

RhinoCeros

RhinoCAM® 

D
AN
IELS
DIGITAL
FABRICATION

Rhino CAM 2020/2023 Manual
AXYZ 4008
Roland MDX 540

About this manual

Each page in this manual follows a general layout of three columns. Read left, to right, to the bottom of each column, then move to the next.

Important keywords are **bolded**, values are **highlighted** in red. On images, areas of interest are highlighted with red arrows or squares/circles. External links will precede a ↗.

Table of contents

The table of contents can be found on the next page. Chapter titles and sections are hyperlinked. You can click on a chapter title or a section to quickly navigate. You can click on the **video** ↗ button to watch the video for that chapter.

Chapter cards

Clicking on chapter titles along the bottom bar will also take you to its chapter card. The section titles are hyperlinked, and to the right, corresponding timestamps are hyperlinked for the accompanying video for that chapter.

This manual was updated September 18, 2023.

You can see the section you are currently in.



Click here to go to the table of contents.

Click on a chapter to quickly go to its title card.

Contents

Introduction

Basics	4
Typical Operations	6
Installing & Licensing	8
The UI	10
Tools	13

Getting Started

Setting Up	17
Regions	19
Simulating	22
Feeds & Speeds	24
Creating Toolpaths	27

2D Toolpaths

What are 2D Toolpaths	29
Common 2D Settings	30
Hole Pocketing	33
Pocketing	36
Profiling	39
Engraving	42

3D Toolpaths

What are 3D Toolpaths	45
Common 3D settings	46
Horizontal Roughing	48
Clear Flats	50
Horizontal Finishing	52
Parallel Finishing	54
Radial Machining/Spiral Machining	56

4D Toolpaths

Equipment	59
Setting Up	60
Roughing	63
Finishing	69

Submitting

Estimating Machine Time	74
Calculating Cost	74
Submitting Process	74

Troubleshooting

Common Issues	76
---------------	----

Quick Links:

1. Feeds & Speeds chart
2. Videos Playlist ↗
3. How to access FTP ↗
4. How to setup VPN ↗

Introduction

[video ↗](#)

Basics	4	-
Typical Operations	6	-
Installing & Licensing	8	Skip to 0:10 ↗
The UI	10	Skip to 1:41 ↗
Tools	13	Skip to 2:46 ↗

Basics

Basics

A CNC Router moves a fast-spinning spindle and tool in a number of axes (usually 3). Each axis is controlled by an individual stepper or servo motor.

RhinoCAM is used to translate toolpaths into directional instructions that the CNC machine can understand.

To achieve complex motions, RhinoCAM instructs the CNC machine to move multiple axes together, each with its own speed and direction.

This manual will cover the **AXYZ 4008 3-Axis CNC Router**, and the **Roland MDX 540 4-Axis CNC Mill**.



Setting up a RhinoCAM Project

1. Choose your stock

What material and size are you cutting and is it available in the shop? Often using a slightly larger stock and cutting out your pieces with a profile cut works best.

2. Reference your geometry

You will mostly use curves to define your toolpaths.

3. Pick your tool(s)

You can use this guide as a base to pick a tool, but it may change depending on what's available.

4. Cut Depth/Cut Parameters

You can adjust this parameter to customize the number of cuts you will use. The number of cuts you choose depends on the material being cut.

5. Set your Feeds and Speeds

You can use the table provided in this guide as a base to choose your feeds and speeds. These values are heavily dependent on the material.

6. Simulate

You can see exactly what the CNC will do through the simulation function. Using this feature, you will see what your end result will look like, and if there are any collisions you need to take care of.

FAQ

1. Why did I get a pop up saying units are not supported?

RhinoCAM will only work if your Rhino file units are set to **inches**.

2. Can someone look over my file before I submit?

TAs work every weeknight from 6-10 pm to run files and help set up files. You can email cncmilling@daniels.utoronto.ca to set up a time to get feedback on your file before you submit.

3. When will I be scheduled to mill?

Once you have uploaded your file to the **FTP**, and emailed cncmilling@daniels.utoronto.ca with your submission form, a TA will book you into the next available slot that works with your schedule.

4. How long will my mill take?

Once everything is setup in your file, you can right-click on your setup, and select **"Information"** to get an estimation. Please note, you should add **40-60%** extra time to your estimation.

Notes

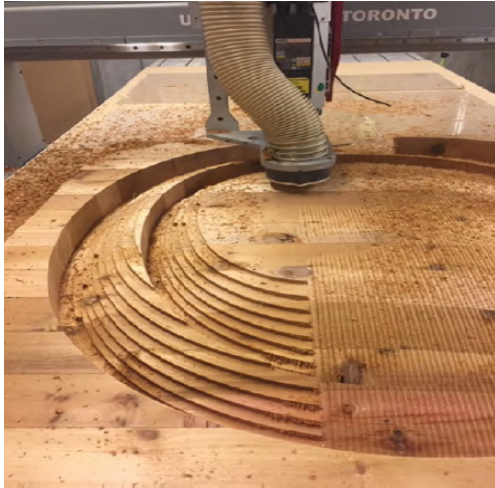
1. It can take up to 1 week during busy times to mill your file after you have submitted. Please plan ahead.

2. The toolpaths are based on your geometry. Having a clean model is critical to creating clean toolpaths.

3. This manual does not cover every toolpath available in RhinoCAM. If you are looking to create more specific toolpaths, you can contact a TA.

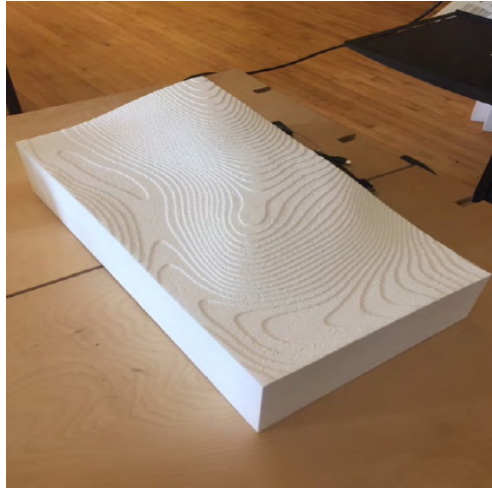
4. If you do not connect to the VPN while using RhinoCAM, you will not be able to save your toolpaths.

Typical Operations



Horizontal Roughing + Parallel Finishing

Almost 95% of projects are these two operations. It is used for creating smooth 3D surfaces quickly by first removing as much material with a larger bit, then going back and forth usually with a smaller bit to smooth out the contours.



Horizontal Roughing + Clear Flats

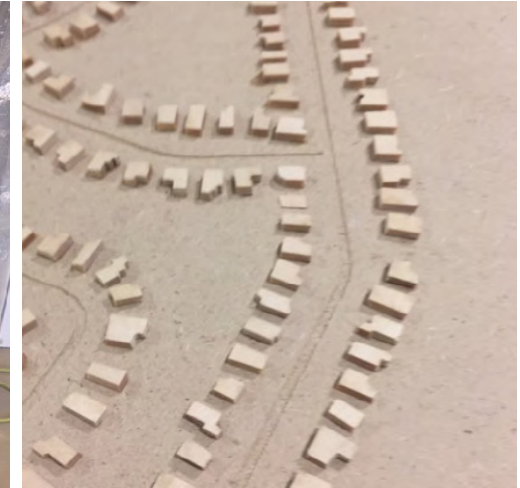
This is often used for stepped contour models. If you already have a stepped contour model in Rhino, you can use these two toolpaths to first remove the bulk of material, then use a smaller tool to finish the rest of the flat areas.



Profiling

This toolpath is usually used to cut out parts from a stock.

Profiling can also be used to clean up vertical edges after operations, such as the pocketing operation.

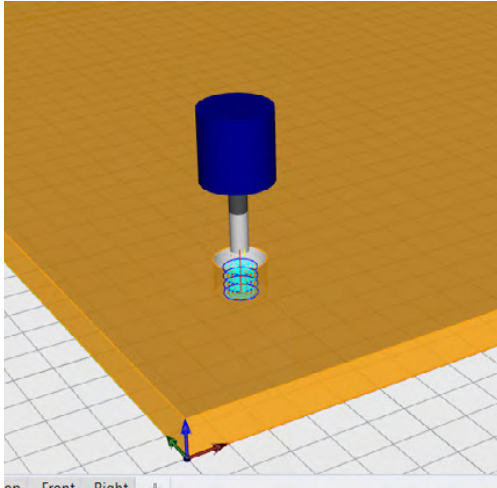


Engraving

This toolpath is often used to create engravings into the stock, for example to create roads or paths on a site model.

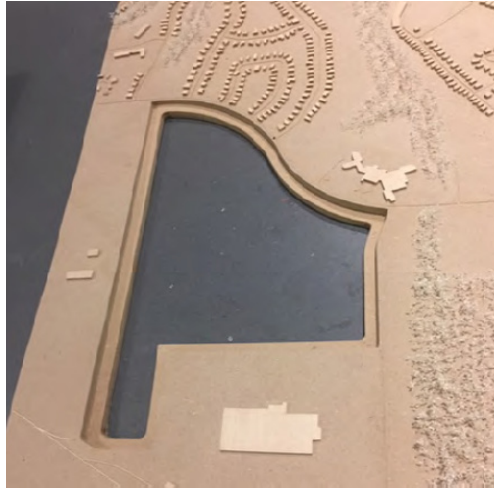
Engraving can also be used to clean up hard edges after a parallel finishing operation.

Typical Operations ctd.



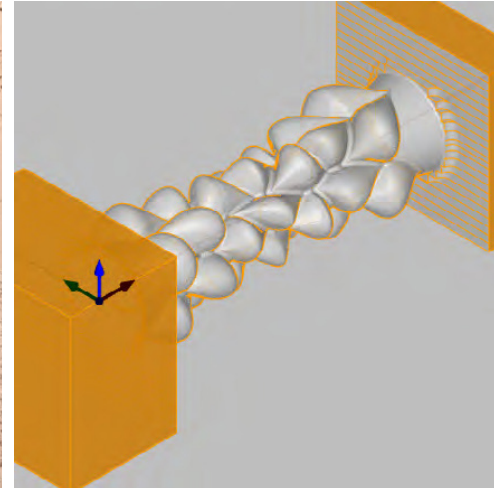
Hole Pocketing

This toolpath is used to create perfectly circular holes, which could be used for inserting a dowel or screw holes into your model.



Pocketing

This toolpath is often used to create inserts within a model. You can use this toolpath for models that require a plug.



4 Axis Milling

4 Axis operations allow the creation of mills that would be impossible with 3 Axis milling, as it can rotate the stock to mill all around the object.

Installing & Licensing

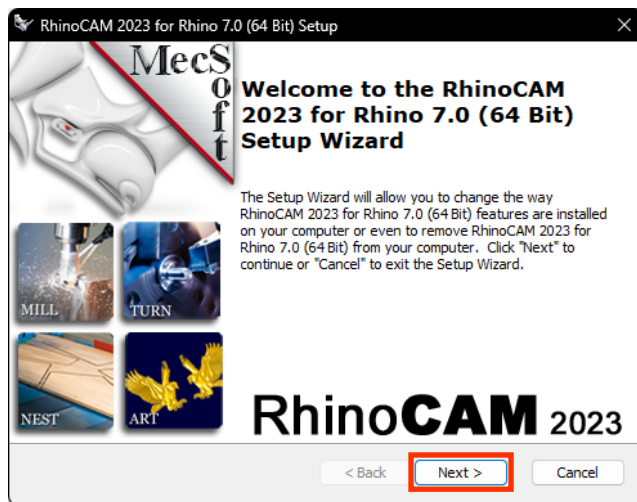
Installing RhinoCAM

The RhinoCAM 2023 installer can be downloaded from the following link:

<https://mecsoft.com/demo-rhinocam/>

Alternatively, the installer can be found on the FTP under “Groups > CNC Milling”. To connect to the FTP server, follow [this guide](#) ↗

Run the installer, then follow the instructions to complete the installation.



Licensing RhinoCAM

Once installed, launch Rhino. Simply close the two new pop-up windows that appear.

If the RhinoCAM 2023 Machining Browser does not appear when Rhino loads, you can enter “RhinoCAM2023” as a Rhino command to bring it forward.

In the Machining Browser window, click on “Preferences”, then navigate to the “Licensing” tab.

Check “Using Network Locked License Server”, and set the following values:

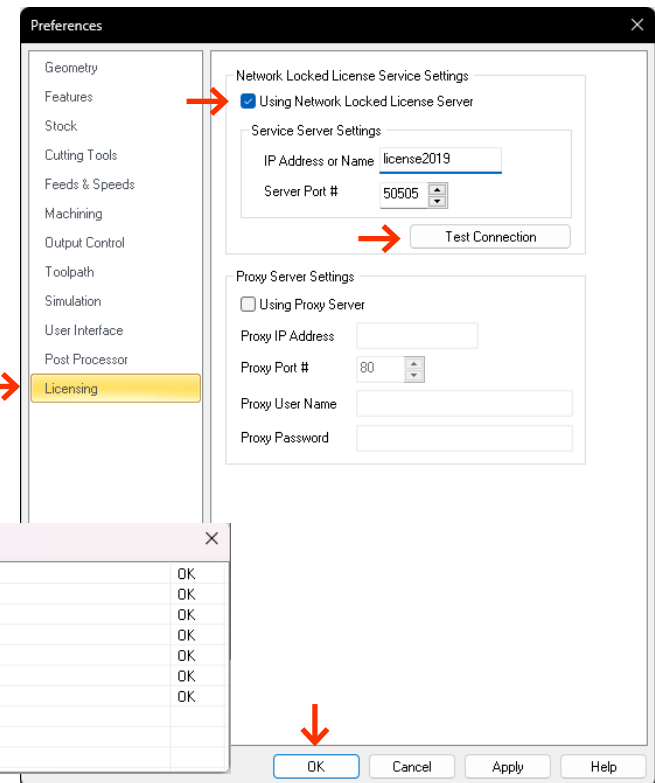
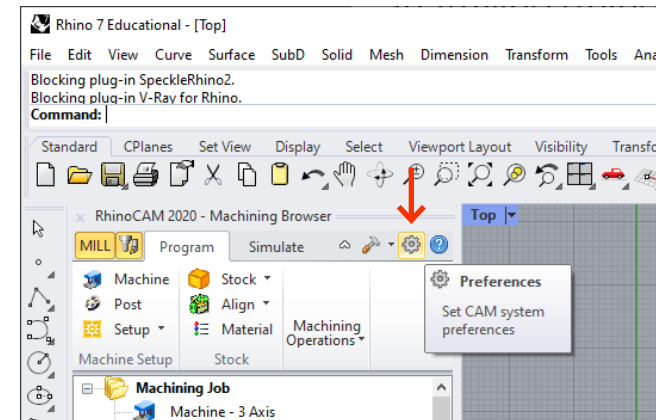
IP Address or name: **license2019**
Server Port #: **50505**

Click “Test Connection” to ensure the license server can be reached. You should see “OK” for each item in the “Diagnostics” window.

If you do not see “OK” for each item, ensure you are connected to the Daniels network either through a VPN or directly.

Click “OK” to close this window.

Restart Rhino to apply the license.



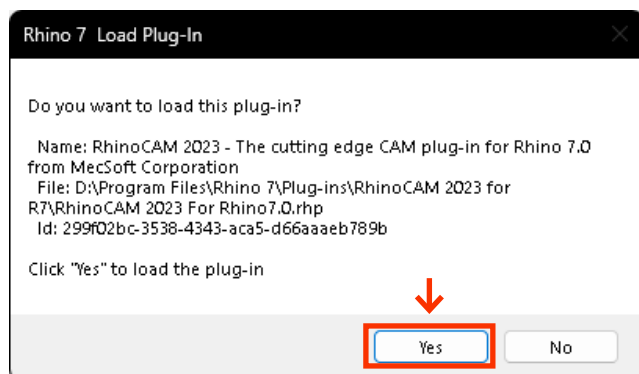
Manually loading RhinoCAM

Once RhinoCAM is installed, it will load every time Rhino is launched. To load Rhino faster, you may choose to only load RhinoCAM only when you wish to use it.

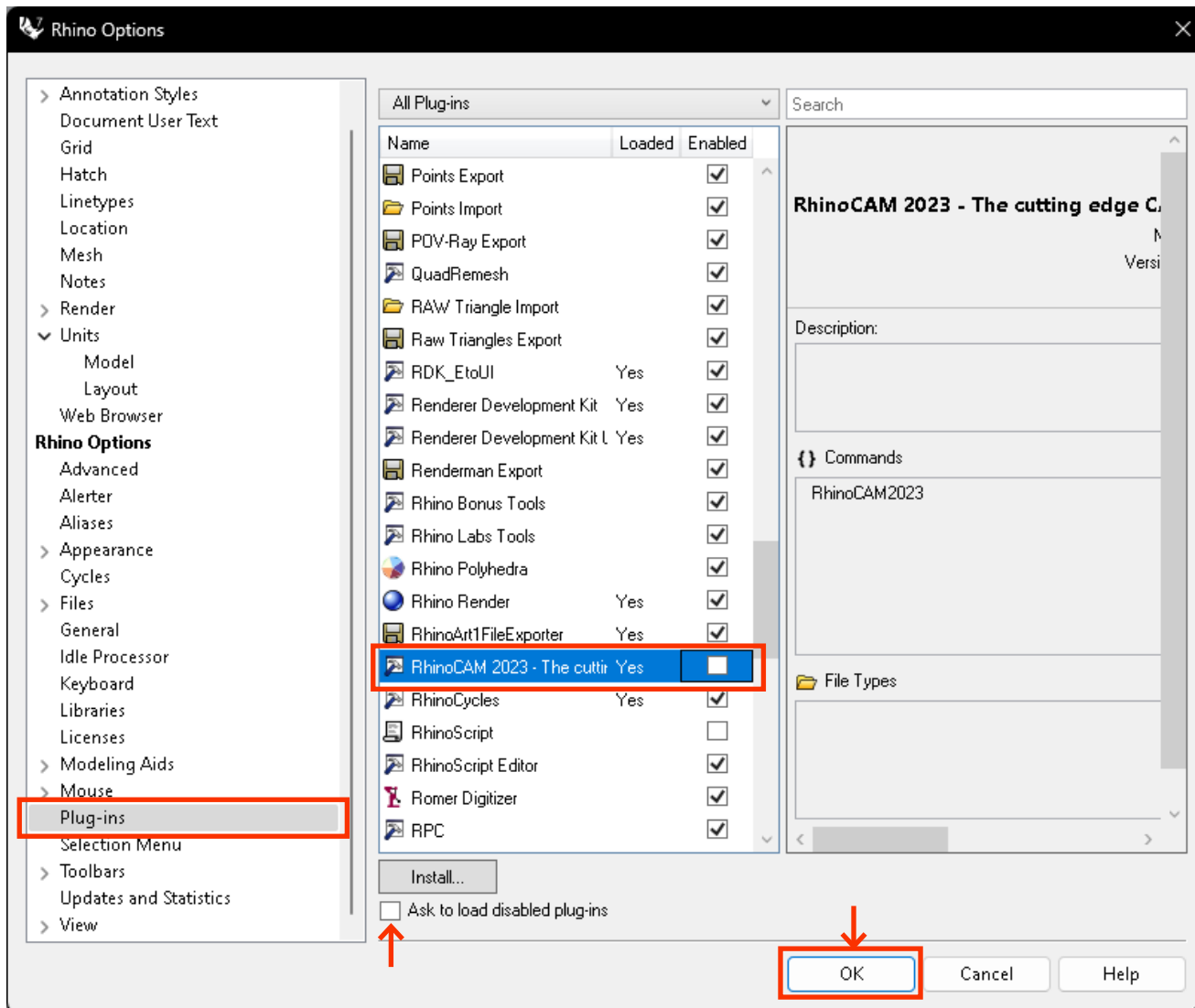
Enter “Options” as a command. In the Rhino Options window, navigate to **Plug-ins > RhinoCAM 2023**. Deselect **RhinoCAM**, and select “Ask to load disabled plug-ins”.

Click “OK”.

Now when launching Rhino, you will be prompted to load RhinoCAM:



Selecting “No” will prevent Rhino from unnecessarily loading **RhinoCAM**. Select “Yes” when you wish to use **RhinoCAM**.

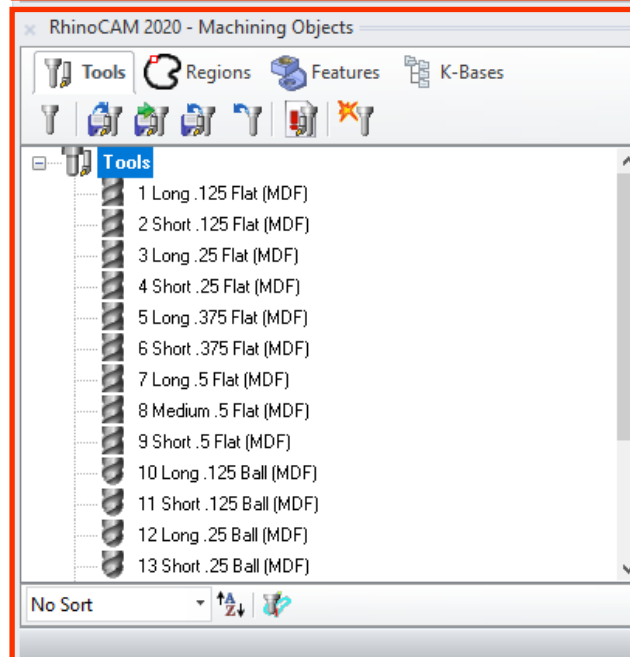
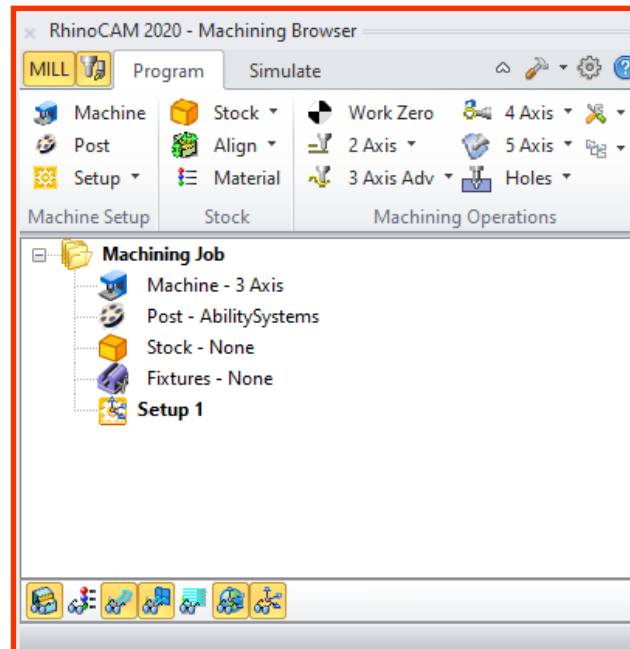


The UI

The RhinoCAM UI consists of two main windows: **Machining Browser** and **Machining Objects**. If you do not see these windows when you launch Rhino, click **RhinoCAM2023 > Mill** from the menu bar.

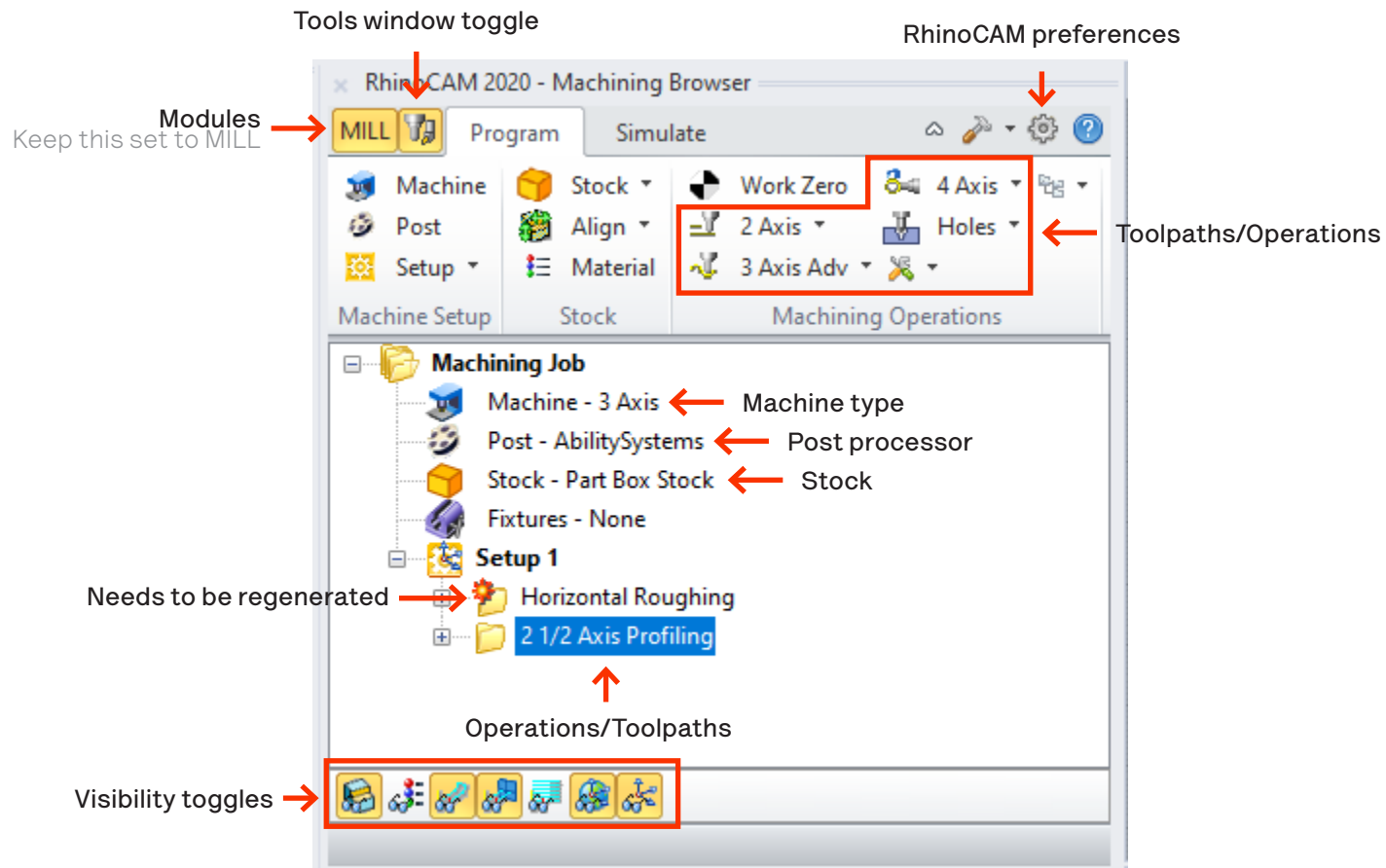
From the **Machining Browser** window, you can: edit preferences, create/edit setups, set stock, select operations, simulate, toggle visibility options.

Most of what you do will be in this window.

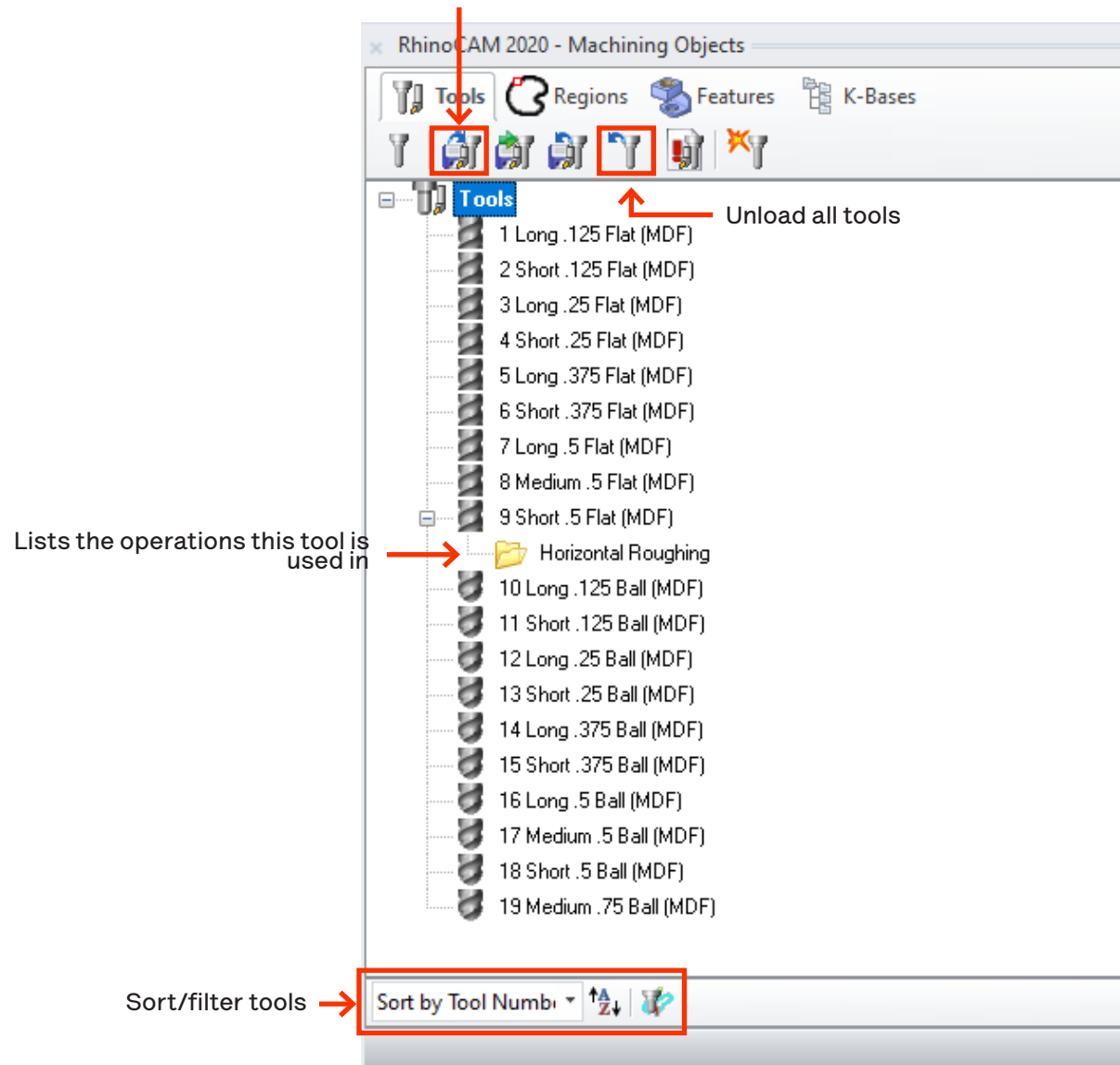


From the **Machining Objects** window, you can: load/edit tools, select machining regions, add custom bridge points using regions.

If you do not see this window, click on the **Machining Objects toggle** button shown on the next page.



Load tool library



Tools

There are three main forms of tools/bits/endmills.

Tools of the same form are usually available in various different diameters. A larger tool can remove material faster, but cannot get into the nooks and crannies.

Try not to use the smallest tool for your entire operation. Instead, use larger tools for roughing, then use a smaller tool for finishing up. This will save you time, and prolong tool life.

You can find several tool libraries for milling in the following Daniels **FTP server** directory:
Groups > CNC Milling > Tools

How to access FTP ↗

Download them to your computer so you can reference them in RhinoCAM.

Tool libraries are arranged by material type. **Only import tools appropriate for the material you will be working with.**



Flat/Square

Used for flat surface finishes, pocketing, profiling, and toolpaths that require 90° bottom corners.



Ball-nose

Used mainly for 3D toolpaths which produce smooth 3D surfaces.

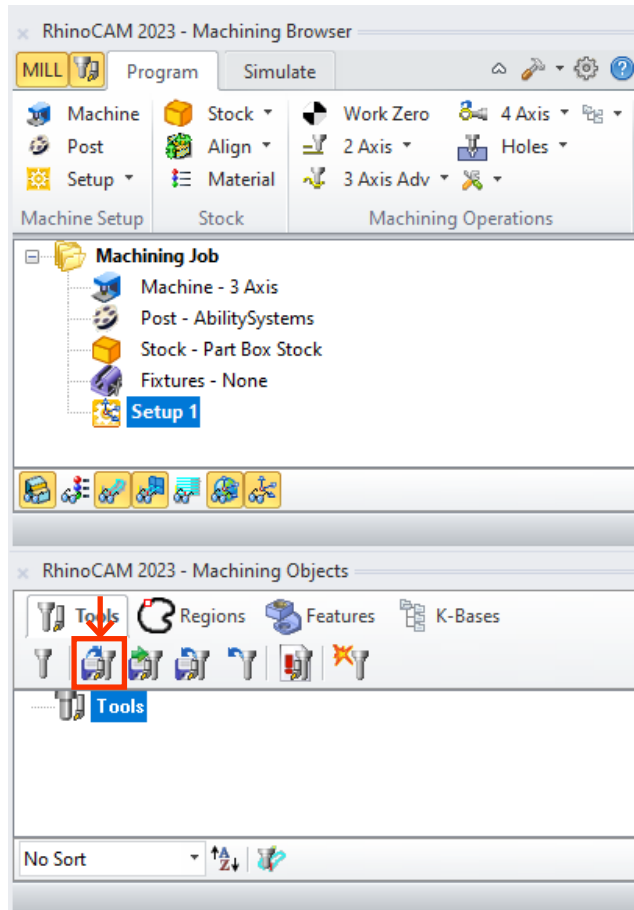


Chamfer/V-bit

Can be used for V-caving, or for creating chamfered edges.

Tools ctd.

To load tools, in the Machining Objects window, click on the “Load Tool Library” button.

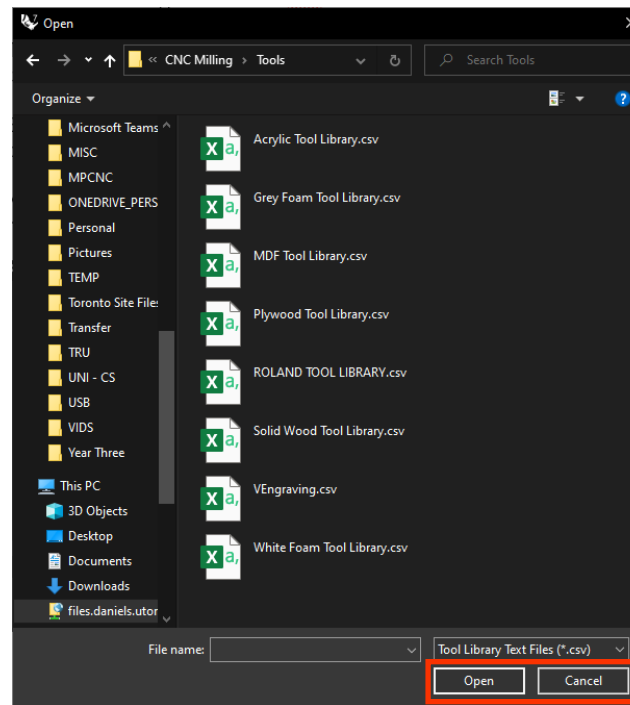


When presented with the open file dialog window, ensure the file type is set to “.csv”

Then navigate to **Groups > CNC Milling > Tools** on the **FTP server**.

Select the Tool Library you want to import for your project, then click “Open”.

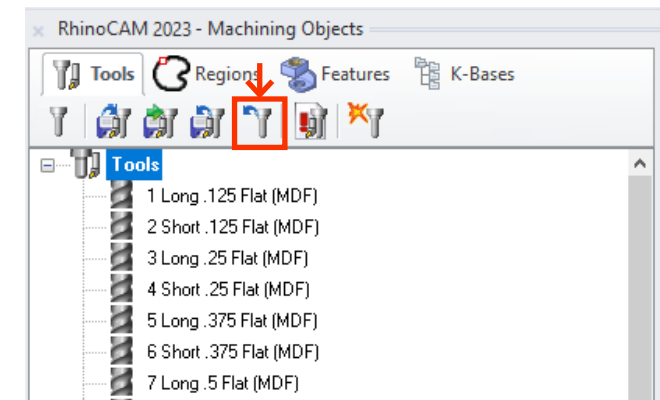
Only load the tools appropriate for your material.



The tools will be imported to your project.

The number to the left of each tool is the tool number. The CNC machine uses this number to guide the auto tool changes.

You can unload tools to load in a different library by using the “Unload Tools” button.

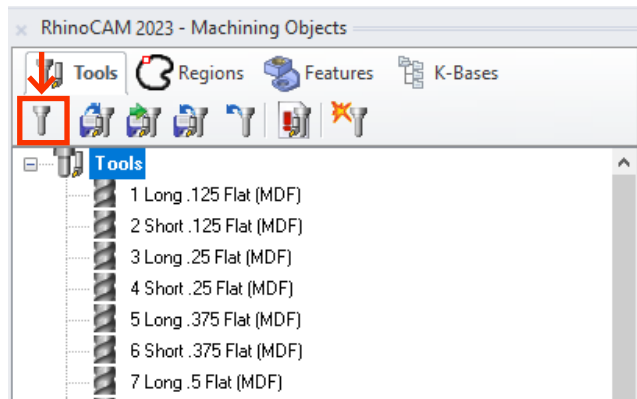


Creating your own tools

You can create your own tools or edit existing tools by inputting the correct tool measurements into RhinoCAM.

To create your own tool, in the Machining Objects browser, click on the “**Create/Edit Tools**” button.

This will bring up the **Create/Select Tool** window.



In this window, **start by entering a new name for the tool.**

Then, enter all of the appropriate information, such as the **Shank Diameter**, **Tool Length**, **Shoulder Length**, **Flute Length**, and **Tool Diameter**. The tool preview will update as the values are entered.

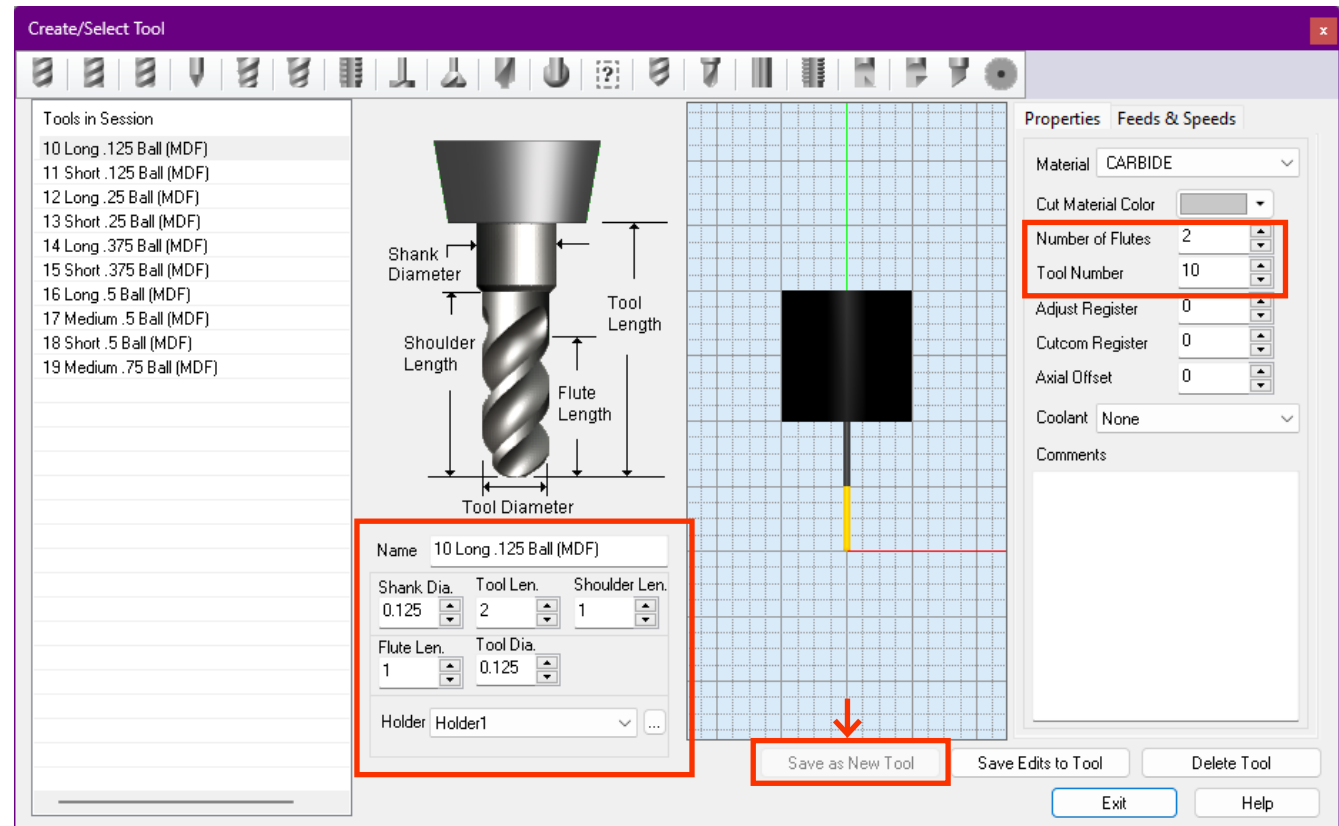
On the right, enter the **Number of Flutes**, and a unique **Tool Number** that is not used by any other tools in your current setup. The machine uses the tool number as reference when picking up tools via **ATC (Automatic Tool Change)**.

You can optionally enter the preferred **Feeds & Speeds** for this tool in the appropriate tab. This will be covered in the next chapter.

To save the tool, click “**Save as New Tool**”. Note that it will not be clickable until you enter a new tool Name.

Then click “**OK**”.

You can now use your newly created tool in machining operations.



Getting Started

Setting Up	17	Skip to 0:19 ↗
Regions	19	Skip to 1:56 ↗
Simulating	22	Skip to 4:25 ↗
Feeds & Speeds	24	Skip to 6:14 ↗
Creating Toolpaths	27	-

Setting Up

Setting up your file

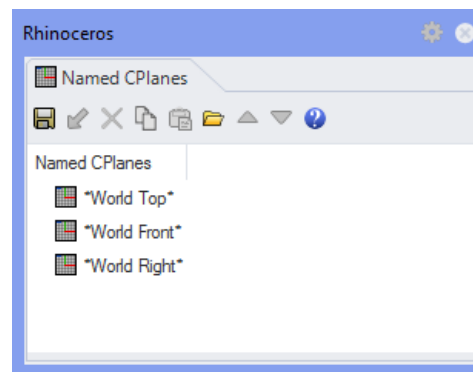
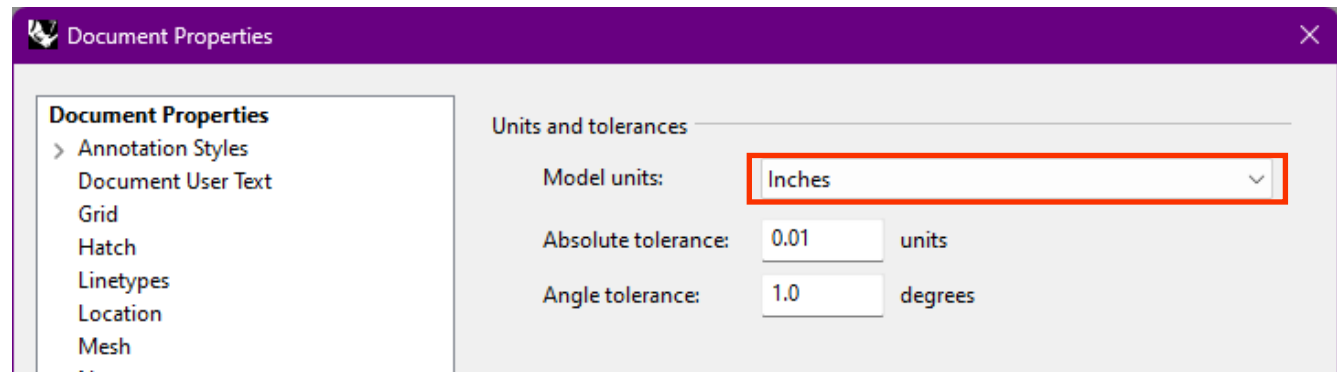
To begin, export only what you want to cut as a separate Rhino file, then open it. Unnecessary geometry may lead to errors in your toolpaths.

1. Your **file units** must be in **inches**. Enter “**Units**” as a command to bring this window forward.
2. Your **CPlane** must be set to “**World Top**”. Enter “**NamedCPlane**” as a command to bring this window forward.

For 2D operations, ensure your control geometry lies on the **top surface of your stock**.

For 3D operations, ensure your 3D model is entirely within the stock, and as close to the top surface of the stock as possible. This will ensure the CNC router has to remove the least amount of material to reach your model.

You are now ready to define the stock.



Defining your stock

Enter “**Box**” as a Rhino command, then type “**0**” to start a box at the origin. Then, enter the **X**, **Y**, and **Z** dimensions of your stock. Press enter to finish the command to create your stock in Rhino.

Dimension your stock in Rhino as accurately as possible to your physical stock. It is especially important to know the thickness of your stock in order to cut entirely through it without cutting into the spoilboard or stopping short of the bottom of the stock.

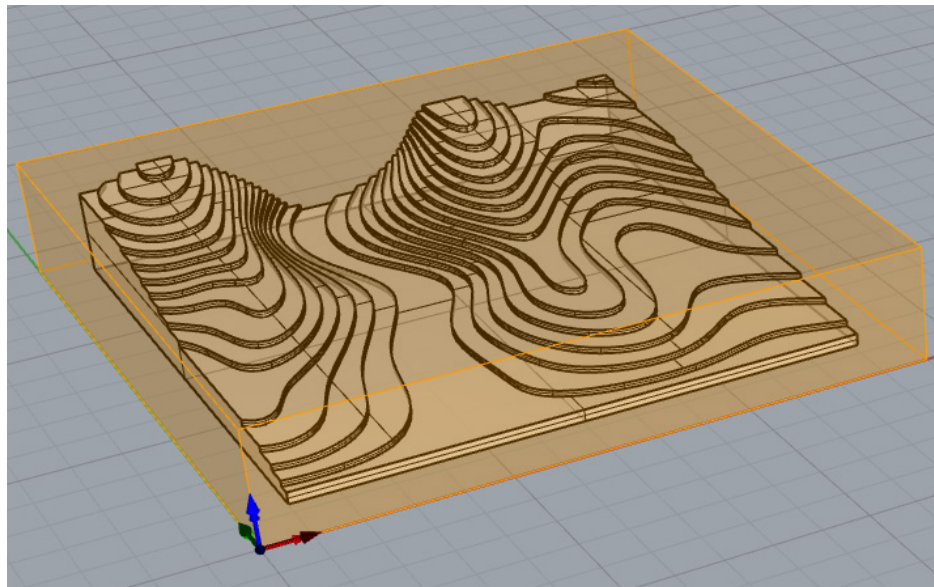
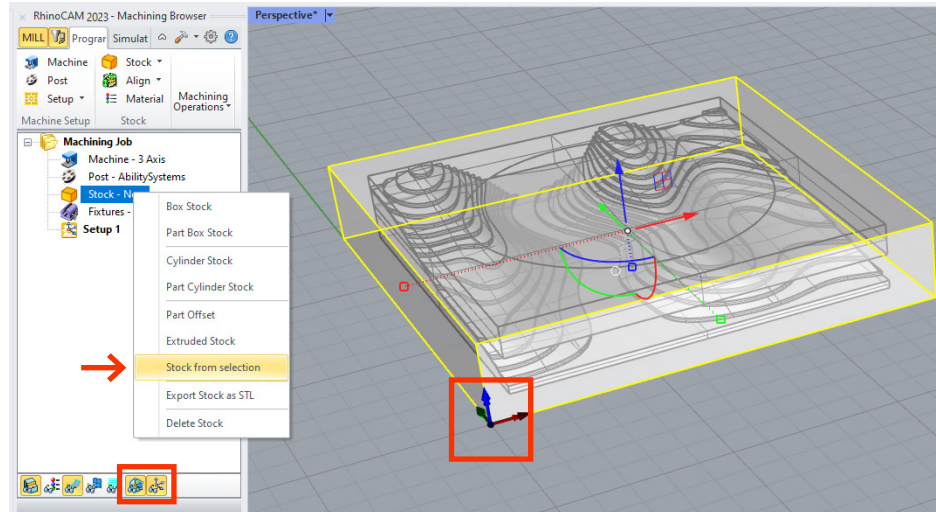
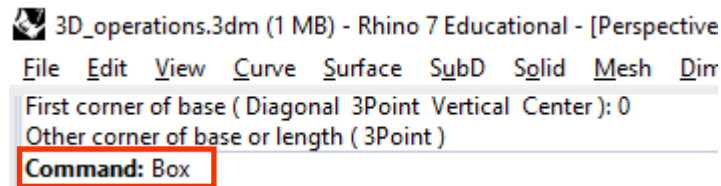
Ensure the bottom left corner of your stock is at the world/machine origin as shown.

With the Rhino stock selected, right-click Stock and set “**Stock from Selection**” in the Machining Browser window.

Create a layer for your Rhino stock, then hide it after you have defined your stock in RhinoCAM. You may see a ghosted view of your stock that you cannot select. This is the RhinoCAM representation of your stock, you can toggle it via the **visibility bar**.

If you do not hide your Rhino stock, it may yield unintended results from your toolpaths, especially in 3D operations.

Again, for 2D operations, ensure your control geometry is on the top surface of your stock.



Regions

After you have defined your stock, you can optionally define your control geometry as “Regions”.

This is an optional step because you can select control geometry directly from the **Control Geometry** tab of every operation.

The Regions tab is located in the “**Machining Objects**” window, beside the **Tools** tab.

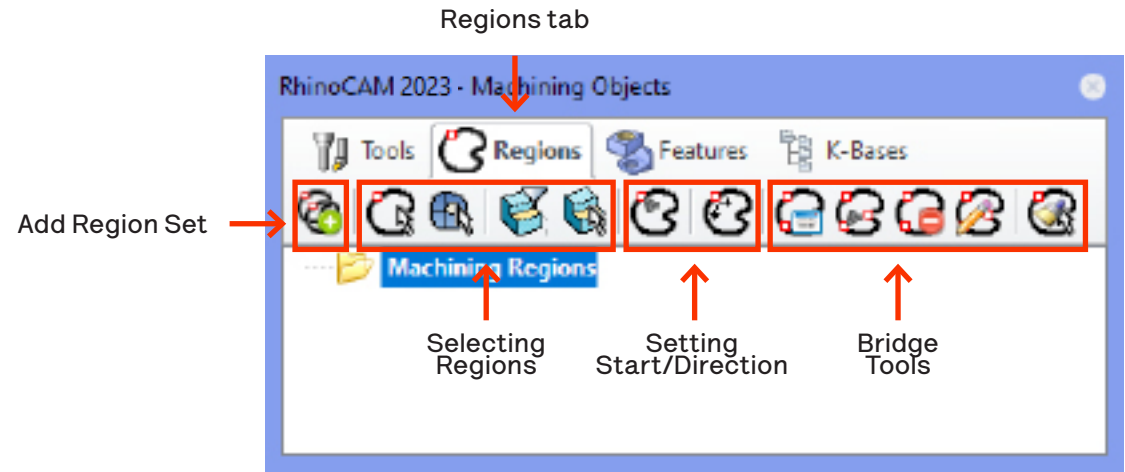
Instead of having to repeatedly select curves for each of our operations, you can define regions to then quickly select for the appropriate toolpath.

You can also change the direction of the toolpath, change the starting point (**profile** and **engrave** only), and add **custom bridge points** via regions.

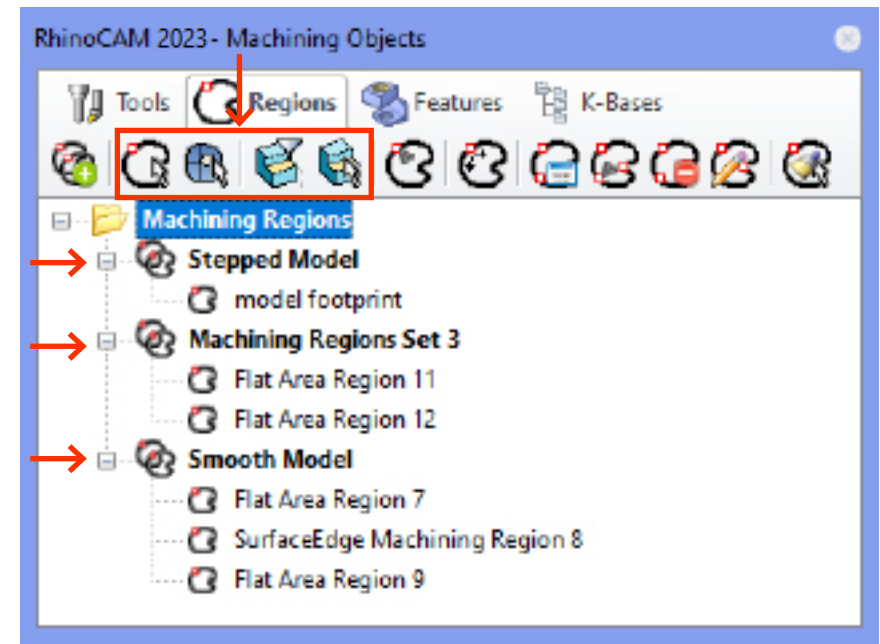
Region Sets can be used to group multiple regions to select them together. You can drag and drop regions into **Region Sets** to better organize your regions.

When working on toolpaths, switch to the **Tools** tab in order to hide region preview in your viewport.

There are 3 ways to select regions for your toolpaths. We will cover them in the next pages.



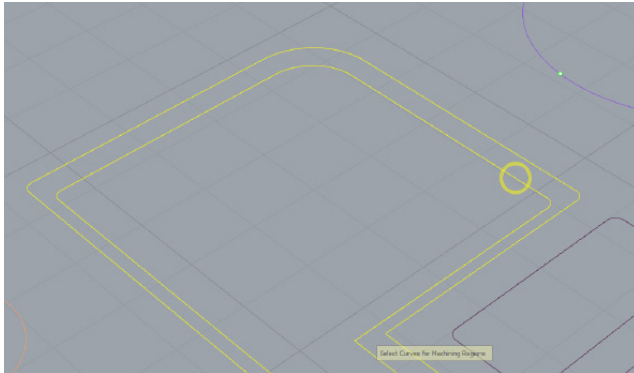
Select curves, select surface edges, flat area selection filters, select flat areas



Regions ctd.

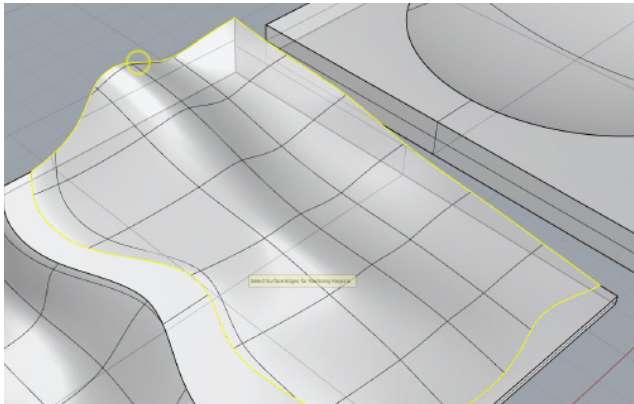
1. Selecting Curves

This selection tool allows you to exclusively select any Rhino curves. It is useful for **Profile** and **Engrave** toolpaths as it enables **custom bridges**.



2. Selecting Surface Edges

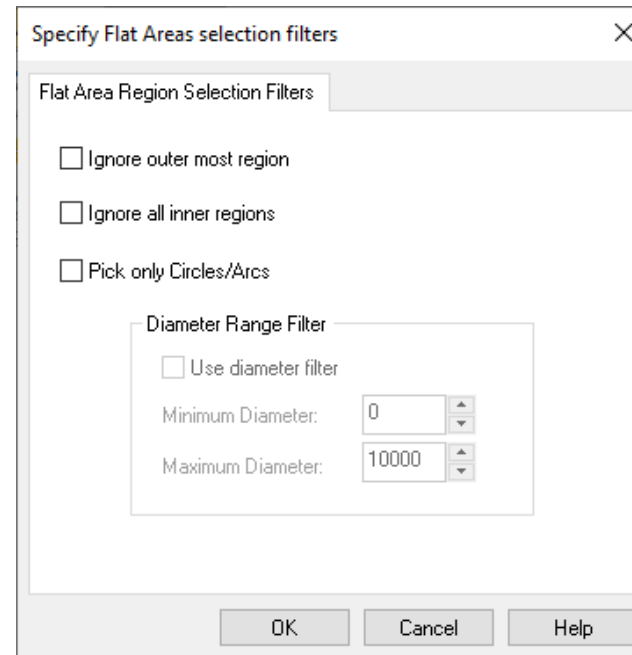
This selection is useful for selecting boundaries of curved 3D surfaces.



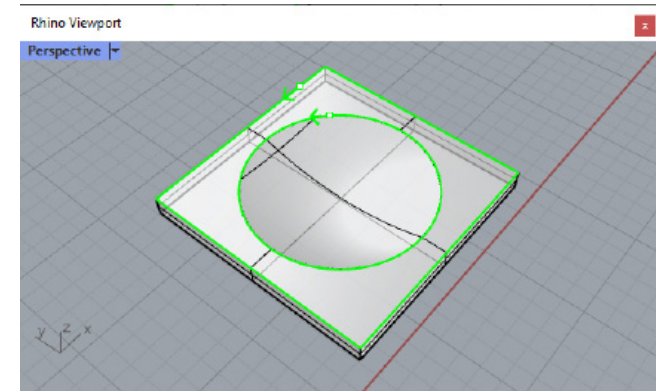
3. Selecting Flat Surface Edges

This tool makes it easy to select both internal and external boundaries of flat surfaces automatically.

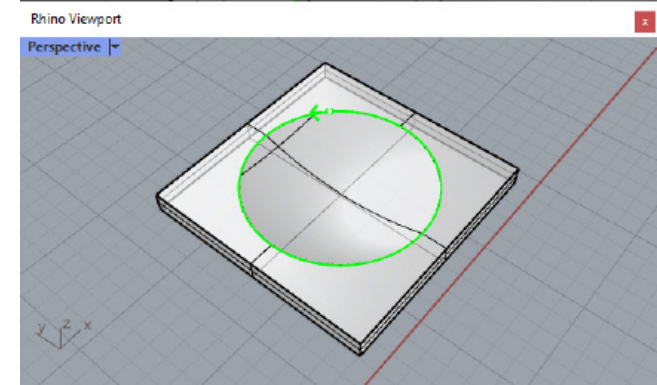
Before selecting surfaces, ensure the **Flat Area Selection Filters** are set up correctly.



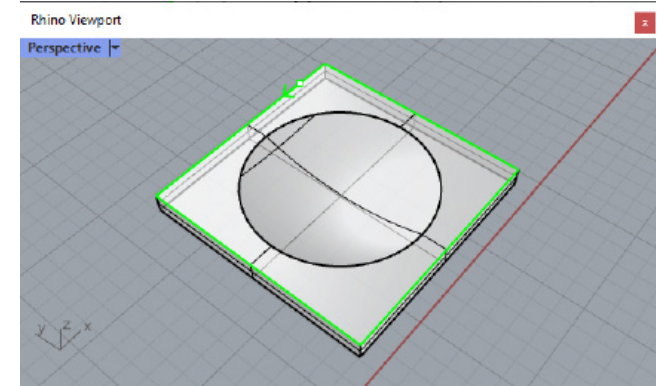
In general, use **Select Curves** for 2D curves, and **Select Surface Edges** for 3D models.



No selection filters



Ignore outer most region



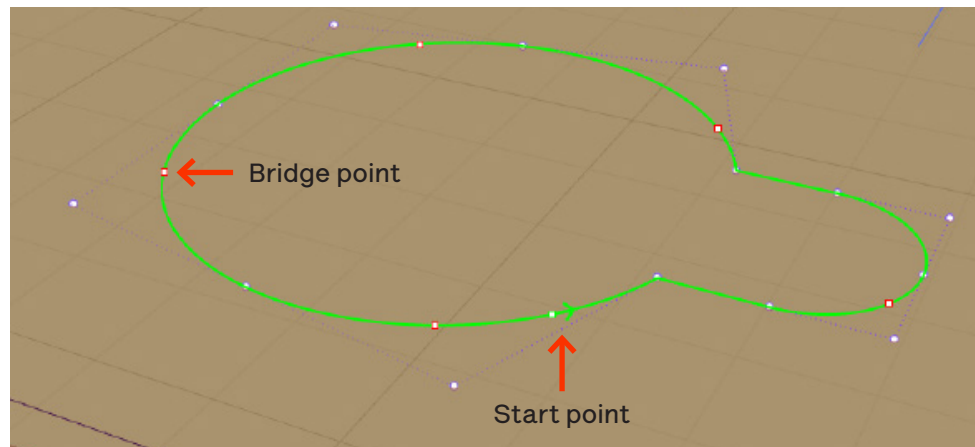
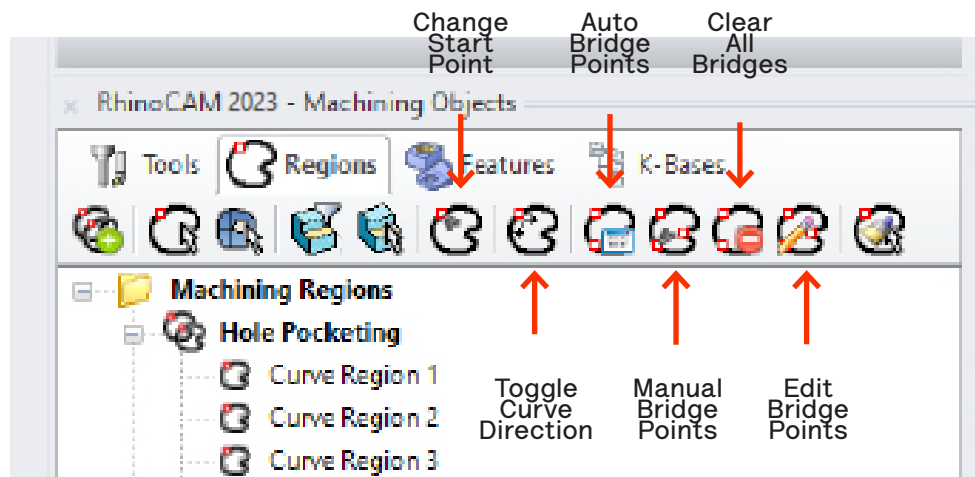
Ignore all inner regions

Custom bridge points

The **Regions** feature also allows you to add or edit **custom bridge points** on selected regions.

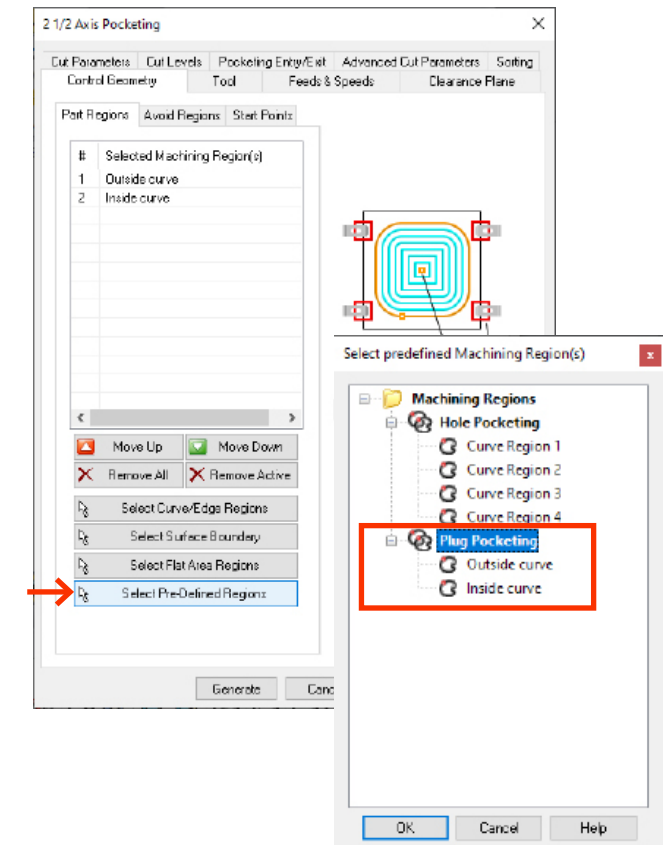
Bridges are useful in holding your piece to the larger stock when profiling. Pieces that become dislodged during milling may become damaged, or damage the machine.

You can also set the **startpoint** used by profiling and engraving operations, as well as the **cut direction**.



Using regions in toolpaths

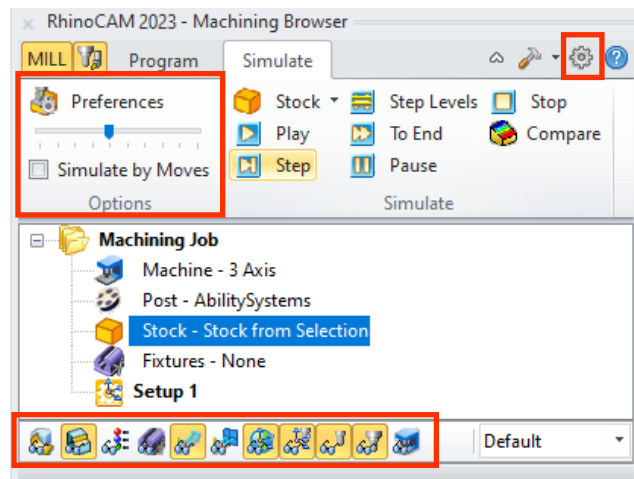
Once **Regions** are setup, you can pick them when creating a new operation/toolpath.



Simulating

Switching to the **Simulate** tab allows simulation and playback of the established milling operations. You can come back to this section after you have set up your toolpaths in the next chapters.

The slider in the options column of the **Simulate** tab allows you to control simulation speed. You can deselect “**Simulate by Moves**” to further slow down your simulation. At any point, you can **Stop**, then click **To End** to see the end result. **Step** allows you to run through the simulation step by step.

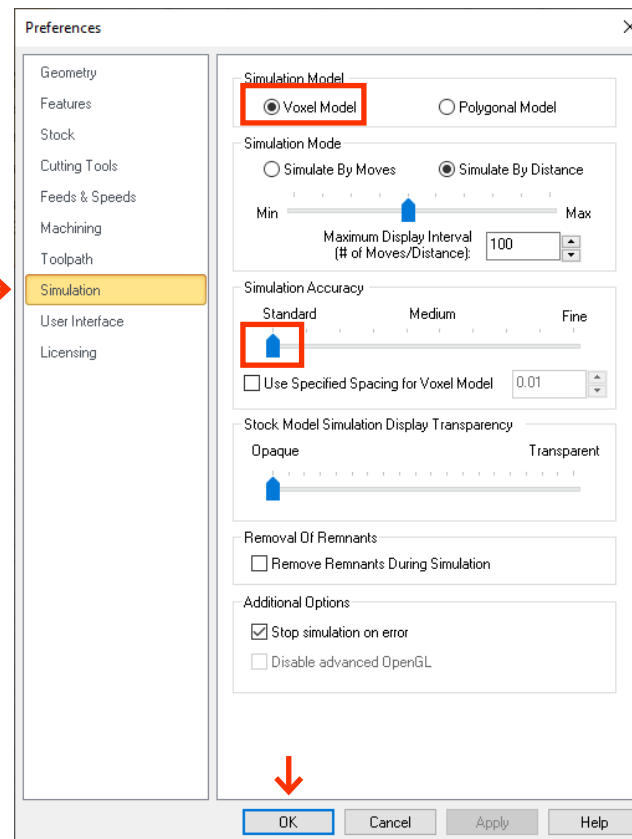


The **visibility bar** has additional toggles while in this tab. You can toggle: your part, your stock, toolpath previews, tool visibility, holder visibility.

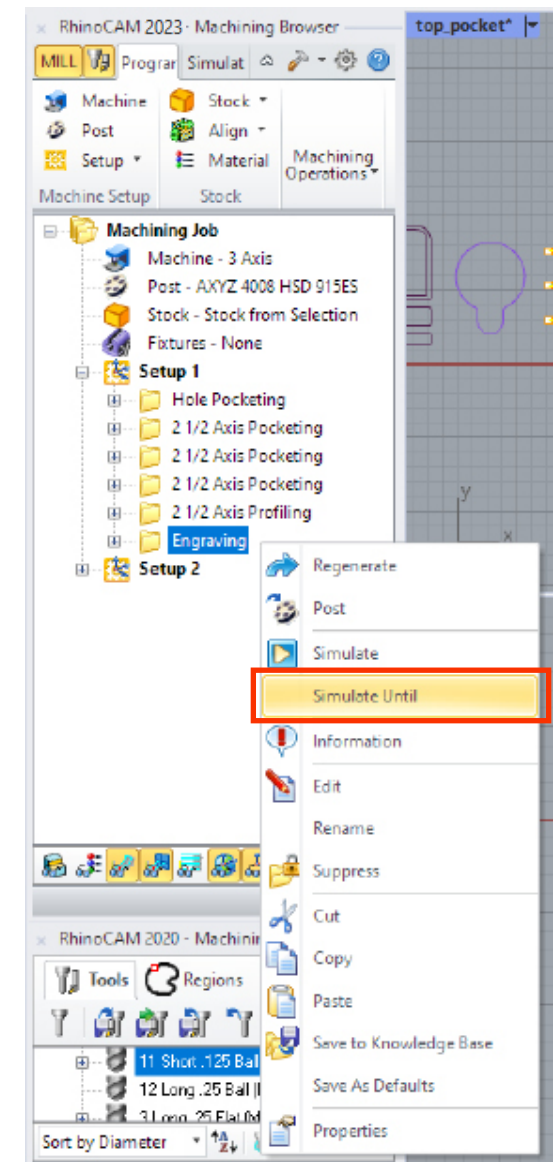
Click the **gear icon** to go open the **preferences** window. Under **Simulation**, you can switch between “**Voxel Model**” and “**Polygonal Model**”. The **Polygonal Model** is more accurate, but slower to compute.

Under **Simulation Accuracy**, you can adjust the simulation accuracy using the slider. A more accurate simulation will take longer to compute, and may produce a choppy simulation.

On larger models with a lot of complexity, using a **Voxel Model with standard accuracy and reducing Maximum Display Interval** can speed up the simulation playback.



You can select “**Simulate Until**” on any toolpath to instantly run through all toolpaths before it and present you the final result.

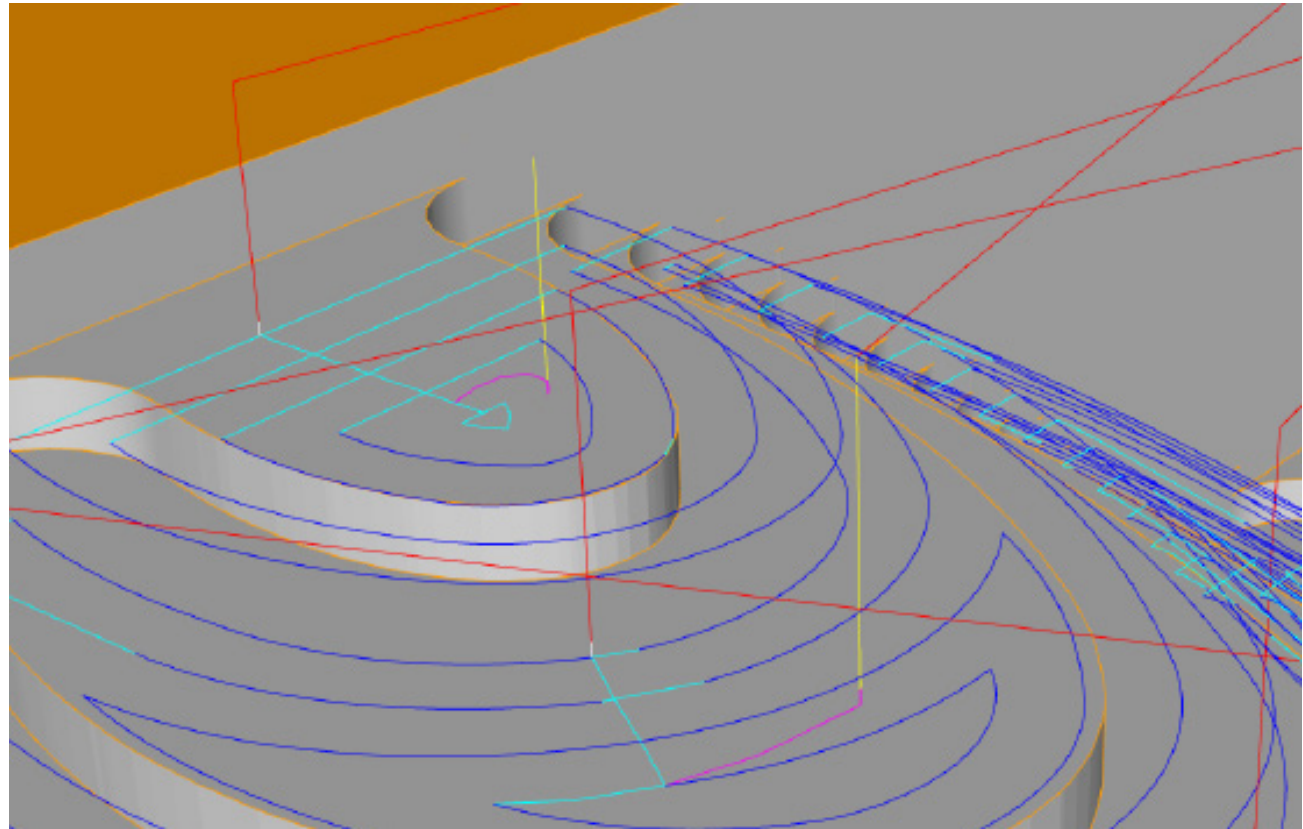
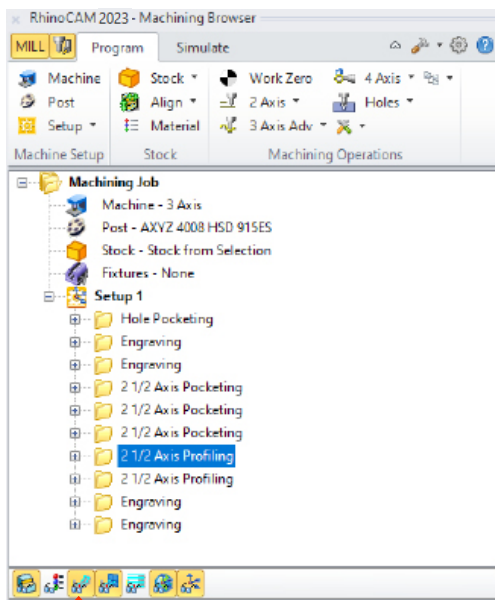


Toolpath preview

You can always preview your toolpaths by first selecting the toolpath in the **Machining Browser** window, then ensuring “**toolpath visibility**” is toggled on in the visibility bar below.

The toolpath preview allows you to see every movement of the tool in the selected toolpath.

It can be useful in quickly identifying issues with your toolpath, such as not enough stepdown levels, or incorrect control geometry selection.



The line colours in toolpath preview have meaning.

Yellow lines are where the tool enters the stock.

Red vertical lines are where the tool exits the stock,
Red horizontal lines are transfer moves that show the tool heading to its next entry location.

Magenta curves are where the tool begins cutting into the stock. They may appear as spirals, or ramps along a path as set in the Entry/Exit tab of the toolpath.

Cyan curves are linear cut moves. They are mostly used for straight moves, but can be broken up to approximate curved moves.

Blue curves are arc cut moves.

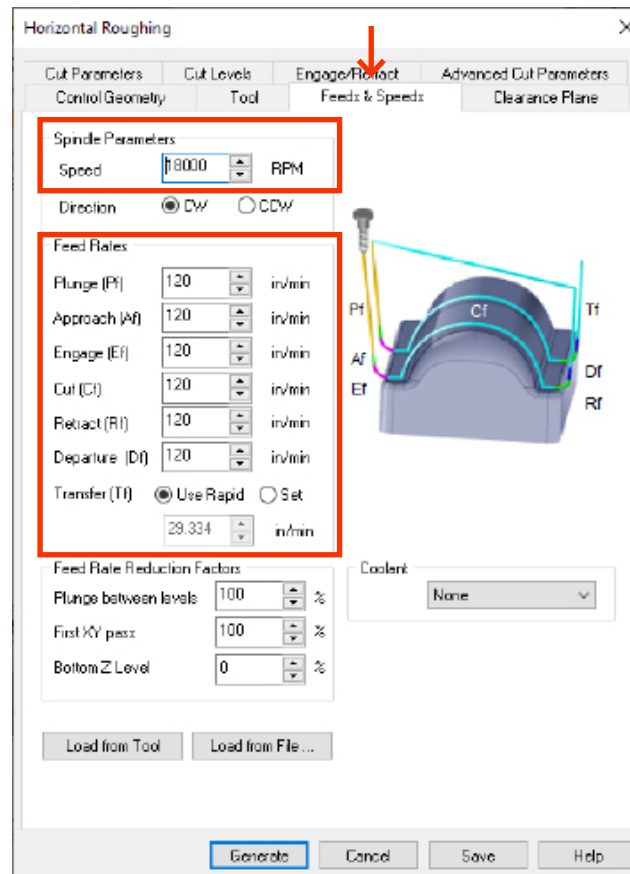
Feeds & Speeds

For every operation the **Feeds & Speeds** of the machine need to be set. The **speed** is how fast the tool spins within the spindle, the **feed** is how fast the tool moves into the stock.

The speeds and feeds are dependent on the operation you are performing and what material you are using. The chart below outlines the speeds and feeds for each operation and material.

They can be set under the **Feeds & Speeds** tab of each operation and example of this is to the right.

You can use the chart on the next two pages as reference when setting **Feeds & Speeds** for your toolpaths.



A TA will finalize your Feeds & Speeds.

	MDF			Plywood			Solid Softwood			Solid Hardwood		
	Roughing	Finishing	Profiling	Roughing	Finishing	Profiling	Roughing	Finishing	Profiling	Roughing	Finishing	Profiling
Speed	16000 RPM			16000 RPM			16000 RPM			18000 RPM		
Feeds	300 in/min	300 in/min	120 in/min	150 in/min	175 in/min	125 in/min	250 in/min	150 in/min	125 in/min	120 in/min	150 in/min	100 in/min
Suggested Tools	0.5"-0.75" Round or Flat	0.5"-0.75" Round or Flat	0.25"-0.75" Flat	0.5"-0.75" Flat Compression or Chipbreaker*	0.5"-0.75" Round or Flat Compression**	0.25"-0.75" Flat	0.5"-0.75" Round or Flat	0.5"-0.75" Round or Flat	0.25"-0.75" Flat	0.5"-0.75" Round or Flat	0.5"-0.75" Round or Flat	0.25"-0.75" Flat
Suggested % Step Down of Tool Diameter	75%	25%	50%	50%	25%	50%	50%	25%	50%	25%	25%	25%

Chipbreaker** bits will leave serration marks on your vertical surfaces. *Compression** bits decrease the chance of plywood layer delamination during milling.

	White Foam (gray bits)			White Foam (gold bits)*			High Density Foam		
	Roughing	Finishing	Profiling	Roughing	Finishing	Profiling	Roughing	Finishing	Profiling
Speed	16000 RPM			6000 RPM			16000 RPM		
Feeds	200 in/min	100 in/min	25 in/min	200 in/min	100 in/min	25 in/min	500 in/min	300 in/min	200 in/min
Suggested Tools	0.5"-0.75" Round or Flat	0.25"-0.75" Round or Flat	0.25"-0.75" Flat	0.5"-0.75" Round or Flat	0.25"-0.75" Round or Flat	0.25"-0.75" Flat	0.5"-0.75" Round or Flat	0.25"-0.75" Round or Flat	0.25"-0.75" Flat
% Step Down of Tool Diameter	100%	100%	100%	100%	100%	100%	100%	100%	100%

Note: White Foam stepover for
Parallel / Horizontal / Spiral
Finishing can be set to:

10-25% for smooth (slower)
80-100% for scallops (faster)

*Gold bits are longer, but are more
fragile

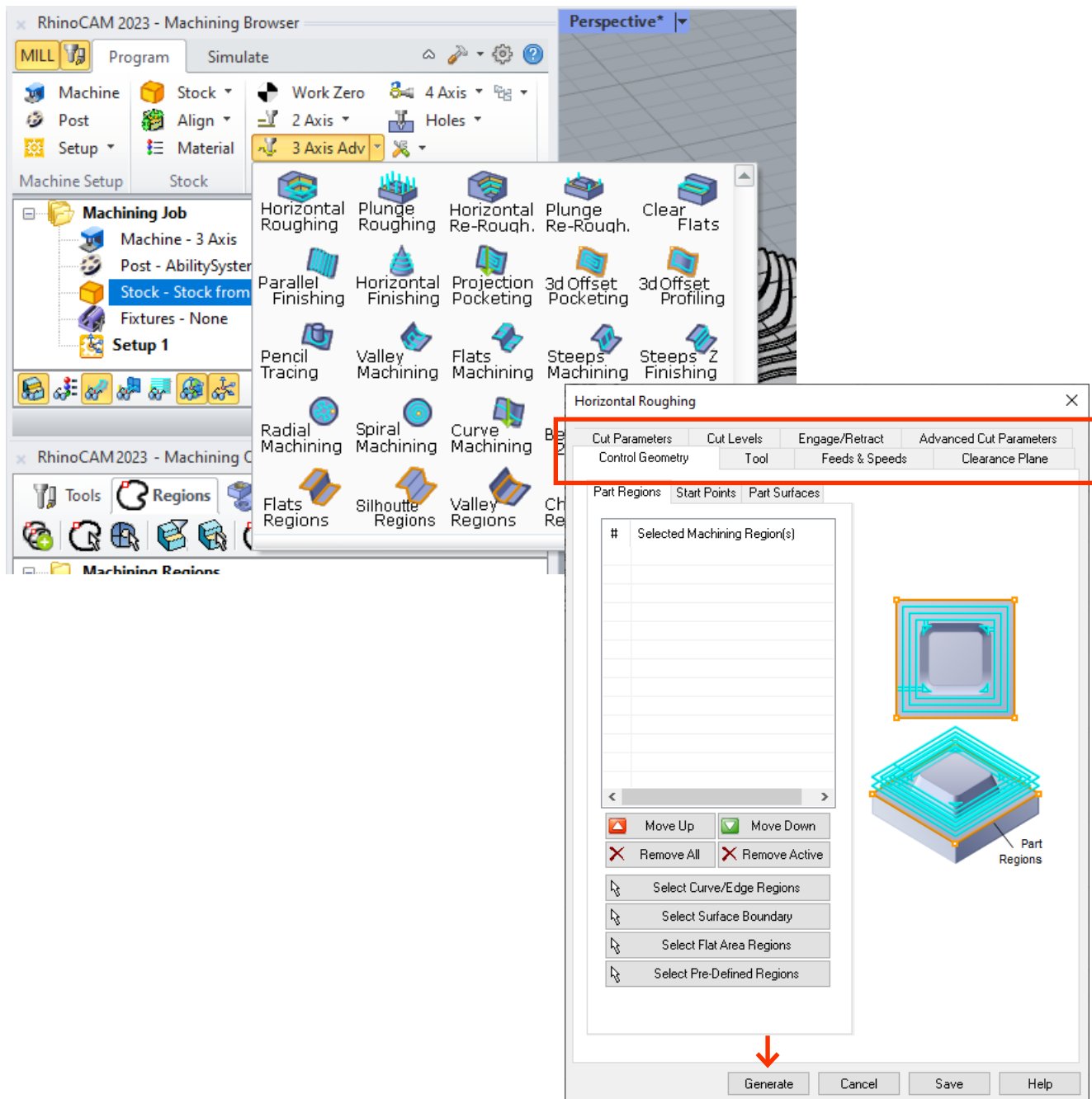
Creating Toolpaths

For most projects, you will be doing one or more **roughing passes**, followed by one or more **finishing passes**.

A **roughing pass** will use the largest allowable bit to remove as much material as fast as possible. **Finishing passes** will generally use a smaller bit to remove little material, at a slower pace to “finish” your part.

When you select a toolpath from either the 2 Axis, 3 Axis, or 4 Axis drop-down, a new window will open where you can set up the toolpath. This window has several tabs such as **Control Geometry**, **Tool**, **Feeds & Speeds**, **Clearance Plane**, **Cut Parameters**, **Cut Levels**, **Engage/Retract**, **Advanced Cut Parameters**, and more.

Once the tabs are setup according to this guide, you can click “**Generate**”, and the toolpath will get added to your project.



2D Toolpaths

video ↗

What are 2D Toolpaths	29	-
Common 2D Settings	30	-
Hole Pocketing	33	Skip to 1:00 ↗
Pocketing	36	Skip to 4:18 ↗
Profiling	39	Skip to 9:17 ↗
Engraving	42	Skip to 11:09 ↗

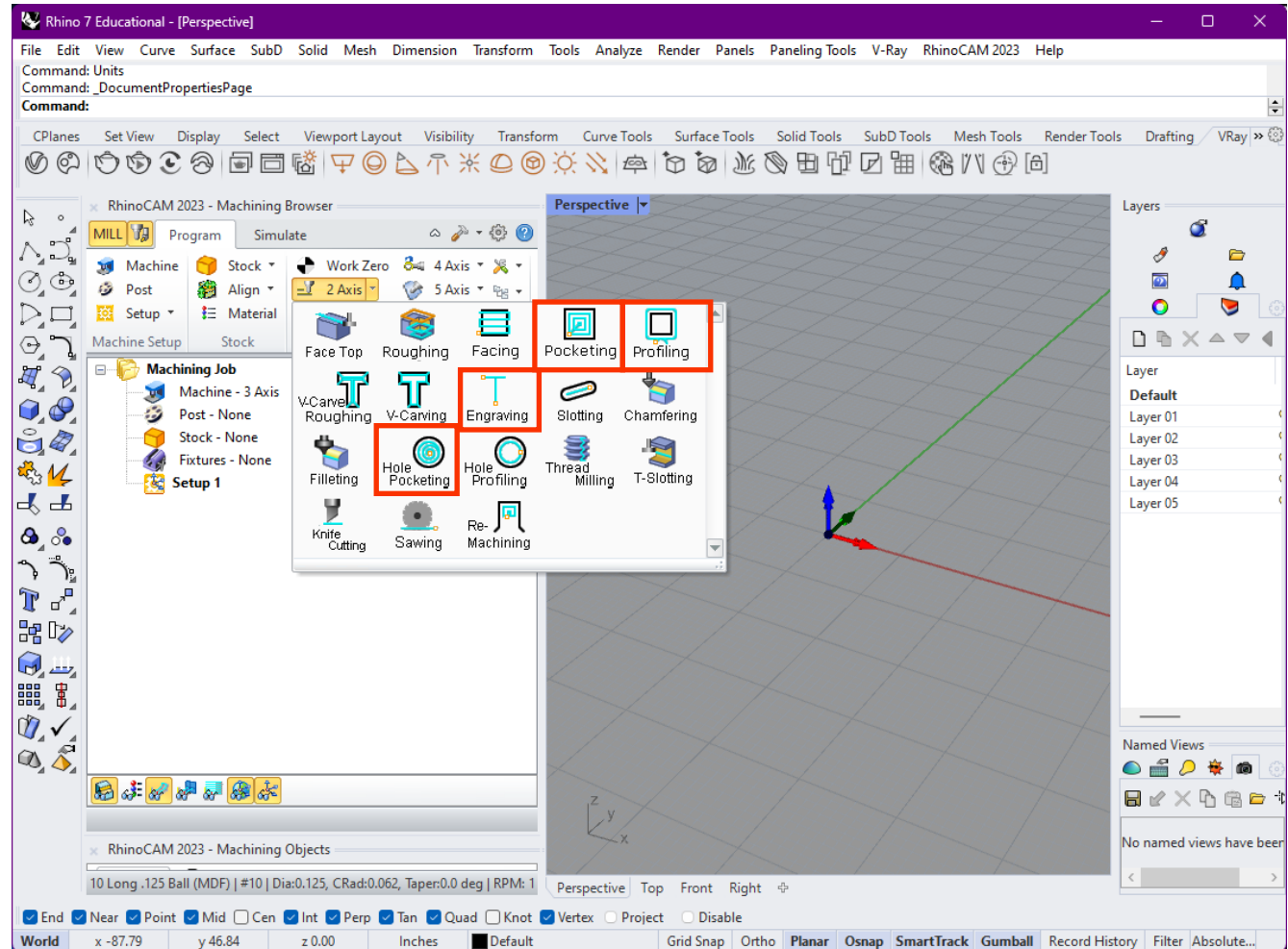
What are 2D Toolpaths

2D Toolpaths still cause the machine to move in three dimensions. However, **2D toolpaths** only cut while moving in a maximum of two dimensions, whereas **3D toolpaths** can cut while moving in the x, y, and z directions simultaneously.

There are two exceptions to this: V-Carve, and Engrave (if given non planar curves), though we will not cover them in detail in this manual.

You can select **2D toolpaths** from the **2 Axis** drop-down under the **Program** tab of the **Machine Browser** window.

In this manual, we will be covering **Hole Pocketing**, **Pocketing**, **Profiling**, and **Engraving**.



Common 2D Settings

In this section, we will cover some tabs that are common across all basic **2D Toolpaths**. Tabs not covered in this section will be covered in the individual toolpath sections.

When you select a toolpath, you will be greeted with the toolpath setup window as seen on the right.

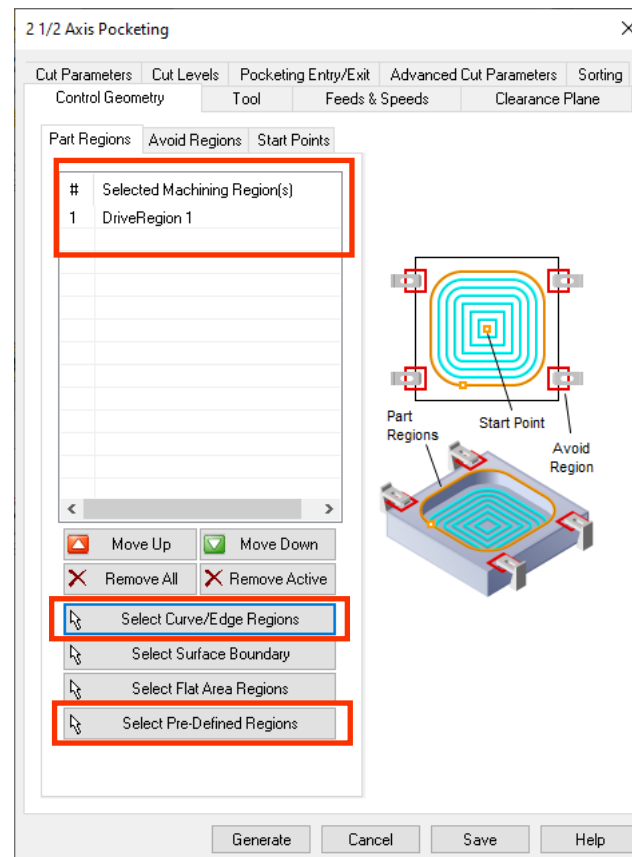
When setting up a new toolpath, always go through every tab, even if you are not changing anything in that particular tab.

Cut Parameters and **Cut Levels** are usually the settings you will spend most time tweaking. We will cover these tabs in the individual toolpath pages as the settings under these tabs change with the toolpath.

Control Geometry

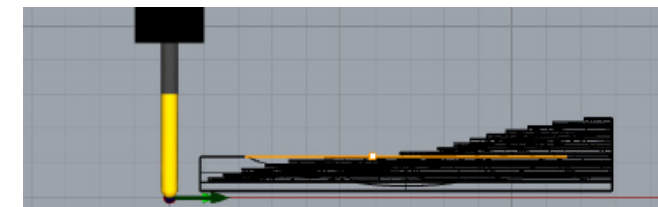
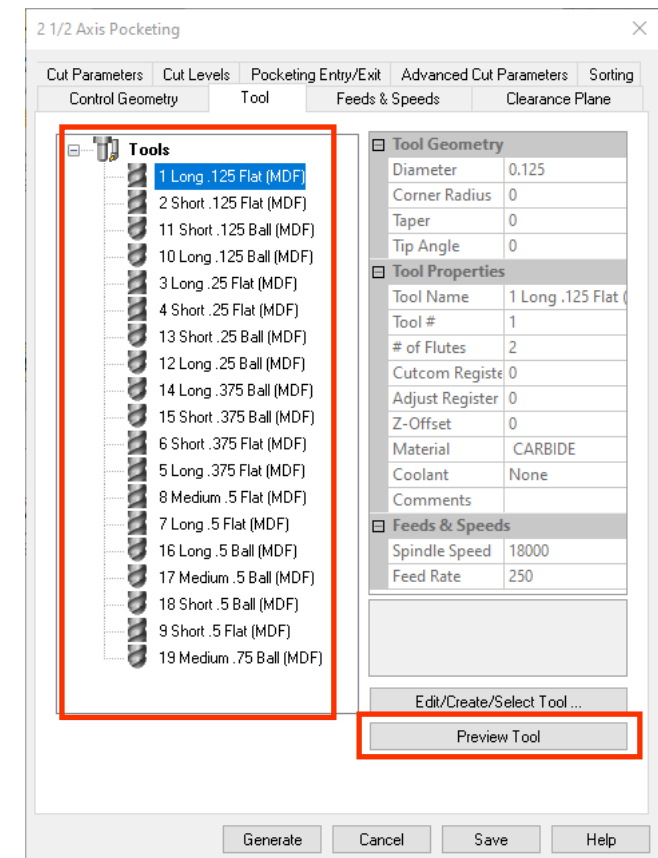
This is always the first tab you need to setup when creating a new toolpath. First, select your **control geometry**, either directly by selecting the **Curve/Edge Regions**, or using **Pre-Defined Regions**.

Any control geometries you select will appear in the **Part Regions** list.



Tool

In this tab, you can pick the tool for this operation. You can preview the selected tool by clicking “Preview Tool”.

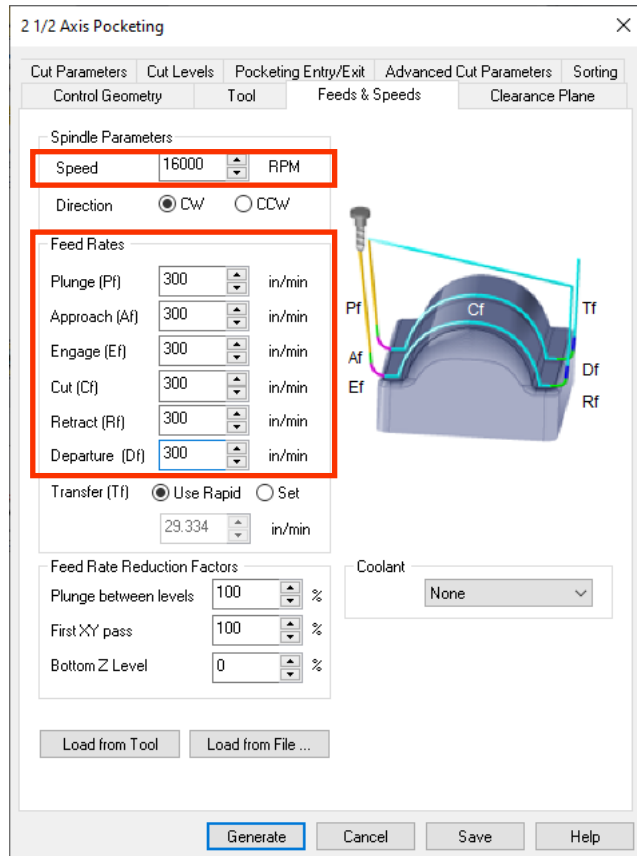


Common 2D Settings ctd.

Feeds & Speeds

Set your feeds and speeds according to the material, type of operation, and tool you selected.

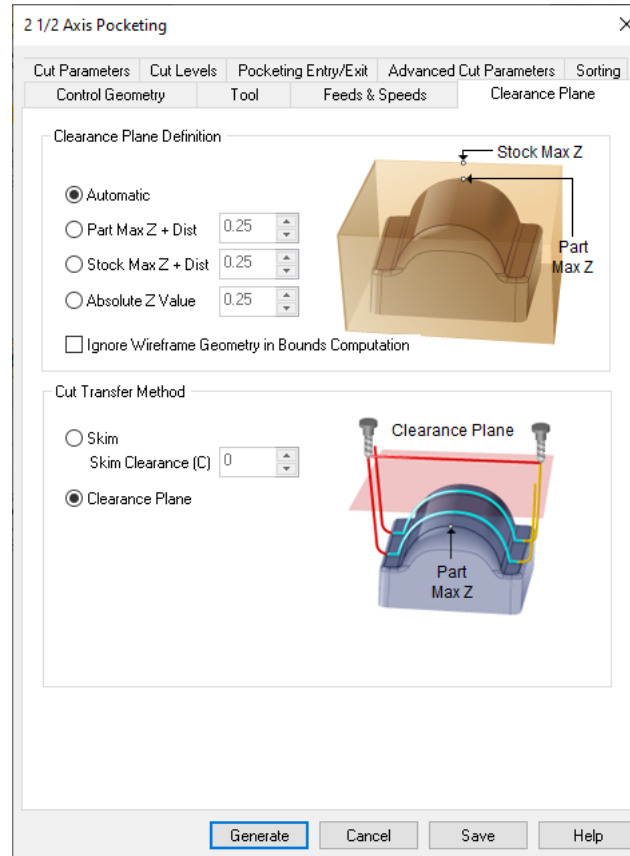
Refer to the **Feeds & Speeds** table.



Clearance Plane

Leave the **Clearance Plane** tab as is.

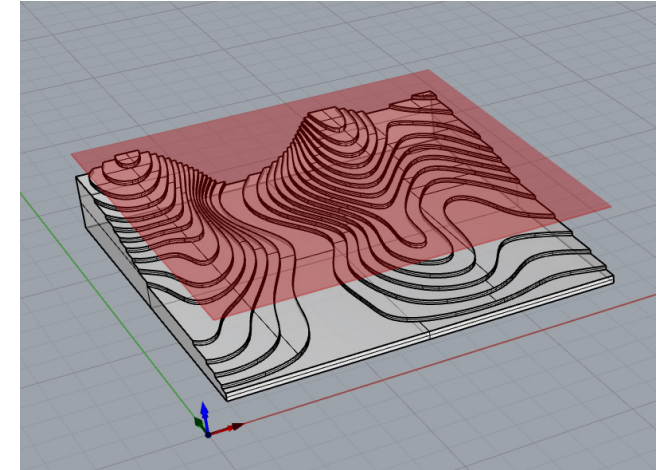
It defines a safe plane above the stock on which the tool expects to travel without obstruction. It will be automatically calculated using the topmost geometry in your file.



Clearance Plane ctd.

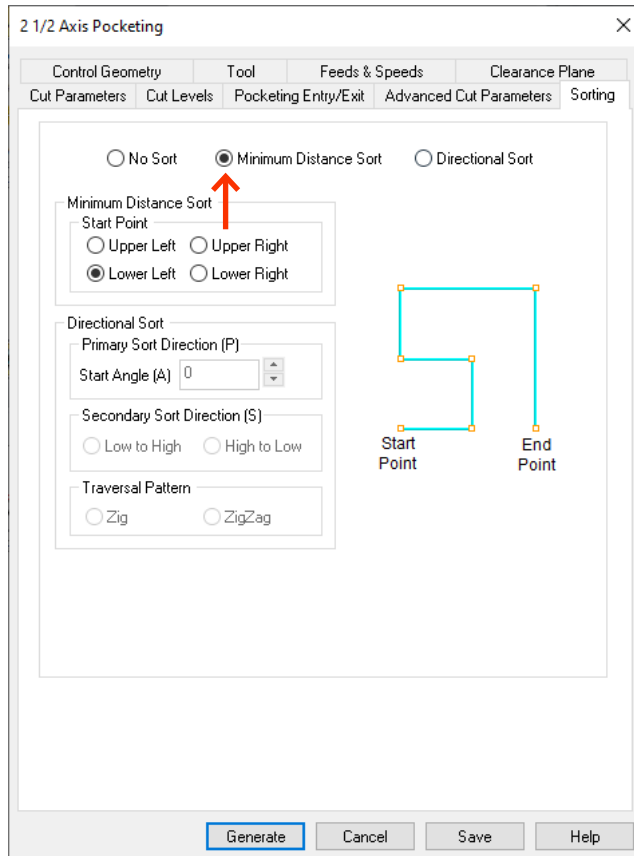
When in the **Clearance Plane** tab, you will be able to see a preview of the clearance plane in red.

If you do not see this red preview, ensure there are no rogue geometries in your file above your model.



Sorting

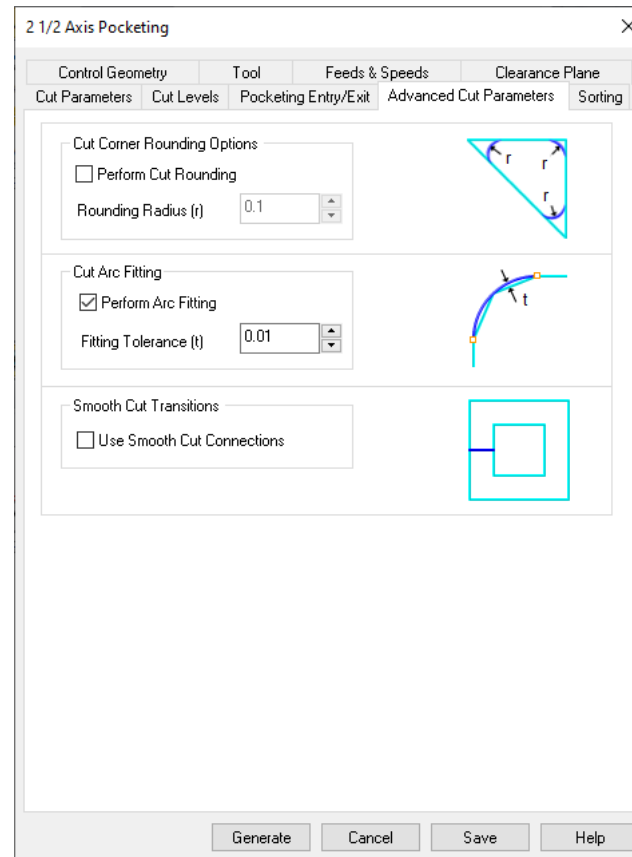
Always set **Sorting** to “**Minimum Distance Sort**” when you have multiple control geometries. This will speed up toolpaths that need to move between many entry points.



Common 2D Settings ctd.

Advanced Cut Parameters

You can leave this tab as is.

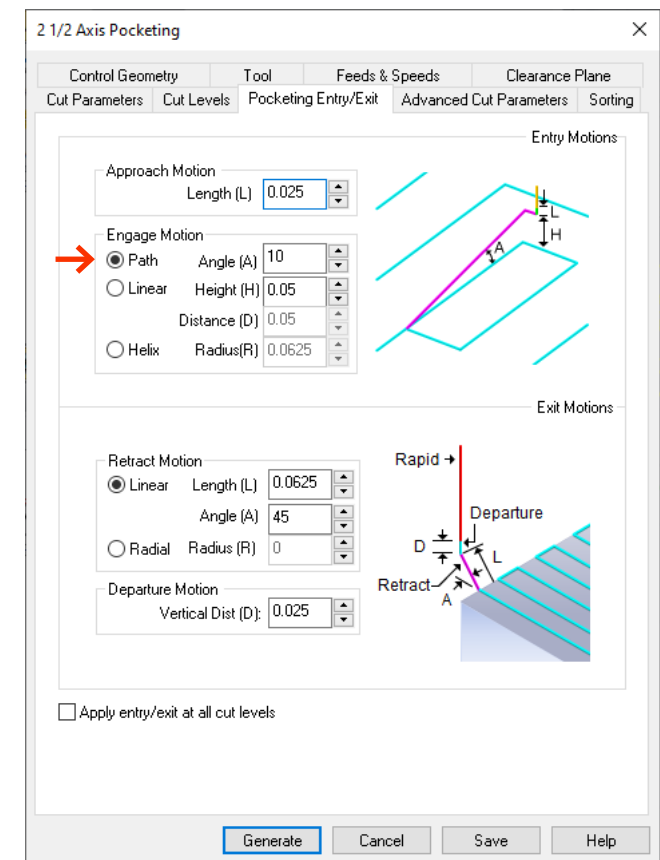


Entry/Exit

This tab defines how the tool enters the stock, and leaves it.

Always set **entry motions** to “**Path**”. Always set **exit motions** to “**None**” if possible, otherwise use the default.

You can adjust the **Length**, **Angle**, and **Height** of the **engage/retract** motions as needed.



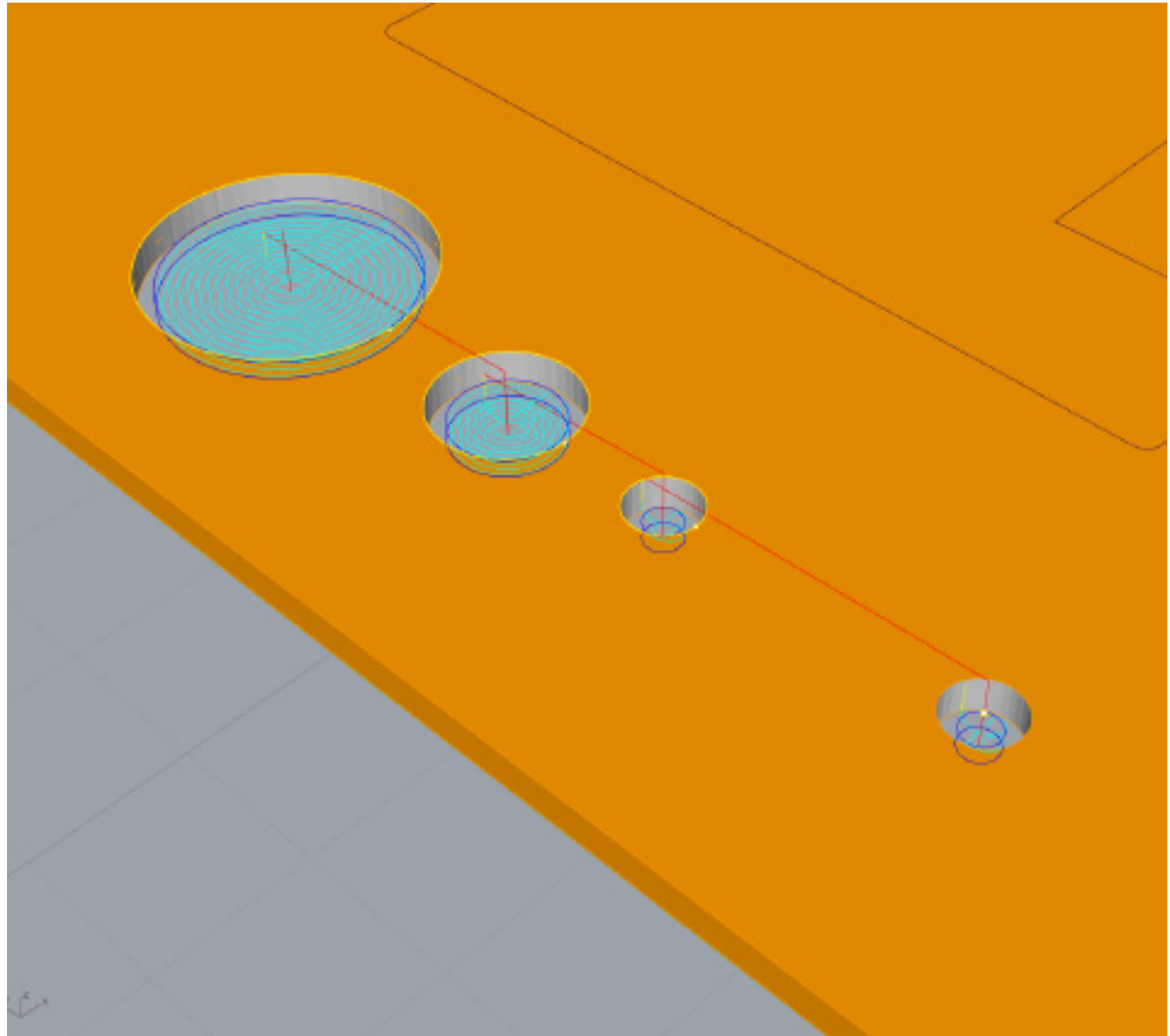
Hole Pocketing

This operation will mill circular holes in your stock.

You can select closed circular curves, or points as your control geometry. Curves that are not circular, or not closed cannot be used with **Hole Pocketing**.

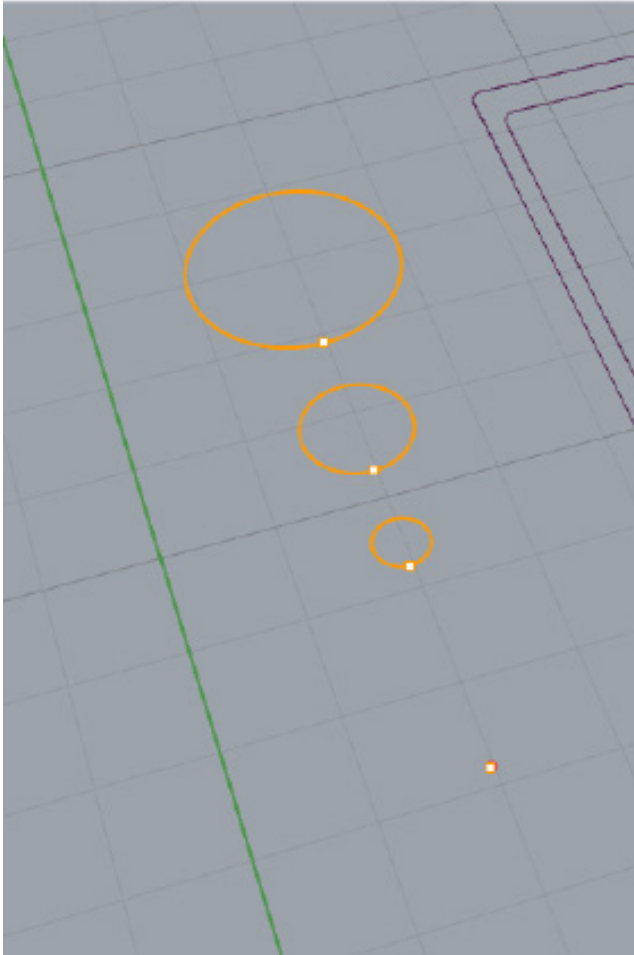
Circles with diameters less than or equal to the tool diameter will also not be milled via this operation.

To set up all other tabs not shown in this section, refer to the **Common 2D Settings** section.



Hole pocketing ctd.

Select your geometry. You can select closed circular curves, or even points. You will assign diameters to these points later.



Cut Parameters

Ensure your control geometry is on the top surface of your stock. When **Location of Cut Geometry** is set to **At Top**, RhinoCAM will start cutting from where the control geometry is, and finish cutting below it with the depth you set.

Set your **Hole Depth**. If you want to mill all the way through your stock, **Hole Depth** should equal your stock thickness.

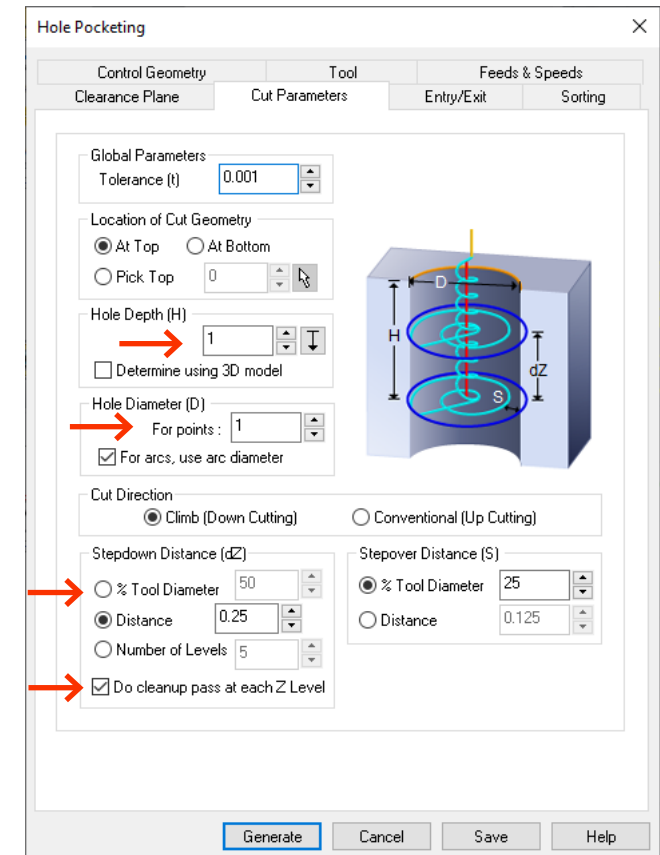
You can set **Hole Diameter** for any points you have selected as your control geometry. The toolpath will replace any points you have selected with holes of the diameter set here.

Stepdown controls how deep the tool goes into the stock at each cut level. Going too deep in one go may cause too much resistance on the tool, and cause it to break. Set the **Stepdown** according to the **Feeds & Speeds** table.

Select **“Do cleanup pass at each Z level”**.

Ensure you have setup all other tabs according to the **Common 2D Settings** section.

Click **“Generate”**.



Drilling small holes

Cut Parameters

The **Hole pocketing** operation cannot pocket holes smaller than or equal to the tool diameter.

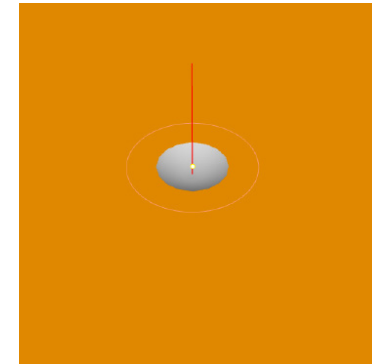
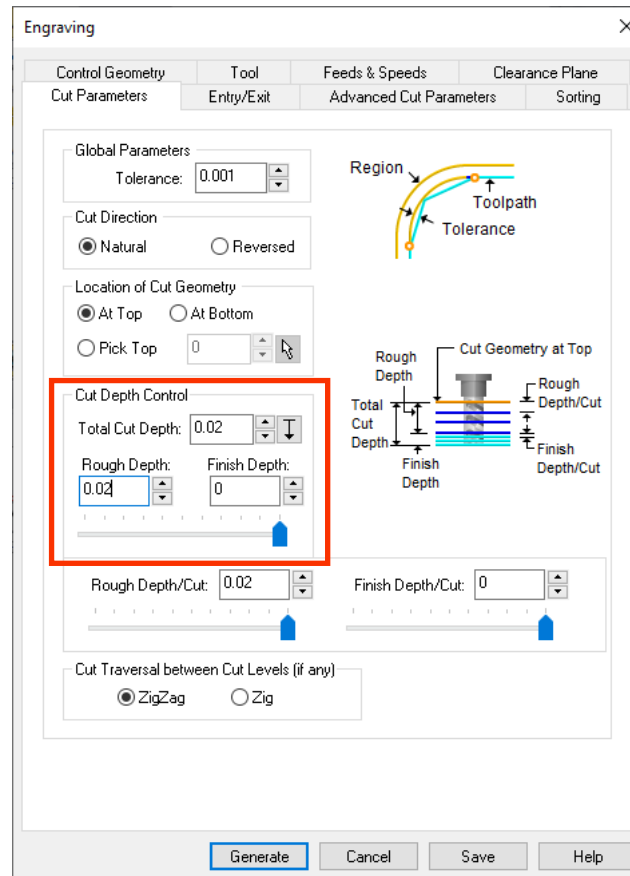
A work-around is to use the **Engrave** operation to **centermark** the hole for later drilling using a hand-held drill.

Select a point centered in the hole, then use a cut depth of **0.02"**.

You may want to use a **ball-bit** for this operation.

These **centermarks** will make it easier to know where to drill, and will prevent the drill bit from "walking".

Drilling holes can cause fires if done incorrectly. Talk to a TA about drilling holes if required.



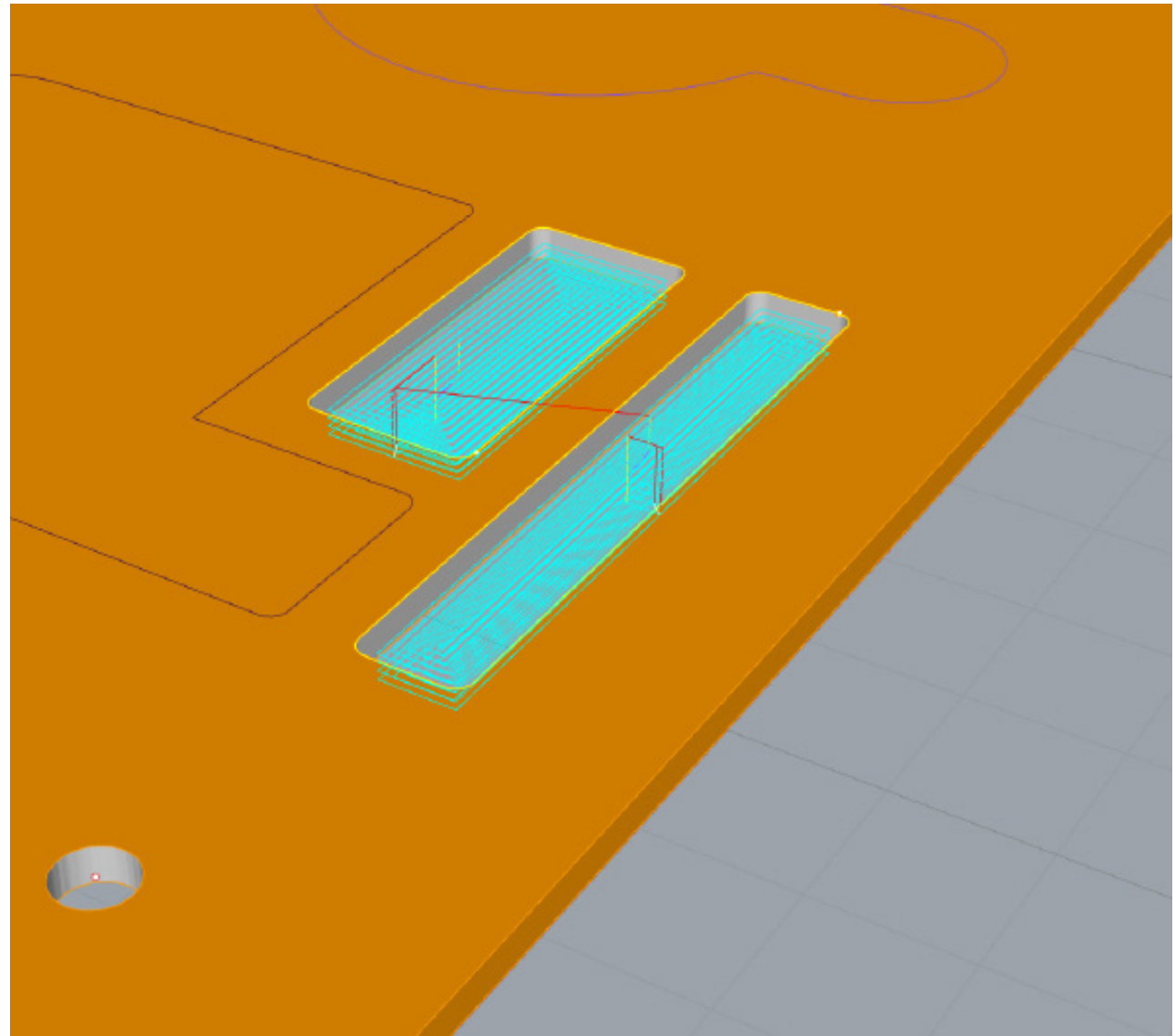
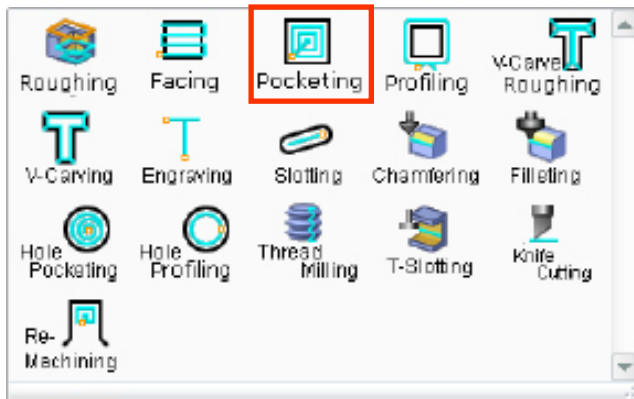
Pocketing

This operation will mill a pocket given an outline geometry.

It is suggested to run a **Profiling** operation after **Pocketing** to leave a cleaner surface finish.

You can select closed curves, or surface boundaries as control geometry.

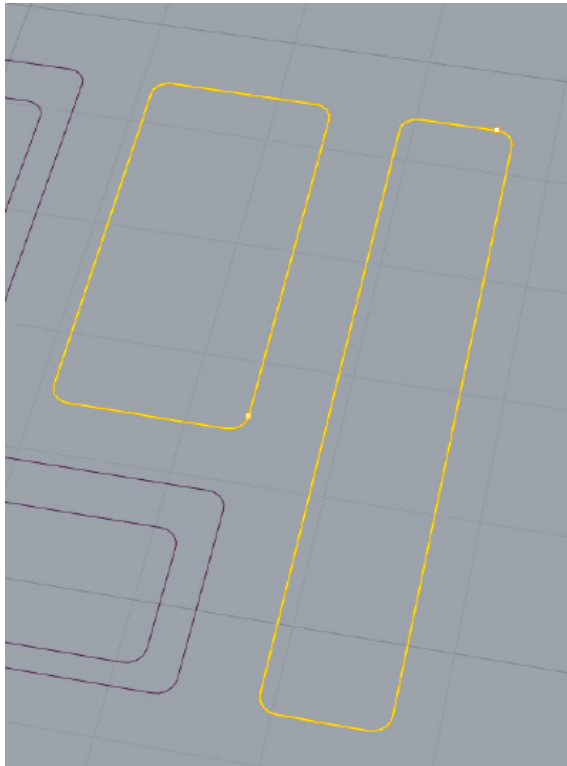
To set up all other tabs not shown in this section, refer to the **Common 2D Settings** section.



Pocketing ctd.

Select control geometry

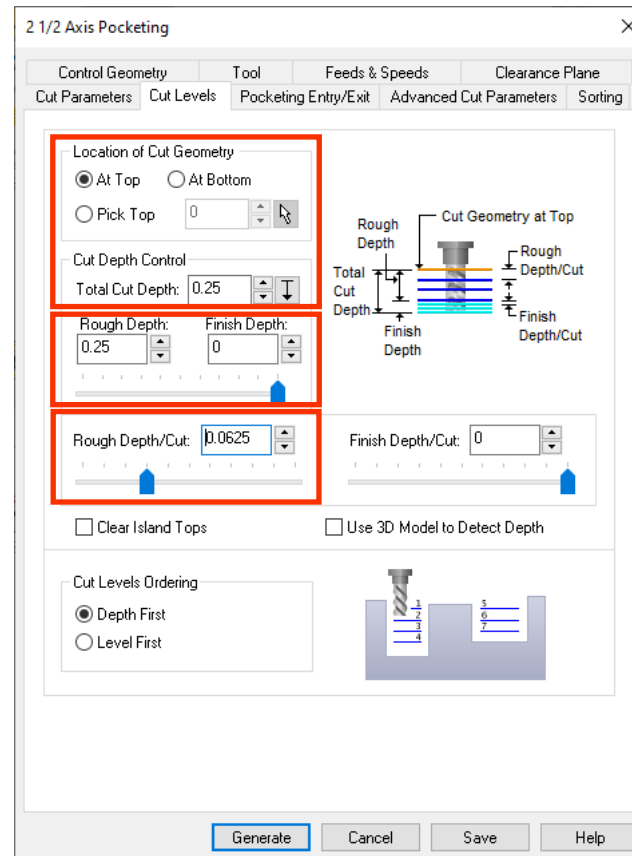
Select your control geometry. You may wish to add fillets to your geometry according to the tool you will be using. If you require sharp corners, you will need to file them manually later.



Cut Levels

Again, ensure your control geometry is on the top surface of your stock, set **Location of Cut Geometry** to “At Top”.

Set your pocketing depth, and set the **Rough Depth/Cut** (stepdown) according to the **Feeds & Speeds** table for the material, and tool you are working with.

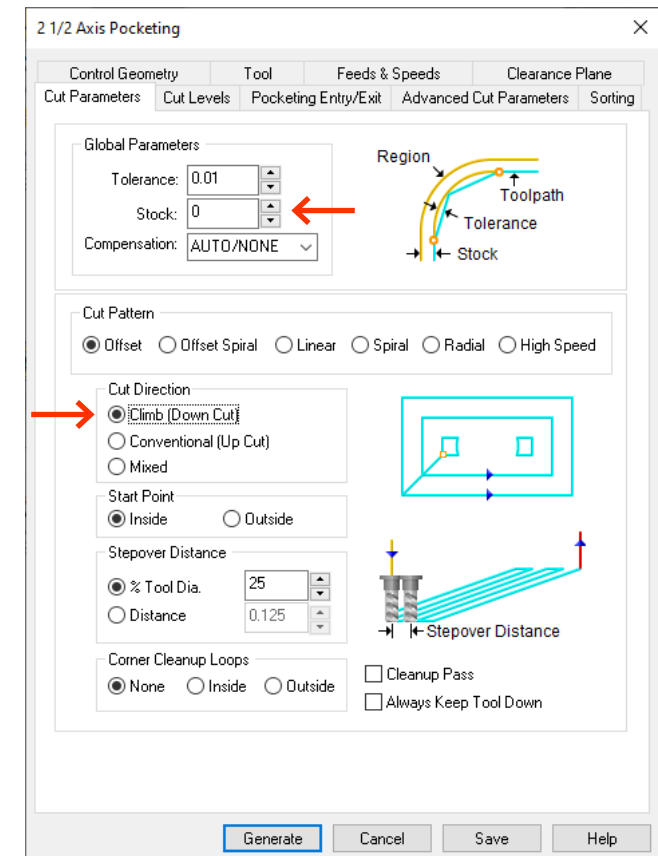


Cut Parameters

You can set **Stock** to 0 if this will be the only operation you wish to use. Otherwise, set this to 0.025” to leave some material behind for the next operation using this **control geometry**.

To learn more about **Cut Patterns**, see the next page.

Cut Direction can be set to “Mixed” for foam and softer materials, otherwise select “Climb” for a cleaner wall finish.

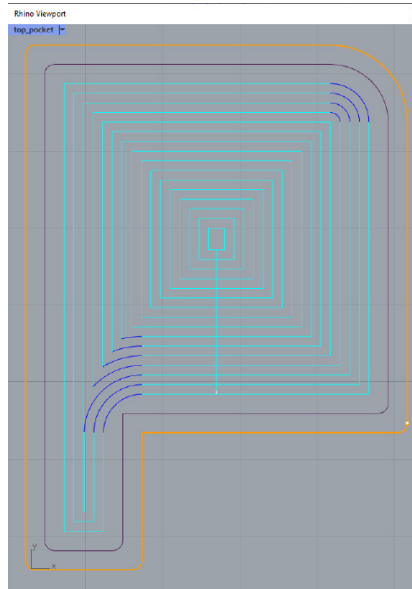


Cut Pattern

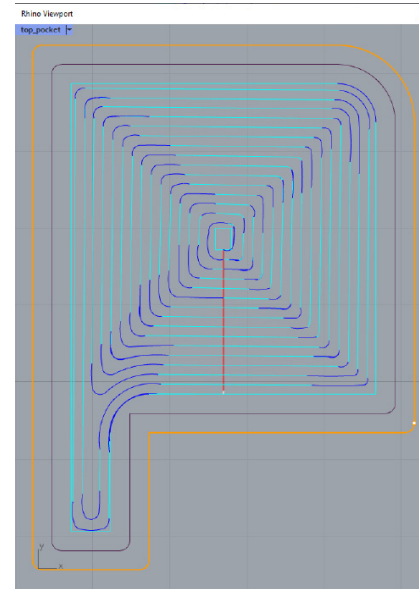
Some toolpaths such as the **Pocketing** toolpath allows you to choose the **cut pattern**. This pattern may appear on the bottom surface of your cut, depending on the material.

Other than the “**High Speed**” pattern which has an advantage in optimizing tool load, and prolonging tool life when milling hard materials, the other cut patterns can be chosen freely depending on the type of surface finish you are looking for.

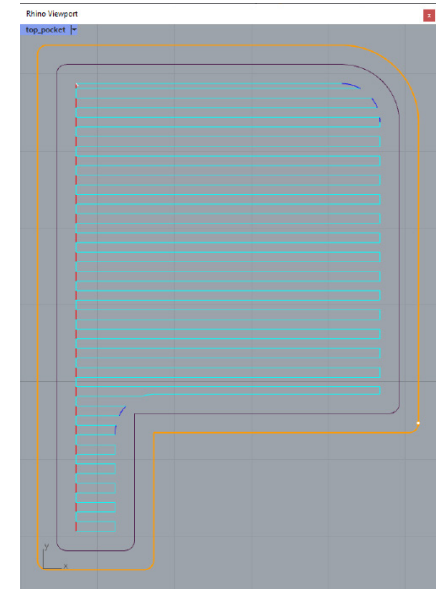
Note that some patterns take longer to cut.
“**Offset**” is almost always the fastest cut pattern.



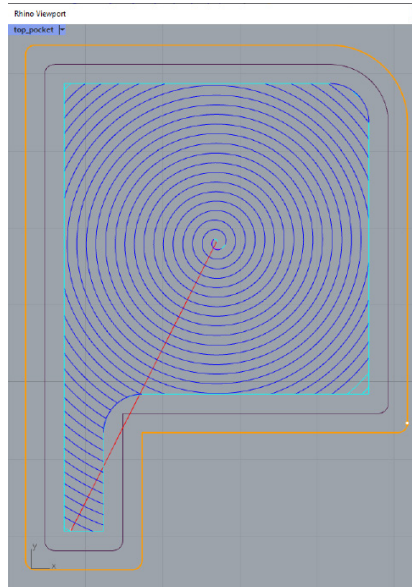
Offset: 1.28 min



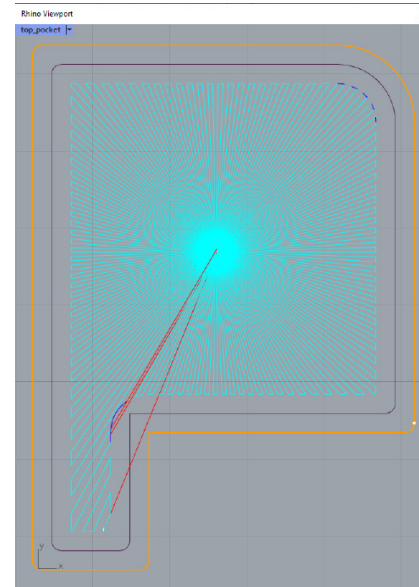
Offset Spiral: 1.31 min



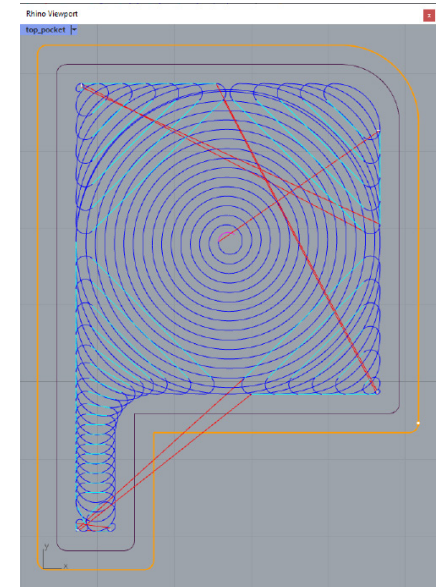
Linear: 1.41 min



Spiral: 2.32 min



Radial: 4.19 min



High Speed: 2.79 min

Profiling

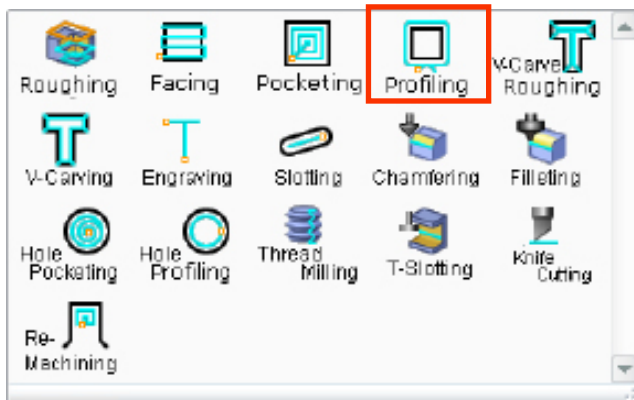
This operation will follow the inside or outside of a profile, usually to cut out a shape. A **flat tool** is recommended for this toolpath.

You have the option to leave **tabs/bridges** with this operation so that your part does not come loose after being cut out. A loose part will almost always collide with the tool, causing damage to the tool and to the part.

Profile can be used instead of **pocketing** to clear out a large hole in your stock.

This toolpath can also be used for finishing the vertical surfaces, usually after a **pocketing** operation.

To set up all other tabs not shown in this section, refer to the **Common 2D Settings** section.

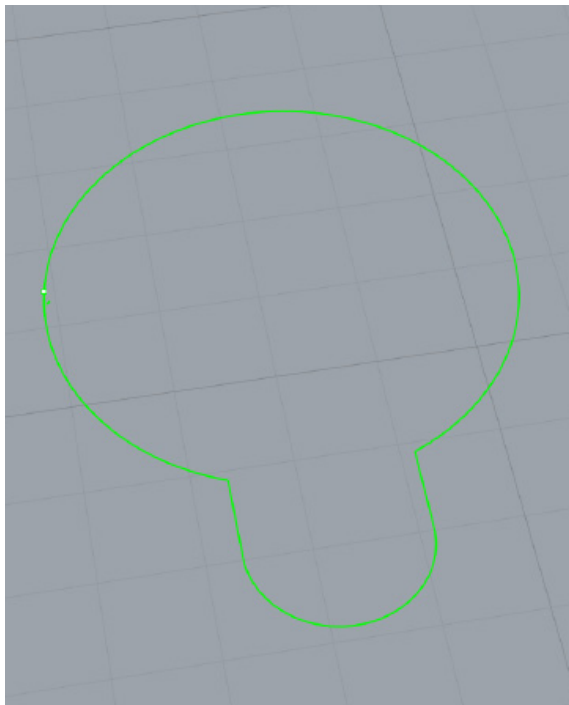


Profiling ctd.

Select control geometry

Select your control geometry either by selecting the curves directly, or by using **Pre-Defined Regions**.

If you select curves using **Pre-Defined Regions**, you will have more control over the placement of **Bridges/Tabs**.



Cut Parameters

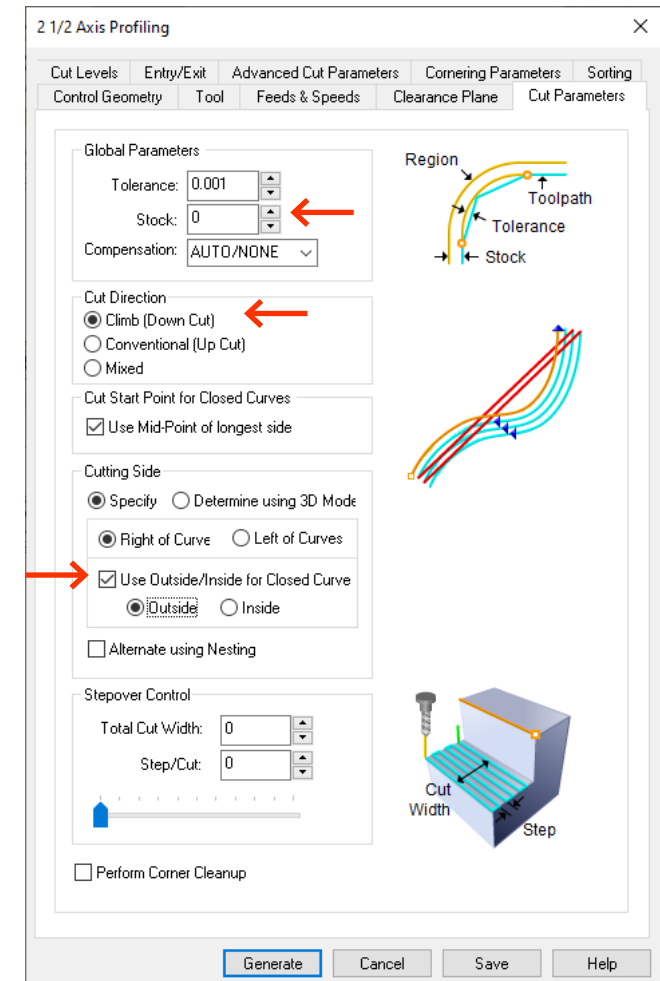
Set **Stock** to leave to **0**.

Uncheck “**Use Mid-Point of longest side**” if using Regions and to start cutting at your defined start point.

Set **Cut Direction** to “**Climb**” for better wall finish.

Set the cutting side to “**outside**” to cut out the shape to keep, or set to “**inside**” to remove material from inside the profile shape.

Adjust the other settings as shown.



Profiling ctd.

Advanced Cut Parameters

In this tab, you can set the **Bridges/Tabs**.

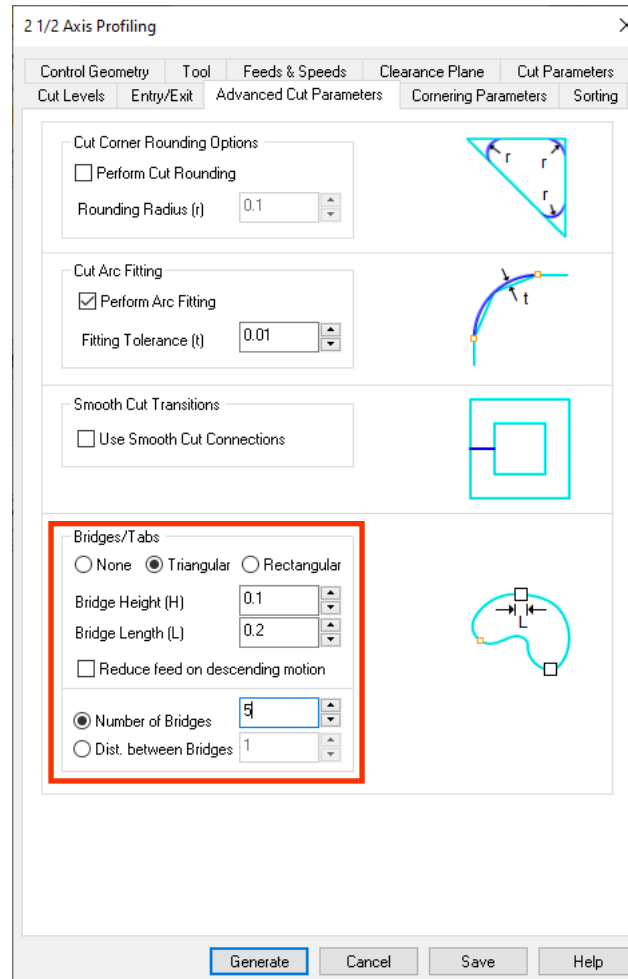
Unless specified using **Regions**, this option will automatically place bridges around your profile geometry according to the number of bridges or distance between bridges set here.

Always add bridges if your **Profiling** operation will cut out a part. Otherwise, the part may become loose and collide with the endmill, causing damage to your part and the machine.

Triangular bridges have a smaller area of contact with the piece, so they may be weaker but are easier to remove and leave a better wall finish when cleaned.

When using **Profiling** as a finishing operation, you do not need to set **Bridges/Tabs** as there should not be any material left to dislodge.

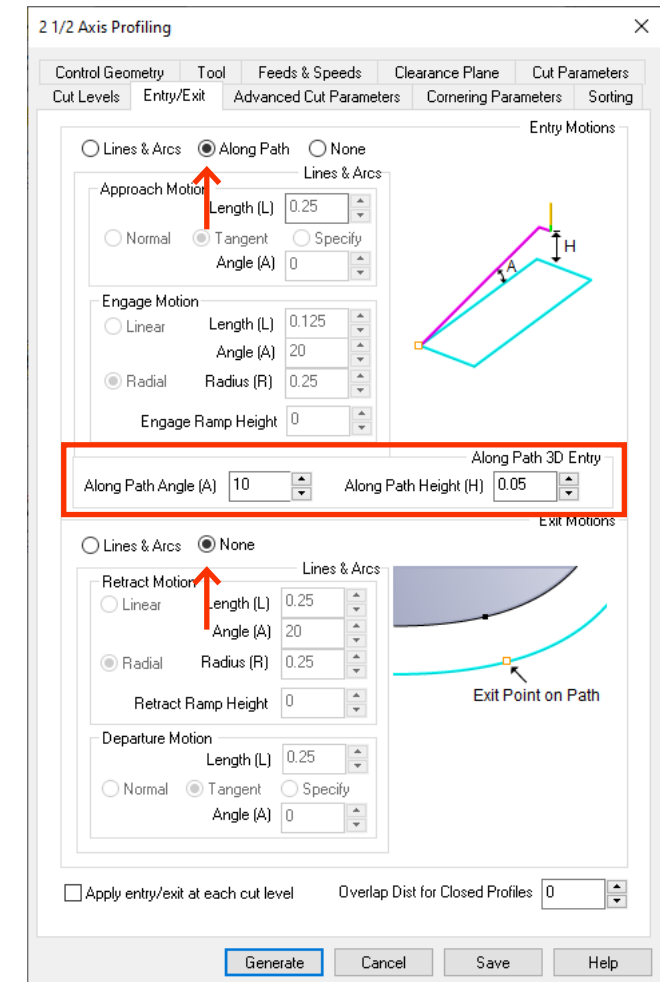
Tip: avoid placing bridges in corners as it will be more difficult to clean up.



Entry/Exit

Set Entry Motions to **Along Path**, and Exit Motions to **None**.

You may need to adjust the Angle and Height if the entry motion is too small or too large.



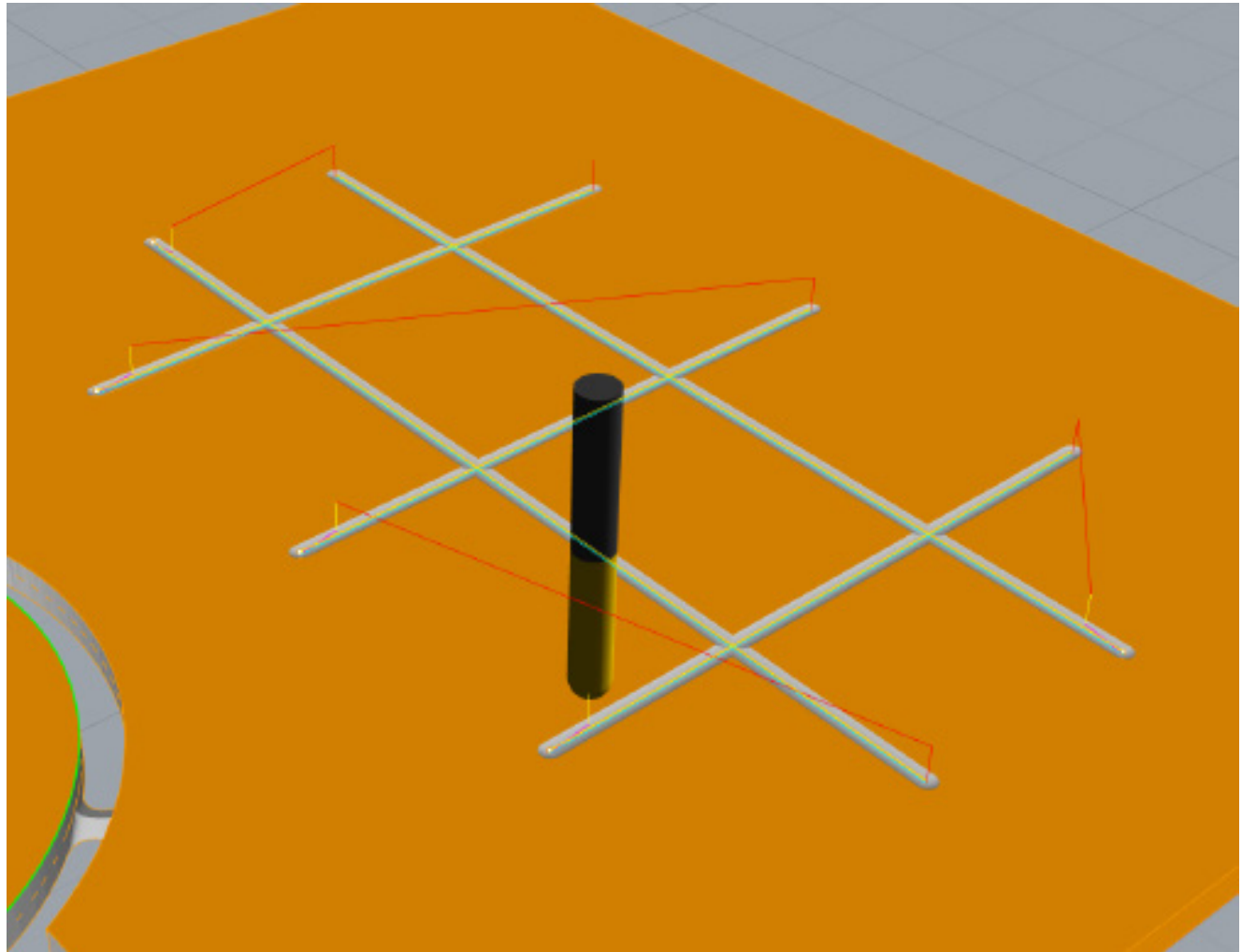
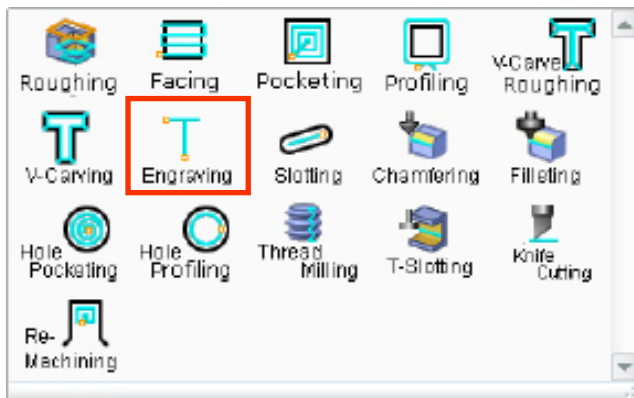
Engraving

Unlike the **Profile** operation which follows inside or outside the control curve, **Engrave** will follow the curve down the middle.

This is useful for engraving road lines, grids, or simple text. You can use **either a flat and a round** endmill for this operation, depending on the engraving effect you desire.

For very shallow engraves, you can set the **feedrate** to **roughing feedrates** for the material you are using. For engraves deeper than **0.0625"**, use the **Profiling feedrate** for the material you are using.

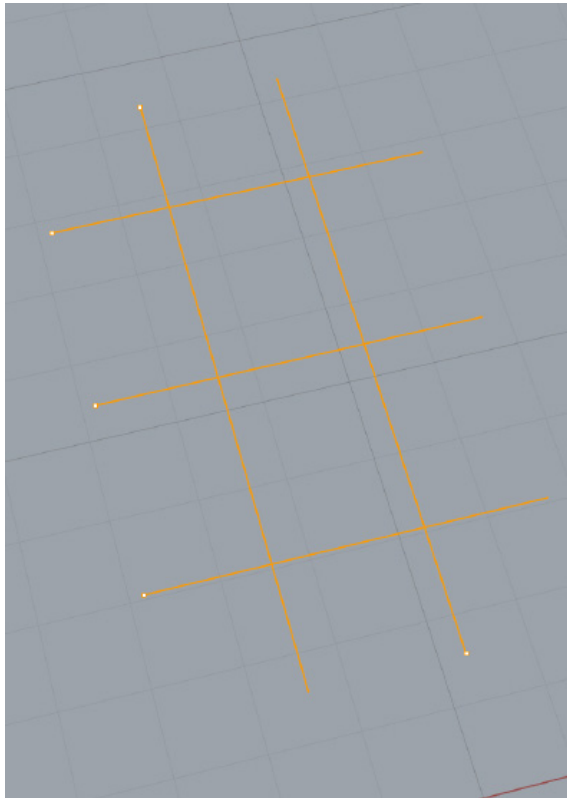
To set up all other tabs not shown in this section, refer to the **Common 2D Settings** section.



Engraving ctd.

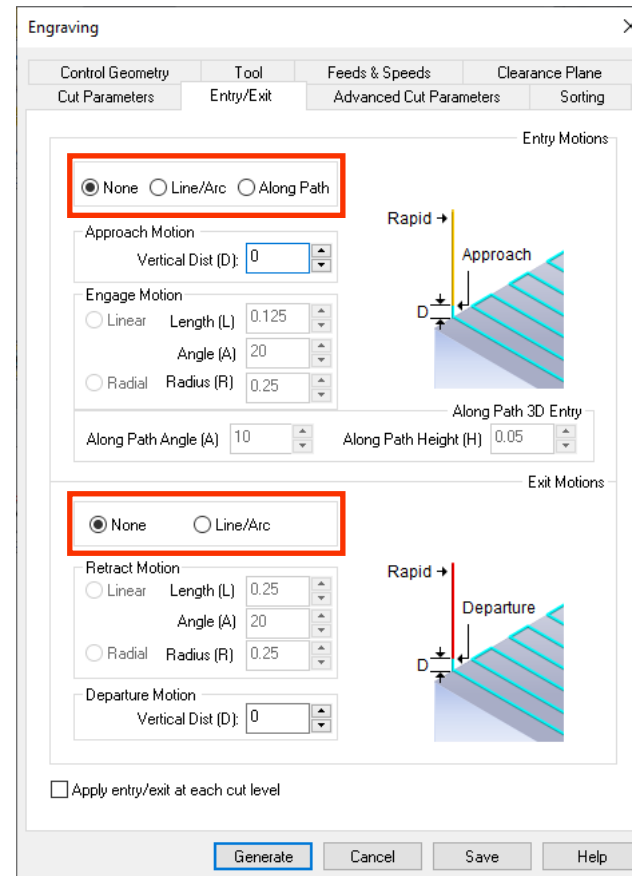
Select control geometry

Select your control geometry. You can use open/closed curves, as well as points.



Entry/Exit

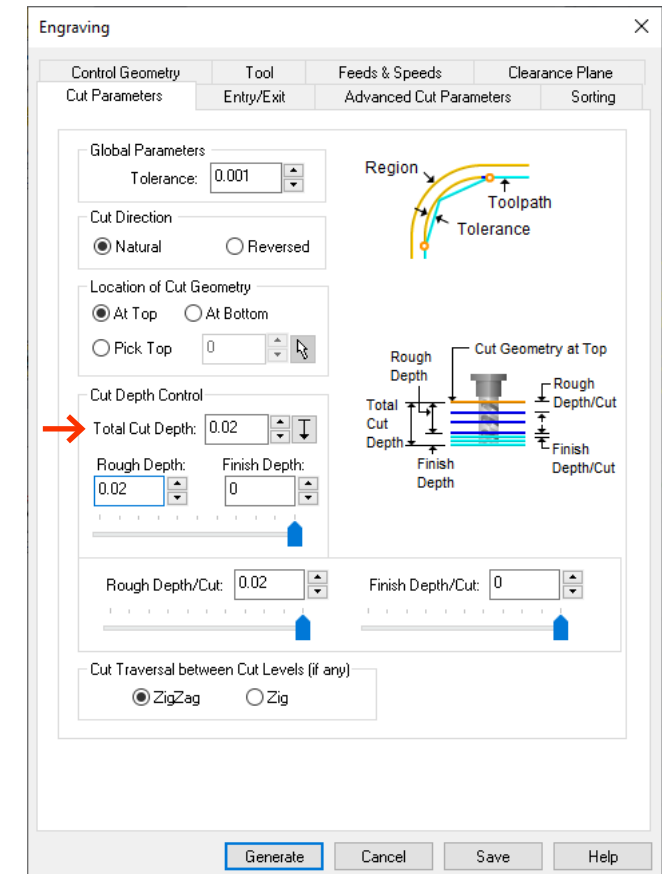
Set **Entry/Exit** to “None” if removing very little material. Otherwise, set **Entry Motions** to “Along Path”, and **Exit Motions** to “None”.



Cut Parameters

For **Cut Depth**, set it to something very small like **0.02** to engrave, as shown.

To engrave deeper, ensure you are using the appropriate **Rough Depth / stepdown**.



3D Toolpaths

video ↗

What are 3D Toolpaths	45	-
Common 3D settings	46	-
Horizontal Roughing	48	Skip to 0:59 ↗
Clear Flats	50	Skip to 2:39 ↗
Horizontal Finishing	52	Skip to 3:35 ↗
Parallel Finishing	54	Skip to 5:05 ↗
Radial Machining/Spiral Machining	56	Skip to 7:15 ↗

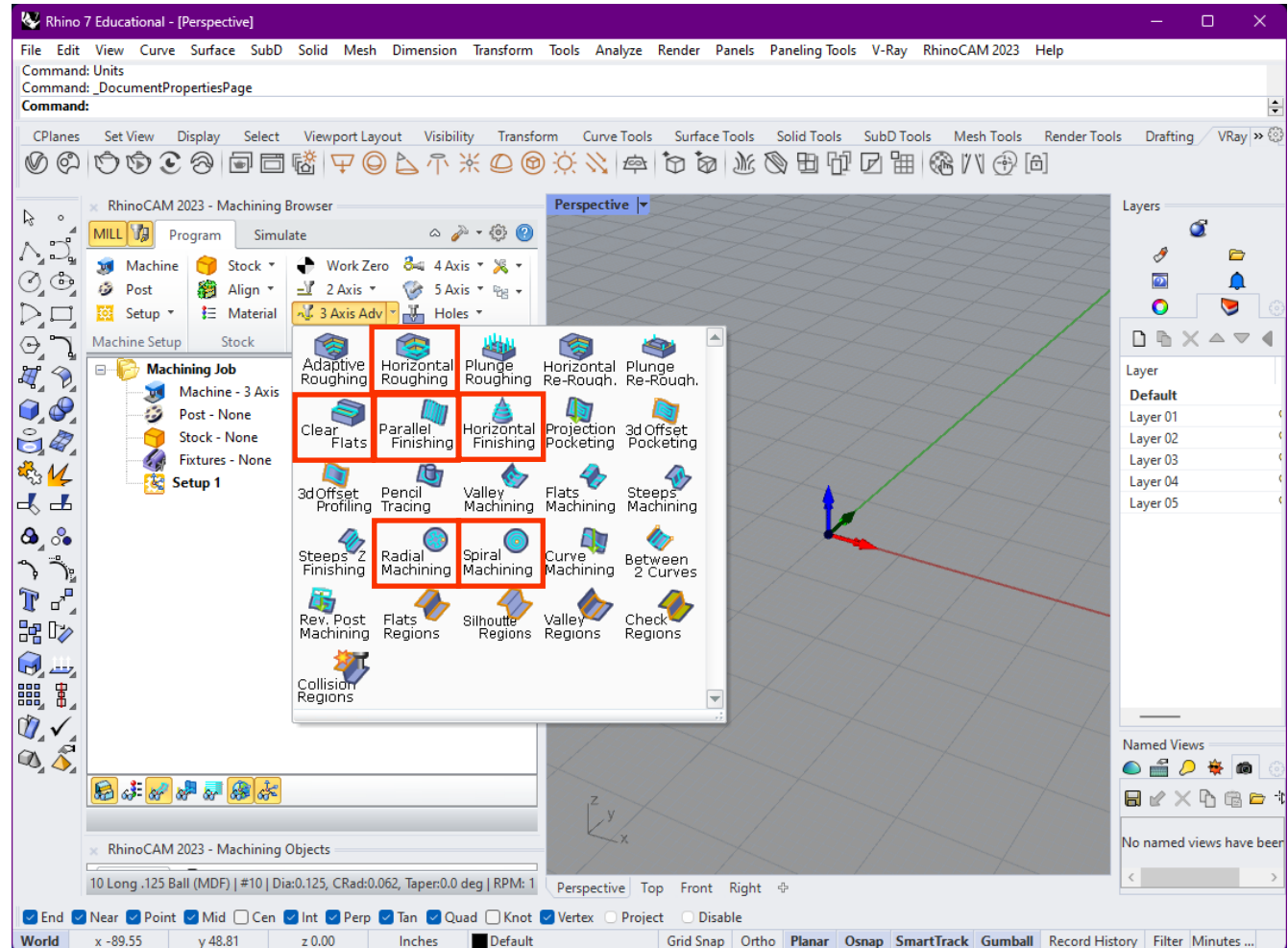
What are 3D Toolpaths

3D toolpaths allow you to take full advantage of the 3 axis motion system of the CNC router. These toolpaths allow the machine to cut while moving all three axes simultaneously.

You can select **3D toolpaths** from the **3 Axis Adv** dropdown under the **Program** tab of the **Machine Browser** window. Most of these toolpaths only require a general boundary to stay within as their control geometry; they calculate the cut depth and cut area automatically using the 3D model.

Common toolpath combinations include:
Horizontal Roughing + Clear Flats,
Horizontal Roughing + Parallel Finishing.

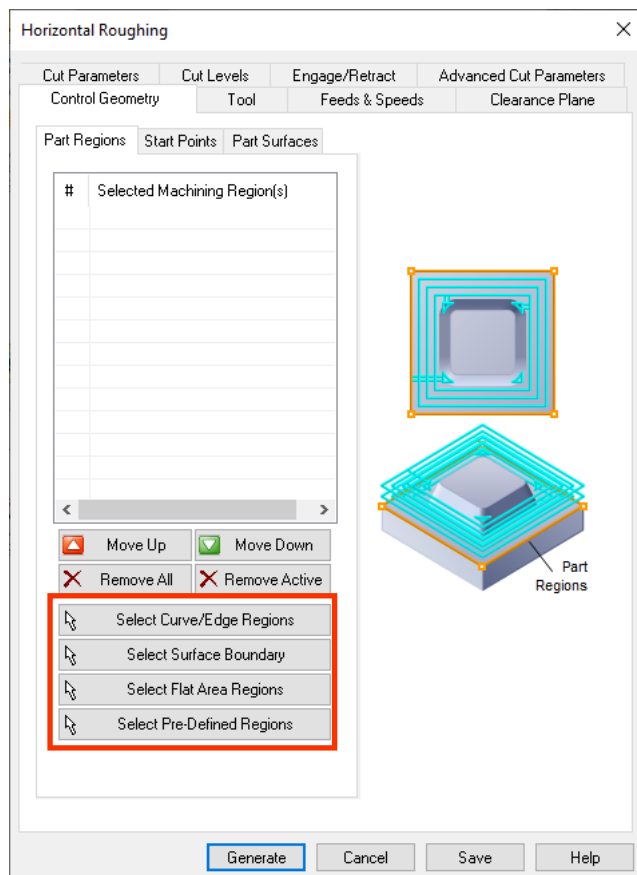
In this manual, we will be covering **Horizontal Roughing, Clear Flats, Parallel Finishing, Horizontal Finishing, Radial Machining, and Spiral Machining.**



Common 3D settings

Control Geometry

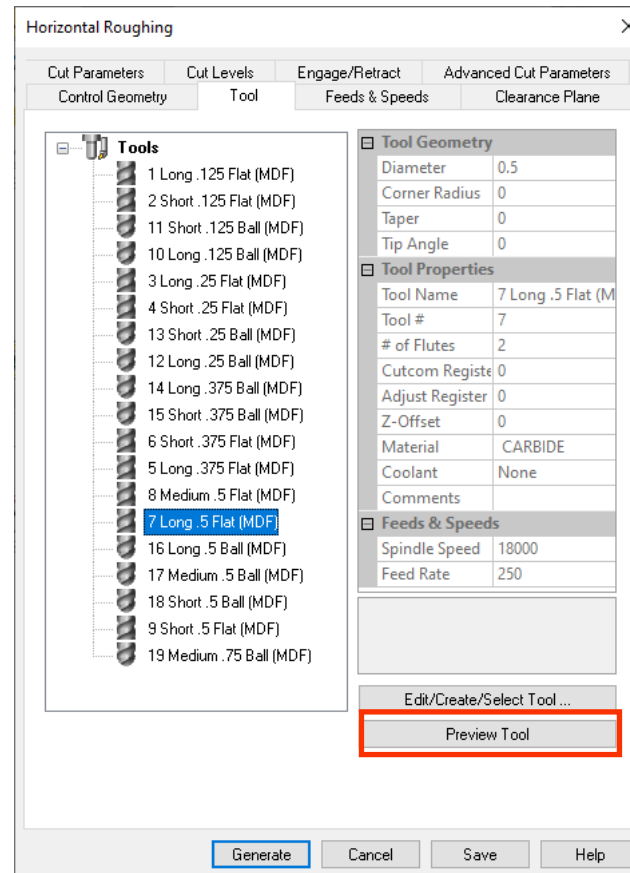
First, select your control geometry. The toolpath is smart enough to recognize what it needs to remove, as long as it is given a boundary region to stay inside.



Tools

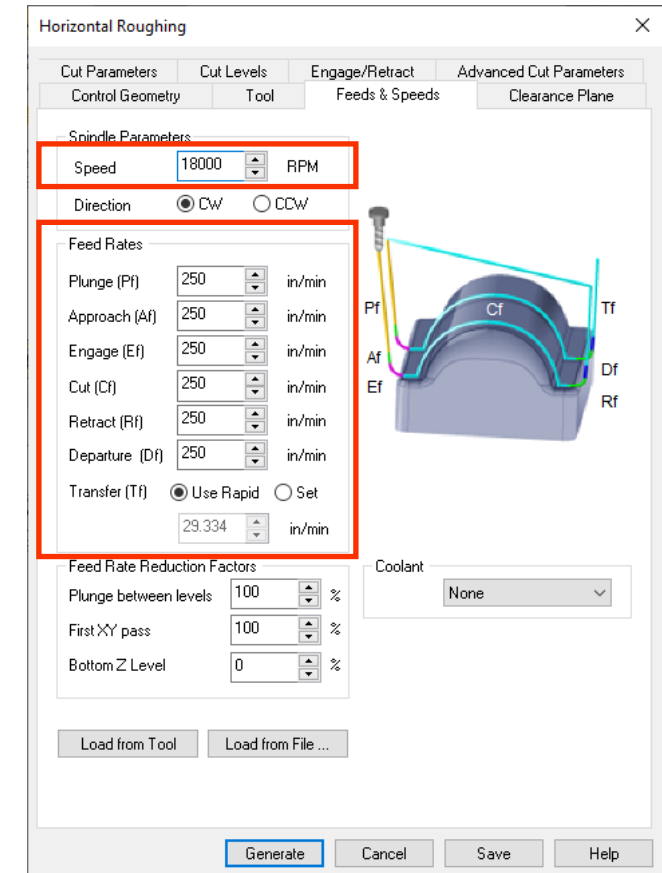
Flat bits are mainly used for **Roughing, Pocketing**, and for the **Clear Flats** toolpaths. **Ball-bits** are usually used for **Parallel Finishing** operations.

You can preview your tool using the “**Preview Tool**” button below.



Feeds & Speeds

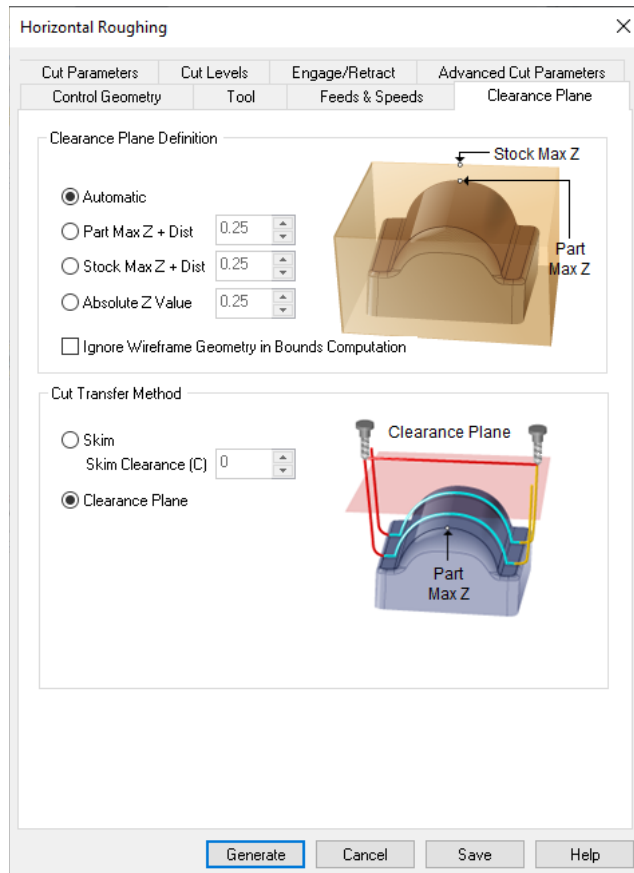
As with 2D operations, set your **Feeds & Speeds** according to the chart.



Common 3D settings ctd.

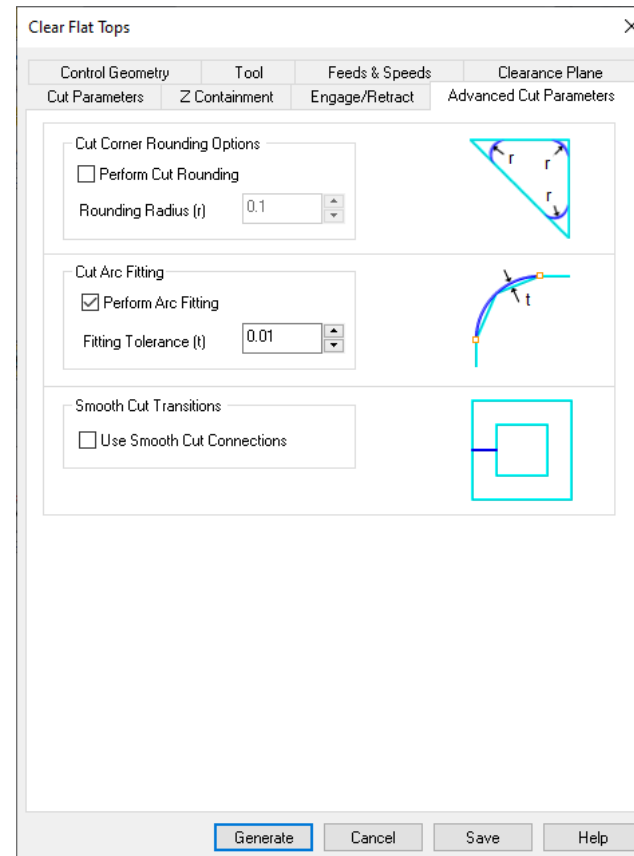
Clearance Plane

Leave this tab as is.



Advanced Cut Parameters

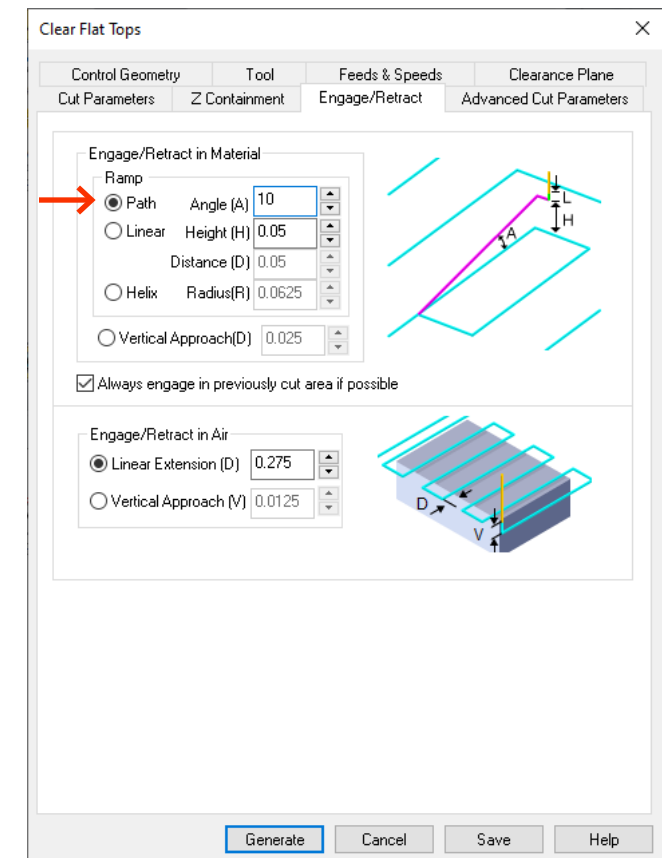
Leave this tab as is.



Engage/Retract

This tab is similar to **Entry/Exit**. It defines how the tool engages and retracts from the stock as it is cutting.

Always set **entry motions** to **Along Path**. Always set **exit motions** to **Linear** if possible, otherwise use the default.

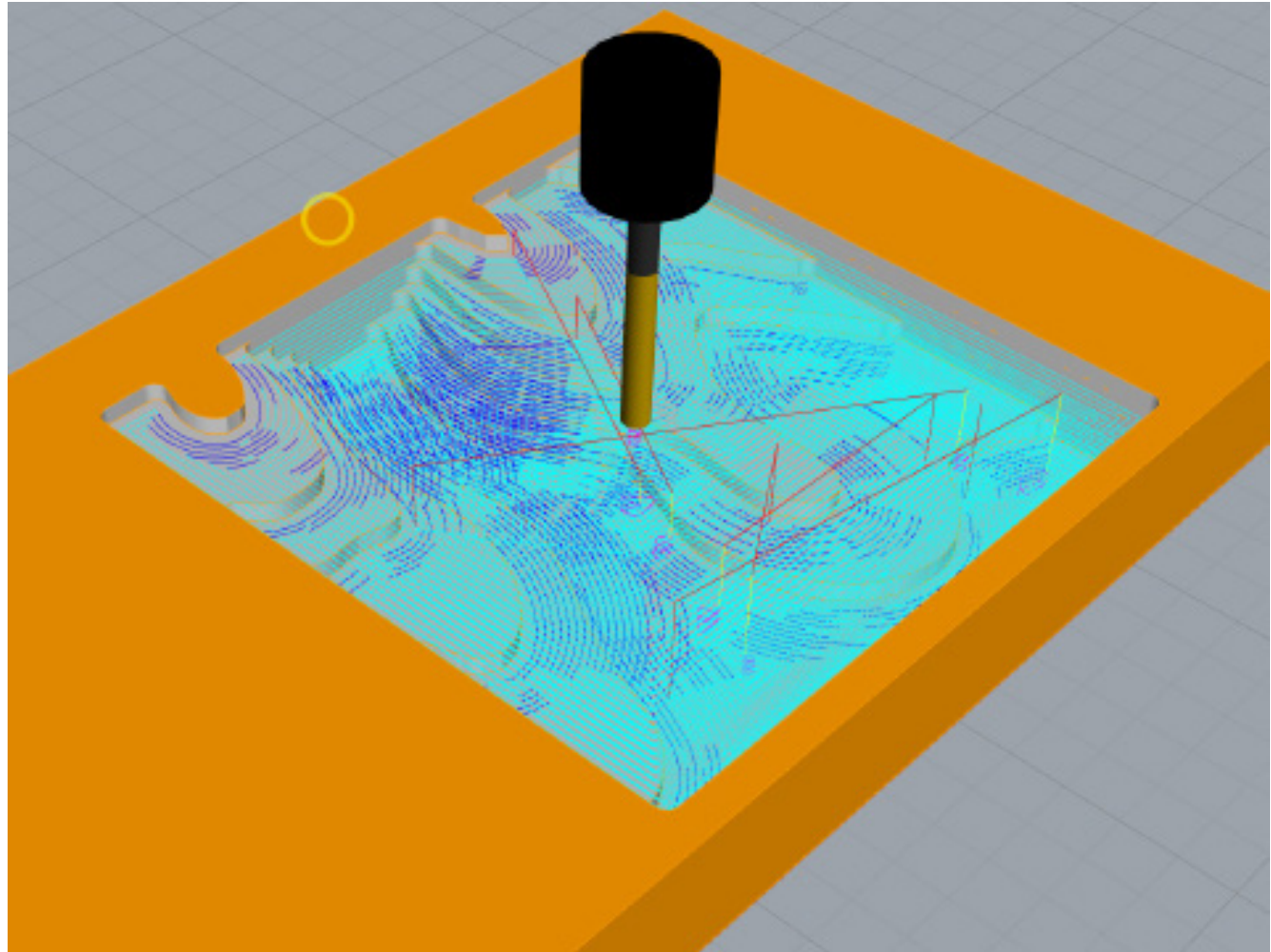
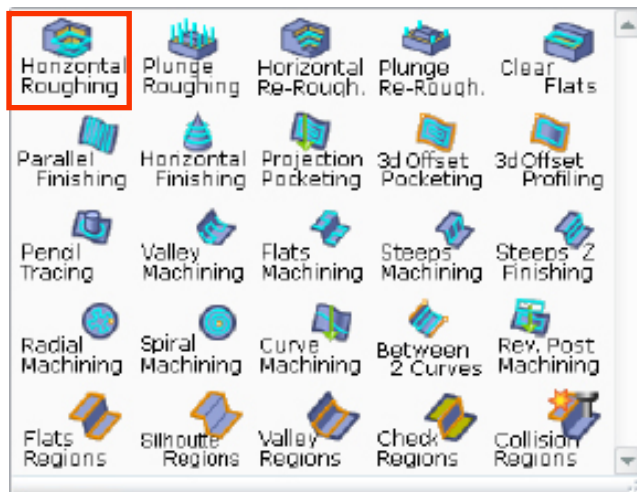


Horizontal Roughing

This is usually the first operation in any project. This operation removes the bulk of the material in your stock so that the toolpaths afterwards will encounter lesser resistance when finishing.

Roughing recognizes curvature in your Rhino model to compute toolpaths automatically, given a boundary.

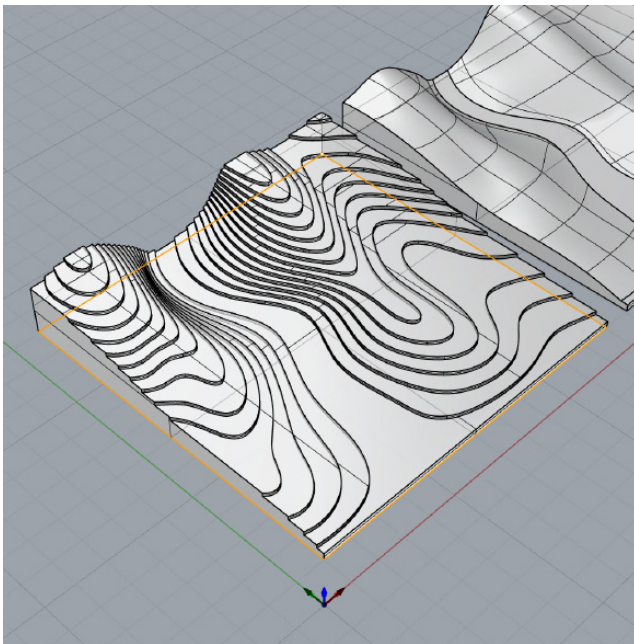
To set up all other tabs not shown in this section, refer to the **Common 3D Settings** section.



Horizontal Roughing ctd.

Select Control Geometry

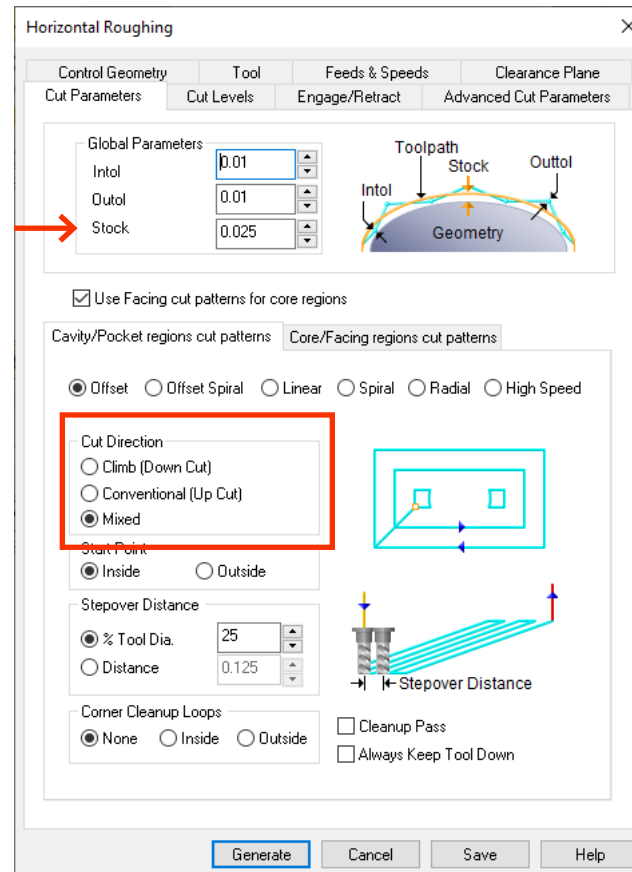
Choose your **control geometry** by selecting a rectangle around your model. The toolpath will automatically determine where inside this boundary it needs to cut. **Ensure all other tabs have been setup as shown previously.**



Cut Parameters

You should leave some stock behind if you intend to use finishing toolpaths.

Set **Cut Direction** to “**Mixed**”, as wall finish is usually not important for roughing.

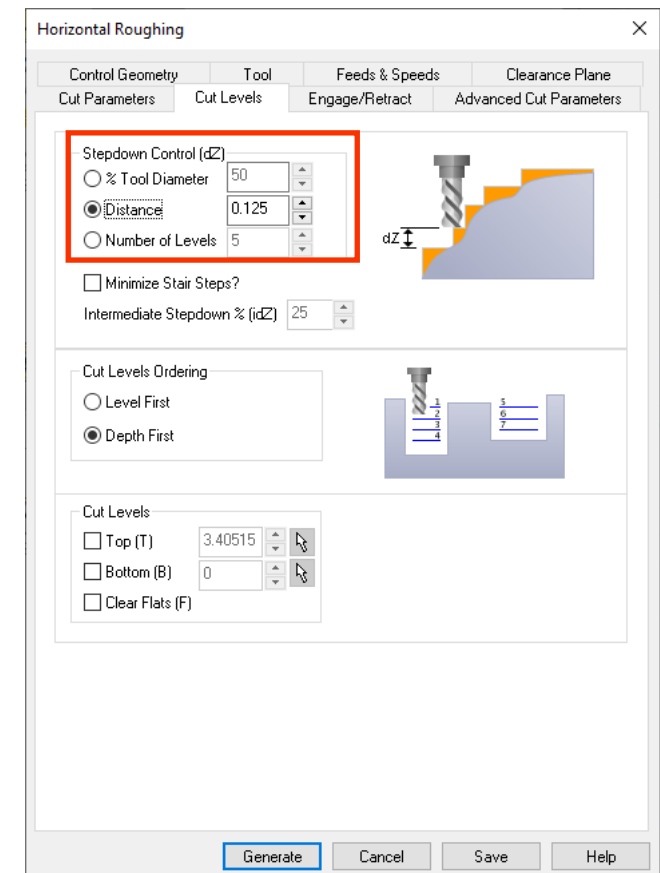


Cut Levels

Set the **stepdown** distance here.

If you have a **stepped model**, set the **Distance** equal to the height of your steps, as long as it does not exceed the limits set in the **Feeds & Speeds** chart.

Click “**Generate**”.



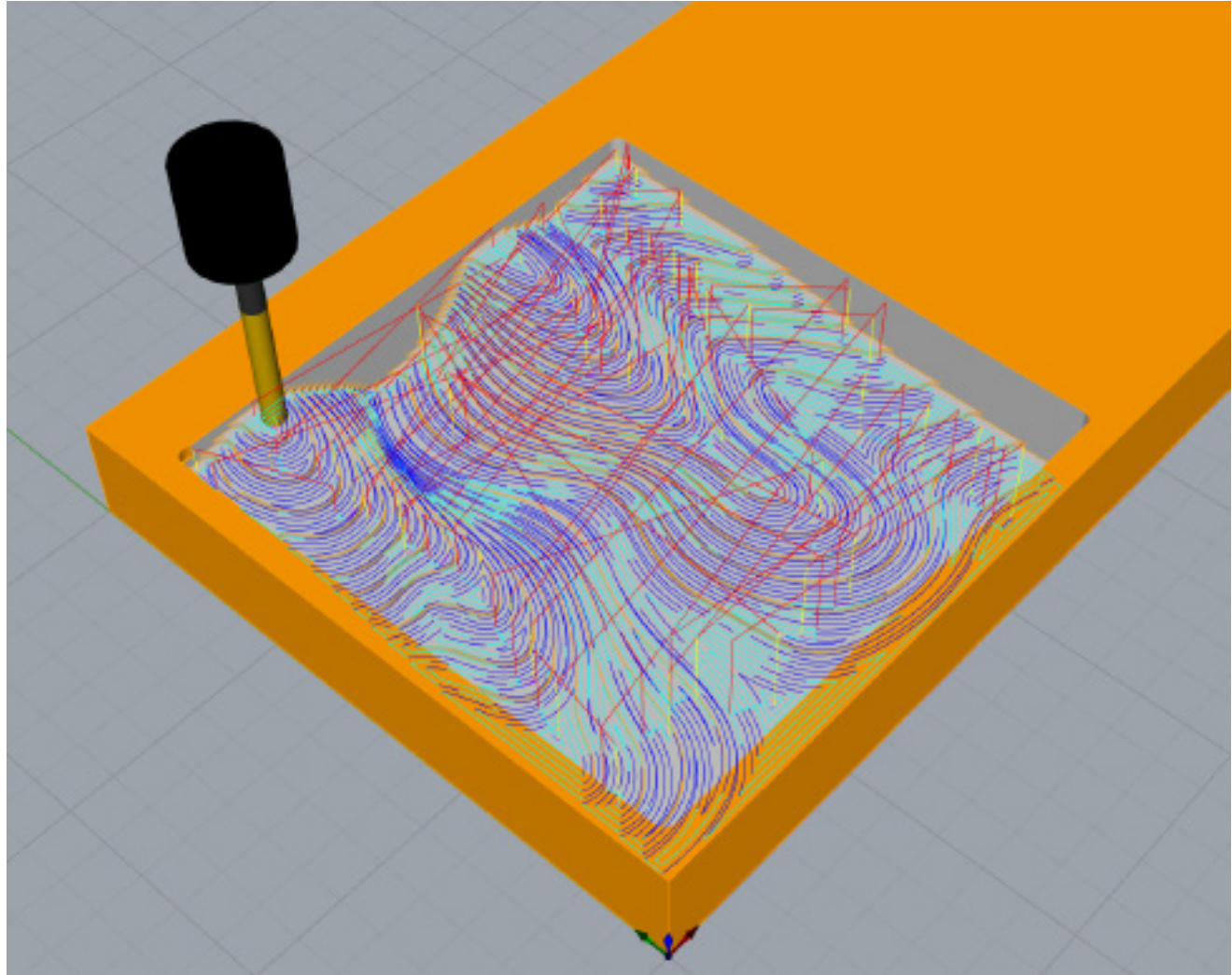
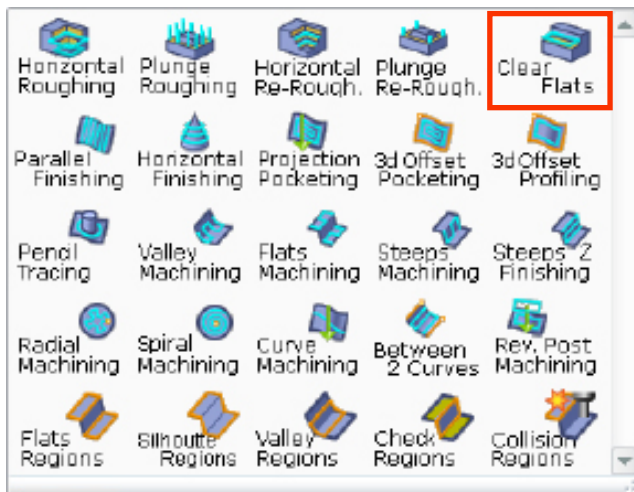
Clear Flats

This toolpath can be used for a stepped contour model.

You must do **Horizontal Roughing** before using **Clear Flats**.

As the name suggests, this tool will clear any flat areas in your model. A **flat tool is required** for this toolpath.

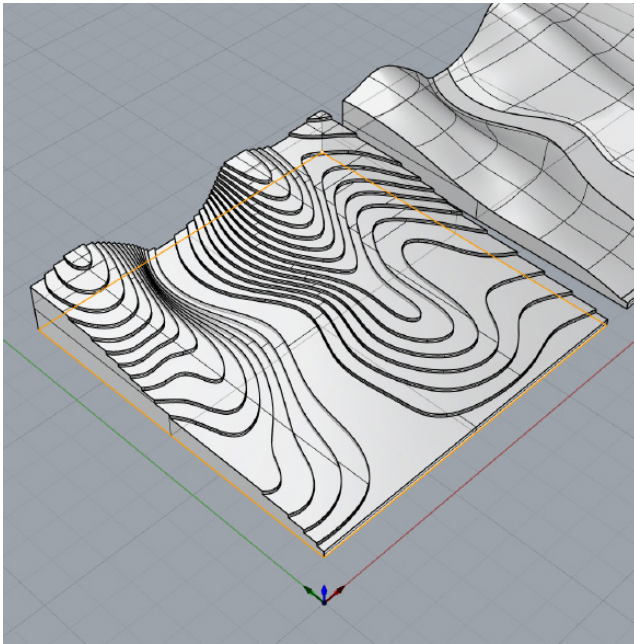
To set up all other tabs not shown in this section, refer to the **Common 3D Settings** section.



Clear Flats ctd.

Select Control Geometry

Choose your **control geometry** by selecting a rectangle around your model. The toolpath will automatically determine where inside this boundary it needs to cut. **Ensure all other tabs have been setup.**



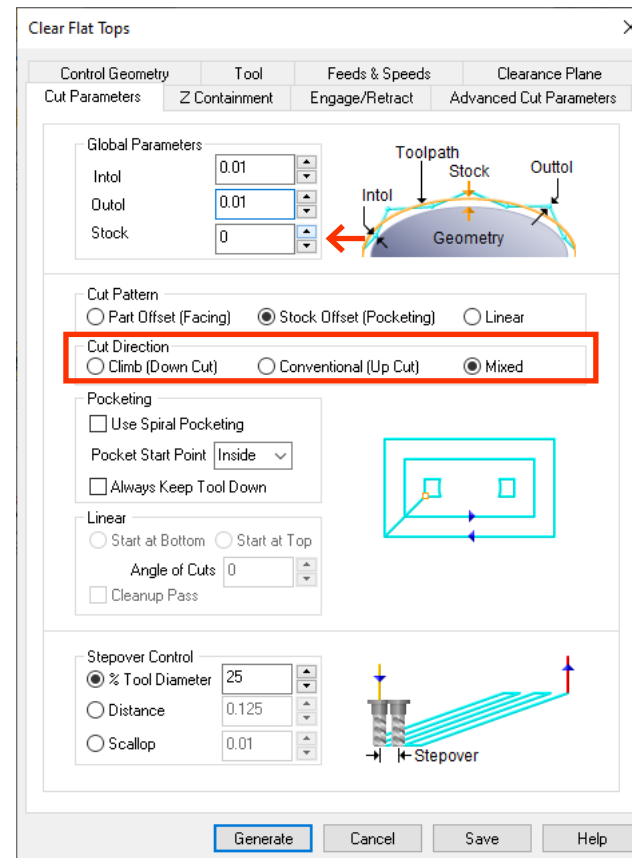
Cut Parameters

Stock should be set to **0**.

Cut direction can be set to “**Climb**” for a cleaner surface finish. In foam, you can set it to “**Mixed**”.

This toolpath may take several minutes to compute depending on how complex your model is.

Click “**Generate**”.



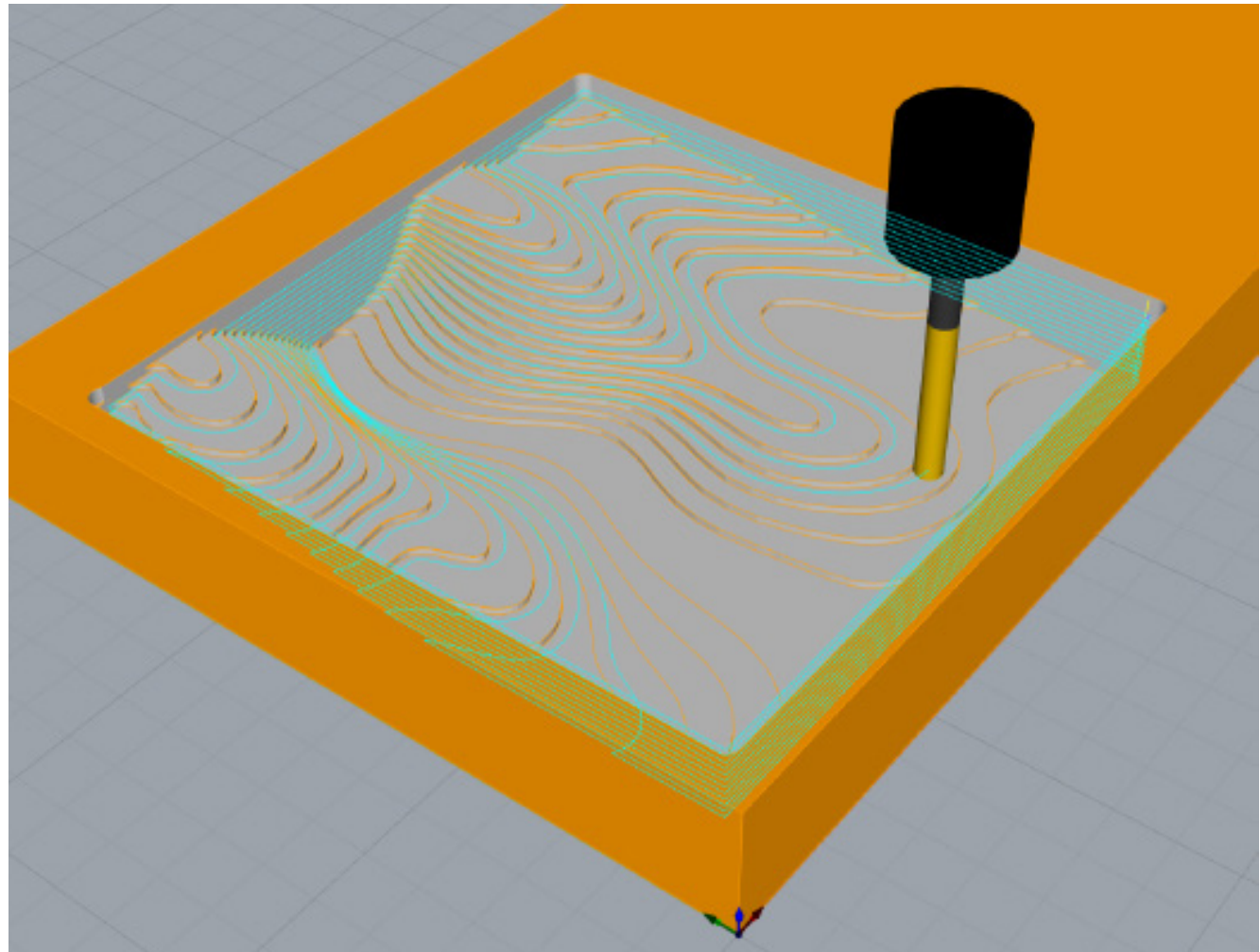
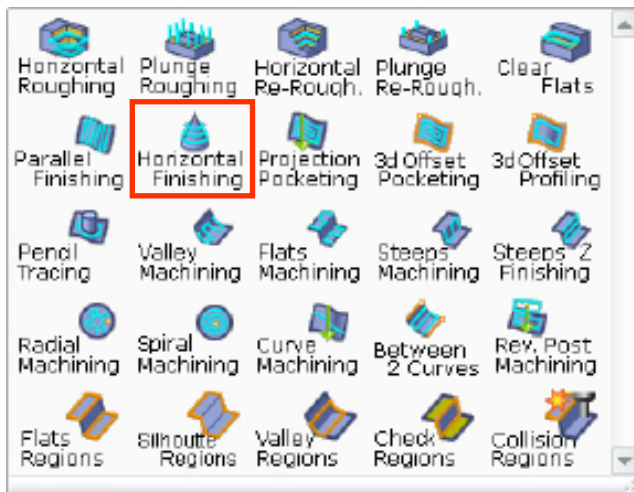
Horizontal Finishing

Horizontal Finishing can be used to clean up all vertical surfaces after a **Clear Flats** operation if desired.

Horizontal Finishing expects to encounter very little material when cutting. Ensure at the least you ran a **Roughing** operation first.

If your model is mostly vertical, or if the flat areas in your topography model are narrower than your tool diameter, you can use **Horizontal Finishing** instead of **Clear Flats**.

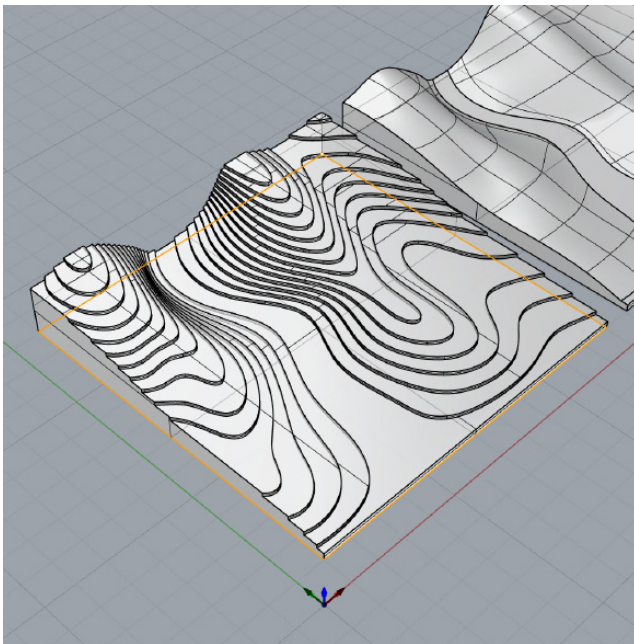
To set up all other tabs not shown in this section, refer to the **Common 3D Settings** section.



Horizontal Finishing ctd.

Select control geometry

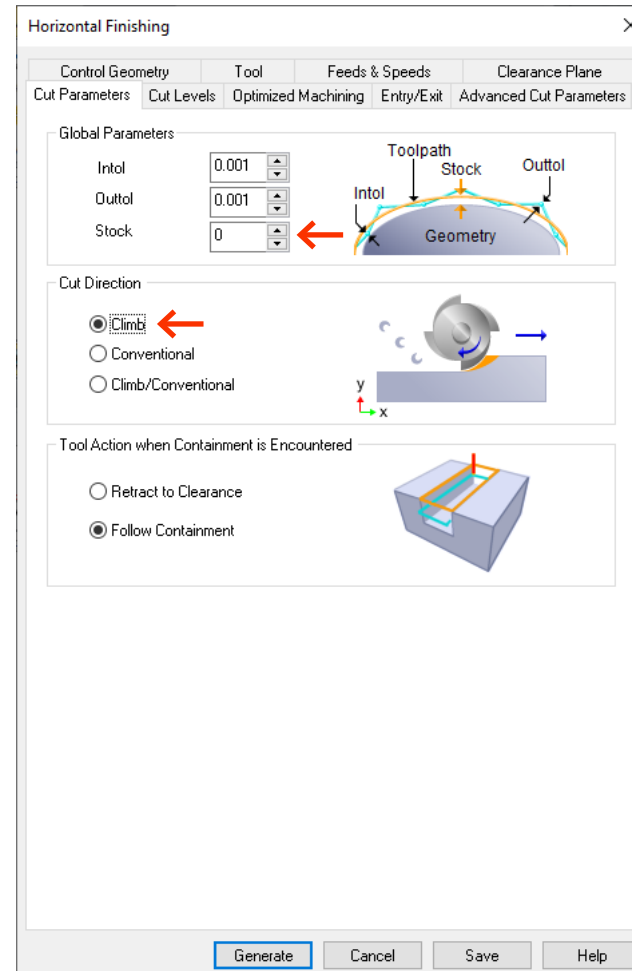
Choose your **control geometry** by selecting a rectangle around your model. The toolpath will automatically determine where inside this boundary it needs to finish. **Ensure all other tabs have been setup.**



Cut Parameters

Set **Stock** to **0**.

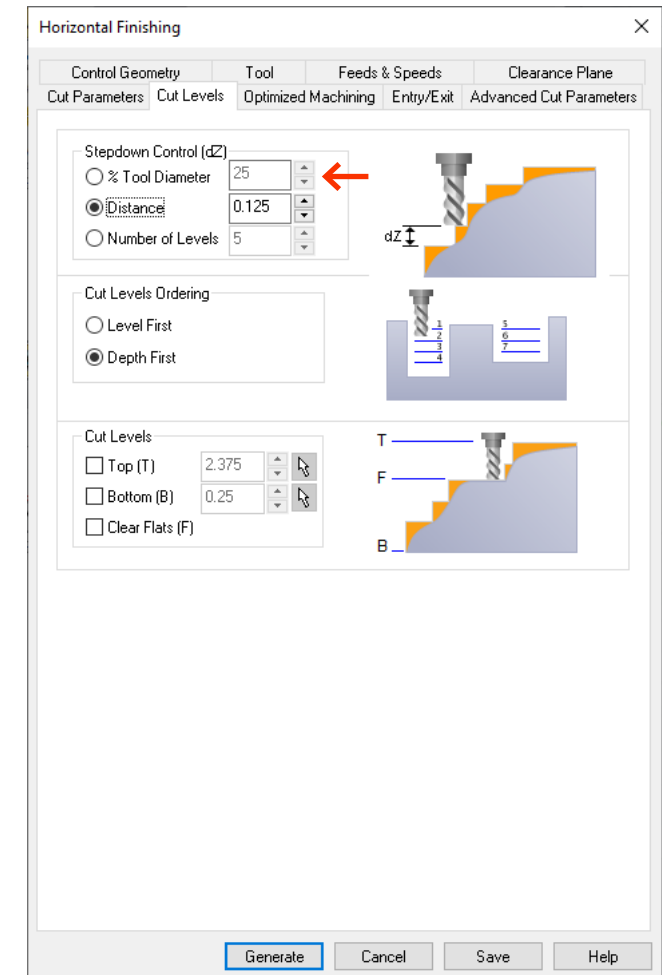
Cut direction should be set to “**Climb**” for better wall finish.



Cut Levels

Set **stepdown** distance to equal to your model step height, to a maximum of **50%** of your tool diameter, or according to the **Feeds & Speeds** table.

Click “**Generate**”.



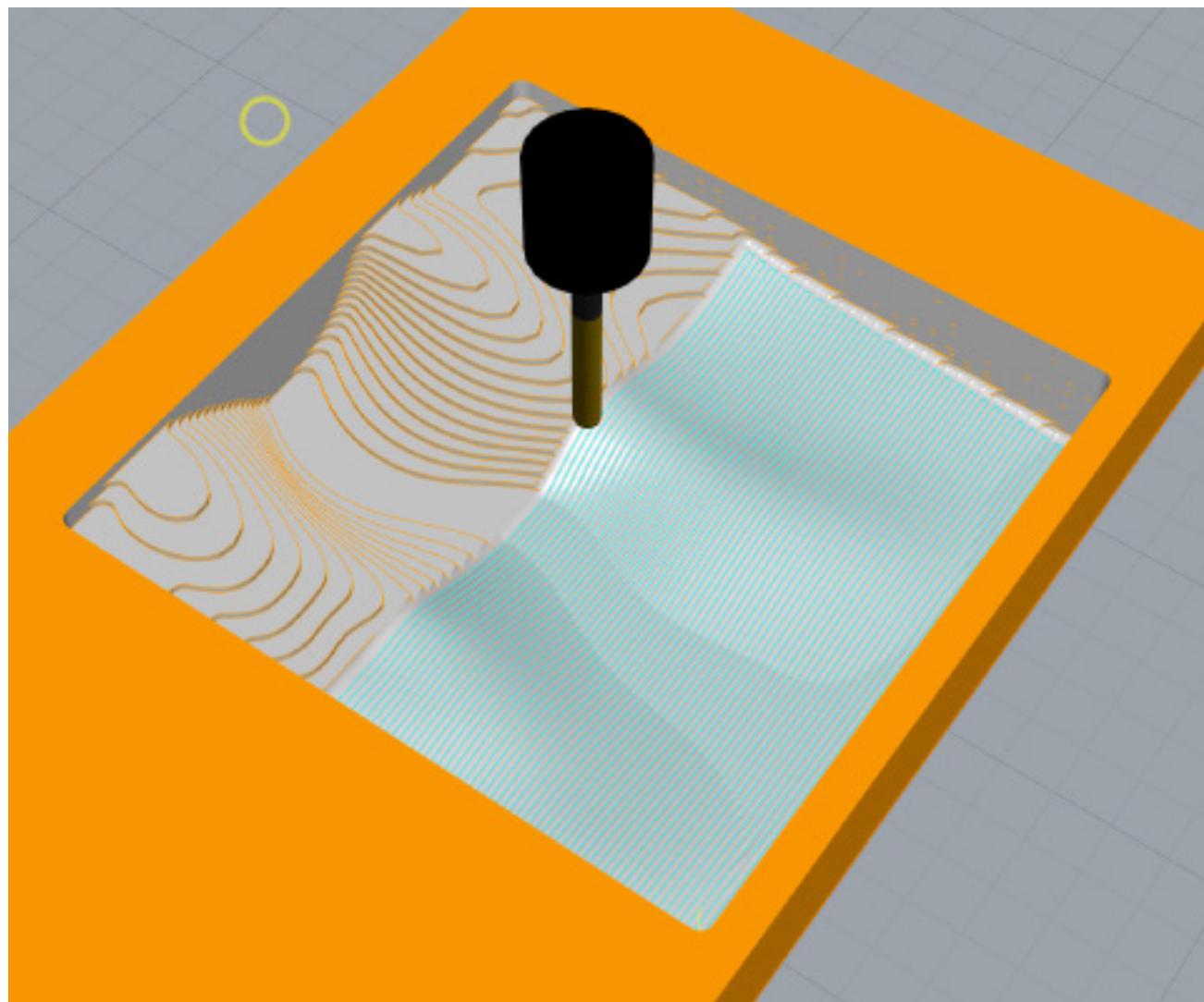
Parallel Finishing

This toolpath is used to create smooth 3D surfaces. It can be run using both a **ball**, or **flat** end mill.

A **ball-bit** is recommended for most **Parallel Finishing** operations.

You must do a **Horizontal Roughing** operation before using **Parallel Finish**.

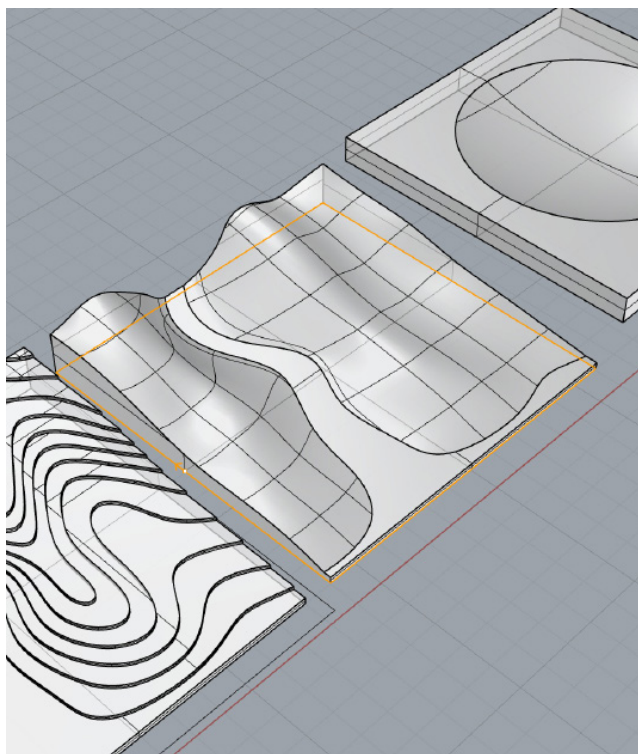
To set up all other tabs not shown in this section, refer to the **Common 3D Settings** section.



Parallel Finishing ctd.

Select Control Geometry

Select a closed boundary around your model as your **control geometry**. Ensure all other tabs have been setup.



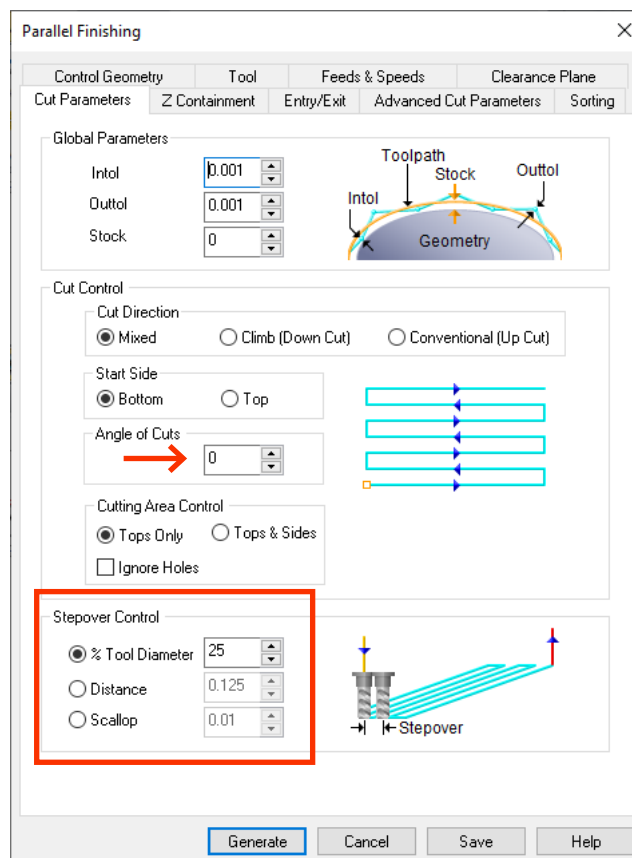
Cut Parameters

You can specify the **angle of cuts** for the parallel passes. An angle of **0** means the tool will move **left to right**.

By default, a **stepover** of **25%** of tool diameter is used. A smaller stepover will result in smaller scallops, at the cost of time.

For white foam, you can use **10% - 25%** for clean finishes, and **80% - 100%** for a finish with scallops.

Click **“Generate”**.



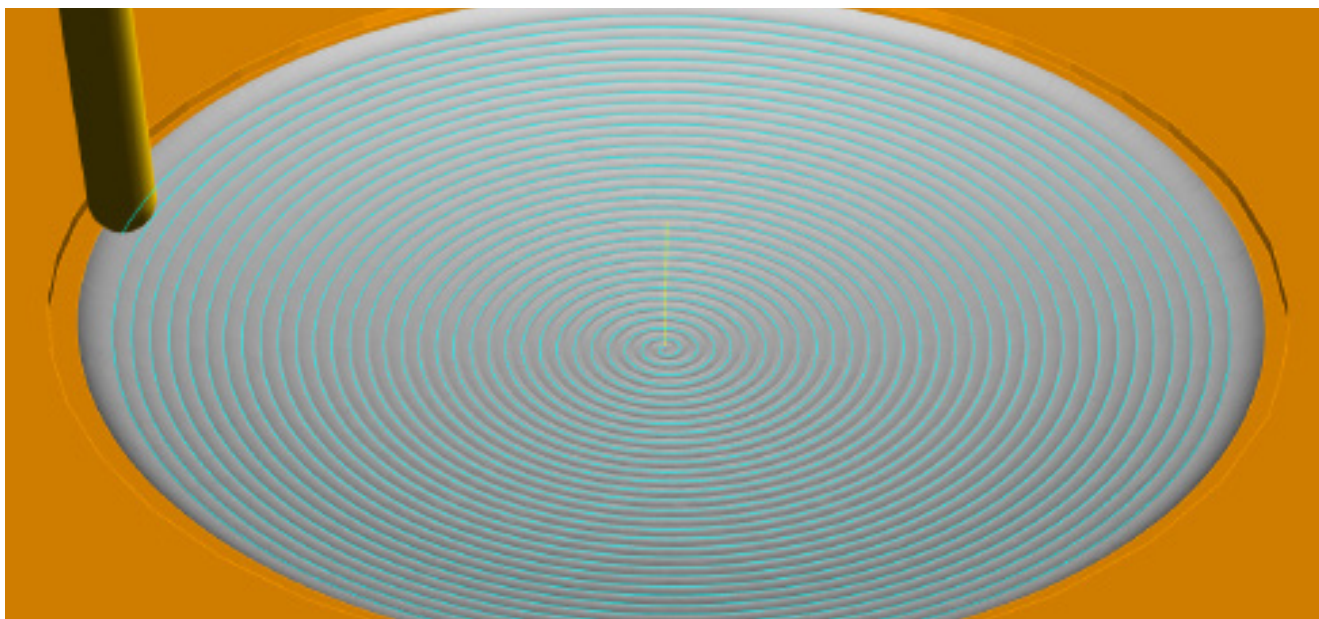
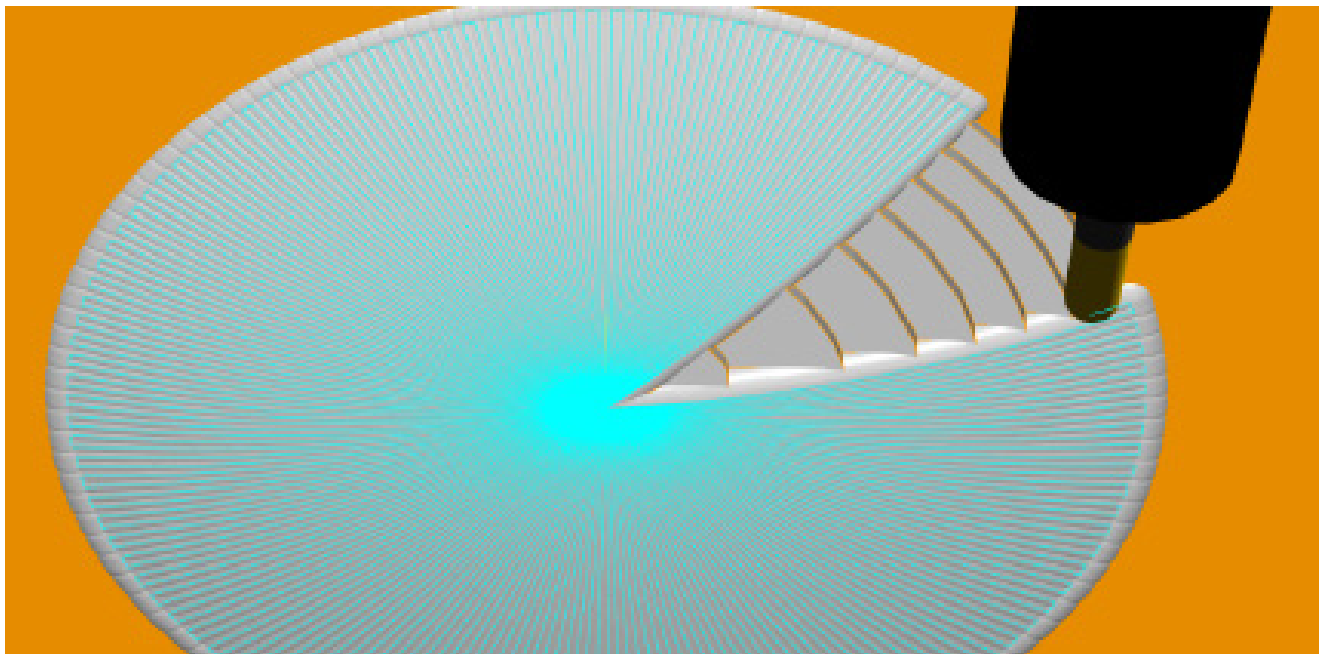
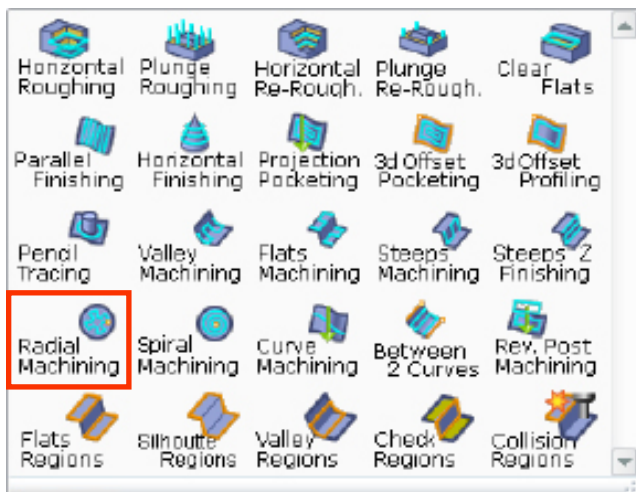
Radial Machining/ Spiral Machining

Radial Machining and **Spiral Machining** can be used as an alternative to **Parallel Finishing** for circular curved features, to leave a different 3D surface finish.

You can use both a **flat** and **ball-bit**, however a **ball-bit** is recommended.

You must do **Roughing** prior.

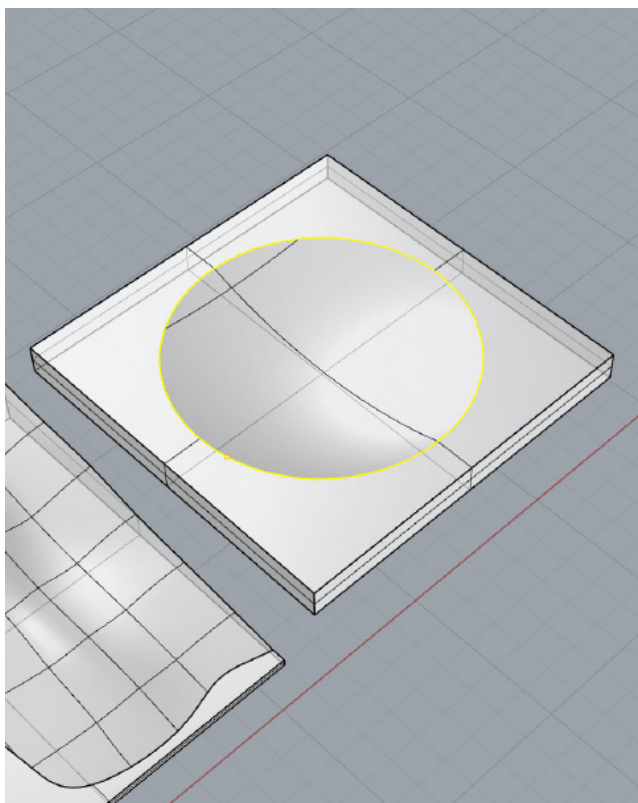
To set up all other tabs not shown in this section, refer to the **Common 3D Settings** section.



Radial Machining/Spiral Machining ctd.

Select Control Geometry

Select your geometry. It does not have to be circular. Ensure all other tabs have been setup as shown previously.



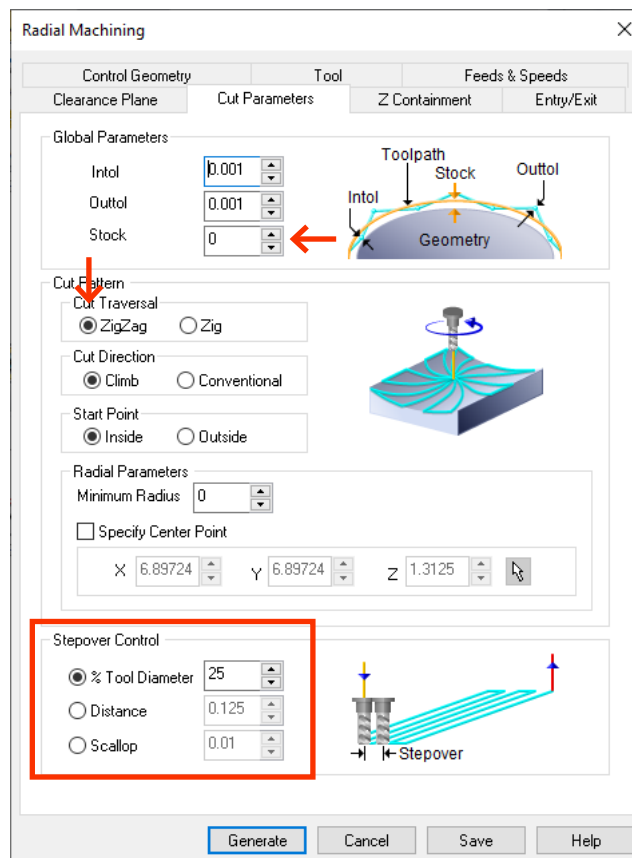
Cut Parameters (Radial Machining)

Set **Stock** to **0**.

Set cut traversal to **"ZigZag"**, cut direction can be left as is.

Lower **stepover** percentage results in a better surface finish at the cost of time. You can set surface finish between **25% - 50%** for a decent surface finish.

Click **"Generate"**.

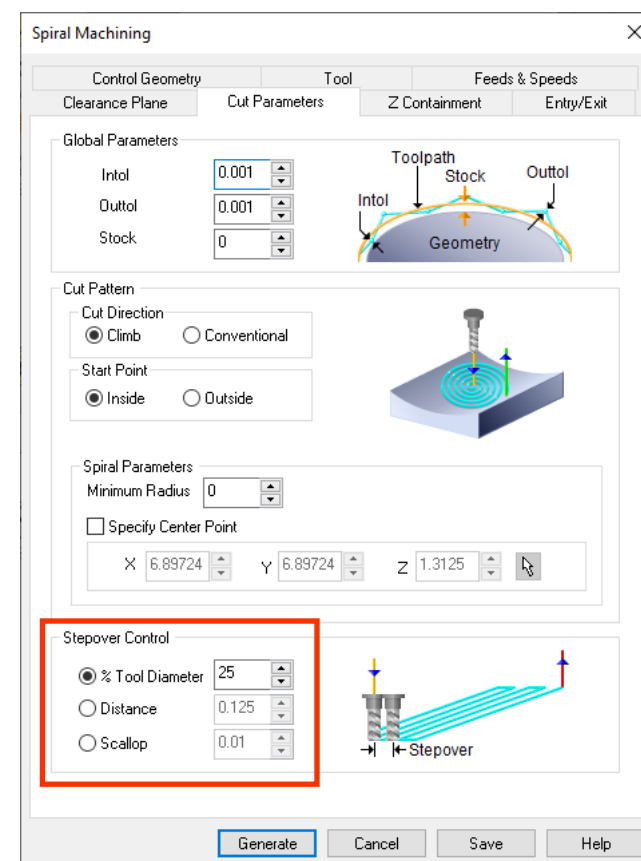


Cut Parameters (Spiral Machining)

These settings can be set as shown.

Like **Radial Machining**, you can adjust the **stepover** percentage to achieve a better surface finish at the cost of time.

Click **"Generate"**.



4D Toolpaths

video ↗

Equipment	59	-
Setting Up	60	Skip to 0:10 ↗
Roughing	63	Skip to 3:10 ↗
Finishing	69	Skip to 6:06 ↗

Equipment

The **Roland MDX 540** is a small scale CNC 'lathe' that allows for the fabrication of highly detailed components and geometries that would be impossible to achieve in **3-axis milling**. A rotary axis allows the stock material to turn slowly during milling, allowing the tool to gain access to areas that would be considered 'undercut' on a **3-axis** machine.

Both **2D** and **3D** operations can be performed as isolated cuts from different orientations within the same file, and a small selection of **4-axis** operations allow for simultaneous movement of all axis.

We have carefully calibrated our machine to share the same RhinoCAM programming techniques that are already used on the **3-axis XYZ 4008**.

It is expected that students will have read and familiarized themselves with both **2D** and **3D** milling operations detailed in this manual before approaching **4-axis milling**.



Setting Up

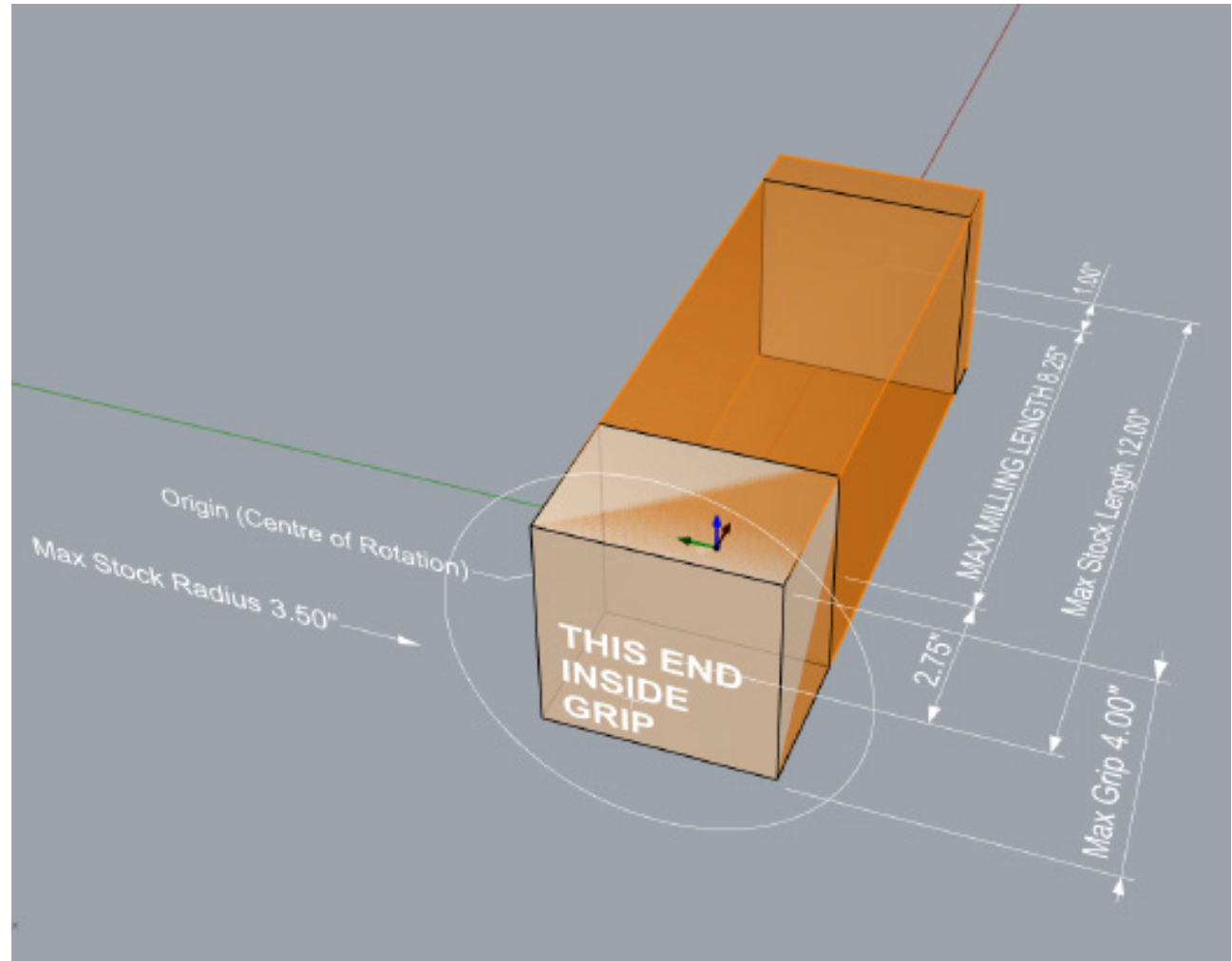
Setting up your file

A **template** ⤴ file for the **Roland MDX 540** can be found on the FTP under **Groups > CNC Milling > Submissions > ROLAND MDX 540**. Please copy the template file and tool library to your computer.

When positioning your model to cut, it should be centered along the Rhino **x axis**, and should only exist in the positive direction. You will notice a part of the stock in the negative x direction; this is the material dedicated to fixturing clamps for positioning, and the tool cannot travel here.

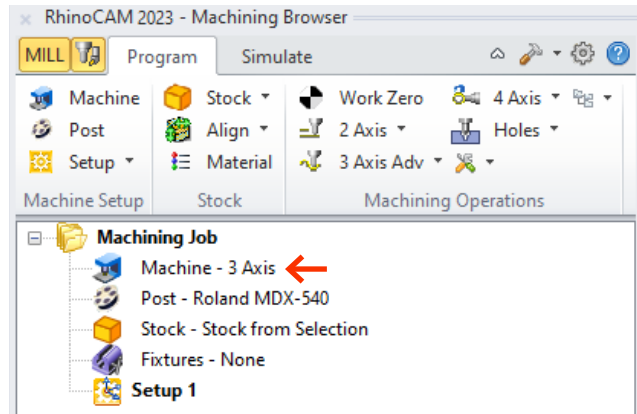
The template file includes annotations to guide you. It is important to note these key dimensions:

Max stock length:	12" (includes fixturing stock)
Max fixture opening:	4"
Max millable length:	8.25"
Max stock radius:	3.5"

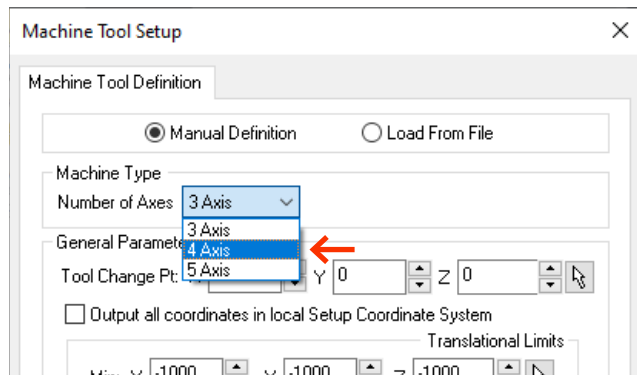


Setting up your file ctd.

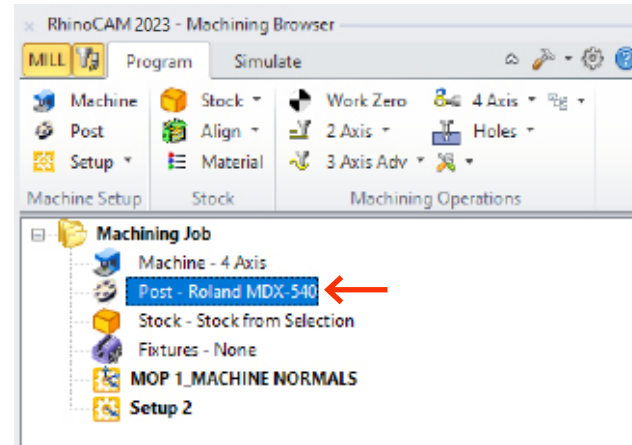
Double-click on **Machine** to open the **Machine Tool Setup** window.



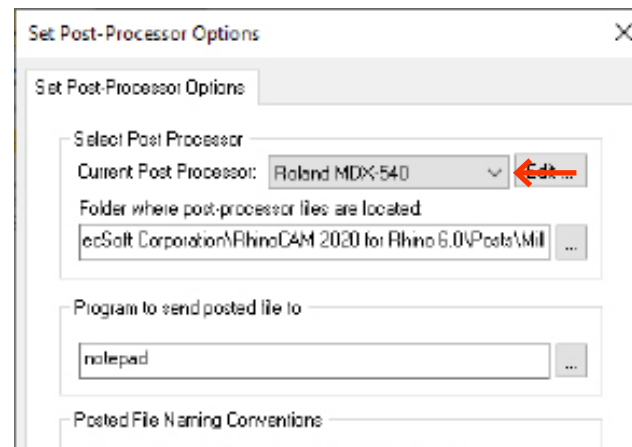
Set the number of axes to **4 Axis**, instead of **3 Axis**, and click “OK”.



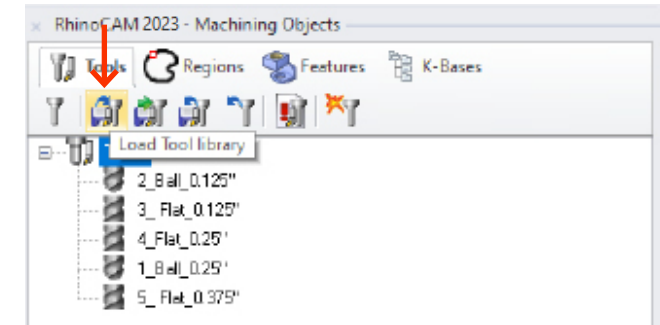
Then, double-click on **Post** to open the **Set Post-Processor Options** window.



Set the current post processor to **Roland MDX 540**, and click “OK”.



Then, load the **Roland** tool library from the FTP.



You are now ready to move onto the next step.

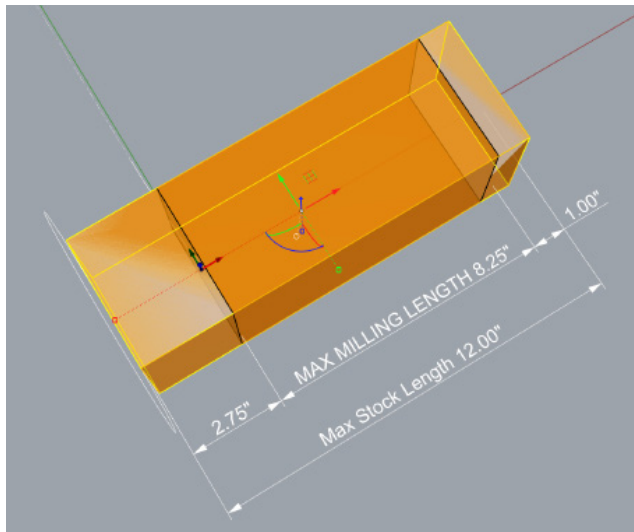
Setting up your stock

Create a box that represents the exact dimensions of your stock.

Your stock can be shorter than the max milling length, as long as it extends exactly **2.75"** into the negative x direction, and has **1"** of work holding space at the end of it.

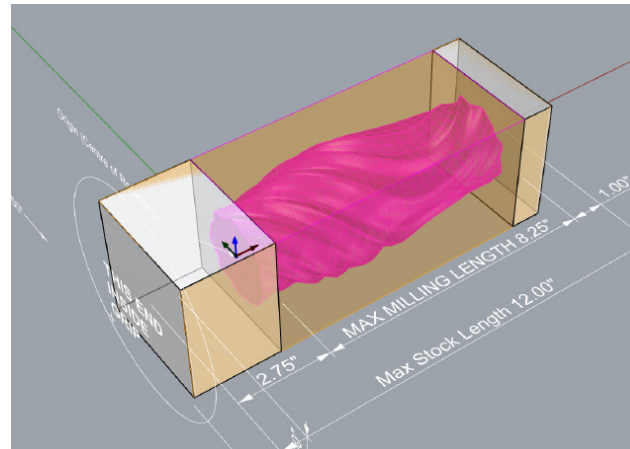
It may be helpful to model these regions as separate geometry, placed in the **"GRIP STOCK"** layer.

Then, same as before, right click **Stock**, and set **"Stock from Selection"** from the **Machining Browser** window.

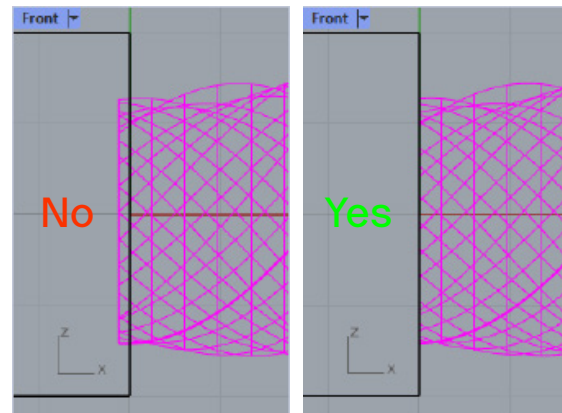


Placing your model

Position your desired geometry within the millable region, **centered on the x axis**. Your geometry should be **fully enclosed within your stock**, and **must be touching the fixturing material on both sides**.



Ensure your geometry ends at the **fixturing stock**. If it extends beyond the **fixturing stock**, or does not reach the **fixturing stock**, trim or scale your model so that it ends at the **fixturing stock** on both sides.



The fixturing stock is what the machine holds onto on both sides to spin the stock. These areas are not millable.

Make sure to hide your stock using Rhino layers.

You must hide your **stock** material in Rhino, but you do not need to hide the **fixturing stock**.

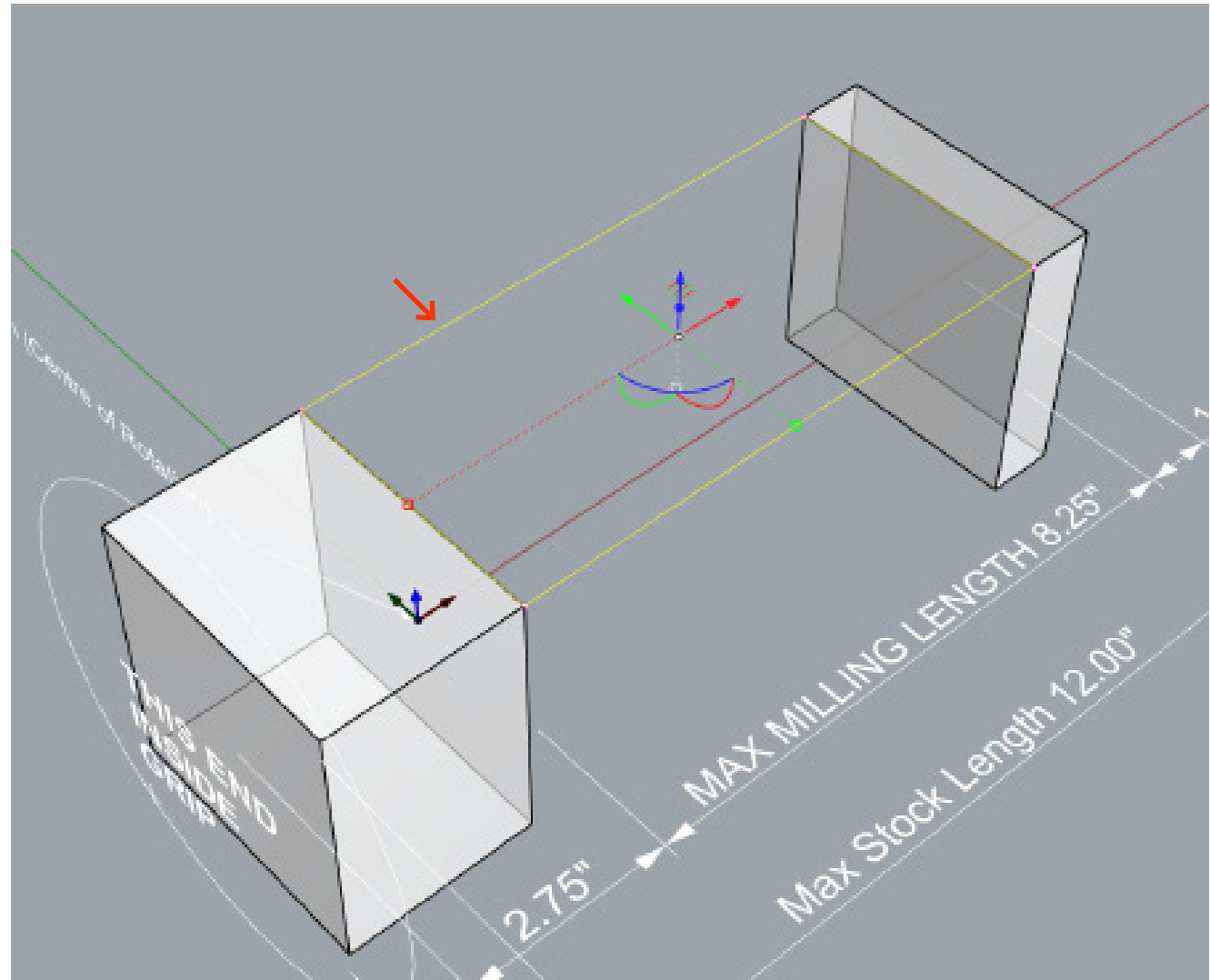
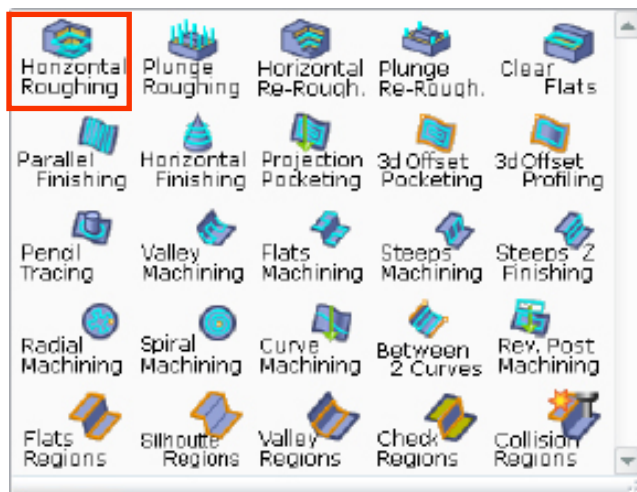
Roughing

2.5 Axis Horizontal Roughing

In 4-axis milling on the Roland MDX 540, we can also use 2-axis, and 3-axis toolpaths.

To define our roughing operation, we must first create a **rectangle** between the two fixturing areas to define our **control geometry**.

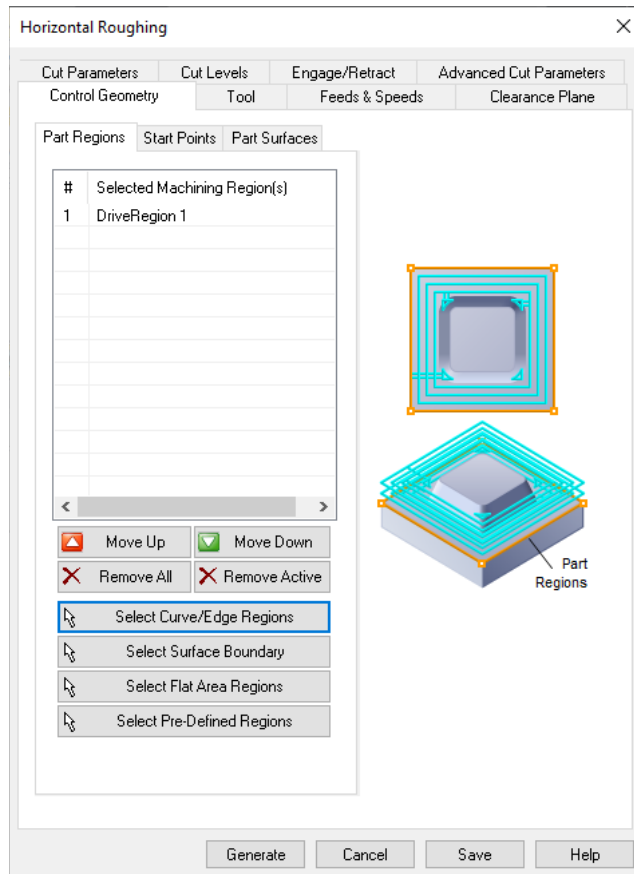
Then we can select the **Horizontal Roughing** toolpath from the **3 Axis Adv** dropdown.



2.5 Axis Horizontal Roughing ctd.

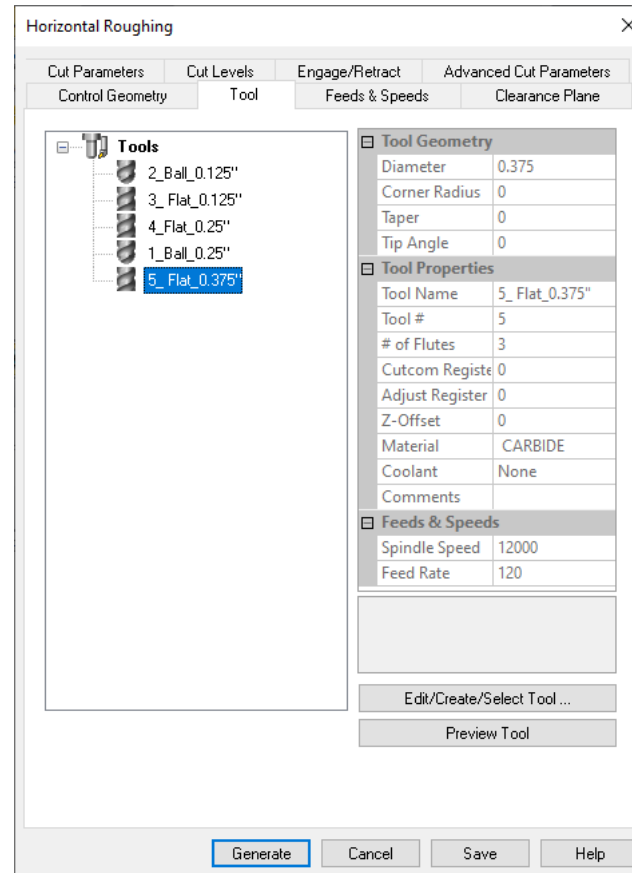
Control Geometry

Select the **rectangle** you just created as your **control geometry**.



Tool

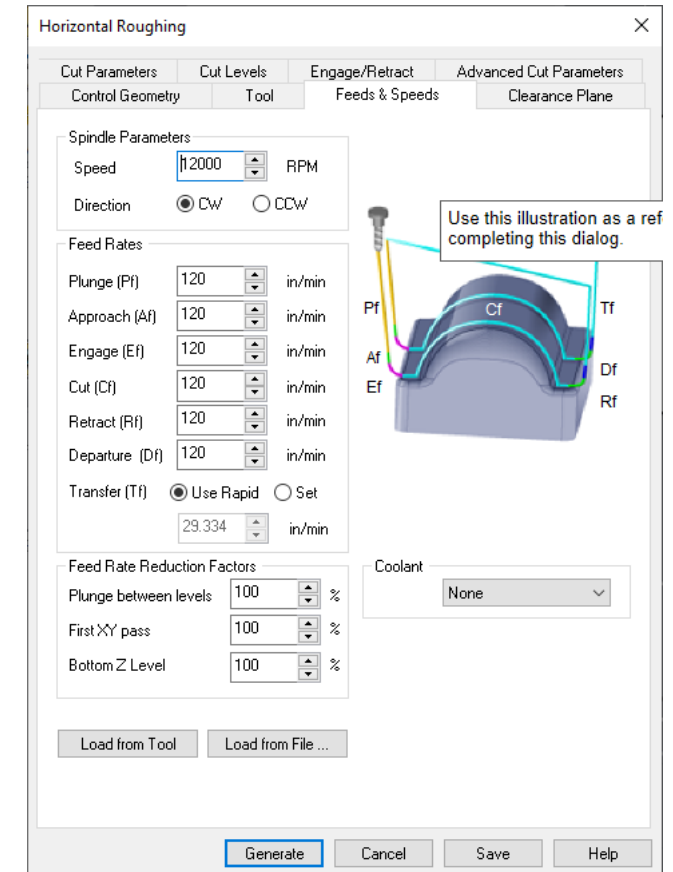
Select the **largest tool** to speed up the **Roughing** toolpath.



Feeds & Speeds

A TA will set your **Feeds & Speeds**.

The **Roland** can spin up to a maximum of **12000 RPM**.

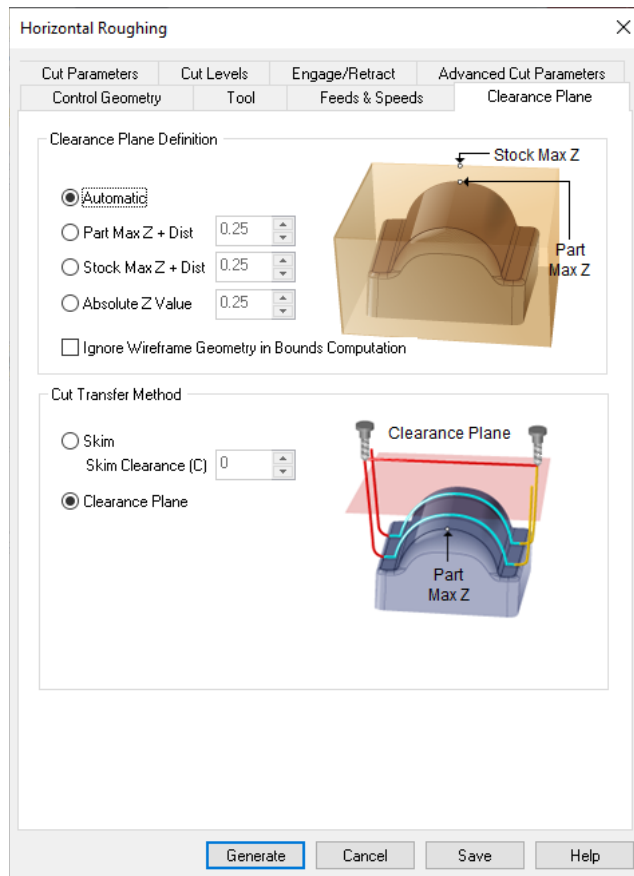


2.5 Axis Horizontal Roughing ctd.

Clearance Plane

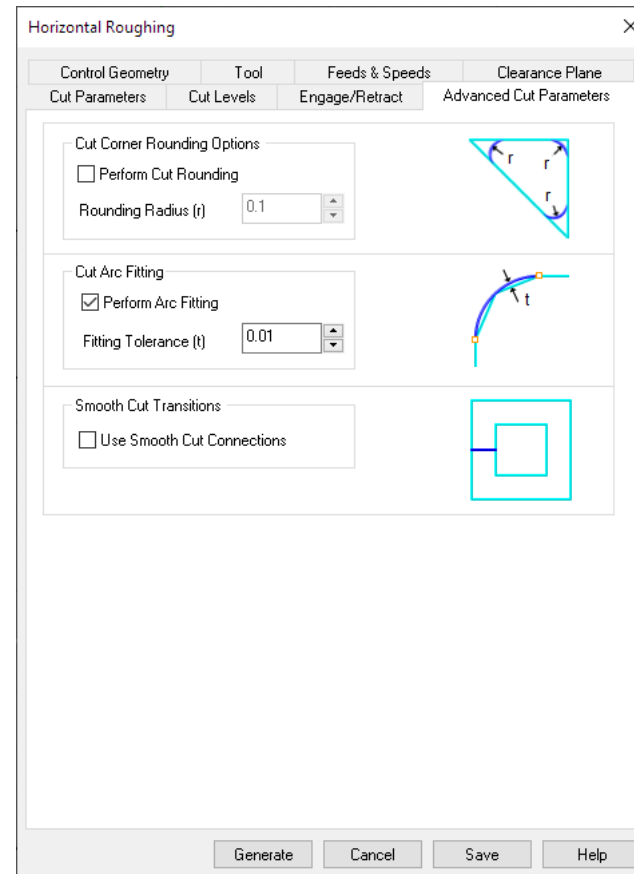
Leave this tab as is.

Verify the clearance plane is visible above your part.



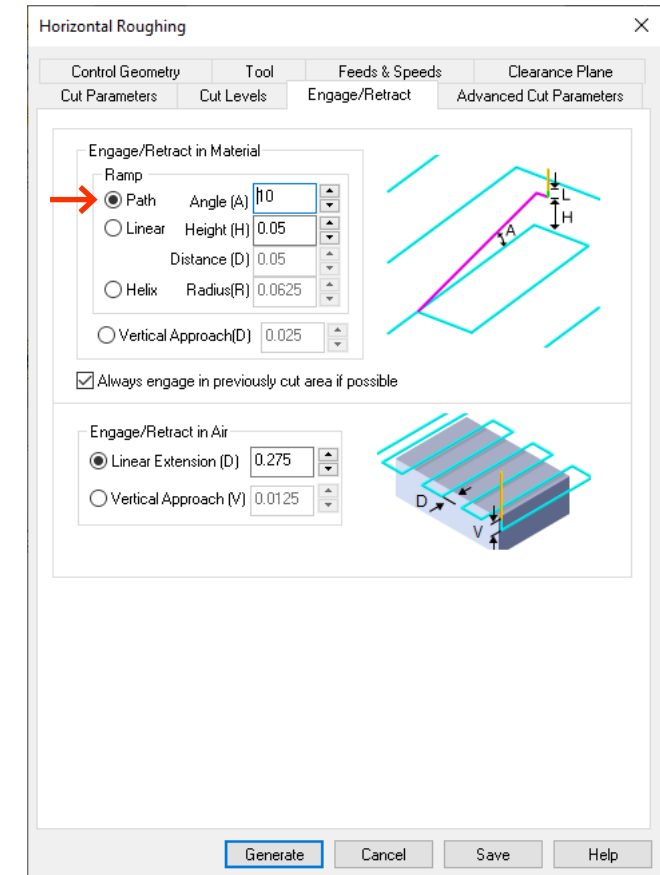
Advanced Cut Parameters

Leave this tab as is.



Engage/Retract

Ensure **Engage/Retract** is set to "Path". Leave the other settings as is.

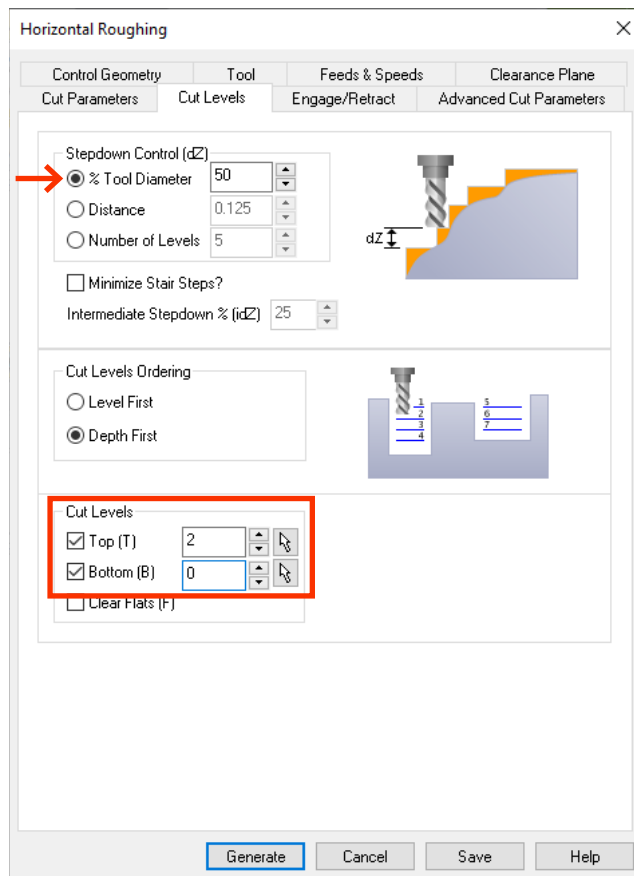


2.5 Axis Horizontal Roughing ctd.

Cut Levels

For roughing, set the **stepdown** to **50%** of tool diameter.

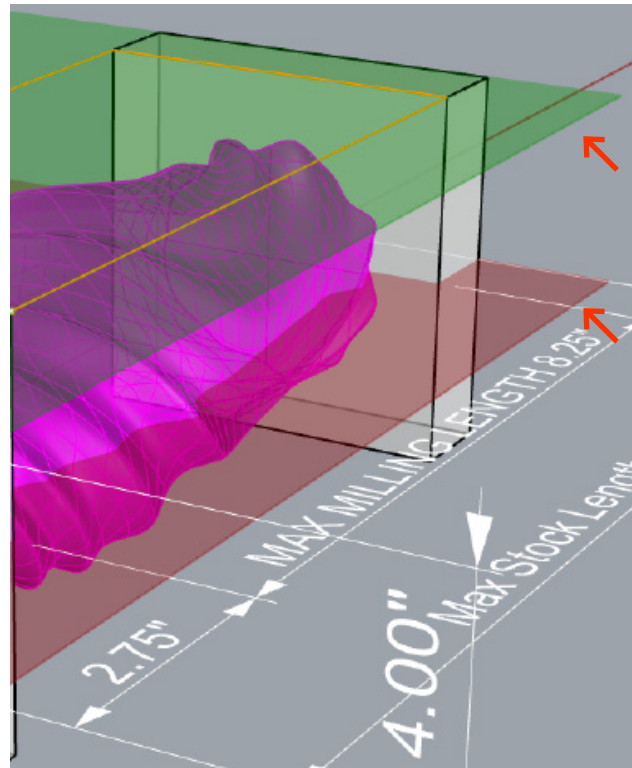
In **Cut Levels** section, select both “**Top**” and “**Bottom**”.



Set “**Top**” to the top of your stock by clicking on the **cursor icon** to the right, then selecting a point on the top face edge of the fixturing stock.

Set “**Bottom**” to **0**. We’re going to rough from both top and bottom of the stock to avoid undercuts, so setting bottom to 0 will have both operations meet in the middle.

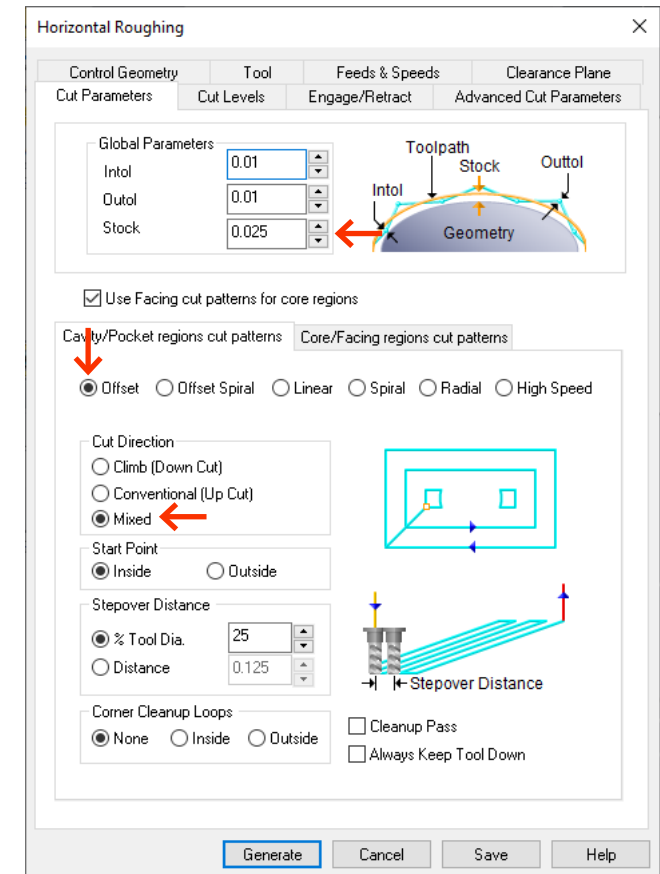
The **green plane** represents the top of the cut, the **red plane** represents the bottom of the cut. Your planes should look like this.



Cut Parameters

Set **stock** to leave at **0.025**". Set **cut pattern** to “**Offset**”, cut direction can be left on “**Mixed**”. Leave the other settings as is.

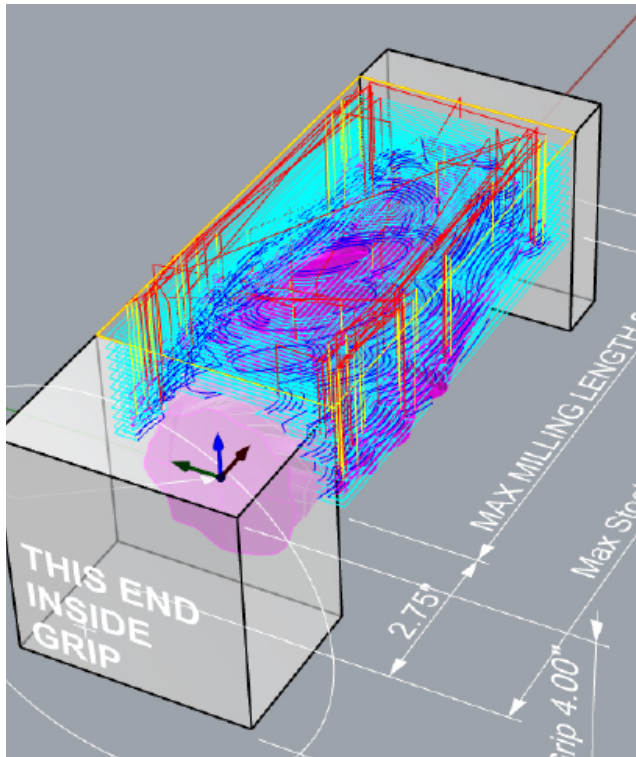
Click “**Generate**”.



2.5 Axis Horizontal Roughing ctd.

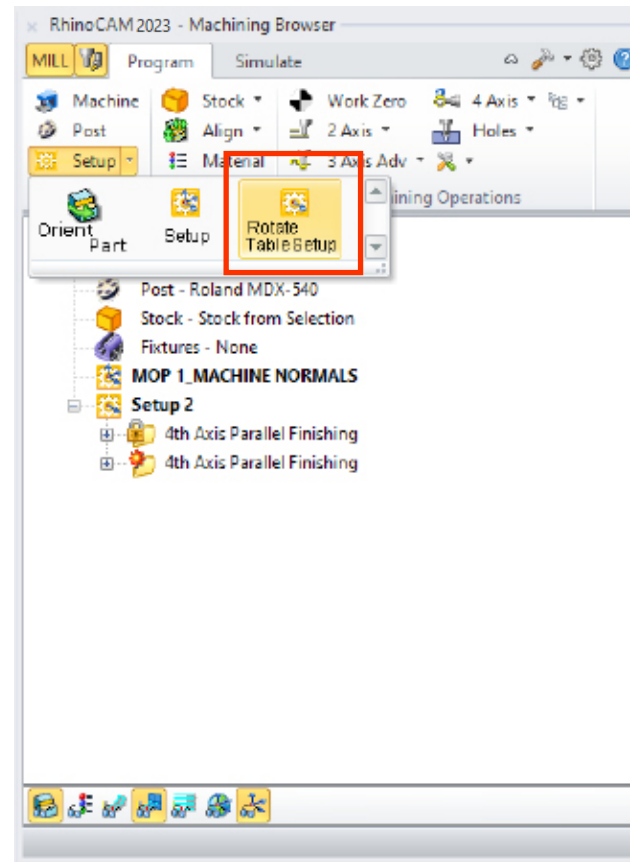
Once the toolpath has been generated, it will look like this. Toolpaths will only be populated on the top side.

Now we must rough the other side.



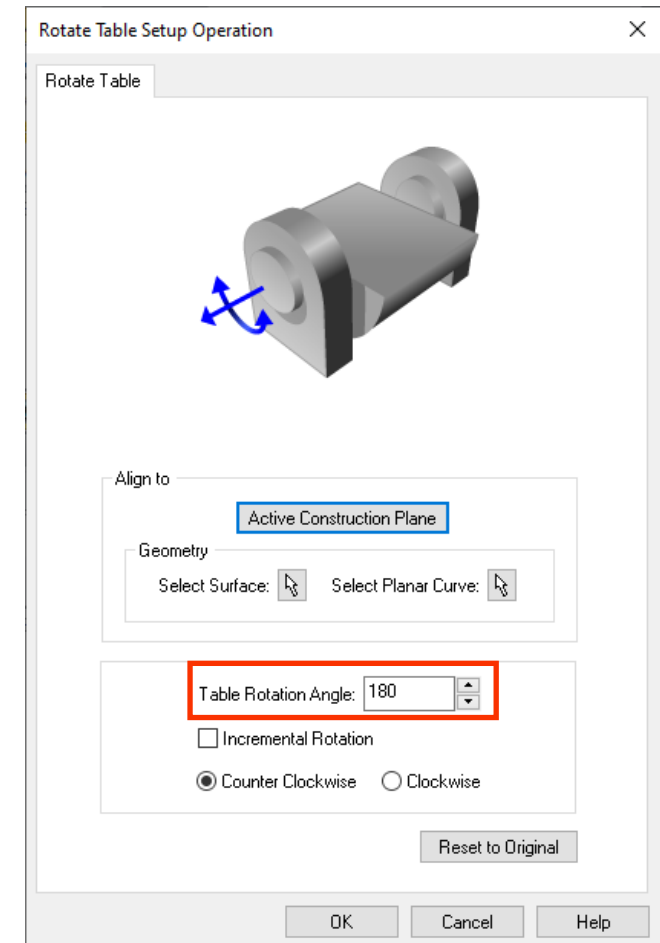
To rough the other side, we need to rotate our setup.

From the **Setup** dropdown, select **Rotate Table Setup**.



The **Rotate Table Setup Operation** window will open. Set the **Table Rotation Angle** to **180**, to rotate the stock **180°**.

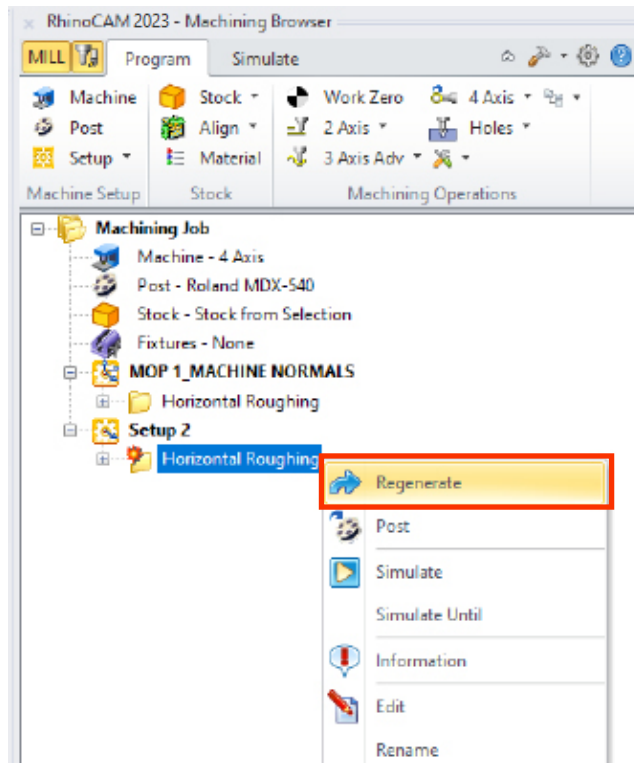
Click "OK".



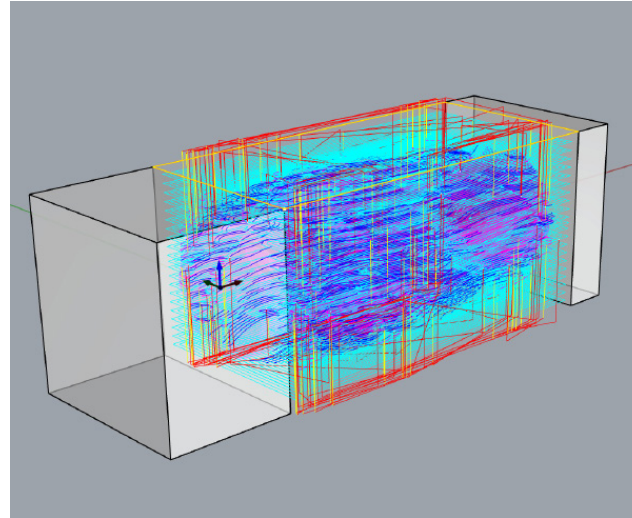
2.5 Axis Horizontal Roughing ctd.

RhinoCAM will create a new setup called **Setup 2** that is rotated **180°** like we set.

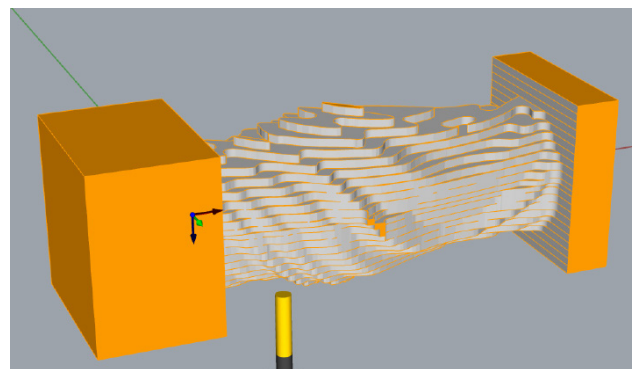
Copy and paste the previous **Horizontal Roughing** operation into this **new setup**. Then **Regenerate** it.



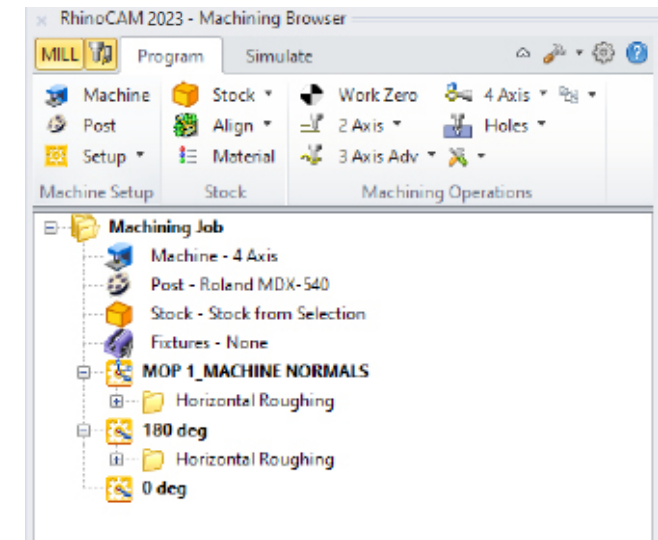
You will now see toolpaths on the bottom side of the stock. Note that this is a Rhino quirk, in reality it is the stock that is rotating. The tool always approaches from above.



You can **simulate both operations** in order to view the roughed out geometry.



For the next operation, reset the rotation by again creating a new **Rotate Setup**, and setting table rotation angle this time to **0°**.



It may be helpful to rename the setups according to the stock angle set.

We are now ready to add the finishing step.

Note: If you do not see your toolpaths on the bottom side of your model, delete the second setup, and try rotating again, making sure to enter the values exactly as shown in the last page.

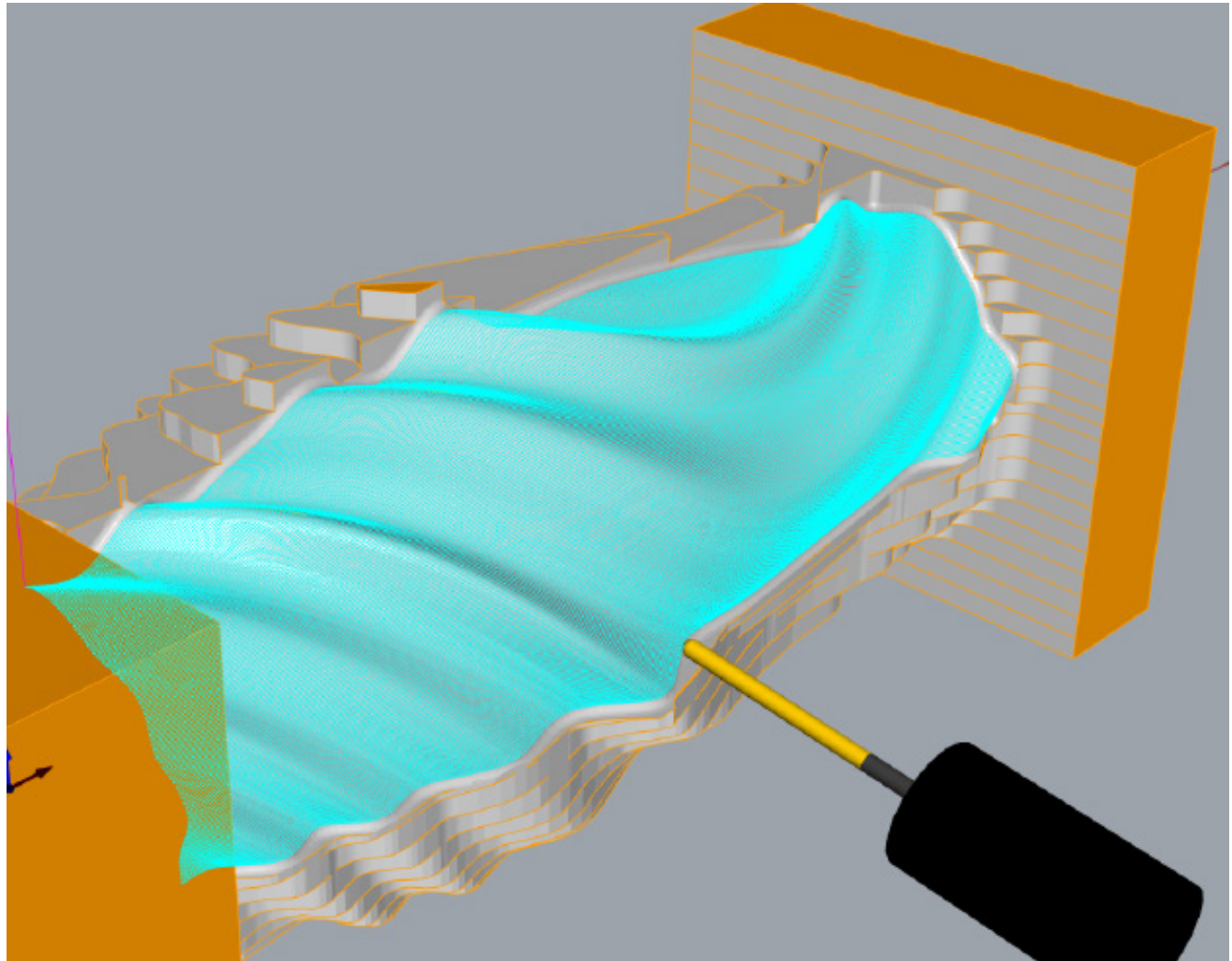
Finishing

4 Axis Parallel Finishing

Like **3 Axis Parallel Finishing**, this operation moves the tool left to right following the contours of the model. In **4 Axis**, the **stock is also rotating** while this happens. This toolpath is found under the **4 Axis** dropdown in the operations column.

Make sure to do **Roughing** from both sides before Parallel Finishing.

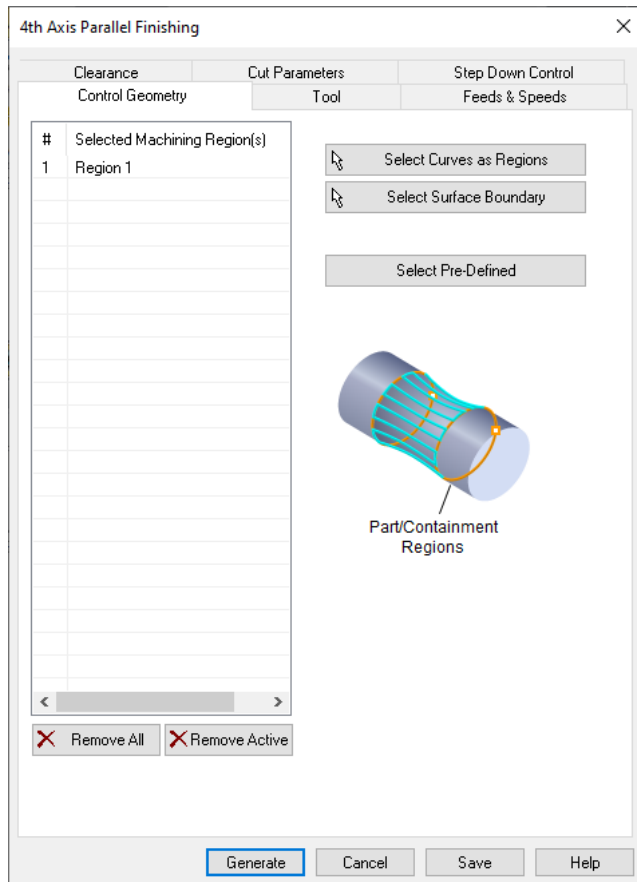
Note that while you may see the tool spinning around the stock in simulation, in reality, the stock spins, and the tool always approaches from above.



4 Axis Parallel Finishing ctd.

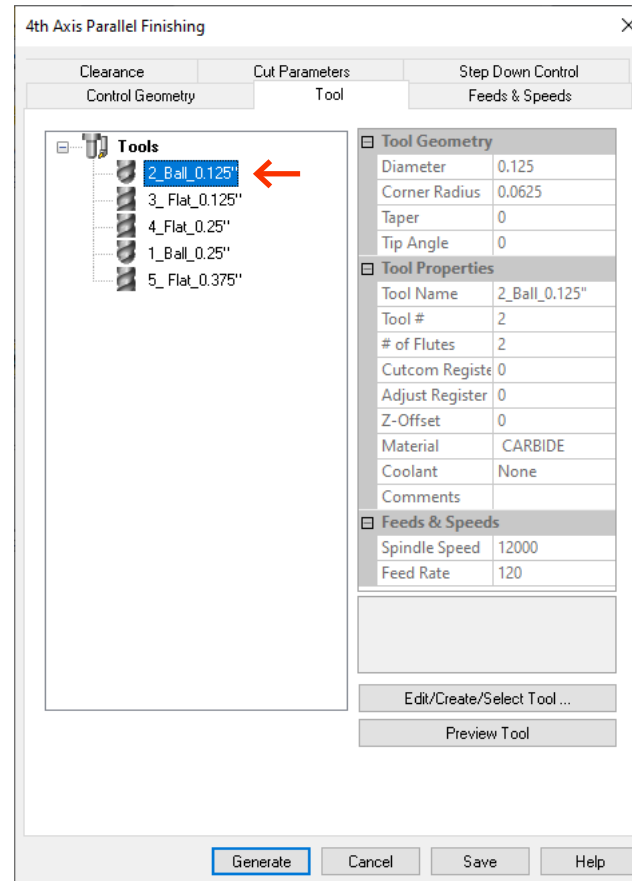
Control Geometry

Do not select any control geometry for this toolpath. If there is pre-selected control geometry here, remove it.



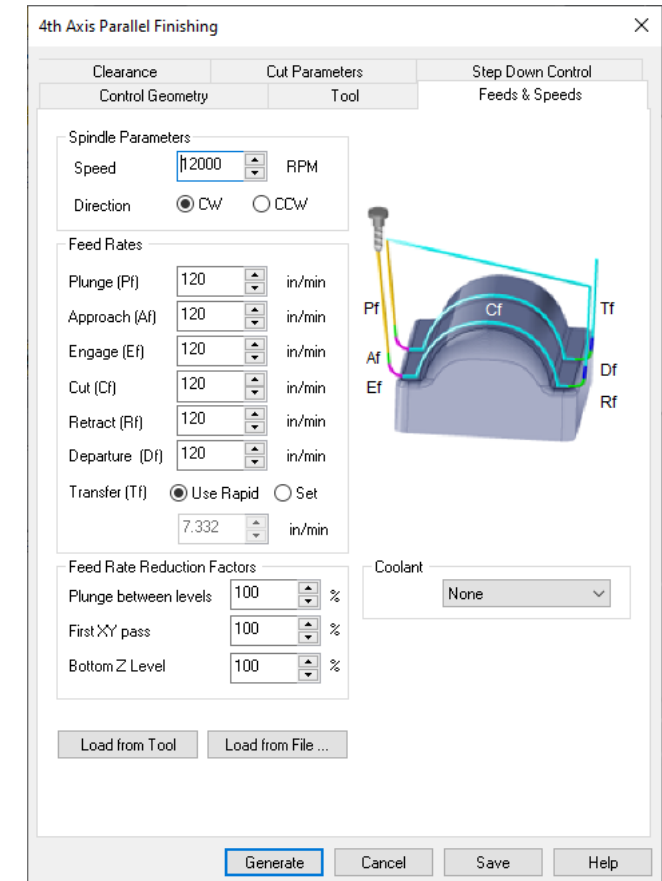
Tool

Choosing a small ball bit will provide a good amount of details.



Feeds & Speeds

This tab can be left as is. Again, note that the maximum spindle RPM will be **12000**.



4 Axis Parallel Finishing ctd.

Step Down Control

We will not be using **stepdowns** since we should have already roughed out the stock.

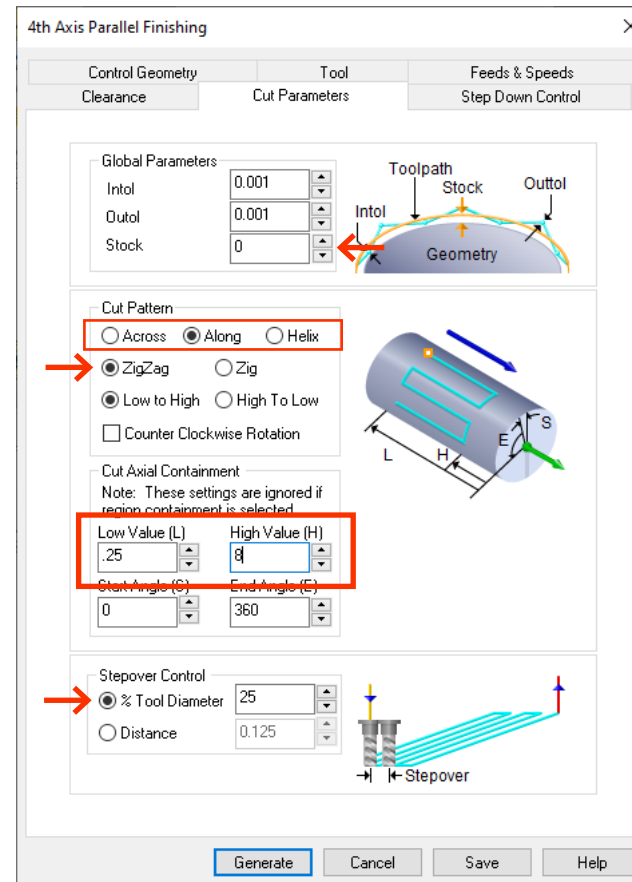
You can leave this tab as is.



Cut Parameters

You can choose your **cut direction** depending on the finish you want.

Cycle through them to learn what patterns they create, and choose the one appropriate for your model. In most cases, you can use **“Along”**.

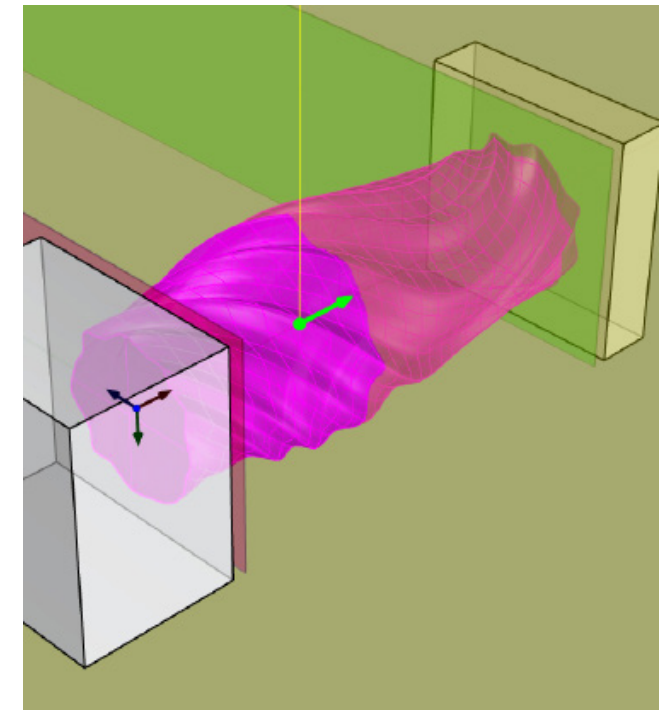


Cut Parameters ctd.

In Cut Axial Containment, set the **Low Value** to **0.25**, and **High Value** to **the distance between the two fixturing blocks minus 0.25**. In our case, we've set **High Value** to **8** because the distance between is **8.25**.

We don't want the tool to cut into our fixturing stock, so we add this offset towards both ends of our mode. You will see both a **red plane** and **green plane** appear on both sides of the model.

Set the **stepover** based on the quality of finish you are expecting. Set it to **25%** for a good balance between quality and time.

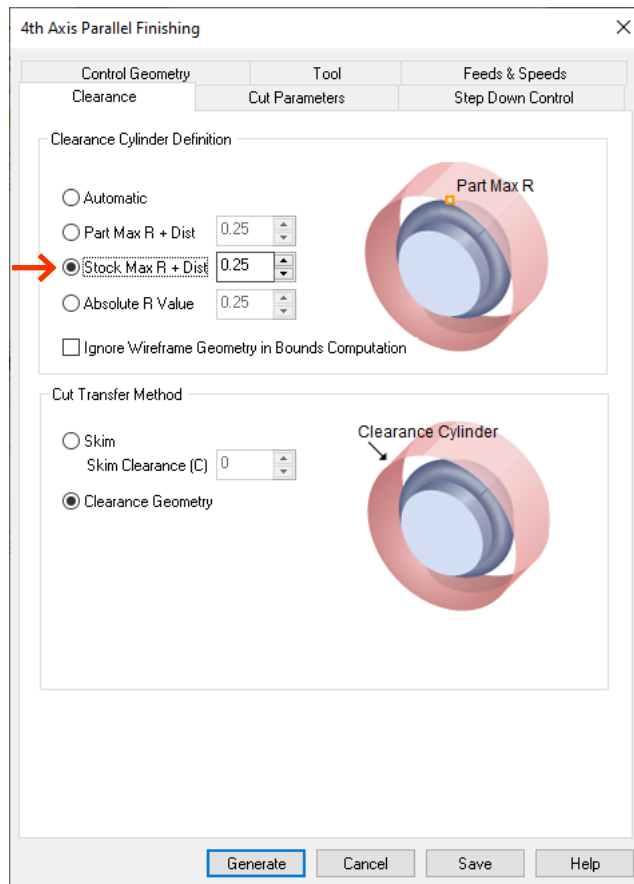


4th Axis Parallel Finishing ctd.

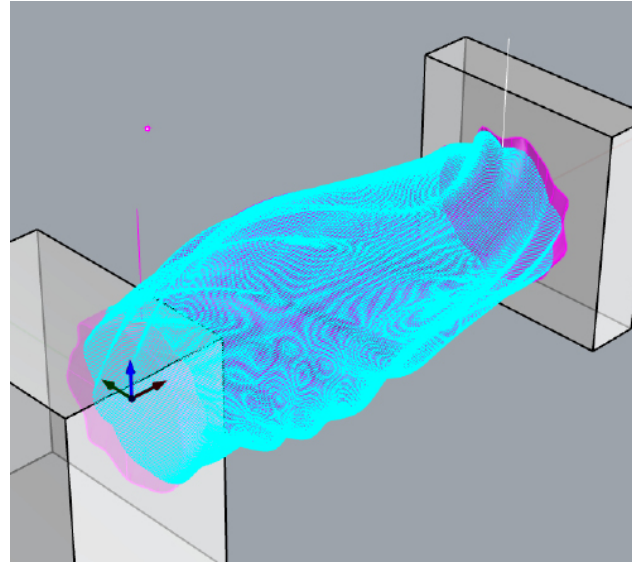
Clearance

Set this to **Stock Max R + 0.25"**. You will see a **red cylinder** appear around your model.

Click **"Generate"**.

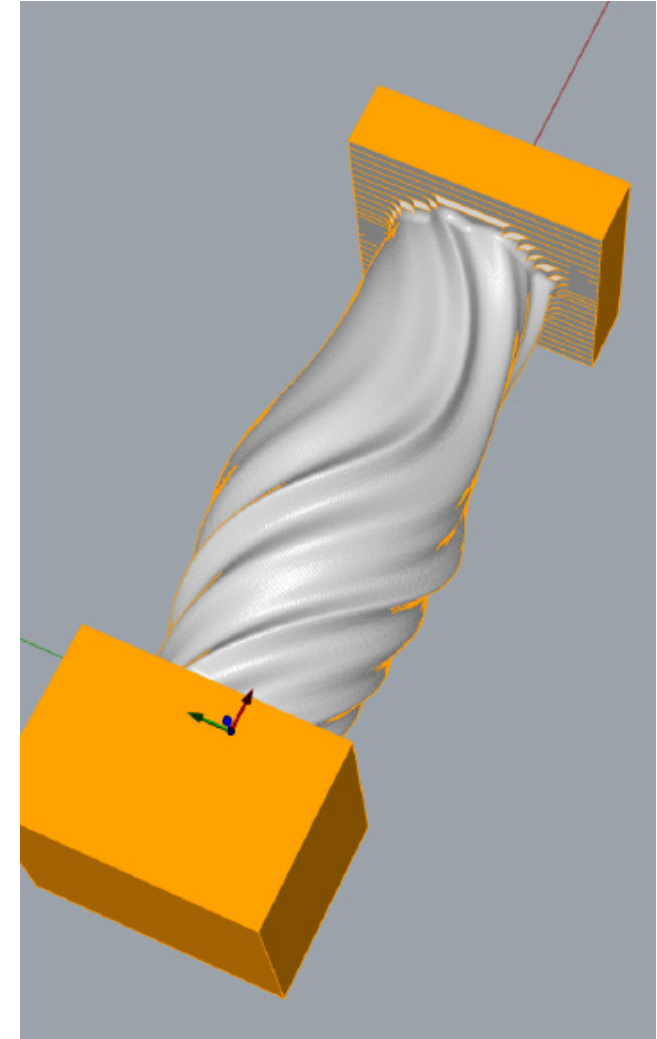


After generating the toolpath, you can preview it.



Notice how the toolpath does not extend all the way to the end of the model. This is because we set **Axial Containment** in the **Cut Parameters** tab.

You can now simulate the toolpath.



Note that **0.25"** of space at the ends of the model are not milled. You can factor this into your model by leaving a **buffer zone** when modeling.

Submitting

Estimating Machine Time	74	Skip to 0:21 ↗
Calculating Cost	74	Skip to 0:54 ↗
Submitting Process	74	Skip to 1:13 ↗

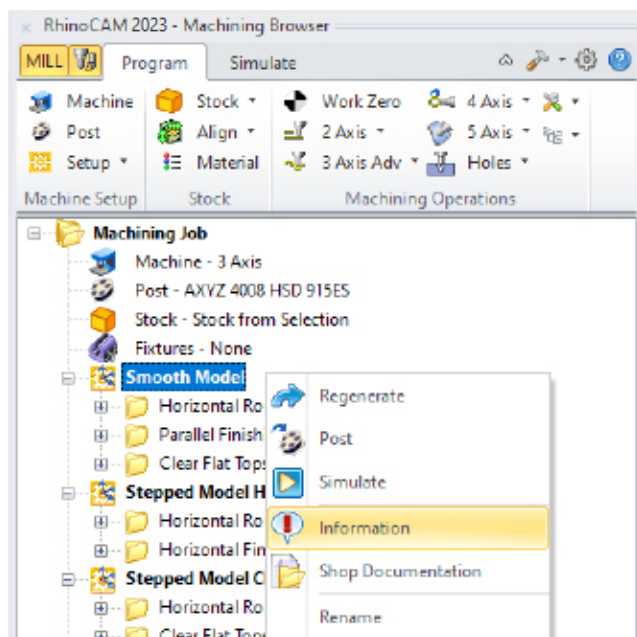
[video ↗](#)

Submitting a Project

Estimating Machine Time

Before submitting a project, run through a simulation of all toolpaths.

Then, estimate your milling time by right-clicking on a setup, and selecting **Information**.



In the **Machining Operations Information** window, you will see the **name of the operation**, **tool name**, **tool number**, **feedrate**, as well as the **estimated machining time**.

In reality, the machine time will be much higher. Add an extra **40% - 60%** to the machine time to make a more realistic estimation.

Name	Status	Tool	Tool #	Cut Feed	# of GOTOs	Machine Time
Smooth Model						
Horizontal Roughing	Clean	7 Long .5 Flat (MDF)	7	300.00 in/min	14859	50.48 min
Parallel Finishing	Clean	16 Long .5 Ball (MDF)	16	300.00 in/min	14963	4.12 min
Clear Flat Top w/ ...	Clean	7 Long .5 Flat (MDF)	7	300.00 in/min	277	1.10 min
					Setup total	55.70 min

In the case of this project, adding **60%** on top of the **55.70 min** gives us **89.12 minutes**, or approximately **1.5 hours**.

Calculating Cost

The CNC Lab charges **\$20 per hour**. From our last example, the estimated **1.5 hours** would cost us approximately **\$30**. You will be charged based on the actual milling time, including setup.

Payment for the CNC time is due immediately after the job has concluded, and must be paid through your **Daniels account**.

Submitting Process

Once you have simulated all toolpaths, and estimated the machine time, you are ready to submit.

1. Name your file in the following format with your name and date:
lastname_firstname_ddmmyyyy.3dm
2. Then, create a folder in the FTP server under **Groups > CNC Milling > Submissions** and name it "**lastname_firstname**"
3. Copy your **.3dm** file into this newly created folder.
4. Download and complete **this submission form** ↗
5. Email the above form to **cncmilling@daniels.utoronto.ca**

Once the file is reviewed, verified and approved by the CNC TAs, you will be provided with your scheduled milling time. We will try to schedule for your requested time, but please be amenable to other dates.

You are required to be present through your entire milling time.

Failure in following the above steps may result in your milling job not being processed.

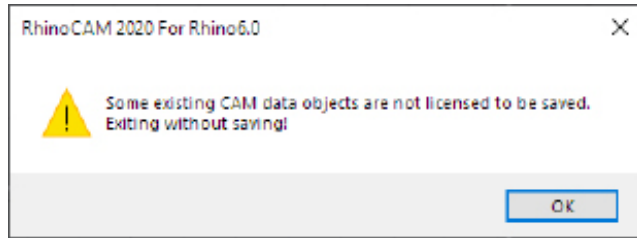
Troubleshooting

video ↗

Cannot save	76	Skip to 0:10 ↗
Nothing to cut	76	Skip to 0:50 ↗
No geometry found	76	Skip to 1:21 ↗
No active part	76	Skip to 1:42 ↗
Clearance plane too high	76	Skip to 1:56 ↗
Features not being milled	77	Skip to 2:49 ↗
Tool digging into stock unexpectedly	77	Skip to 3:23 ↗
Simulations not running as expected	77	Skip to 4:12 ↗
Simulations running too fast	77	Skip to 4:43 ↗
Cannot see tool when simulating	77	Skip to 4:54 ↗
Shank collision	77	Skip to 5:10 ↗

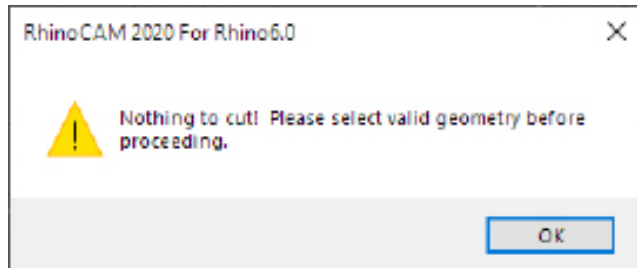
Common Issues

Cannot save



This occurs when you launch Rhino without being connected to the VPN first. Follow the steps outlined on [page 10](#), then save.

Nothing to cut



This error occurs when **generating/regenerating 3D toolpaths**. Ensure that your Rhino stock is hidden.

No geometry found



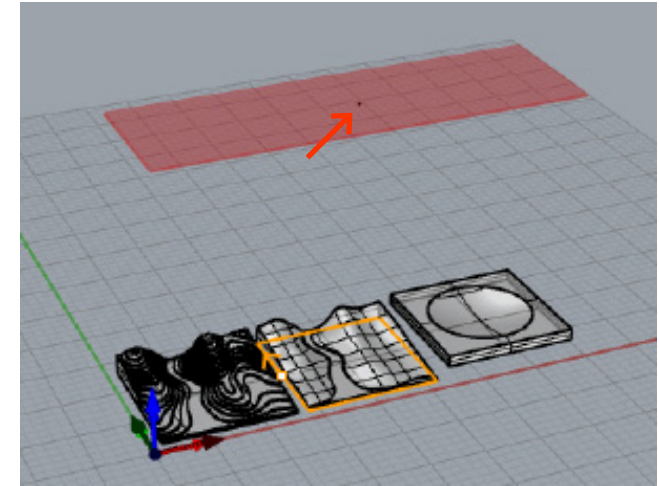
This may occur when **regenerating** an existing toolpath while its control geometry is hidden. Ensure the **control geometry** is visible, and is set correctly.

No active part



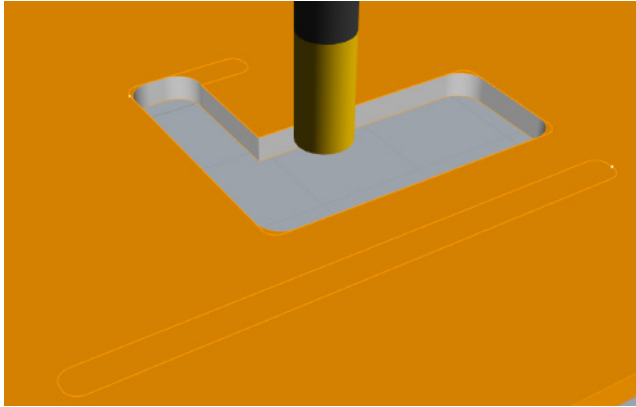
This may occur when trying to create a **3D toolpath** with no **3D geometry** visible in your file. Ensure your model is visible.

Clearance plane too high



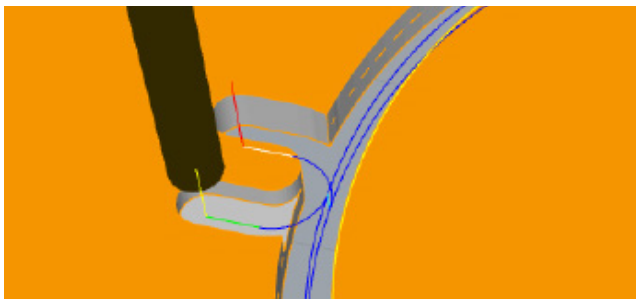
You can see the clearance plane while on the **Clearance Plane** tab of the toolpath setup window. Ensure there is no rogue geometry present in your Rhino file.

Features not being milled



Ensure that your tool diameter is small enough to fit in your control geometry. **Choose a smaller endmill** if possible.

Tool digging into stock outside of control geometry

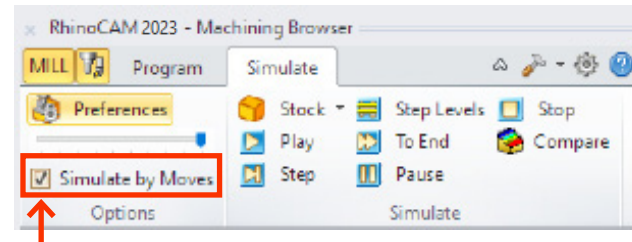


This occurs when your Entry/Exit parameters are not setup correctly. Set **Entry Motions** to **“Along Path”**, and **Exit motions** to **“None”**.

Simulations not running as expected

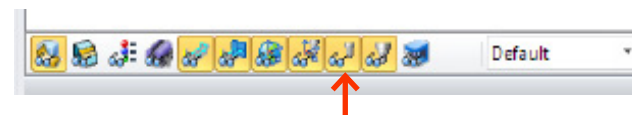
Sometimes, simulations may not update after you regenerate a toolpath. Try doing **“Simulate Until”** on the last toolpath. If this does not fix the simulation, go to **Preferences > Simulation**, switch the **Simulation Model**, click **“Apply”**, then switch it back. This will clear the simulation cache.

Simulations running too fast



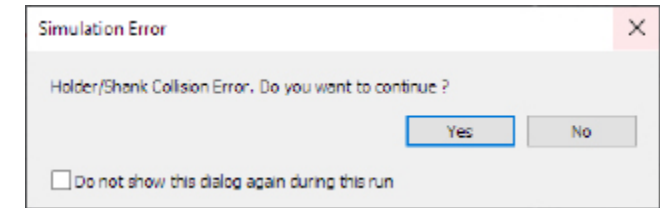
Deselect **Simulate by Moves**, and lower the slider.

Cannot see tool when simulating



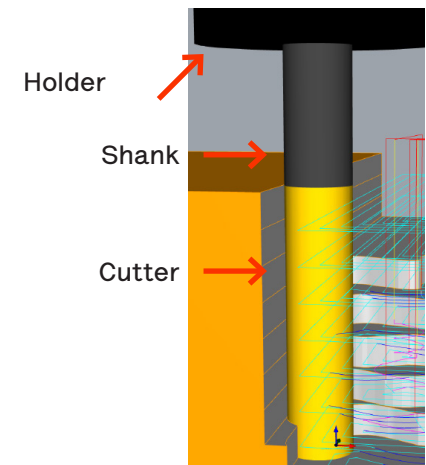
Ensure **tool visibility** is toggled on.

Shank collision



You may receive this error when the tool holder collides with your part. This mainly happens when the **tool is too short for the job**.

RhinoCAM does not differentiate between a **holder/shank collision**. **Shank collisions** may not be dangerous. Try setting the flute length equal to tool length in **Tool Settings** to eliminate the shank. Then try running the simulation again.



If you encounter the message again after changing your flute length, **use a longer tool**.

End of Manual

[Go to Contents](#)