Rhino 6

RhinoCAM® MecSoft

AXYZ 4008 + Roland MDX 540

CONTENTS

BASICS

BEFORE YOU BEGIN

To download RhinoCAM:
https://www.daniels.utoronto.ca/info/current-students/undergraduate/it-support/your-laptop-software

The license is not working if you are on FreeMILL and you will not be able to save your CAM data.

It can take up to 1 week during busy time to mill your file after you have submitted. PLAN AHEAD!

The millpaths are based off your geometry, having clean geometry is critical to creating clean toolpaths.

Basics of Setting up a RhinoCAM

1 **Set your Stock** - What material and size are your 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 either use curves to create you millpaths or use curves to outline a surface you want to mill.

3 **Pick your tool(s)** - You can use this guide as a base to pick a tool, but it is dependent on the tools available and will be finalized by the TA running your file.

4 **Cut Depth/Cut Parameters** - This is were you can customize your cuts. This will be based on the material being used, the desired finished and time.

5 **Set you Feeds and Speeds** - Depending on your material and what operation you are doing you will set your feeds and speeds accord to the chart in this guide.

6 **Simulate** - You can simulate exactly why the CNC will do and see the final result of your mill. You can use this tool to see if you machining operations are doing exactly what you need them to.

Frequently Asked Questions

Why did I get the pop up saying units are not supported? RhinoCAM will only work if your Rhino file is in inches.

Can someone look over my file before I submit? TAs work every weeknight from 6-10 to run files and help set up a files, you can e-mail CNCmilling@daniels.utoronto.ca to set up a time to get feedback on your file before you submit.

When will I be scheduled to mill? Once you have uploaded your file to the FTP and e-mailed CNCmilling@daniels.utoronto.ca with your submission form a TA will book you into the next available time spot that works with your schedule.

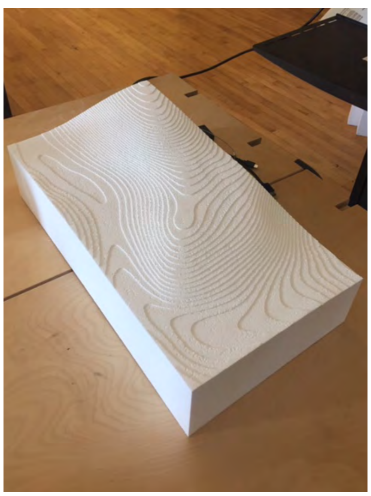
How long will my mill take? Once everything is set up in your fill including feeds and speeds you can right click on your setup and click information to get a time estimation for your mill. It often takes a bit longer than the estimation, plan for up to 4 hours for setup, running the mill and clean up for a typical mill.

TYPICAL OPERATIONS

There are many more operations than these that create different toolpaths these are just the most typical operations, consult a TA on if it's possible to create a specific toolpaths.

Horizontal Roughing + Parallel Finishing

95% of mills are these two operations. It is used for 3D surfaces. It quickly removes excess materials with a large flat tool and then zig-zags back and forth with a smaller end mill for the final finished surface.



Horizontal Roughing + Clear Flats

This is often used to create stepped contour models. If you already have a stepped contour model in rhino you can use Horizontal roughing to remove a lot of the material with a large flat tool then Clear Flats finished the surface.

Profiling

Is used for cutting pieces out of a stock.

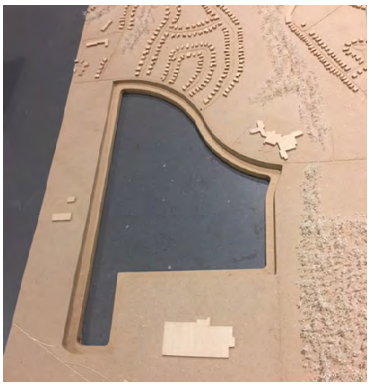
Profiling can also be used to clean up hard edges after a parallel finish which can leave scallops on the edge which it zig-zag across. This only works with flat surfaces, angled surfaces engraving needs to be used for clean up.

TYPICAL OPERATIONS

Engraving

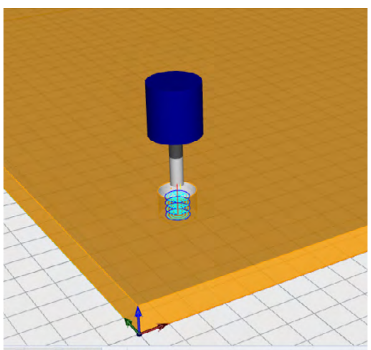
Is often used to create engraves into the stock, this could be for roads or paths on a site model.

Engraving can also be used to clean up hard edges after a parallel finish which can leave scallops on the edge which it zig-zag across.



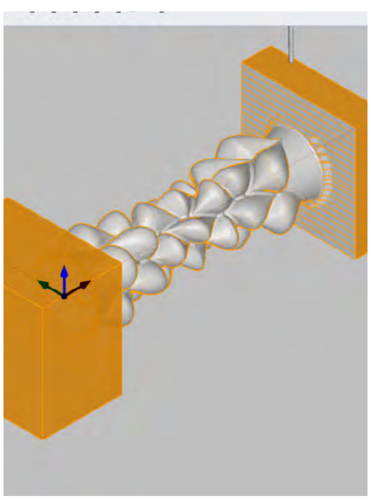
Pocketing

Is often used to create inserts within a model, this could be used for a site models that require a plug.



Hole Pocketing

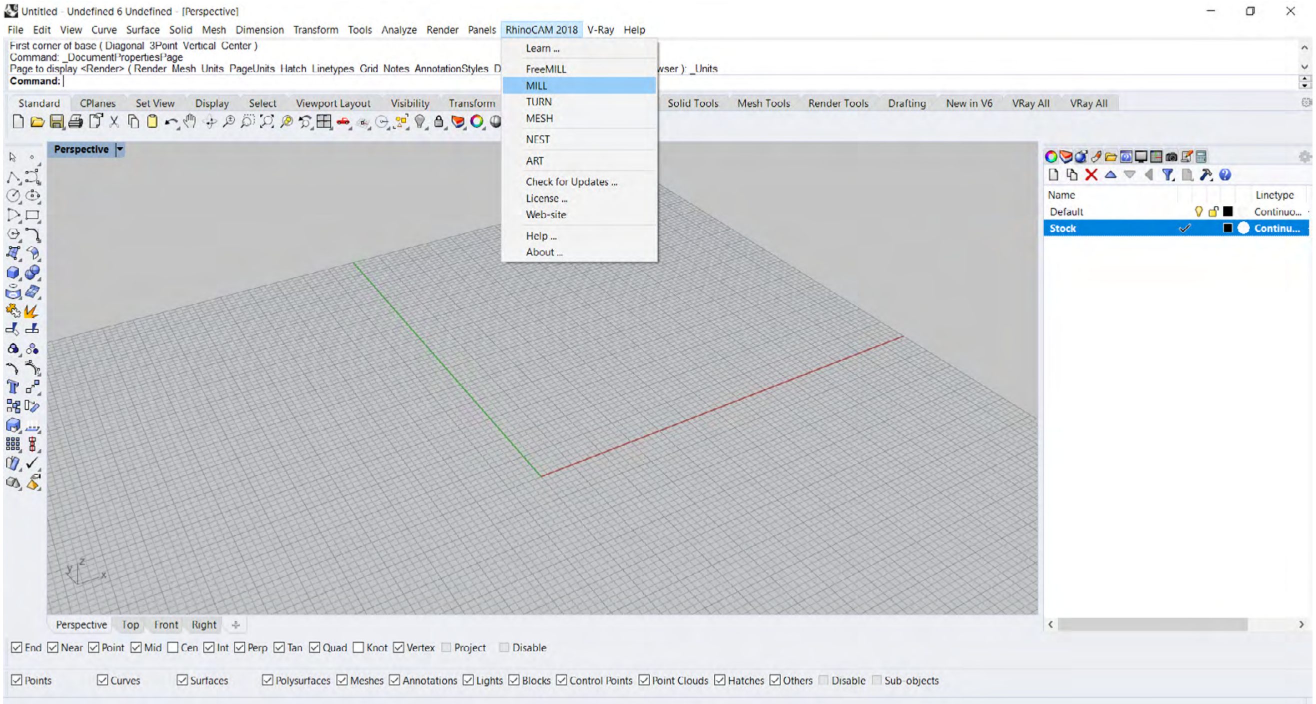
Is used to create perfectly circular holes, which could be used for inserting a dowel or screw holes into your mill.



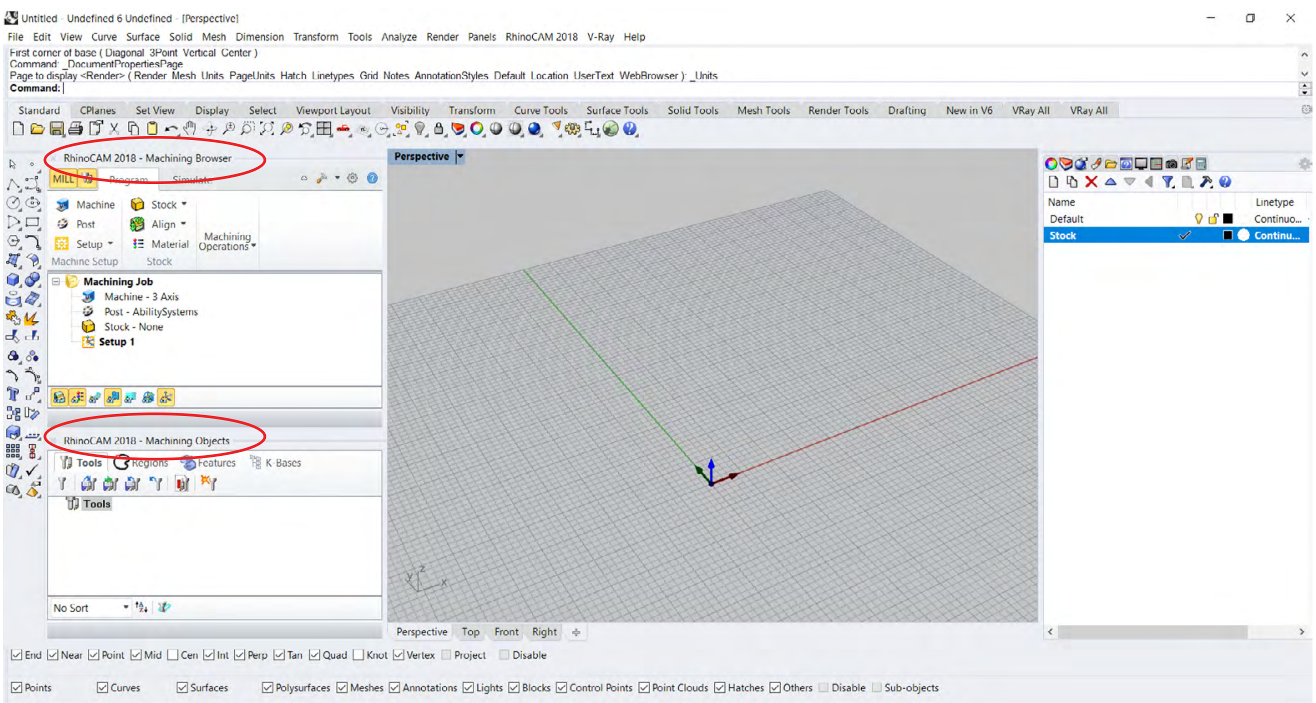
4 Axis Milling

Is used to create mills that would be impossible with 3 Axis milling, as it can rotate the lathe as it mills to mill all around the object.

LOADING RHINOCAM



To operate RhinoCAM launch the Machining Browser Tab and the Machining Objects Tab. These can both be found in the RhinoCAM Drop down menu located along the top of the Rhino menu. Alternatively you can also type RhinoCAM into the command bar.



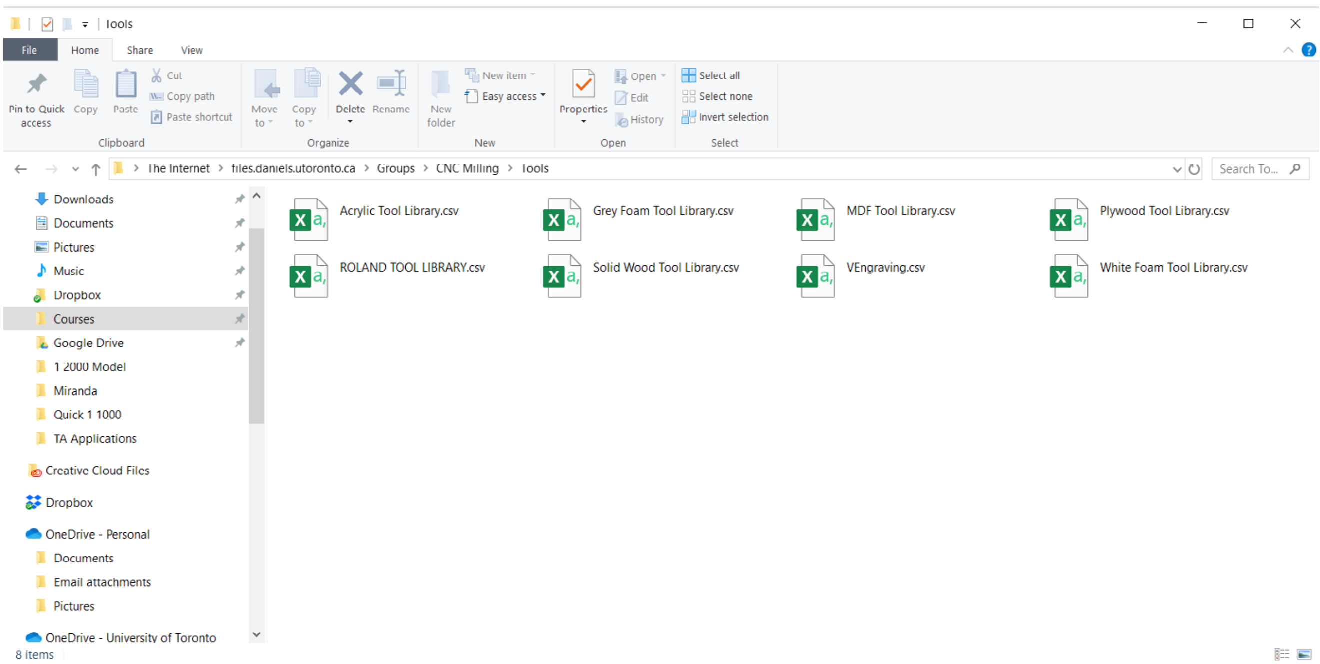
Once launched you can dock the two menus on the left side of the screen, both menus are necessary for setting up and running milling operations.

Your units must be in inches for the program to launch.

LOADING TOOL LIBRARIES

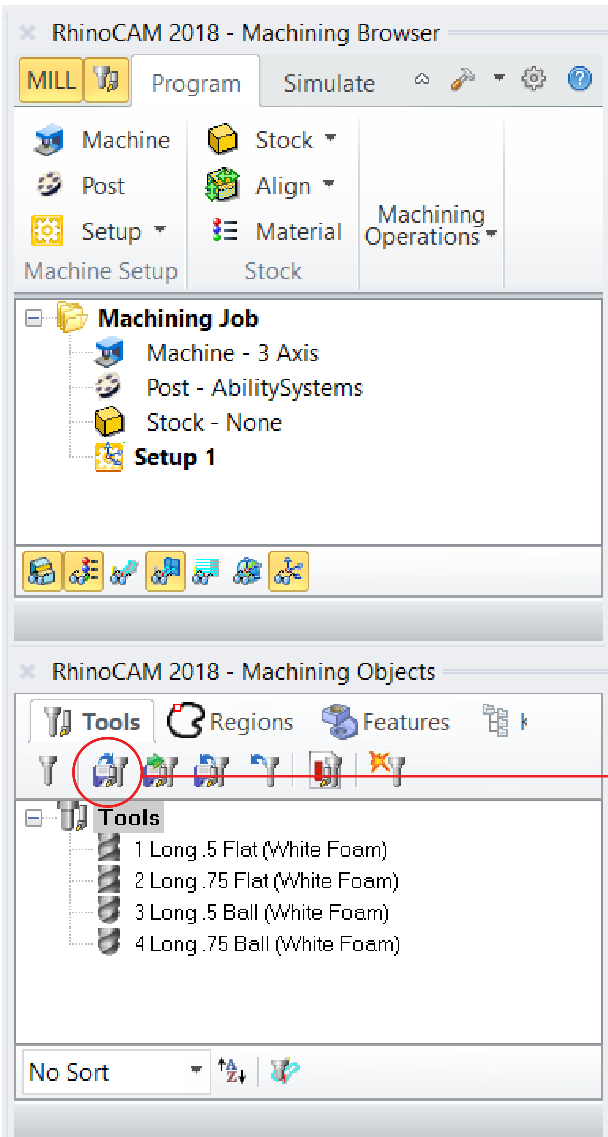
To find the Daniels Tool Libraries first go to:
ftp://files.daniels.utoronto.ca/Groups/CNC_Milling/Tools/

Download these CSV files to your computer directly so you can reference them in RhinoCAM later.



The Tool Libraries have been customized to each material, and are based off of commonly used tools found in the Lab. Later in the tutorial you can find how to create and edit your own end mills for programming if your job has unique finish requirements.

LOADING TOOL LIBRARIES



To load the tool library go to the Machining Objects Menu while under the tools tab click the load tool library button.

Only load the tools specific to your material.

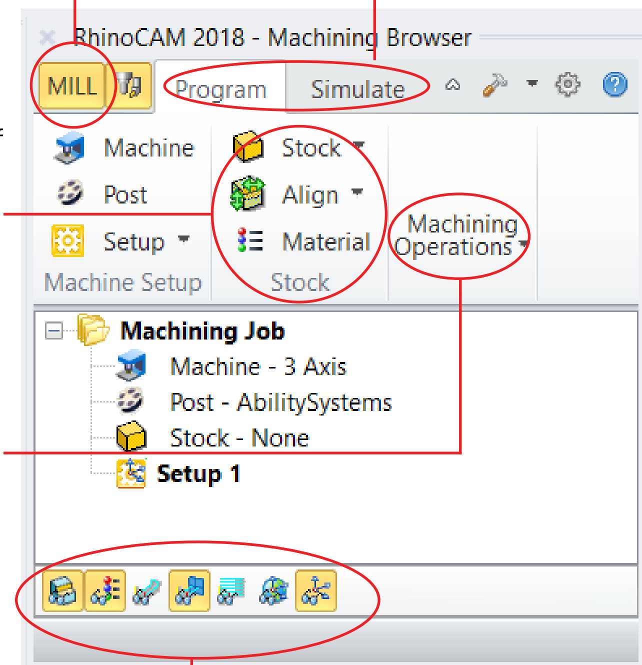
Once loaded the library will display all of the common tools used for your choice of material.

INTERFACE AND LAYOUT

The Machining Operations menu contains two tabs, program and simulate.

The Program tab and associated functions are used to set up your milling operations. The Simulate Tab allows for simulation and analysis of the milling paths and order.

On the far left hand side of the display tabs, **make sure MILL is displayed and highlighted**. Clicking here will toggle between MILL and TURN. TURN is not a function available on the labs CNC Router.

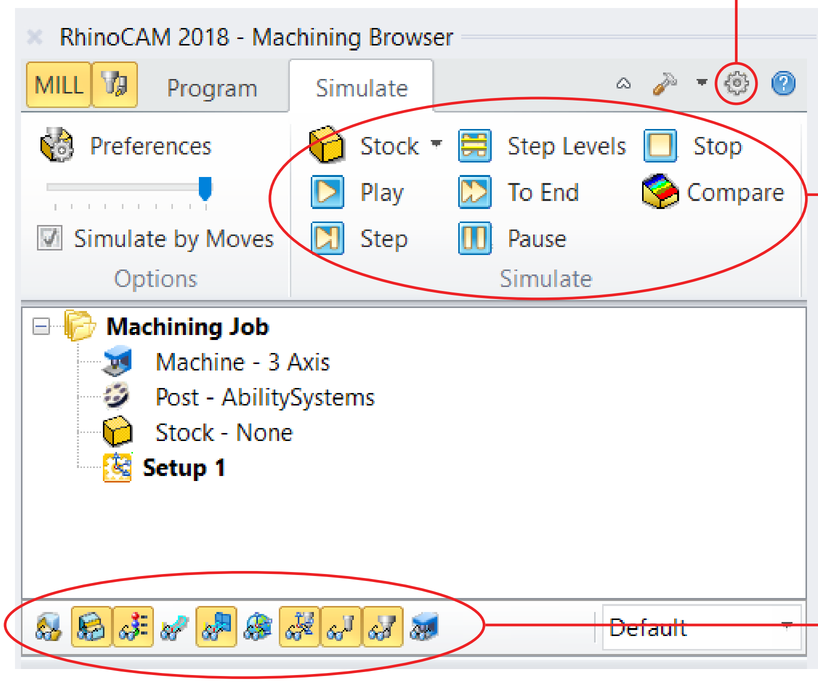


The stock align and material menu allow for the creation of stock, alignment to the working surface and the material texture of the stock. The stock drop down menu and creation of stock is covered on page __ along with setting material texture.

The machining operations contain 2 axis, 3 axis adv and 4 axis cutting operations, the drop down menus contain all necessary functions to create milling paths for 2d and 3d cutting.

The display buttons are located in the bottom left of the machining operations menu. From left to right they display the stock visibility, material texture visibility, toolpath visibility, toolpath level visibility and Machine CSYS visibility.

INTERFACE AND LAYOUT

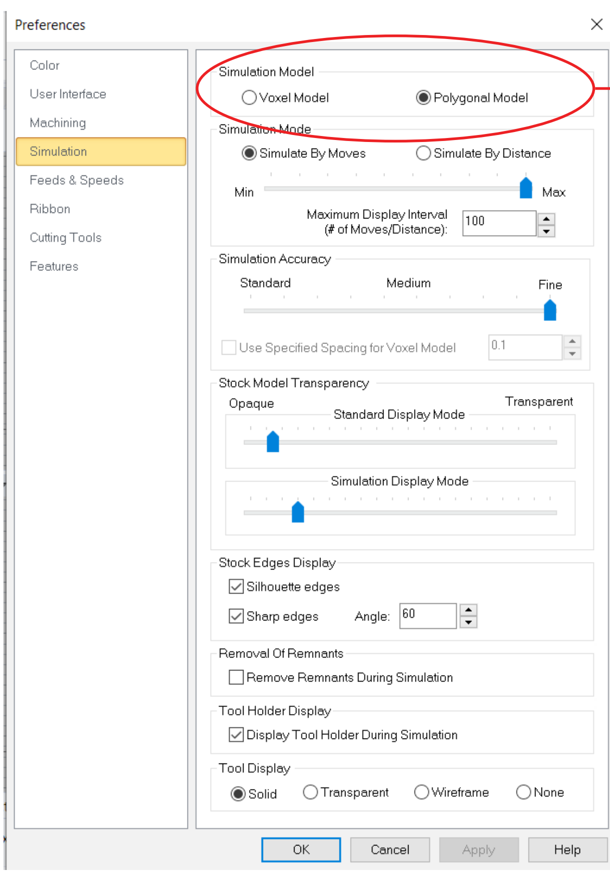


Preferences Button

The Simulate Menu allows simulation and playback of the established milling operations. You can process by playing the Milling operations or stepping through the set incrementally with the step button. Pausing will allow you to fast forward to the end with the 'To End' button.

The compare button lets you compare your input geometry to your milled geometry.

The visibility displays are located on the bottom left of the Simulation menu. The buttons allow to toggle on part, stock, material, toolpaths, Machine CSYS, Tool, Holder and Machine Visibility.

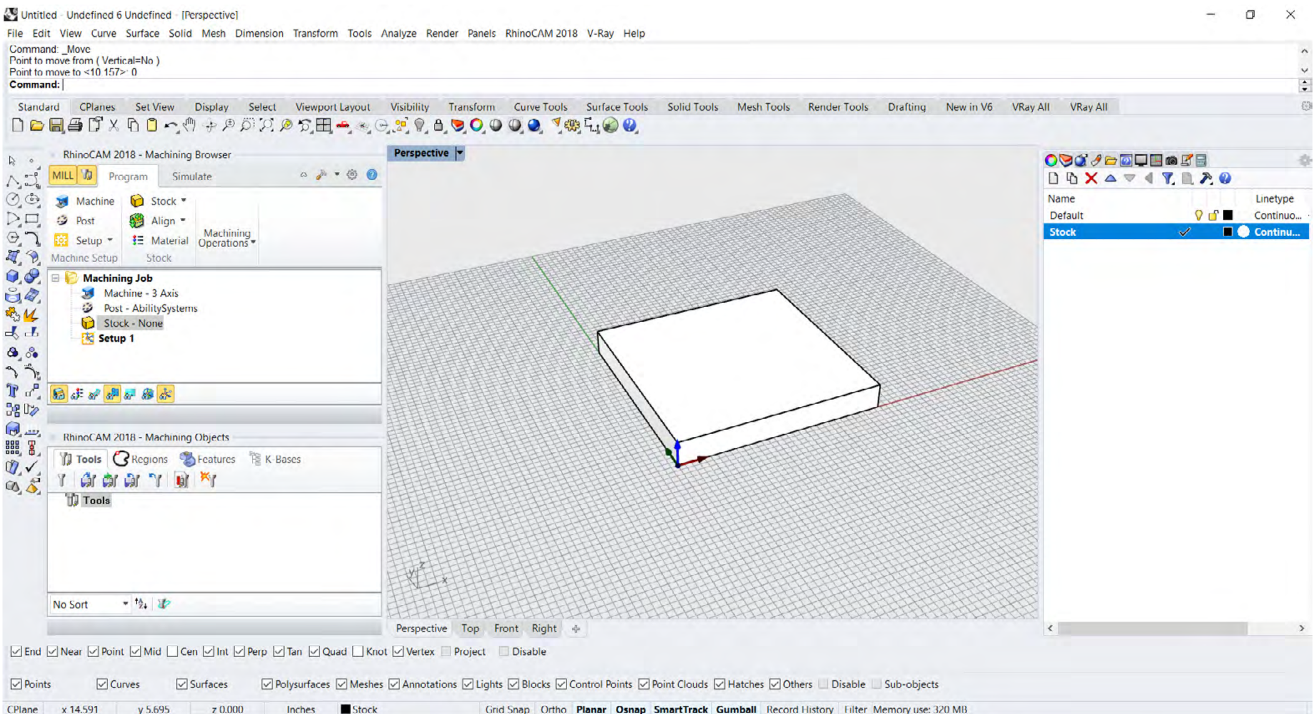


The simulation Preferences can be changed with the preferences button. The simulation model can be switched from a voxel model to a polygonal model. This will display greater simulation accuracy but also will slow down the simulation.

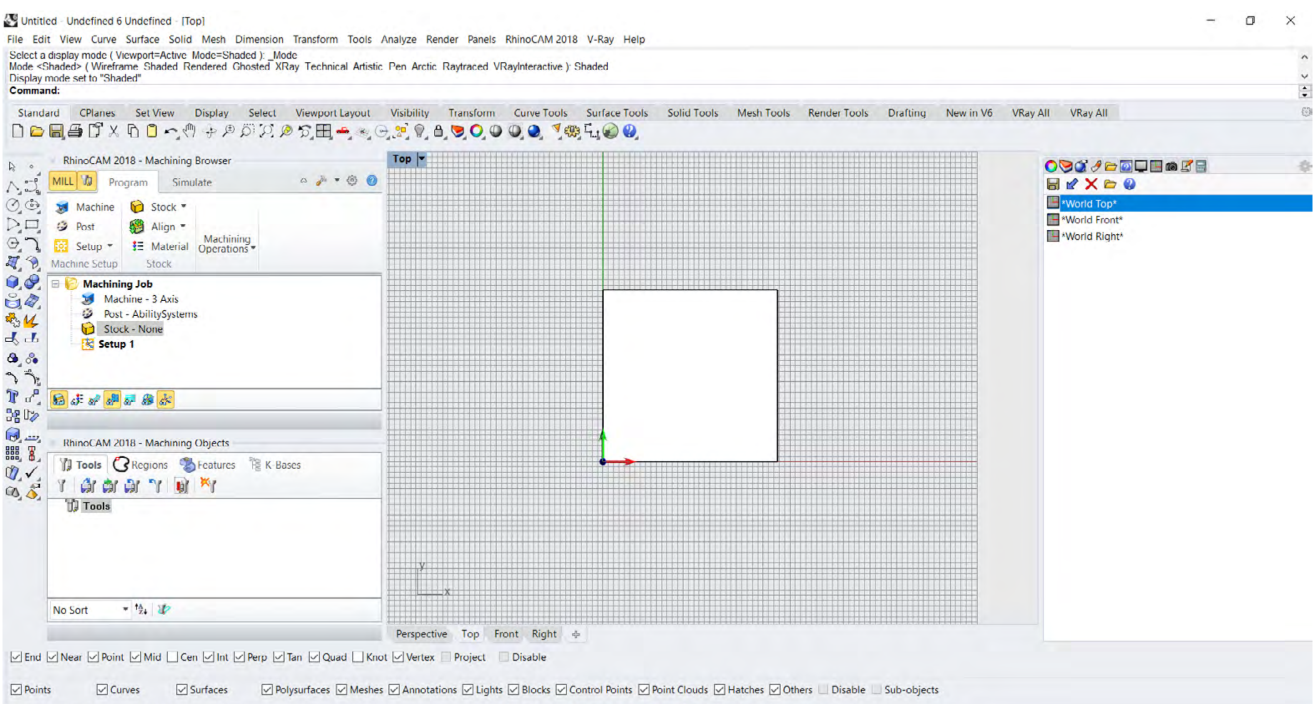
The simulation accuracy can be shifted between standard and fine, this will create a greater resolution on the simulated milling paths. On a large model with multiple complex milling paths switching to a voxel model with standard accuracy and reducing the display intervals will speed up the simulation, greater accuracy will create slower simulation.

STOCK SET UP

Creating stock in RhinoCAM is done by creating a solid poly object (usually a box) to match the physical dimensions of the real stock material to be milled.

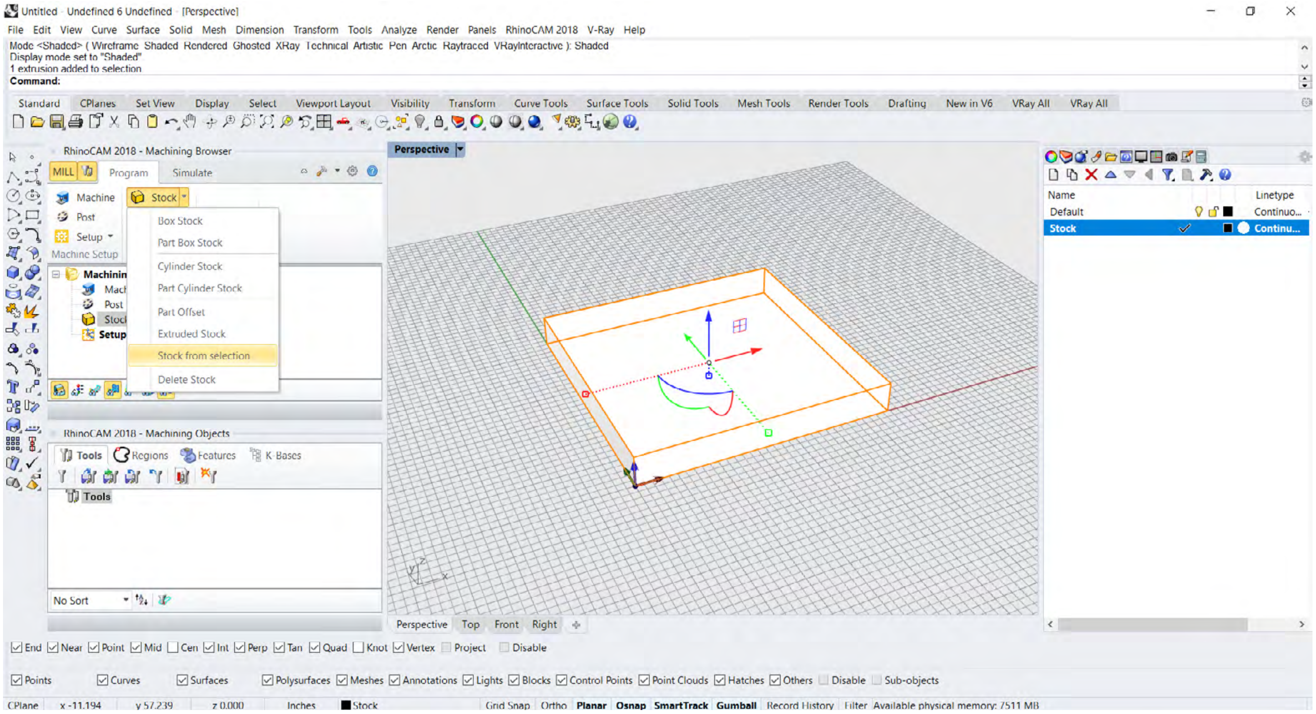


Create a stock layer, then create the stock size required to match the physical material stock in Rhino.

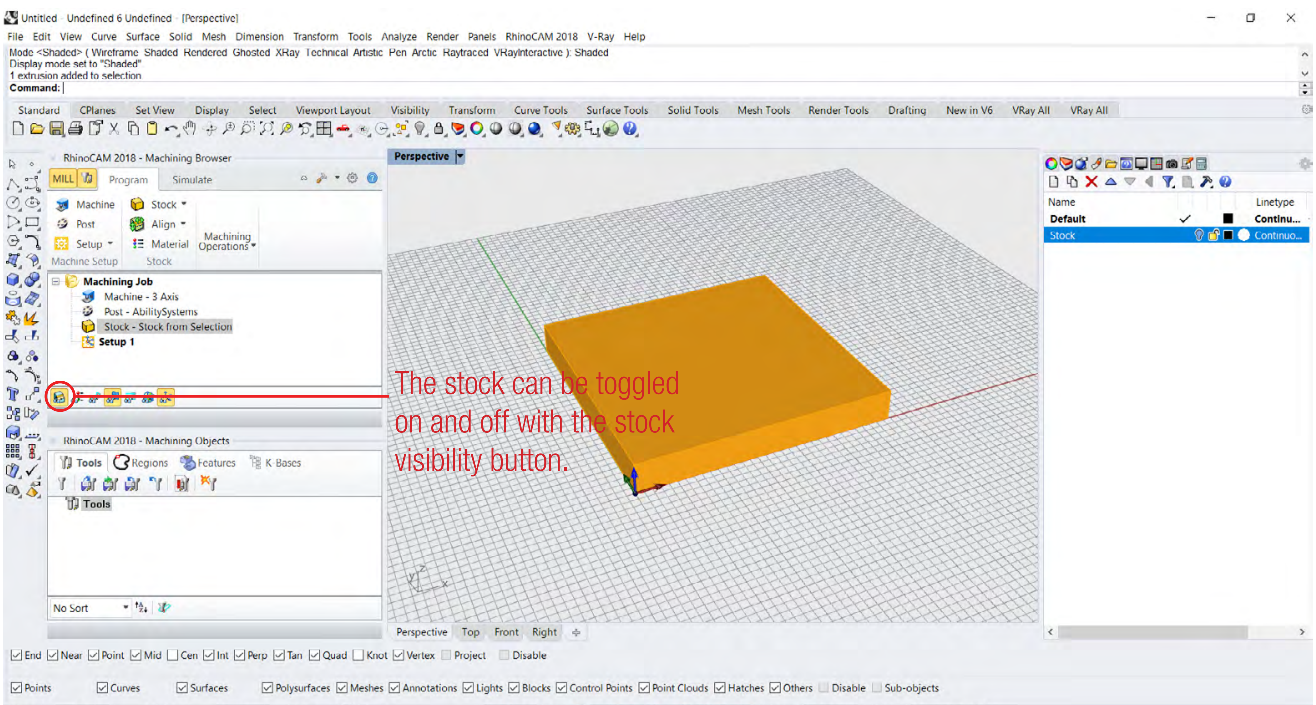


Ensuring that your C Plane is at *World Top* Move the bottom left hand corner stock to the origin in rhino (0,0,0) this corresponds to the origin of the CNC. The Red, Green and Blue arrows indicate the X,Y and Z direction of the CNC bed.

STOCK SET UP



Highlight the created stock, from the machining operations menu select the stock drop down menu, then select stock from selection. This will create a stock from the object.



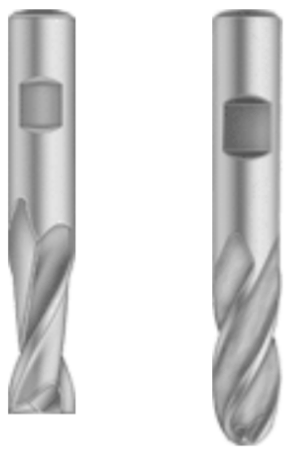
Once the stock has been created you can turn off the stock layer, the ghosted stock you see in the program menu is the RhinoCAM stock. All milling paths and operations will be applied to this stock.

SETTING UP A MILLING FILE

To begin setting up your RhinoCAM file have either your 2D geometry (curves) or 3D geometry (Surfaces or Meshes) ready and exported into a new rhino file. Remove any unnecessary geometry from the milling file, only maintain necessary curves and surfaces for the milling operations. Any unnecessary geometry can easily be selected and lead to errors in the assignment of milling procedures and operations.

Best practice is to have your curves on layers with each machining operation you want to do.

Establish the size of your stock and model, make sure there both the material size available for your mill and the end mill lengths and cutting depths. The smaller the end mill diameter the shorter the cutting depth. Always make sure you can cut to the desired depth with the selected end mill.



Tools come in different lengths and diameters, the Daniels tool library (covered previously in how to load the tool library) has a range of end mills from 1/4” to 3/4” in flat end mills and ball end mills. The end mills have a overall length, cutting depth and both shank diameter and mill diameter.

Different end mill sizes and types will leave different tooling effects on the material surface. RhinoCAM will simulate these paths and effects based on selection of end mill and tooling paths.

Once you have your 2D geometry ready for 2 axis milling, see the section on 2 Axis Machining Operations.

If you have 3D surfacing see the section covering 3 Axis Machining Operations.

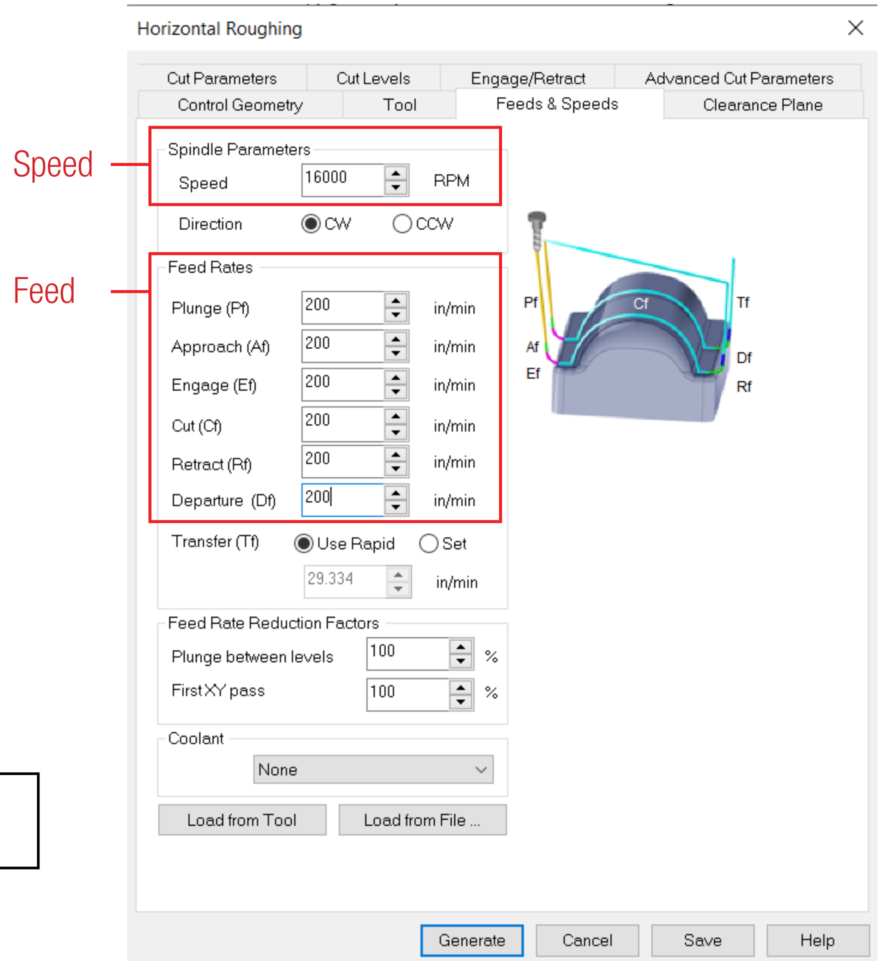
FEEDS AND SPEEDS

For every operation the feeds and 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 and Speeds tab of each operation and example of this is to the right.

A TA will finalize your Speeds and Feeds.



Material	Operation	Speed	Feed	Suggested Tool	% Step Down of the Tool Diameter
White Foam	Roughing	16,0000	200 in/min	0.5" - 0.75" Round or Flat	100%
	Finishing		100 in/min	0.25" - 0.75" Round (or Flat)	100%
	Profiling		25 in/min	0.25" - 0.75" Flat	100%
High Density Foam	Roughing	16,0000	500 in/min	0.5" - 0.75" Round or Flat	100%
	Finishing		300 in/min	0.25" - 0.75" Round (or Flat)	100%
	Profiling		200 in/min	0.25" - 0.75" Flat	100%

FEEDS AND SPEEDS

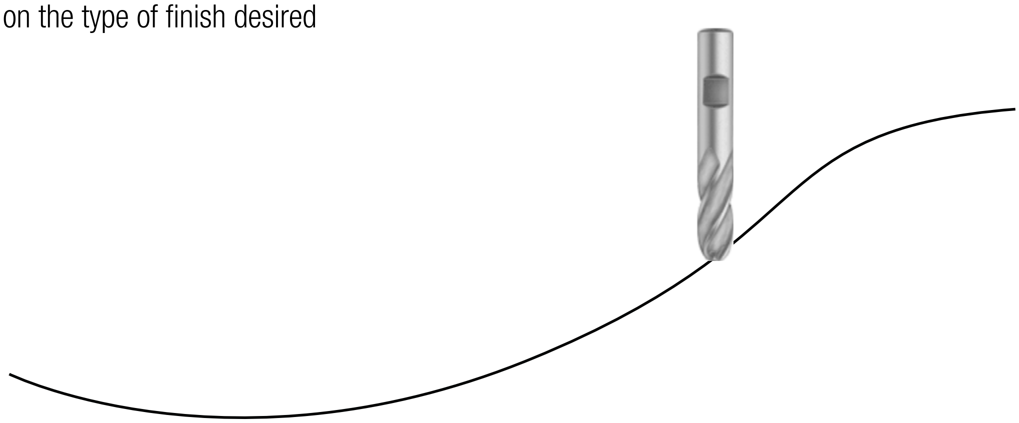
		Speed (RPM)	Feed	Suggested Tool	% Step Down of the Tool Diameter
Solid Soft Wood	Roughing	16,0000	250 in/min	0.5" - 0.75" Round or Flat	50%
	Finishing		150 in/min	0.25" - 0.75" Round (or Flat)	25%
	Profiling		125 in/min	0.25" - 0.75" Flat	50%
Solid Hard Wood	Roughing	18,0000	120 in/min	0.5" - 0.75" Round or Flat	50%
	Finishing		150 in/min	0.25" - 0.75" Round (or Flat)	25%
	Profiling		100 in/min	0.25" - 0.75" Flat	50%
MDF	Roughing	16,0000	300 in/min	0.5" - 0.75" Round or Flat	75%
	Finishing		300 in/min	0.25" - 0.75" Round (or Flat)	25%
	Profiling		120 in/min	0.25" - 0.75" Flat	50%
Plywood	Roughing	16,0000	150 in/min	0.5" - 0.75" Round or Flat	50%
	Finishing		175 in/min	0.25" - 0.75" Round (or Flat)	25%
	Profiling		125 in/min	0.25" - 0.75" Flat	50%

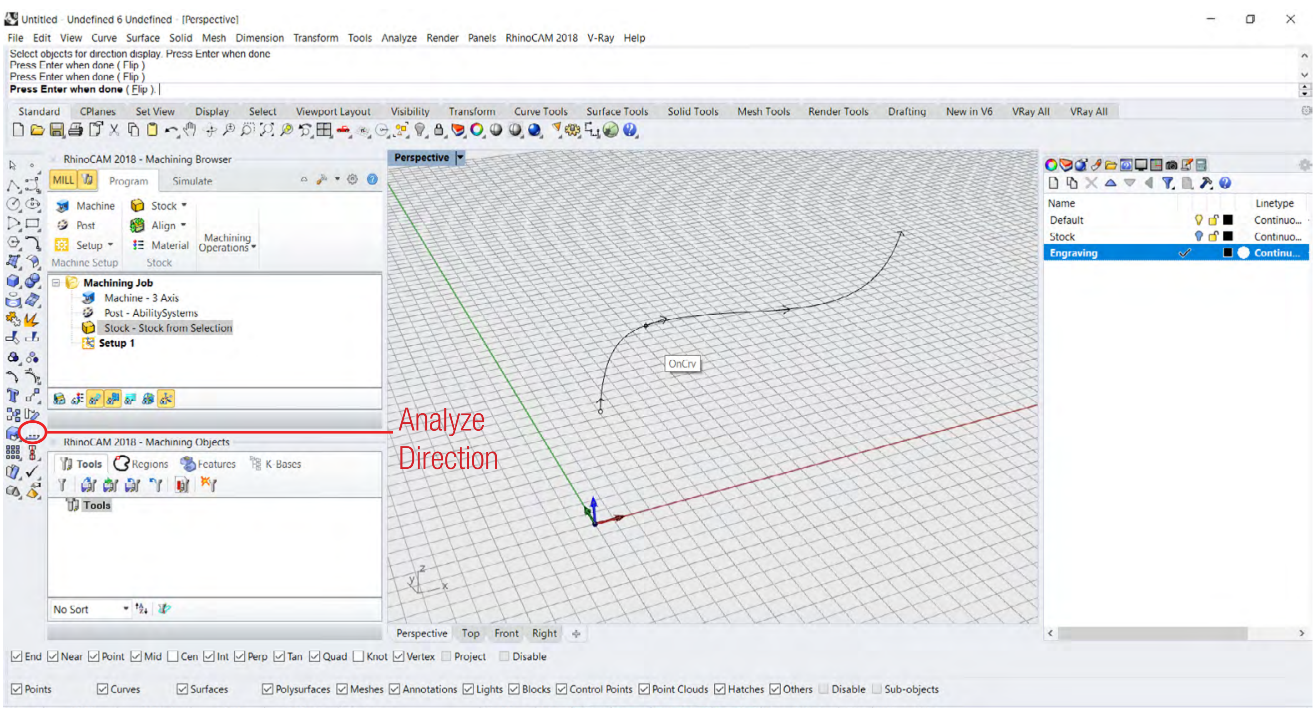
2 Axis
Machining Operations

ENGRAVING

Engraving Milling paths follow curves along the center of the end mill to the assigned curve or line. The end mill cutting direction will follow the direction of the curve or line, to find the direction of the curve in Rhino you can enter the command _Dir or press the analyze direction button.

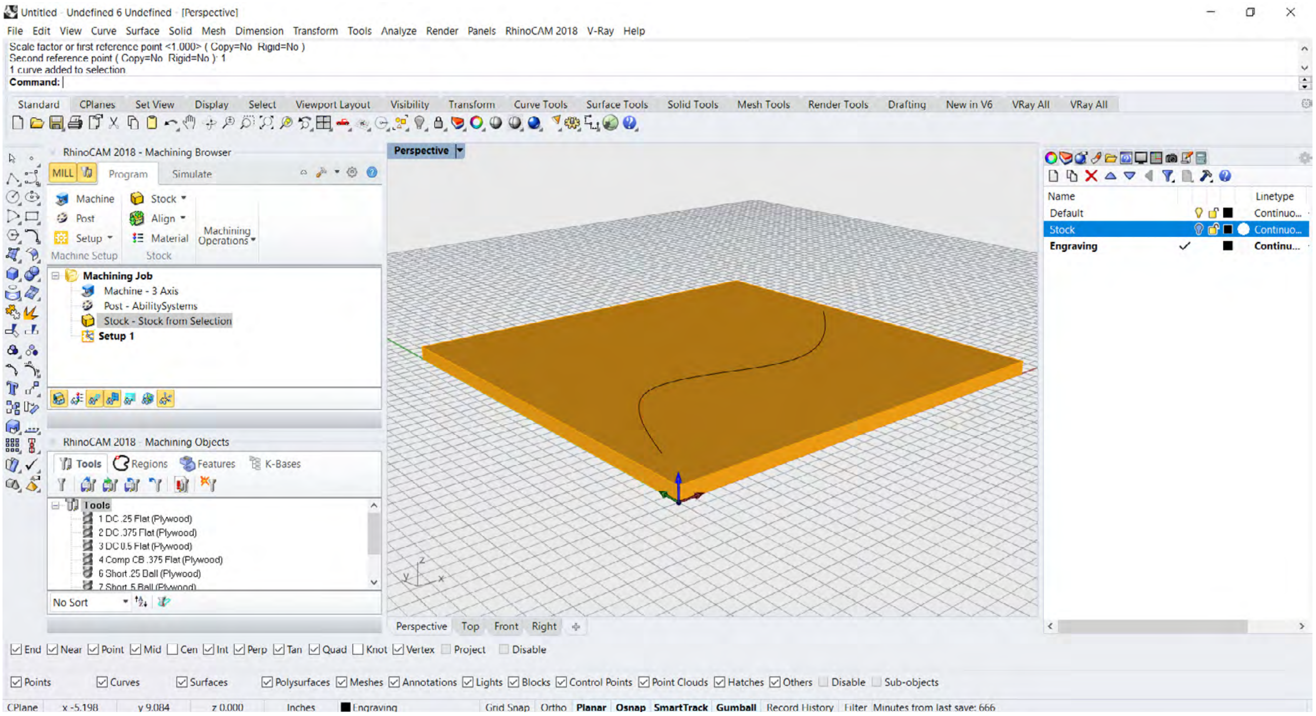
Round or Flat tools can be used on engraving depending on the type of finish desired



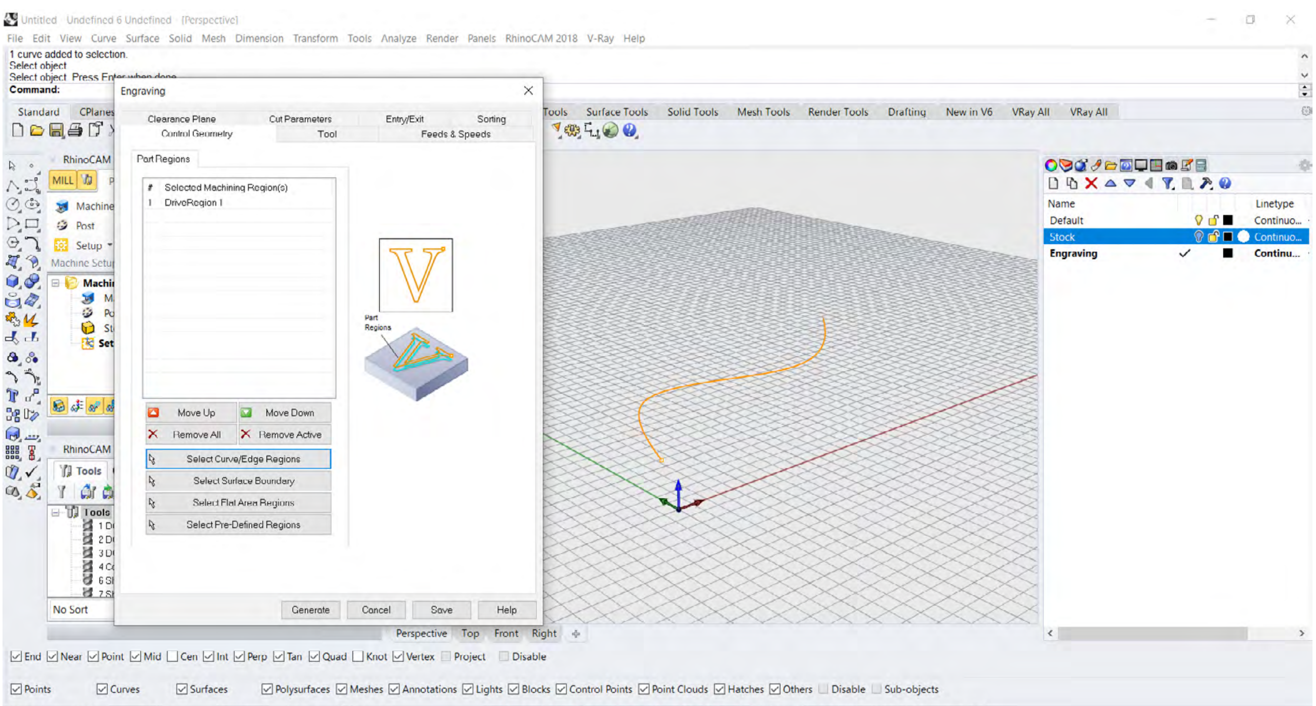


The direction of the curve can be flipped while in the Analyze direction command. These curvature directions will be important in assigning 2D milling operations as the line/ curve direction will dictate the cutting direction in your milling operations.

ENGRAVING

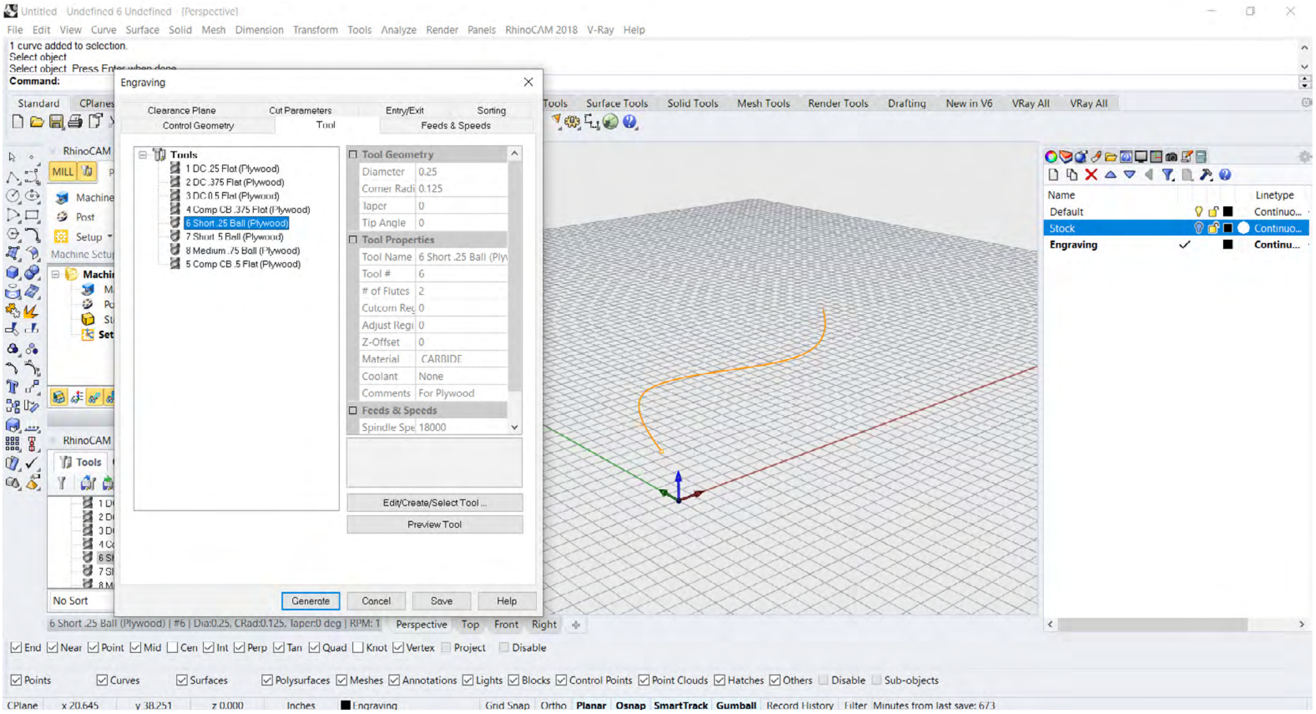


The engraving curve is aligned to the top surface of the stock material. In the cut parameters tab in the machining operations for engraving you can set the cut depth.

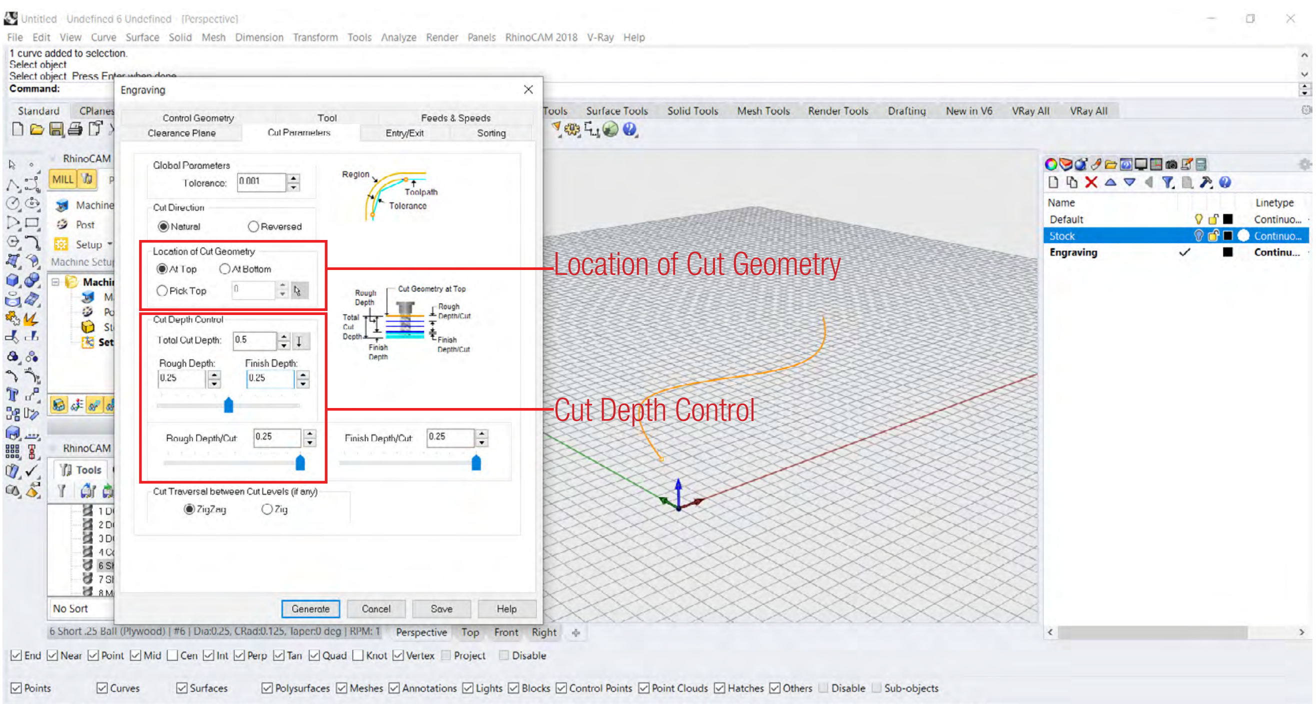


Select Engraving from the 2D milling operations tab, press Select Curves/Edge Regions and select the curves or regions you want to engrave to create your milling path(s).

ENGRAVING

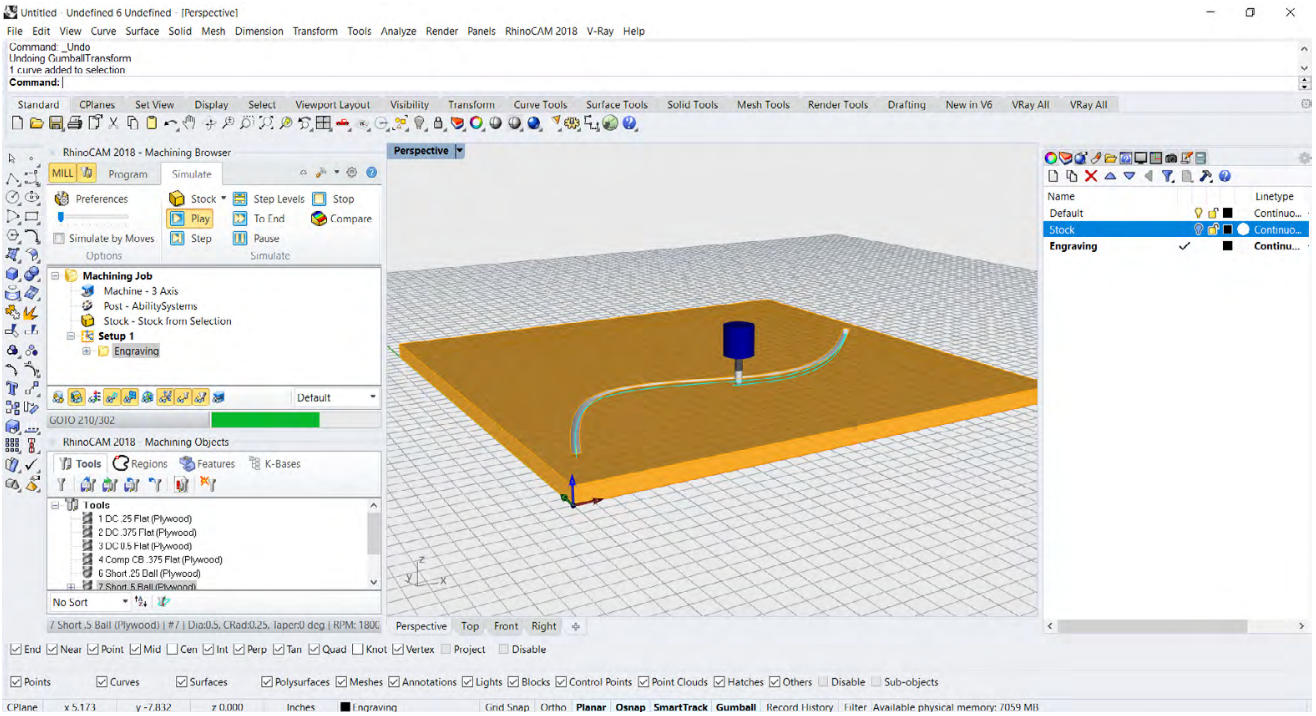


The tool diameter selected will determine the width of the engraving line, as the tool center point follows the selected engraving line the tooling path with be the tool diameter from the center of the line. A flat or round tool can be used depending on the desired finish.



Under the Cut Parameters tab, the cutting direction, depth and geometry location is chosen. The geometry currently in the example is located at the top and the cut depth is 0.5 inch, the rule is never cut deeper than half the diameter of the tool, so since we are using a 0.5 inch bit, we set the rough depth to 0.25 so the cut is done in two passes. For less dense materials ie. foam we can cut 100% of the tool.

ENGRAVING



Once the milling paths have been generated select the simulate tab and run the engraving paths to see the toolpath result.

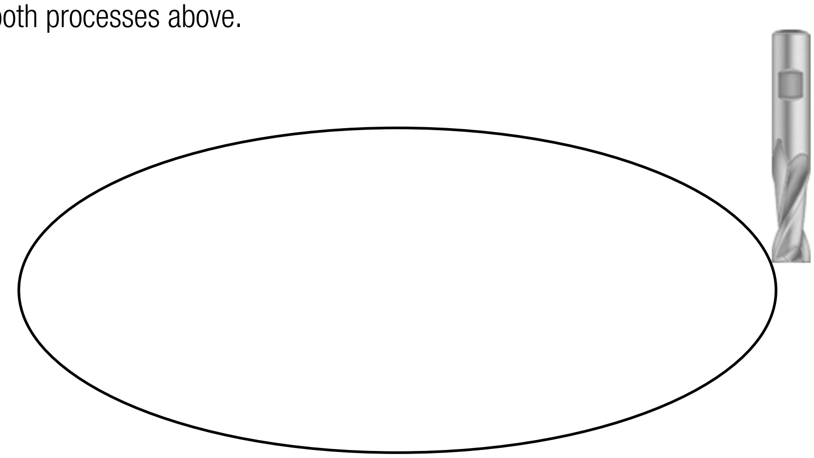
While this example uses a curve that is flat, and engraving is in 2 Axis milling you can also use curves that changes in elevation. Engraving can be used to clean up edges of parallel finishes that change in elevation in more advanced setup of CAM files.

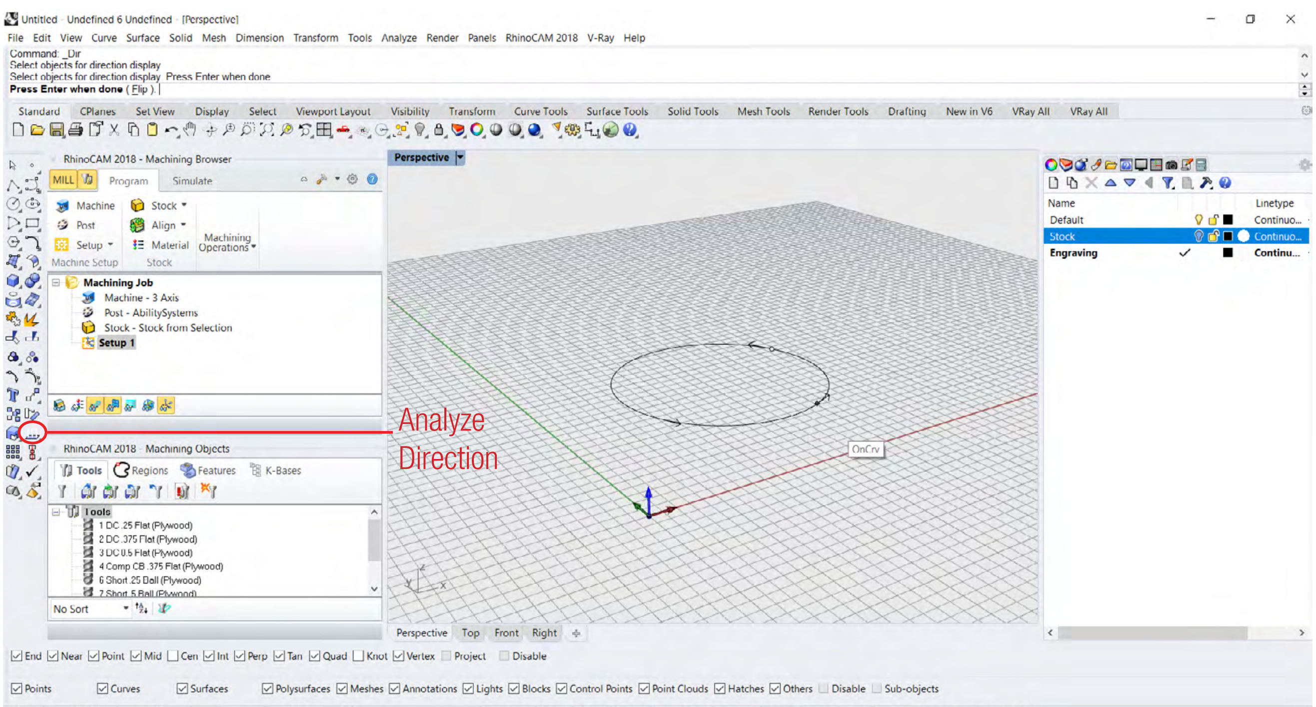
PROFILING

Profiling Milling paths follow curves along the directional normal of the curve, the tool will always assign itself to the right side of the directional normal (unlike engraving which goes down the center of the curve). To change the cutting direction of the profile the directional normal can be changed in Rhinoceros or the setup parameters in the profiling pass can be changed. Profiles can either be performed on open or closed curves.

Profiling is often used to cut pieces out or a larger stock or cleaning up geometry after a parallel finish.

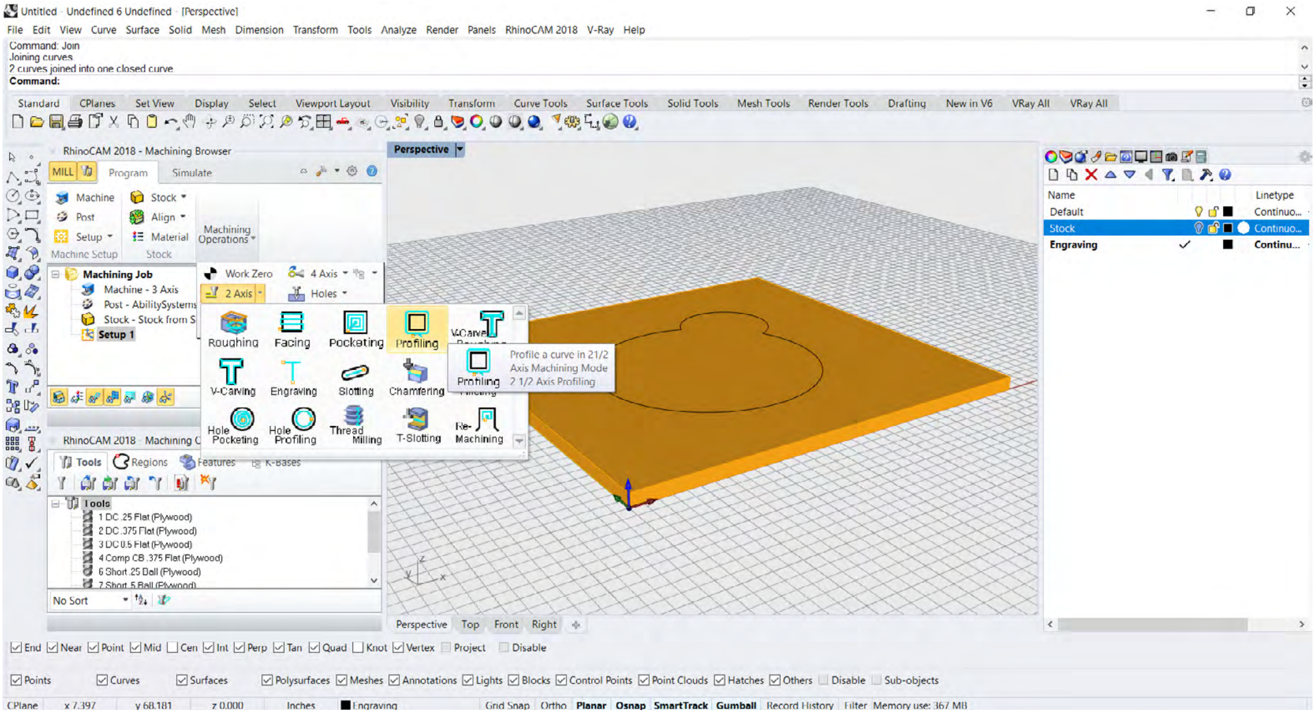
Flat tools work best for both processes above.



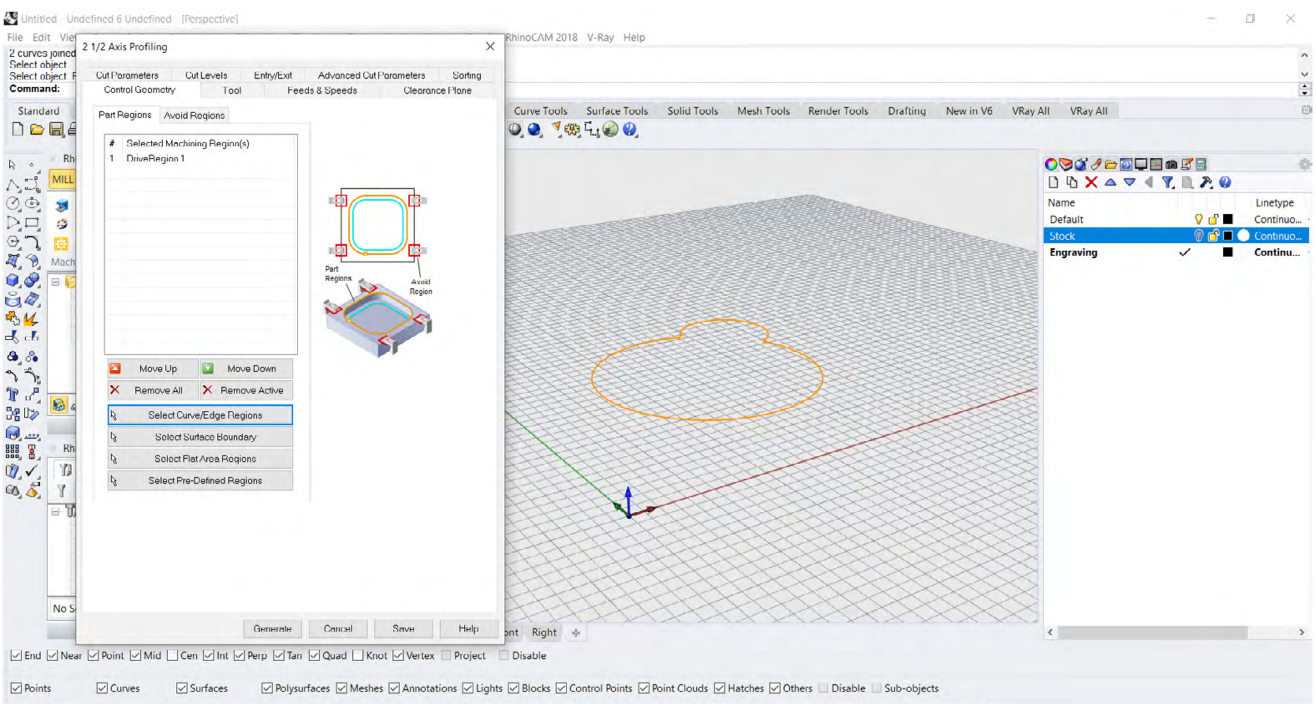


The direction of the curve can be flipped while in the Analyze direction command. These curvature directions will be important in assigning 2D milling operations as the line/ curve direction will dictate the cutting direction in your milling operations.

PROFILING

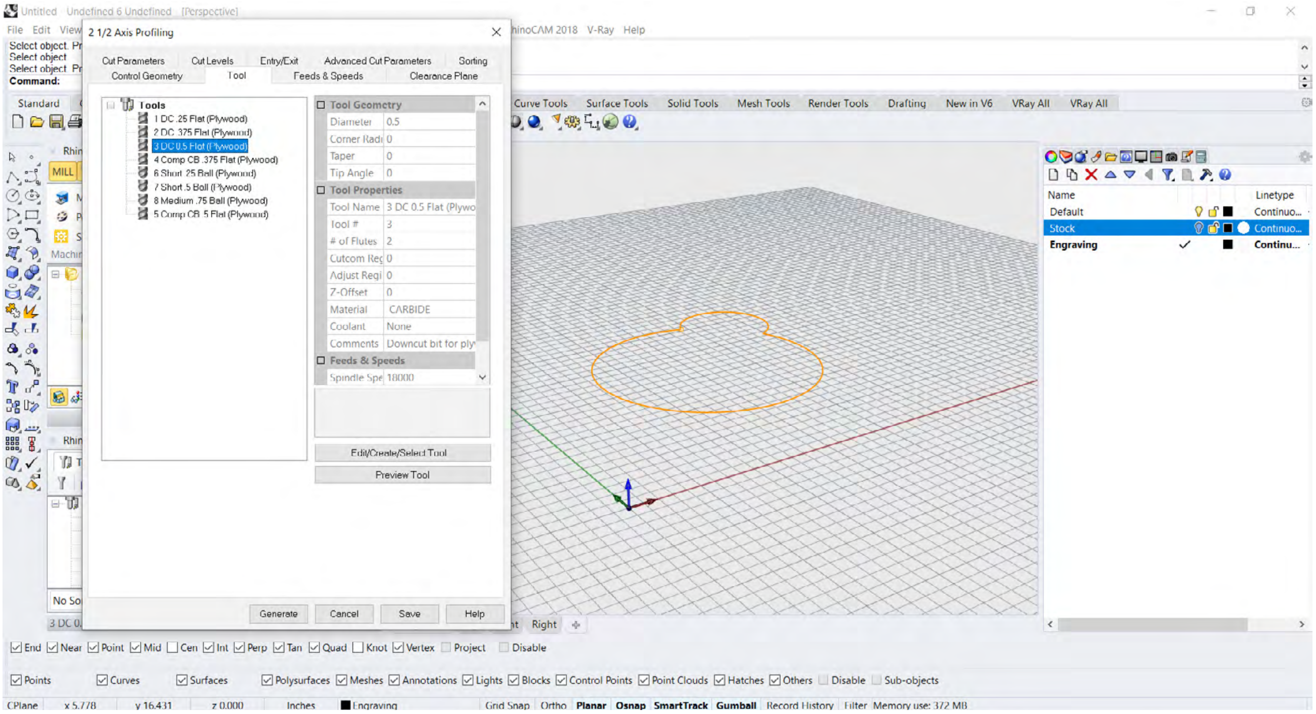


The profile curve is aligned to the top surface of the stock material. In the cut parameters in the machining operations for engraving the cutting depth can be established.

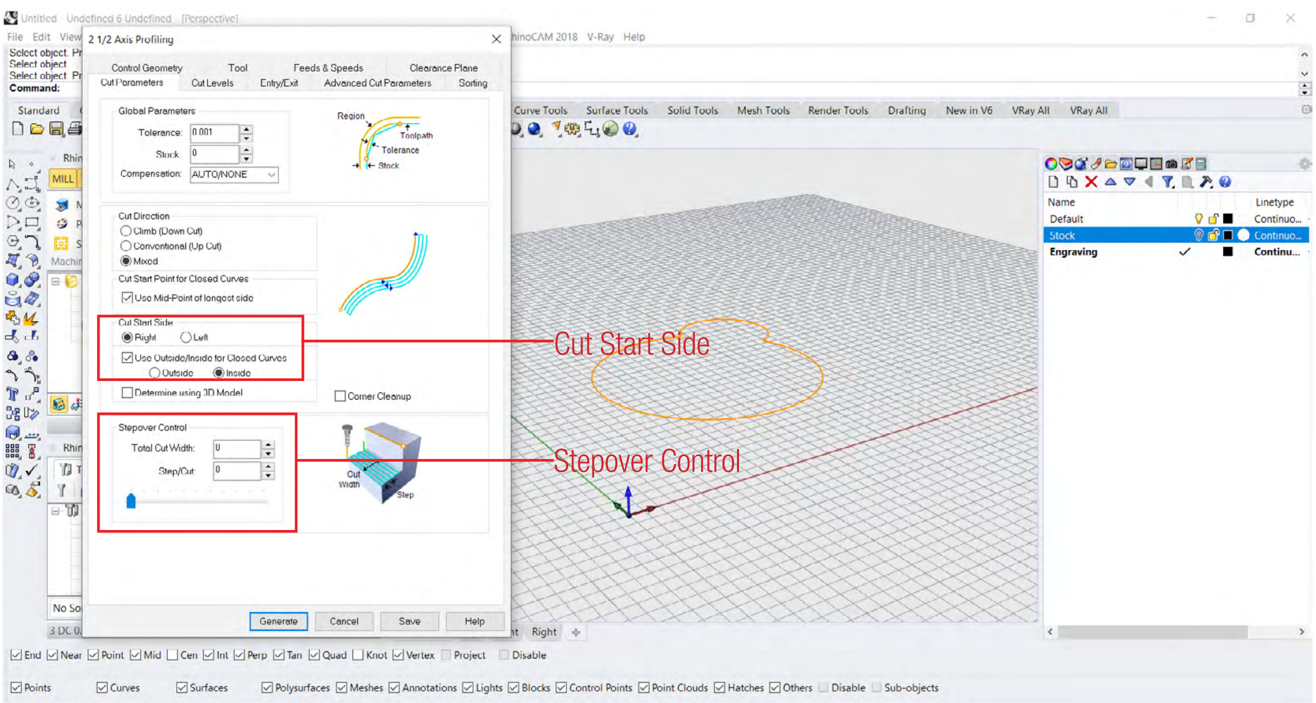


Select Profiling from the 2D milling operations tab, press Select Curves/Edge Regions and select the curves or regions you want to profile to create your milling path(s).

PROFILING

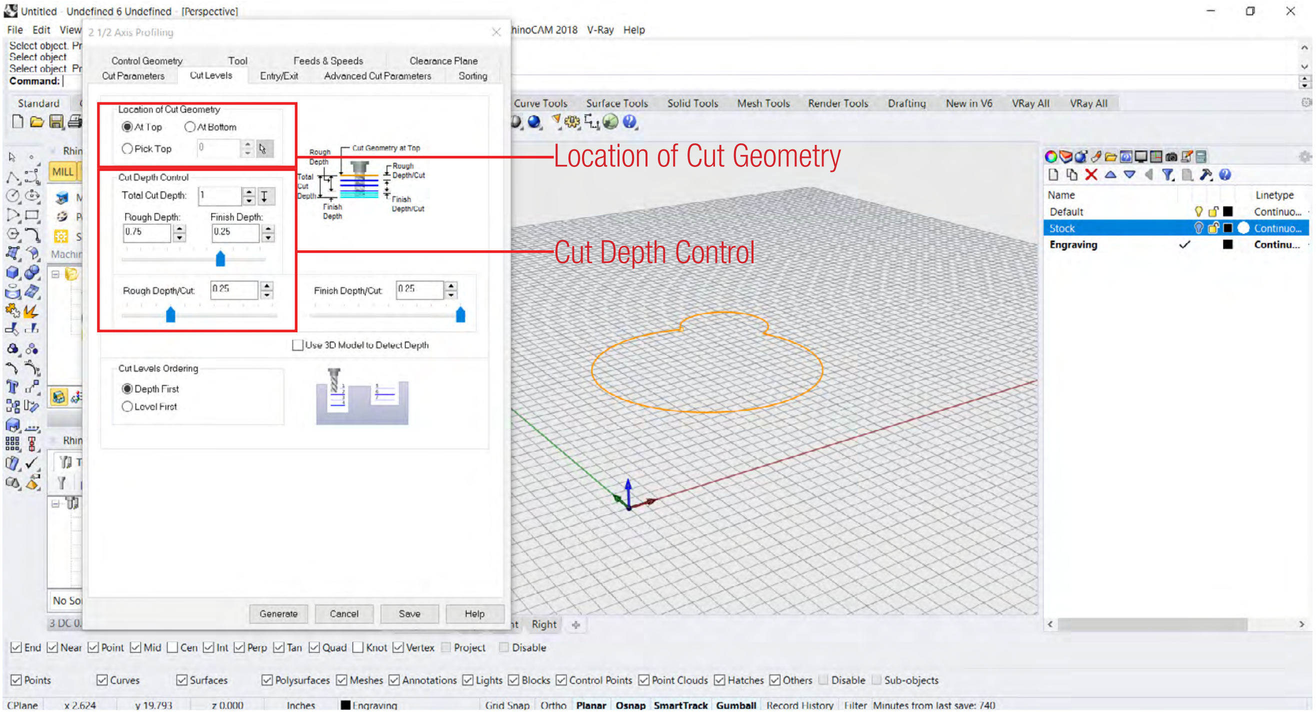


The tool diameter selected will determine the width of the profiling line. Flat Tools work best for profile cuts.

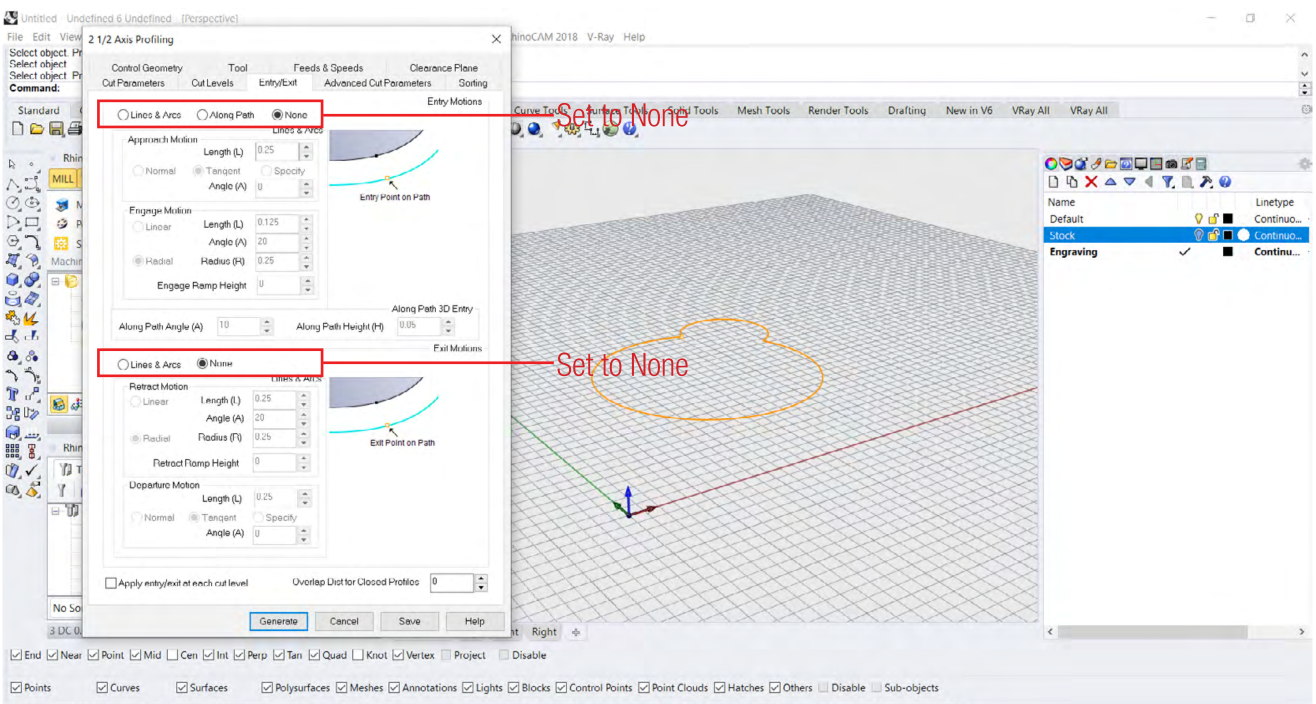


In the Cut Parameters tab the cut direction can be changed and for closed curvature either the inside or the outside of a closed curve can be chosen to cut. A offset of the profile can be setup with Stepover Control, the cut width and the step over to the desired width can be set up here.

PROFILING

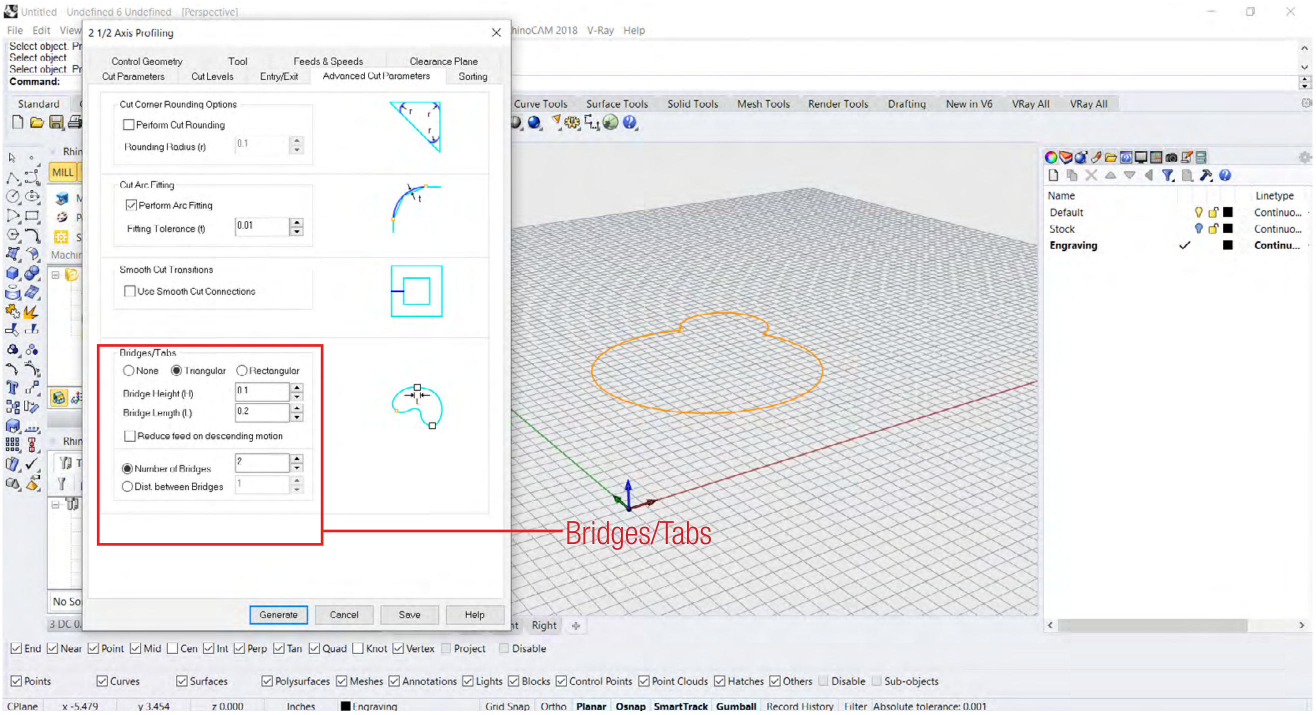


Under the Cut Levels tab the depth and geometry location is chosen. The geometry currently in the example is located at the top and the cut depth is 1 inch, the rule is never cut deeper than half the diameter of the tool, so since we are using a 0.5 inch tool, we set the rough depth to 0.25 so the cut is done in four passes. For less dense materials ie. foam we can cut 100% of the tool.

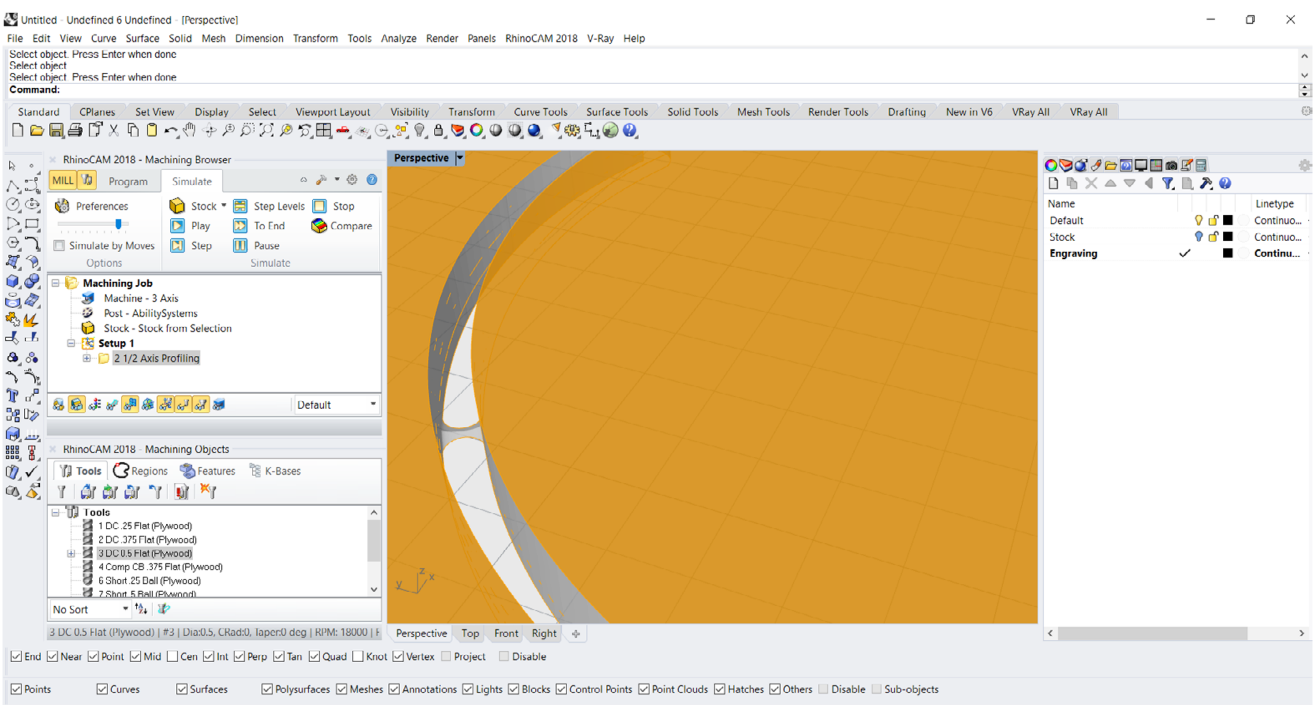


Under the Entry/Exit tab the entry and exit is set. They are automatically set up to Lines & Arcs but they need to be set to None for both the entry and exit.

PROFILING

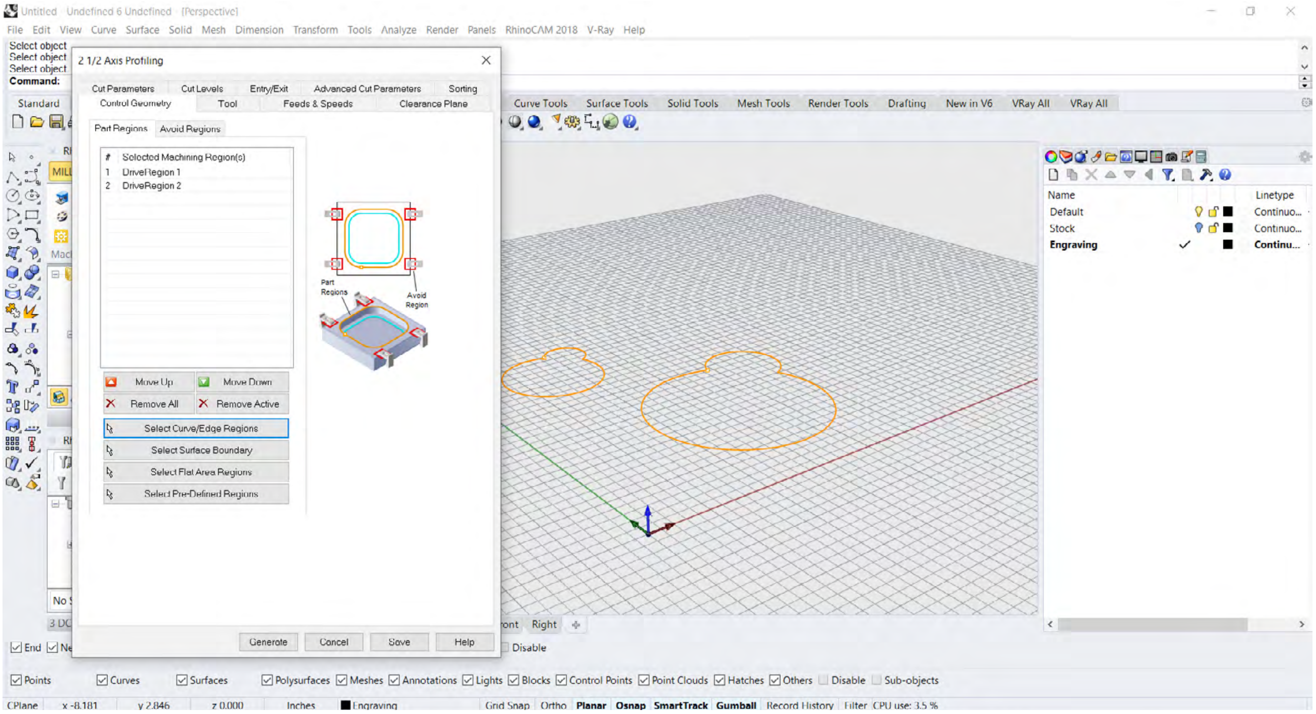


When cutting a profile out of a larger surrounding material the inner cut can lose suction to the CNC vacuum bed once cut. This can cause it to shift and bump into the tool. To prevent this bridges can be created, these create small tabs that link the cut piece to the larger material. The height, length and number of bridges can be set. Once cut the bridges can be hand cut to release the profile from the material. Triangular bridges are used in most cases as they are easier to remove.

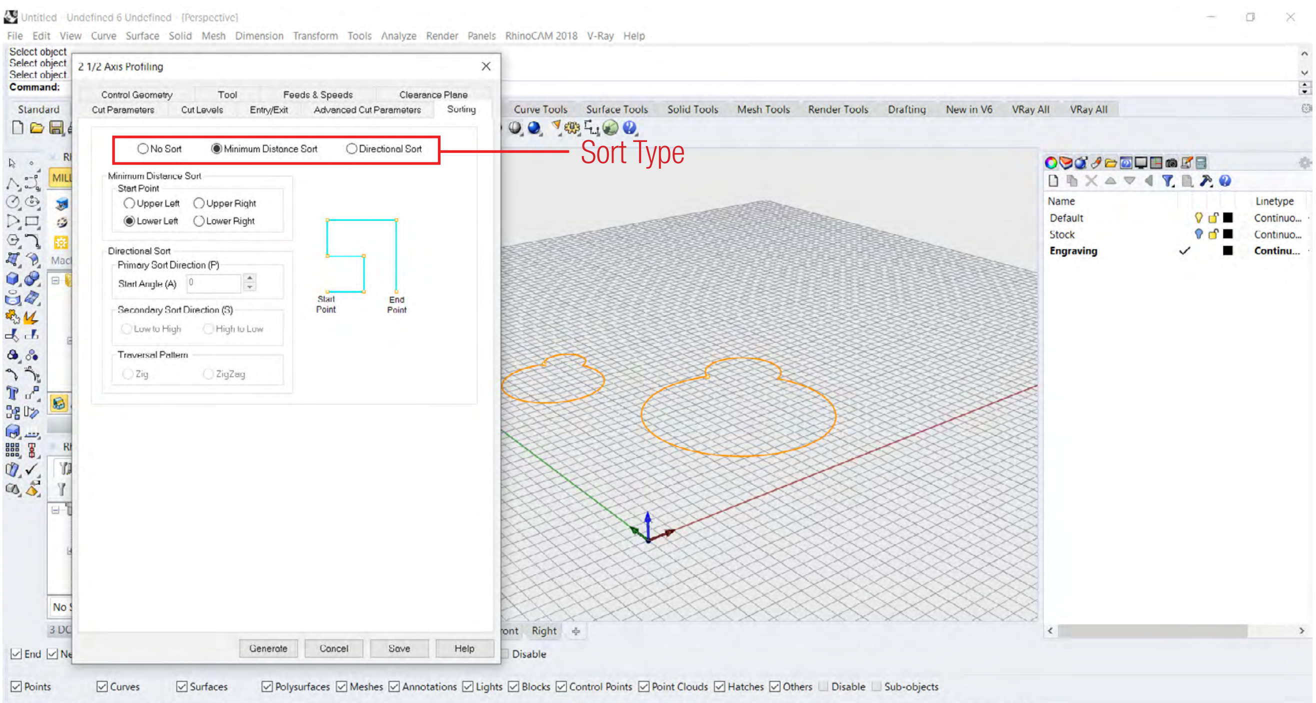


Once the milling paths are generated select the Simulate tab once the simulation has been run the bridges are visible linking the profile cut back to the main sheet material.

PROFILING



For multiple profiles, select all the drive regions as long as the cutting operation performed remains the same. Under the Sort tab select minimum sorting distance.

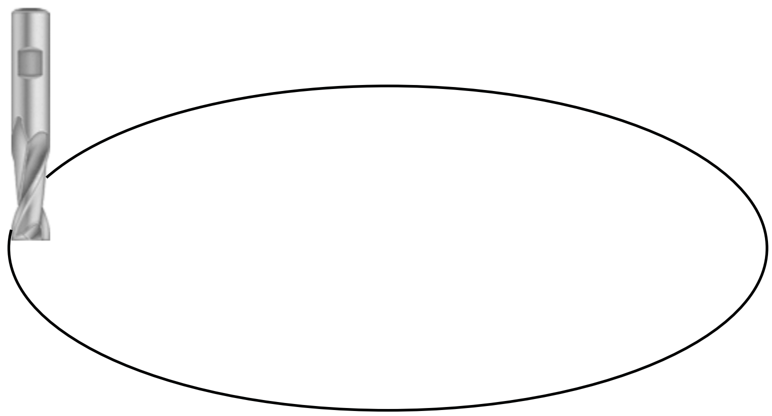


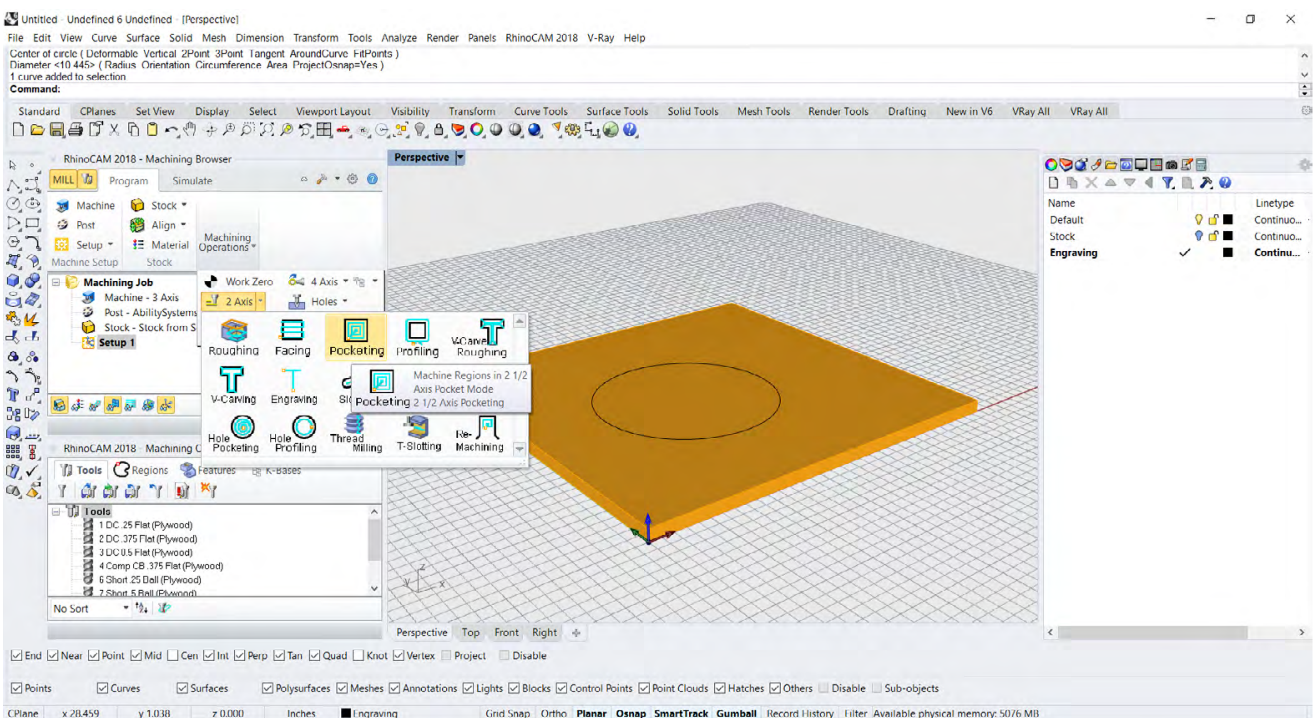
If multiple profiles are being cut in a single sheet the cutting order can be set up under the sorting tab, this allows the shortest path generated from a starting point.

POCKETING

Pocketing milling operation will mill out a selected region, either within a bounding regions our between two bounding areas. The generated paths will populate the region with milling paths to remove the desired material to a depth assigned in the cutting depth operation.

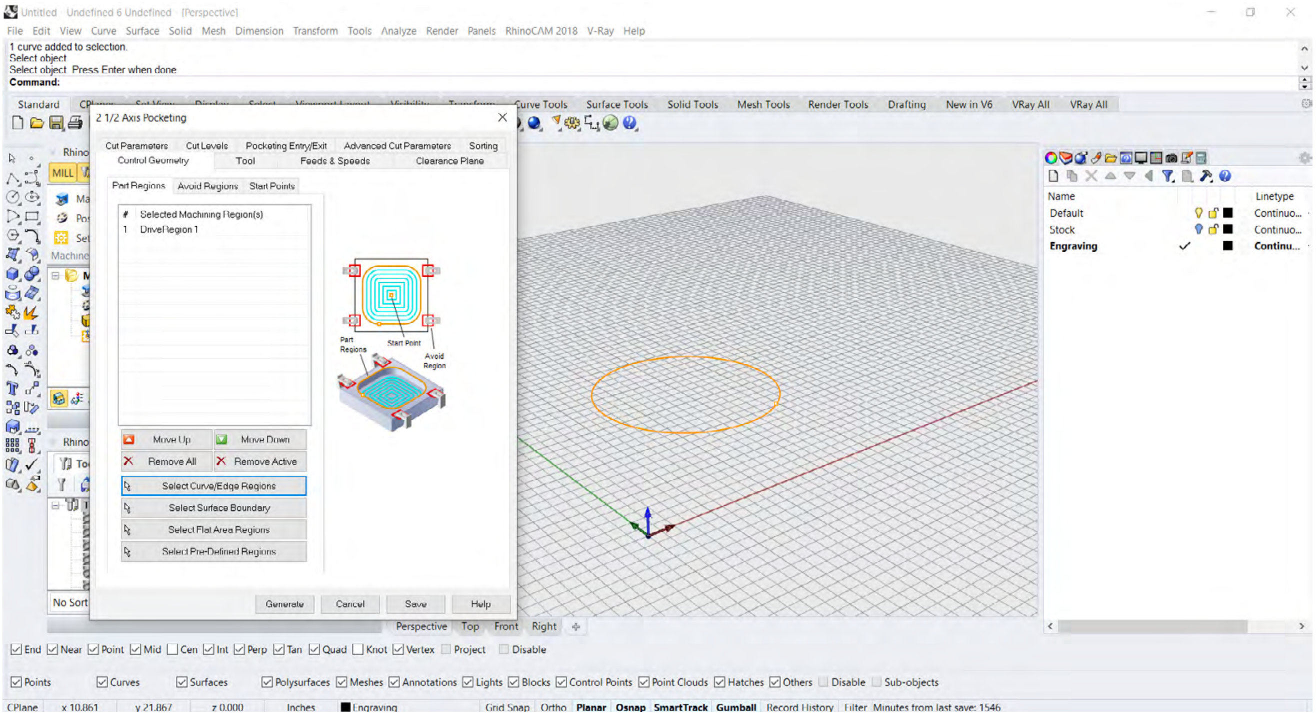
Flat tools work best to create a flat surface of the pocketing or to cleanly cut out a shape.



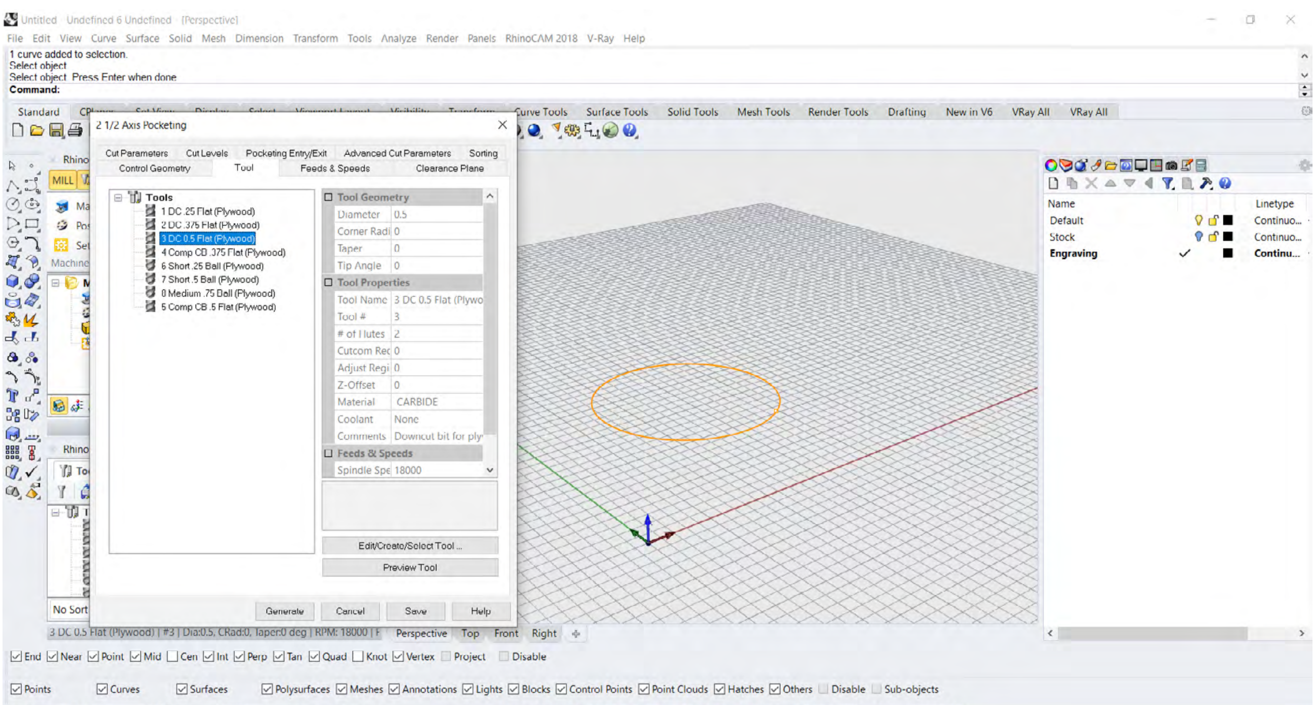


The Pocketing curve is aligned to the top surface of the stock material. In the cut levels tab in the machining operations for Pocketing you can set the cut depth.

POCKETING

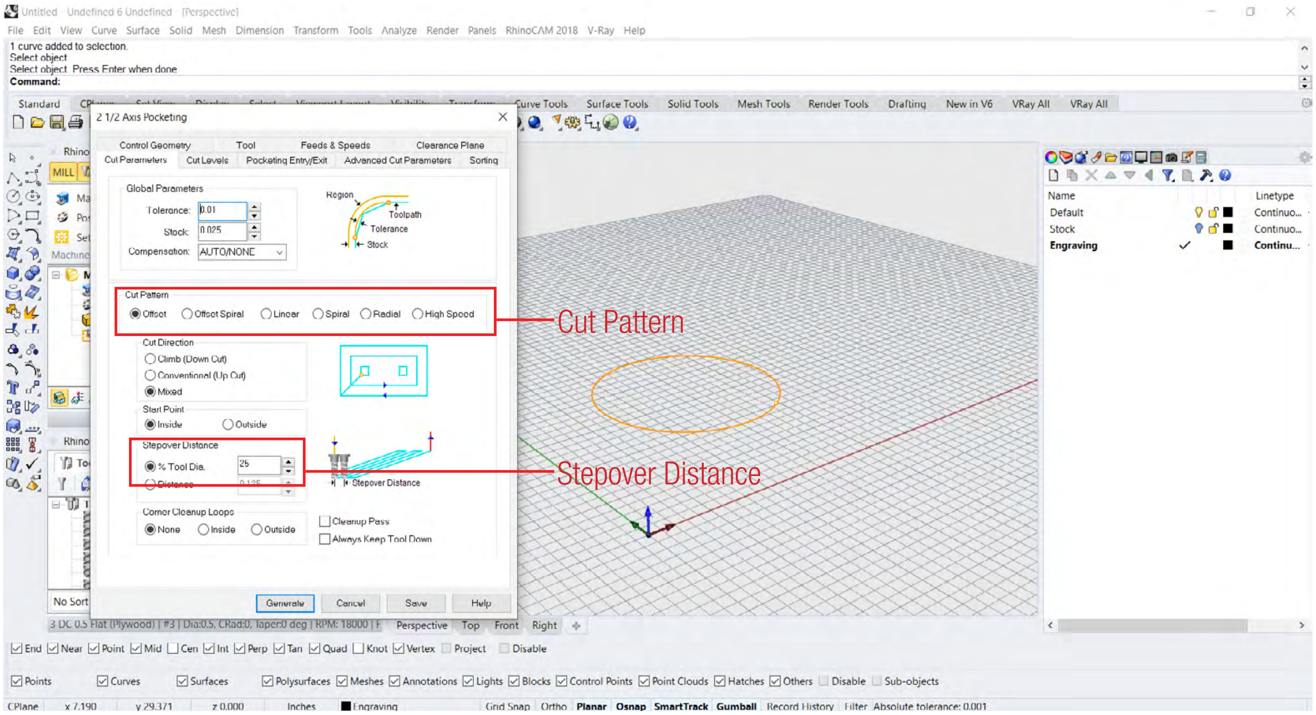


Select Pocketing from the 2D milling operations tab, press Select Curves/Edge Regions and select the curves or regions you want to profile to create your milling paths. **You must use a closed curve for Pocketing to work.**

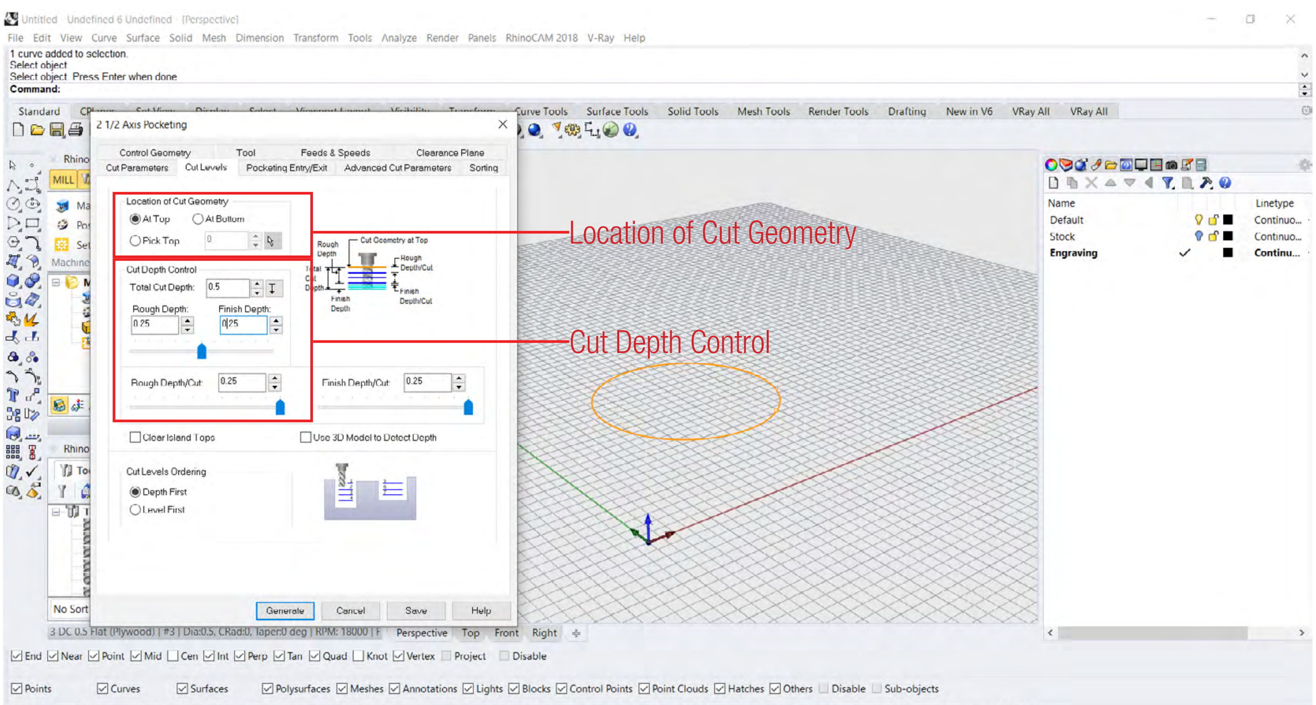


Select the tool for the operation, a 0.5 or larger flat tool is recommended for this operation.

POCKETING

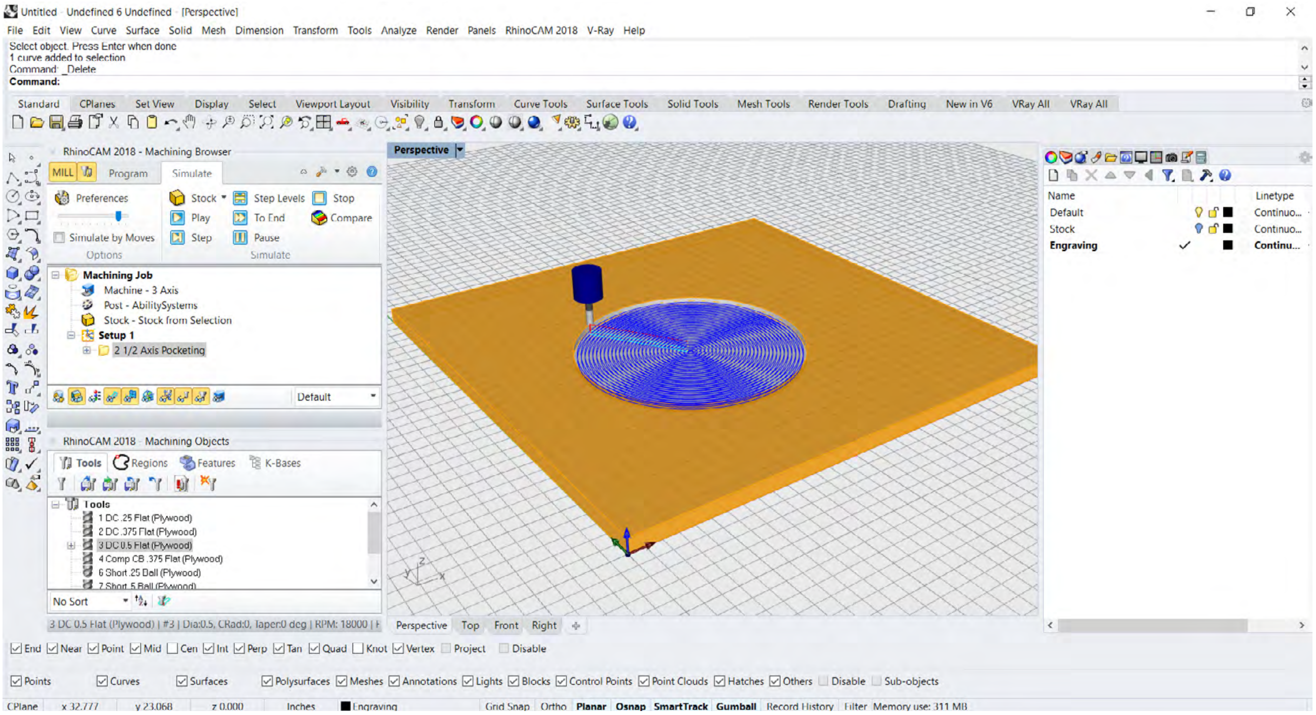


Under Cut Parameters the pocket can be cut using either Offset, Linear, Spiral or Radial cuts. The choice of operation will be dependent of the geometry to be pocketed, for the curvature shown the offset cut will be used starting from the inside to periphery of the geometry. Set the stepover to 50% to quickly remove material.



Under the Cut Levels tab the depth and geometry location is chosen. The geometry currently in the example is located at the top and the cut depth is 0.5 inch, the rule is never cut deeper than half the diameter of the tool, so since we are using a 0.5 inch tool, we set the rough depth to 0.25 so the cut is done in two passes. For less dense materials ie. foam we can cut 100% of the tool.

POCKETING



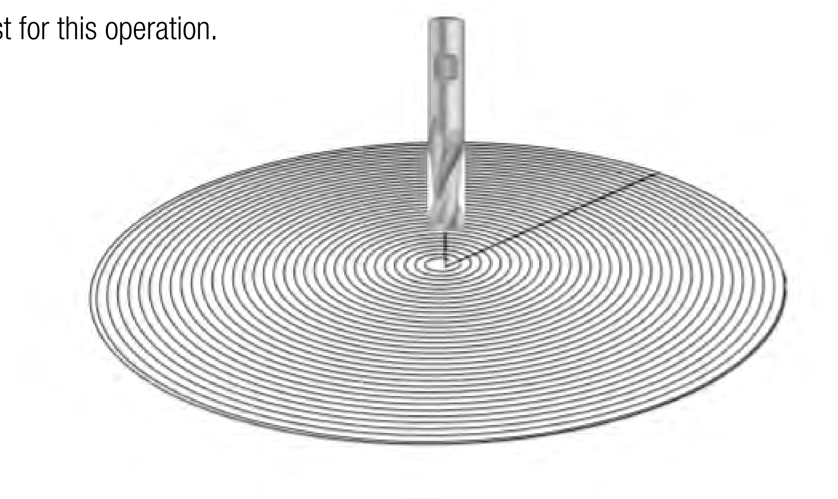
Once the milling paths have been generated select the simulate tab and run the pocketing paths to see the toolpath result.

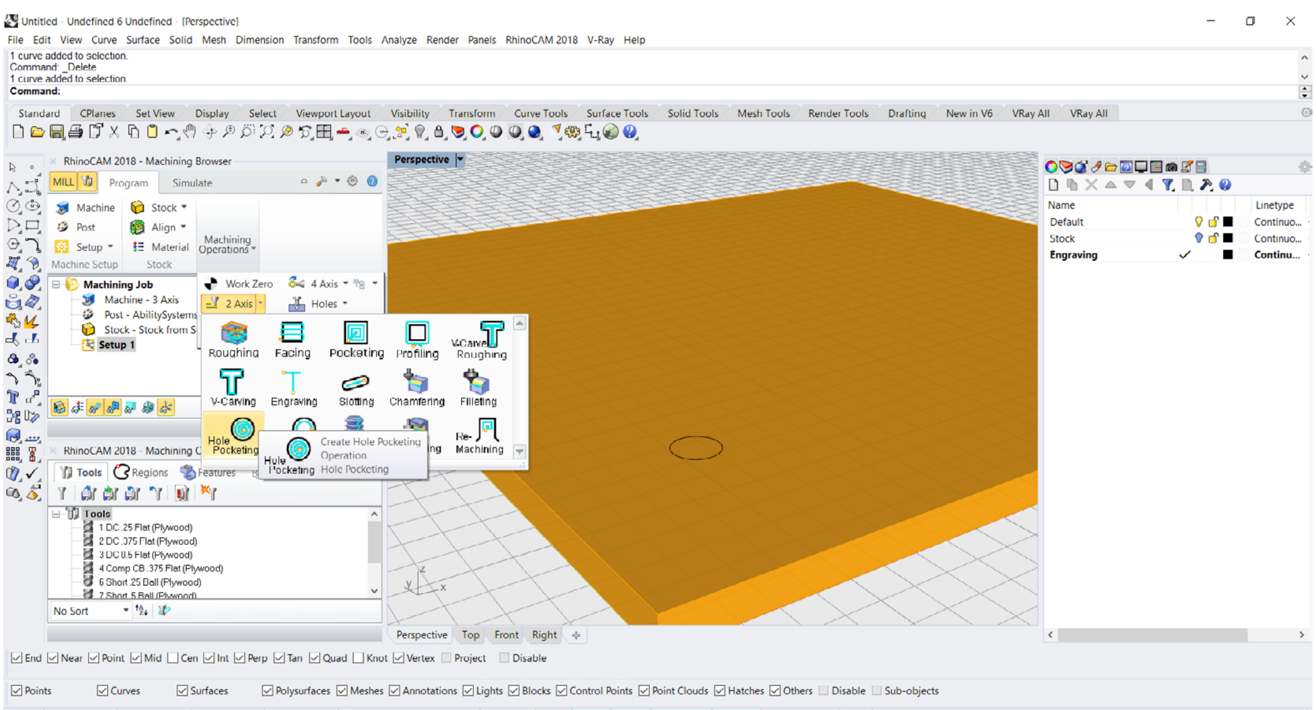
HOLE POCKETING

Hole Pocketing operation can be used to create spiral tooling paths to cut a large hole diameter. Similar to a pocketing pass but with a spiral pattern which begins at the center and progresses outwards.

This operation works well to create perfectly cut small circular holes as the toolpath is a spiral.

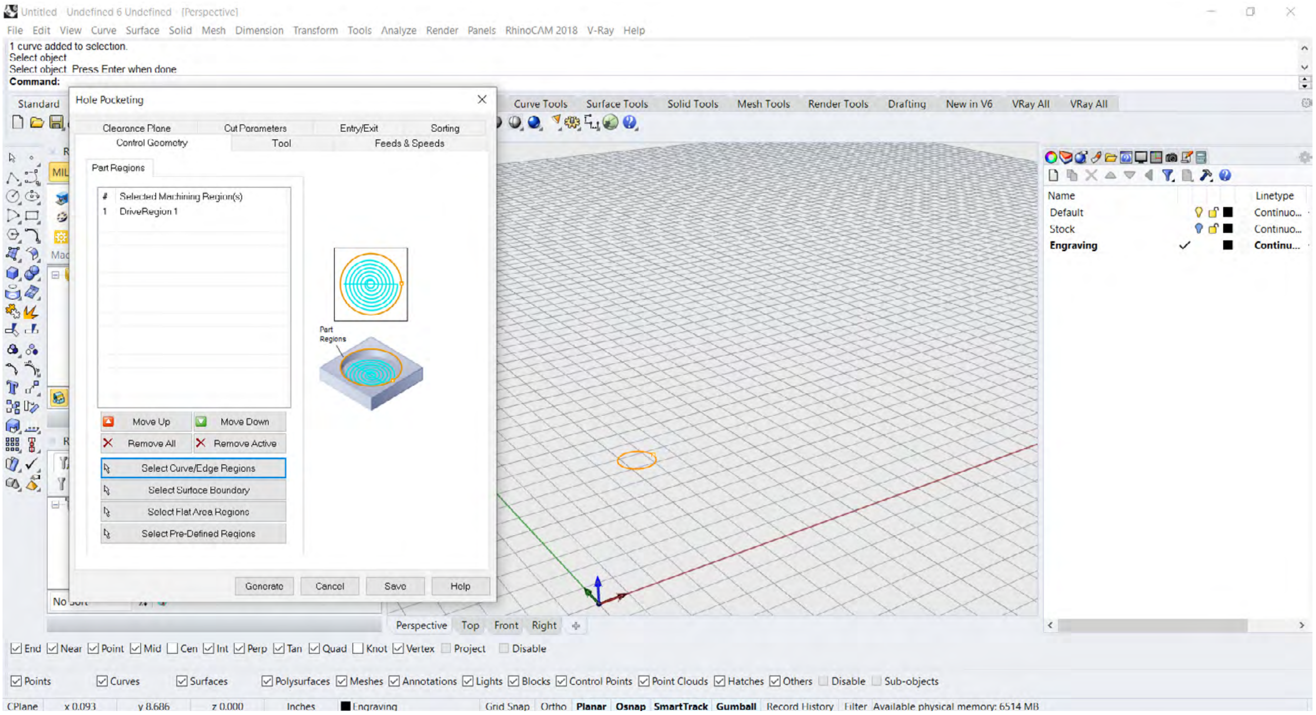
A Flat tool works best for this operation.



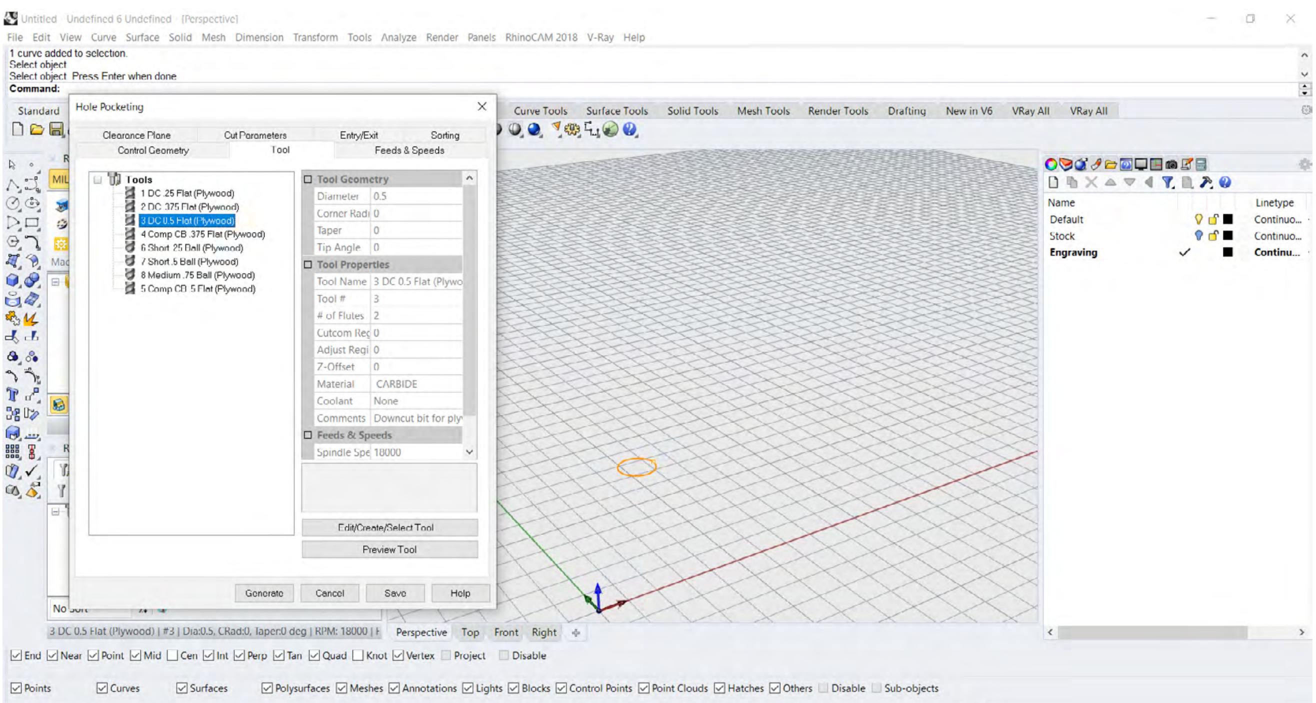


The Hole Pocketing curve is aligned to the top surface of the stock material. In the cut parameters tab in the machining operations for Hole Pocketing you can set the cut depth.

HOLE POCKETING

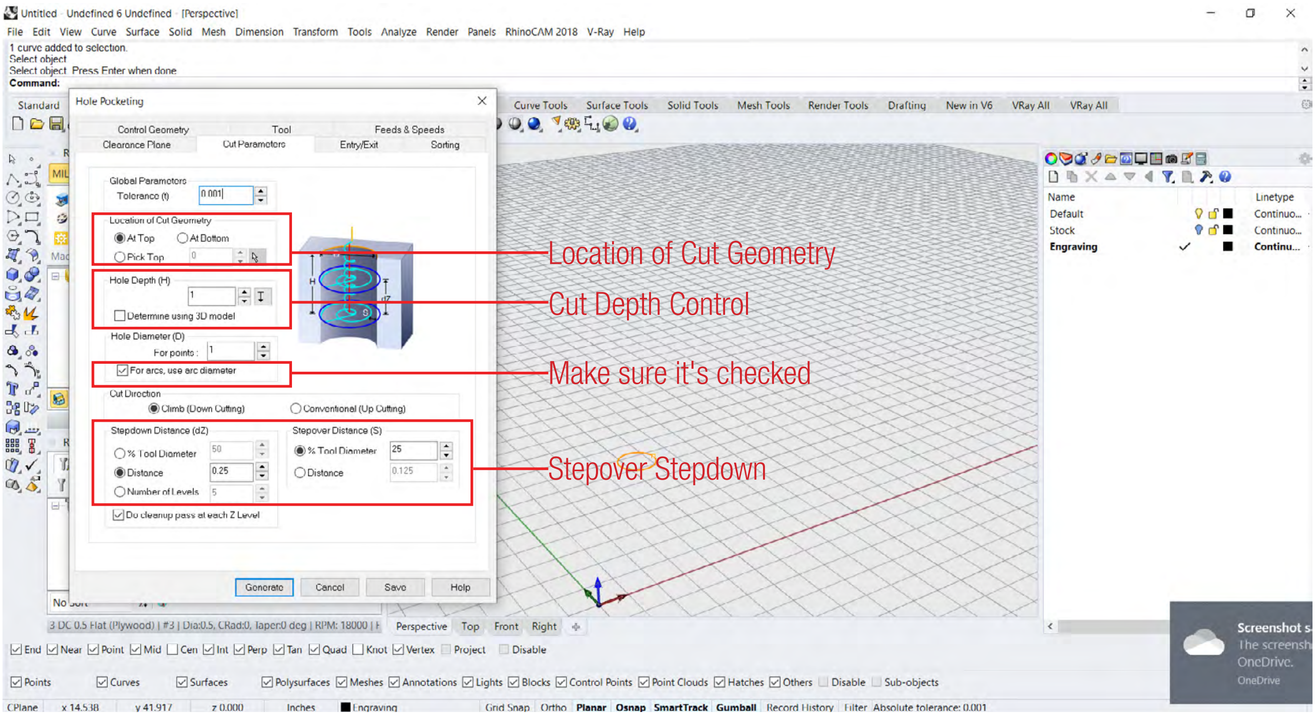


Select Hole Pocketing from the 2D milling operations tab, press Select Curves/Edge Regions and select the curves or regions you want to profile to create your milling paths. **You must use a closed curve for Hole Pocketing to work.**

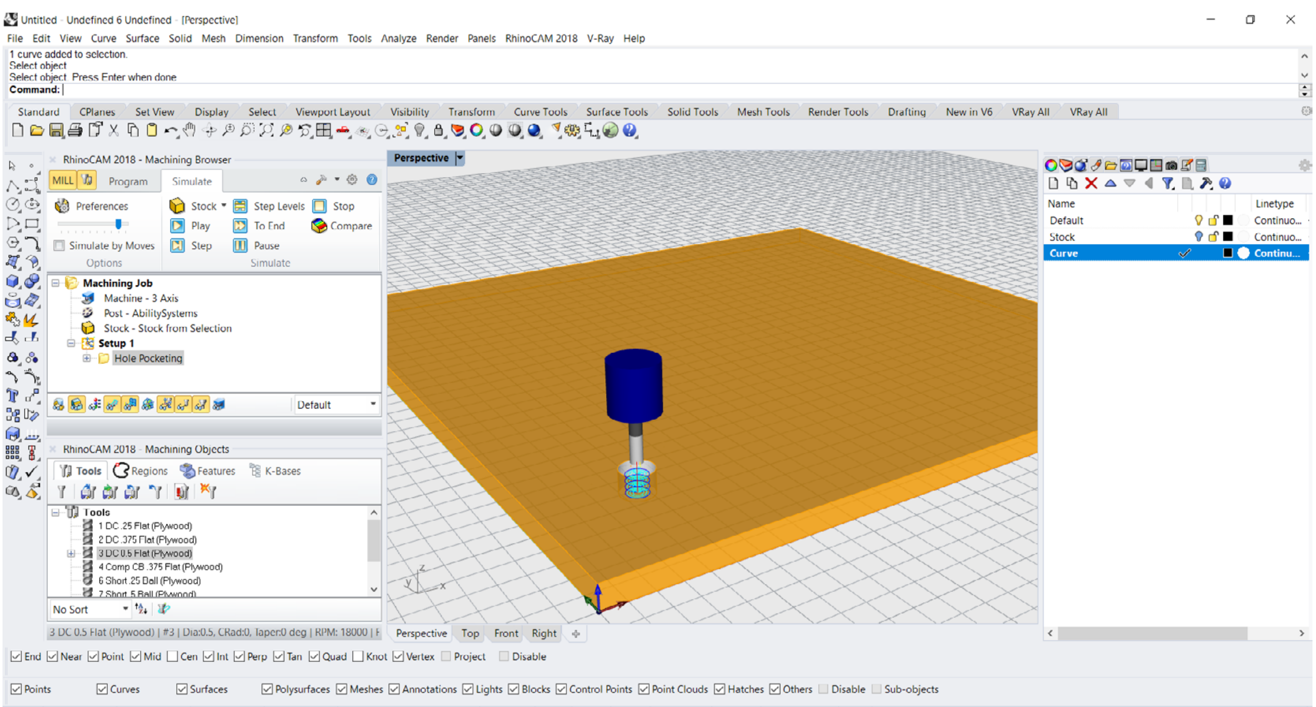


Select the tool for the operation, a flat tool is recommended for this operation, the size is dependent on the size of hole.

HOLE POCKETING



Under the Cut Parameters tab the depth and geometry location is chosen. The geometry currently in the example is located at the top and the cut depth is 1 inch, the rule is never cut deeper than half the diameter of the tool, so since we are using a 0.5 inch tool, we set the stepdown distance to 0.25 so the cut is done in four passes. The stepover is set to 25% which could be increased depending on the material to speed up the mill time. For less dense materials ie. foam we can cut 100% of the tool.



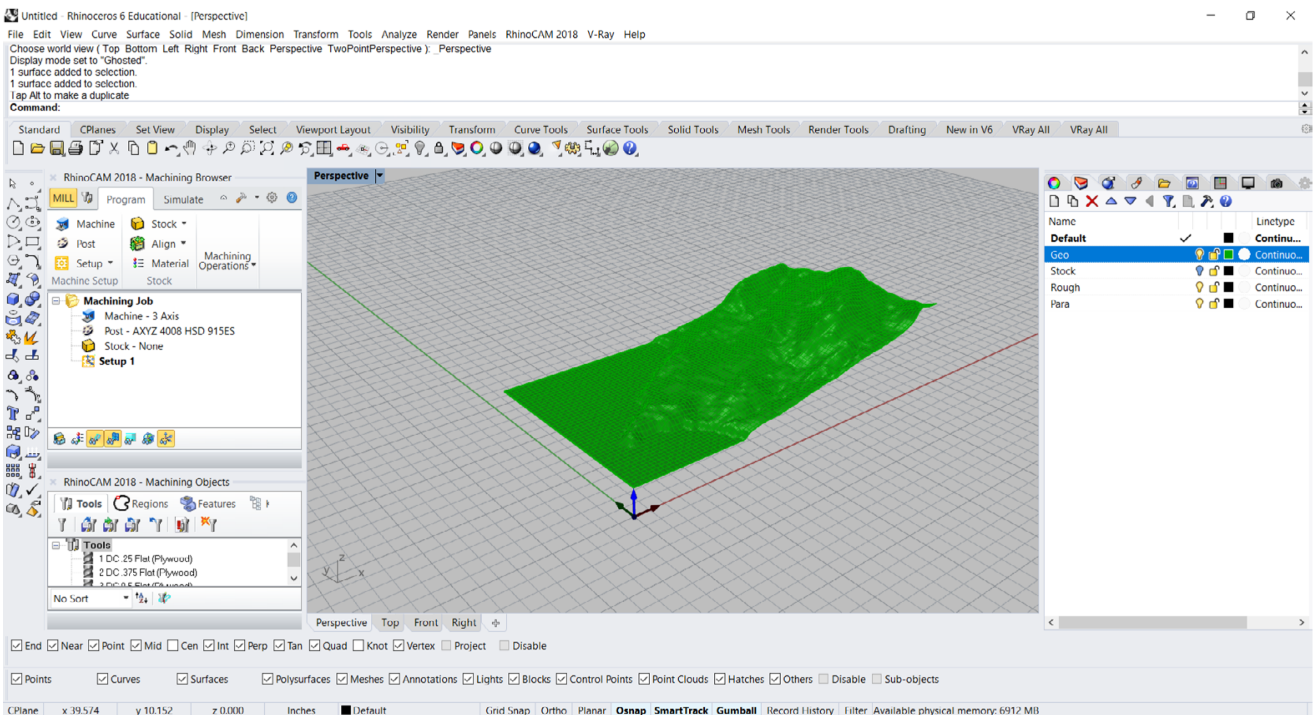
Once the milling paths have been generated select the simulate tab and run the pocketing paths to see the toolpath result.

3 Axis Machining Operations

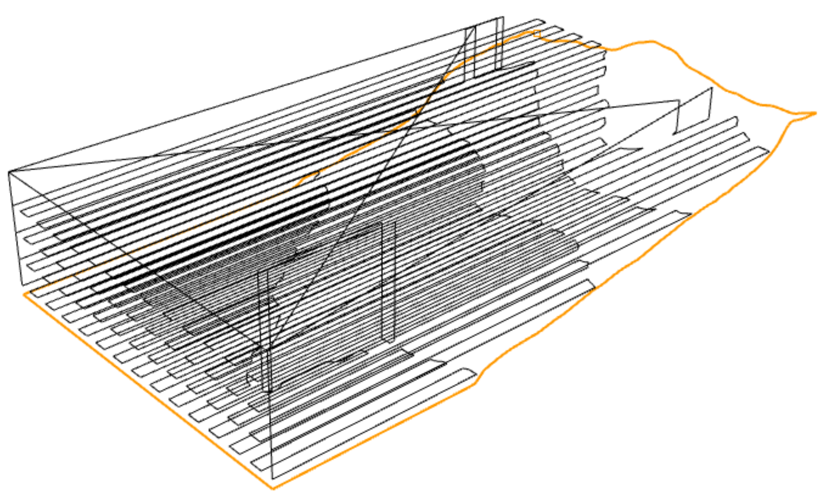
HORIZONTAL ROUGHING

Horizontal Roughing is located in the 3 Axis milling operations, the operation is used to quickly remove a lot of materials before the finishing operation usually parallel finishing or horizontal finishing but there are other operations available as well. After setting up Horizontal Roughing you get the error that there is nothing to cut, you material maybe flat enough that roughing is not necessary and you can go right to finishing.

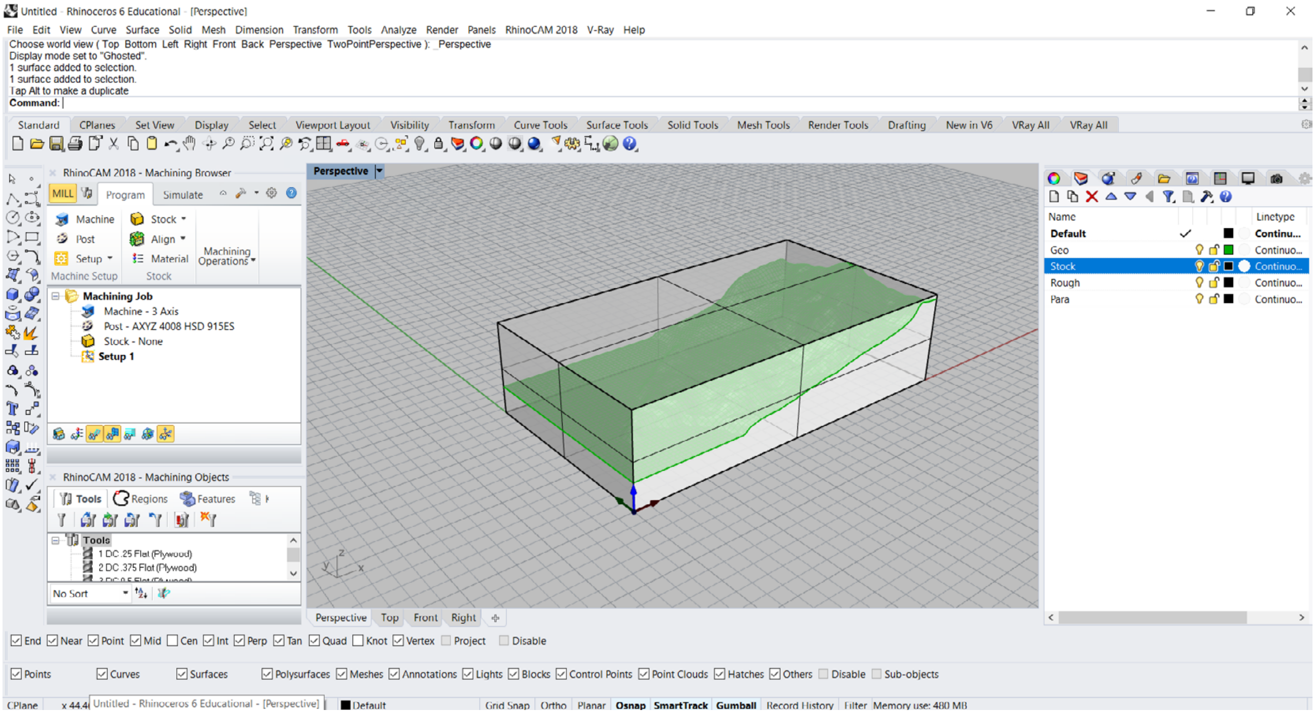
A large flat tool is recommended for this operation.



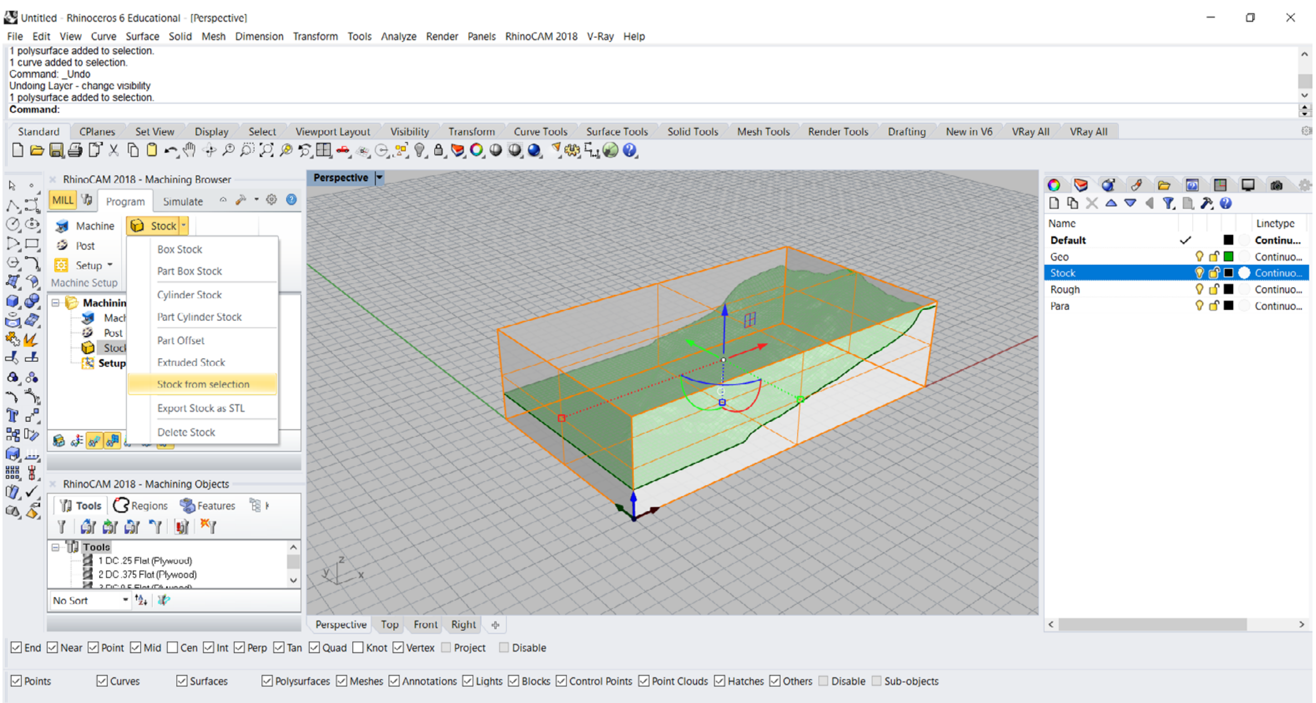
In the sample above a terrain file has been imported into Rhino, the surface was generated from contour data. The surface will be placed in a piece of stock that will match the physical stock to be milled. Both meshes and nurbs surfaces can be used for 3d milling.



HORIZONTAL ROUGHING

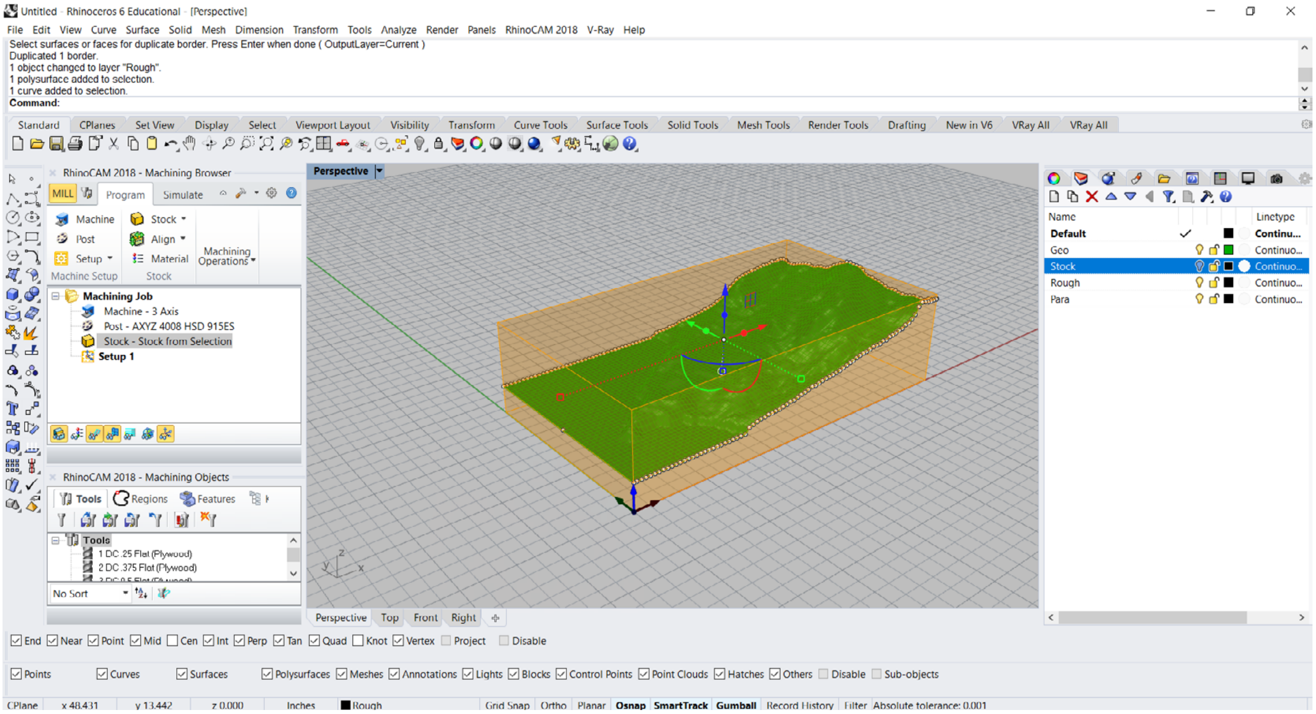


Create a closed polysurface (usually a box) to match the physical milling stock, place the surface within the stock, try to add a bit of space between the top of the stock and the surface in case your stock is a little shorter. The deeper the terrain is inset into the stock the more material there is to remove for the finished topography.

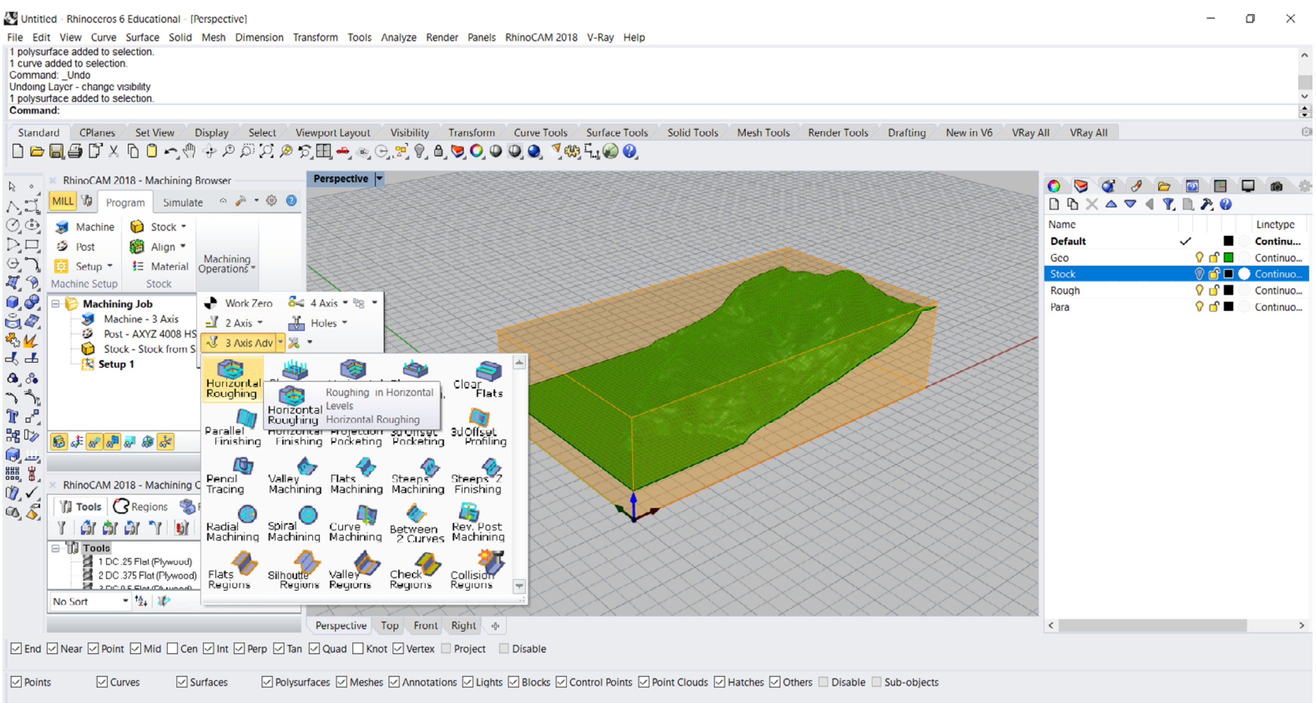


To set you stock in RhinoCAM highlight the box and from the program tab choose stock then stock from selection. This will create a stock in RhinoCAM to be milled from that is ghosted in orange.

HORIZONTAL ROUGHING

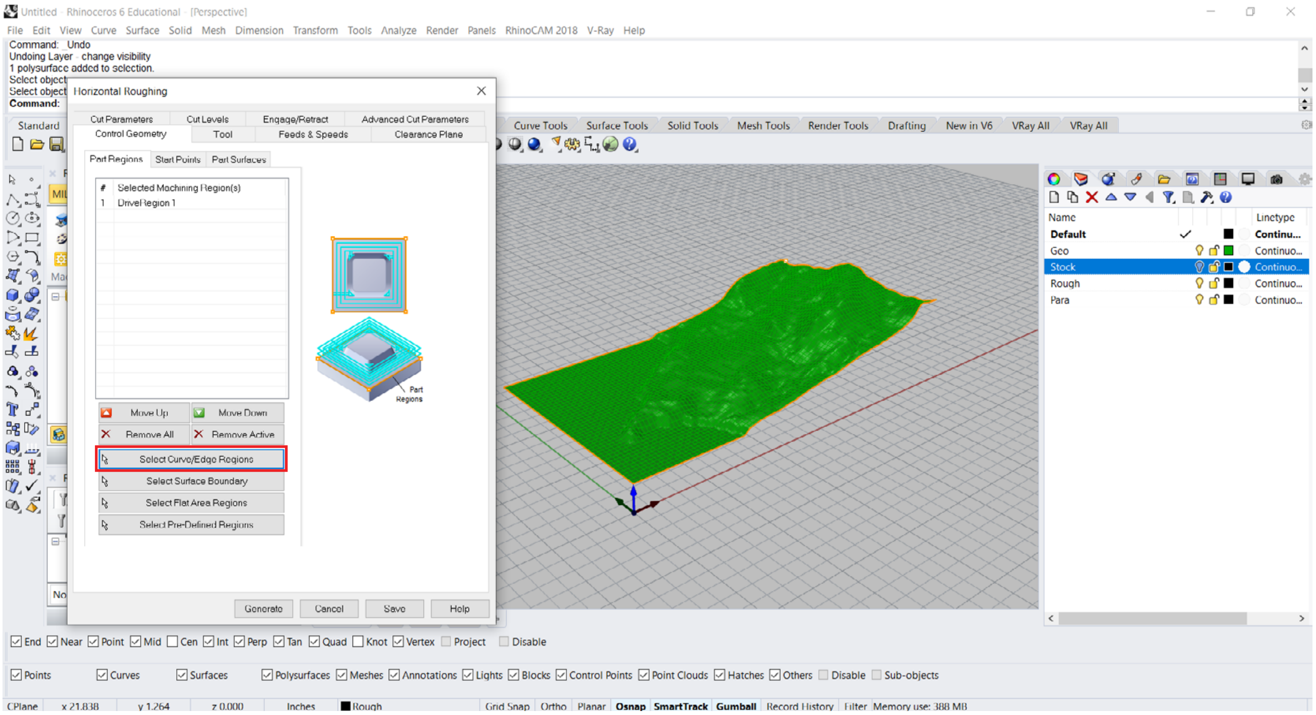


Create a curve that outlines the area that you want milled. These areas are called a bounding region. The curve can be along the edge of the surface like seen here or projected flat. The Dupboarder command in rhino is a quick way to get the edge of surface. Best practice is to keep the curves on separate layers, labelled with the milling operation you are performing with those curves.

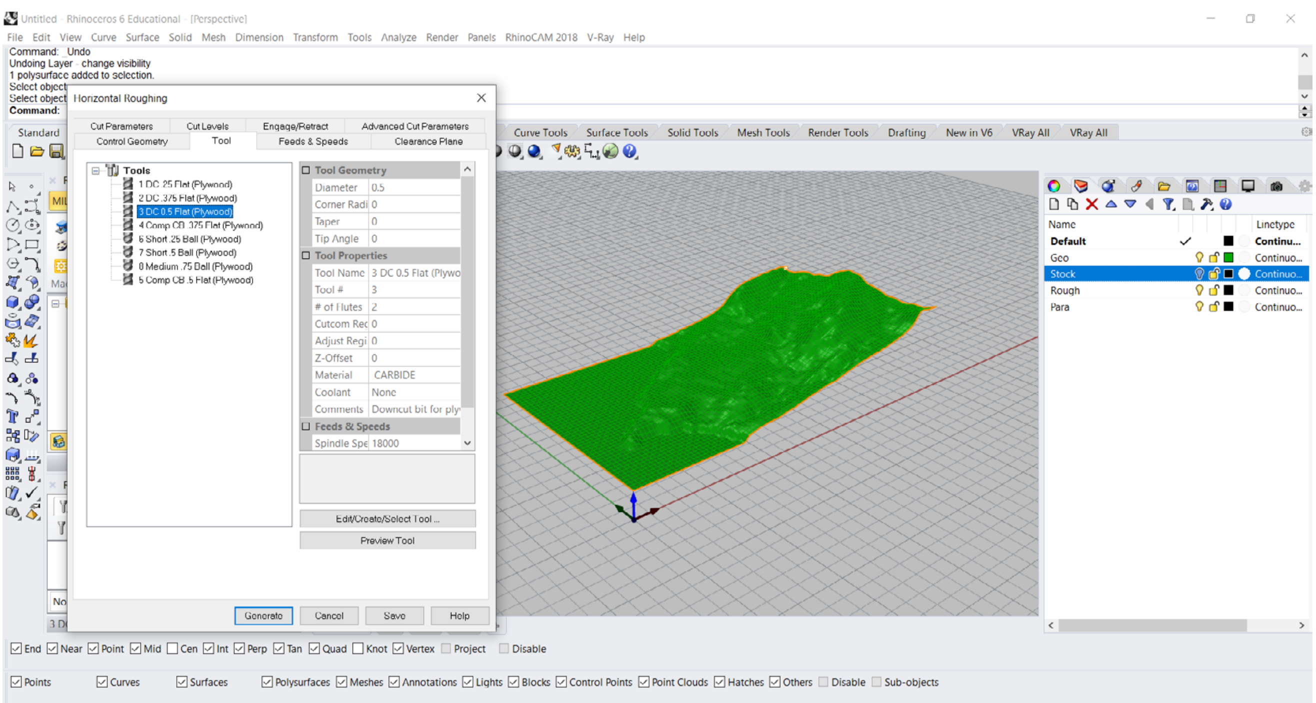


Under the 3 Axis Adv tab, select Horizontal roughing.

HORIZONTAL ROUGHING

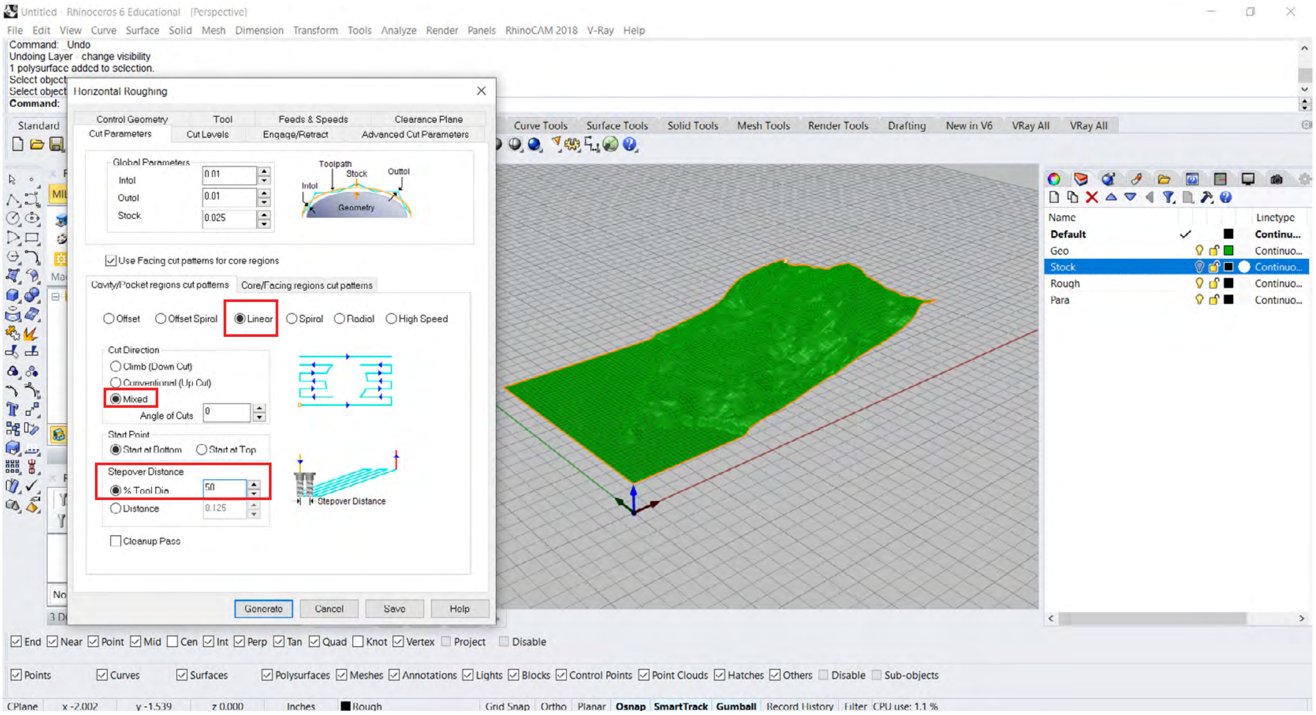


Press Select Curves/Edge Regions and select the curves or regions you want to profile to create your milling paths.

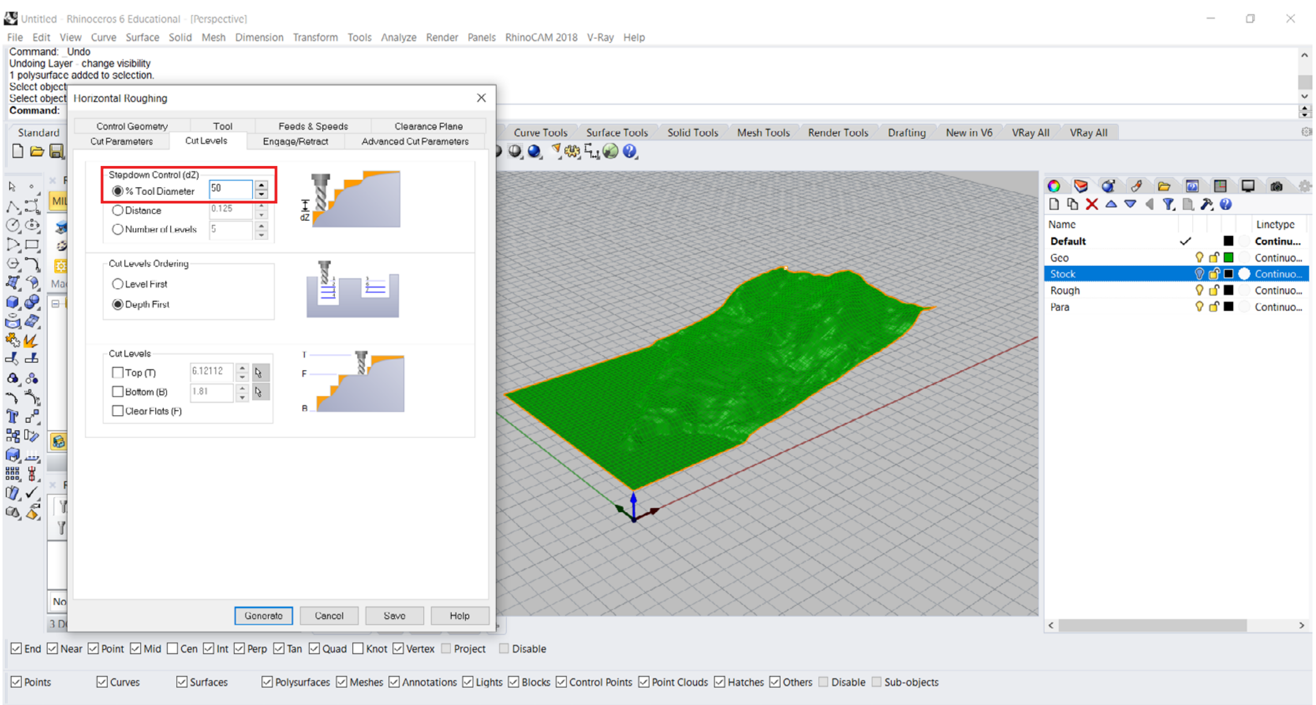


Select the tool to be used for the roughing operation, typically a larger diameter flat tool will be used to remove the material to approximate the rough surface.

HORIZONTAL ROUGHING

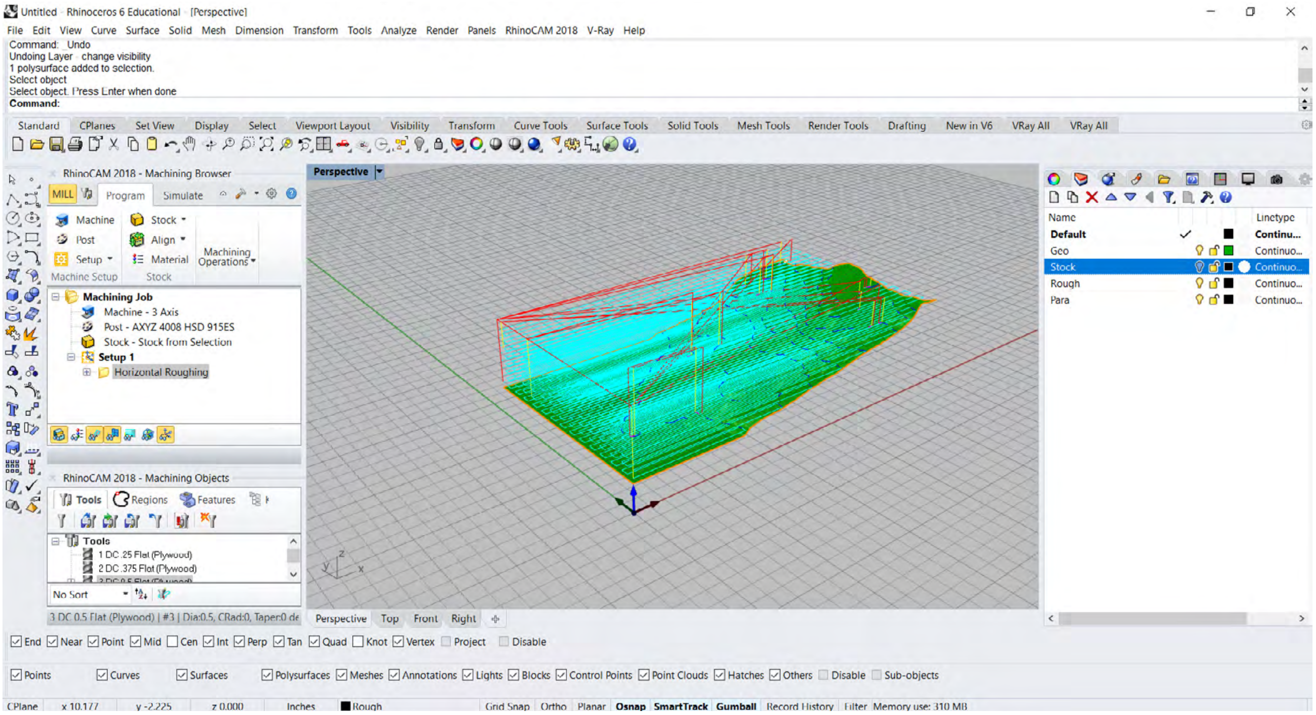


Under Cut Parameters set the cut pattern to linear and the cut direction to mixed for materials that cut direction isn't necessary (ie Foam). For stepover control set to 50% for dense materials and for foam this can be increased to 100%.

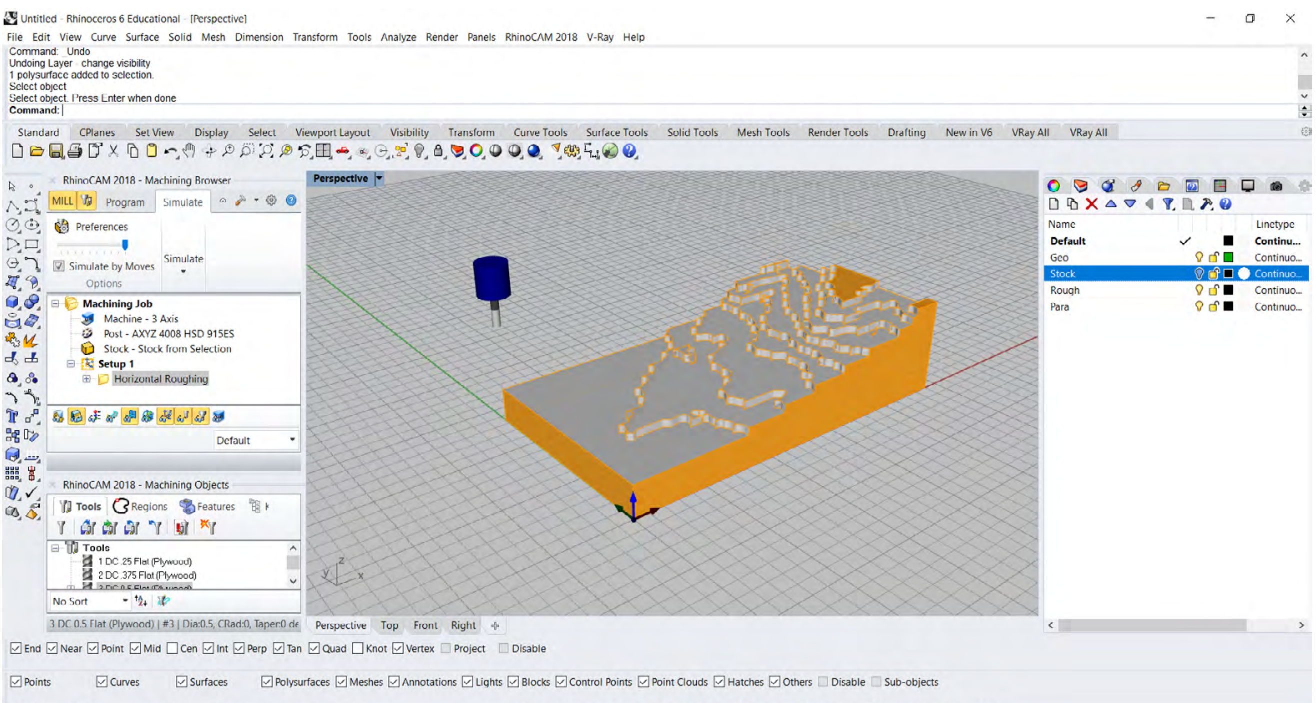


Under Cut Levels the stepdown control can be set, this can be set to 50% for dense materials and for foam this can be increased to 100%.

HORIZONTAL ROUGHING



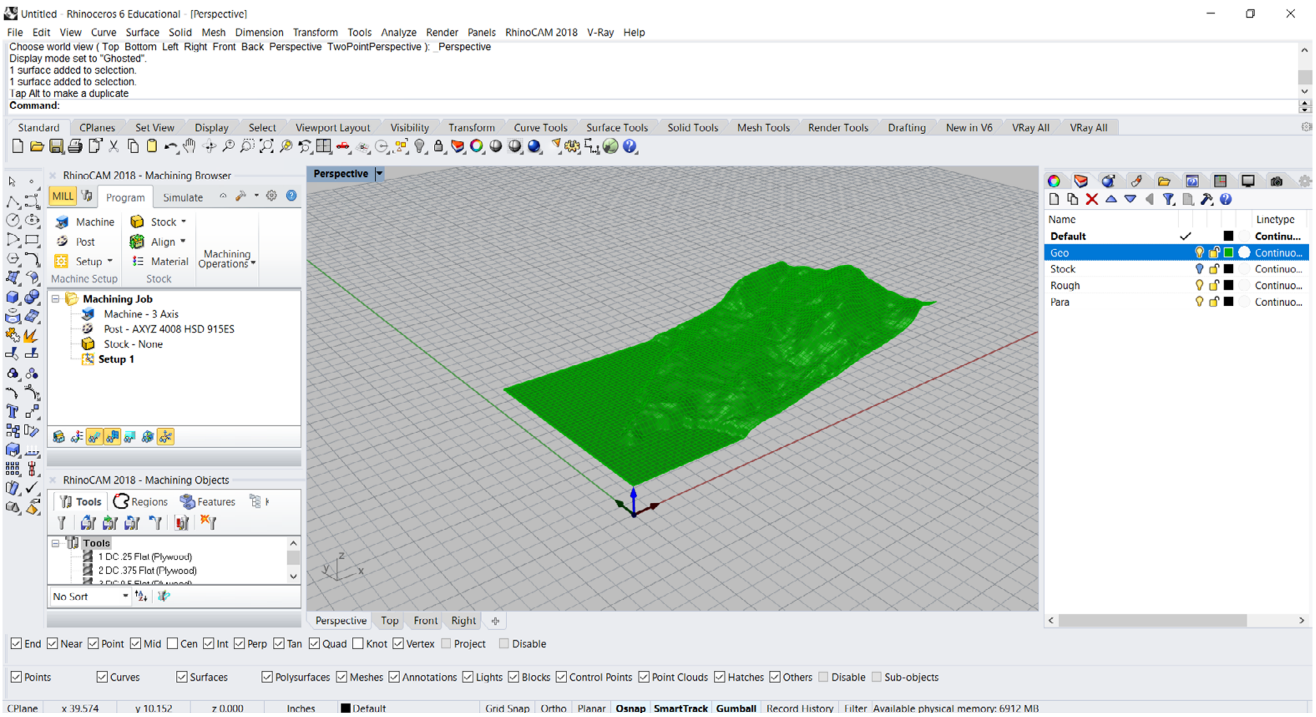
Press generate once all the setup has been completed, the tooling paths with levels will populate the surface.



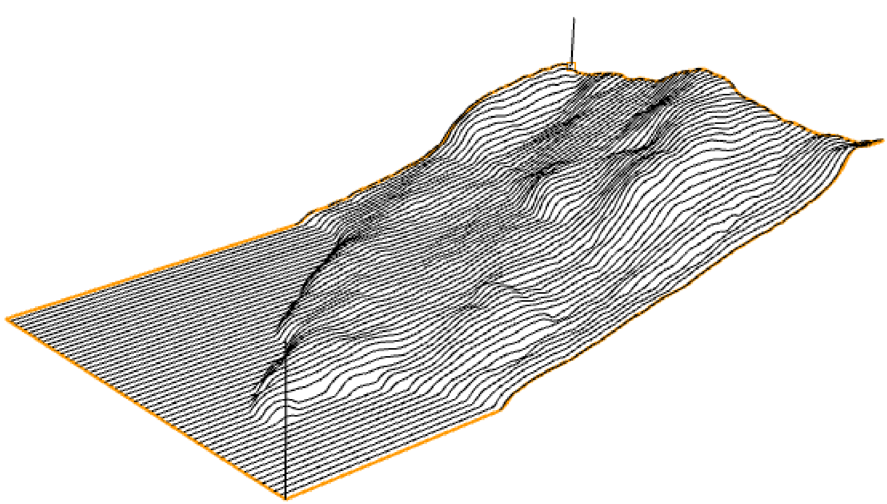
Run the simulation under the simulate tab. If there are collisions or errors in the tooling path setup the will turn the folder icon of the milling operation red after the simulation is complete. Consult a TA if you cannot resolve the issue.

PARALLEL FINISHING

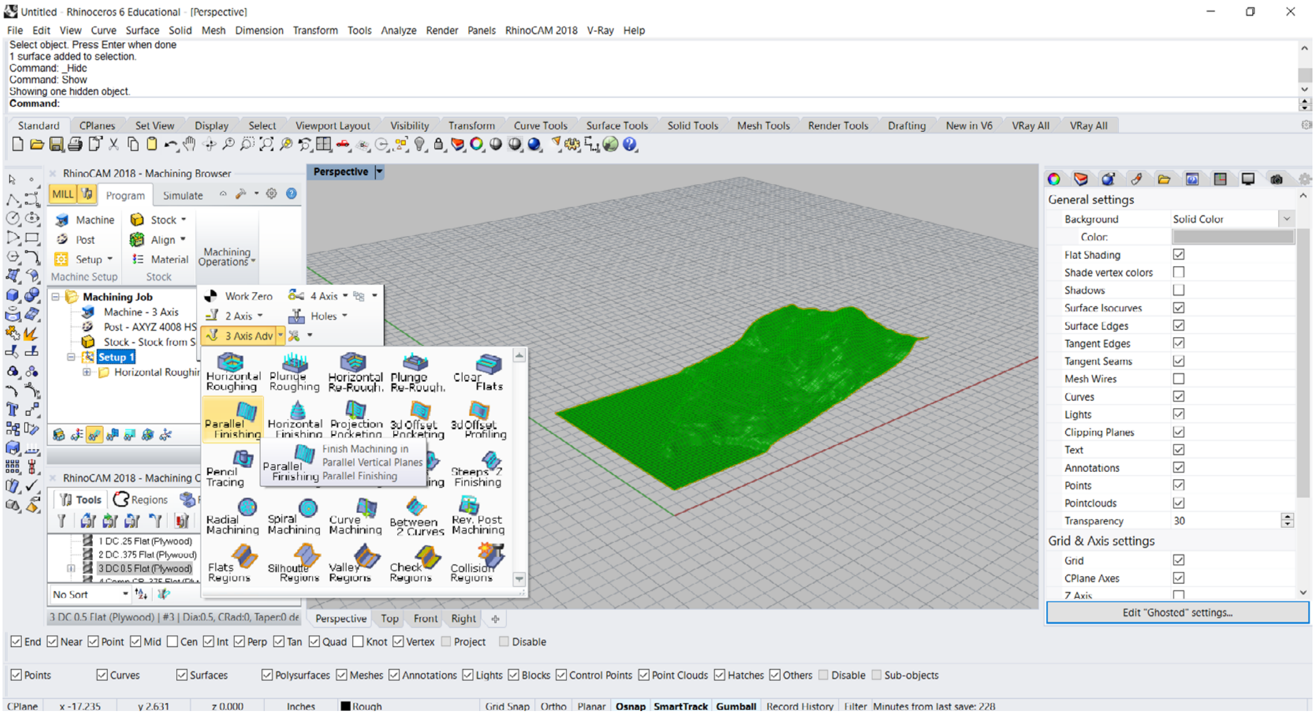
Parallel Finishing operation follows the finish surface with the chosen endmill, the tooling paths create different material effects depending on the endmill chosen. For tight geometry a small endmill is necessary to fit into the terrain or surface, a larger diameter endmill can be used but it will approximate the surface geometry where it cannot fit into areas.



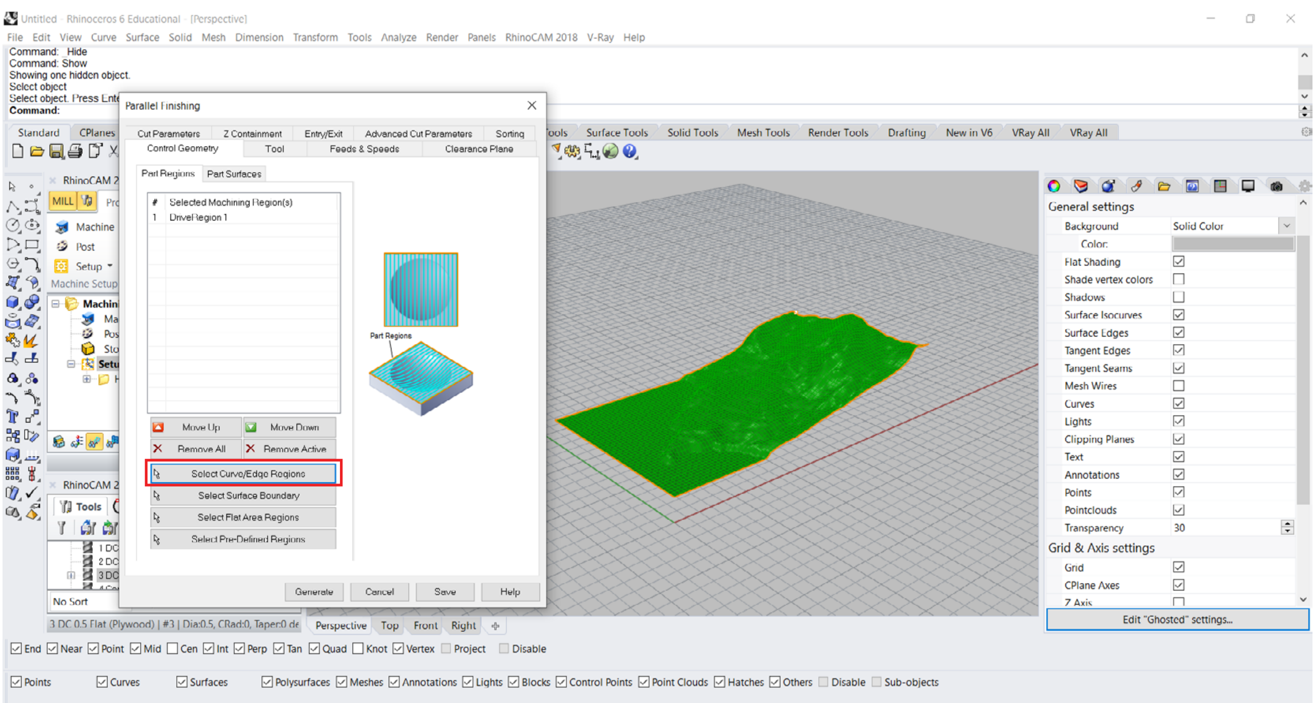
You must do Horizontal Roughing before doing Parallel Finishing.



PARALLEL FINISHING

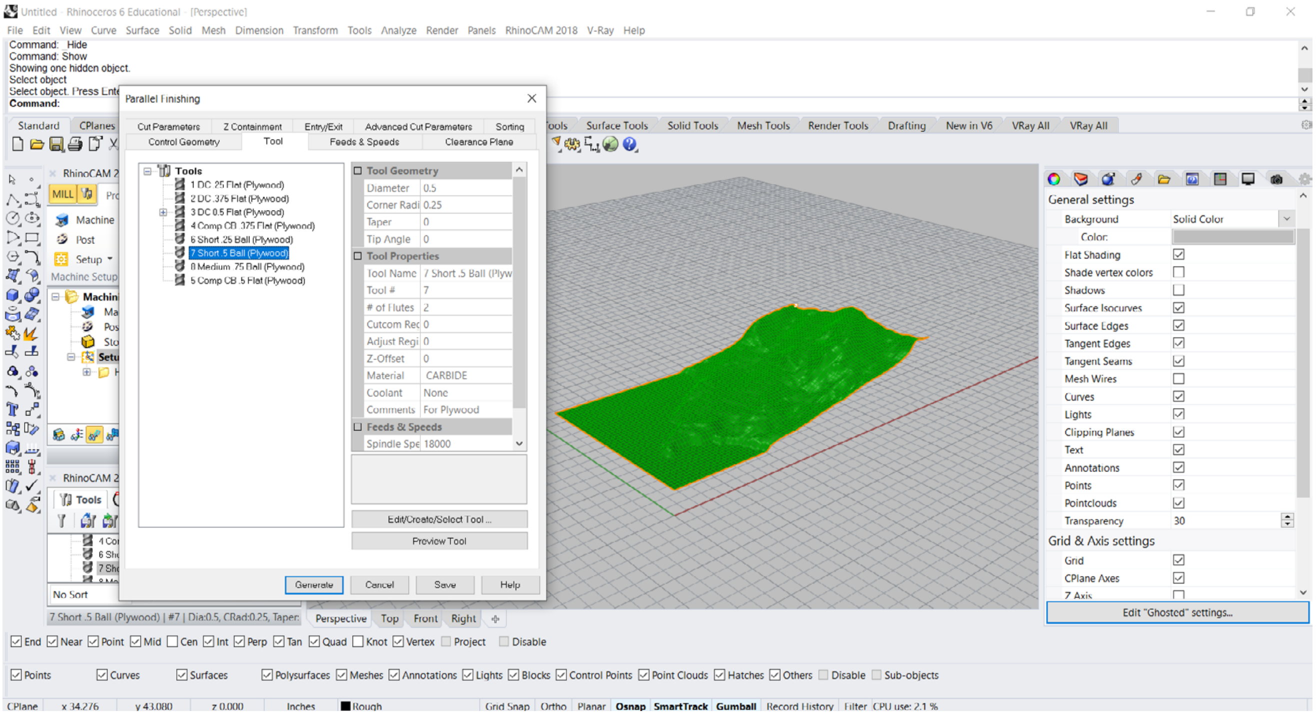


Select Parallel Finishing from the 3 Axis Adv drop down menu.

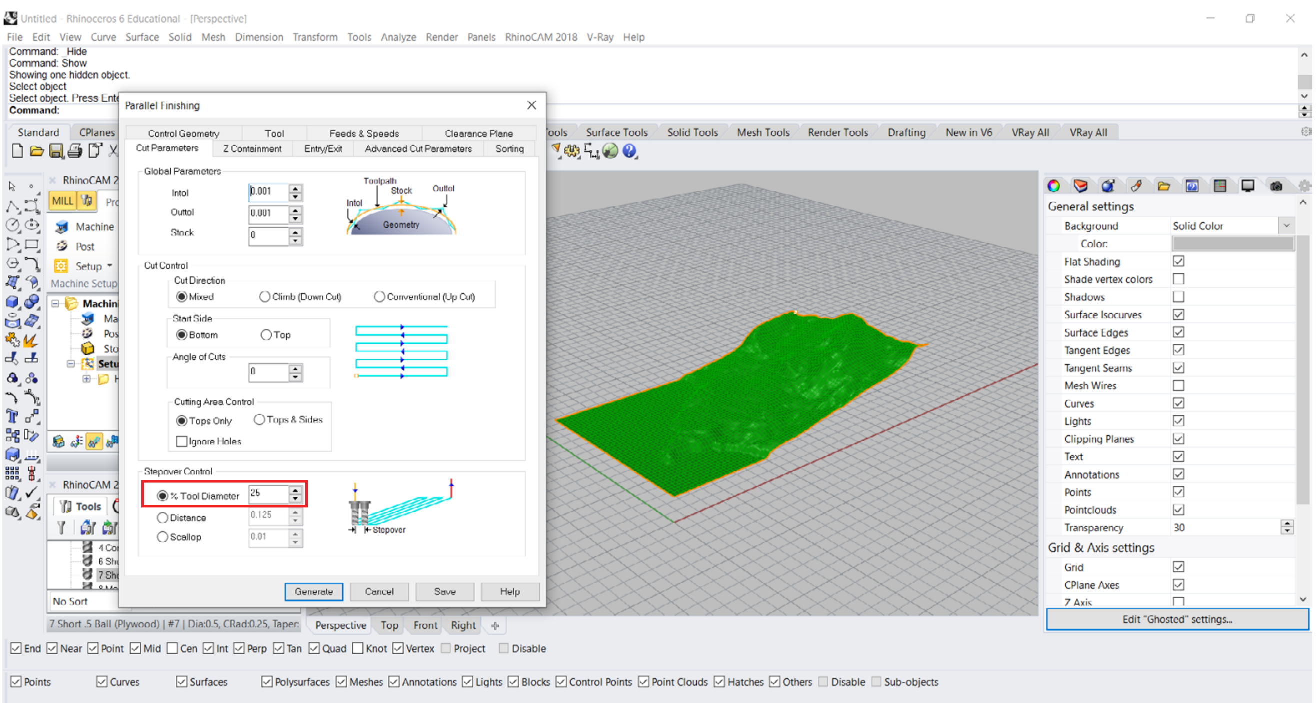


Press Select Curves/Edge Regions and select the curves or regions you want to profile to create your milling paths. This could be the same curve as the Horizontal Roughing.

PARALLEL FINISHING

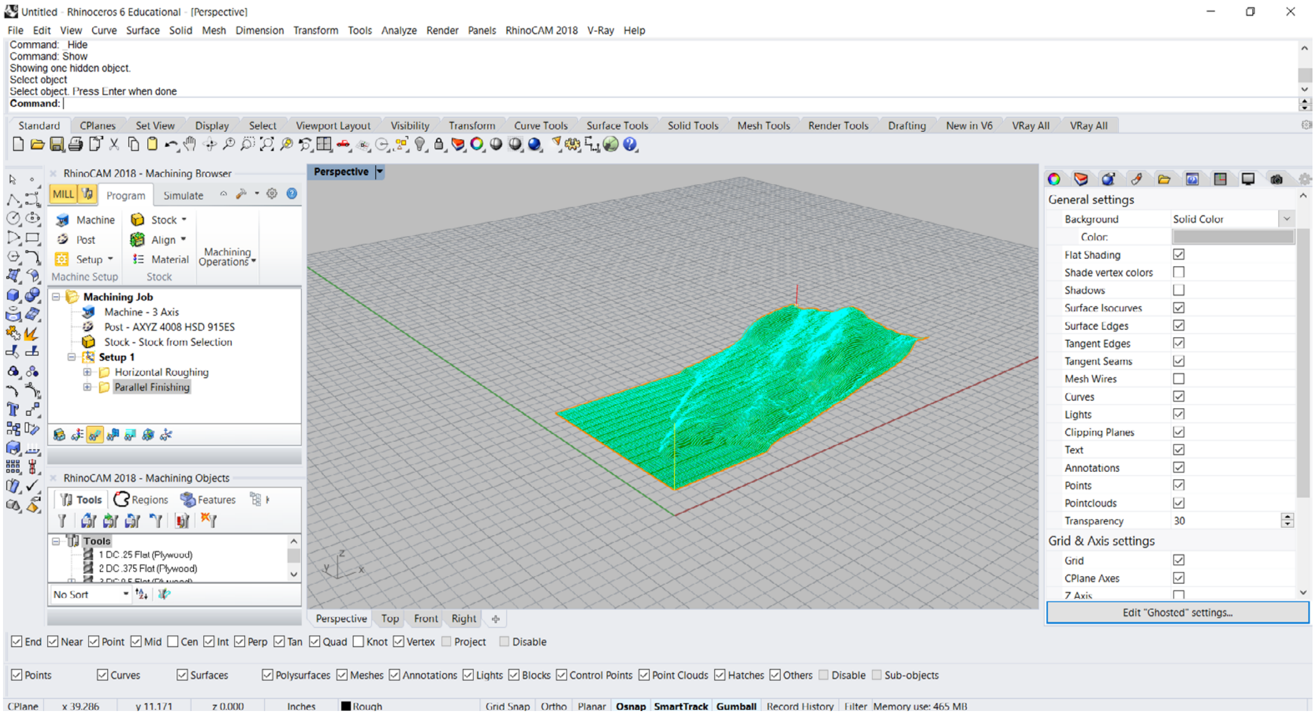


Select the tool to be used for the finishing operation, typically a ball tool will be used but this could be changed dependent on the what kind of finished is desired.

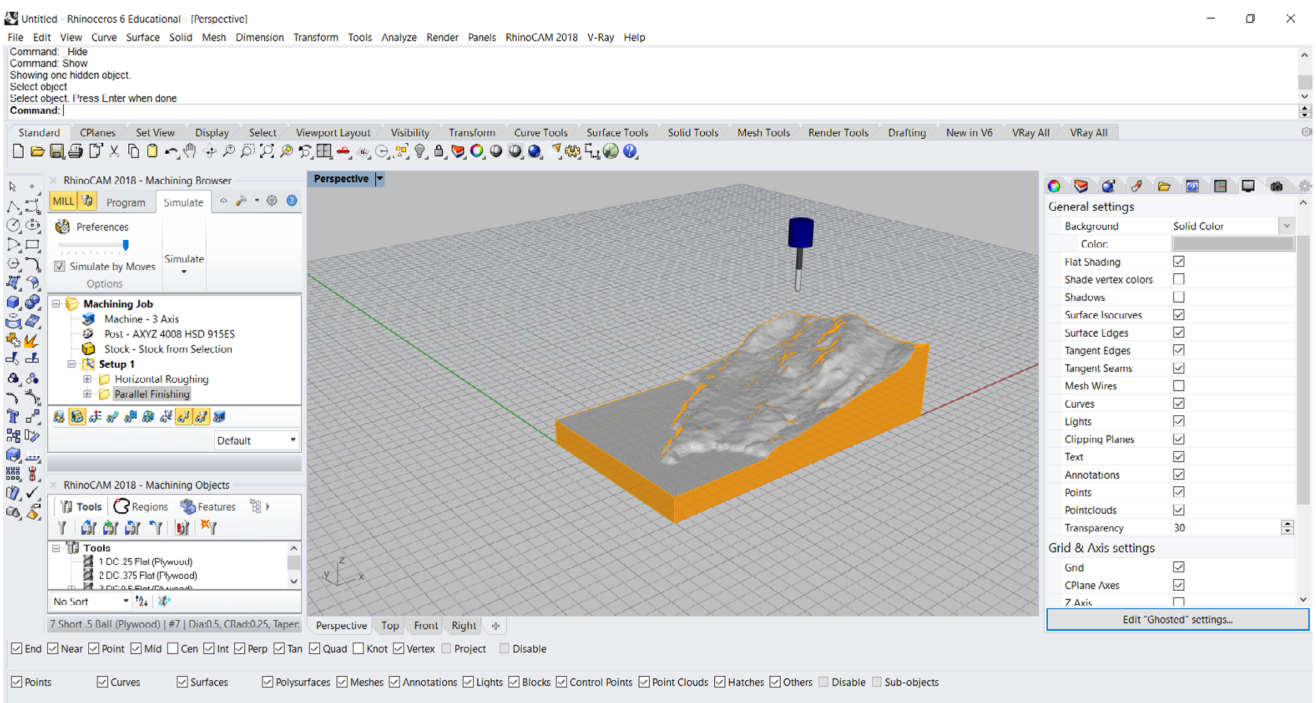


Under Cut Parameters the step over can be set to 25%. This will be the final finish of you model, increasing the step over will decrease the mill time but will result in a larger scallop.

PARALLEL FINISHING



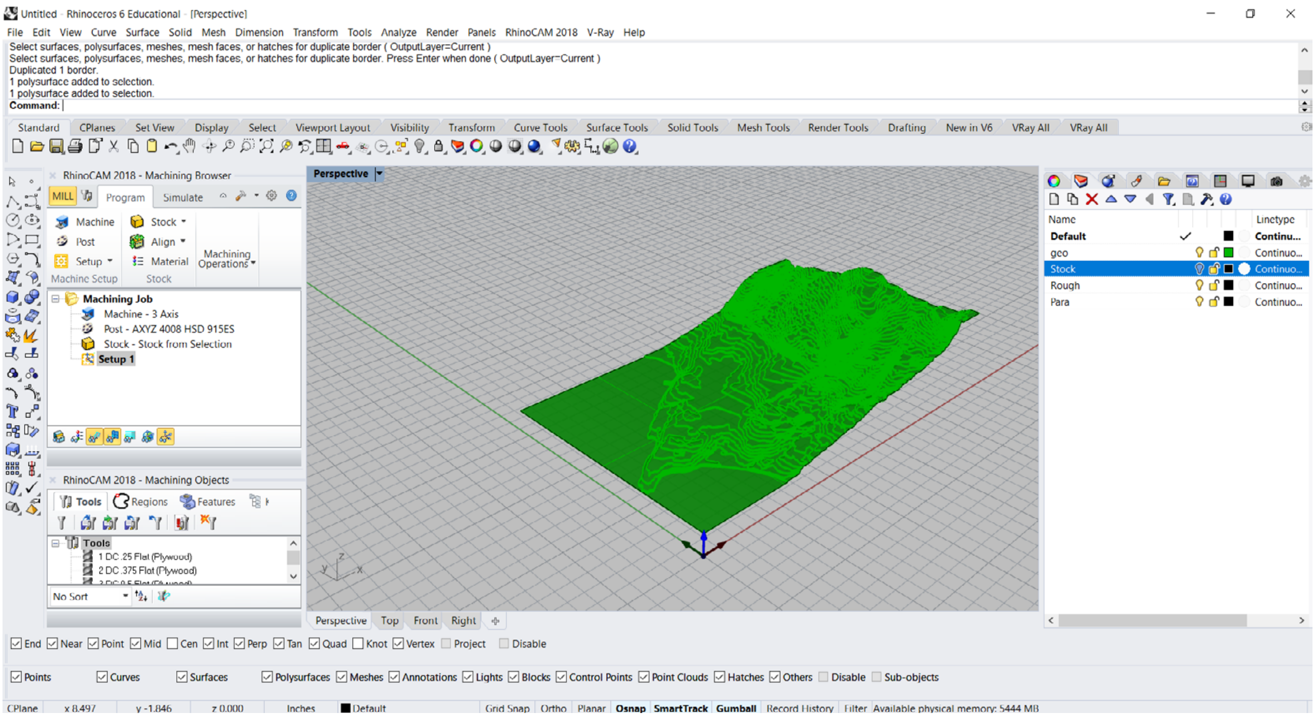
Press generate once all the setup has been completed, the tooling paths with levels will populate the surface.



Run the simulation under the simulate tab. If there are collisions or errors in the tooling path setup the will turn the folder icon of the milling operation red after the simulation is complete. Consult a TA if you cannot resolve the issue.

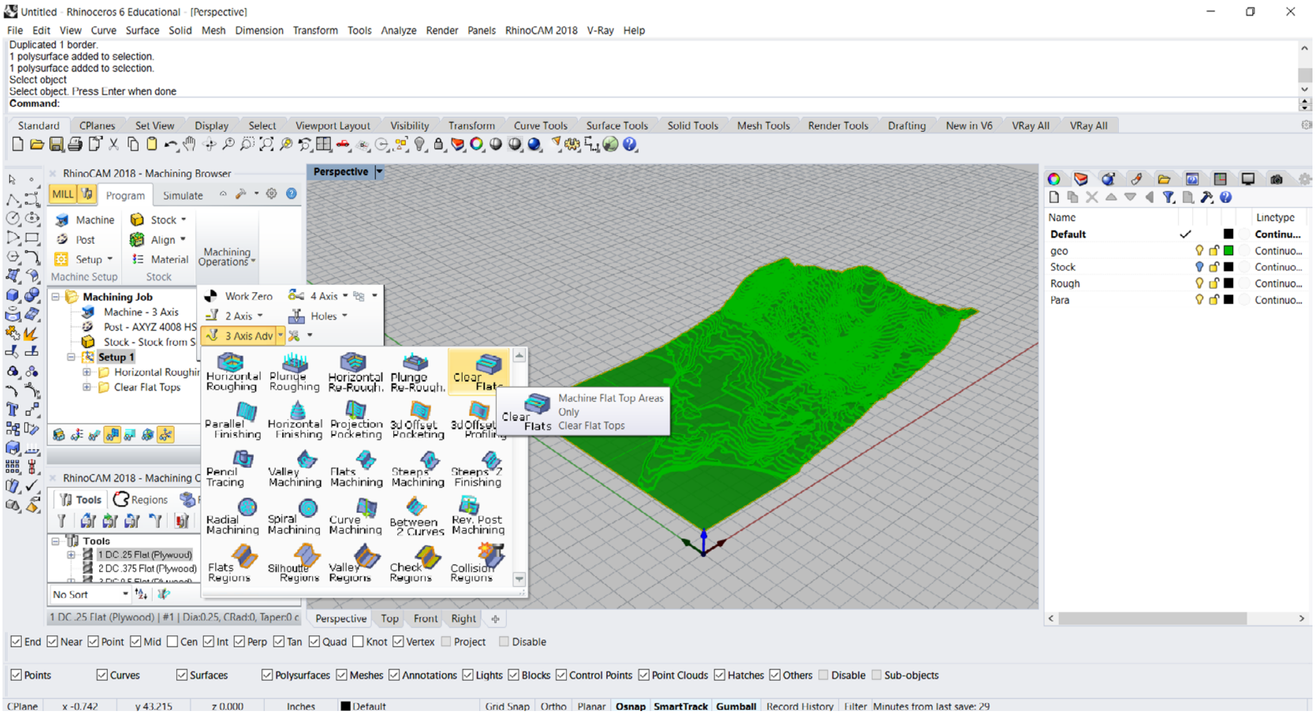
CLEAR FLATS

Clear Flats can be used to for stepped contour models. As it name says it will clear all flat areas. This operation works better than Parallel Finishing for contour models as it will follow along the contour steps creating a clean edge as opposed to scallops which occur when Parallel Finishing goes against the curve. A flat tool is required to create the stepped contour.

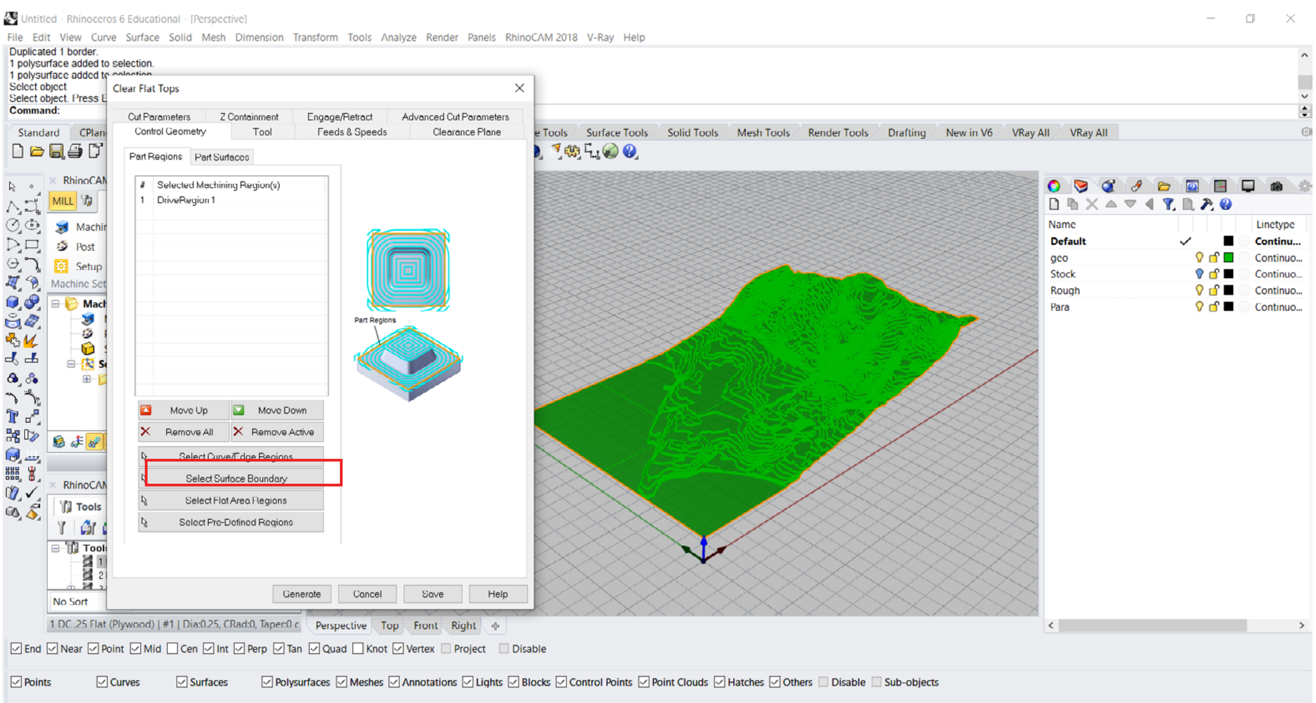


You must do Horizontal Roughing before doing Clear Flats. Clear Flats can take a while to Generate depending on how complex your geometry is.

CLEAR FLATS

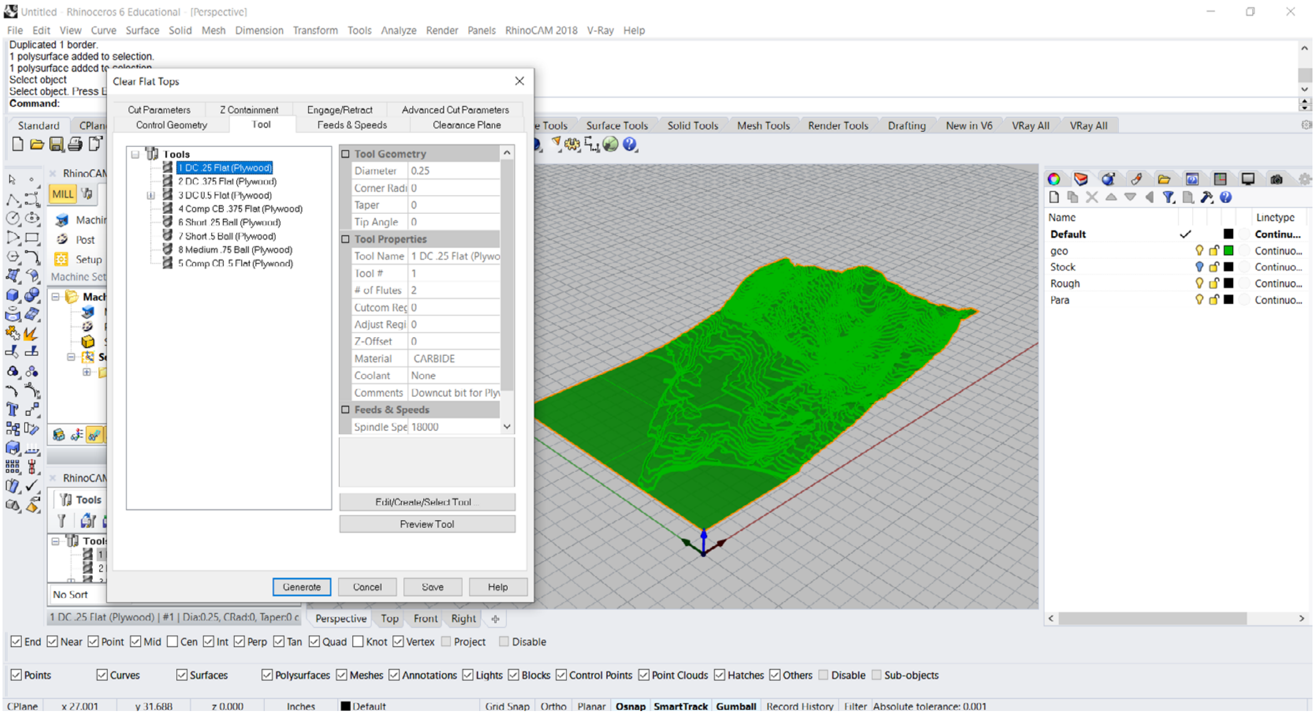


Select Clear Flats from the 3 Axis Adv drop down menu.

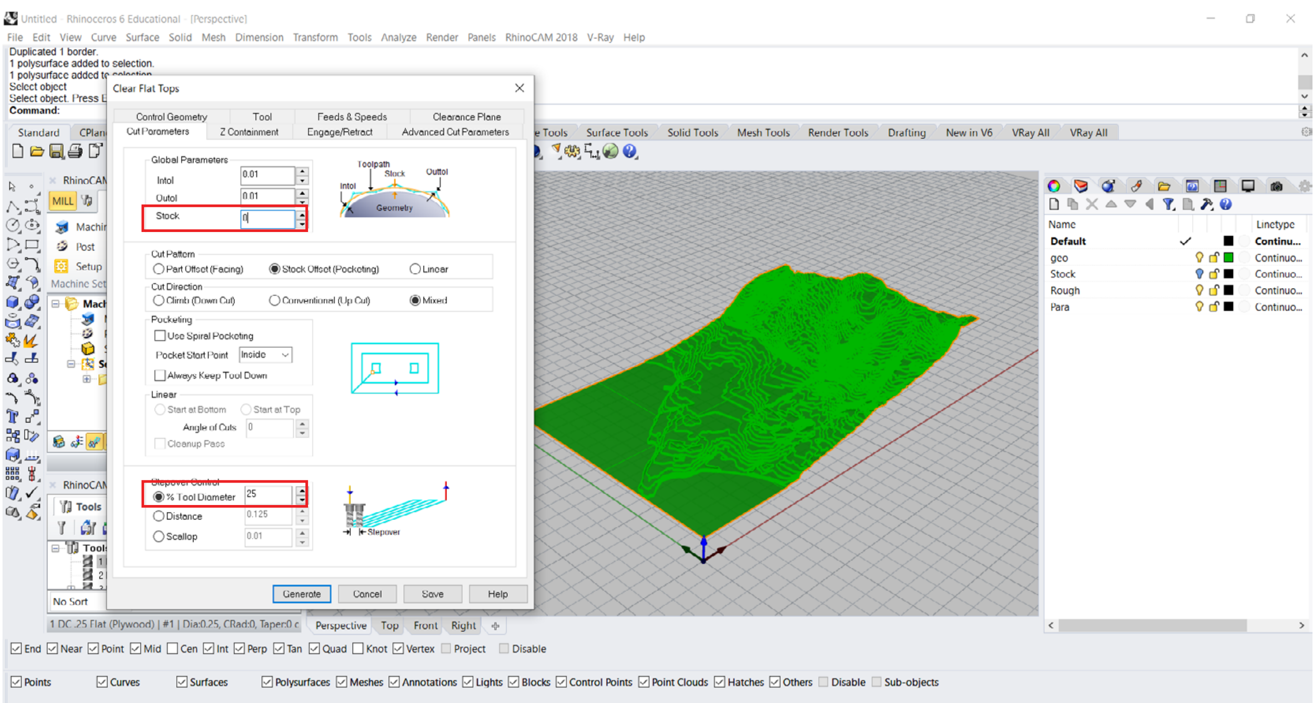


Press Select Curves/Edge Regions and select the curves or regions you want to profile to create your milling paths. This could be the same curve as the Horizontal Roughing.

CLEAR FLATS

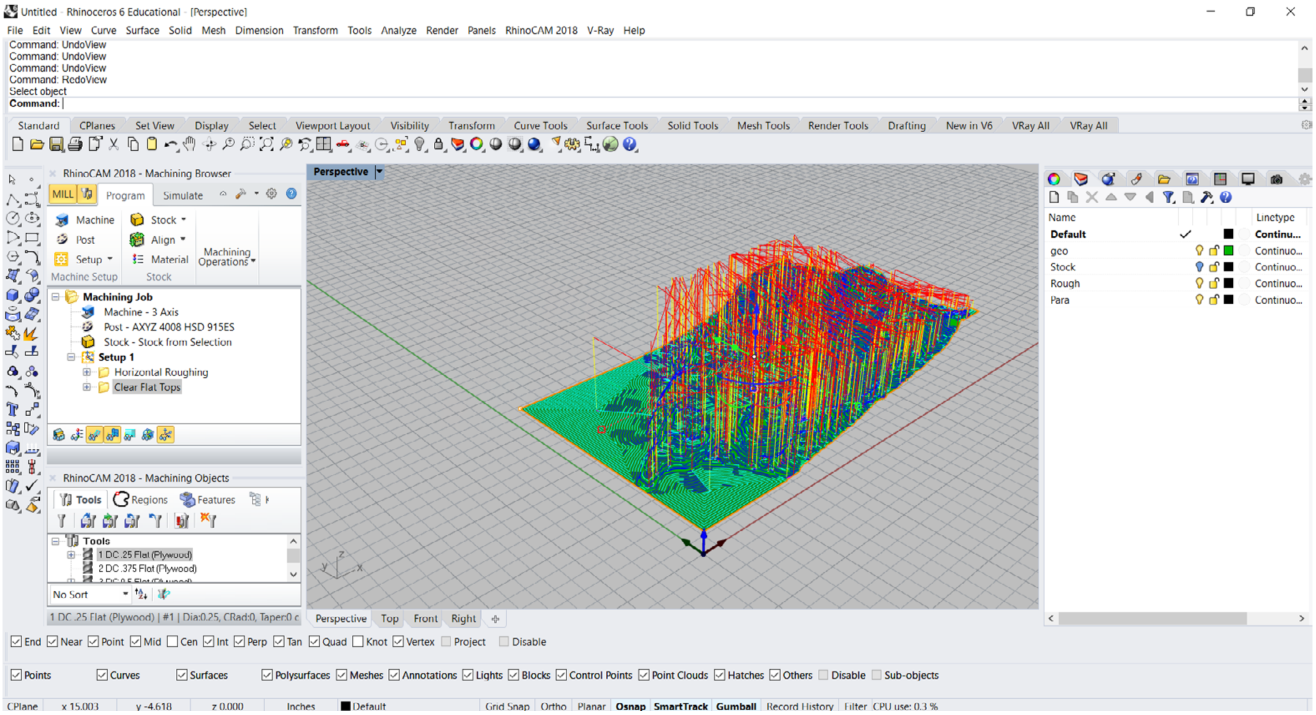


Select the tool to be used for the finishing operation, a flat tool is required to create the stepped contour.

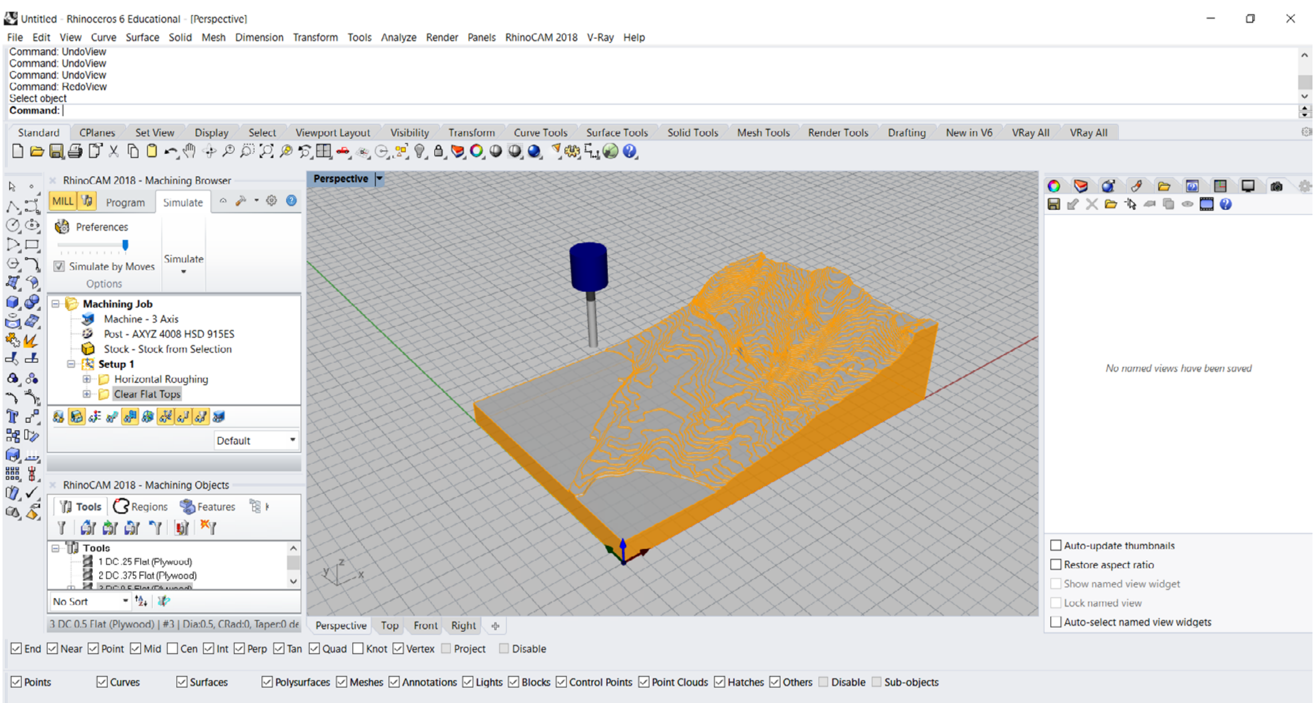


Under Cut Parameters the stock should be set to 0 so it removes all the excess material and the step over can be set to 25%. This will be the final finish of you model, increasing the step over will decrease the mill time but will result in a larger scallop.

CLEAR FLATS



Press generate once all the setup has been completed, the tooling paths with levels will populate the surface.



Run the simulation under the simulate tab. If there are collisions or errors in the tooling path setup the will turn the folder icon of the milling operation red after the simulation is complete. Consult a TA if you cannot resolve the issue.

4 Axis Machining Operations

EQUIPMENT

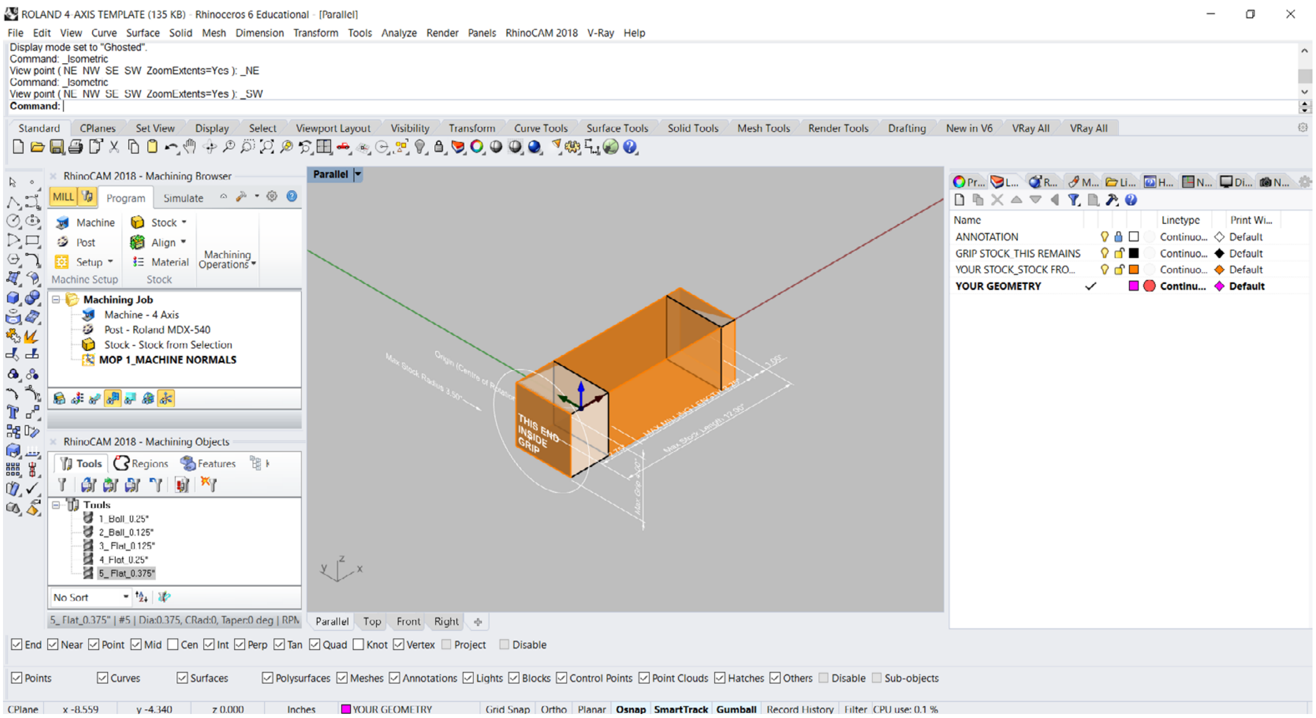
The Roland MDX 540 is a small scale CNC 'lathe' that allows for the fabrication of highly detailed components and geometries that would be impossible to achieve in 3-axis milling. A rotary axis allows the stock material to turn slowly during milling, allowing the tool to gain access to areas that would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

FILE PREPARATION



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 the tool library directly onto your computer drive.

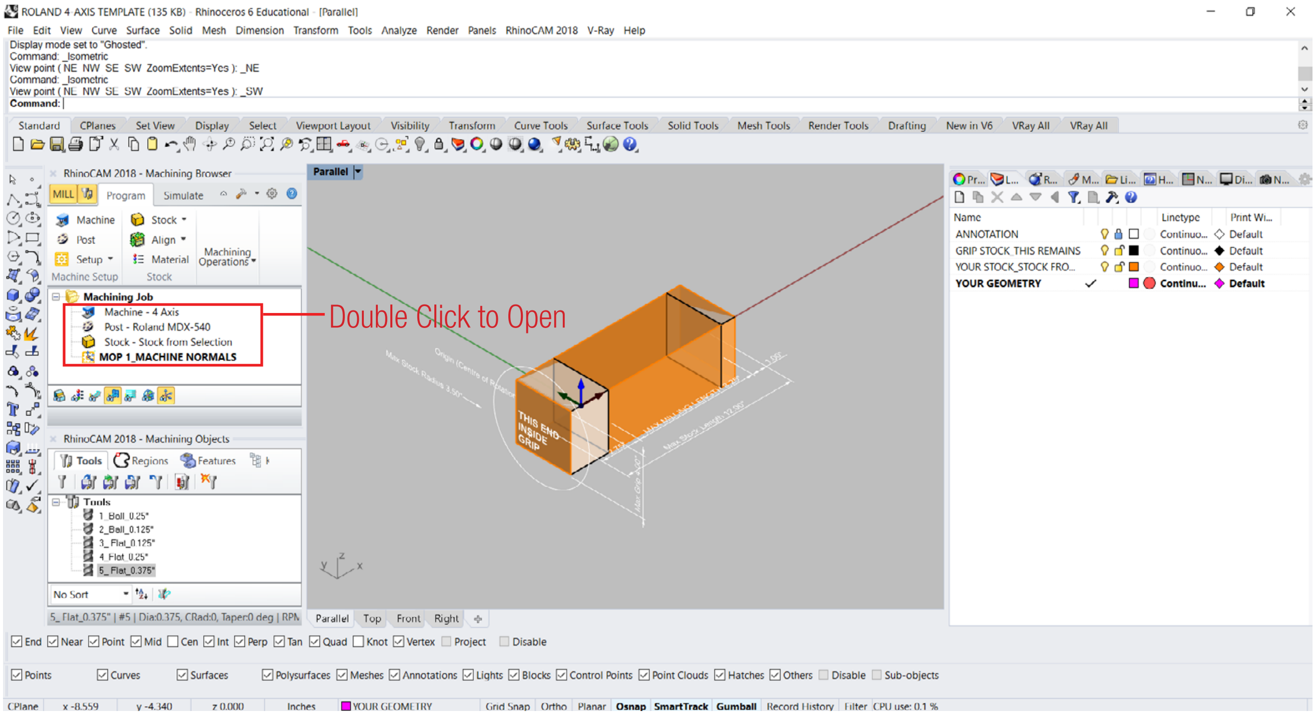
When positioning geometry for the Roland, the x axis acts as the center of rotation. Your geometry should be positioned along the x axis, and should exist only in the positive direction. You will notice that part of the stock material exists on the negative side; this is the material dedicated to the fixturing clamps for positioning, and the tool cannot travel here.

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

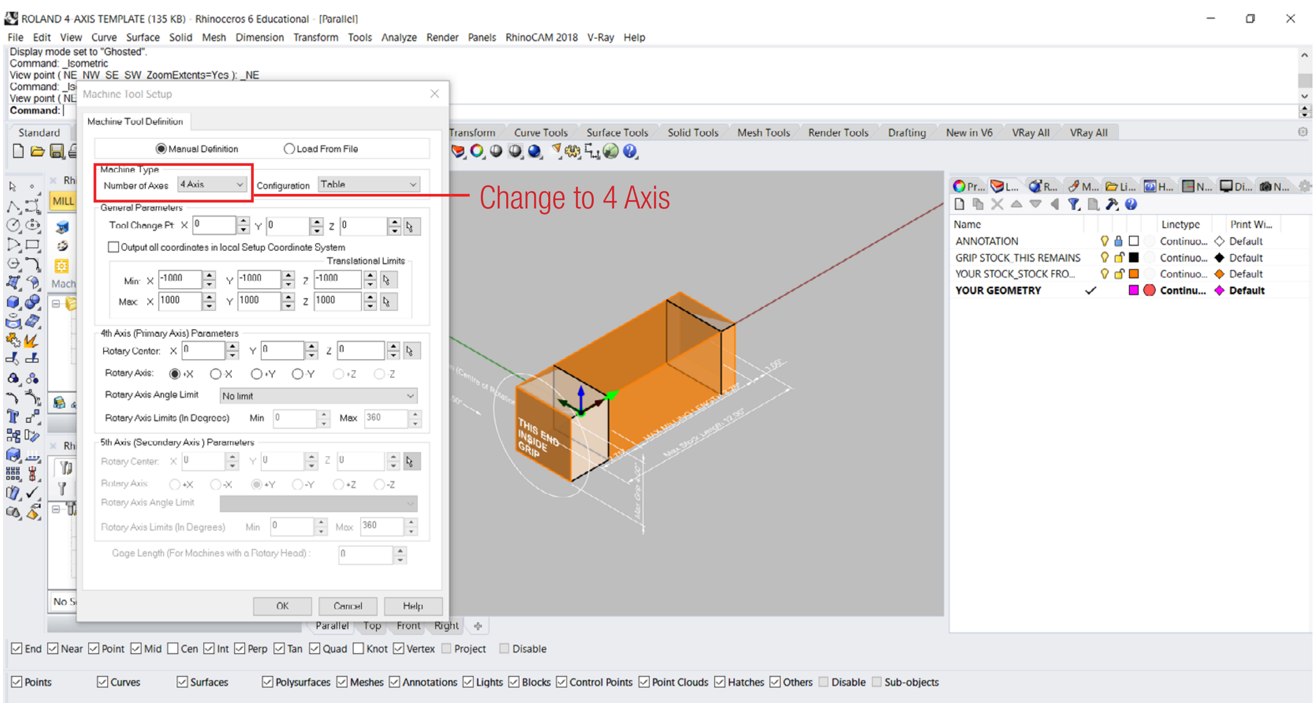
Max stock length = 12” (includes material left clear for fixturing)
Max fixture opening = 4”
Max millable length = 8.25”

All stock must fit within a 7” diameter rotation

FILE PREPARATION

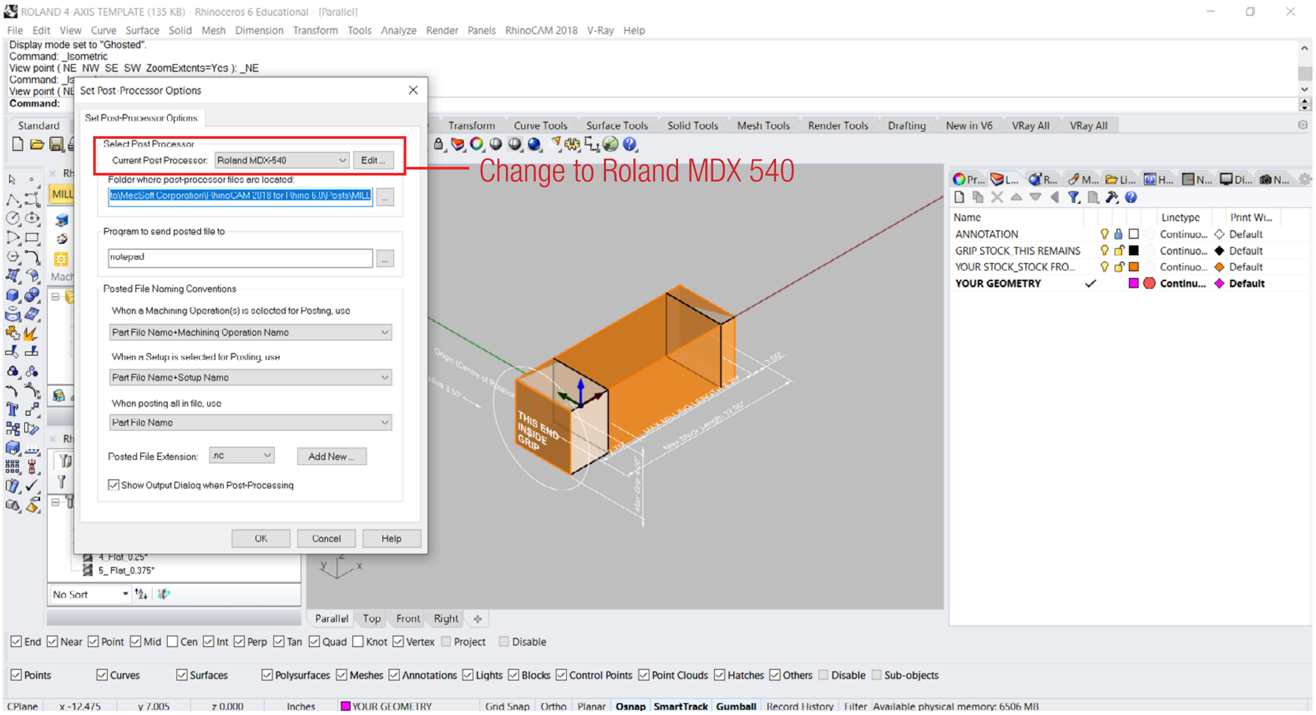


Under Machining Job you will change the Machine from 3 axis to 4 axis, and the Post Processor to the Roland MDX 540. You will also set your ‘stock from selection’ as you would in any other 2 or 3 axis milling

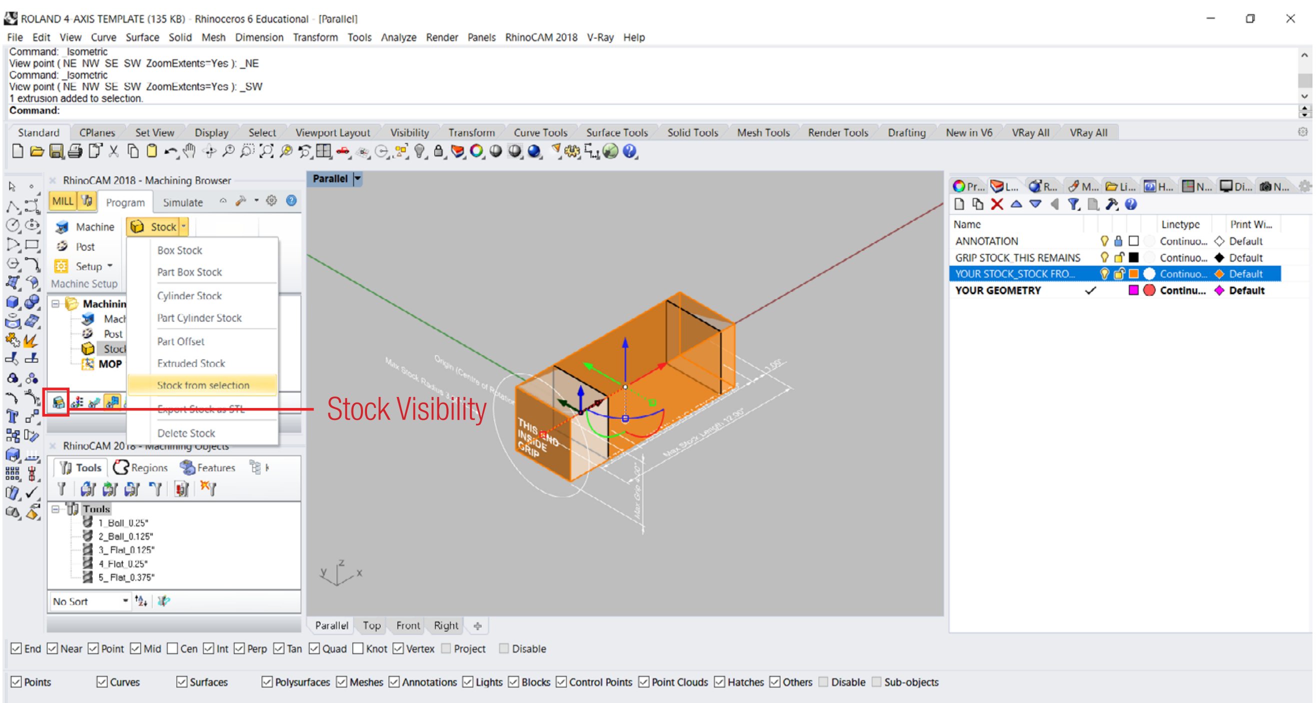


Double Click on 'Machine' and change Number of Axises from 3 Axises to 4.

FILE PREPARATION

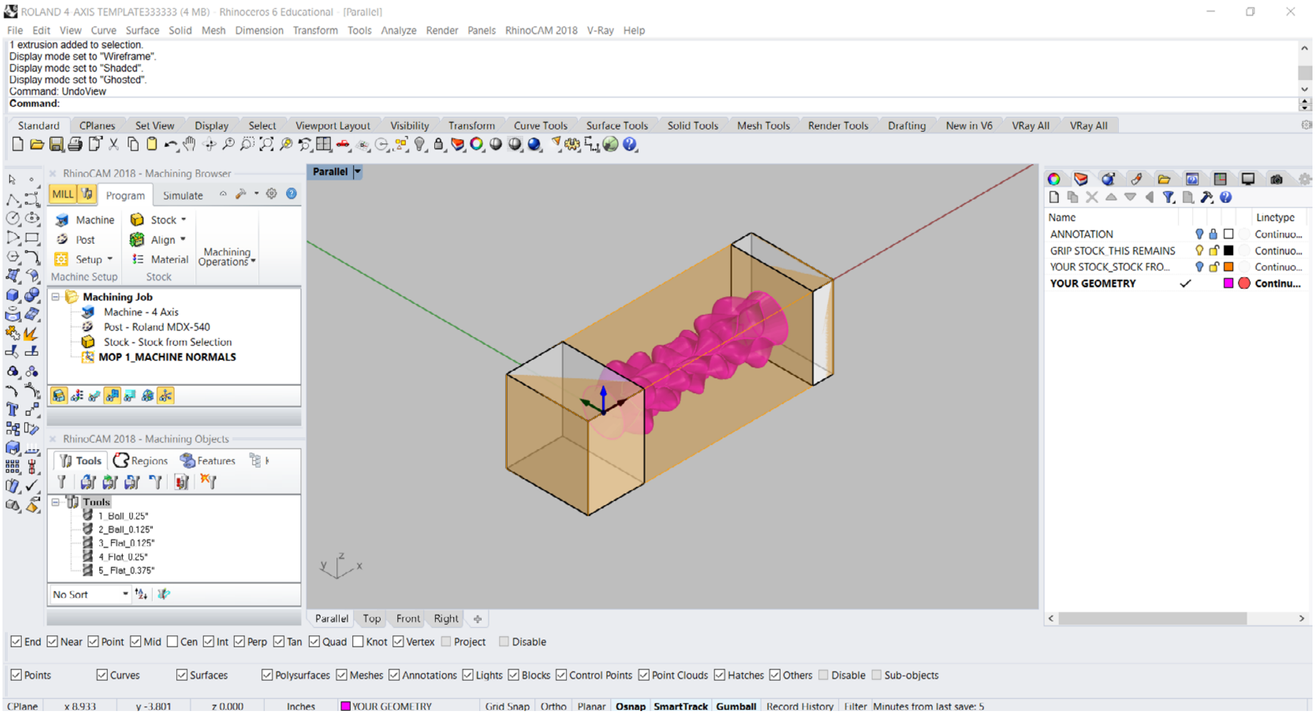


Double click on 'Post' and set the Current Post Processor to 'Roland MDX 540

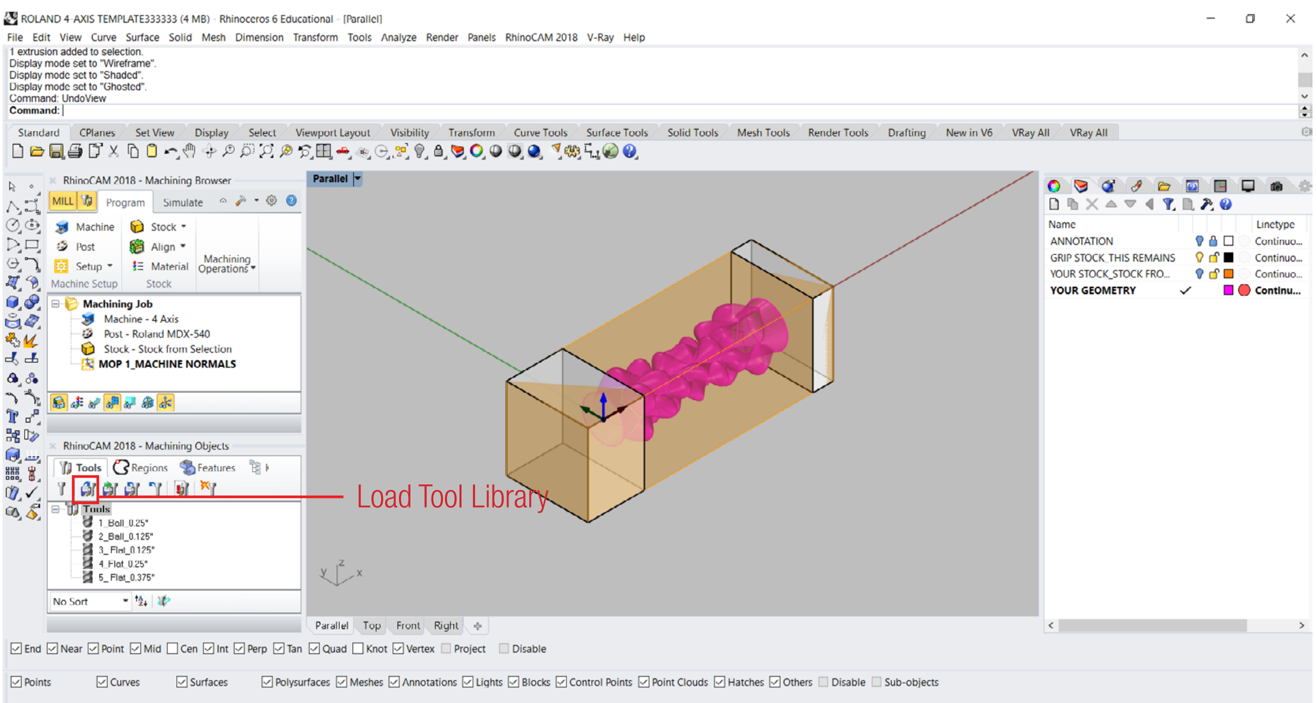


Select your stock material, and click ‘Stock’ and choose ‘stock from selection’. You should now see your stock ghosted in orange when you toggle stock visibility on and off.

FILE PREPARATION

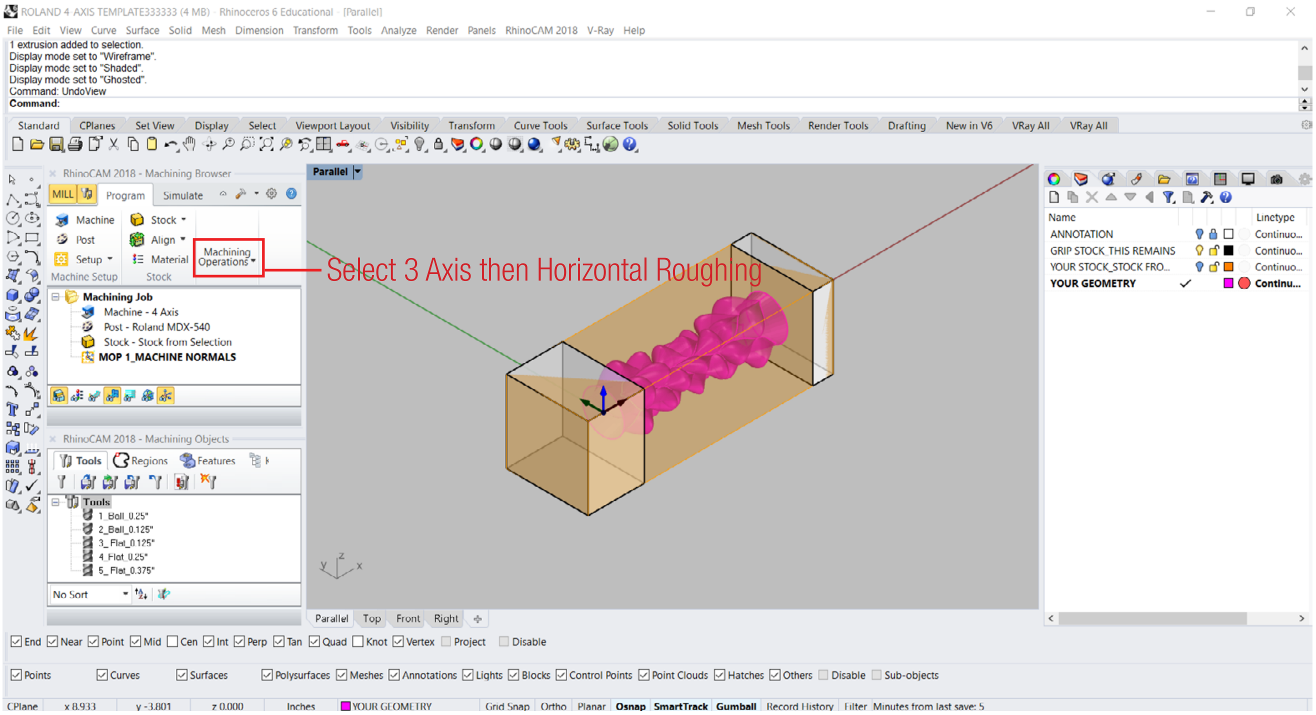


Position your desired geometry within the millable region, centered on the x axis. Your geometry should be fully encased within your stock material, and attach to the leftover fixturing material at either end. Geometry cannot simply float within the bounding volume

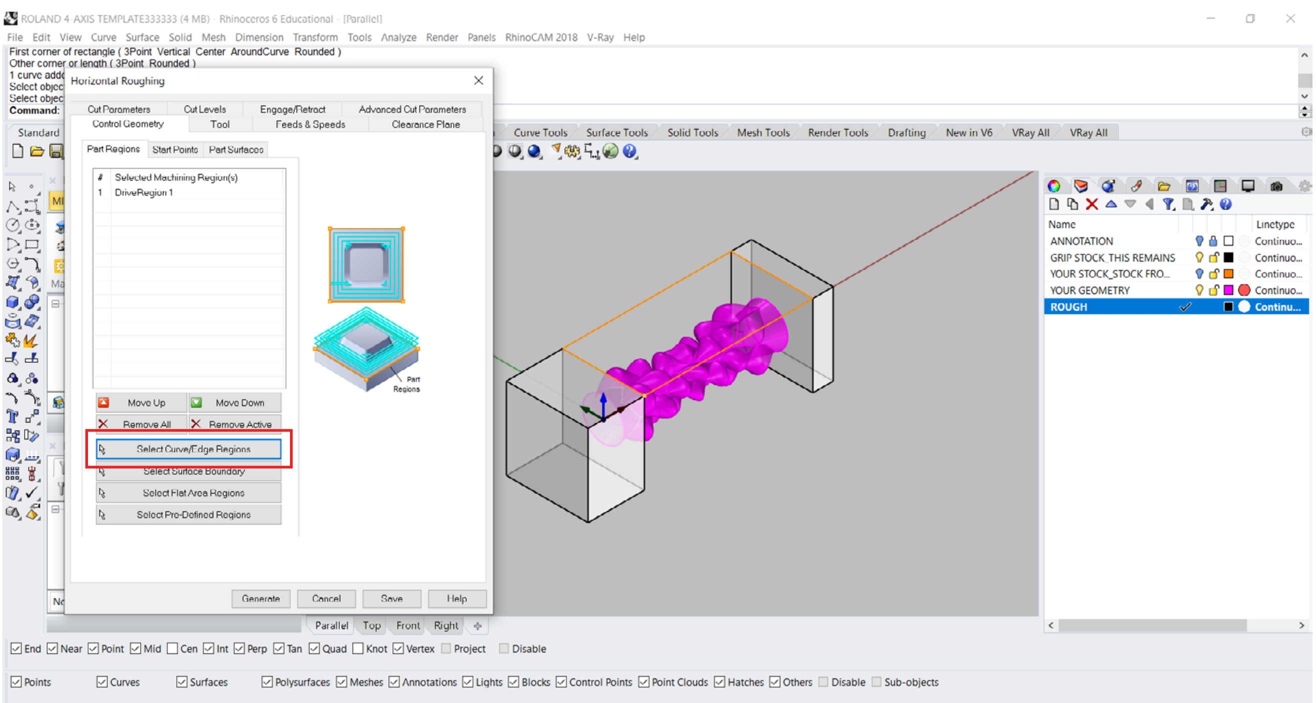


Load the Roland Tool Library you downloaded from the ftp.

FILE PREPARATION

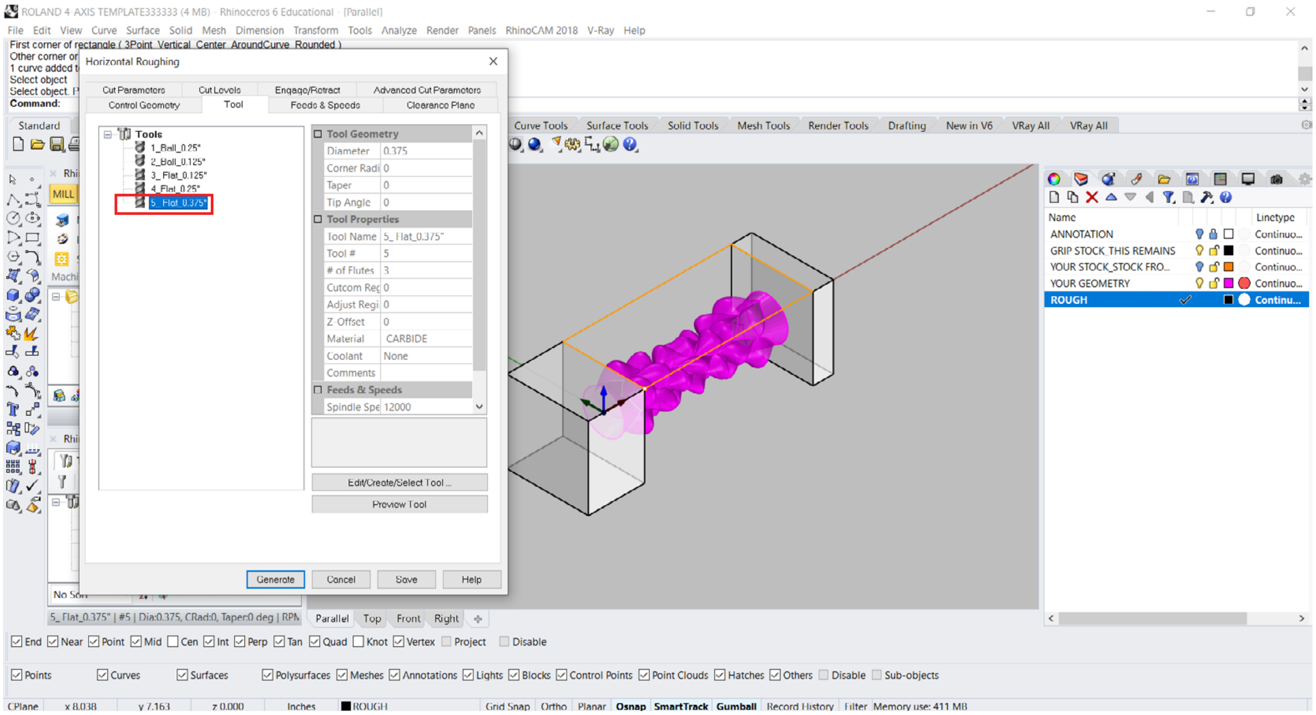


We will use 3 axis milling operations to perform a roughing pass from both top and bottom in order to clear out stock material before finishing passes can be run. Select 3 Axis Adv. operations from the Machining Browser, and choose Horizontal Roughing.

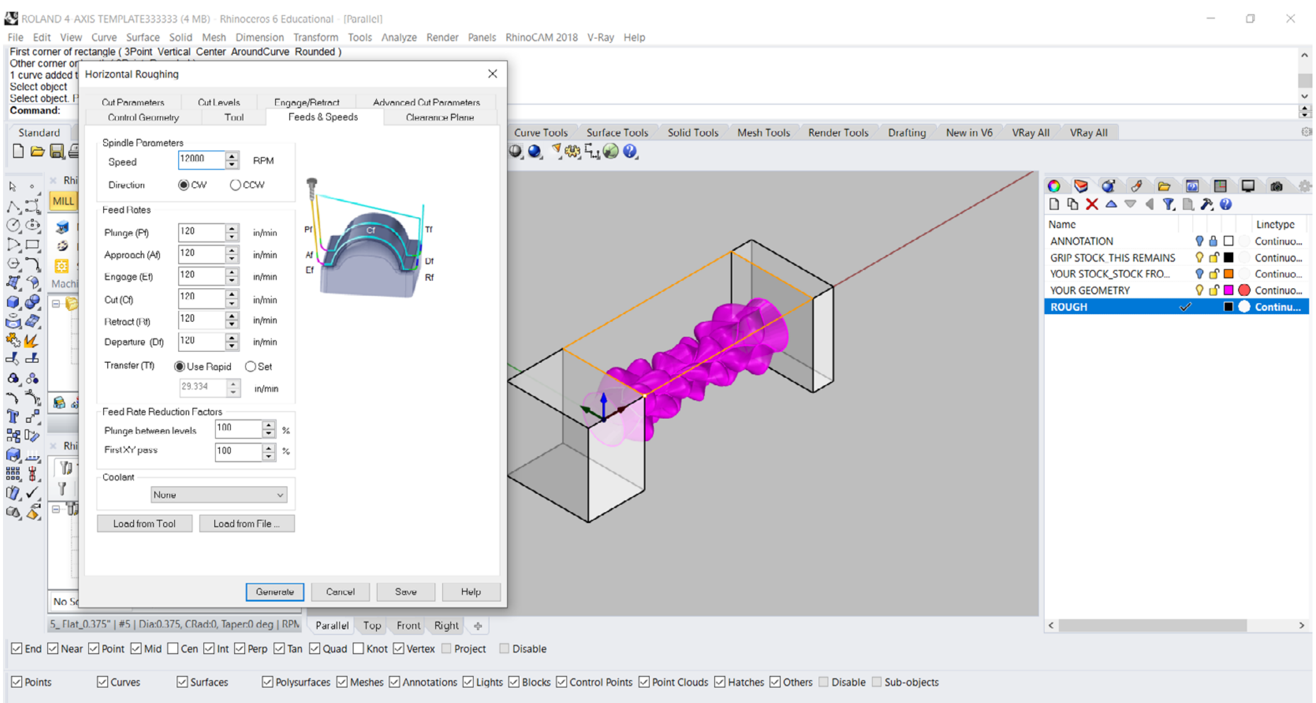


Select a drive region (closed curve) that represents the perimeter of the machinable area.

FILE PREPARATION

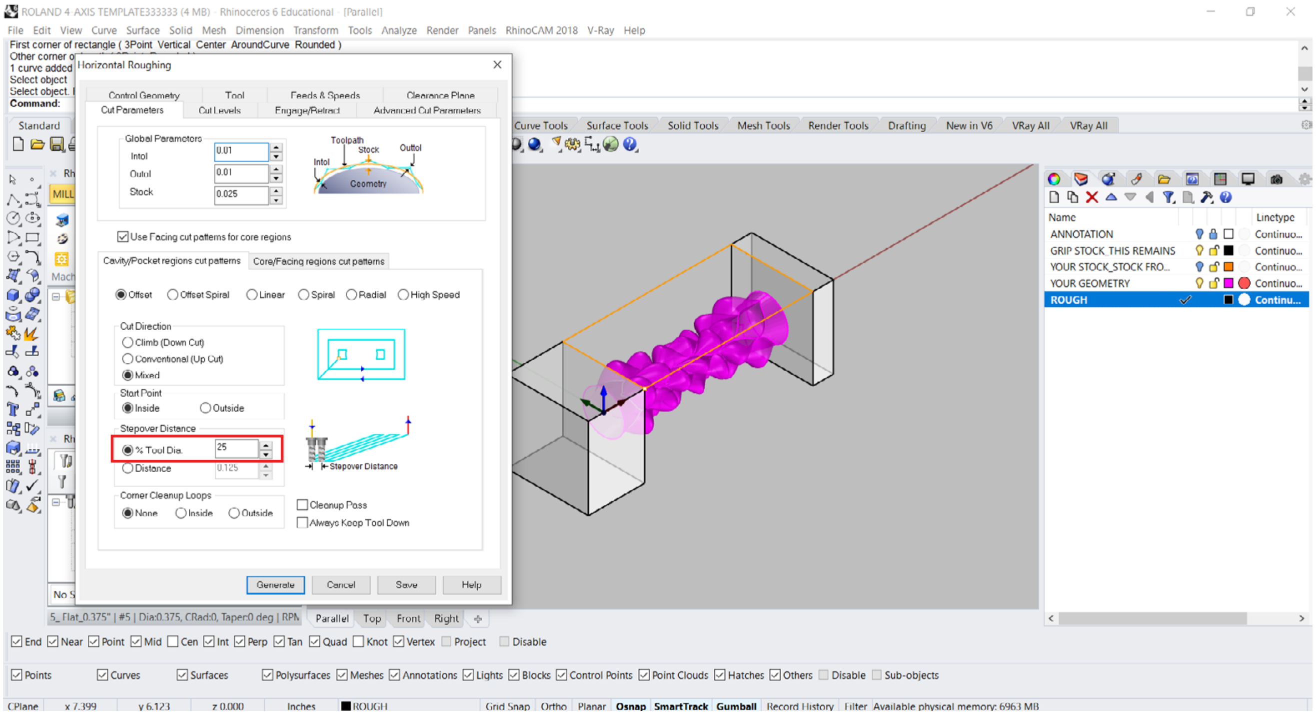


Select the tool you wish to use. In this case, using the largest tool at our disposal will speed up milling rates and allow us to remove more material more quickly.

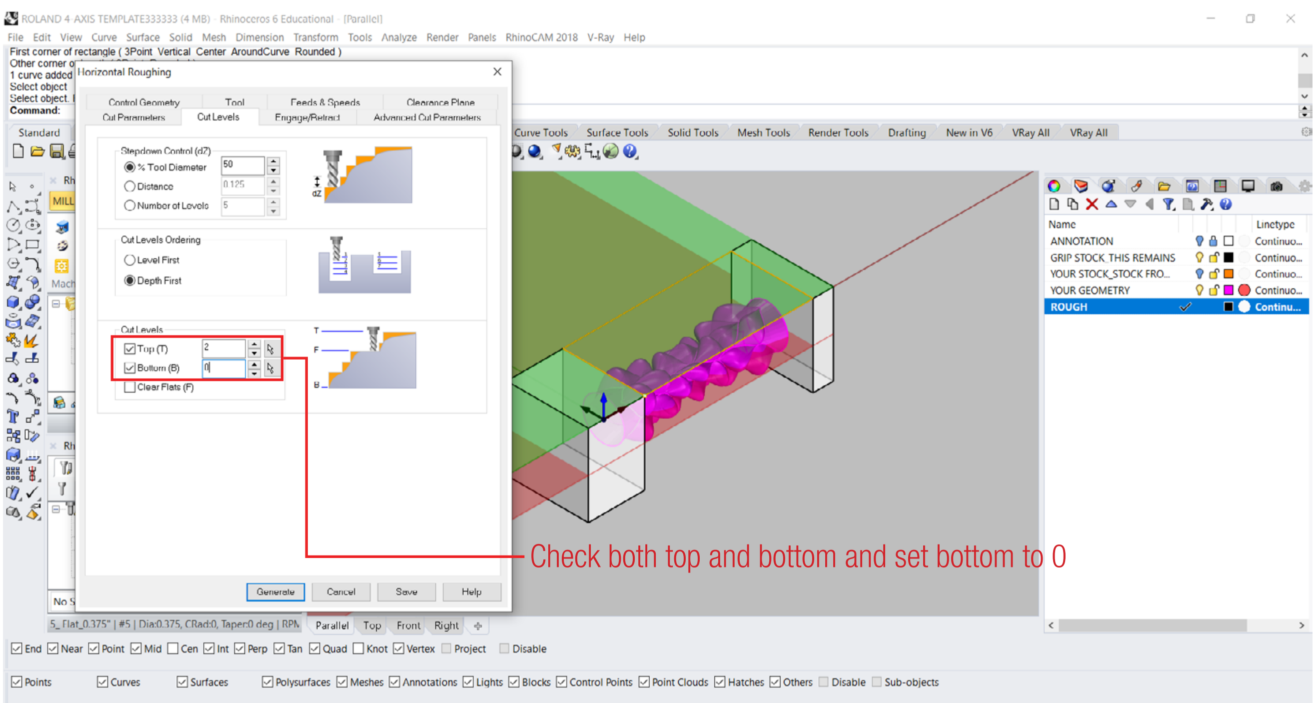


If you have loaded the Roland Tool Library correctly, these values should fill in automatically. A milling TA will review these with you prior to milling, as these values will change for each material. Note that the Roland MDX 540 has a maximum spindle RPM of 12 000 vs the 18 000 of our AXYZ 4008.

FILE PREPARATION

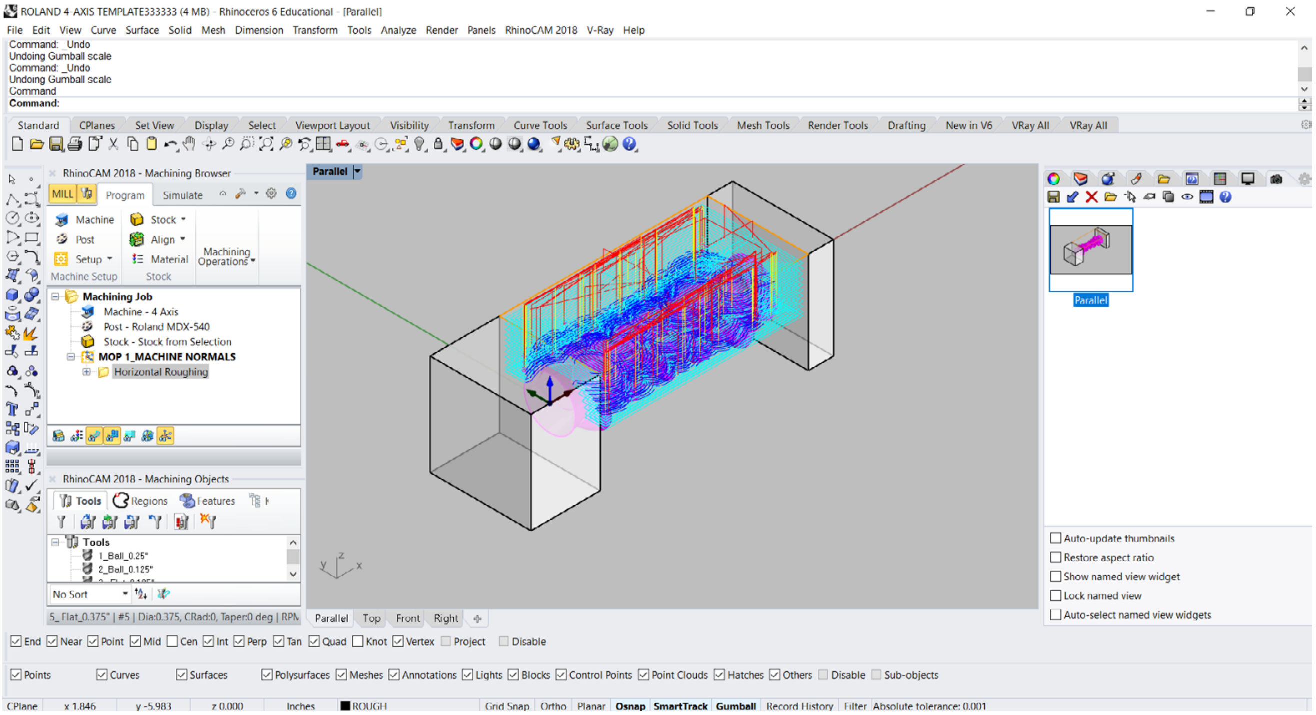


In this case, the default stepover of 25% is desirable. This could be adjusted based on material choice, review this with a milling TA prior to starting your job.

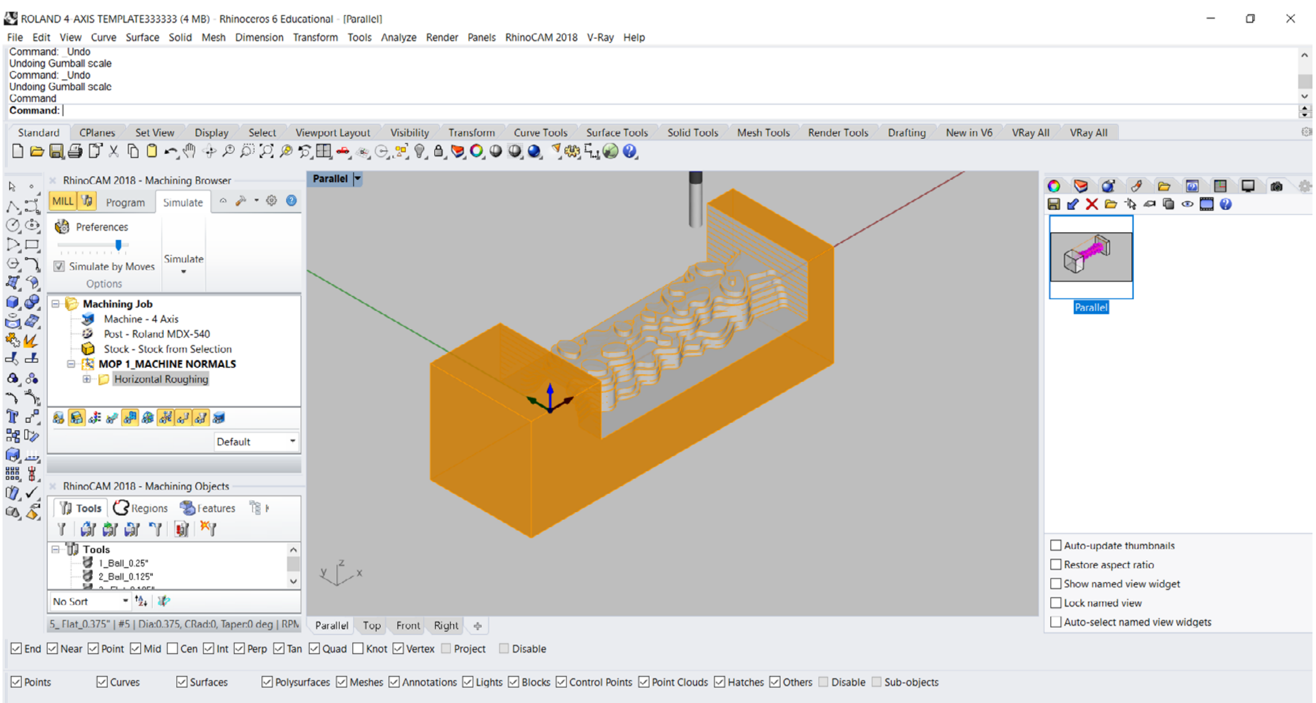


In the Cut Levels Tab, we need to set a manual Top and Bottom values to our cut. We are going to rough from both top and bottom to avoid undercuts, so setting the bottom to zero will have both operations meet in the middle.

FILE PREPARATION

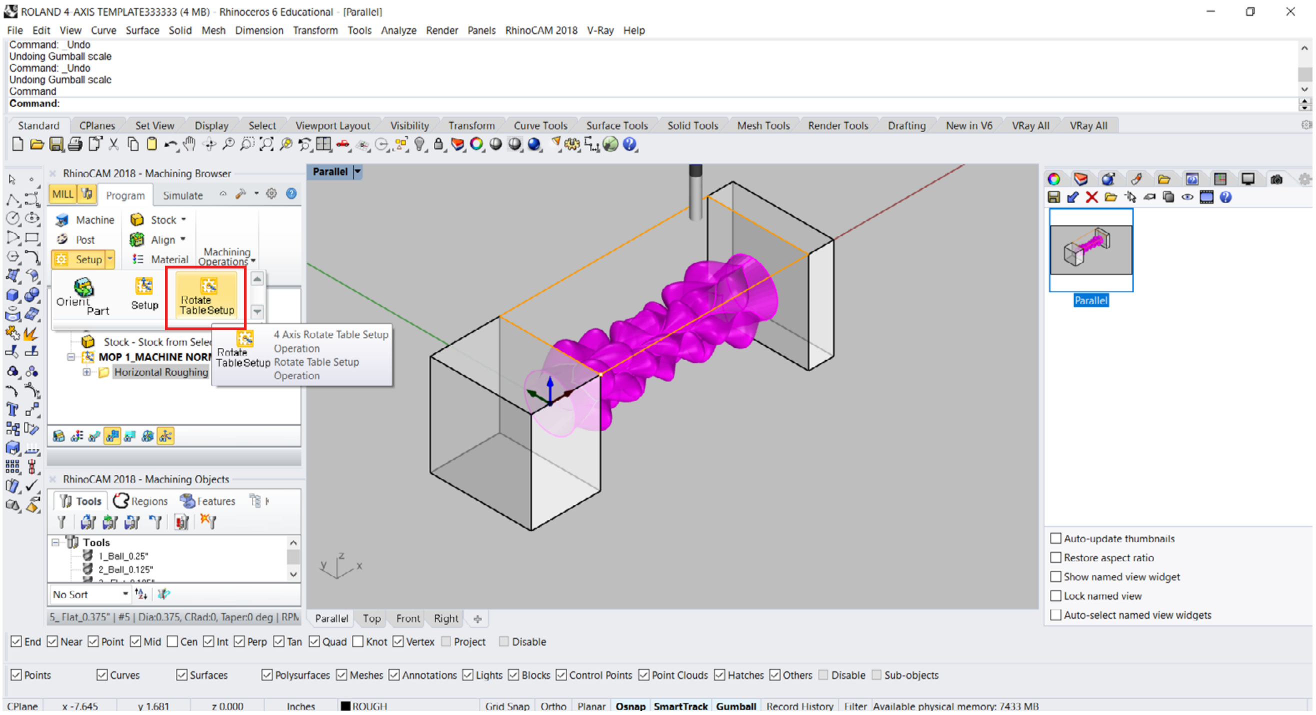


Generate this operation. Toolpaths will be visible as you toggle “ToolPath Visibility” on and off

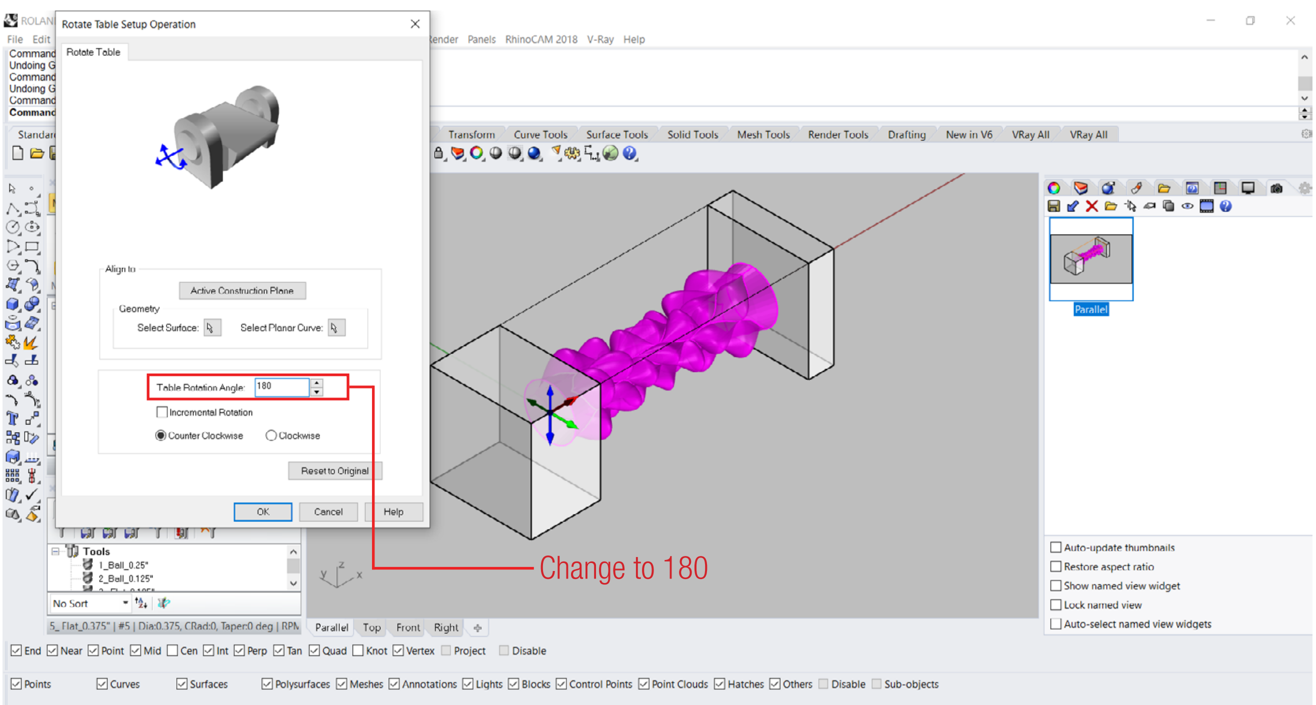


Highlight the Horizontal Roughing operation and click ‘play’ in the Simulations tab. Notice that we have only gone to 0, and the bottom side remains.

FILE PREPARATION

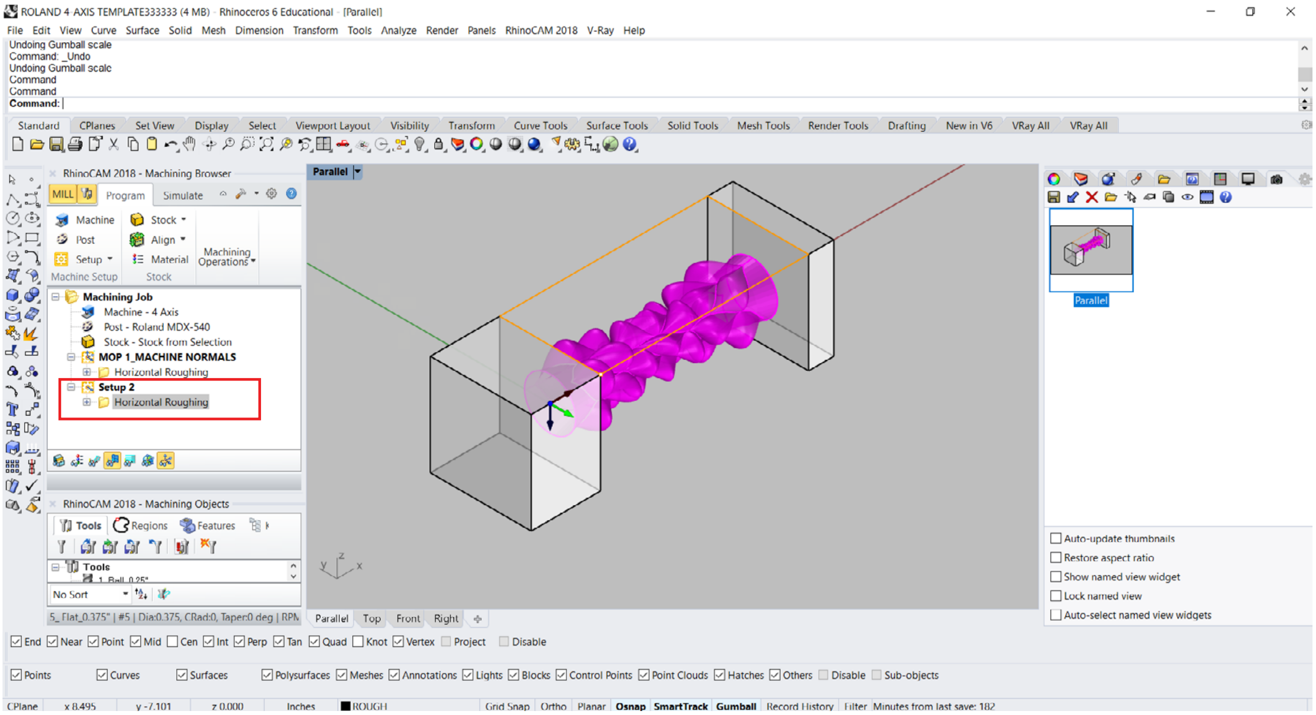


The Machine Operation Set (CSYS Setup) we used for the first roughing pass uses the Rhino axis' to determine part orientation, ie. Rhino Z is 'up'. This is the default setup type. Now, we need a Machine Operation Set that will change the orientation by 180 degrees in order to mill from the bottom. Click setup, then Rotate Table Setup.

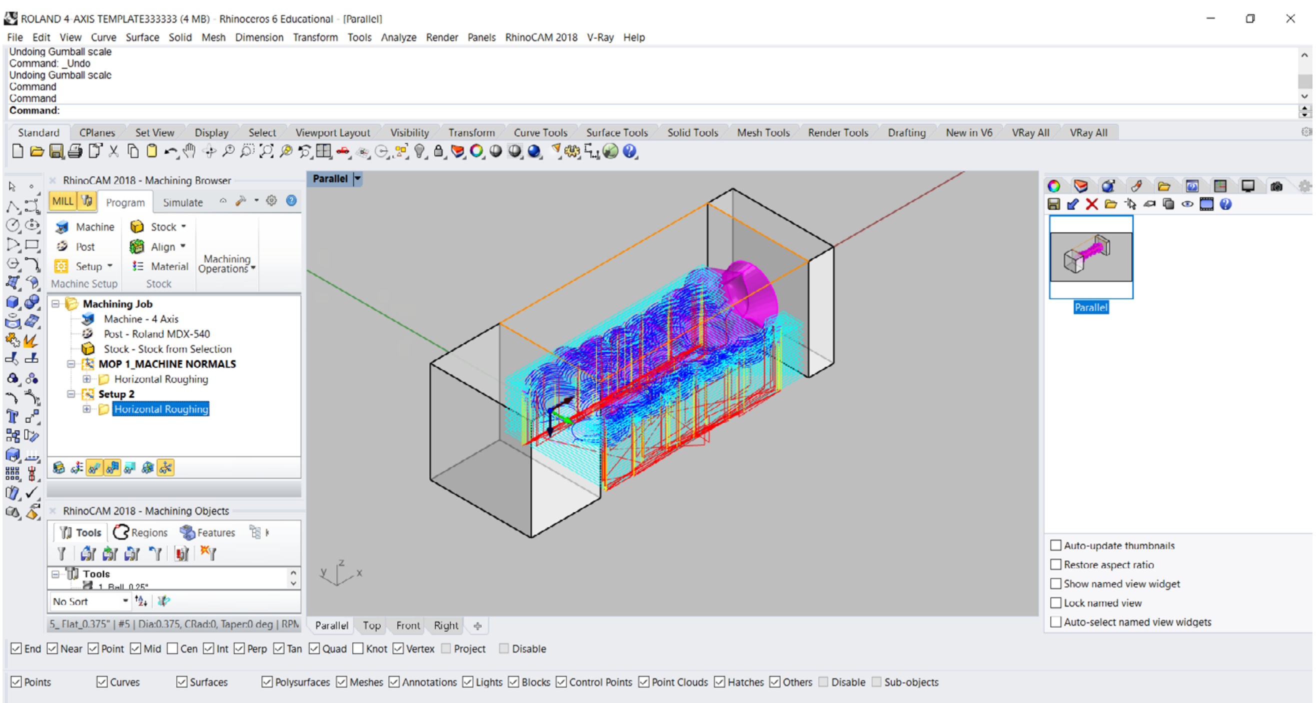


Change the rotation value to 180 degrees. Notice in the Rhino window that the axis arrows at the origin have rotated to match.

FILE PREPARATION

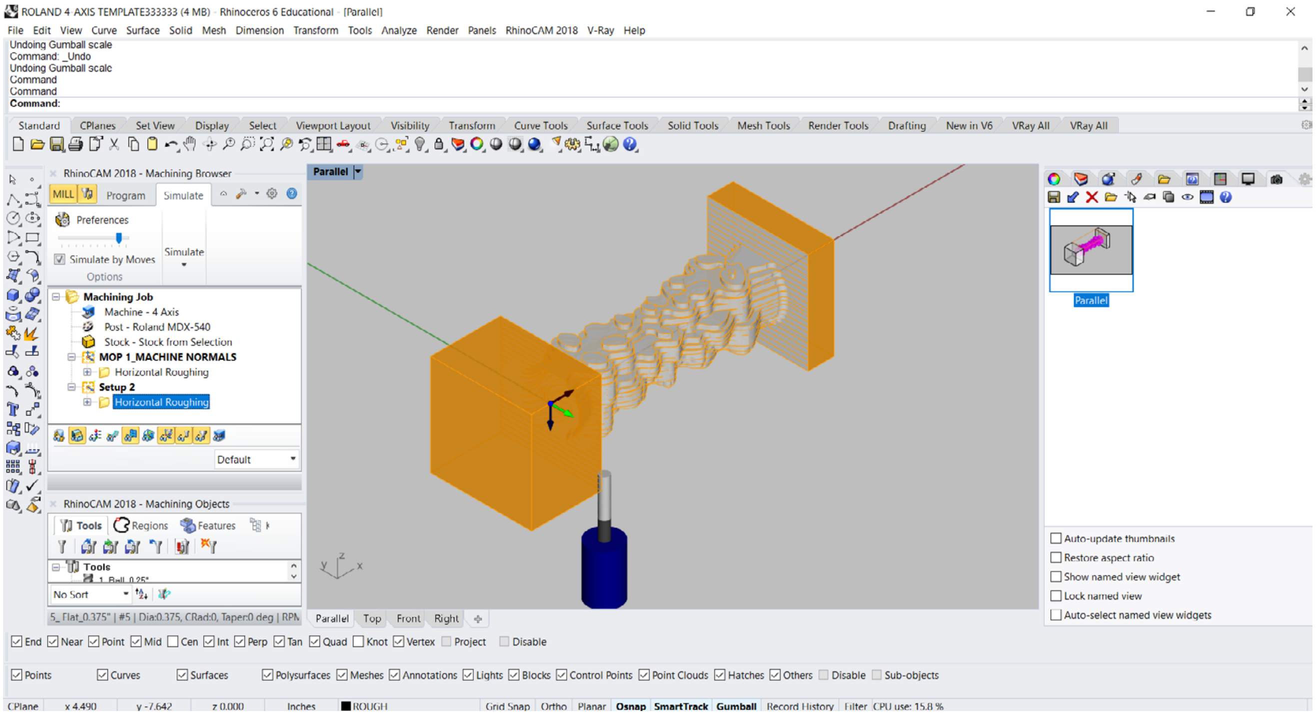


Copy and paste the original Horizontal Roughing operation into this new Setup. Regenerate this operation.

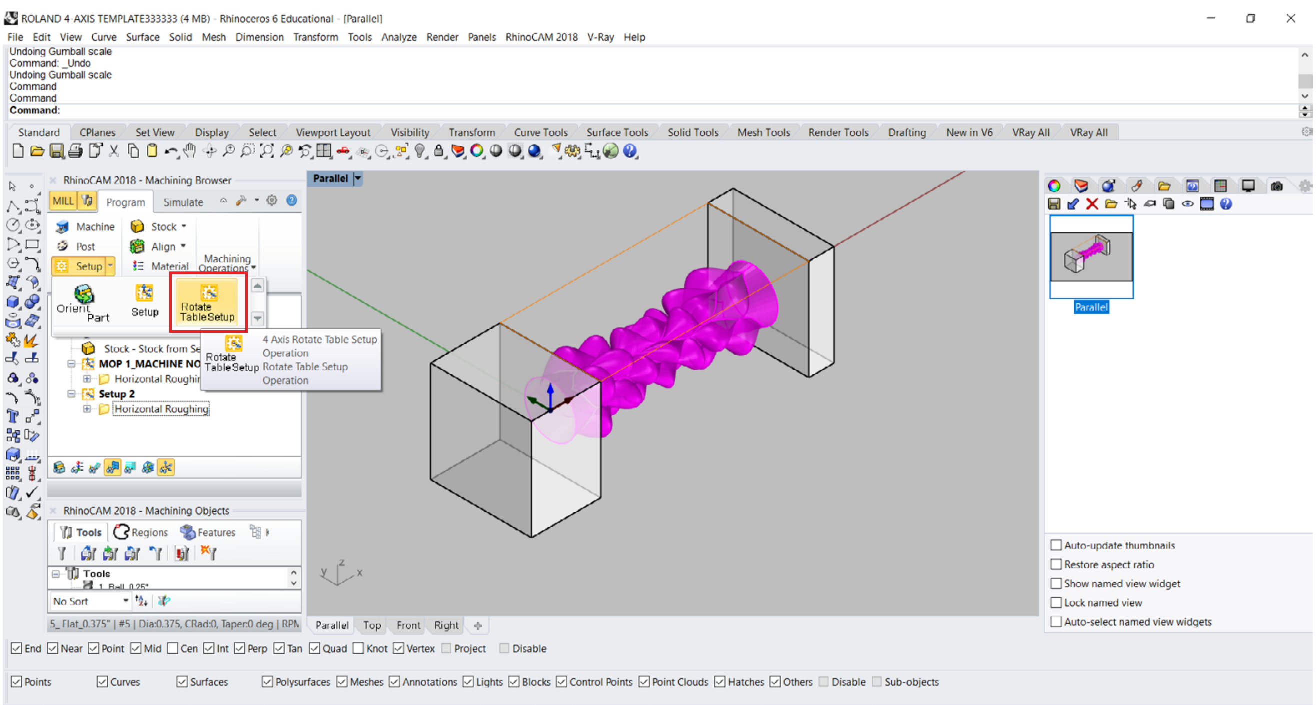


Your toolpaths should now be generated from the underside of the stock. Note that this is just a Rhino quirk, in reality it will be the piece that is rotating, while the tool always approaches from above.

FILE PREPARATION

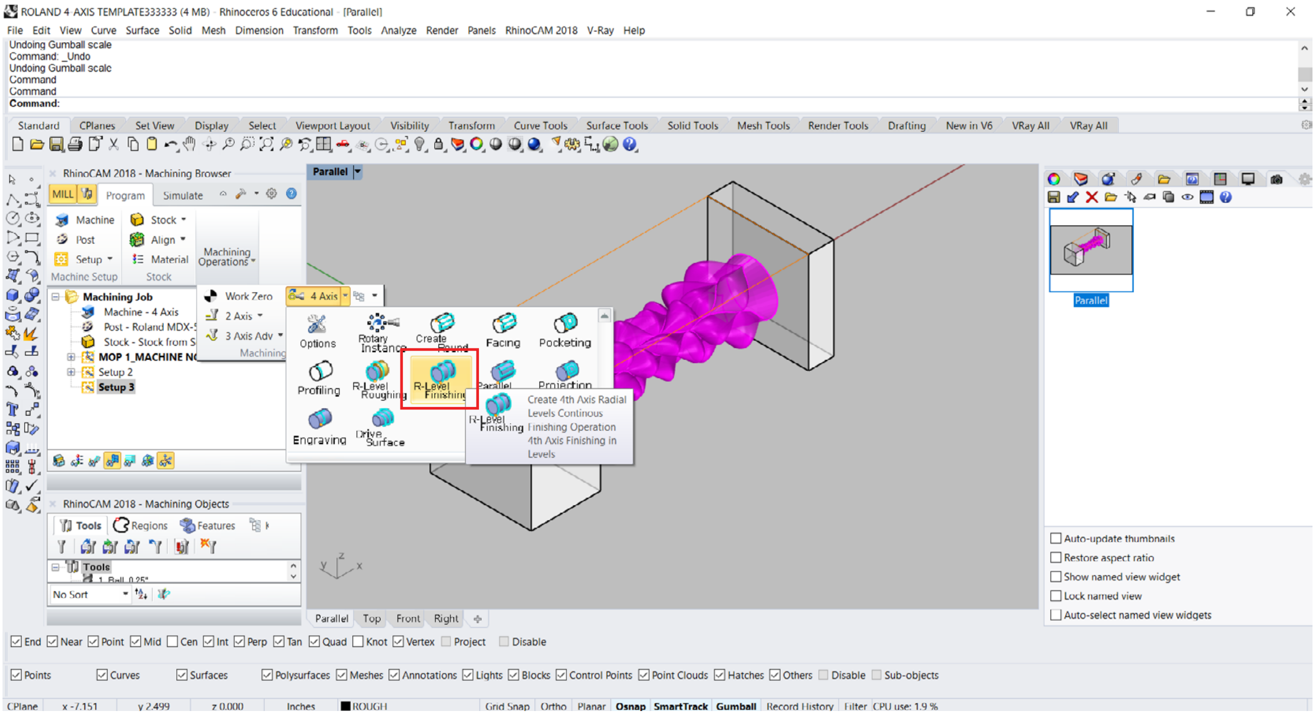


Simulating both operations should show you a roughed out geometry from both top and bottom.

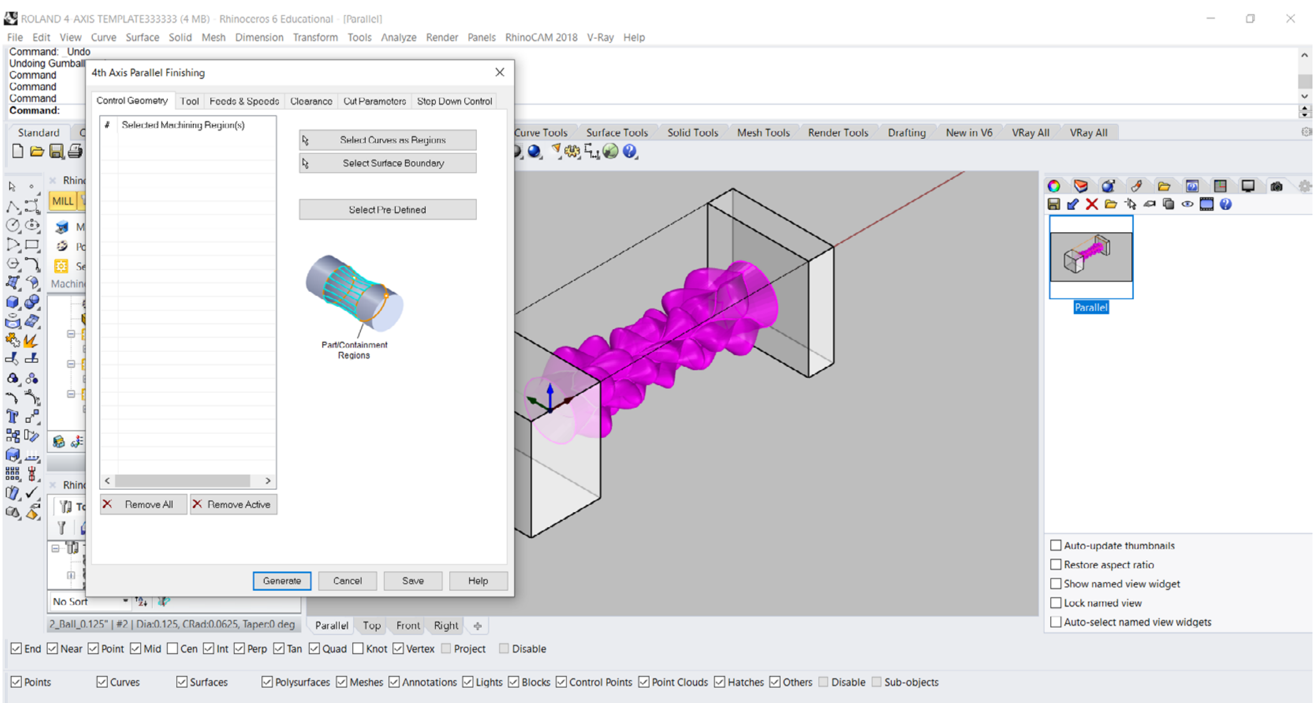


Now, create a new CSYS setup to return to the original orientation by setting the table rotation angle to 0.

FILE PREPARATION

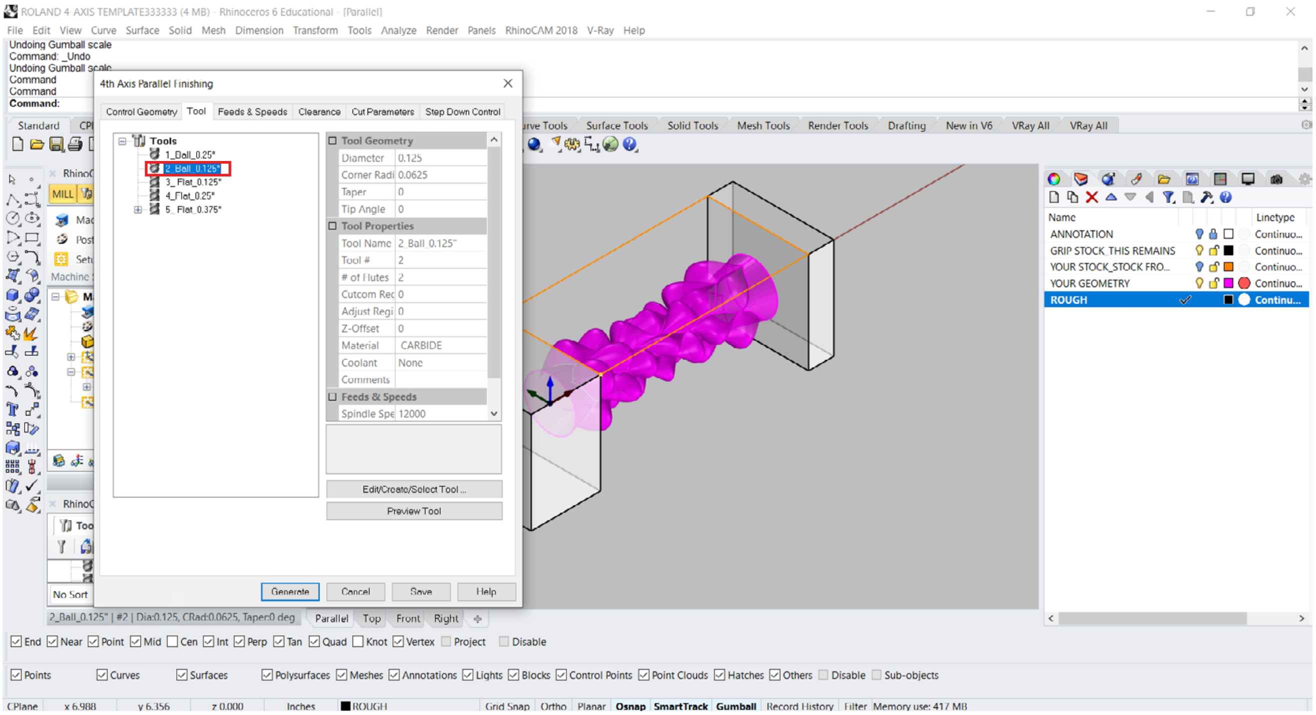


Click the 4 Axis tab and create a 4 Axis Parallel Finishing operation

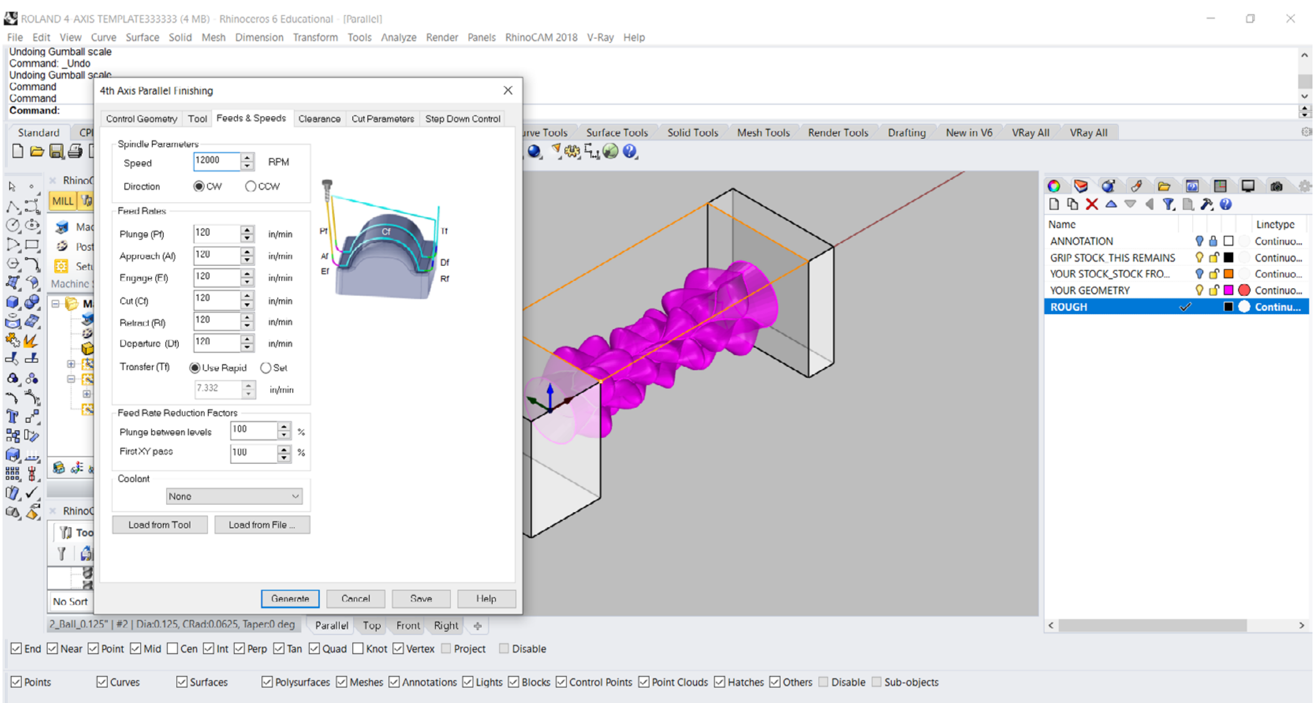


This type of operation does not require a drive region. We will define its bounding conditions later. If there is drive region already set up make sure to remove it.

FILE PREPARATION

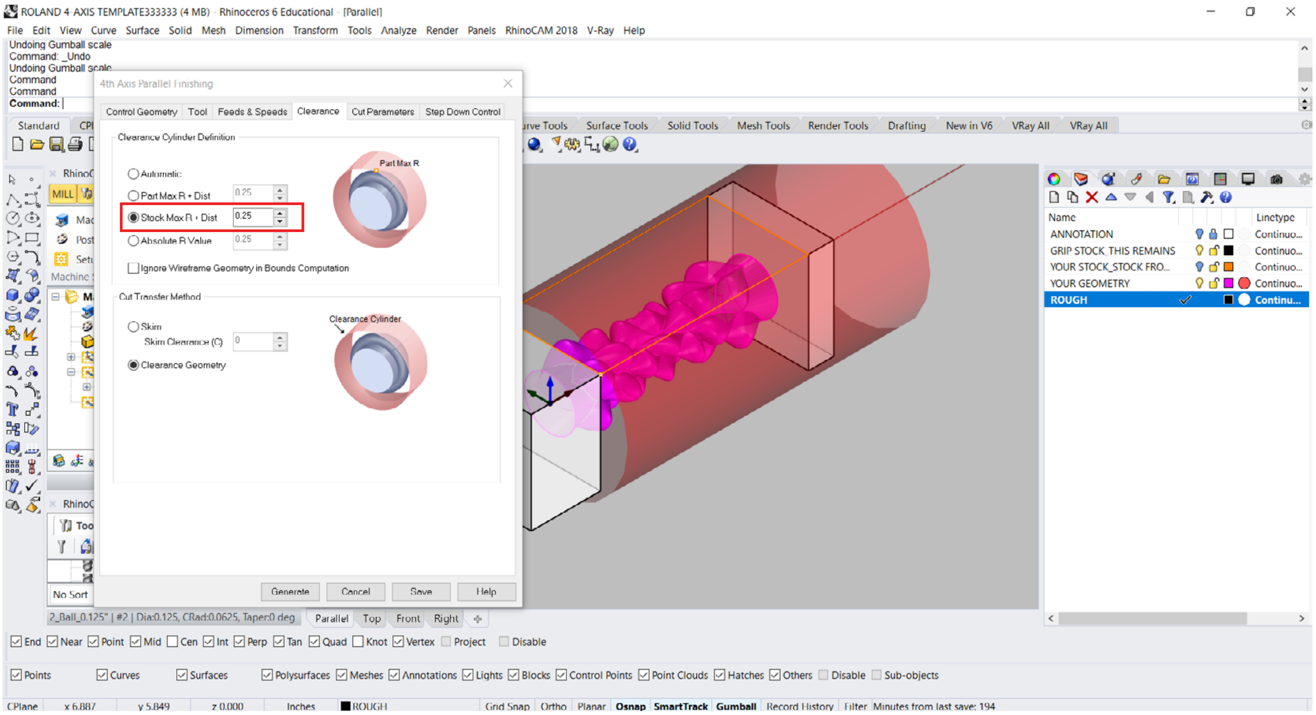


Select the tool you wish to use. In this case, a 1/8" ball tool will give us ideal resolution.

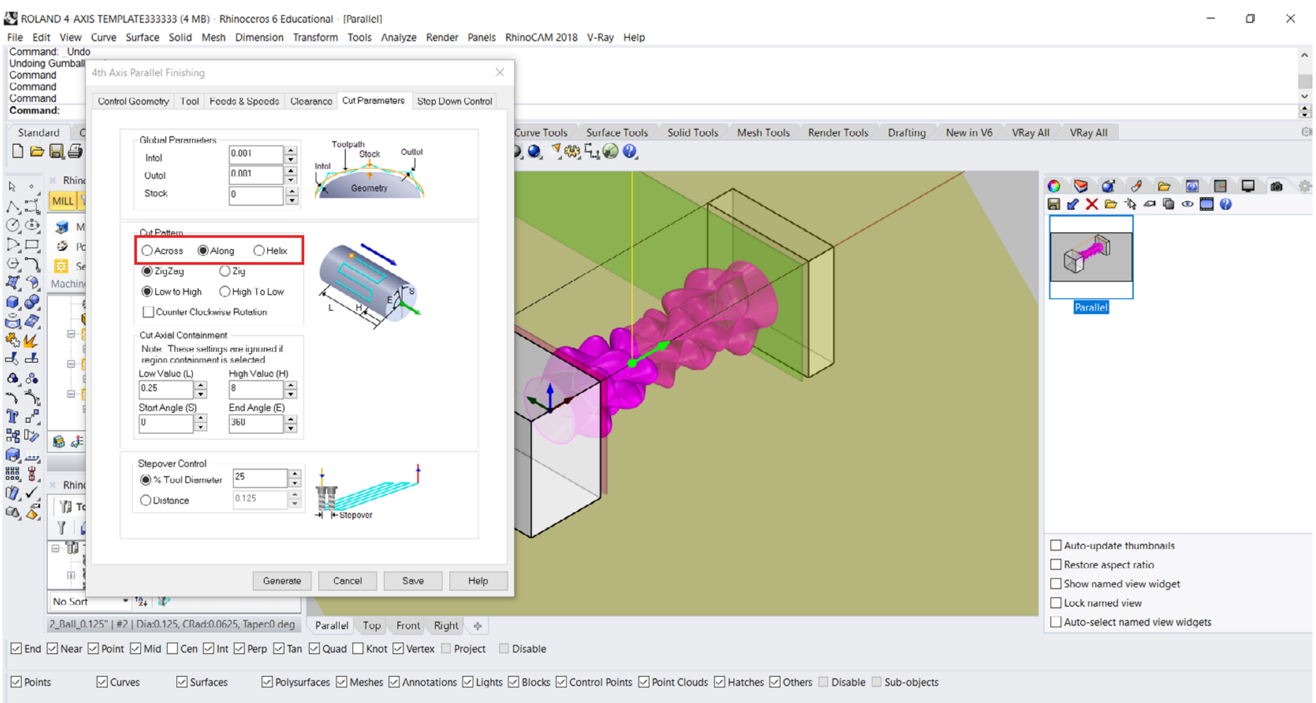


Again, feeds and speeds load automatically from the Roland Tool Library. Review these with a milling TA prior to starting your job.

FILE PREPARATION

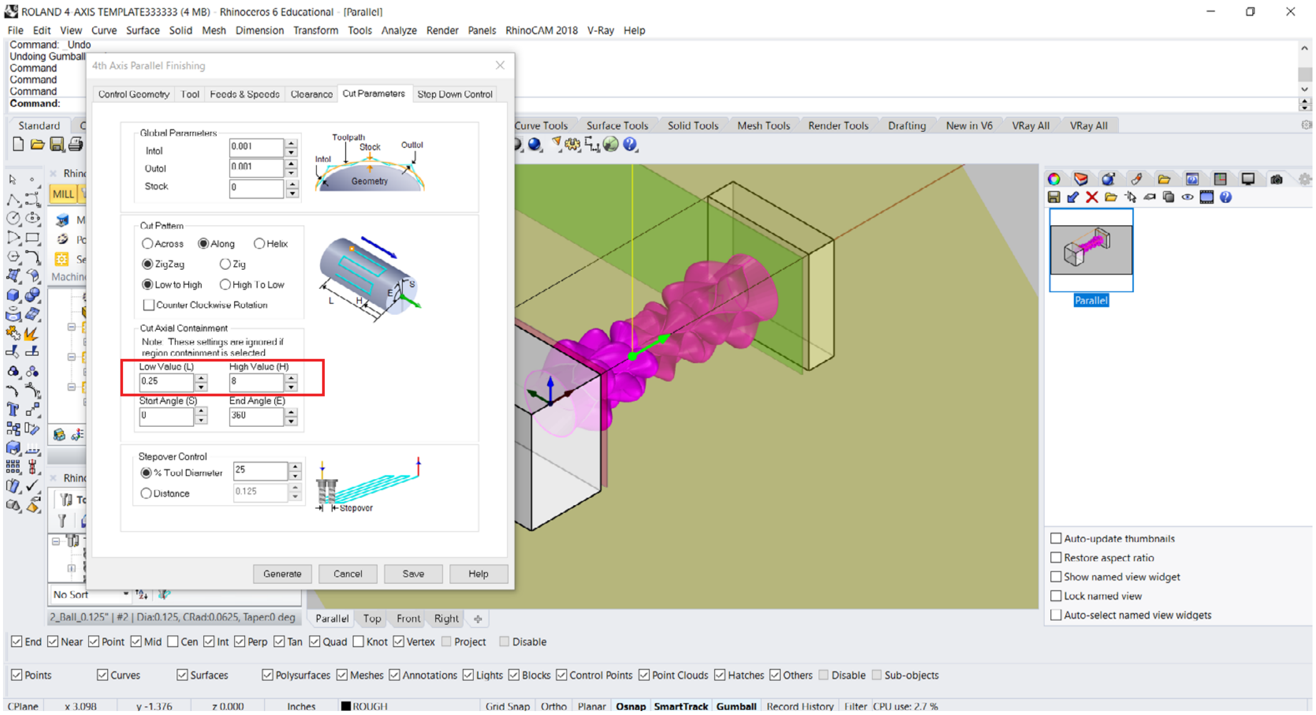


In the clearance plane tab, select stock max + 0.25”. You should see that the clearance plane is represented by a cylinder here rather than a flat plane.

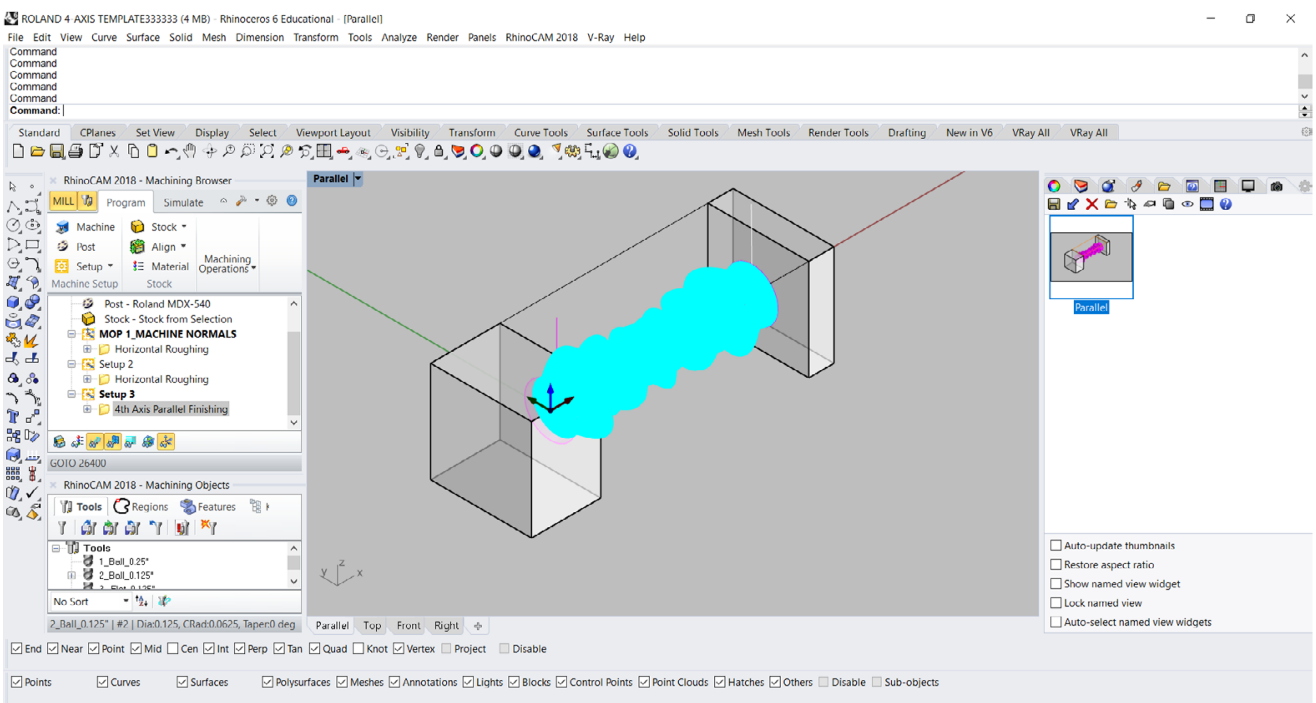


In the cut parameters tab, we need to define a few variables. First, whether we want to cut along the axis vs across. This is a choice you can make depending on your geometry, for now the along axis works fine.

FILE PREPARATION

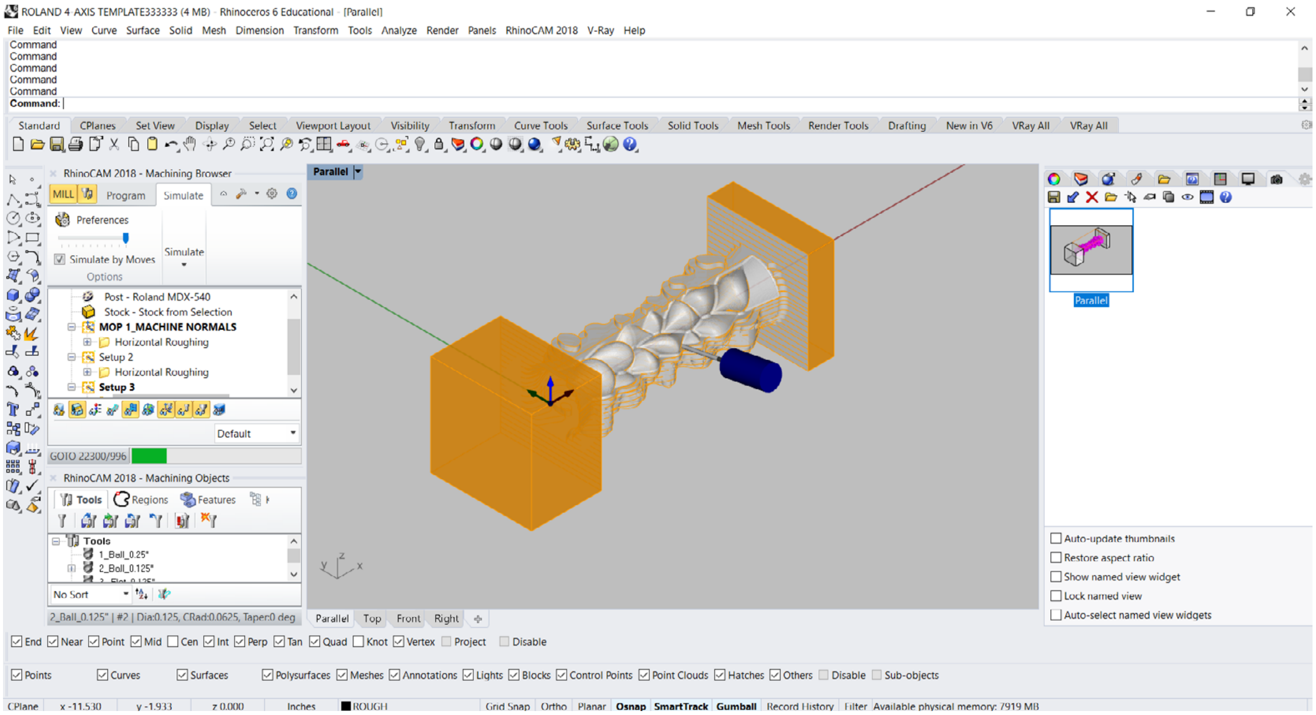


Next, we define axial containment. We want to keep the tool off of those harsh vertical faces to reduce cutting strain, so we tell it to only cut along the x axis between 0.25” and 8” (our actual geometry runs 0” to 8.25”). The default 25% stepover is fine for now.

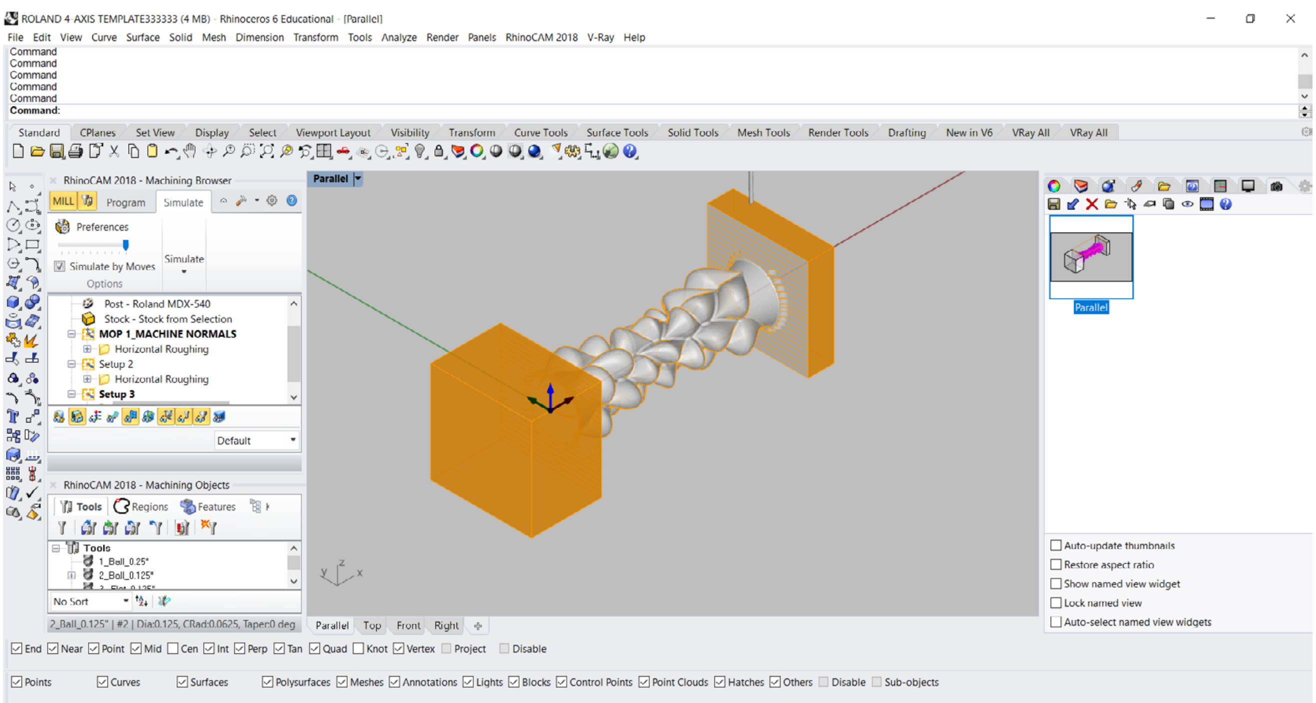


Generate this operation to view its toolpaths. Notice how they stop short of the vertical faces of our fixturing geometry.

FILE PREPARATION



Simulate this operation to view your results. Again, in reality the part spins, not the approach of the tool. It's just a RhinoCAM simulation quirk.



Toggle off your toolpaths to view the final result.

Save your file, and follow proper file submission protocols outlined here: https://daniels.utoronto.ca/step-step-guide-submitting-file-cnc-router