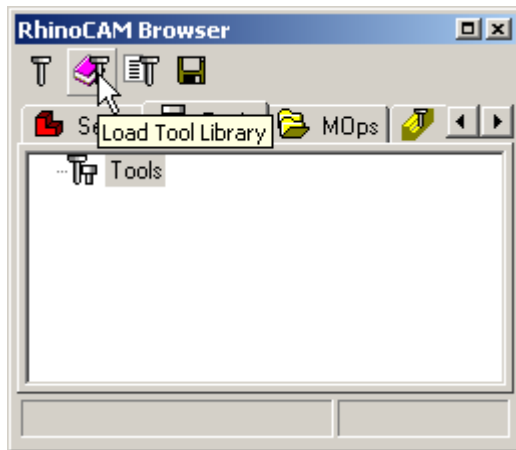
Now to reload the tools from the created tool library, we first delete the existing tools and read the tool library back. To do this:

4. Right-click on the **Tools** root level icon in the Browser and select **Delete All**.

Getting Started with RhinoCAM

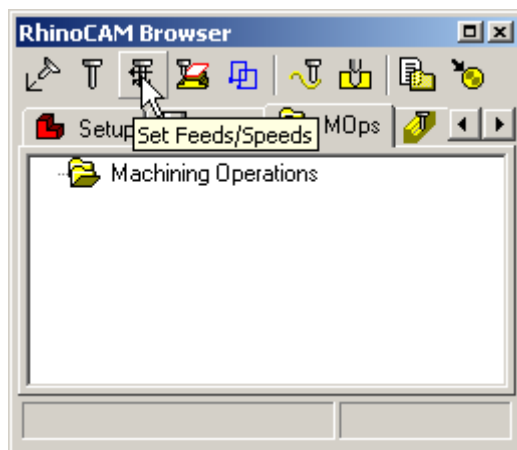


5. To replace the tools, select the **Load Tool Library** button. Select the Dumbells.csv file you just saved, and notice that the tools reappear in the listing.

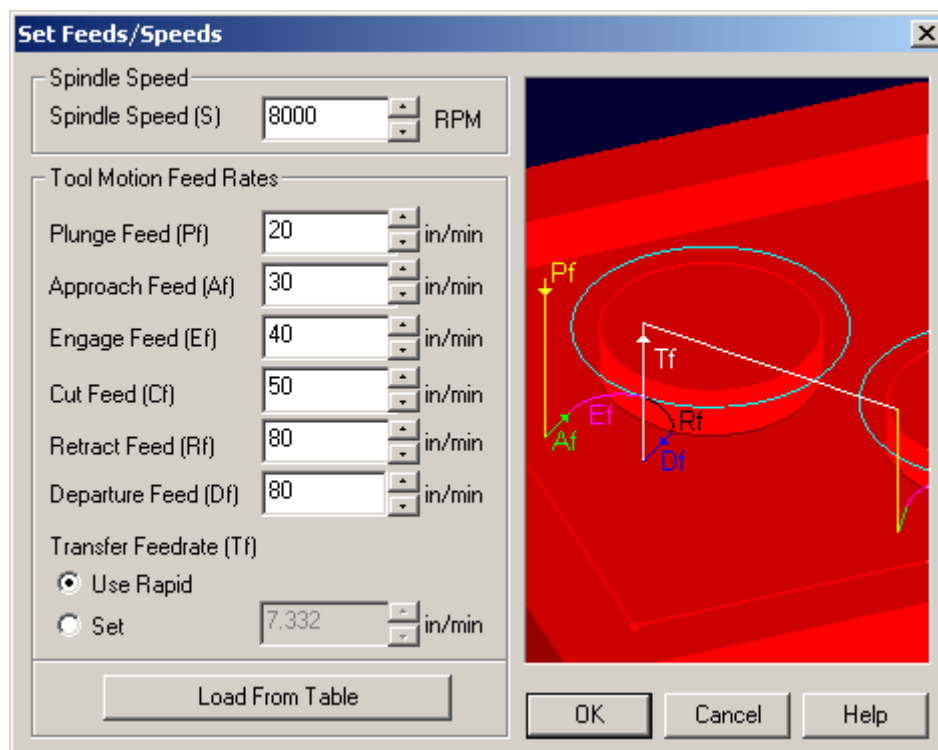


Setting Feeds and Speeds

You can set toolpath feeds and speeds and customize these settings for later use. You can do this using the Set Feeds/Speeds dialog. This dialog can be invoked from the **Set Feeds/Speeds** button found in the **MOps** tab toolbar.



Selecting this button will bring up the dialog shown below:



Getting Started with RhinoCAM

You can assign values for the following:

- ❑ **Spindle Speed:** The rotational speed of the spindle, in RPM.
- ❑ **Plunge Feed:** The vertical approach feed rate before the tool starts to engage in material.
- ❑ **Approach Feed:** The pre-engage feed rate that prepares the tool just before it starts engaging into material, as it starts cutting. These tool motions are dependent on the machining method.
- ❑ **Engage Feed:** The feed rate as the tool starts engaging into material. By default this value is 75% of the **Cut Feed**.
- ❑ **Cut Feed:** The feed rate used when the tool is cutting.
- ❑ **Retract Feed:** The feed rate as the tool stops cutting. By default, this is equal to **Engage Feed**.
- ❑ **Departure Feed:** The post-engage feed rate that prepares the tool just as it stops cutting.
- ❑ **Transfer Feedrate:** Specifies the feedrate of transfer motions (air motions). You can either use the **Rapid** setting of the tool, or set a custom feed rate.



To set feeds and speeds:

1. Select the **Set Feeds/Speeds** button on the MOps tab toolbar of the Browser.
2. Leave the **Spindle Speed** as is and change the **Cut Feed** to 30 in/min. Set the **Approach & Engage Feed** and **Retract & Departure Feed** to 20 in/min. Click OK.

These feeds and speeds will be used during the post-processing of the toolpath.

Creating the Horizontal Roughing Toolpath

In this type of toolpath, RhinoCAM uses stock geometry and part geometry to determine the safe machining region at each user-specified Z level. The safe machining region is the region in which the tool can safely traverse removing stock. Once this machining region is determined, a tool traversal pattern such as a zigzag or offset machining cut pattern can then be applied to remove stock.

***Note:** For 3-axis milling, you can also create and activate regions that act as machining boundaries. An example would be if you wanted to machine only one surface; you could define a region around that surface so that no other areas will be milled.*

Regions are curves that already exist in the model, or curves you create within RhinoCAM. In the **Setup** tab of the Browser, you will see that one region already exists in this file. Regions will not be used in Tutorial 1, since they are not required for 3-axis milling. However, they are required for 2½-axis milling, so you will learn about them in Tutorial 2.



To create the horizontal roughing toolpath:

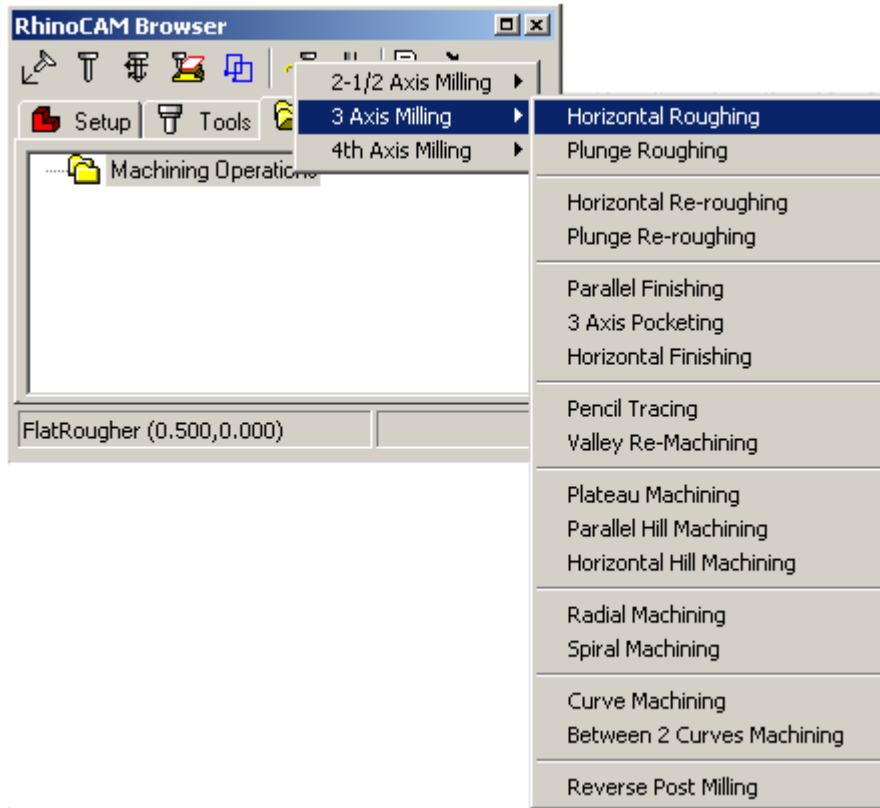
1. All you need for toolpath creation is found on the **MOPs** tab of the Browser. First you must activate the tool to be used. Select **Tool / Create/Select Tool** or click the icon on the Browser.



Select the **Flat End Mill** button, pick **FlatRougher** and click OK. **FlatRougher** is now the active tool,

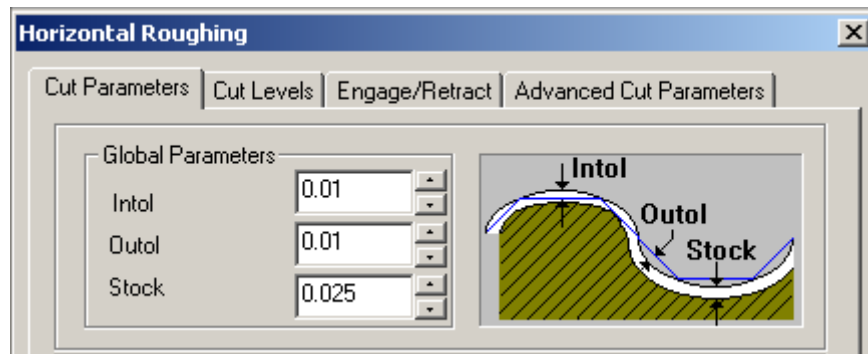
Getting Started with RhinoCAM

2. Select **3-Axis Milling** from the MOPs tab toolbar and select **Horizontal Roughing** from the drop down menu.



The **Horizontal Roughing** dialog is invoked, in which you can set parameters for the toolpath.

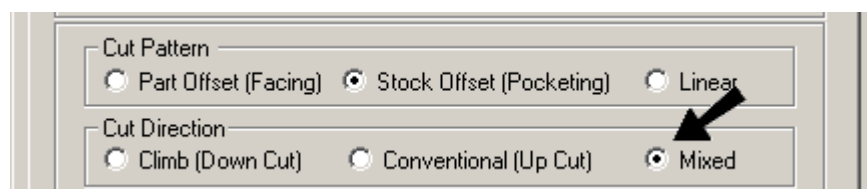
3. Click the Cut Parameters tab. At the top of the window are Global Parameters: Intol, Outol, and Stock.



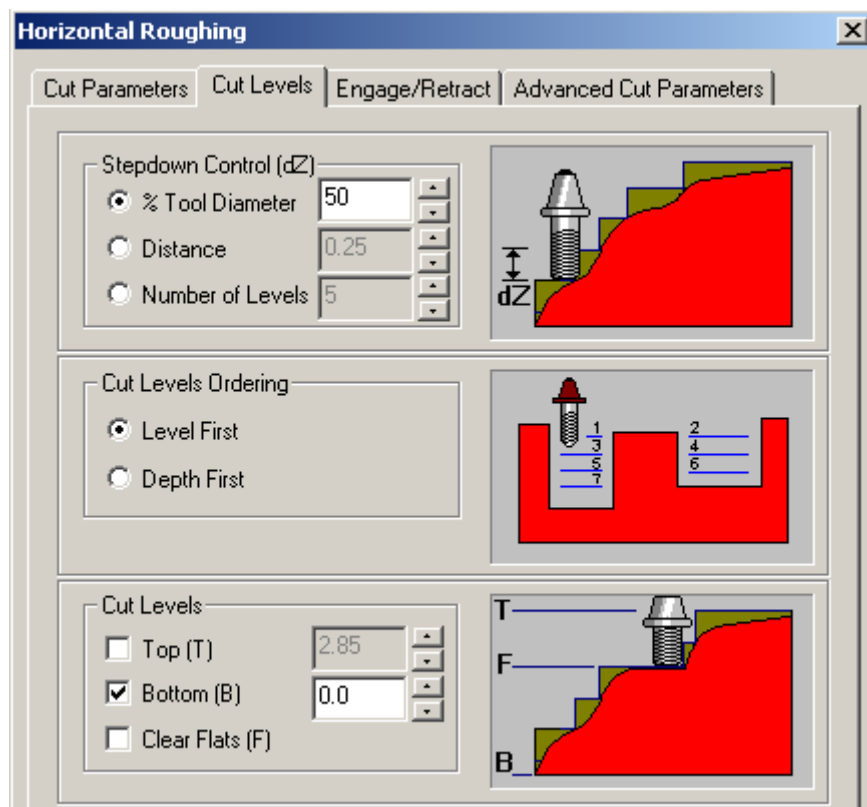
- **Stock:** The thickness of the layer that will remain on top of the part after the toolpath is complete. Roughing operations generally leave a thin layer of stock, but for finishing operations this value is zero.

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

- **Intol:** Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer.
 - **Outol:** Outward tolerance - the maximum thickness of material that can remain above the **Stock** layer.
4. Leave the **Global Parameters** as they are, but change the **Cut Direction** to **Mixed**. This enables the tool to cut either up or down, so that the tool does not have to retract each time the direction changes. Minimizing the number of tool retractions creates a cleaner toolpath.



5. Click the **Cut Levels** tab and set the **Bottom** cut level to 0.0.

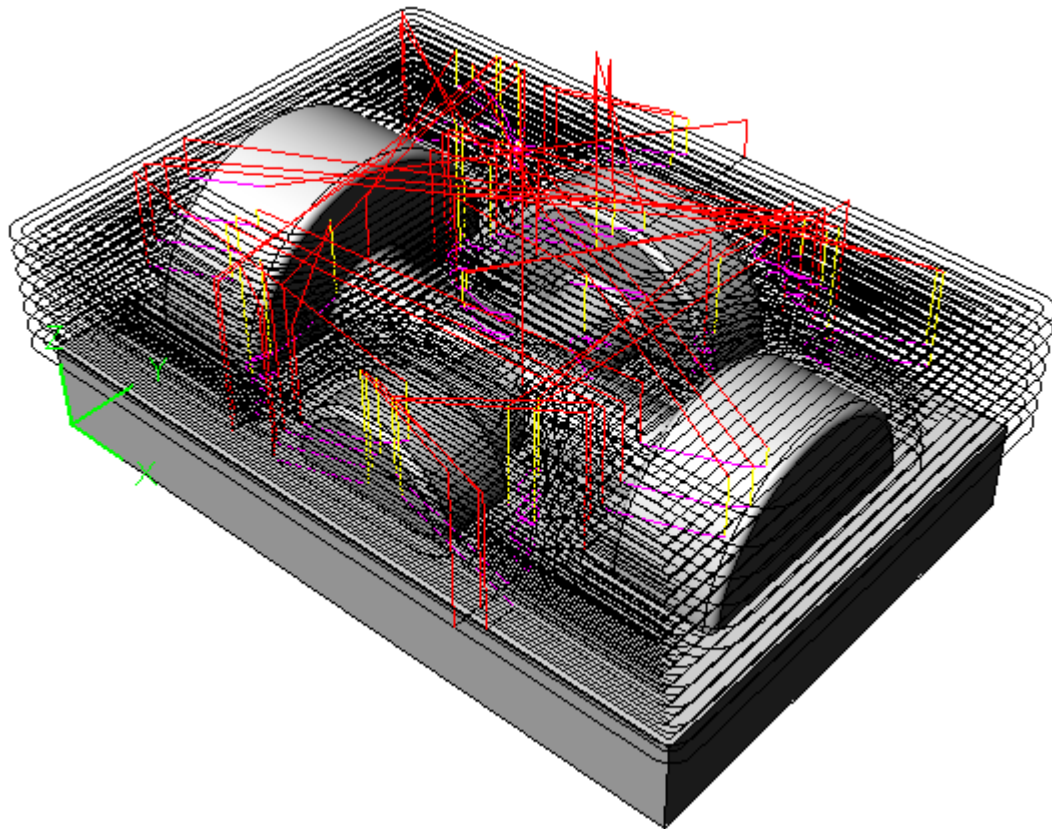


Getting Started with RhinoCAM

6. Leave the remaining parameters in the other tabs as they are, and click Generate, located at the bottom of the window.

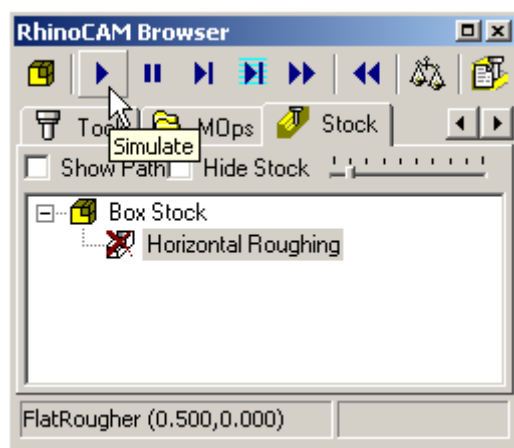
The window will disappear and a dialog box with a progress bar will appear on the screen. When the calculation is complete, this dialog will disappear and the roughing toolpath will appear on the screen.

***Note:** Make sure you are in the Shaded mode to see the toolpath and the simulation. If you don't want to view the toolpath, you can select the Toggle Toolpath display button in the RhinoCAM View toolbar*




Simulation


The VCR commands are used for toolpath simulation. These commands can be found on the **Stock** tab of the Browser.




The RhinoCAM simulator enables you to view your toolpath in action, reflecting what the actual model would look like after machining. Simulation can also be used to catch errors.


For simulation to work, you must have stock geometry defined, and the stock must be displayed.


 **Create/Load Stock:** As you've already seen, this tool is used to create various types of stock material. Generally a stock is created automatically, depending on the type of toolpath you create, but you can always change the stock.


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

 **Pauses:** Stops the simulation.

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

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

 **Simulate to End:** Simulates to the entire toolpath before updating the display.

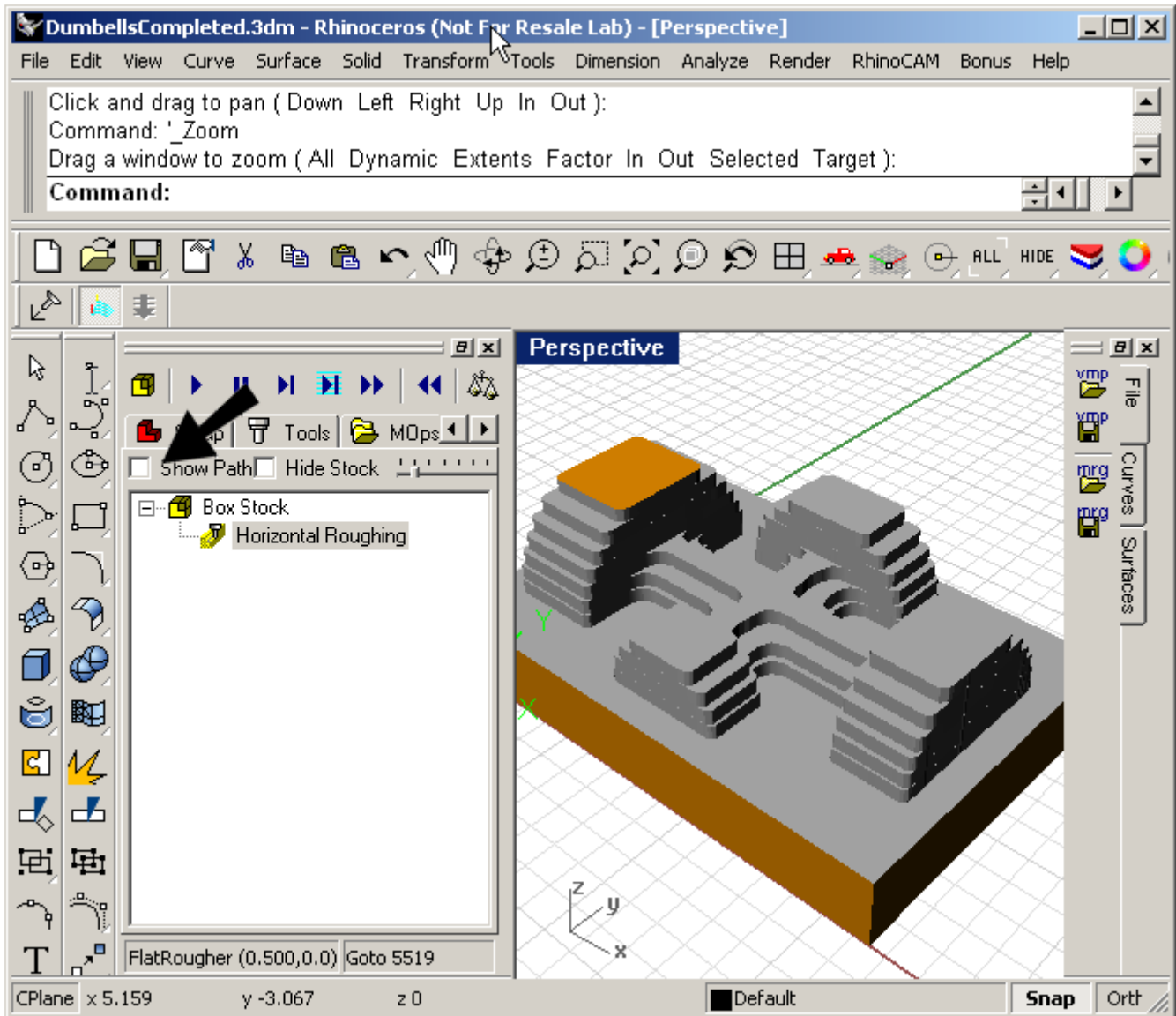
 **Rewind:** Jumps to the start of the simulation. This is used to clear the simulation data if the user stops in the middle of simulating a toolpath and does not want to complete the simulation.

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

Getting Started with RhinoCAM

Simulating the Horizontal Roughing Toolpath

Now that the first toolpath has been created, you can simulate it. To simulate the toolpath, make sure that the created machining operation appears in the stock browser and is selected, and then click **Simulate**. Once the simulation is complete, the cut stock model will be displayed as shown below. To better visualize the simulation model you can un-check the **Show Path** check box to display just the stock model.



Note: You can also try clicking *Step simulation* to view a set number of tool motions at one time, or *Step Z Levels* to see levels removed one at a time.

Tips: You can change the color of the stock and cut stock by selecting *Preferences / Color Preferences* and clicking the color box for *Cut Stock Color*. You can find *Preferences* in the setup tab.

The Machining Operation now has a yellow colored icon that shows that the simulation has been completed.

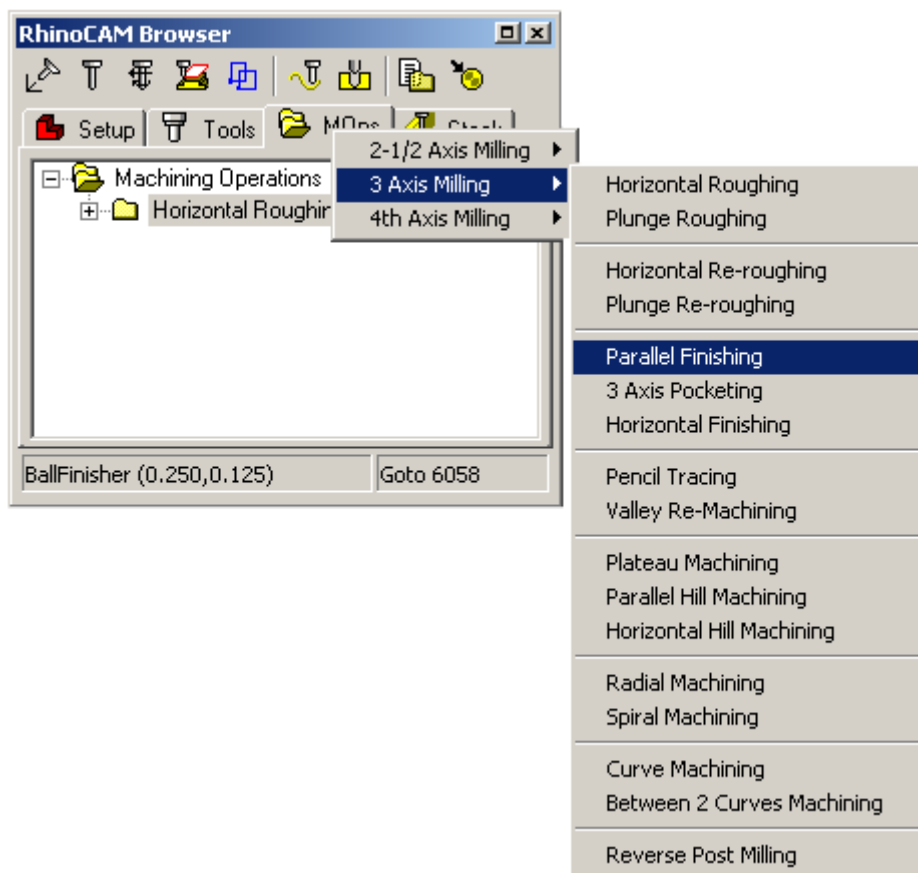
Creating the Pre-Finish Toolpath

Once the roughing toolpath is generated, a pre-finish toolpath can be created to remove the steps left by the roughing process, and to bring the stock closer to the part. There are several methods of generating pre-finishing toolpaths; in this exercise we will use the **Parallel Finishing** method.



To create the pre-finish toolpath:

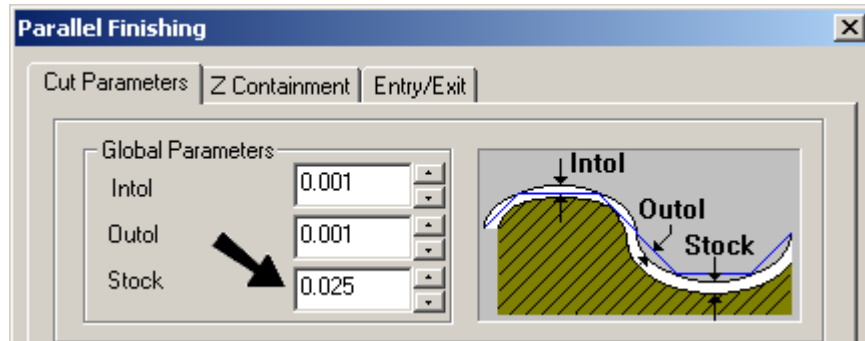
1. Return to the **MOPs** tab.
2. Open the **Select/Create Tools** window, click the **Ball End Mill** tab, and select **Ballfinisher**. Click OK to make this the active tool. (You could also double-click the tool name.)
3. Select **3 Axis Milling / Parallel Finishing**.



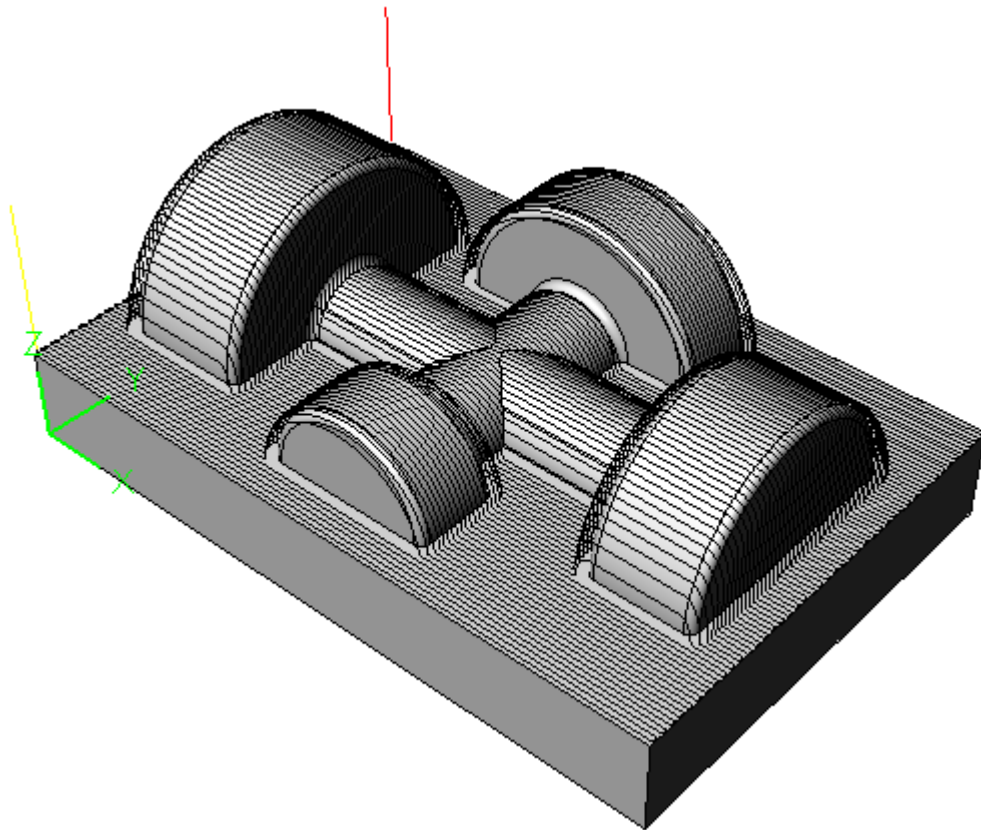
We will use all default settings except for the stock value. Pre-finishing is typically done so that there is a uniform thickness of stock left on the part, to be removed by the final finish path.

Getting Started with RhinoCAM

4. In the **Cut Parameters** tab, enter 0.025 in the **Stock** field. Notice that we leave the same amount of material on the part as in the roughing operation. This is to ensure that there is a uniform amount of stock left for the cutter to remove in the final finishing operation.



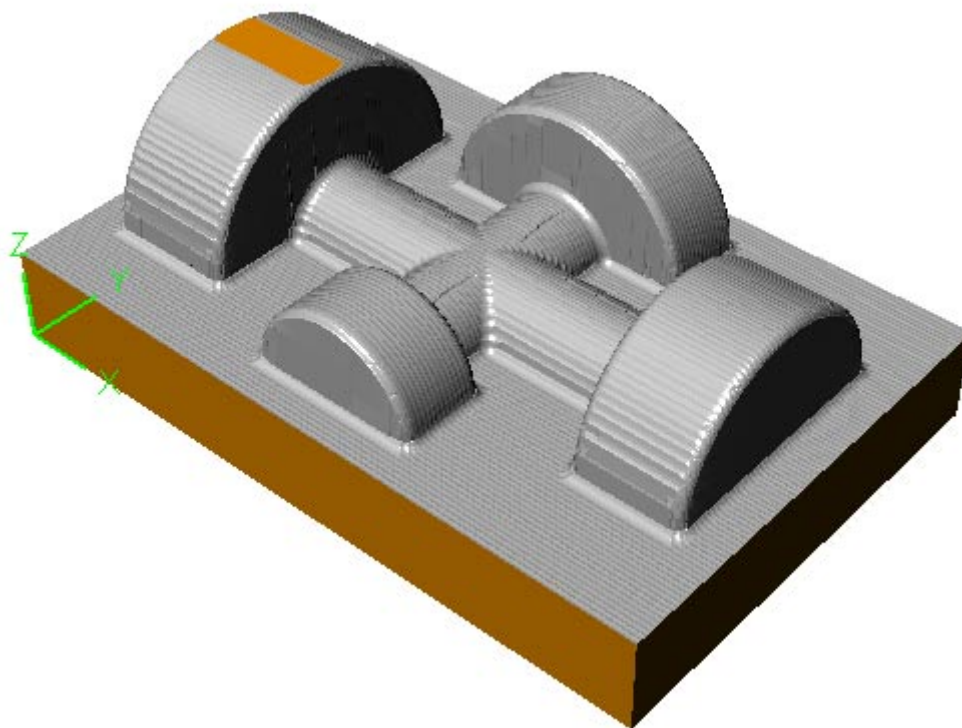
5. Click **Generate** to calculate the toolpath. This toolpath uses a linear zigzag pattern.



**To simulate the pre-finish toolpath:**

1. Switch to the **Stock** tab. Select the pre-finish toolpath to simulate.
2. Click **Simulate** to watch the cut material simulation being performed. Alternatively click on **Simulate to End** button to perform the simulation without any intermediate display of the stock model.

The results show how the stock will look after the pre-finishing toolpath is run.



Creating the Pencil Tracing Toolpath

Not available in RhinoCAM Basic. Basic users – proceed to “Creating the Finish Toolpath.”

In **Pencil Tracing**, RhinoCAM automatically determines all areas of the part where a tool would have more than one contact position. These areas are typically in valleys and corners. The tool is then moved along these areas to remove material.

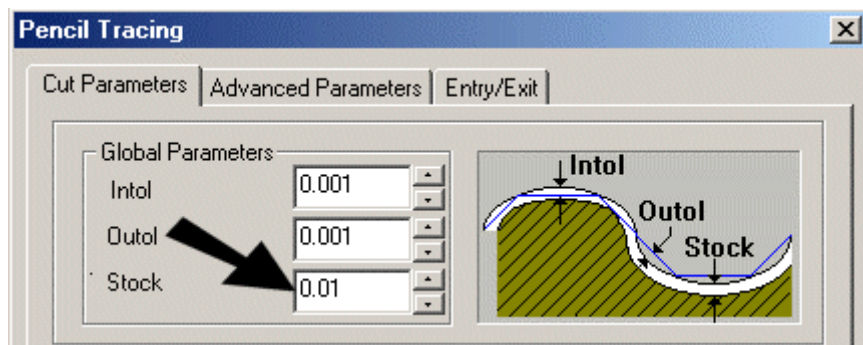
This is a very efficient method to stress-relieve corners and valleys before running a finish toolpath. This allows the finishing cutter to be used without having it encounter significant amounts of material in the valleys. It can also be used as a cleanup operation to remove scallops left in these areas after finishing operations.

Tip: Pencil tracing is typically performed with ball end mills. If there are any fillets in the model, make sure that the pencil tool radius is slightly larger than the maximum fillet radius. This ensures that the tool will be driven along all of the fillet surfaces.

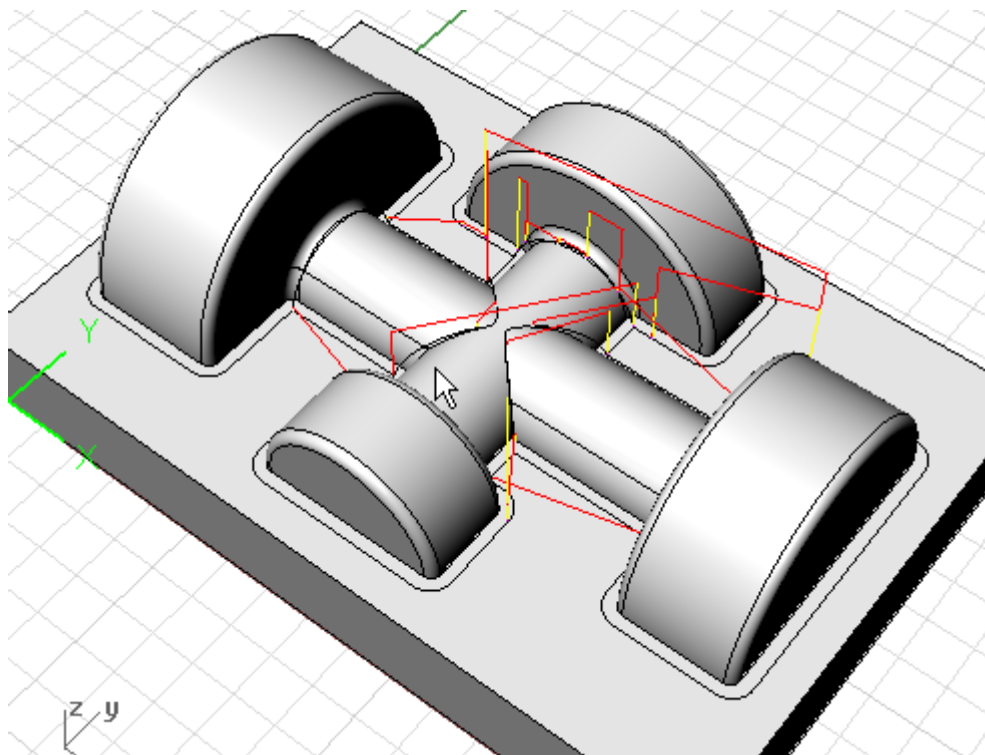


To create the pencil tracing toolpath:

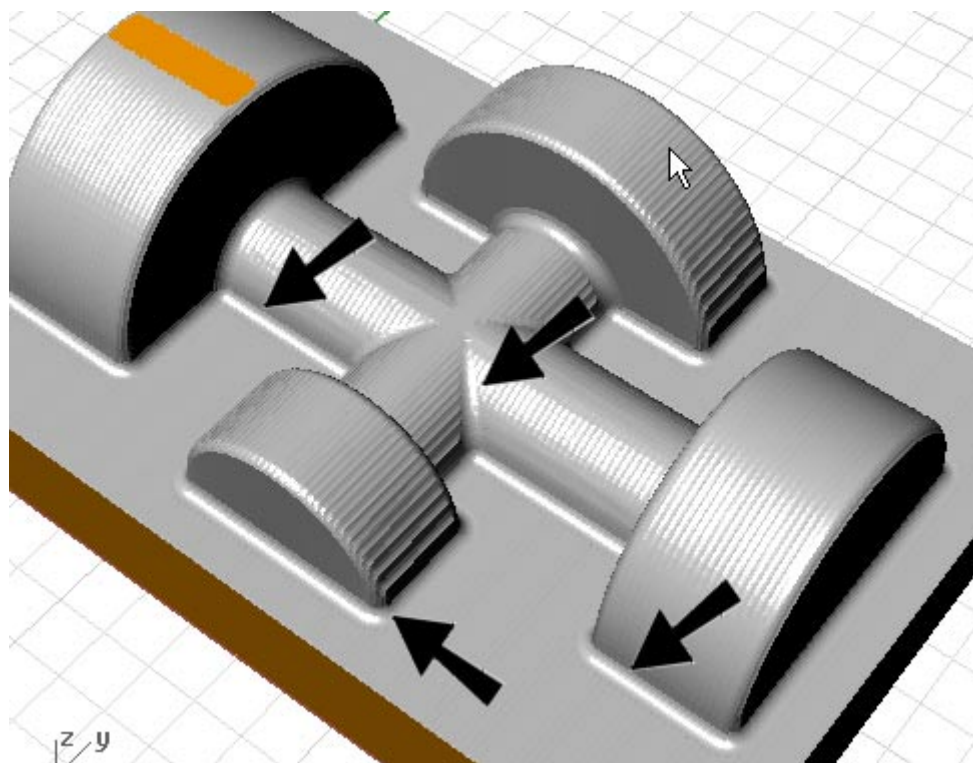
1. Make **BallFinisher** as the active tool.
2. Select **3 Axis Milling / Pencil Tracing**.
3. In the **Pencil Trace Cut Parameters** tab, set **Stock** to 0.01. Leave all other parameters at their default settings.



4. Click Generate



5. Switch to the Stock tab and simulate this toolpath. The pencil trace toolpath removes the scallops along the valleys of the part as shown below.



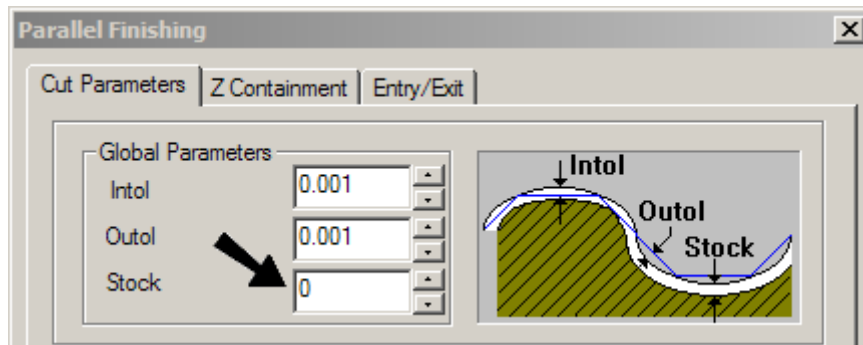
Creating the Finish Toolpath

We use **Parallel Finishing** again as the final step to clean up any remaining material left in the previous tool path using a smaller tool.

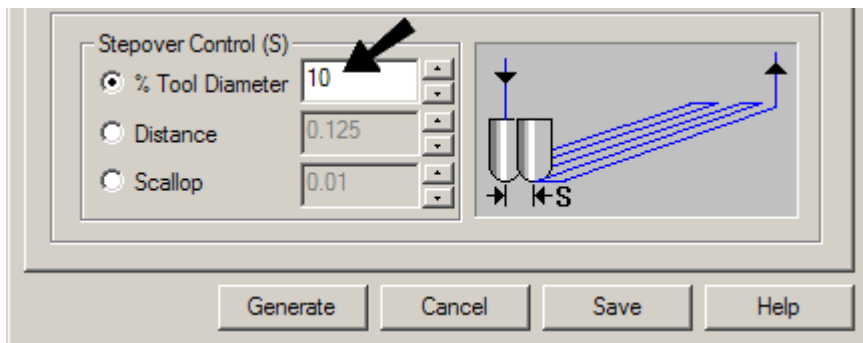


To create the **Parallel Finishing** toolpath:

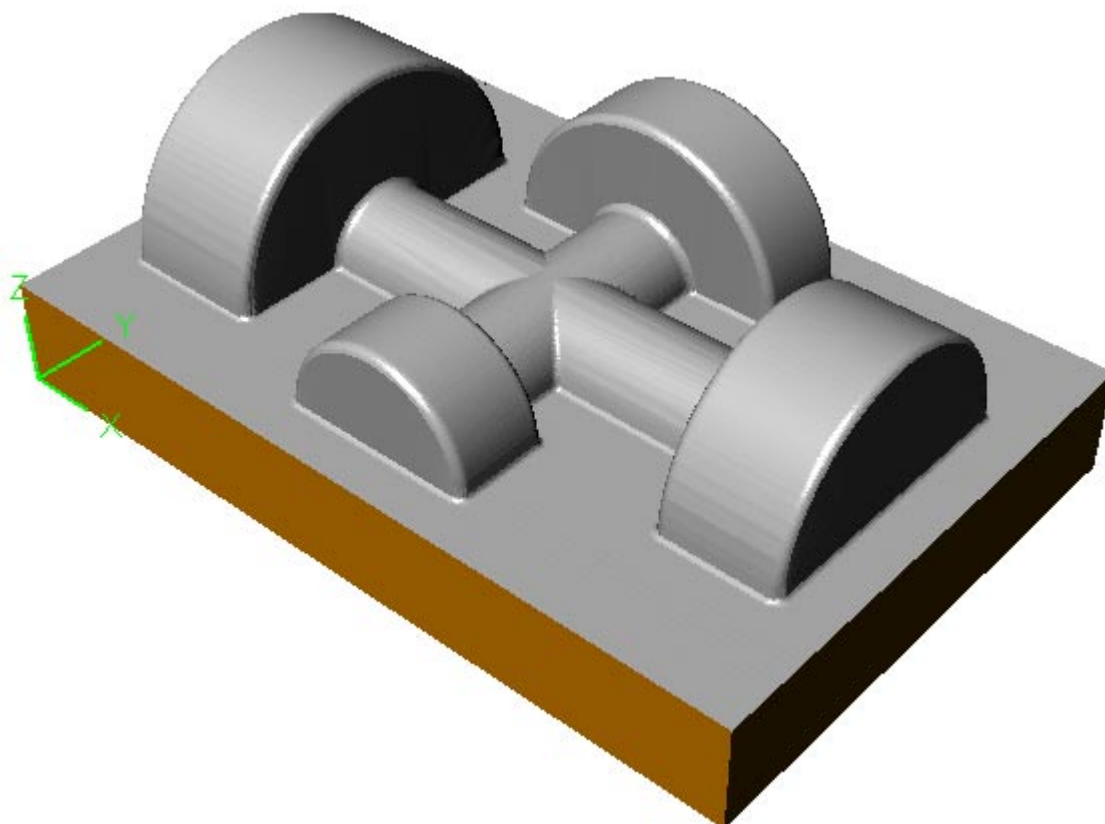
1. Make **FineFinisher** the active tool.
2. Select **3 Axis Milling** and **Parallel Finishing**.
3. In the **Parallel Finishing Cut Parameters** tab, set **Stock** to 0. This is typical of finish operations where we will machine all the way to the net dimensions of the part geometry.



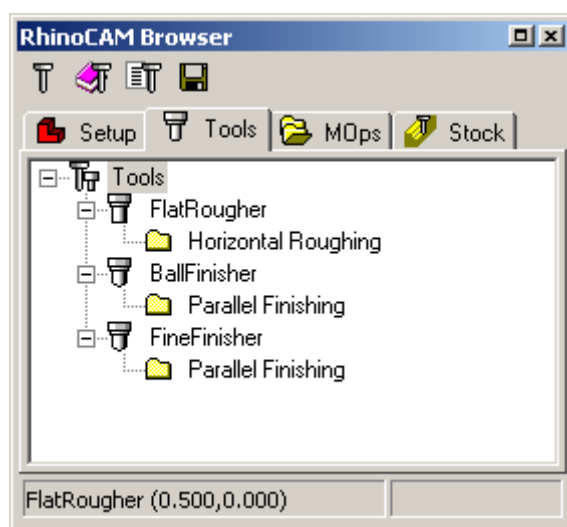
4. In the **Stepover Control** settings specify the stepover value as 10 % of **Tool Diameter** as shown.



5. Click the **Generate** button.
6. Switch to the **Stock** tab and simulate this toolpath. Notice that this final finishing toolpath removes the remaining material left over from the previous operation.



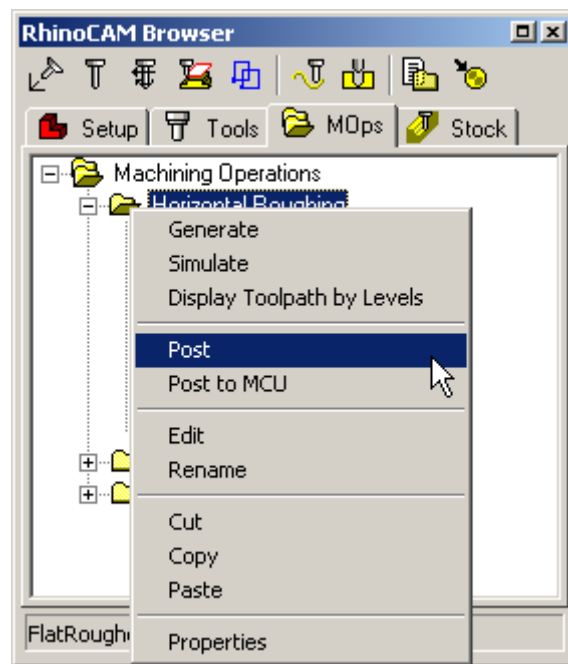
7. Click the Tool tab in the Browser to see each tool, linked to the toolpath that uses it.



Post-Processing

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

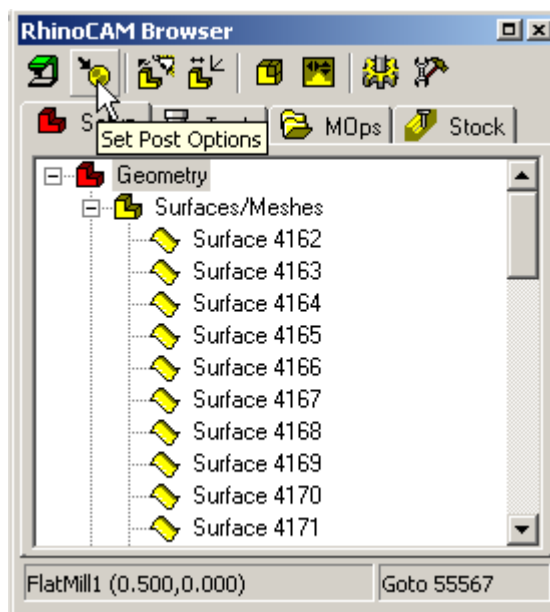
You can post-process an individual toolpath, or all toolpaths at once. For post processing an individual toolpath, right-click on its name in the **MOps** tab of the Browser and select **Post**. You can Right-Click in the **MOps** tab on any machine operations. You will see the options shown in the picture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



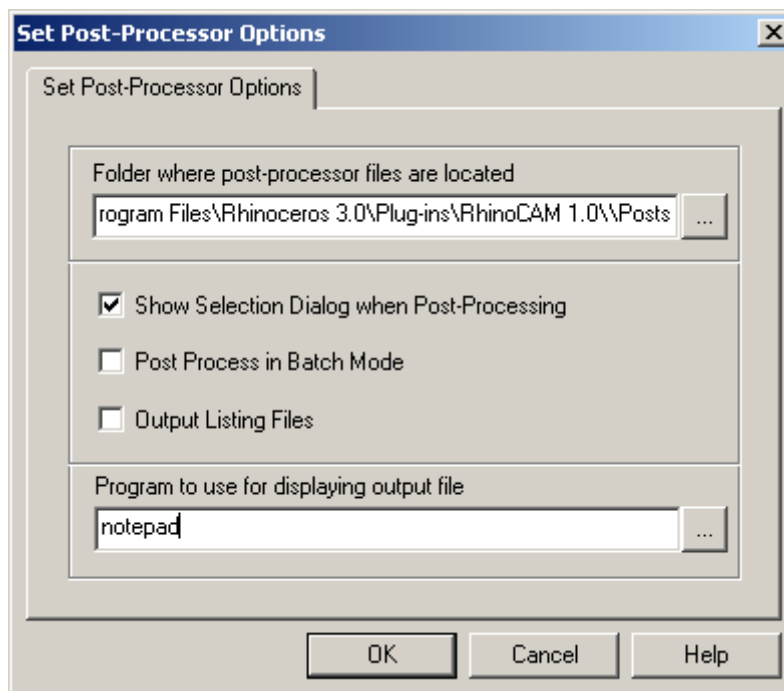
The entire list of toolpaths can be also be post-processed to a single file by right-clicking the root folder in the **MOps** tab and selecting **Post All**.

Set Post Options

Certain post-processor options can be set by selecting the **Set Post Options** button in the toolbar of the **Setup** tab in the browser.



The following dialog is available for selecting the path of Post files and for displaying the output file.



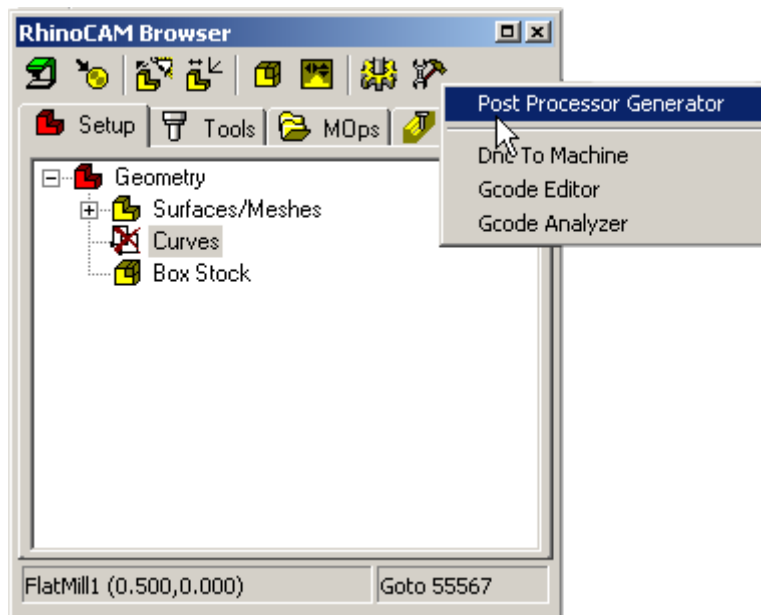
Getting Started with RhinoCAM

By default, RhinoCAM performs interactive post-processing. In other words, RhinoCAM launches the post-processor and waits for the post-processor to complete. You can check **Post Process in Batch Mode** to tell RhinoCAM not to wait for the completion of the post-processor. You can also output the list files and display the selection dialog for each post processing operation.

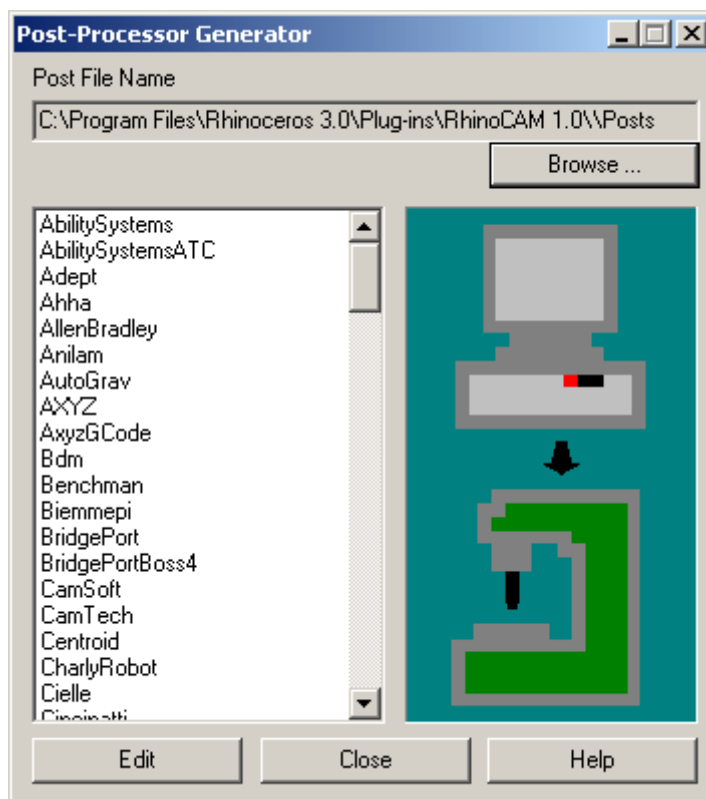
During interactive post-processing, RhinoCAM launches a text editor to display the output file. By default, Notepad is used. This is displayed in the edit box titled **Program to use for displaying the output file**, but you can specify a different program by typing in the full path name of the program or using the browse button to select the program.

Post Processor Generator

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



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



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

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

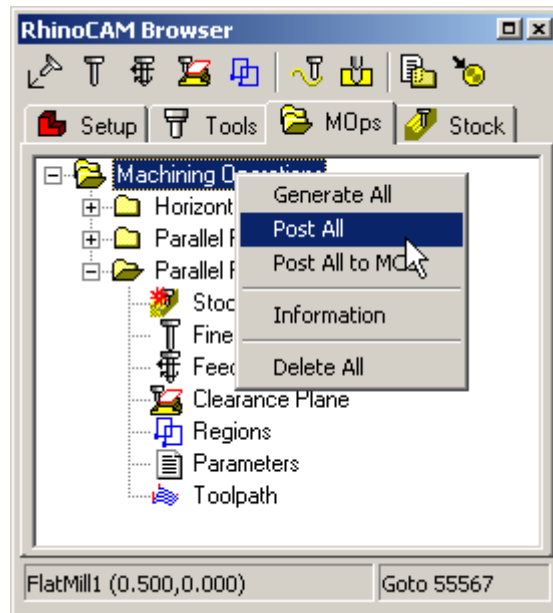


To create the post-processed output:

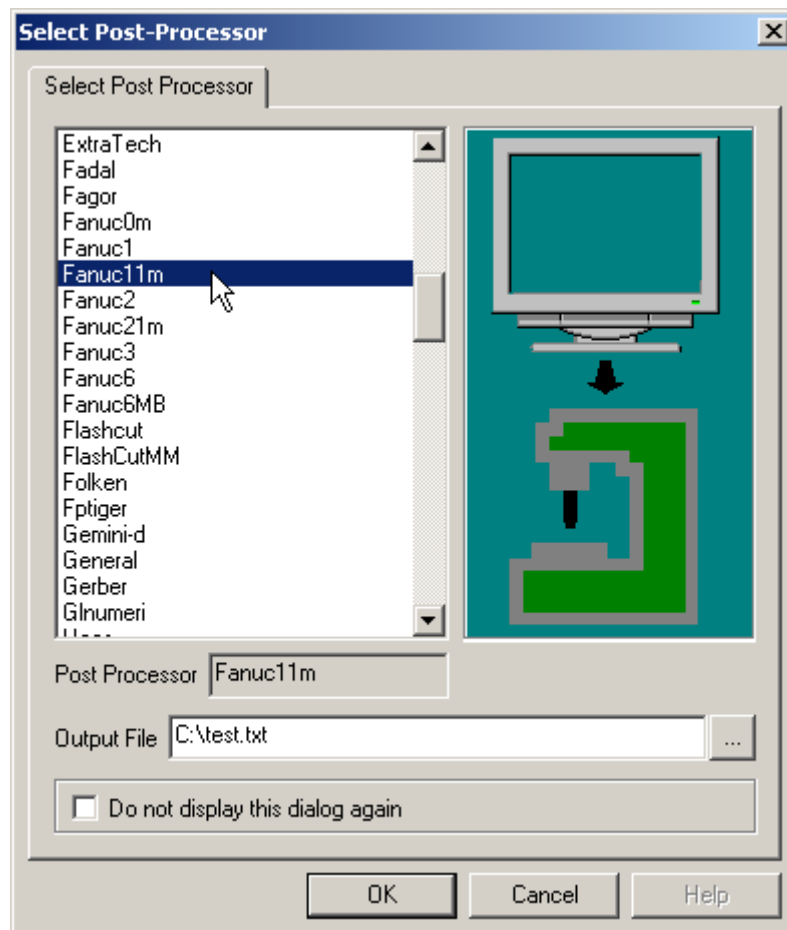
In this exercise we will post-process all of the toolpaths at once.

1. In the **MOps** tab, right-click on the root folder (**Machining Operations**) to open the popup menu. Select **Post All**.

Getting Started with RhinoCAM



2. Browse to the desired output directory and assign a file name for the output. Make sure that this file name is a valid file name and you have permissions to write to the specified file. Then double-click on the post-processor you want to use (such as Fanuc 11m).



3. When complete, the post output file will open in the default text editor (Notepad by default). This file contains all the G-code for your toolpaths.

Note: You can post individual toolpaths by right-clicking on their name in the **MOps** tab and selecting **Post**. The **Post-Process** icon on the **MOps** tab can also be used. To post multiple toolpaths, select each toolpath while keeping **Ctrl** pressed, right-click, and select **Post All**.

Post-Processor Problems

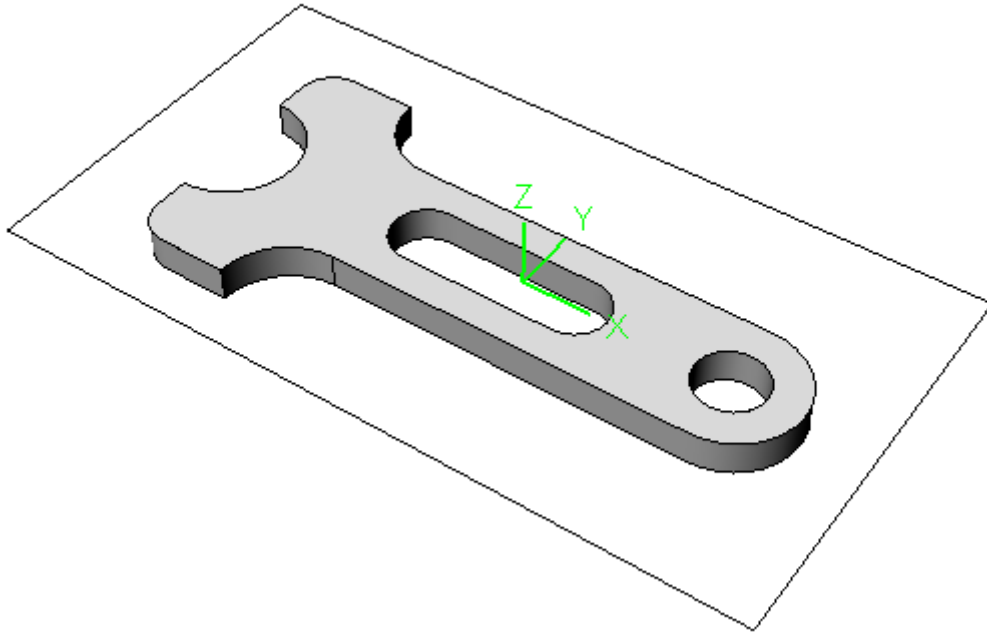
If only two built in posts (APT CLS and Roland CAMM GL) are displayed in the selection dialog, then your **Post** folder is not set correctly. Try the following:

1. Select **Post Process / Set Post Options**.
2. Click the **Browse** icon to change the folder where post-processor files are located.
3. Select the Posts folder located in the RhinoCAM installation folder.
4. If nothing happens when the toolpath is posted, try the following:
5. Select **Post Process / Set Post Options**.
6. Make sure Show Selection dialog when Post Processing is checked.
7. Make sure Post Process in Batch Mode is not selected.
8. Make sure Output Listing Files is not selected.
9. Post the machining operations, making sure you are browsing to the Post folder in the RhinoCAM installation folder. For the output file at the bottom, make sure there is a valid file name (valid path).

End of Tutorial 1!

Tutorial 2: 2½ Axis Milling Operations

This exercise uses the file **spanner.3DM**, located in the **Tutorials** folder. Use **File / Open** to load the part.



This part is suitable for 2½ axis milling because it is prismatic – it is a straight extrusion of curves in the XY plane. The solid geometry is not actually necessary for programming this part, but it is helpful when visualizing the process.

The curves on and around the part are called regions. These are used for defining milling areas, as you will see later in this exercise. The regions are currently located on the same layer as the part, so they have the same color as the part.

Creating Tools and Stock

Before any operations can be performed, you must define the milling tools and the stock geometry.



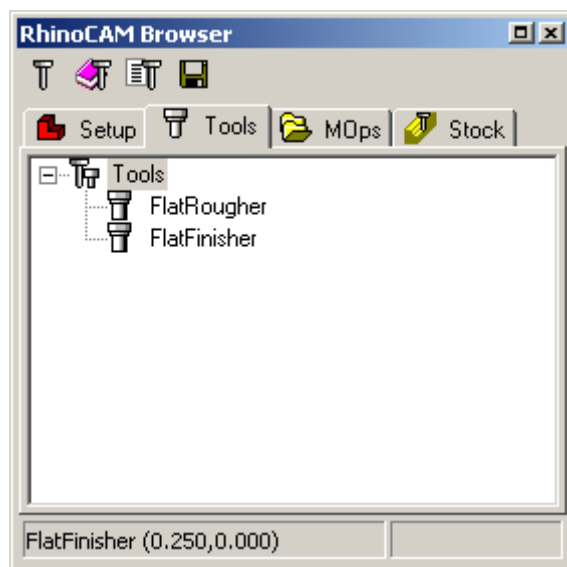
To create the 2½ axis milling tools:

1. Create and save the following two tools:

Flat end mill named **FlatRougher** with a diameter of 0.5 inches

Flat end mill named **FlatFinisher** with a diameter of 0.25 inches

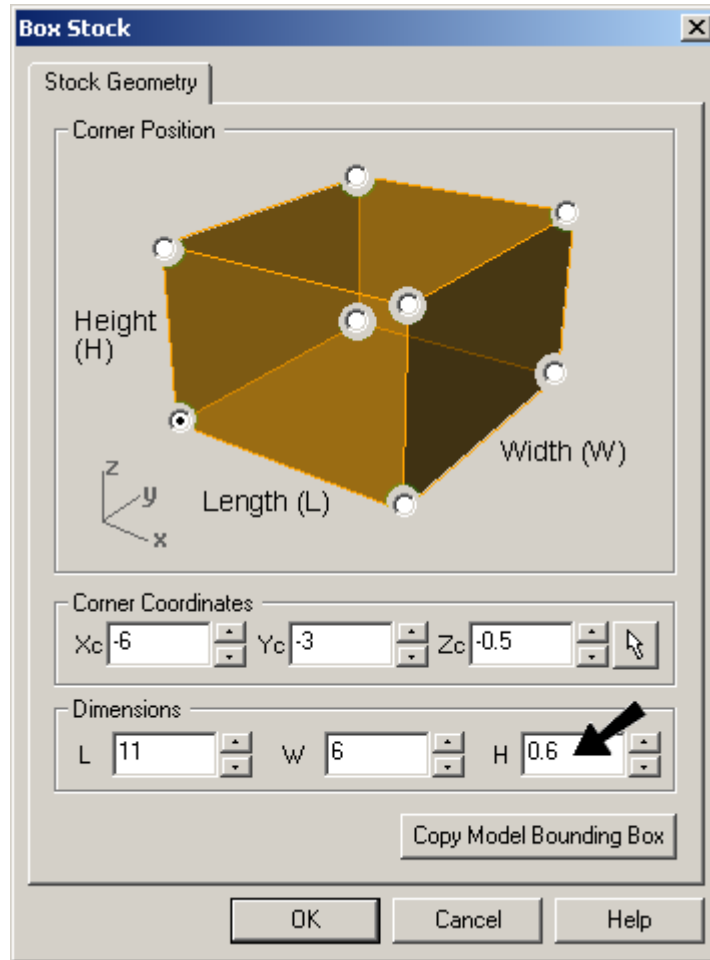
2. Open the **Tools** tab of the Browser to see the tools listed.



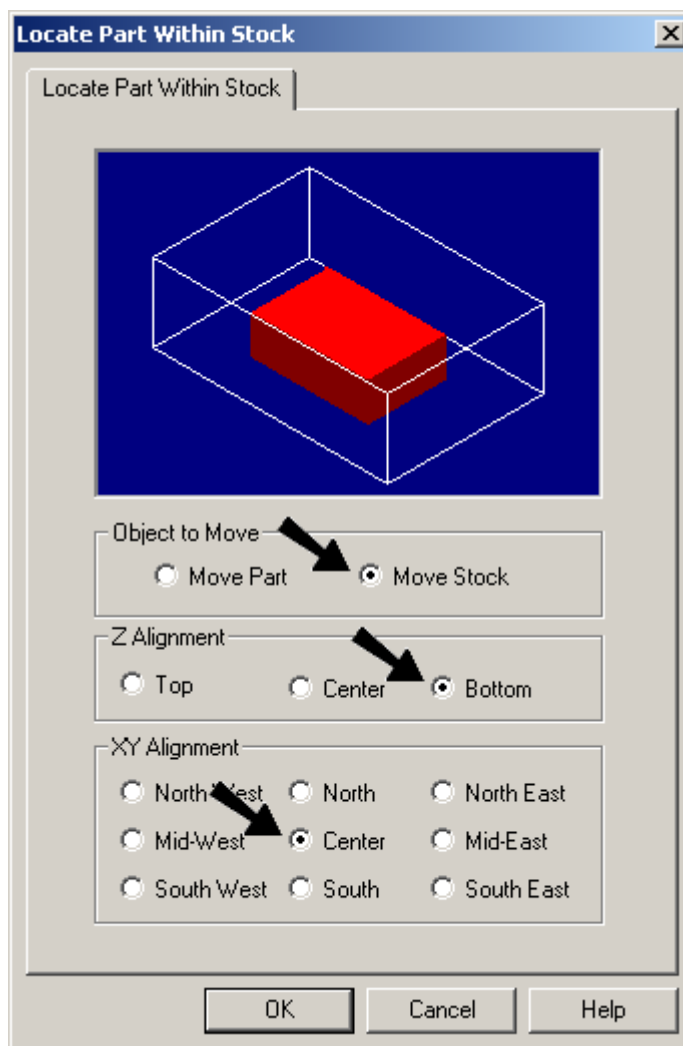
To create the stock:

3. Create a **Box Stock**.
4. The dimensions of the bounding box are automatically calculated. Change the height to 0.6 from 0.5, so that there will be 0.1 inches of stock material above the part.

Getting Started with RhinoCAM



5. Now to locate the stock such that the stock model top is aligned with the top face of the part model, click **Locate Part Within Stock** button on the **Setup** tab.
6. Check **Move Stock** and **Bottom** so that the bottom face of the stock will align with the bottom face of the part. Check **Center** to center the stock about the part in the XY plane.



7. Click OK to locate the stock correctly.

Regions

Regions are curves that already exist in your model, or curves you create within Rhino using the **Curve** tools. Regions serve different purposes in 2½ and 3 axis milling.

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

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

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

Getting Started with RhinoCAM

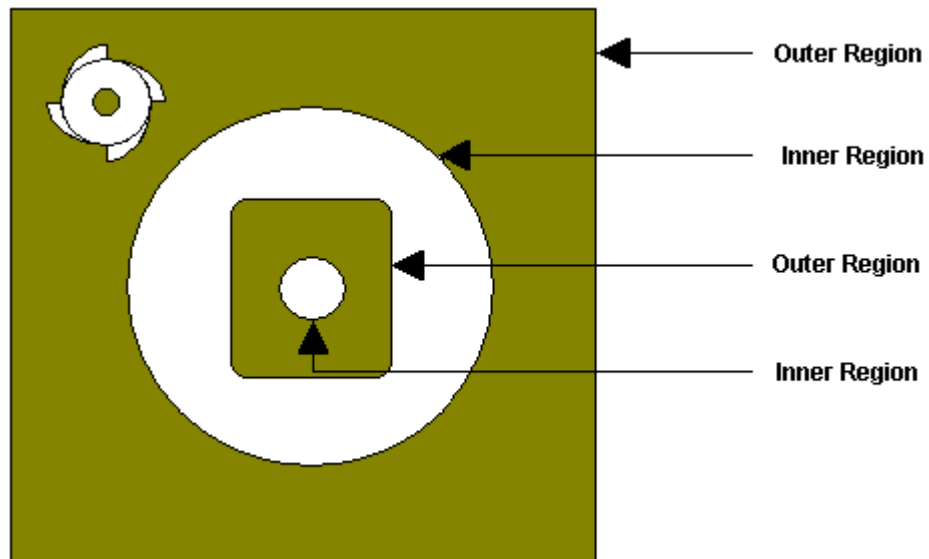
Note: If a region is not defined for finishing operations, the part silhouette is automatically identified and used as a containment region for the tool. For roughing operations, the stock geometry determines the constraints on the tool location.

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

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

Multiple Regions

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



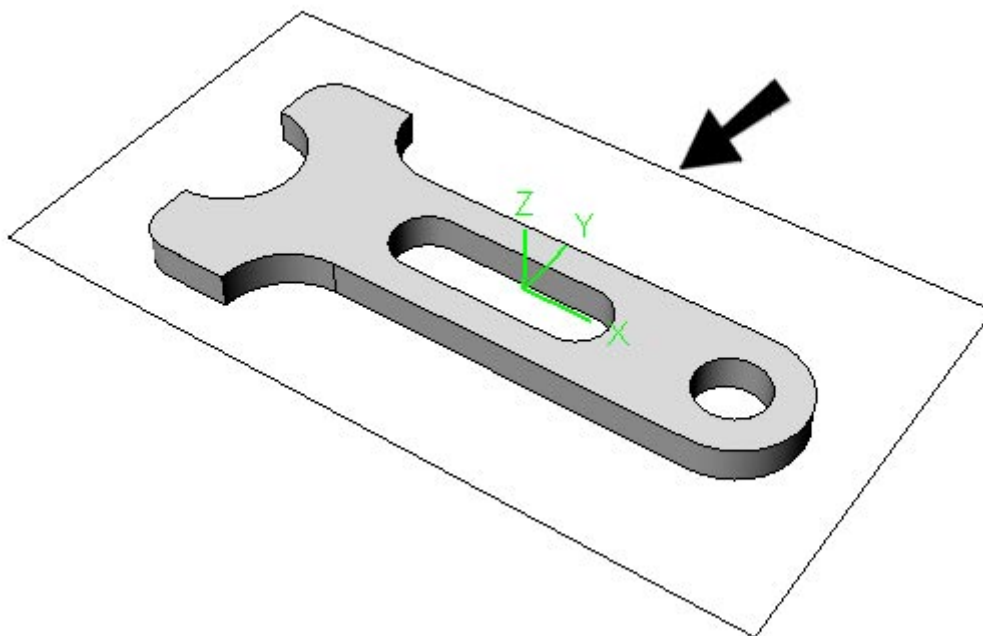
Creating Toolpaths

Four toolpaths will be created – **Facing** to remove the top layer of stock, **Pocketing** for the inner pockets and **Profiling** to finish the outside.



To create the facing toolpath:

1. The first step is to activate the region you want to mill. **Select** the outermost rectangular region in the graphics window by clicking on it.

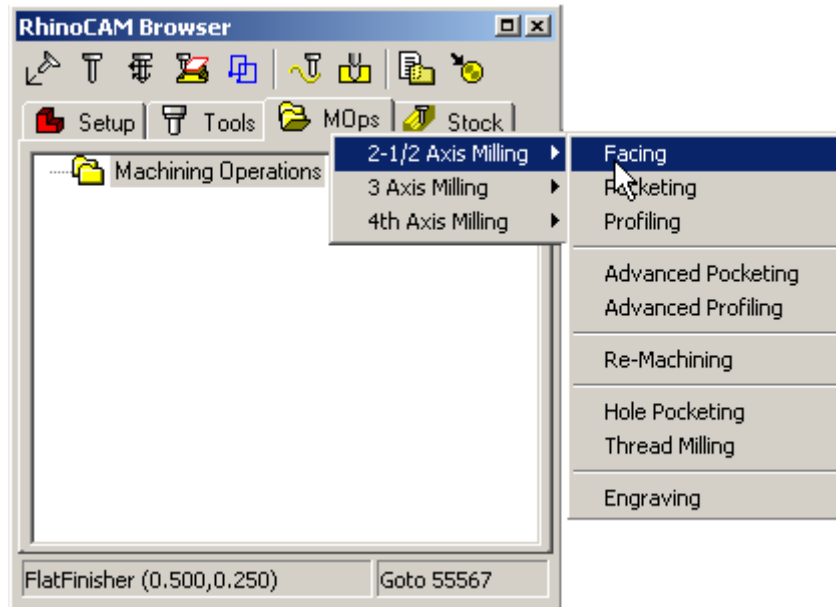


Note: In the **Regions** section of the **Setup** Browser tab, you can see what regions are selected. You can also use the Browser to select and deselect regions.

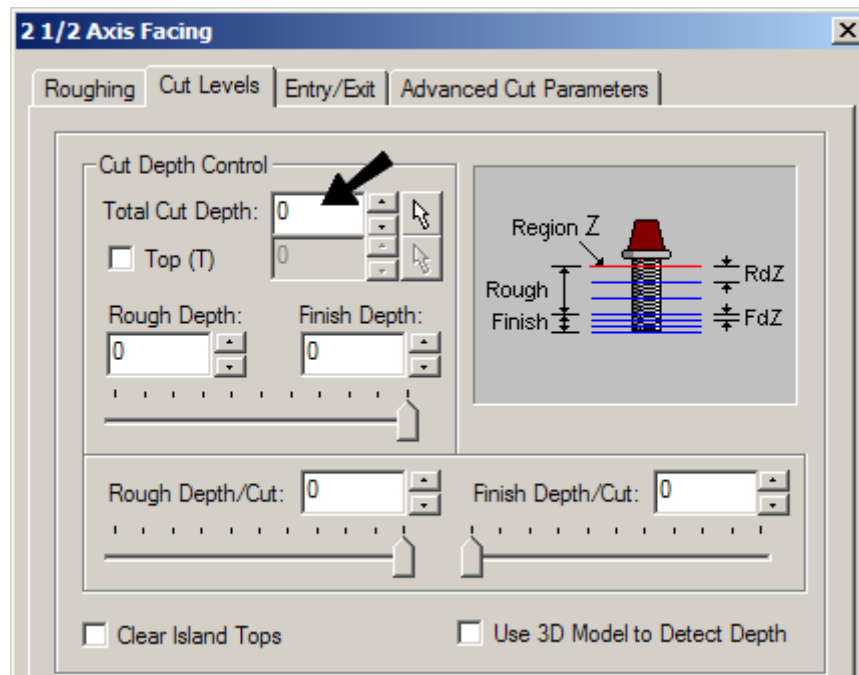
2. Select the **FlatRougher** tool.

Getting Started with RhinoCAM

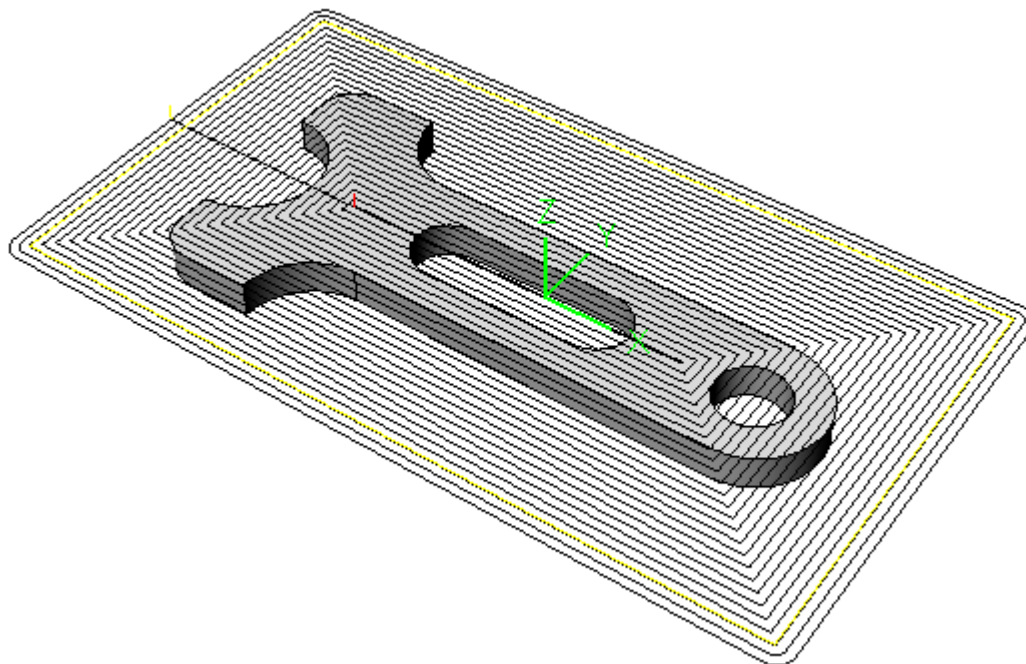
- Once the tool and the region are selected, In the **MOPs** tab select 2-1/2 axis facing operation to clear 0.1 inch off on the top surface of the stock so the stock model is flush with the face of the part.



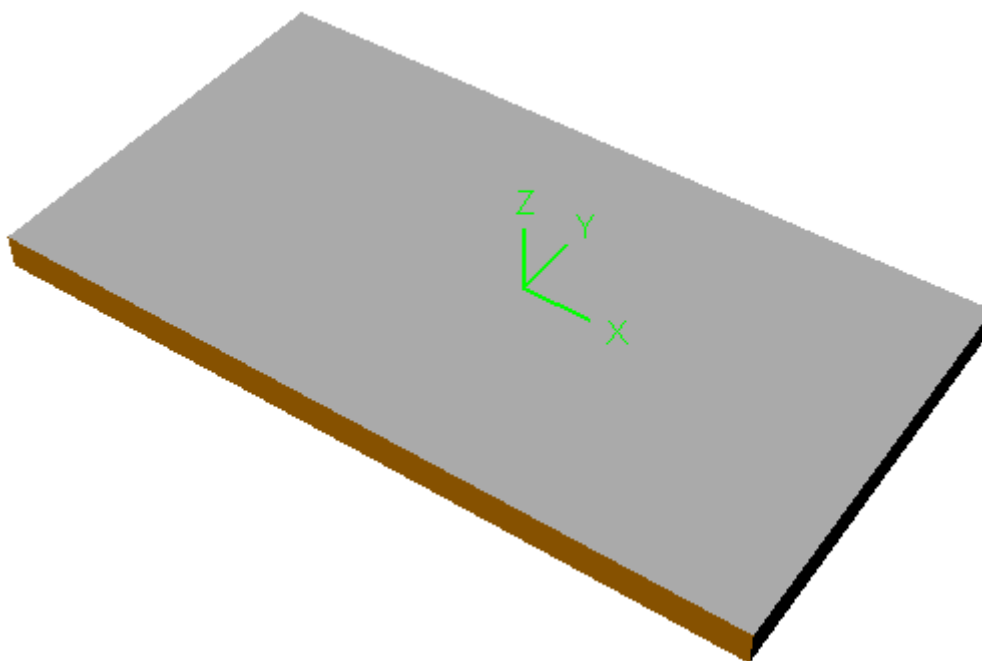
- Select the cut levels tab and set the **Total Cut Depth** to 0. This will create a cut level exactly at the Z value of the selected region and thus exactly flush with the top of the part model.



5. Now press the **Generate** button to create the toolpath. In the shaded mode you will be able to see the tool path as shown:



6. Move to **Stock** tab and press the **Simulate** button to simulate and display the cut stock model.



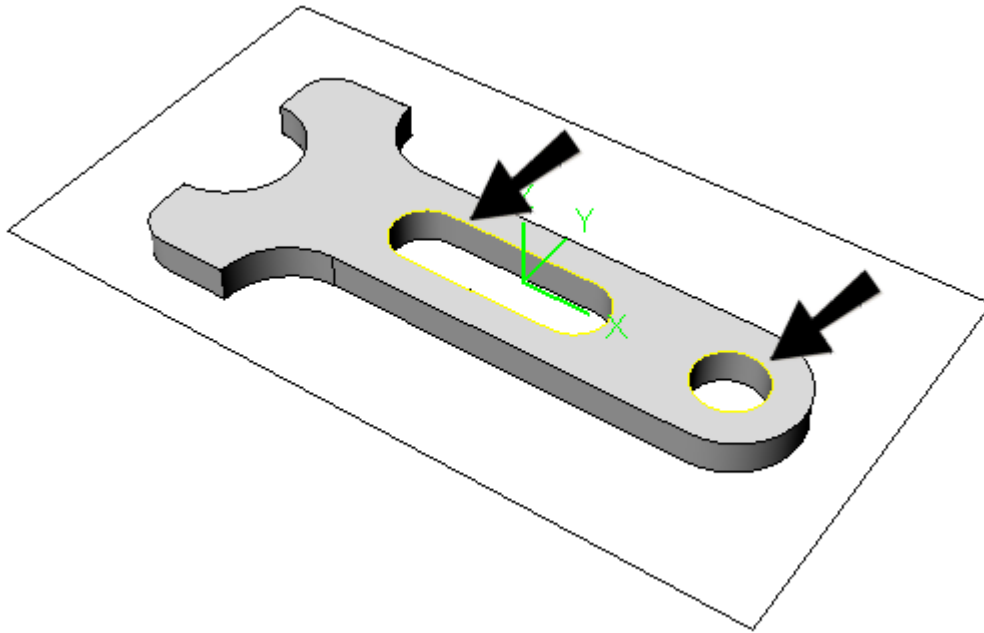
Getting Started with RhinoCAM

*Note: Un-check the **Show Path** check box in the browser to hide the display of the toolpath for better visualization of the simulated stock model.*

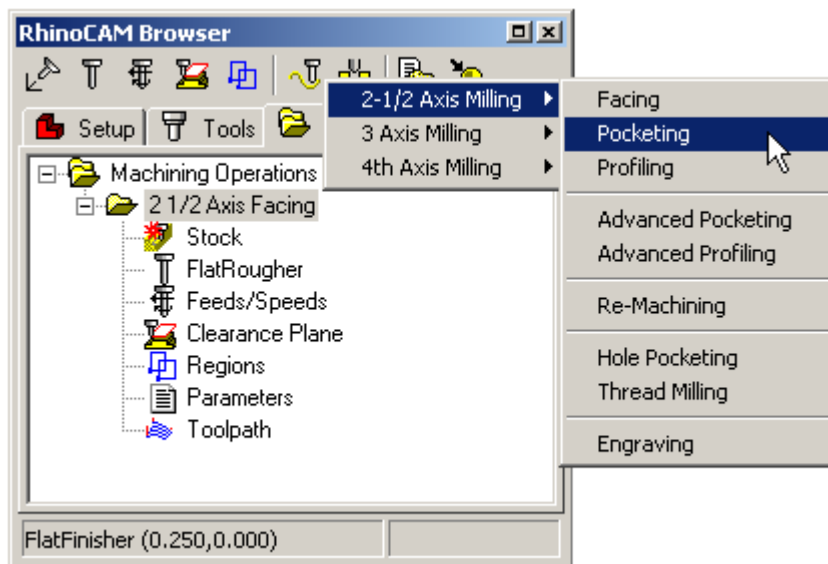


To create the Pocketing toolpath:

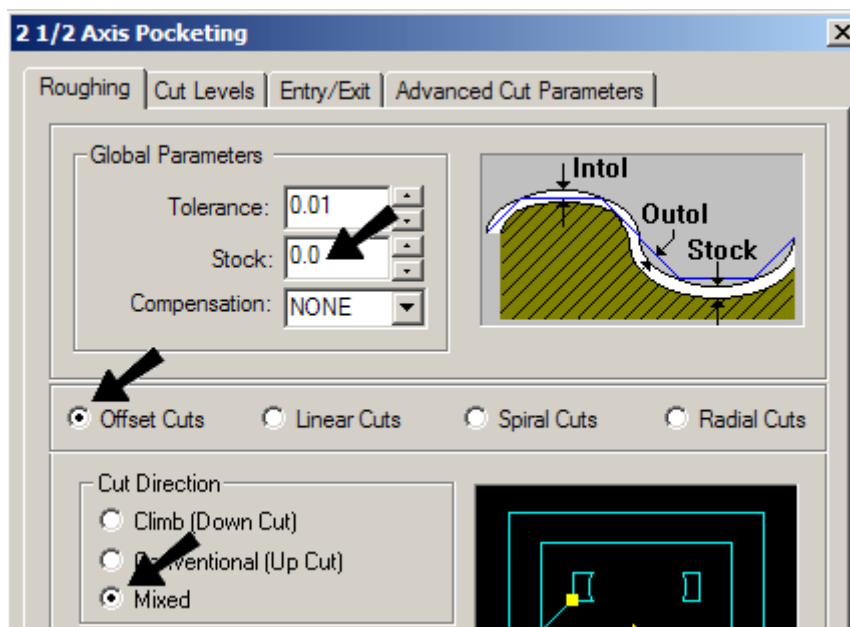
1. Select the two inner regions for creating the pocketing operation



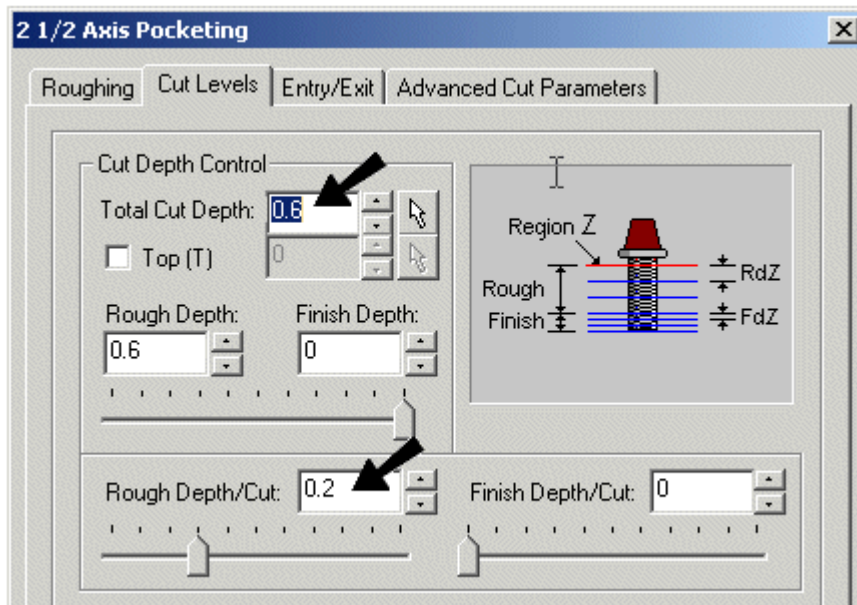
2. Activate the **FlatFinisher** tool
3. Select the **2½ Axis pocketing** machine operations from the **MOPs** tab.



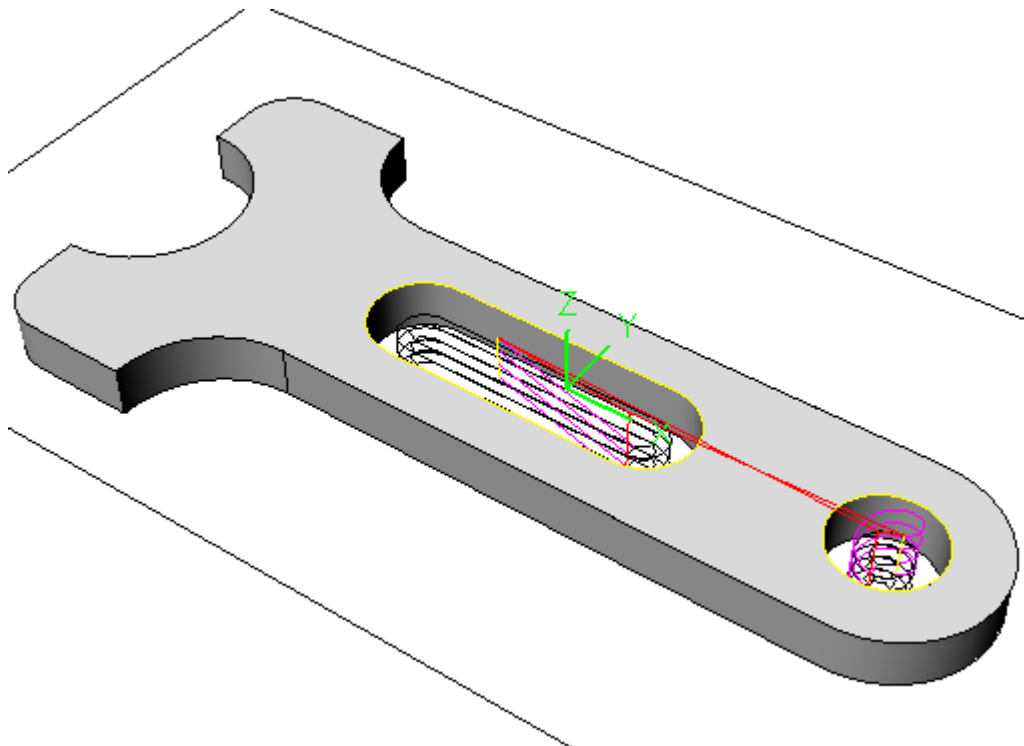
- In the **Roughing** tab, set the **Stock** value to 0.0, check the **Offset Cuts** pattern, and select the **Mixed** cut direction.



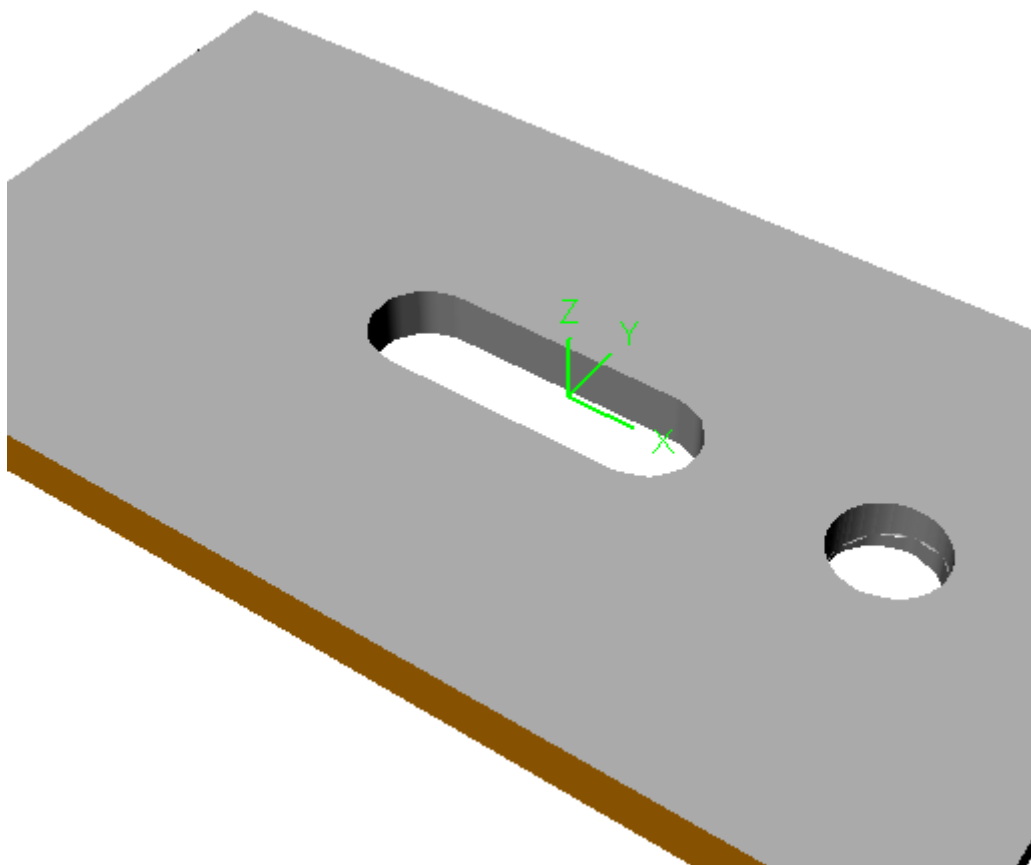
- In the **Cut Levels** tab, set the **Total Cut Depth** as 0.60. This is the thickness of the part geometry. Also set the **Rough Depth/Cut** to 0.2 so that three machining level will be used.



6. Accept the other default parameters and click **Generate**. The pocketing toolpath is generated and displayed as shown.



7. Switch to the **Stock** tab and simulate the pocketing toolpath.

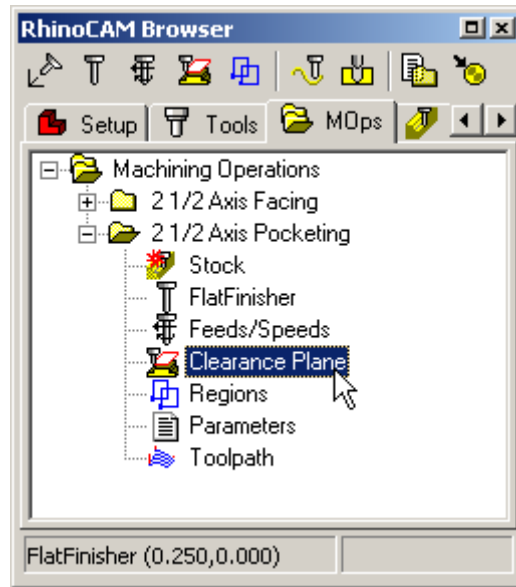


***Note:** To better visualize the simulation model you can un-check the **Show Path** check box in the Browser window to display just the stock model.*

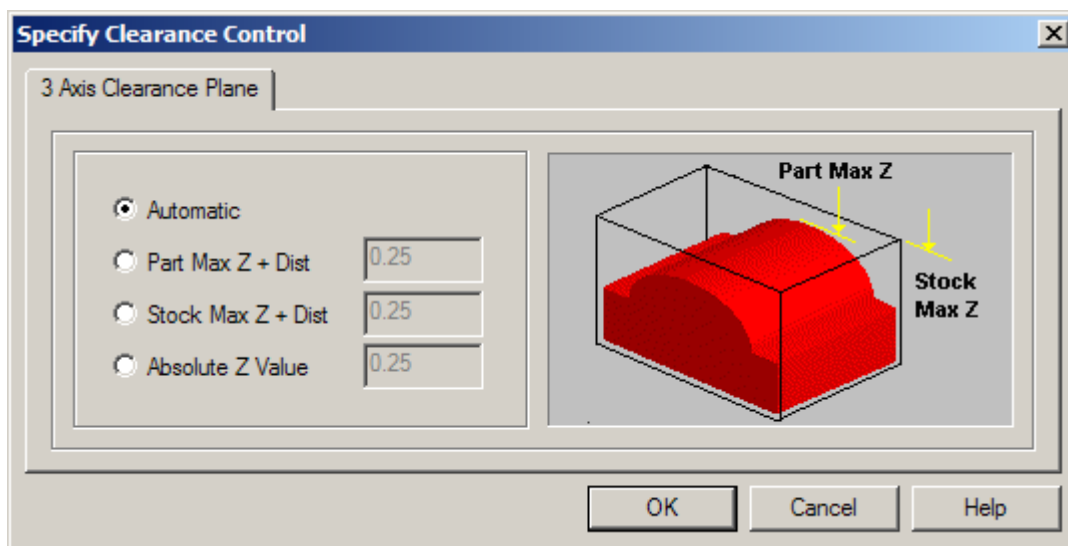
Clearance Plane

The clearance plane is a horizontal plane (set Z level) from which the approach motions start and retract motions end. After retracting, the tool moves rapidly along this plane to the position of the next engage. This plane is typically a certain safe distance above the part geometry.

The clearance plane is defined by double clicking the **Clearance Control** button on the **MOPs** tab.



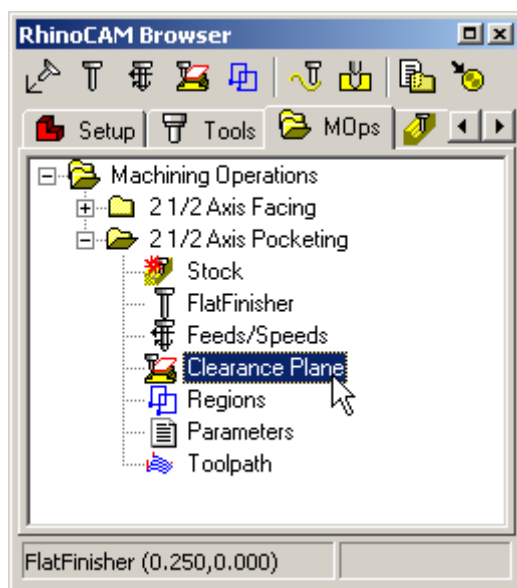
By default (**Automatic** option), the clearance level is calculated by adding a safety distance to the highest point found on both part and stock geometry. This safety distance is set to be the current tool radius. You can set the clearance level to be a set distance from either the part or stock, or enter the absolute Z level.



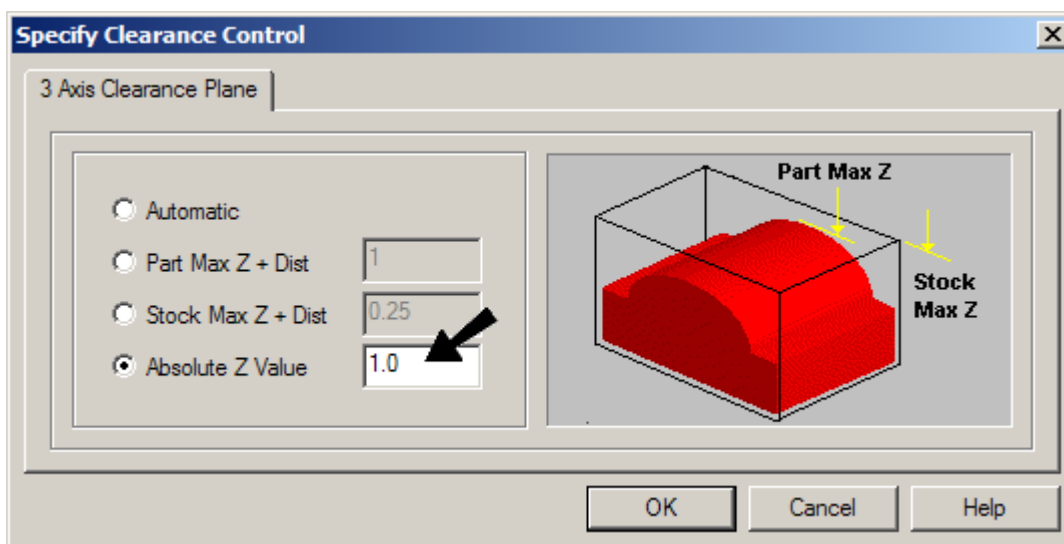


To change the clearance plane for this operation

1. Double-click the **Clearance Plane** icon for this operation in the MOPs tab of the browser.

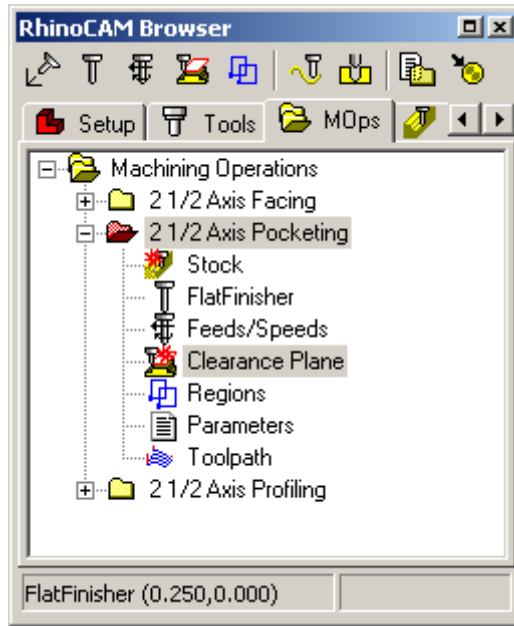


2. Switch from the default **Automatic** to **Absolute Z Value** and set it to 1.0. This will set the clearance plane at the Z value of 1.0 with respect to the World Coordinate System.

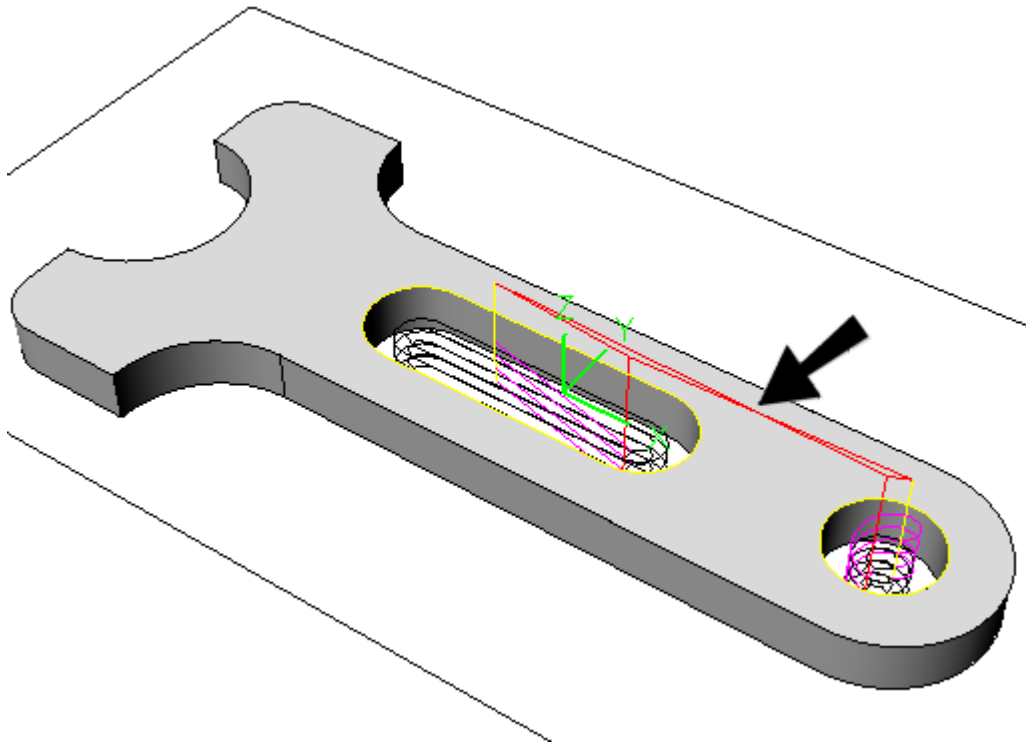


3. Click **OK**. In the Browser, note that the folder icon for this toolpath is red instead of yellow. This indicates that a parameter has changed, and the toolpath needs to be regenerated.

Getting Started with RhinoCAM



4. Right-click on the toolpath name and select **Generate** and you will notice that the toolpath's retract and transfer moves have changed to the new clearance plane.

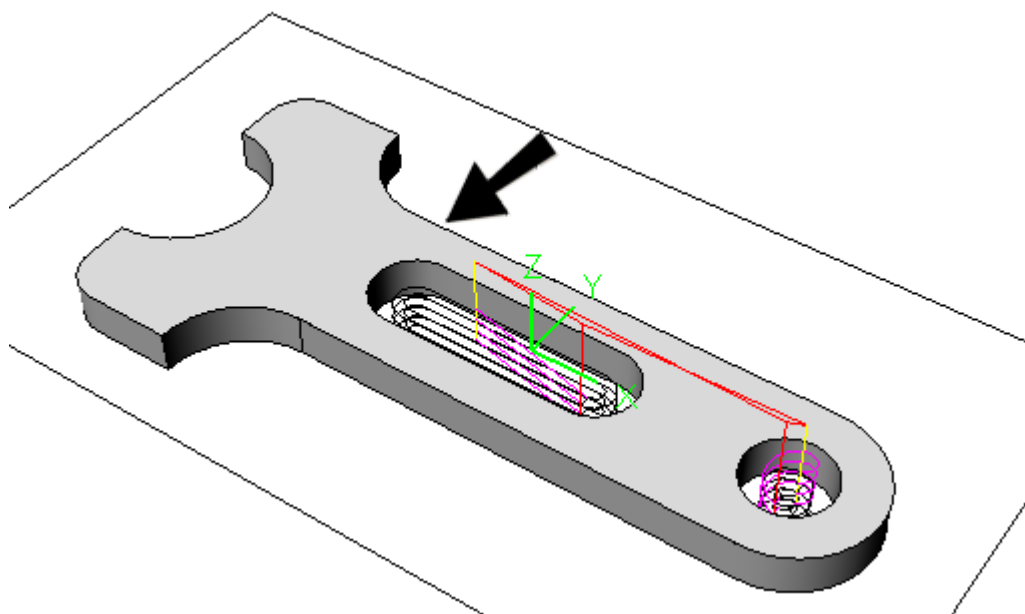




To create the Profiling toolpath:

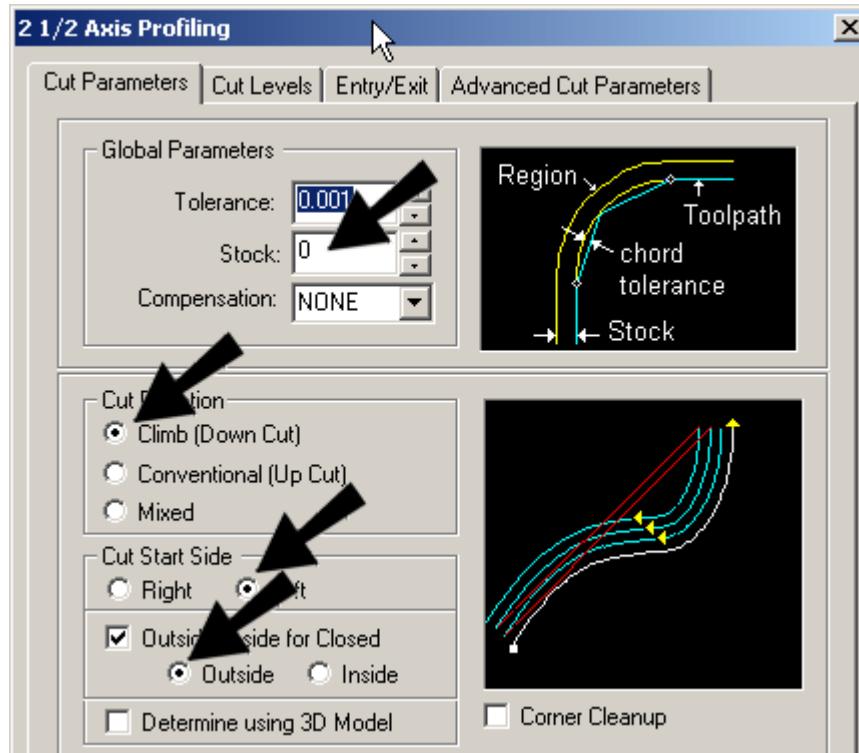
This **Profiling** toolpath uses the same tool as the pocketing toolpath that you just created. You will be using this toolpath to finish the outsides of the part.

1. Switch to the Stock tab to make sure the curves in the part are displayed.
2. Using the graphical selection tool of Rhino single select the curve displayed below.

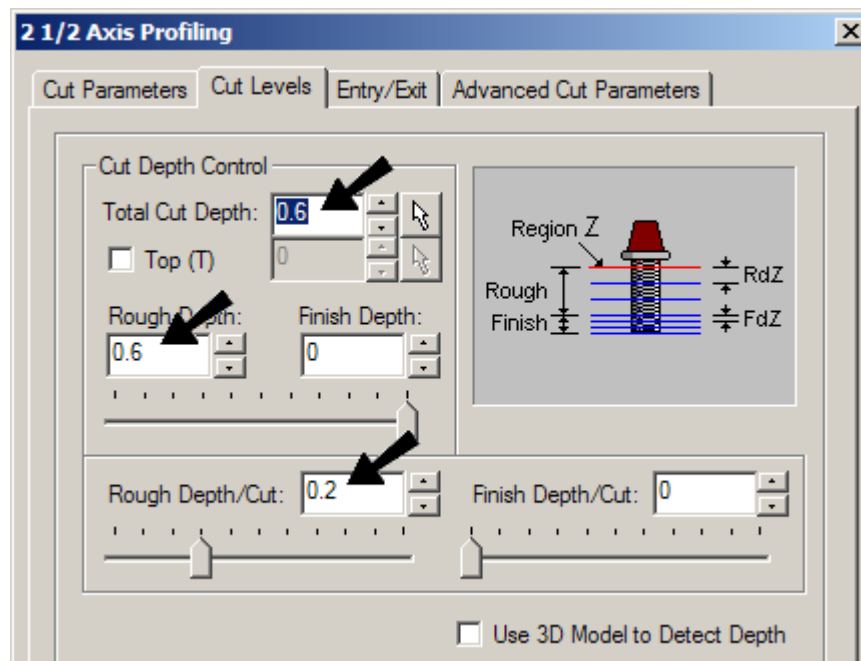


3. The **FlatFinisher** tool should still be active. If it is not, select this tool.
4. Select the **Profiling** cut method from **2½ Axis** cut methods menu in the MOps tab toolbar. In the Parameters tab make sure **Stock** value is set to 0, cut direction is set to **Climb** and the **Cutting Side** is set to **Left** as shown below. Also make sure the button **for Outside/Inside for Closed** is checked and the cut side is chosen as **Outside** as shown. These settings will ensure that in the case of
 - a. Open curves, the cutter will always cut to the left of the curve
 - b. Closed curves, the cutter will always cut on the outside of the curve

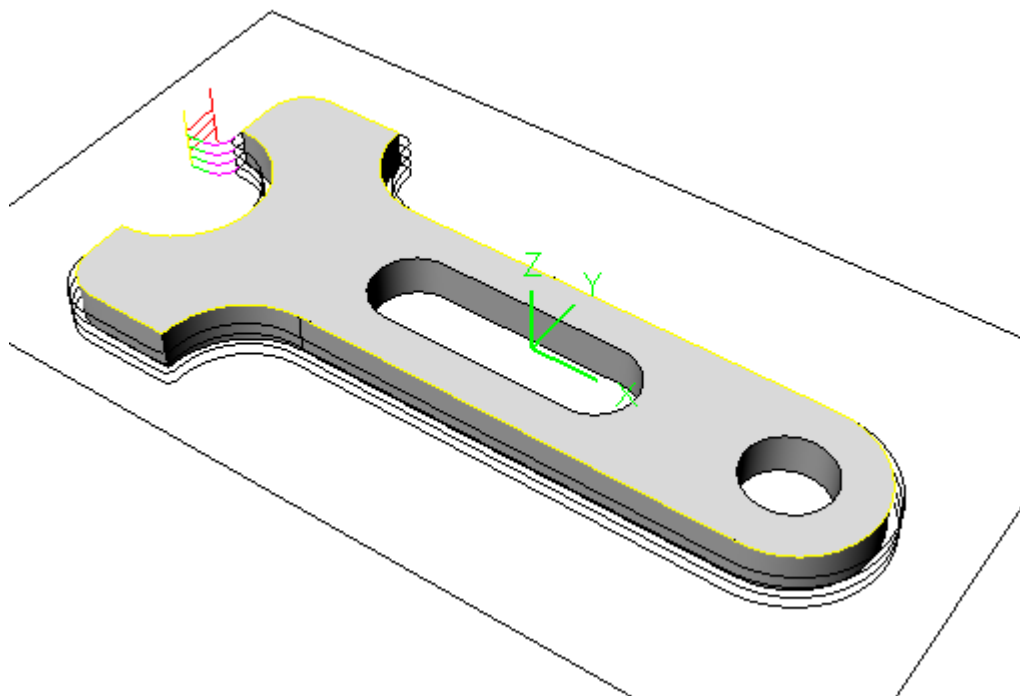
Getting Started with RhinoCAM



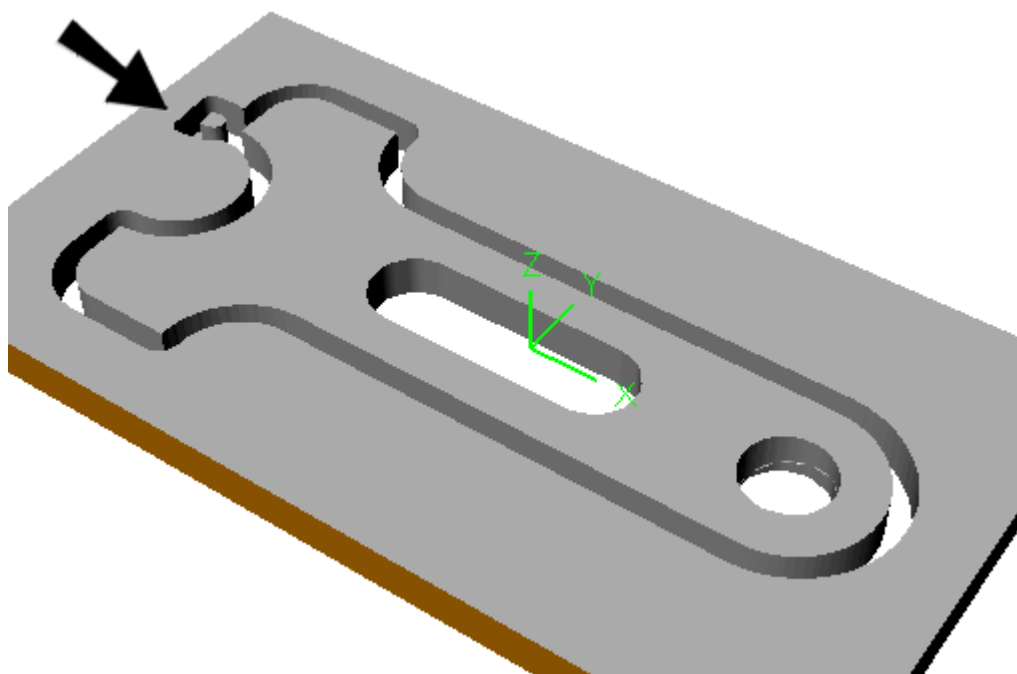
5. Click on the **Cut Levels** tab, and enter the parameters as shown below. This will create a toolpath from the top of the part to the bottom, using three cut levels each spaced 0.2 inches apart as with the previous pocketing toolpath.



6. Accept the other default parameters and click **Generate**. The profiling toolpath is generated and displayed as shown.

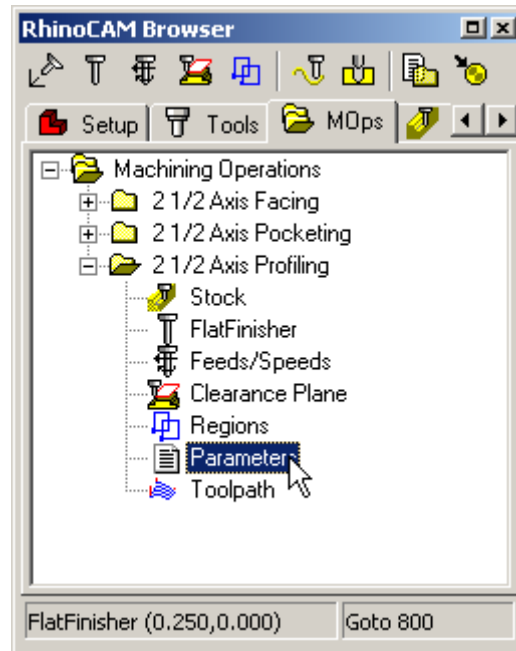


7. Simulate the toolpath using Stock tab **Simulate** button.

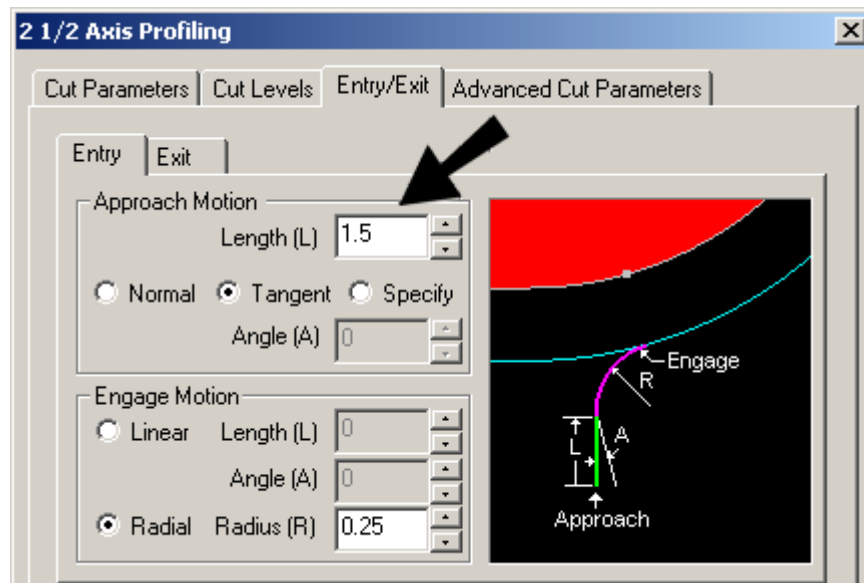


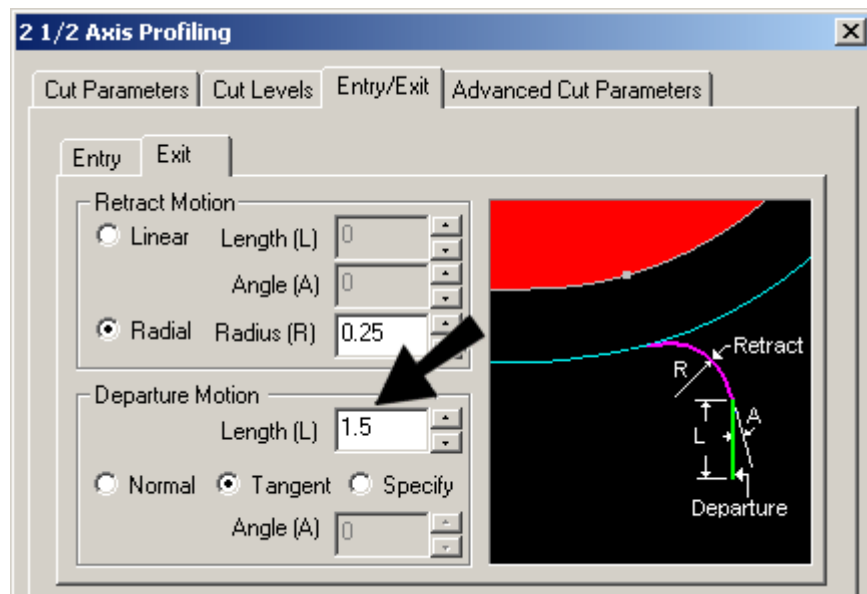
Getting Started with RhinoCAM

- Notice where the cutter engages into material when it starts cutting as shown by the arrow. This is an undesirable situation since a flat tool will plunge straight down into material and contact it with the bottom that does not have a cutting edge. We have to modify the entry and exit of the cutter such that the cutter plunges outside of material and starts cutting with the side of the cutter. To do this we edit the parameters of the operation by double clicking on the **Parameters** icon in the browser corresponding to this machining operation.

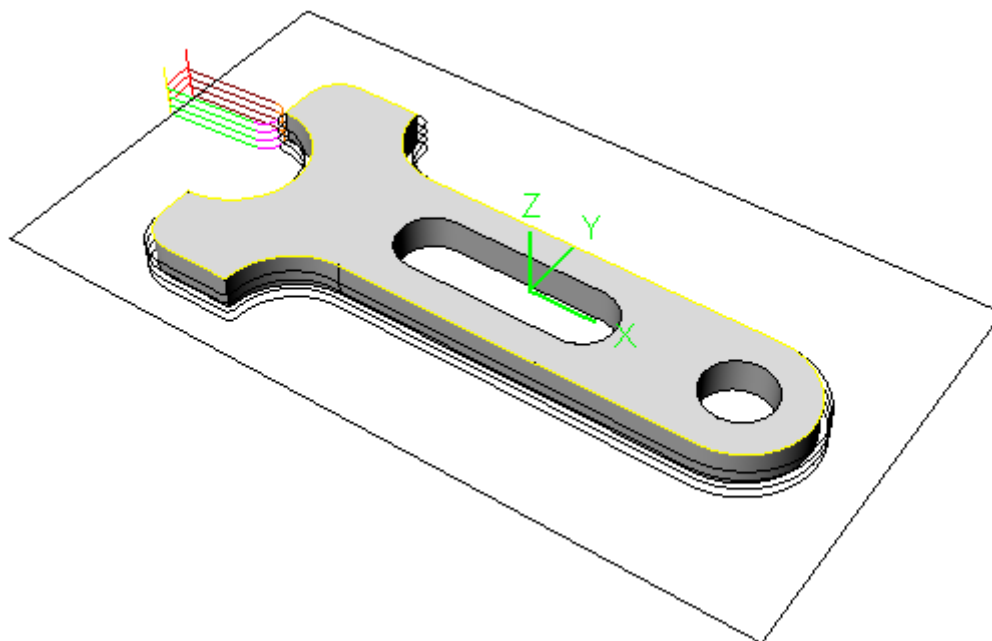


- This will bring up the parameters dialog. Switch to the **Entry/Exit** tab in the dialog and set the **Approach** motion length as 1.5 inches in the **Entry** section of the dialog. In the **Exit** section set the **Departure** motion length as 1.5 inches and generate the toolpath.



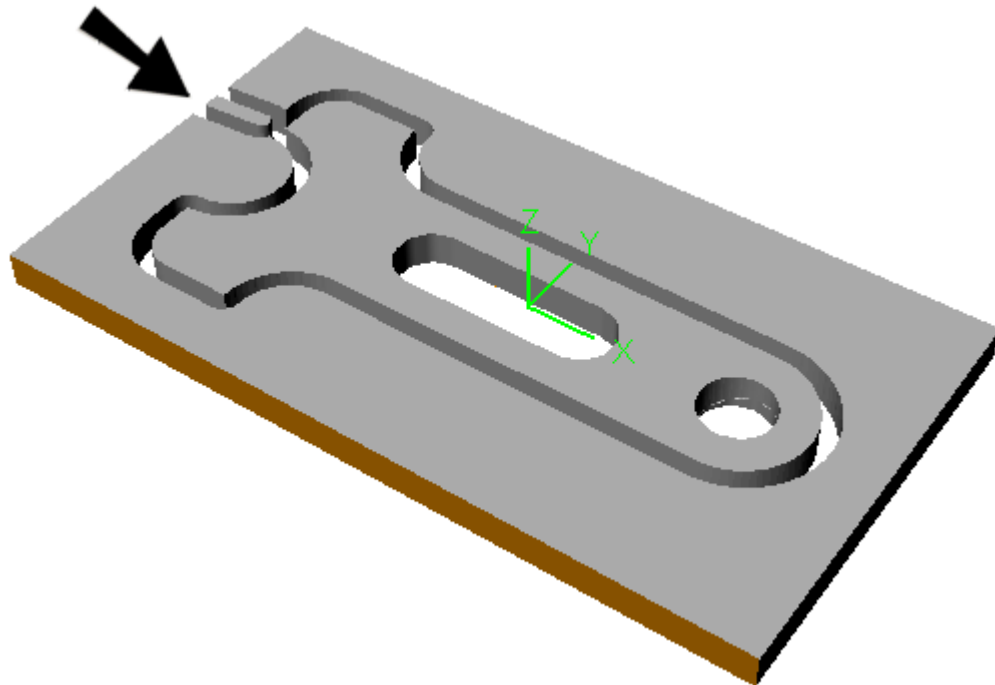


10. This will create the edited toolpath as shown below. Notice that the cutter enters and exits outside the material as desired.



11. Run the simulation from the Stock tab to make sure that the cutter does indeed enter and exit on the outside of the material.

Getting Started with RhinoCAM



12. Now you can post the toolpath as described in Tutorial 1.

End of Tutorial 2!

Tutorial 3: 3 Axis Drilling Operations

This exercise will help you understand how to use the drilling module in RhinoCAM. The following types of drilling operations are available:

Drill: Standard, Deep, Break chip, Counter Sink

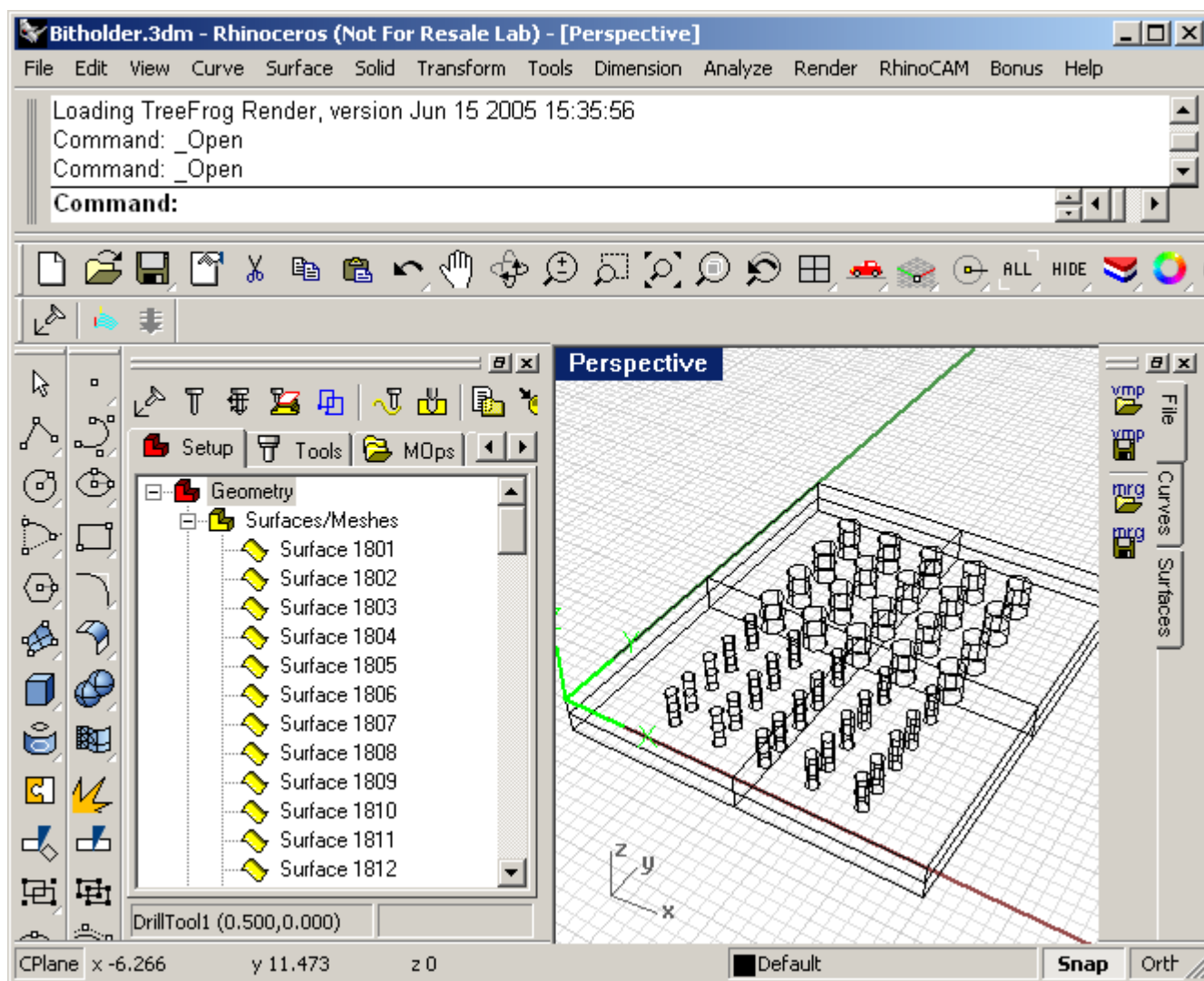
Tap: Clockwise, Counter Clockwise

Bore: Drag, No Drag, and Manual

Reverse Bore



Load the part file, **BitHolder.3DM** to get started for this tutorial.



Regions in Drilling

In Hole making operations, regions are used to determine the location of the hole. As in 2½ axis milling regions act as drive curves. However the regions are used to derive the location of the holes based on the type of regions selected for the operations. If points are selected the location of the points determine the location of the hole. If closed curves such as circles are selected, then the center of these curves are computed and used as the holes' location. Open curves can also be selected, in which case the start point of the curves are used as the location of the holes.

Creating Tools

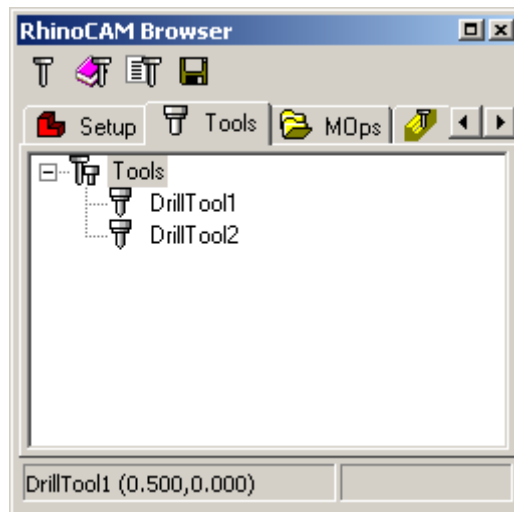
Before any operations can be performed, you must define the drilling tools that will be used in the operation. There are two sets of holes in this file that we want to drill. One set is 0.5 inches in diameter and the other set is 0.25 inches in diameter. So we will create 2 drilling tools one for each size of hole.

1. Create and save the following two drilling tools:

Drill tool named **DrillTool1** with a diameter of 0.5 inches

Drill tool named **DrillTool2** with a diameter of 0.25 inches

2. Open the **Tools** tab of the Browser to make sure the created are tools listed.



3. Select the DrillTool1 as the active tool using the **Create/Select Tool** from the **Tools** tab toolbar button.

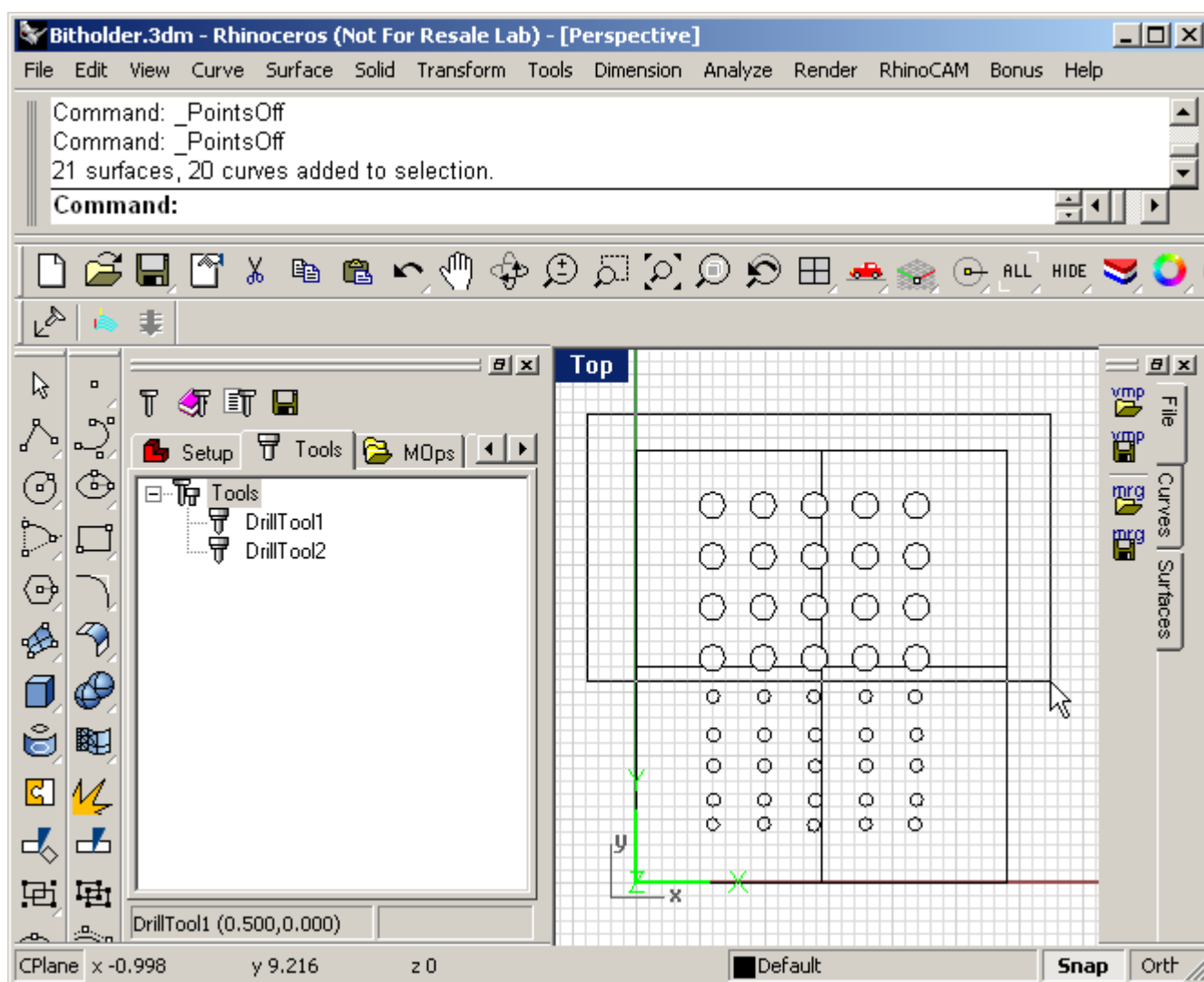
Creating Toolpaths

We will create two operations, the first to drill the larger holes that are of 0.5 inches in diameter and then the second operation to drill the smaller holes of 0.25 inch diameter.



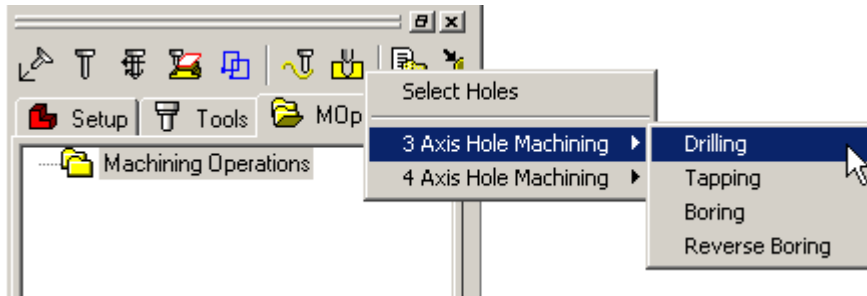
To drill the larger holes:

1. Now first select the group of 0.5 inch diameter holes to drill. The easiest way to do this would be to switch to the top view and using the rectangle selection mode in Rhino to graphically select the larger holes as shown below.

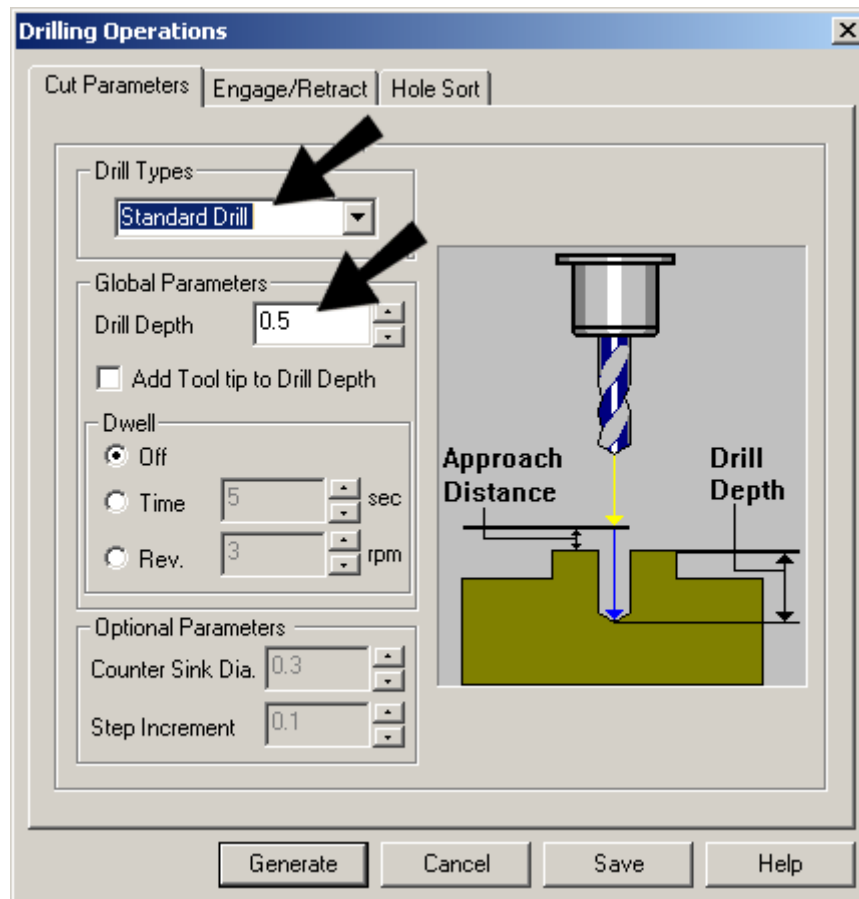


Getting Started with RhinoCAM

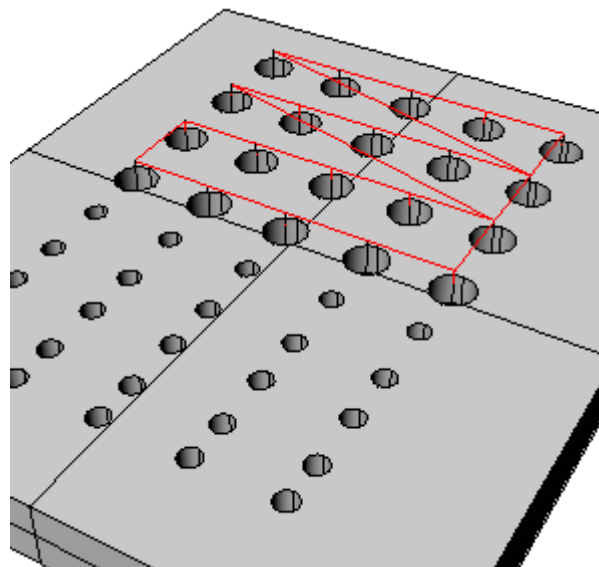
2. Switch to the **MOPs** tab and select drilling operation from the drilling operations menu.



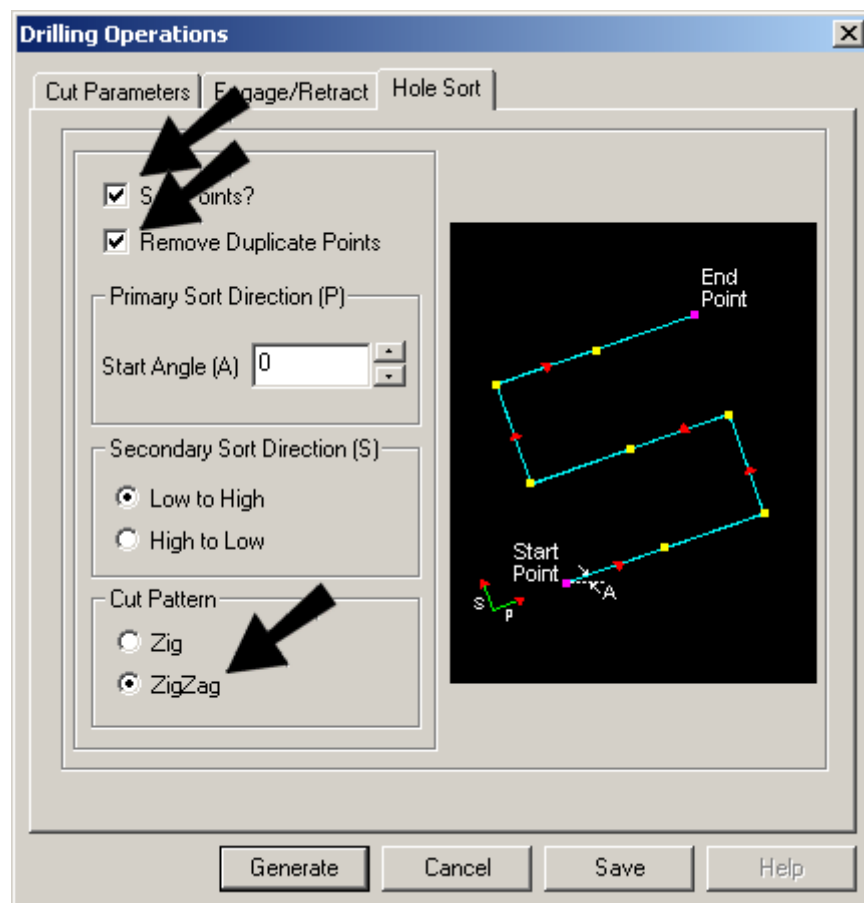
3. Pick the drill type as **Standard Drill** and specify **0.5** as the drill depth.



4. Keep the rest of the parameters as the default values and press **Generate** to generate the toolpath.

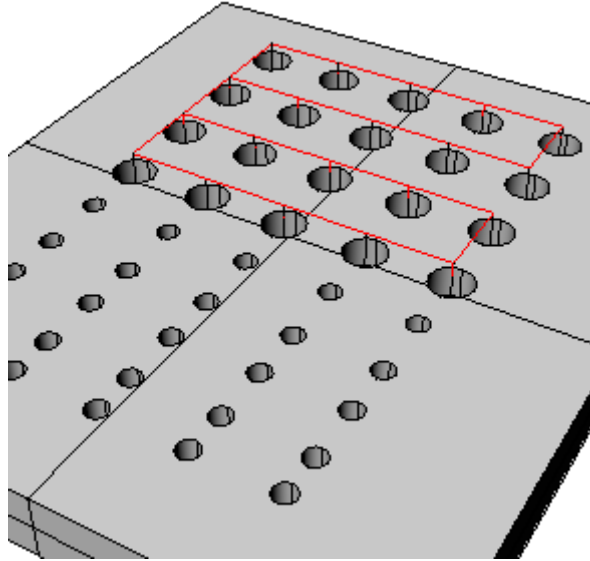


5. You can also use the **Hole Sort** option and change the sequence of the toolpath so as the tool motions are optimized. To do this double-click the **Parameters** icon of the just created drilling operation and choose the Hole Sort tab and select the indicated parameters and press **Generate**.



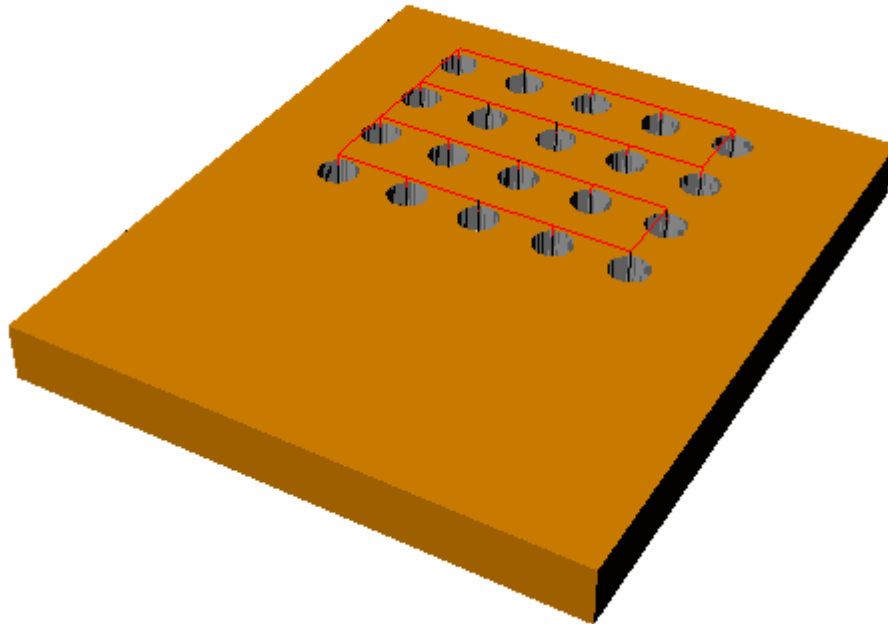
Getting Started with RhinoCAM

A modified toolpath will be generated as shown below:



You will notice that the toolpath travels parallel to the X axis and has a zigzag pattern.

6. In the **Stock** tab, create a **Part Box Stock** model and select the **Simulate** button in the toolbar to simulate the Drilling toolpath.

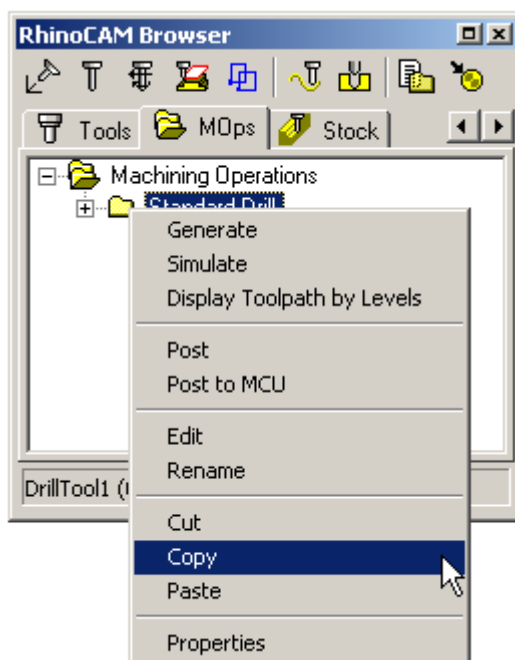




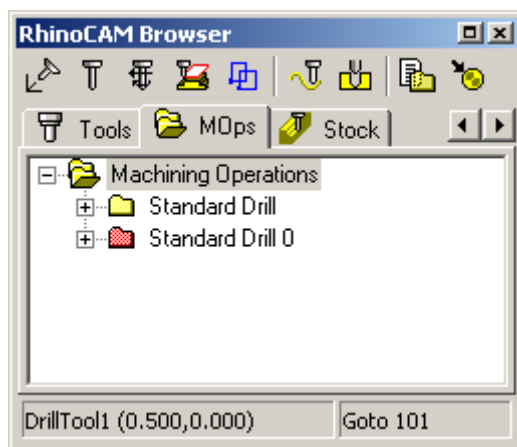
To drill the smaller holes:

There is an easy way to repeat the previous drill operations for drilling smaller holes using the smaller drill tool and selecting new regions.

1. Right-click on the first drill operation that was just created. This will bring up a menu that is shown.



2. Use the copy and paste buttons to copy the created operation to a new drill. A second drill operation is created as shown.

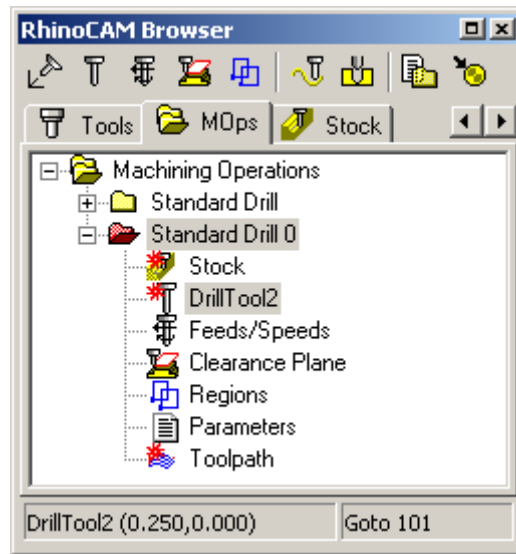


Getting Started with RhinoCAM

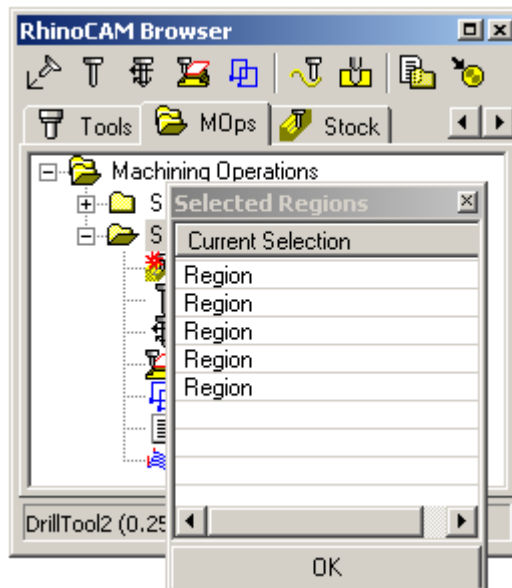
You will notice the icon representing the second drill operation is in red color indicating that the toolpath needs to be generated. We now need to modify the tool and the regions in this operation to create the drilling operations for the smaller holes.

3. Expand the folder for the second drill operation by clicking on the + sign and double-click on **DrillTool1** icon. This will bring up the tool selection dialog. Select **DrillTool2**, the 0.25 dia drill bit.

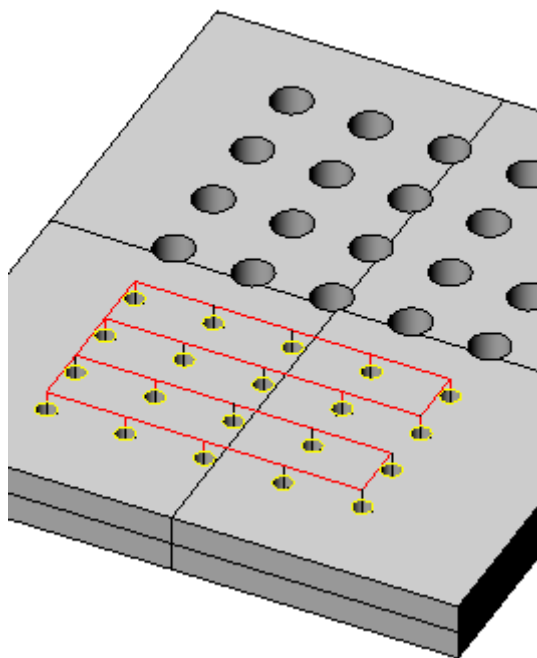
After the selecting the smaller drill, you will see that the tool has been changed in the second machining operation to DrillTool2.



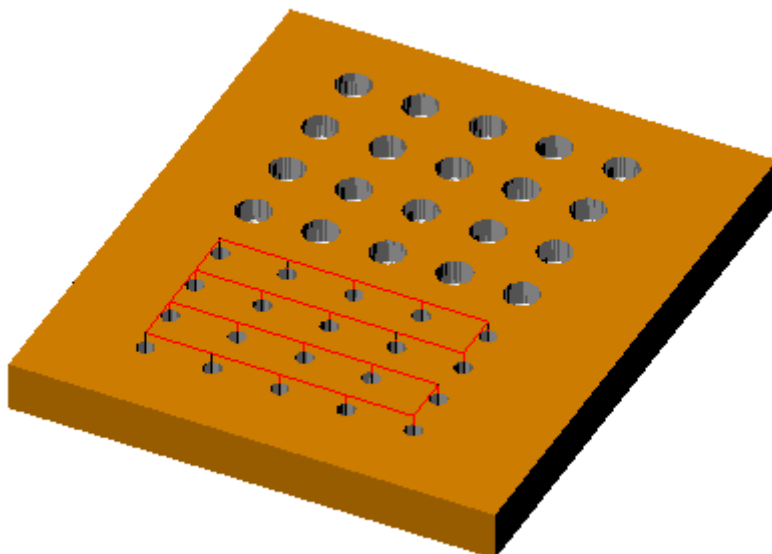
4. Now delete the previously used regions for the larger holes and select the ones representing the smaller holes. Double-click the **Regions** icon of the operation.



5. Now deselect all the regions by clicking on open area outside the part geometry in the graphics view port and all the regions will disappear in the Regions dialog window. You can select all of the smaller holes using Rhino rectangle selection tool as done before with the larger holes.
6. You can keep all the same drilling parameters used in the first operation and generate the toolpath for drilling smaller diameter holes by right clicking on the Mop icon for the second drilling operation and selecting the **Generate** button.



7. In the **Stock** tab, press the **Play** button to simulate the Drilling toolpath.



Getting Started with RhinoCAM

8. You can rename these operations as Large Drill and Small Drill for clarity by Right-clicking and selecting the **Rename** option

You can proceed to use the same selections to generate **Tap**, **Bore** and **Reverse Bore**, by creating the corresponding tools before creating each of these types of toolpaths.

End of Tutorial 3!

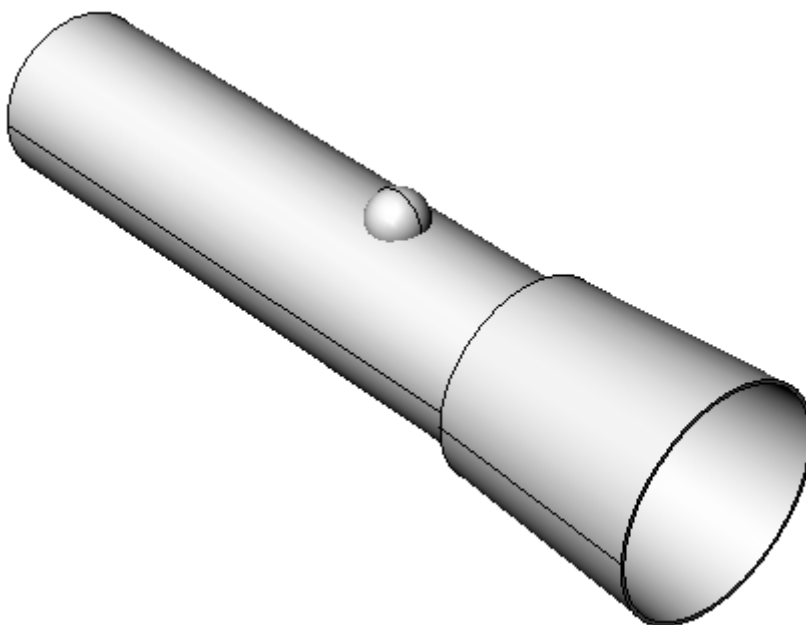
4th Axis Machining Operations

The following section of this guide describes tutorials to help the user get started with 4 Axis machining operations.

***Note:** The 4th axis functionality is an **optional** add-on module that the user can purchase separately to augment the functionality of the base RhinoCAM plug-in product.*

Fourth Axis operations are used to machine parts that cannot be machined with simple 2 1/2 axis or 3 axis machining operations. Since the tool moves only up and down along the Z axis during 3-Axis milling, areas that cannot be seen from above cannot be cut. In such cases, the object could be divided into top and bottom sections and machined separately. RhinoCAM's implementation of indexed fourth axis milling allows the user to do this automatically if the machine tool is equipped with a rotating head or table. Indexing refers to the ability of rotating the part about the X or Y axis and then performing machining in a 3 Axis fashion with the part locked at this new orientation.

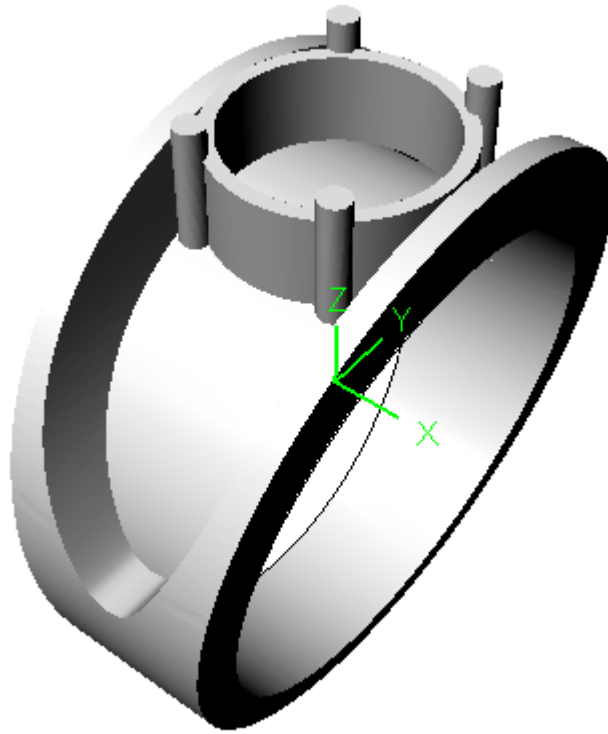
Shown below is a typical part that would require 4 axis milling operations to manufacture the part. Notice that the part has undercut geometry when looking down from the top. This means that there are areas of the part which cannot be reached by a straight tool moving purely up and down. To successfully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



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

Getting Started with RhinoCAM

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



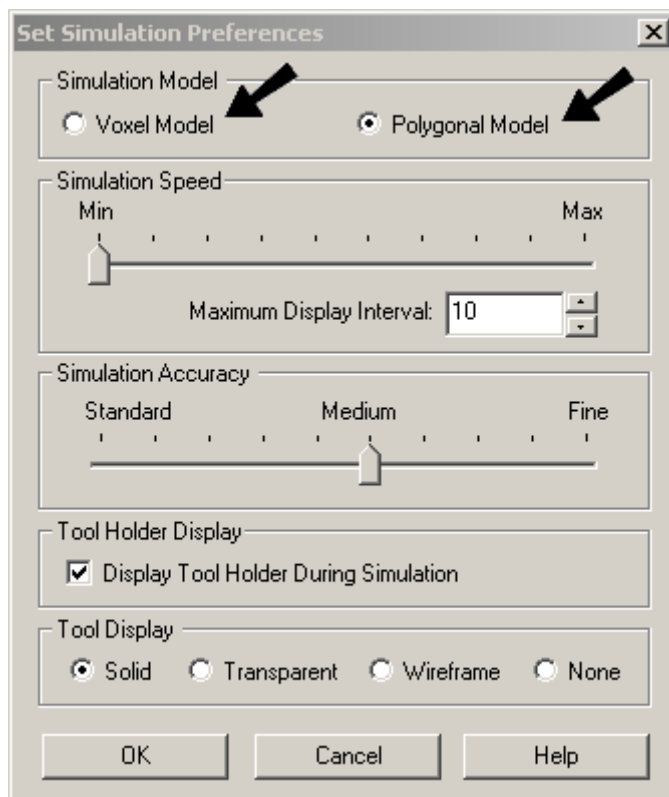
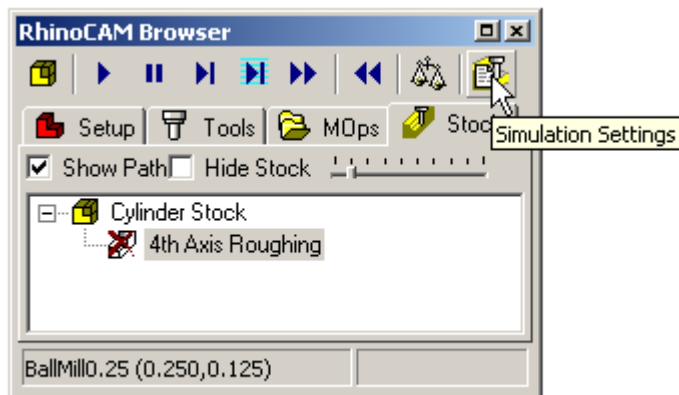
The part shown above is a design of a ring. It can be machined using RhinoCAM's continuous 4th axis machining mode. In the continuous mode, the tool axis is always pointed normal to the rotation axis of the machine tool. In this particular case, the rotary axis of the 4th axis table will be about the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

Simulation during 4Axis Machining

The RhinoCAM 4th Axis add-on product has two material removal simulation modes (or models) available for use. The **Voxel** model can only be used for 2½ and 3 axis toolpaths. The main advantage of this model is very rapid processing times. The **Polygonal** model is necessary for simulating all 4th axis toolpaths. This model is more comprehensive and results in better display quality, however at the expense of speed.

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

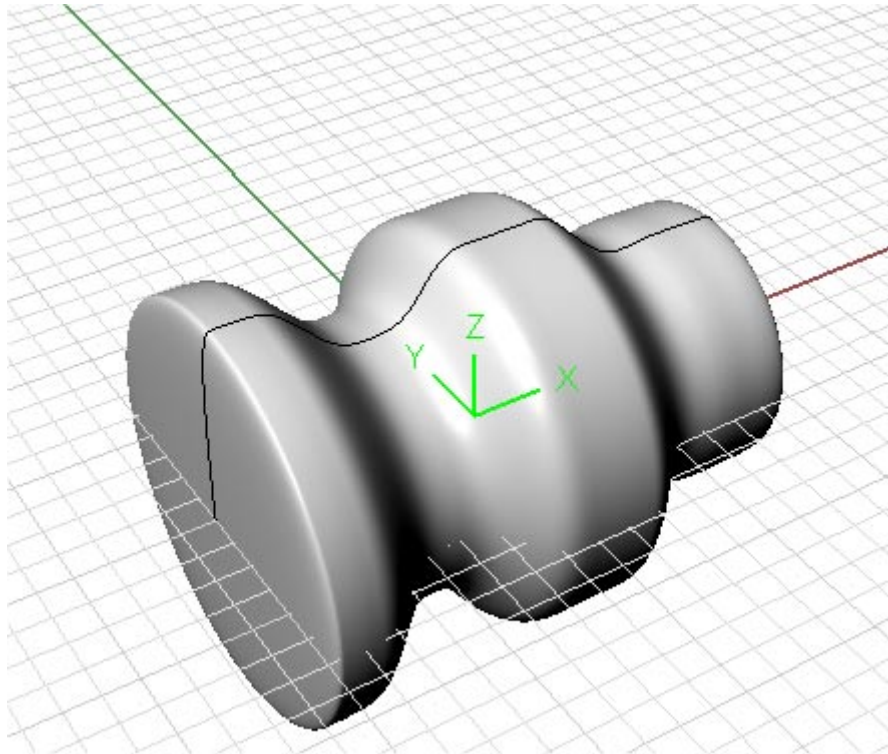
To change the simulation module in RhinoCAM switch to the **Stock** tab in the browser and select the **Simulation Settings** button and select the appropriate model for simulation.



Tutorial 4: 4th Axis Roughing

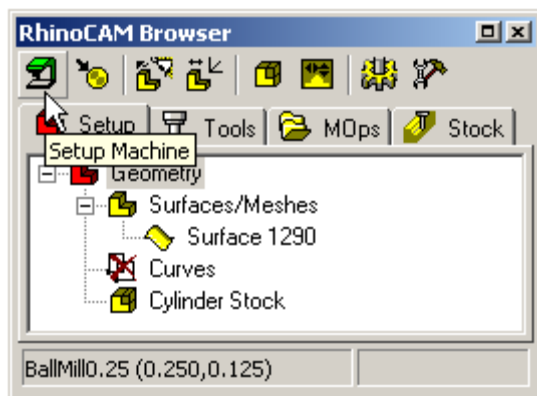
In this tutorial we will use the **4 Axis Parallel Roughing** operation to rough a typical 4 axis part. In this cut method, the tool cuts the stock in successive levels. The spacing between these levels are specified by the user. The stock model after first roughing operation will be as shown below:

Load the starting file **4thAxisRoughing.3dm**. The geometry in the file should like this.

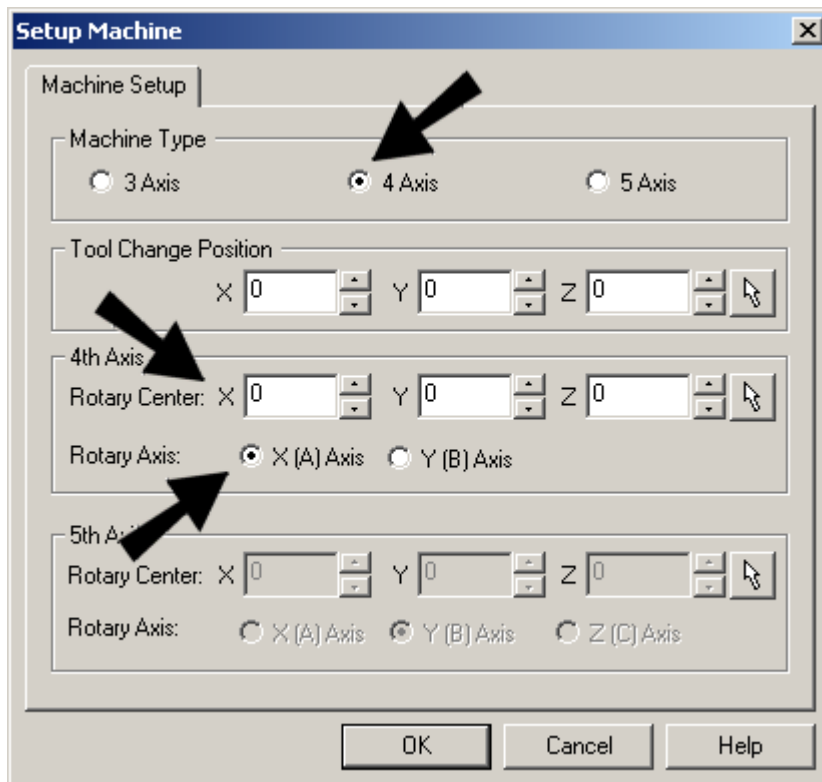


This geometry is a surface of revolution that can be roughed and finished in one 4th axis setup. Care should be taken to hold the part at the ends. Also notice that the machining coordinate system is positioned in the center of the part.

1. Specify the rotary axis of the 4th axis machine to be aligned with the X axis of the machining coordinate system. To do this select the **Setup** tab in the browser window and select **Setup Machine** button on the toolbar.



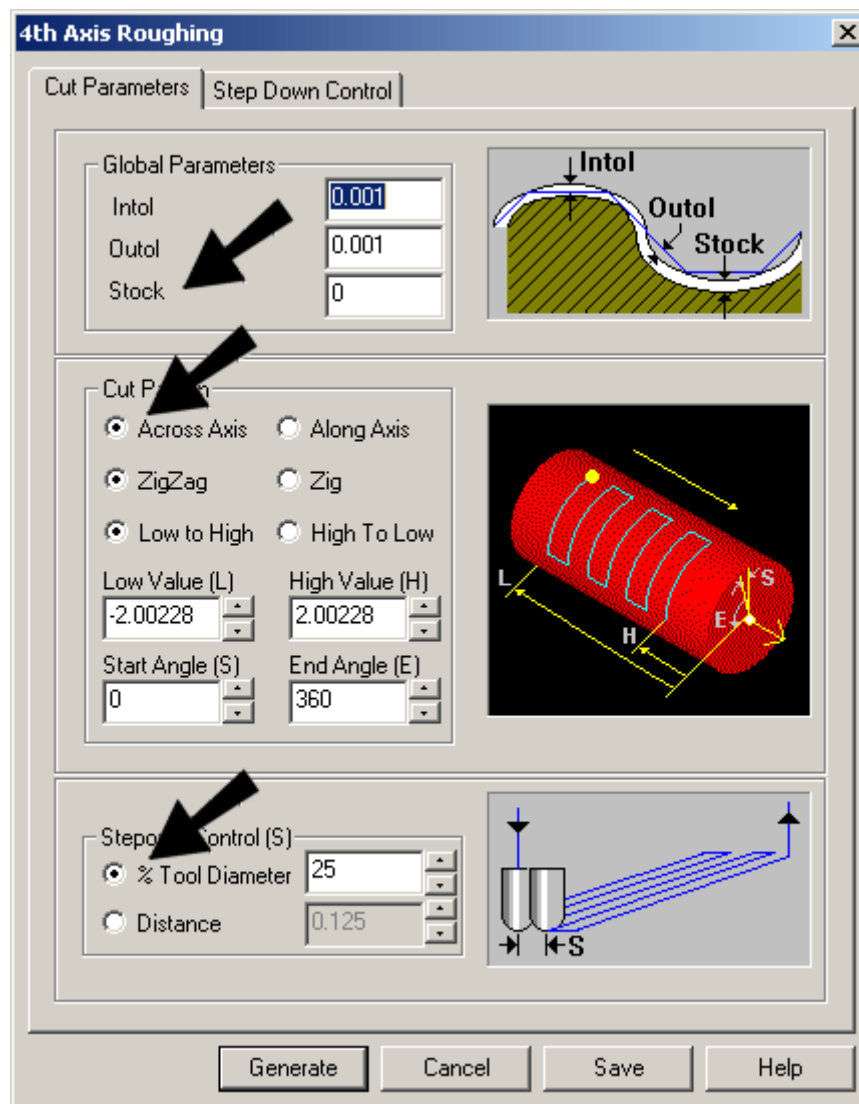
2. In the dialog that comes up make sure the 4th axis **Machine Type** is selected and the Rotary Center and Rotary Axis have the settings as shown below.



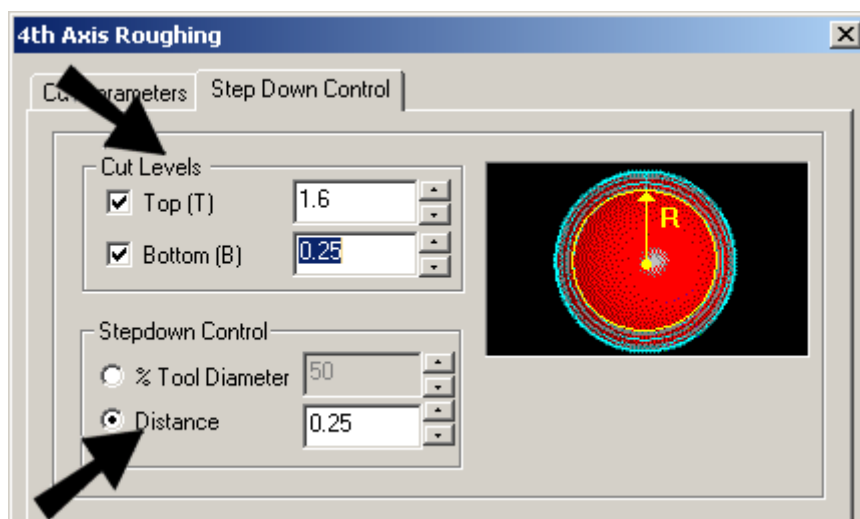
3. Next we create a cylindrical stock model, by selecting the **Cylinder Stock** option and choosing the default values. This will create a cylindrical stock model that encloses the part geometry completely.
4. Next elect a Roughing tool **0.25 Ball mill** that is already saved with the file and select **4th axis Roughing** cut method in the MOPs tab

Getting Started with RhinoCAM

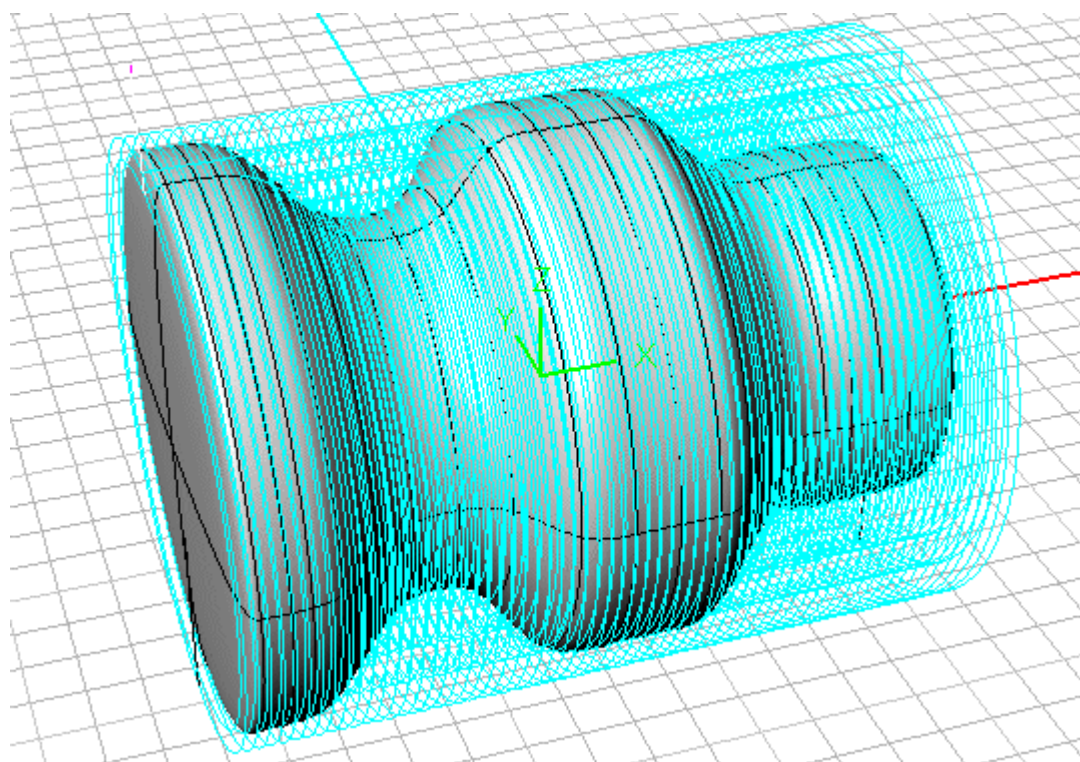
5. Set the values in the cut control dialog box as shown below. The intol and outol control the accuracy of the cutter path. The stock describes any thickness that needs to be left on the final pass. In this case we leave the stock as 0. The cut pattern we choose is used to cut across the rotary axis in a Zig Zag fashion, traversing such that we will go from a low X value to a high X value. These low X and high X values are set in the dialog as shown. The cutter also will travel the entire circumference of the part from 0 degrees to 360 degrees of table rotation for each cut. The step-over between each cut pass is set to be 25% of the tool diameter or 0.125 inches in the case of the cutter being used in the current operation.



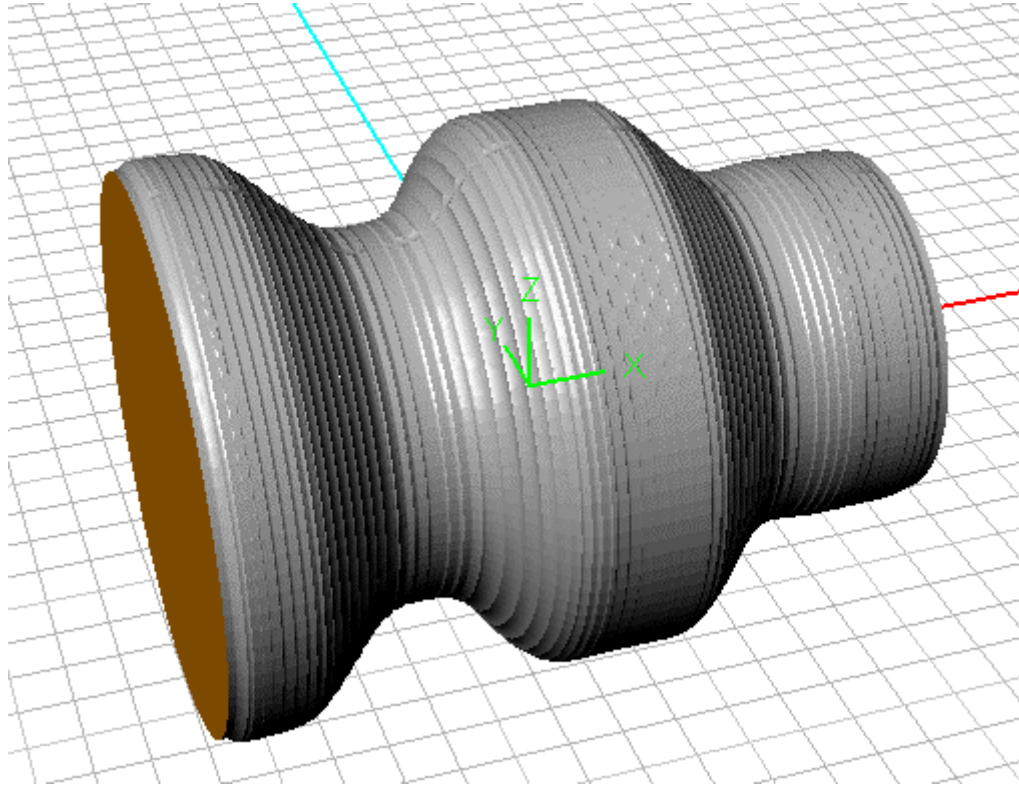
- In the **Step Down Control** page set the step down values to control multiple roughing passes as shown. In the **Cut Levels** parameter box, the top setting defines the top radius value for the first pass and the bottom setting defines the radius value for the last cut. The step-down control defines the distance between each successive roughing passes. In this case it is set to be 0.25 inches.



- Now generate the toolpath. Note the multiple level toolpath.



8. Simulate and you will be able to see the finished part as shown below. You can refer to **4thAxisRoughingComplete.3dm** which is complete with the roughing operation.



End of Tutorial 4!

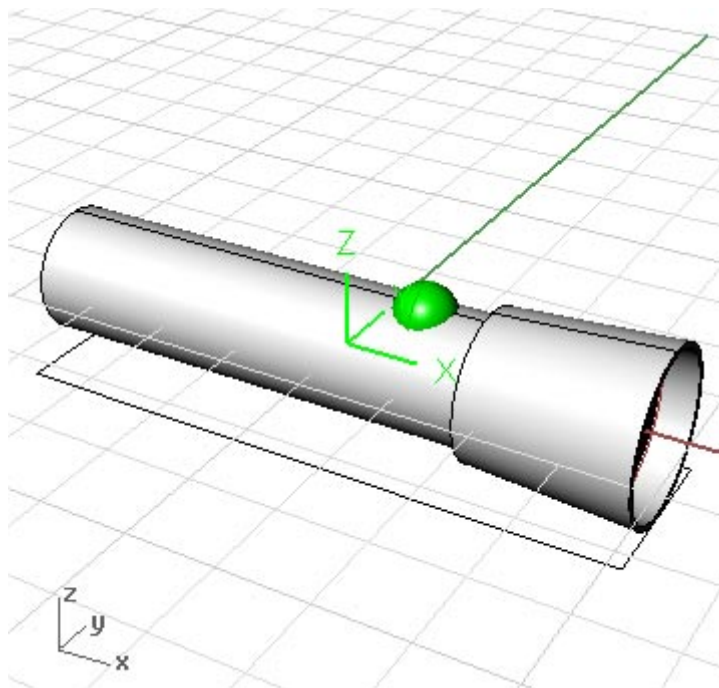
Tutorial 5: 4th Axis Finishing Operation

Purpose

This tutorial is intended to illustrate the combined use of 3 Axis roughing and 4 Axis finishing methods to machine parts that cannot be accomplished using 3 axis machining alone. In this tutorial you will learn to

- Setup the machine for 4th Axis machining
- Use 3 Axis operations to rough out the top half of the part
- Use the **rotate table** command to turn the part 180 degrees
- Rough out the bottom half of the part
- Perform 4th Axis parallel finishing to finish the part

Load the part called **4AxisFlashlight.3dm**. The geometry is shown below.

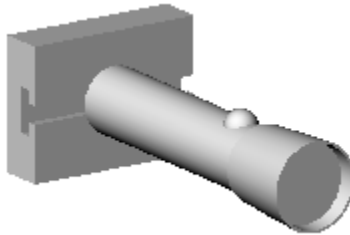


Getting Started with RhinoCAM

Cutting Strategy

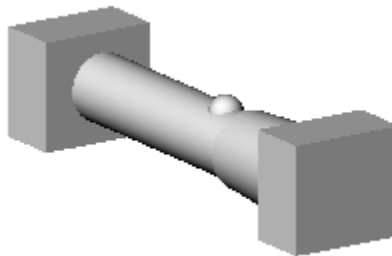
This part cannot be completely machined using just one set of 3 axis operations. We will divide the part geometry into two sections, the top half and the bottom half. We will use the 3 Axis roughing method to rough out the top half, then flip the part over and cut the bottom half.

We first need to determine how the part will be held in the machine. In this exercise we will hold the part so that the axis of the part geometry is aligned with the X axis of the machine coordinate system.



Next, we need to determine which areas of the part should **not** be machined. Some areas must remain un-machined to prevent the part from “falling out” of the stock material. In addition, if there is insufficient material joining the part to the raw stock material, the part may move during machining.

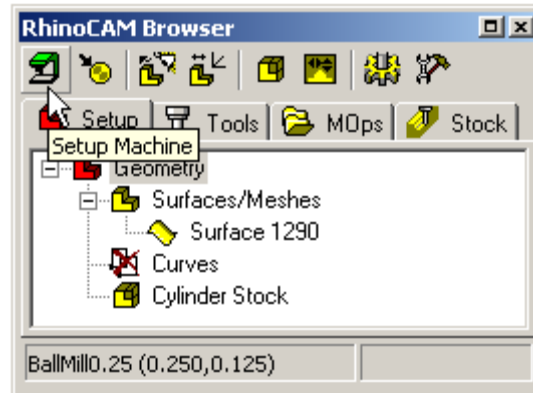
In this exercise we will use a rectangular block of stock as the initial stock volume to machine the part from. We will hold the stock at both ends and also make sure that the toolpath motions stay clear of the clamps holding the part. When this is done, our machined part should look like the picture shown below.



The ends will have to be cut away afterward to obtain the final finished model.

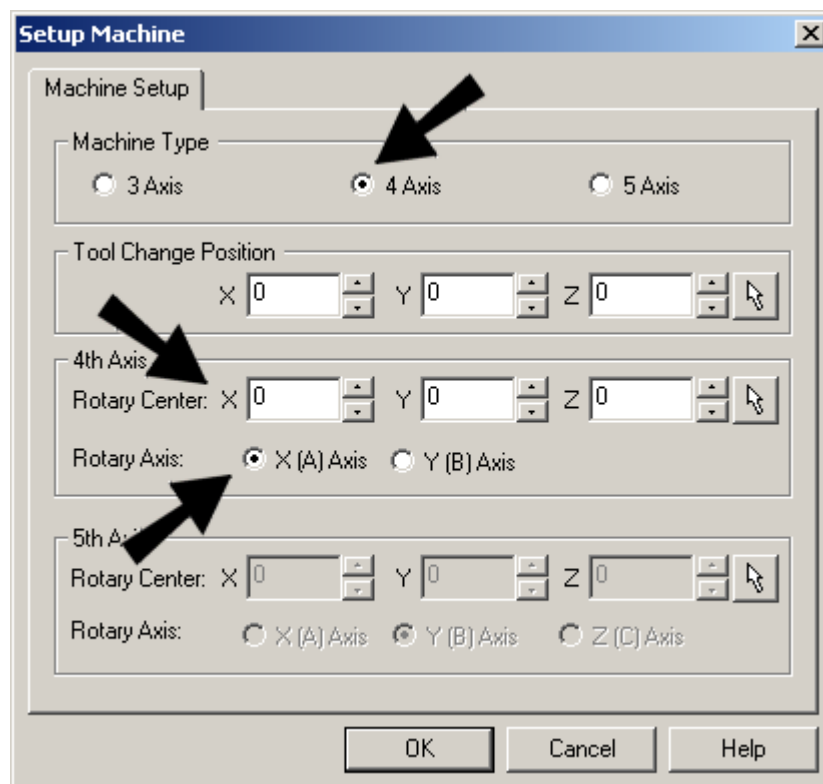
Steps

1. Setup the machine for 4th axis machining. To do this, select the **Setup** tab in the browser window and select **Setup Machine** button on the toolbar.



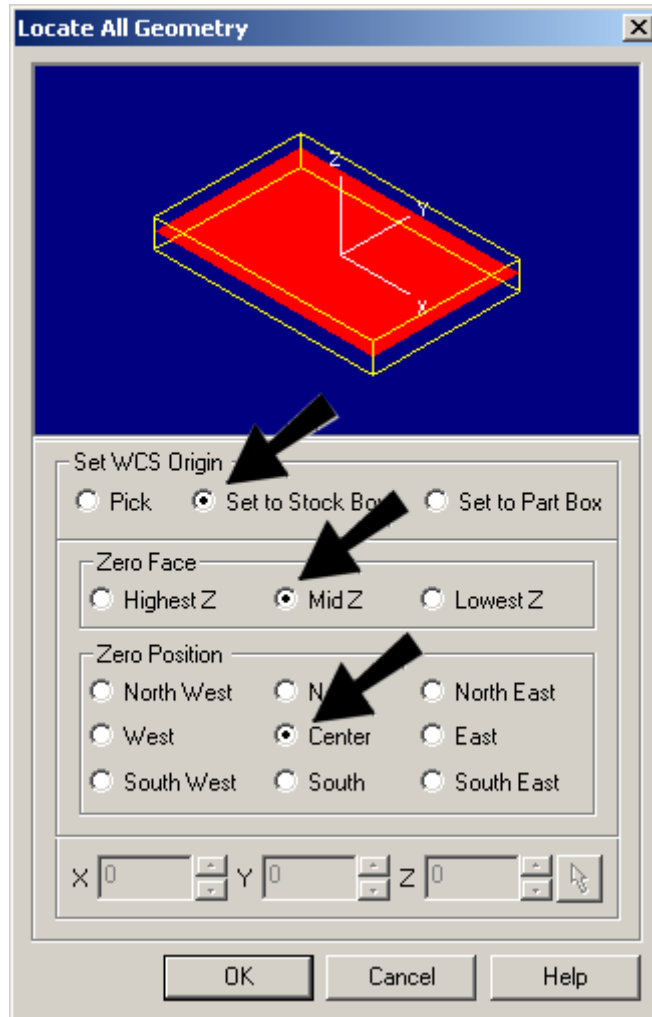
RhinoCAM assumes that the part is located on a rotating table, and that the table can rotate about either the X (A) axis or the Y (B) axis. You will need to specify the rotation axis and the rotation center point. It is recommended to align the machine's rotation axis with the natural rotation axis of the part.

2. In the dialog that comes up, make sure the 4th axis Machine Type is selected and the Rotary Center and Rotary Axis have the settings as shown below.



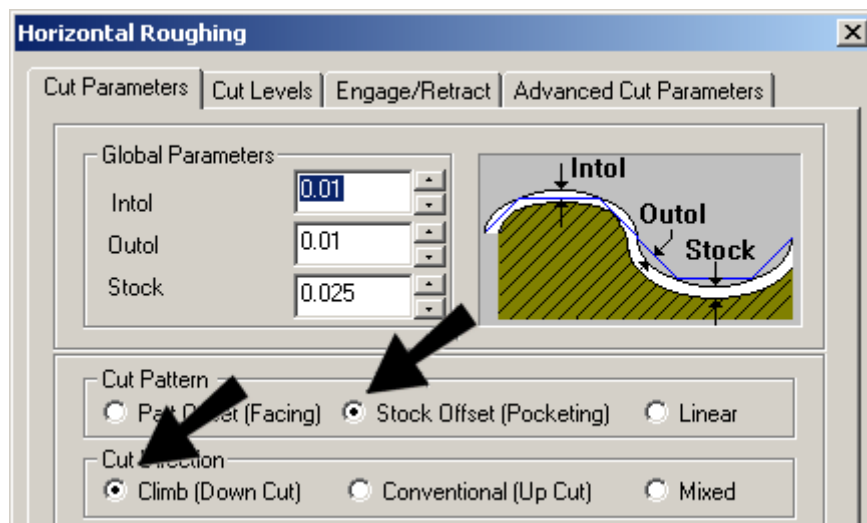
Getting Started with RhinoCAM

- The next step is to locate the machining coordinate system (MCS). In this case, you can see that the MCS is set to be at the center of the part. This is always a good practice when doing indexed machining since it is less confusing to align the rotary axis about the MCS X or Y axis. To do this, select the **Locate Geometry** button in the toolbar and select **Set to Stock Box**, **Mid Z**, and **Center** in the dialog. This will move the geometry such that the MCS is located at the center of the part.

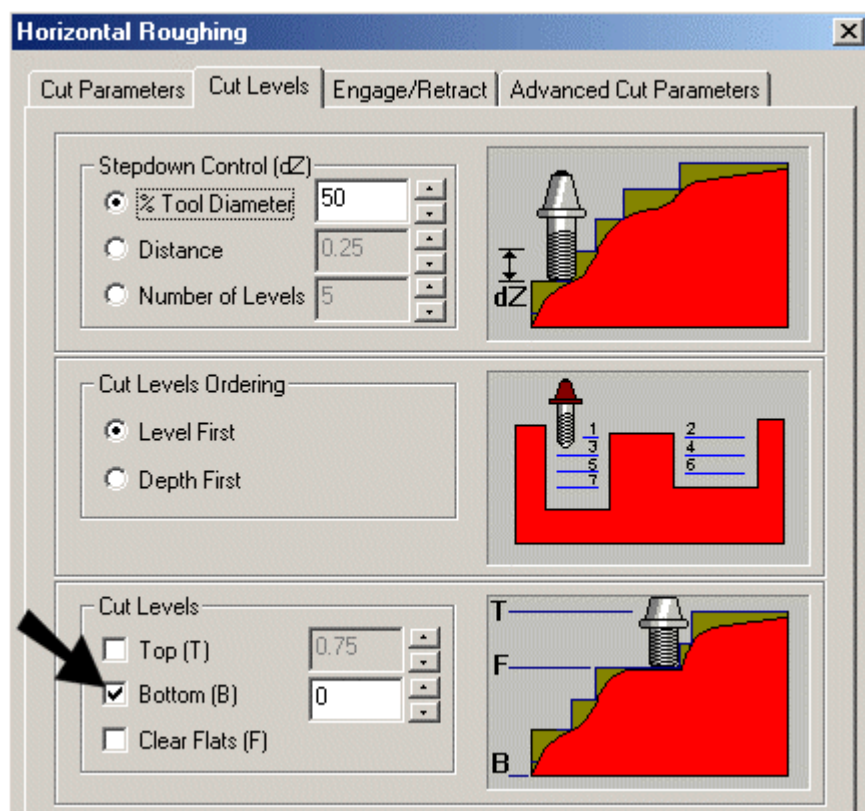


- We will use the 3 Axis **Horizontal Roughing** method to rough out the top half of the part. We will also choose the rectangular region that is in the file as the containment geometry. This is important because we want to make sure that we do not hit the clamps holding the part on the sides. This region will keep the cutter from moving outside this area.
 - Select the **Tools** tab in the **Browser**. This file already contains two tools – a ¼ inch ball mill and a ½ inch flat mill. Activate FlatMill1.
 - If necessary, reset the **Clearance Plane** to **Automatic**.

- Select the rectangular region by using Rhino's single selection command in the graphics window.
- Select the **3-Axis Milling, Horizontal Roughing** toolpath method.
- In the **Cut Parameters** page, select **Stock Offset** Cut Pattern and **Climb** Cutting Direction.

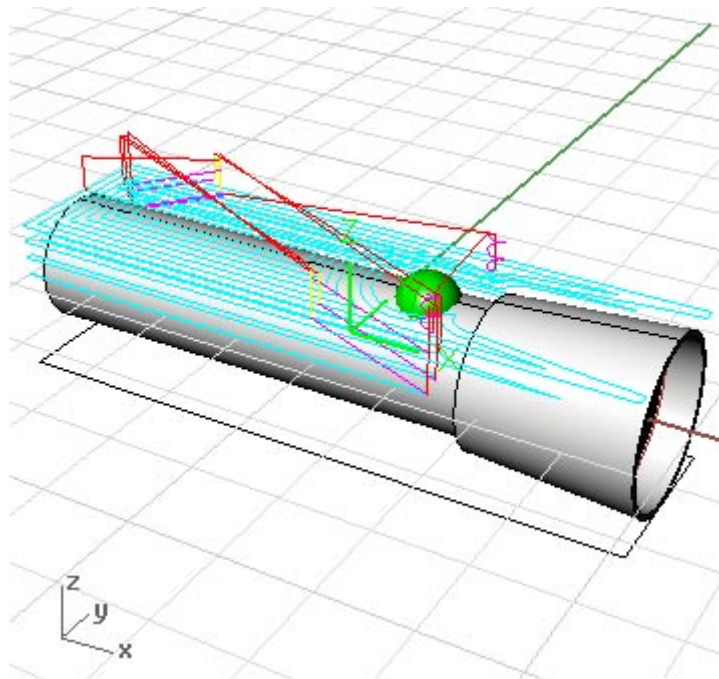


- In the **Cut Levels** page, set **Bottom** to zero so that the machining stops at the center of the part

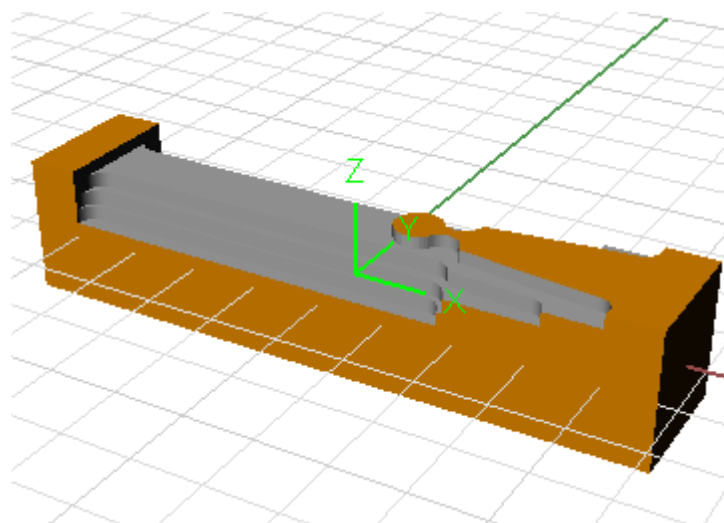


Getting Started with RhinoCAM

- Click Generate to rough the top part.

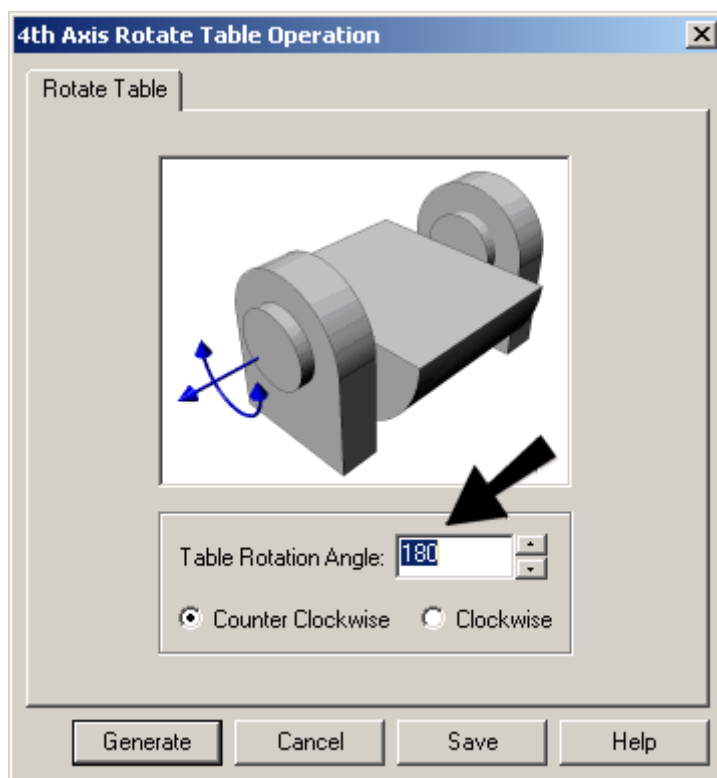


5. Simulate the machining.



Note that the ends of the stock model are not machined.

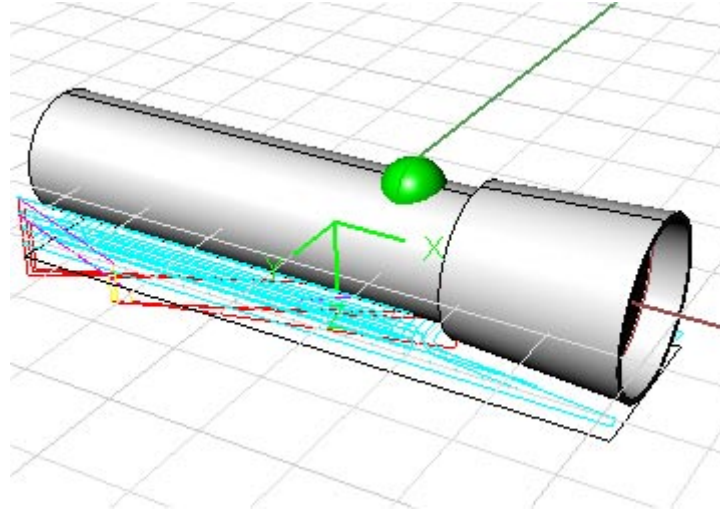
- Next you will rotate the table and rough out the bottom half of the part. Select **4th Axis Milling / Rotate Table**, or use the **4th Axis** icon on the **MOps** tab. Enter 180 degrees and Click **Generate**. A **Rotate Table** machining operation appears in the **MOps** tab. The MCS has flipped and you will note that the Z axis now points downward.



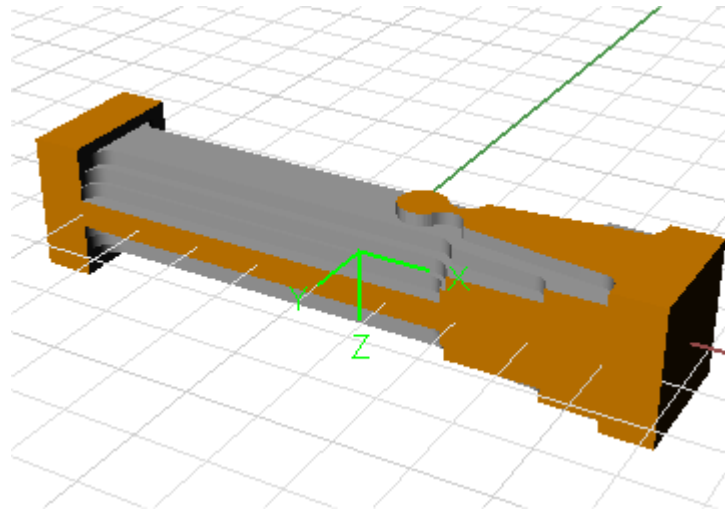
Note: The display of the part does not change as a result of the table rotation. In reality, however, the machine would have turned the table

- After the **Rotate Table** operation, create another **Horizontal Roughing** toolpath with the same parameters as the previous one. The toolpath is created on the underside of the flashlight. You can copy and paste the previous operation by Right clicking the mouse button. Generate the toolpath.

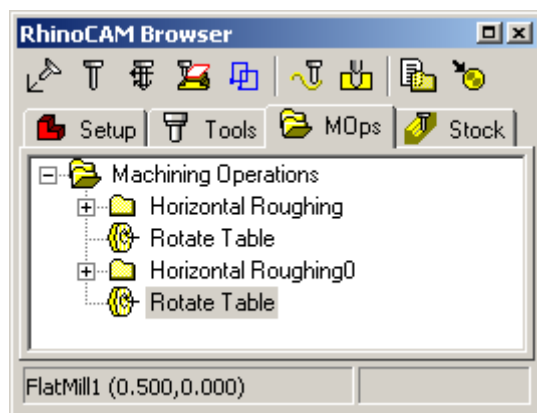
Getting Started with RhinoCAM



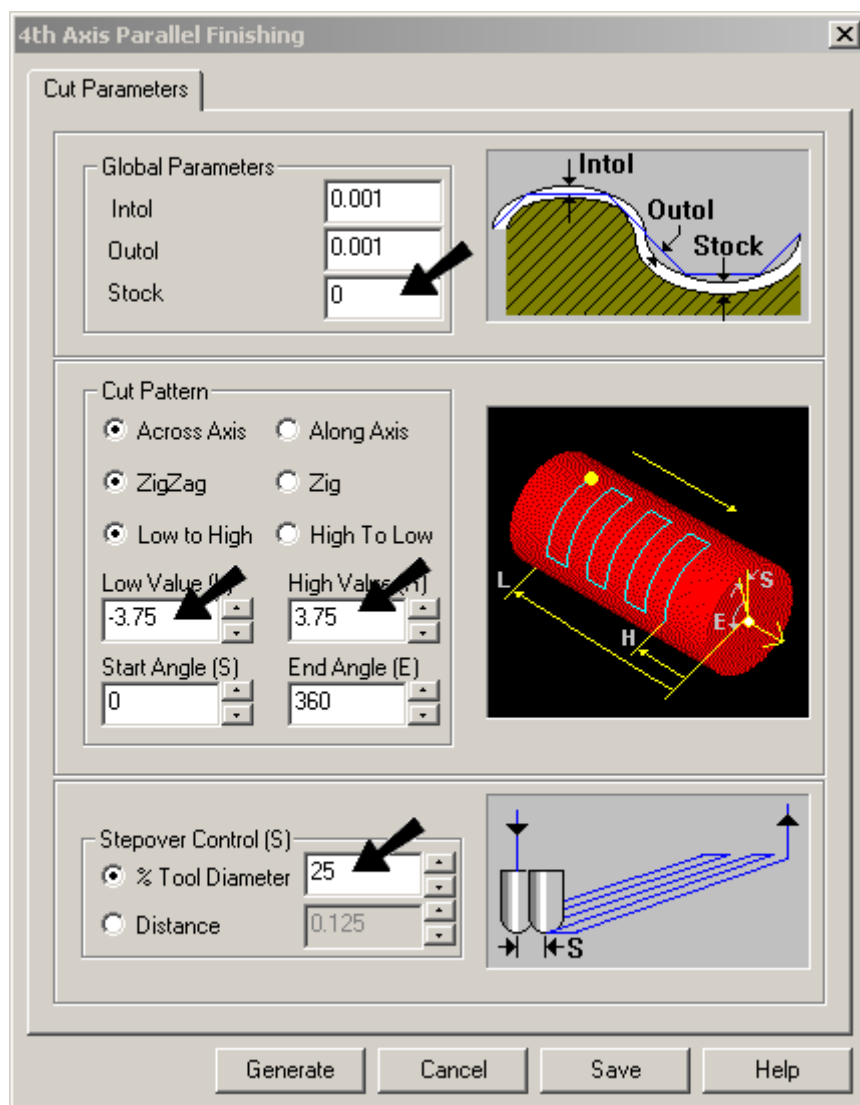
8. Simulate this second toolpath. If you are running the **Voxel** simulation model, you will receive a message that rotated toolpaths cannot be handled. Click Yes to switch to the **Polygonal** simulation model, which means you will have to simulate all toolpaths from the beginning.



9. Create a new **Rotate Table** operation by specifying the rotation angle to zero such that the table is brought back to its original position.
10. Select the last **Rotate Table** operation, so that the new finish toolpath will be created after it.

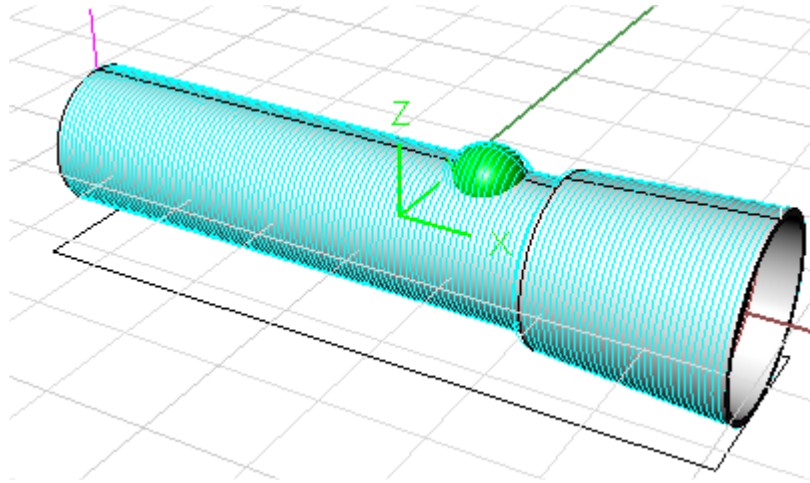


11. After selecting the finishing tool, select **4th Axis / 4th Axis Finishing**. In the **Cut Parameters** page, make sure the remaining **Stock** thickness is set to 0. Specify the Low and High values as -3.75 and 3.75 respectively, so that the cutter does not go past the ends. Set the **StepOver Control** parameter to 25% of the tool diameter.



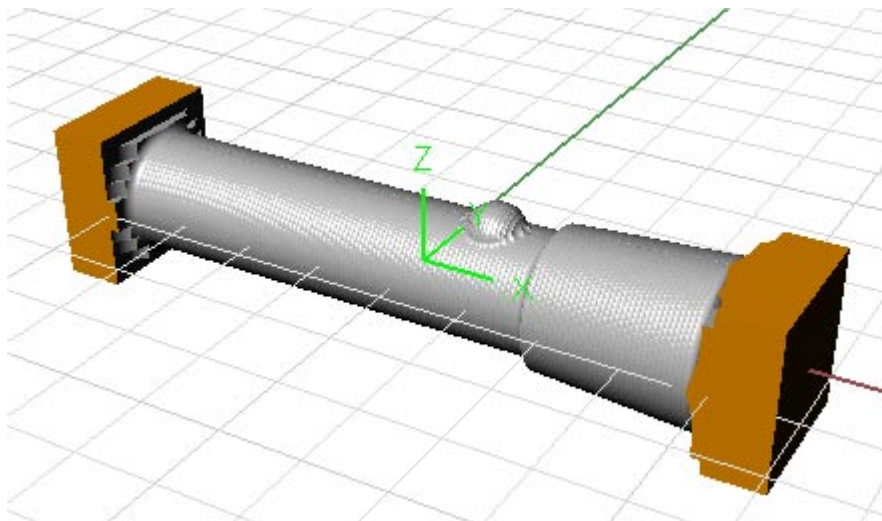
Getting Started with RhinoCAM

12. Click **Generate**. It may take a few minutes to complete.



***Note:** This method does not rely on regions for containing the tool. Regions are inherently 3 axis in nature and are meaningless in 4th axis machining. However, containment of the tool can still be accomplished by limiting the angle of rotation or the length of travel. These values can be set in the **Cut Pattern** section of the **Cut Parameters** tab of the dialog.*

13. Simulate this toolpath. Notice that the ends are still there and will need to be removed once the part is unclamped.



You can refer to the **4thAxisFlashlightComplete.3dm** file, which is the completely programmed part.

End of Tutorial 5!

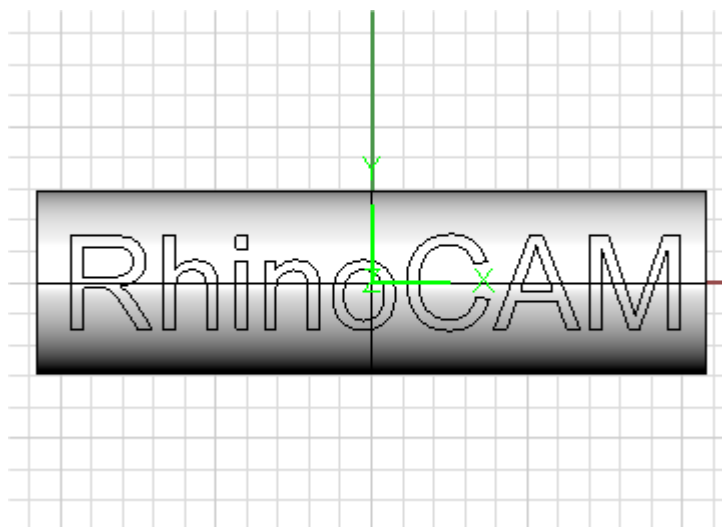
Tutorial 6 – 4th Axis Engraving

Purpose

This tutorial is intended to describe an effortless way to use 4th Axis engraving wherein text is projected on to the surface of a 4th axis part.

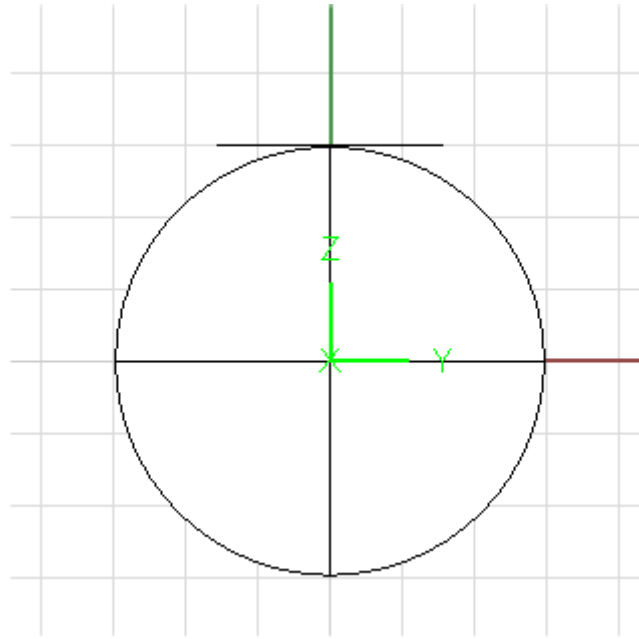
Steps

1. Load the file **4AxisEngraving.3dm**. The top view the part file should look as seen below.

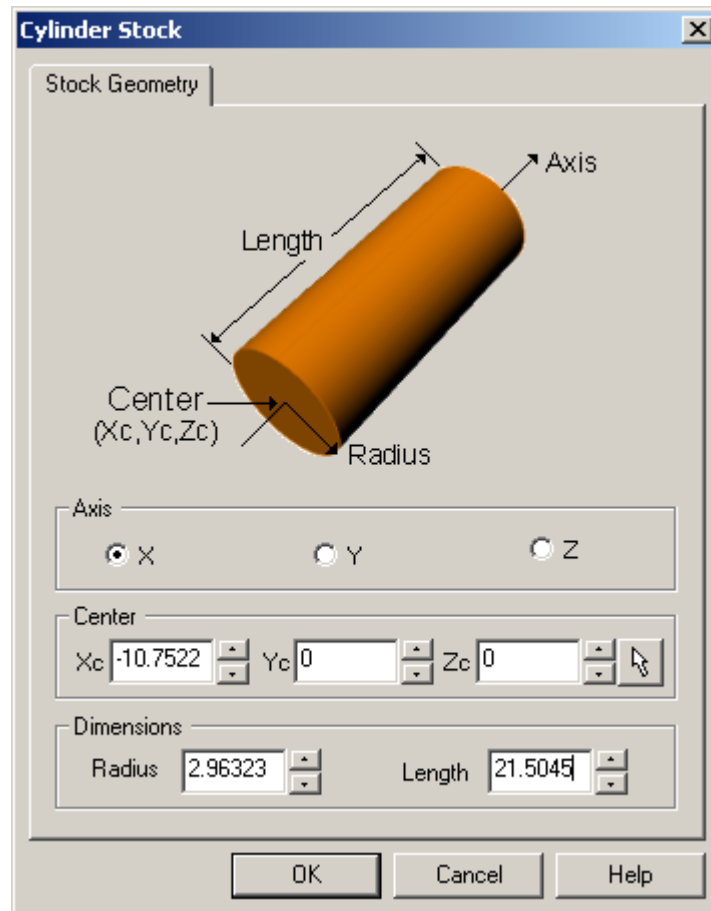


2. Make sure that the text is located above all of the part geometry. To check this, switch to the side view. The text should be seen flush with the top of the part as shown.

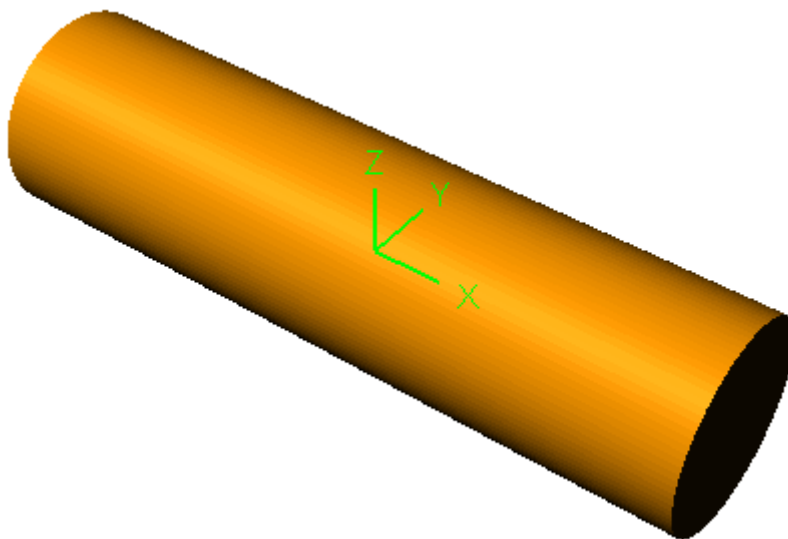
Getting Started with RhinoCAM



3. Select a **Cylindrical Stock**. We will work with the default settings this time.



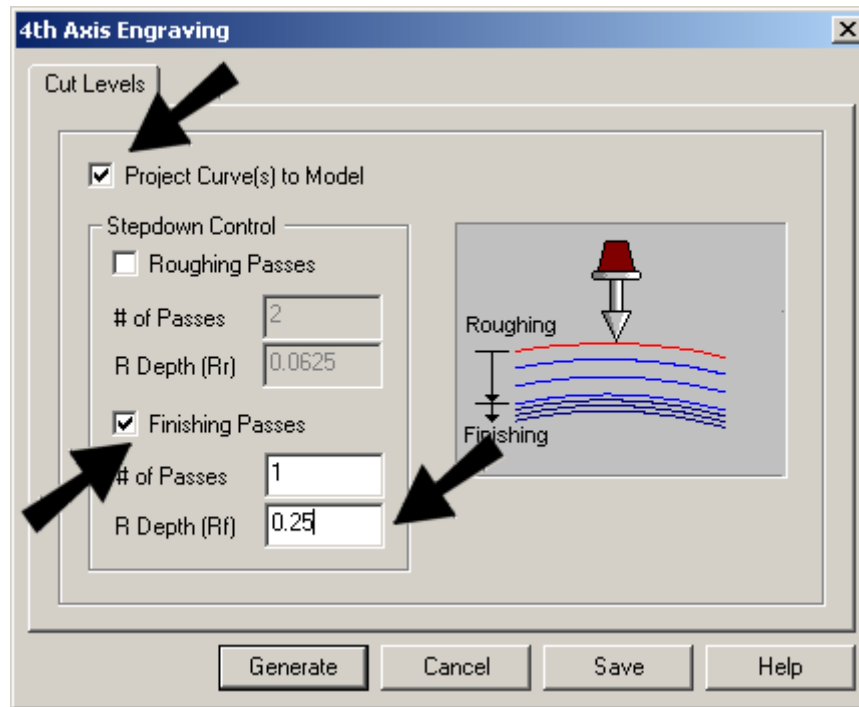
4. Switch to the Stock tab to display the stock model. The stock model should look like this. If the display of the stock model does not resemble the picture below, then make sure you are using the **Polygonal** simulation model. To do this, select the **Simulation Settings** button in the toolbar and select the appropriate radio button in the dialog.



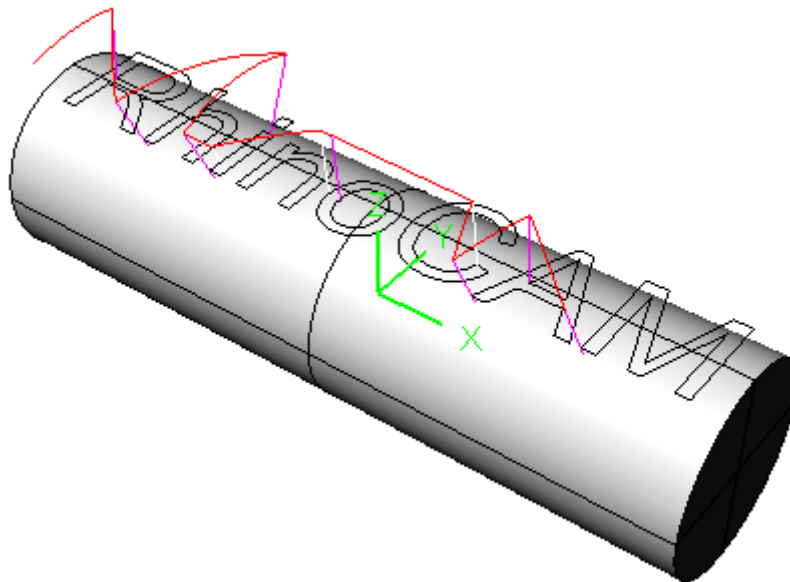
***Note:** To emphasize what was mentioned in the 4th axis introductory session, the RhinoCAM product with the 4th Axis Add-on module has two material removal simulation modes (or models). The **Voxel** model can only be used for 2½ and 3 axis toolpaths. The main advantage of this model is rapid processing times. The **Polygonal** model is necessary for simulating all 4th axis toolpaths. This model is more comprehensive and results in better display quality, at the expense of speed.*

5. Next select the **Scriber** tool that is saved in the part file and make it the active tool.
6. Now select the curves that make up the text by using the Rhino rectangle selection tool in the graphics window.
7. Choose the 4th axis Engraving operation from the MOPs tab. In the parameter dialog box, set the **Project Curves to Model** check box. We will choose to perform a single finishing pass with a cutting depth of 0.25. Enter these values in the dialog box as shown below. To make deeper cuts you could choose multiple passes, a roughing pass followed by a finishing pass.

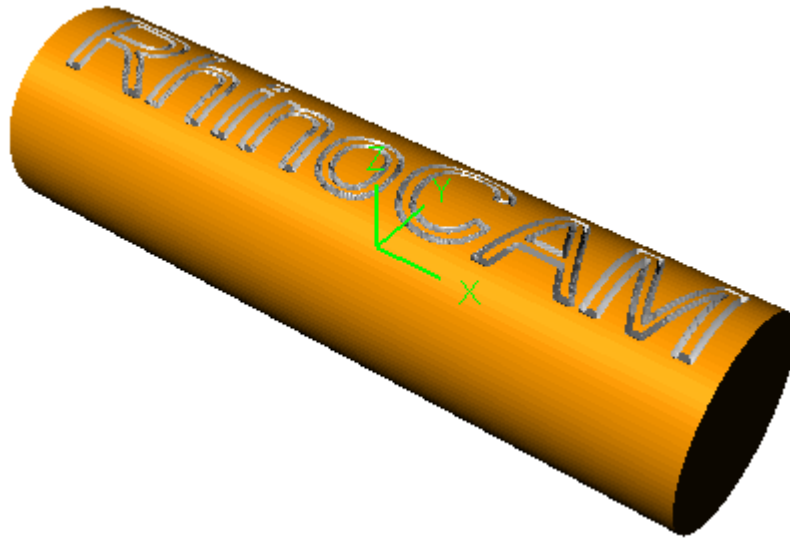
Getting Started with RhinoCAM



8. Generate the toolpath. The toolpath should be displayed as shown below.



9. Switch to the stock tab and simulate. The letters will be engraved as shown. You can refer to **4th AxisEngravingComplete.3dm** part file, which is complete with the programmed operations.



End of Tutorial 6!

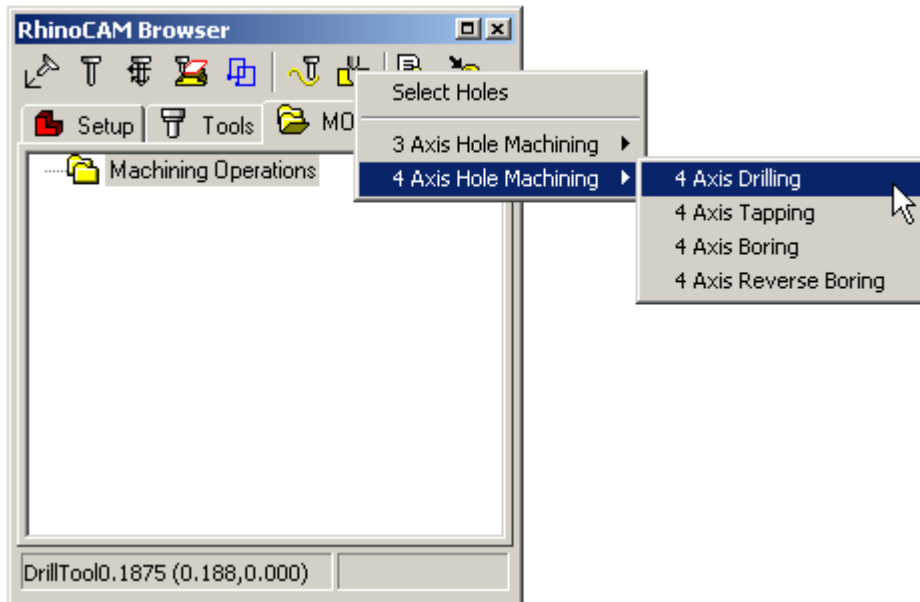
Tutorial 7 – 4th Axis Hole making

Purpose

This tutorial is intended to describe the 4th axis hole making operations. Hole making operations are used to create holes in a part; the hole types varying from simple drill holes, counter sunk holes, through holes to tapped and bored holes. Here you will learn to drill simple holes.

4th Axis Hole Making

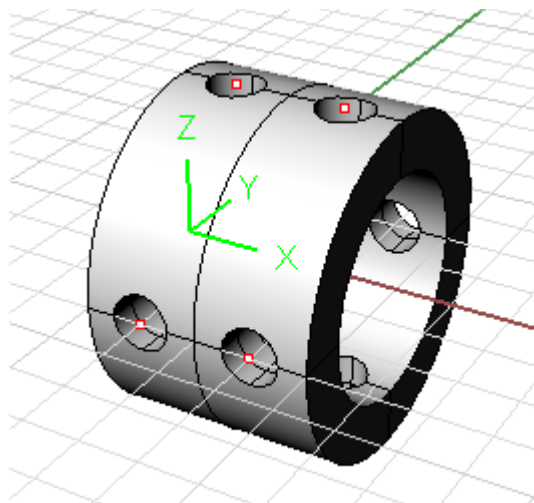
You can access the 4th Axis Hole making options through the hole machining icon in the **MOPs** tab as shown below:



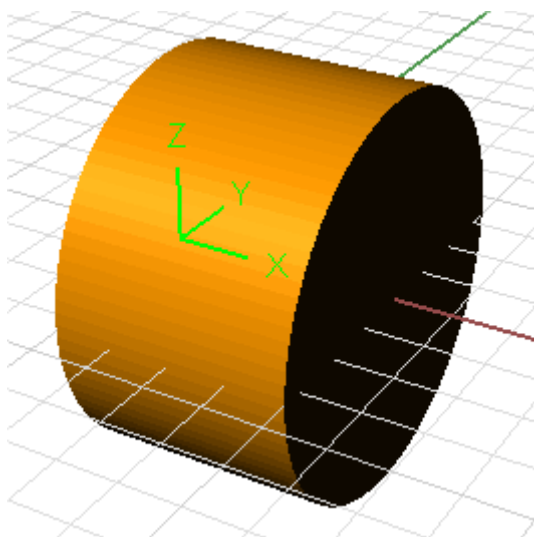
As in any other 4th axis operation, the tool is positioned normal (perpendicular) to the rotary axis. Once the holes (regions) are selected, the dialog boxes are similar to the 3 axis hole making operations. Sorting of holes is also possible to optimize the tool motion.

Steps

1. Load the part file **4thAxisHoles.3dm**. You will see the following part.



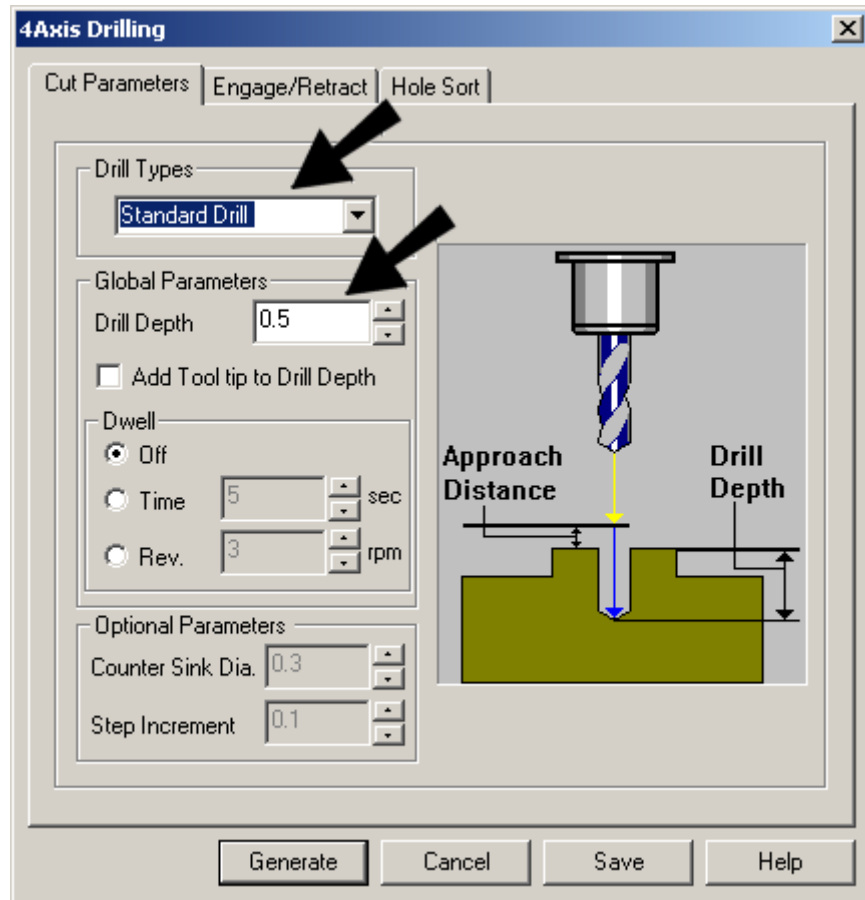
2. Create a **Cylindrical Stock** by choosing the default values in the dialog



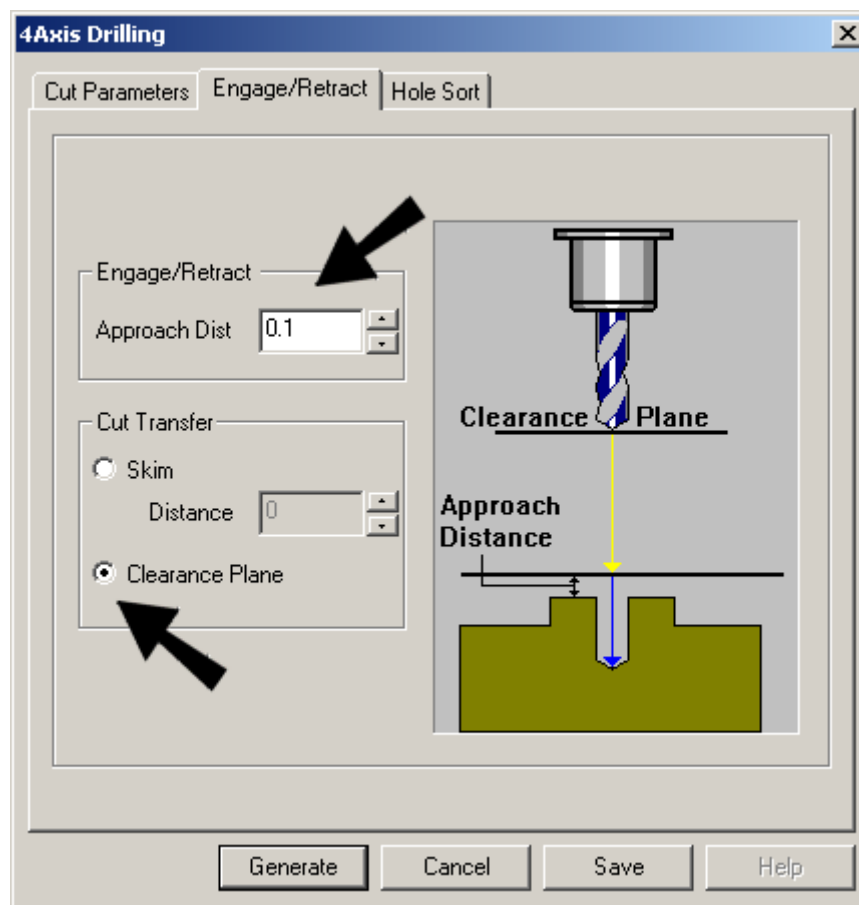
3. Select all the eight centers (points) of the holes in random order using Rhino's single selection tool in the graphics window.
4. Switch to the **Tools** tab and select the 3/16 inch drill bit from the tool dialog.
5. Switch to the **MOPs** tab and set the **Feeds/Speeds** values and the **Clearance Plane** values (using the appropriate buttons) as required for your machine or use the default values.

Getting Started with RhinoCAM

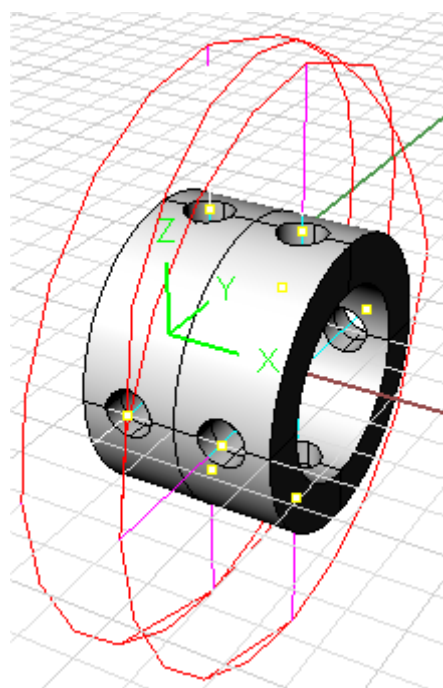
- Pick the **Hole Machining** button and select **4th Axis Drilling**.



- Enter the **Drill Depth** of 0.5 inch. In the **Engage/Retract** page, enter an approach distance of 0.1 and set the **Cut Transfer** to clearance plane.

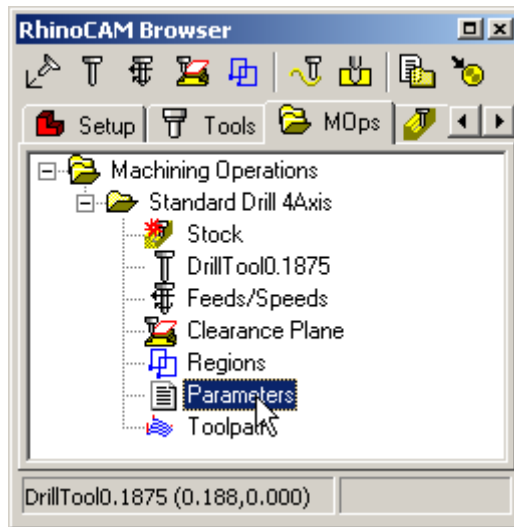


8. Generate the tool path. Notice that the toolpath generation follows the order in which the holes were selected.

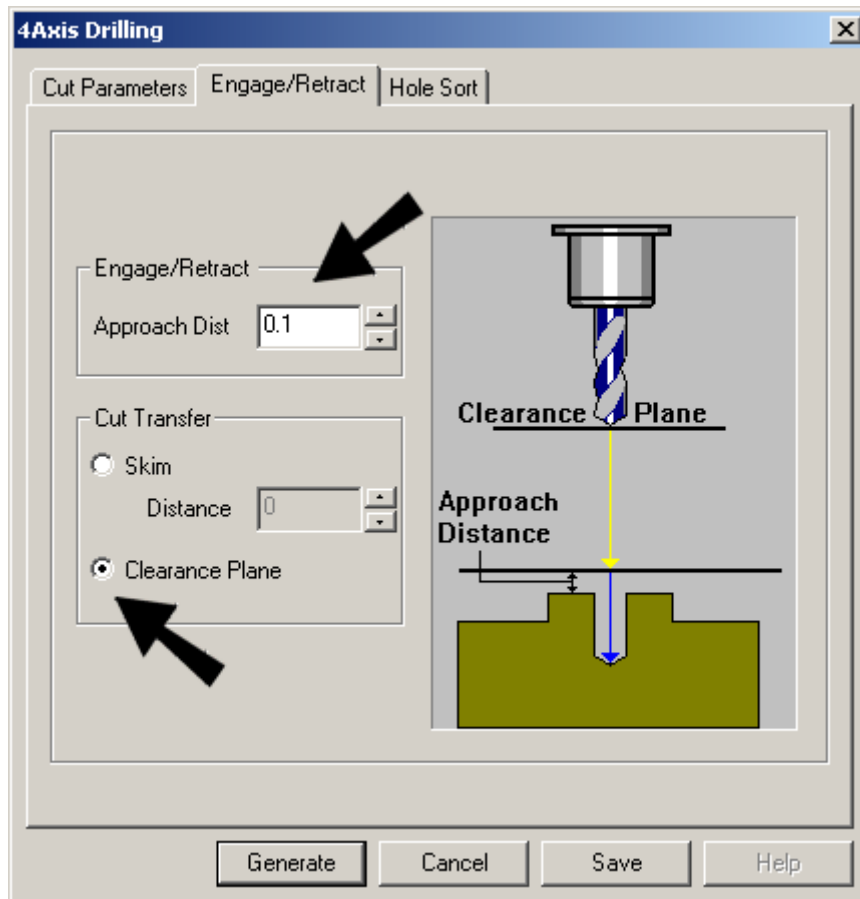


Getting Started with RhinoCAM

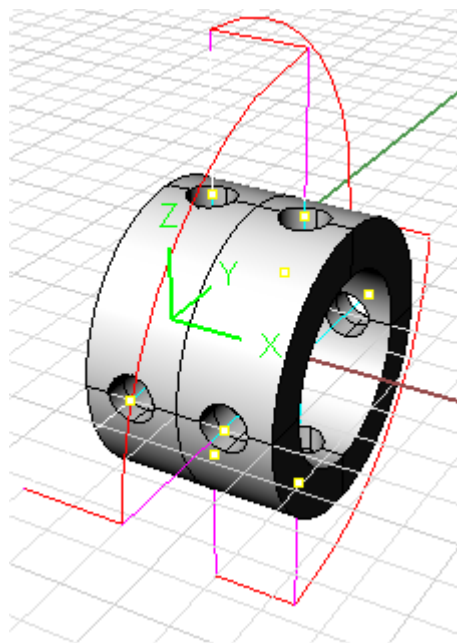
- To optimize the tool travel time, the holes should be picked in the most efficient order. Double-click on the **Parameter** icon of the created operation you just created, as shown below to invoke the dialog box for 4th axis drilling.



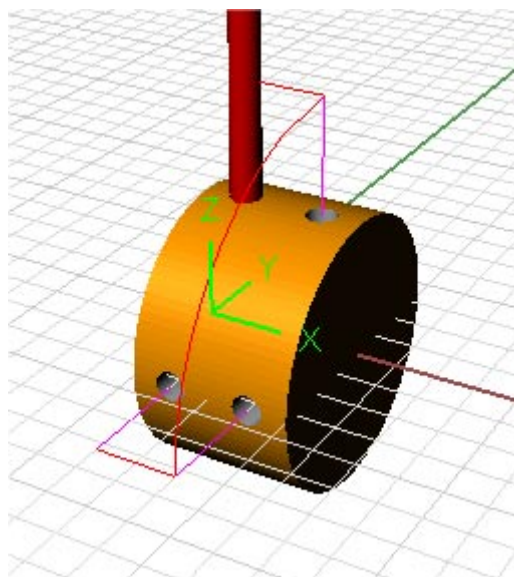
- Select the **Hole Sort** page and choose the options indicated in the picture below. Generate the toolpath.



You will notice that the toolpath follows the specified sorting order. That is, the primary sorting direction is at a 0 degrees to the rotary axis or parallel to the rotary axis. The secondary direction is from the high value to the low value. Additionally the tool travels in a zig-zag fashion from high to low and low to high along the rotation axis as it visits each hole.



11. Switch to the **Stock** tab and simulate the operation. You can refer to **4thAxisHolesComplete.3dm** part file, which is complete with the programmed operations.



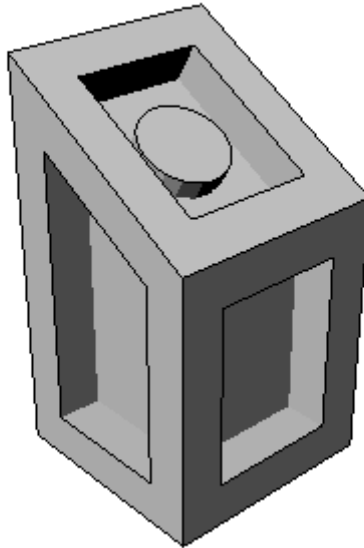
End of Tutorial 7!

Tutorial 8: 5th Axis Machining Operations

Not available in RhinoCAM Basic

Fifth axis machining enables you to change tool direction relative to any set of axes.

Load the part file **5Axis.3dm**.



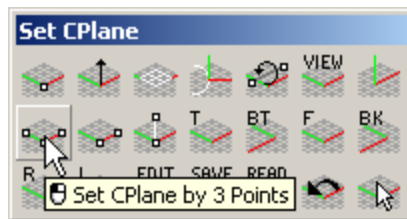
Setting the Construction Plane and MCS

Milling has to be done in three sets of axes – one for the sloped top face, and one for each of the two side faces. By changing the CPlane (construction plane), you can easily change the MCS (Machining Coordinate System).

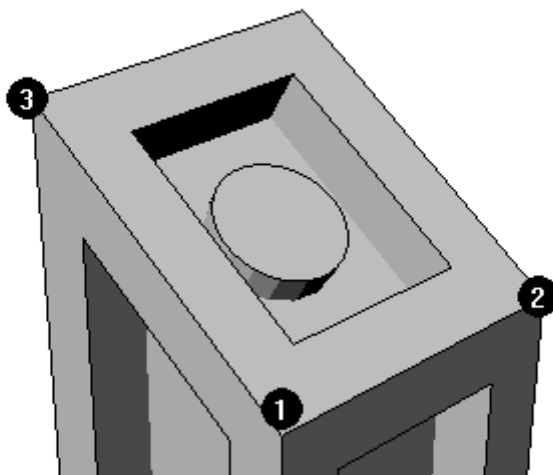


To set the MCS to the top slanted face:

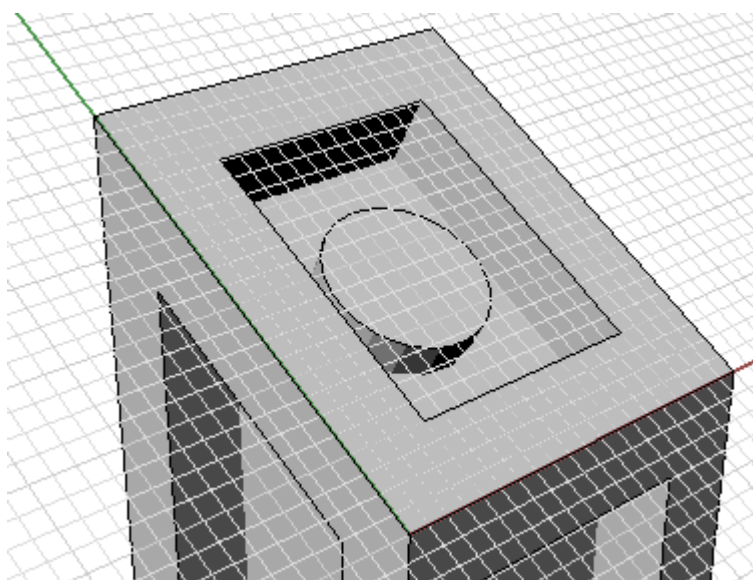
1. First, make sure that the Rhino Cplane is displayed. The grid is used to visualize the current CPlane, and to create geometry (if you want to use grid snaps).
2. Now to change the CPlane to coincide with the top face. Select **Set CPlane by 3 Points**.



3. The first point sets the origin, the second sets the direction of the X axis, and the third sets the Y axis. Select the points in the order shown below. The **Point** snap of the **Osnap** mode must be active.

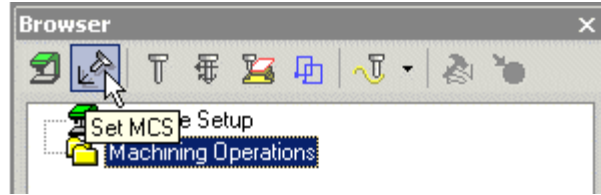


The grid appears along the new CPlane as shown below.

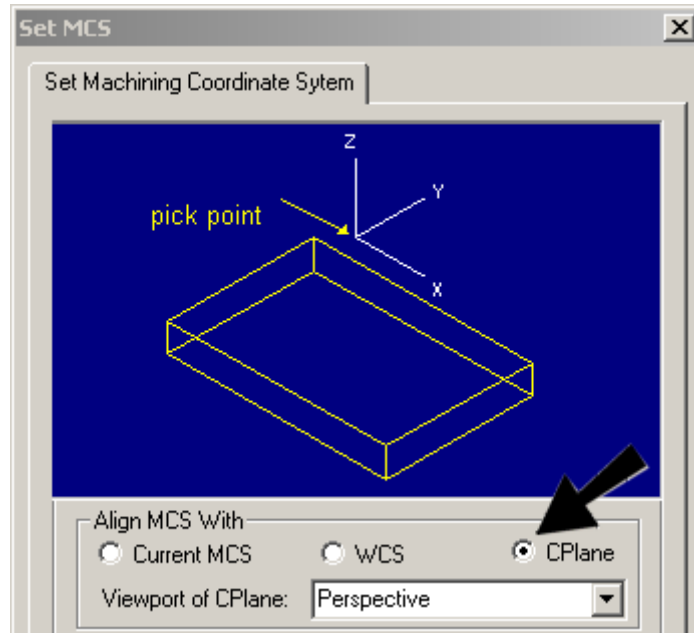


4. In the **MOPs** tab, select **Set MCS**.

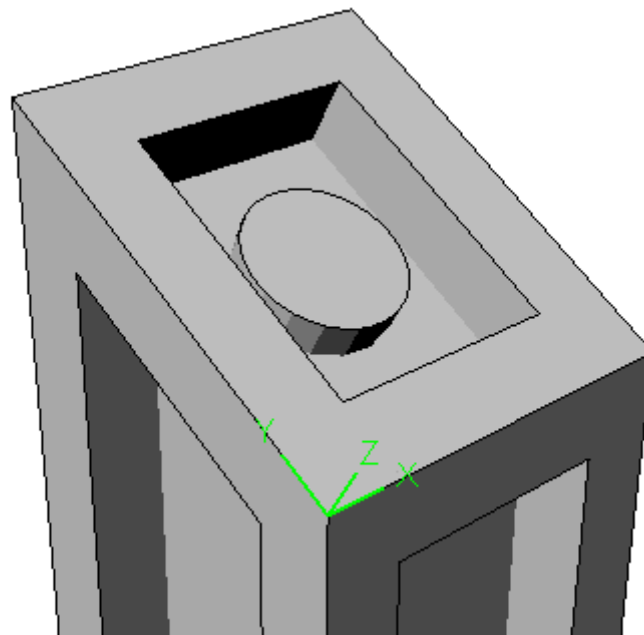
Getting Started with RhinoCAM



5. Check **CPlane** to set the machining axes relative to the top face and select the **Generate** button.

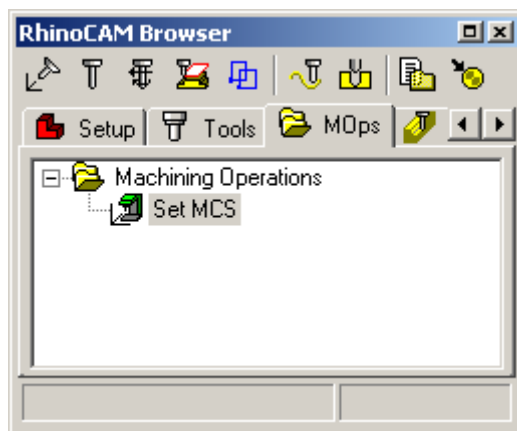


The MCS has moved to the top face, with the X-Y plane coincident with this face.



If the MCS is not displayed as shown, make sure that the **Toggle MCS Display** button in the **View**

toolbar is toggled correctly. Also notice that a Set MCS operation has been created and displayed in the browser as shown below:



Machining the Top Face

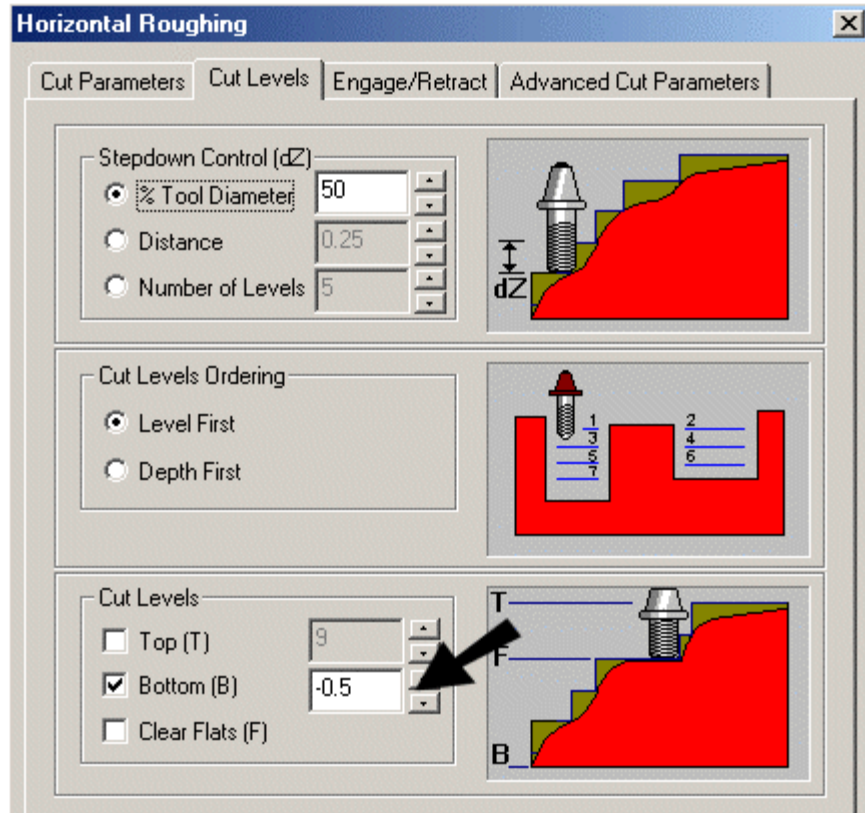
Now that the MCS is oriented correctly, the top face can be machined. First the pocket will be roughed out, then a facing operation will be applied to the top face, then the depressed area will be pocketed.



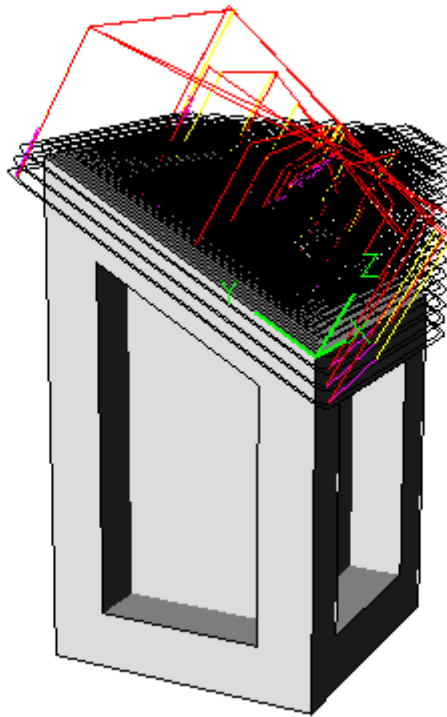
To machine the top face:

1. Activate the FlatMill1 tool (0.5" diameter).
2. Select the **Set MCS** operation in the Browser that has already been created. This will ensure that any subsequent operations will be created after this operation.
3. Create a 3-axis Horizontal Roughing toolpath. In the **Cut Levels** tab set the **Bottom** level to -0.5 – this is the depth of the pocket. Accept all other default parameters.

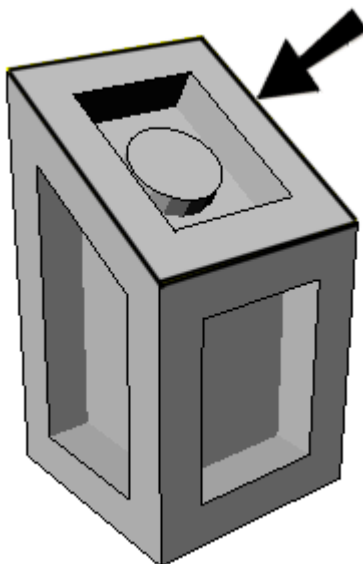
Getting Started with RhinoCAM



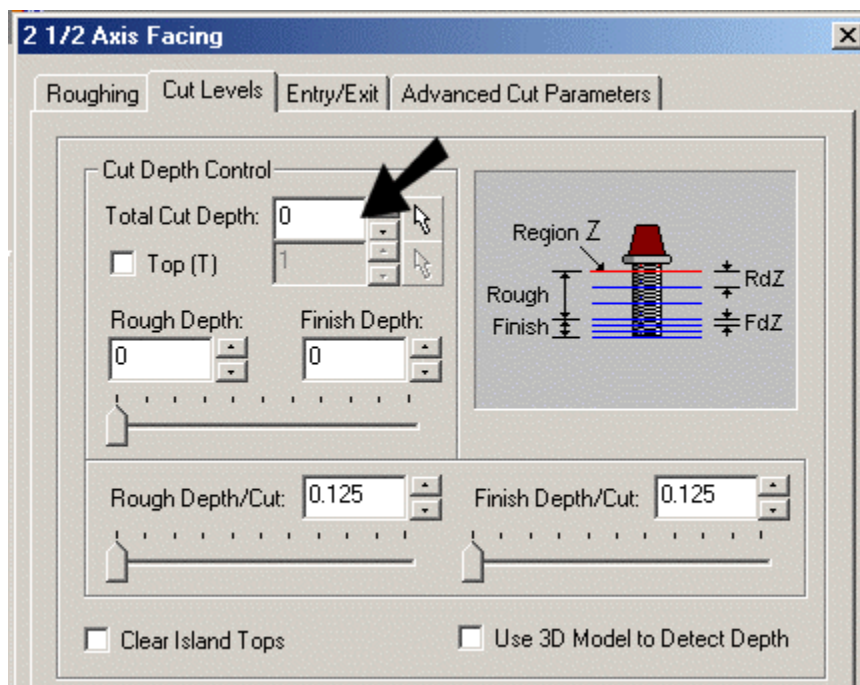
4. Generate.



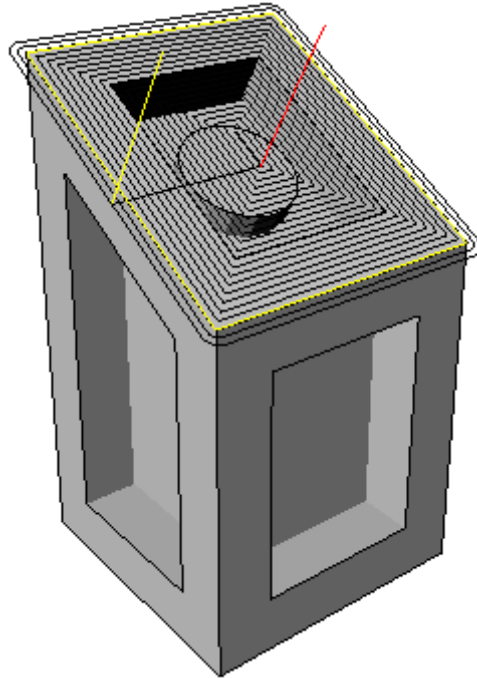
- The next toolpath is a 2-1/2 axis operation and so regions must be defined. Use **Single** select to select the outer boundary of the top face.



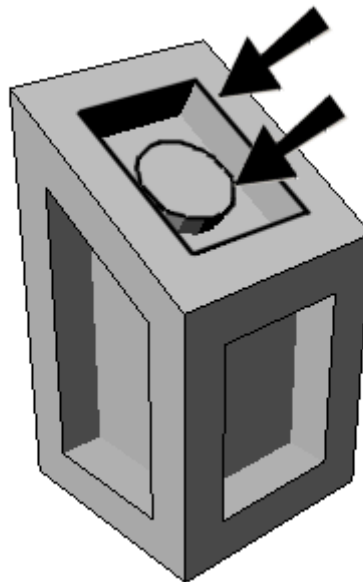
- Create a 2.5 axis Facing toolpath, setting the **Total Cut Depth** to 0.



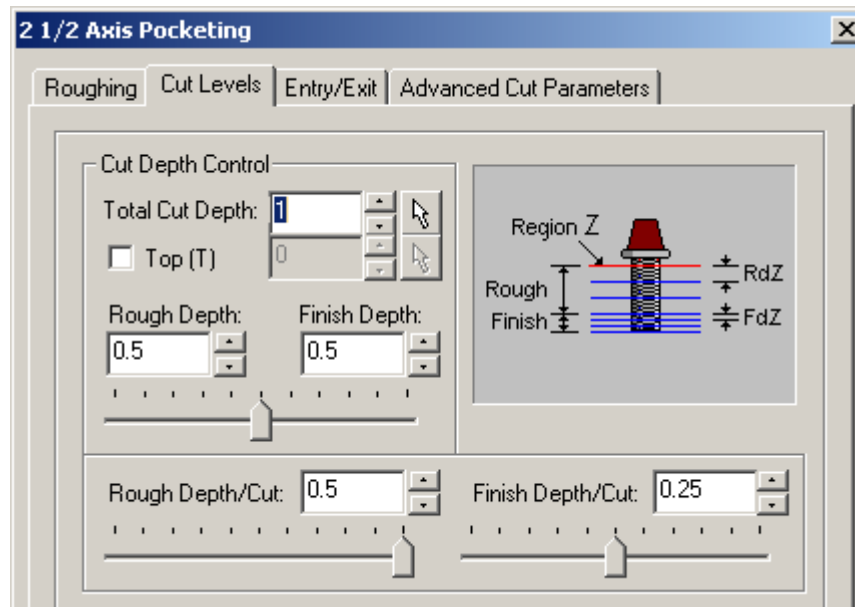
- Generate.



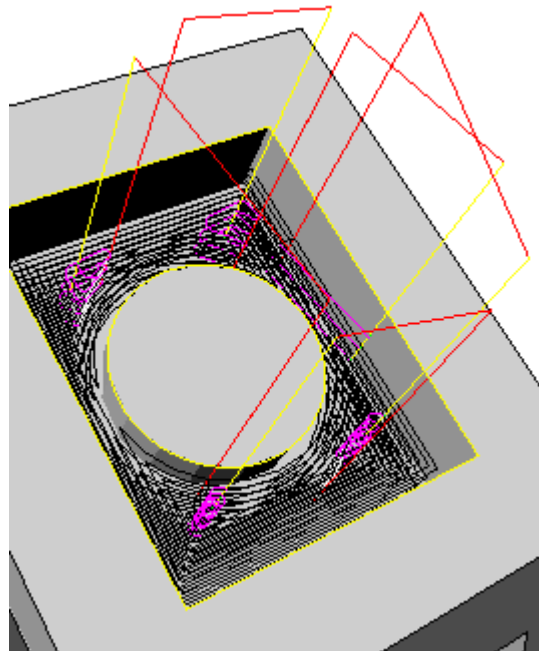
8. For the pocketing toolpath, select both the inner rectangular region and the circular region.



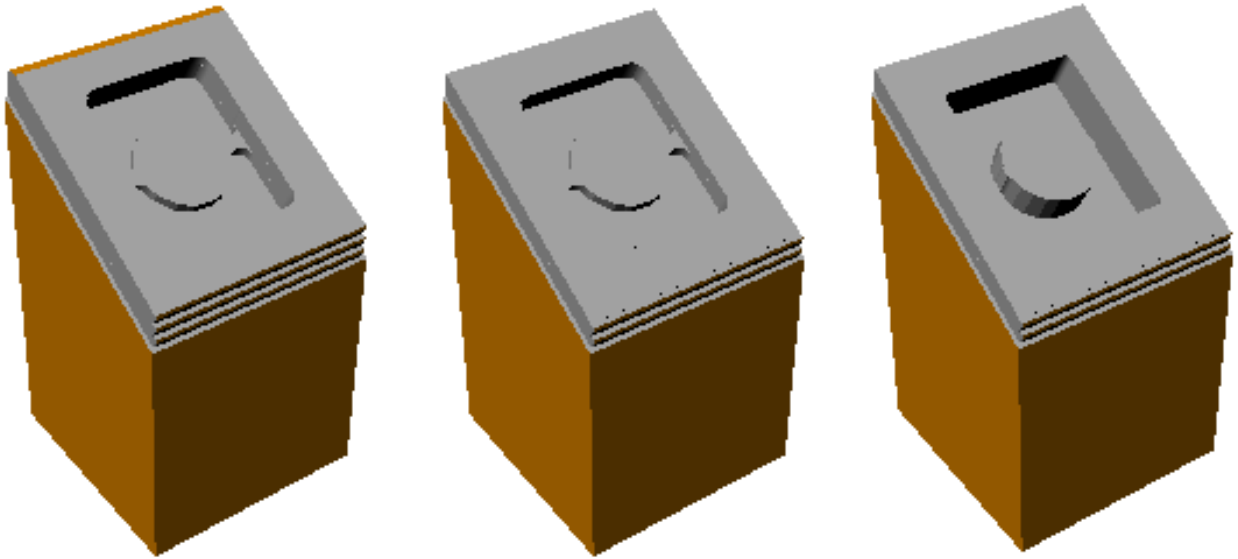
9. Activate the FlatMill2 tool (0.25" diameter) and create a 2.5 axis Pocketing toolpath, setting the **Total Cut Depth** to 1 (the depth of the pocket). Make sure you set the stock to 0.0 so as not to leave any material.



10. Generate.



11. Simulate the three toolpaths created so far.



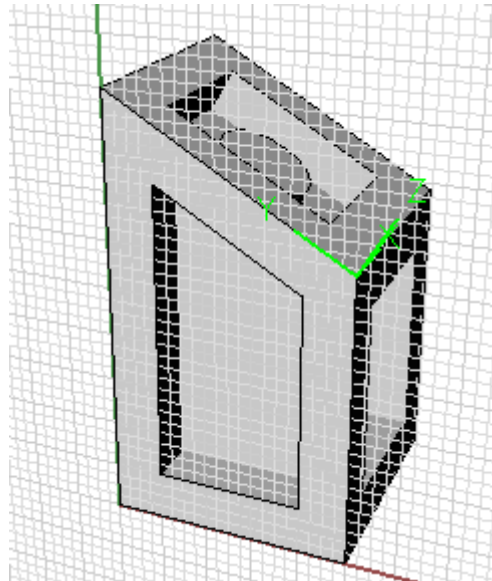
Machining the Side Face

In this section, the MCS will be changed to the side face, and two more toolpaths will be added.

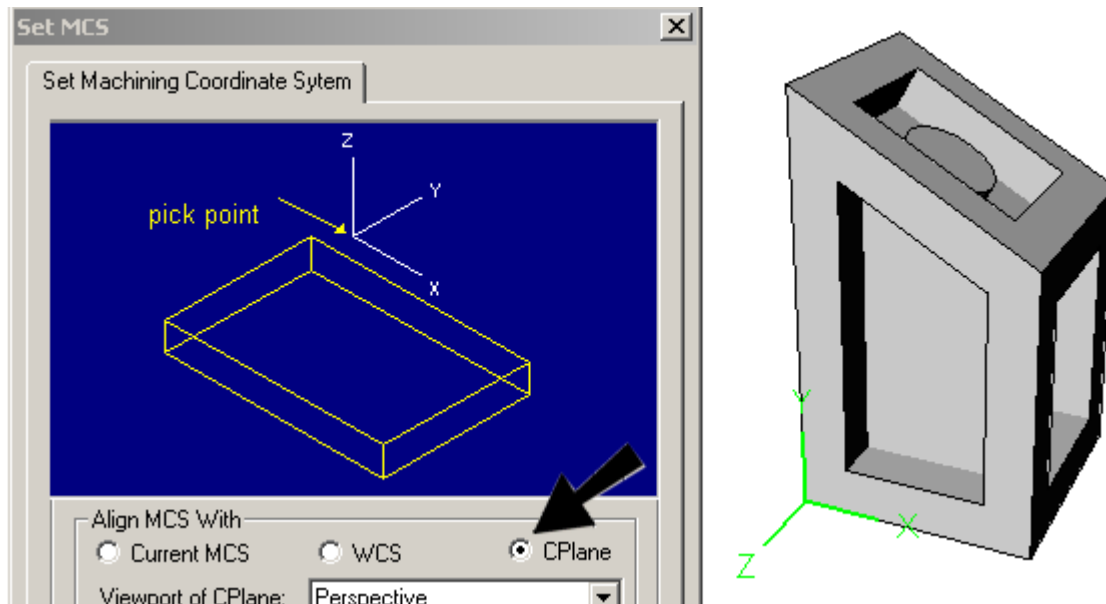


To machine the side face:

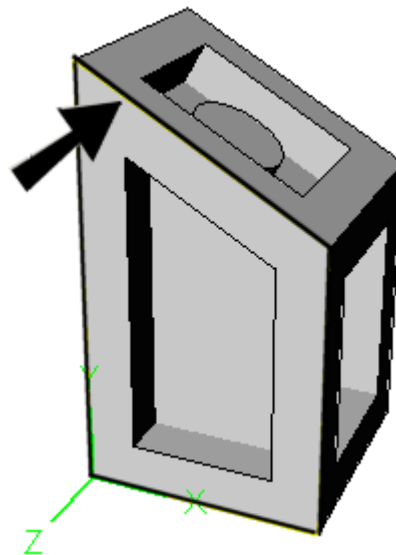
1. Use **CPlane 3 Points** again to change the construction plane to the side face as shown.



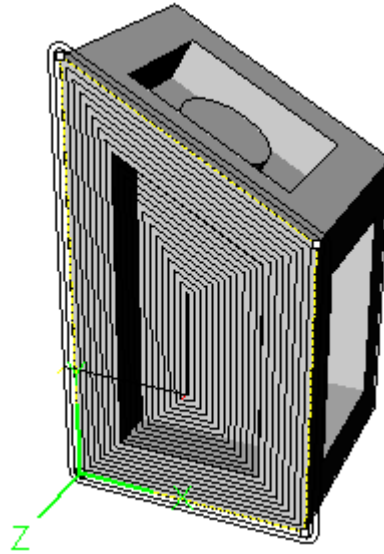
2. Set the MCS to the new CPlane. This should create another Set MCS operation in the browser.



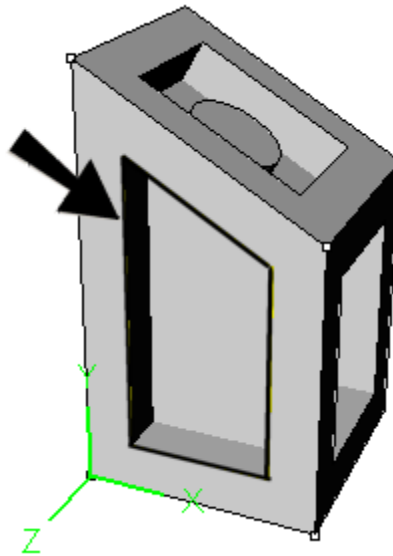
3. Select the outer region of this face.



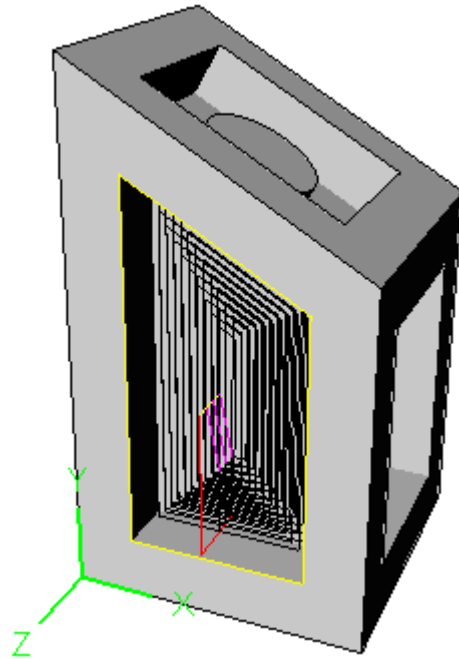
4. Using FlatMill1, create a Facing toolpath with the same parameters as before.



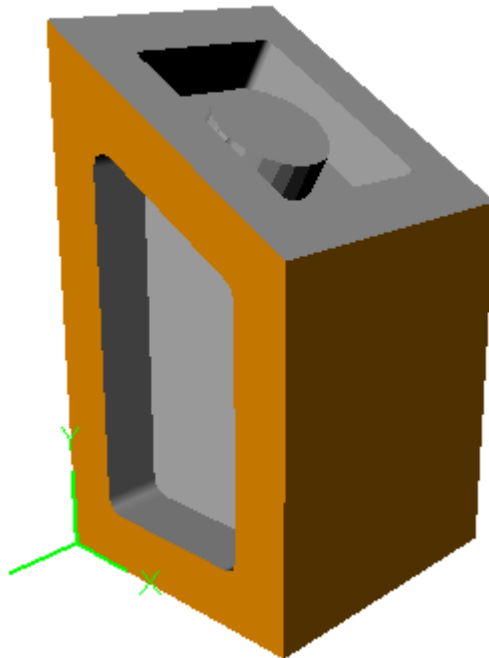
5. Select the inner region.



6. Using FlatMill2, create a Pocketing toolpath, with the same parameters as before.



7. Simulate the three toolpaths created so far.



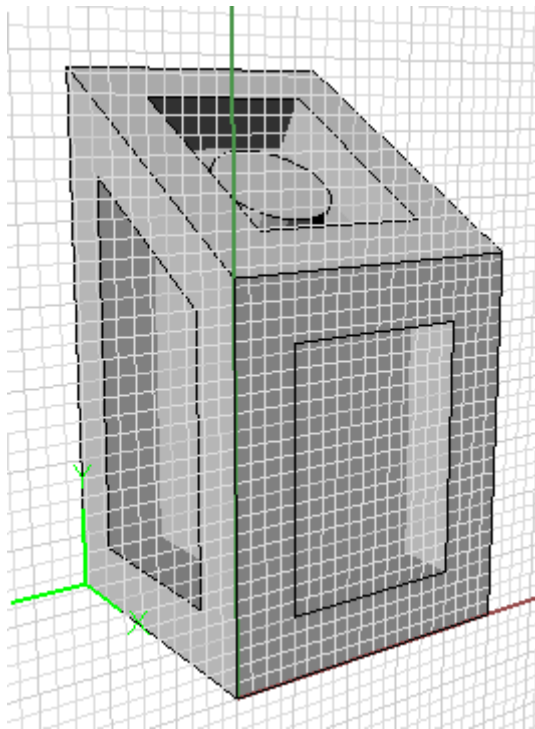
Machining the Front Face

In this section, the MCS will be changed once again, and the same 2.5 axis toolpaths will be applied.

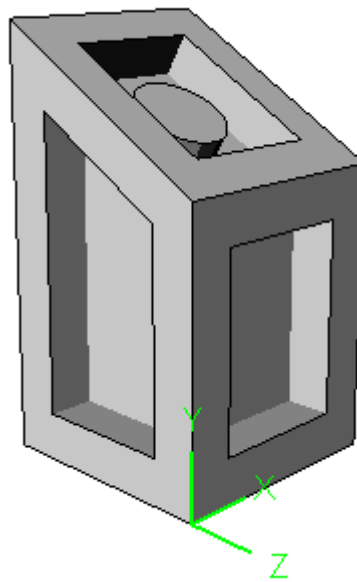


To machine the front face:

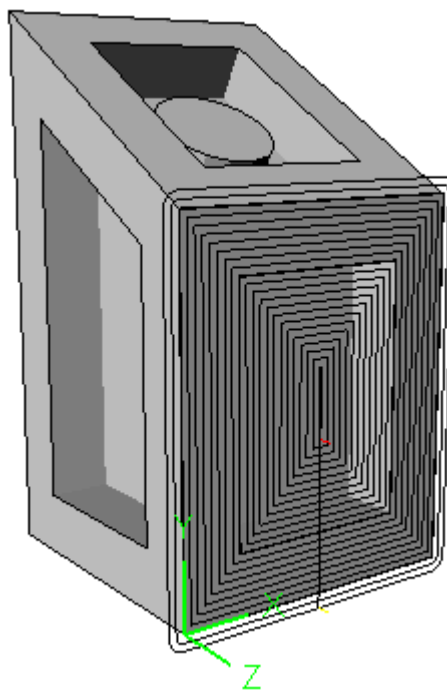
1. Use **CPlane 3 Points** again to change the construction plane to the front face as shown.



2. Set the MCS, using the **Set MCS** button to the new CPlane. This should create another Set MCS operation in the browser. Additionally the MCS should be coincident with the front face now.

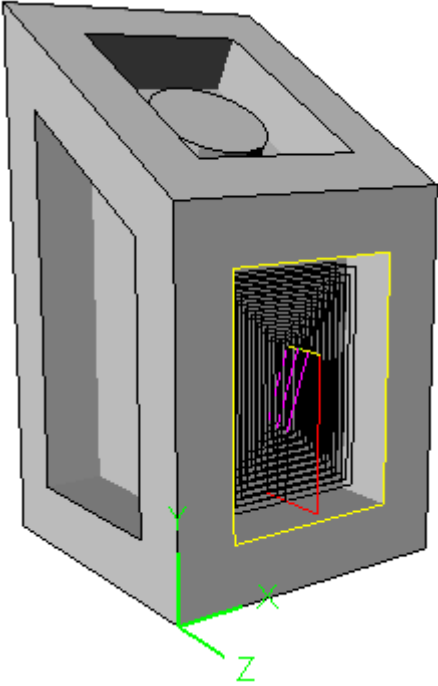


3. Create the Facing toolpath as before, using the outer region of this face.



4. Create the Pocketing toolpath as before, using the inner region.

Getting Started with RhinoCAM



- 5. Simulate all the toolpaths. The tool switches orientation from face to face, in order to machine each face at the correct angle.



Where to go for more help

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

If you need additional help, or if you have any questions regarding RhinoCAM, first try the FAQ section on our web site, www.mecsoft.com or www.rhinocam.com . Most of the common questions that users face are answered here. Also visit the Training section of our web site for more comprehensive training material such as tutorials and videos. You could also visit our Users Forum at our web site to learn from other RhinoCAM users. You are also welcome to contact our friendly technical support team via e-mail at support@mecsoft.com.