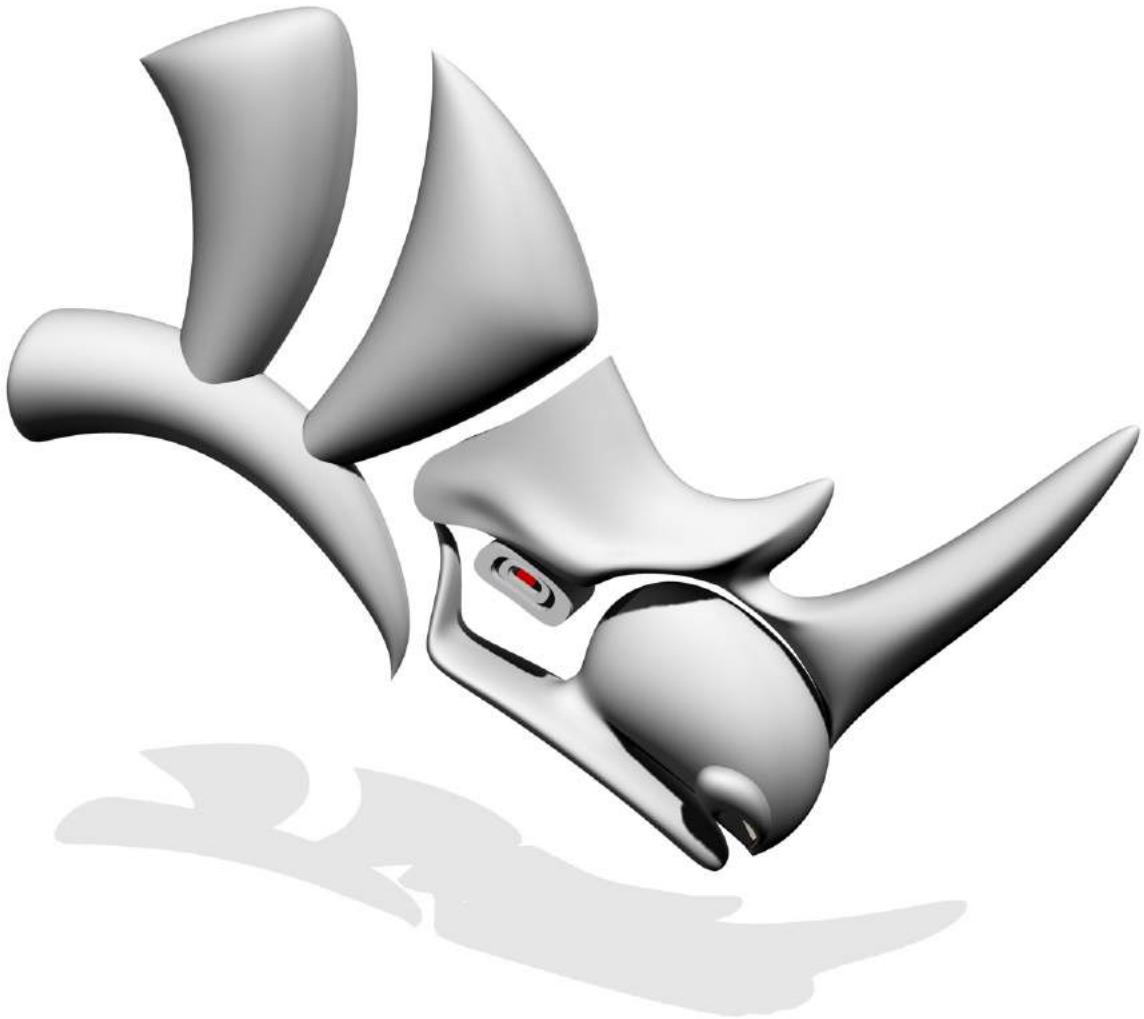


RhinoCAM®

MecSoft

RhinoCAM 2015_for Rhino 5

AXYZ 4008 + Roland MDX 540

CONTENTS_

Loading RhinoCAM P_ 1
Loading Tool Libraries P_ 2 - 3
RhinoCAM Operations Menu P_ 4 - 5
Machine and Stock Set Up P_ 6 - 10
Setting up a file P_ 11

2_AXIS MACHINING OPERATIONS

Engraving P_ 12 - 15
Profiling P_ 16 - 21
Pocketing P_ 22 - 26
Hole Pocketing P_ 27 - 29
Facing P_ 30 - 33

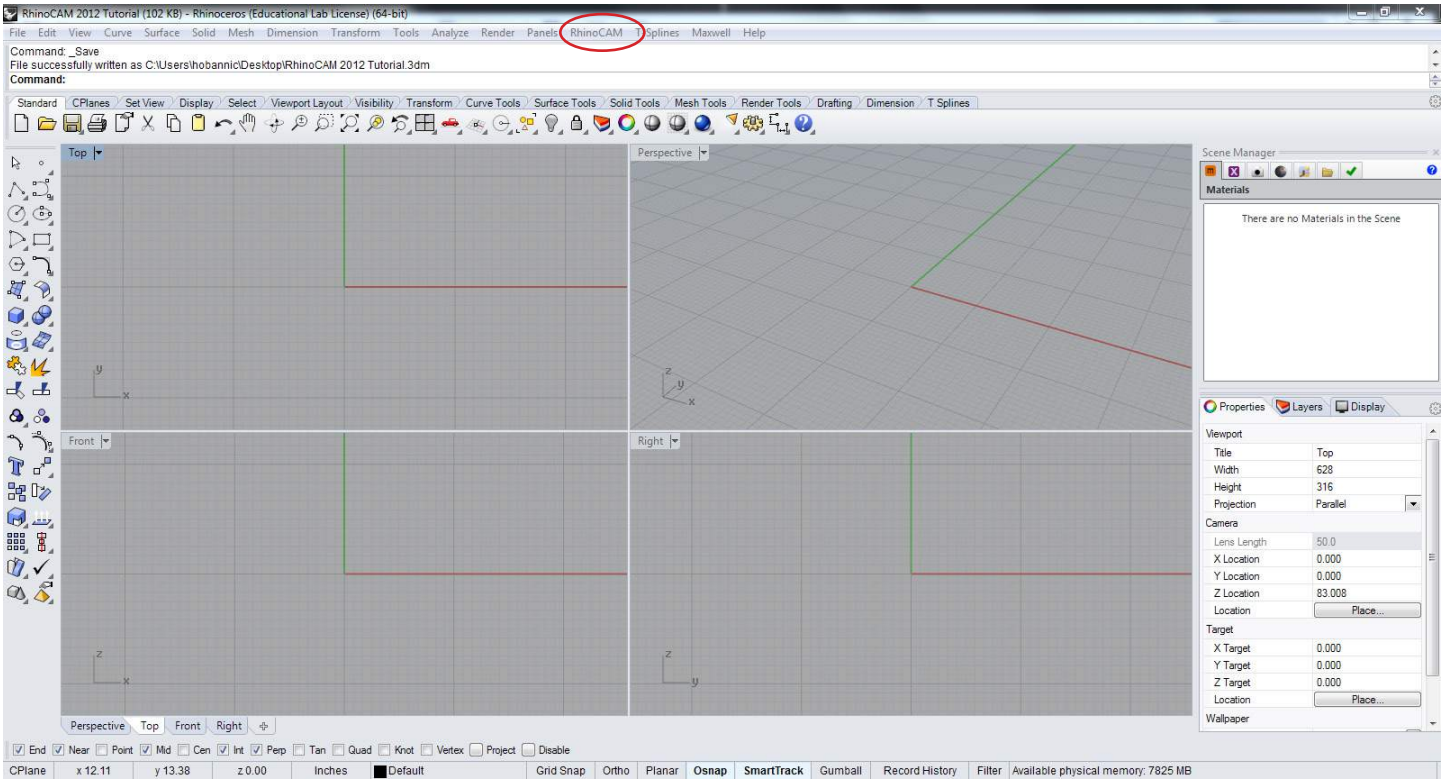
3 AXIS MACHINING OPERATIONS

Horizontal Roughing P_ 34 - 40
Parallel Finishing P_ 41 - 44
Projection Pocketing P_ 45 - 52
Spiral Machining P_ 53 - 61
Horizontal Finishing P_ 62 - 74

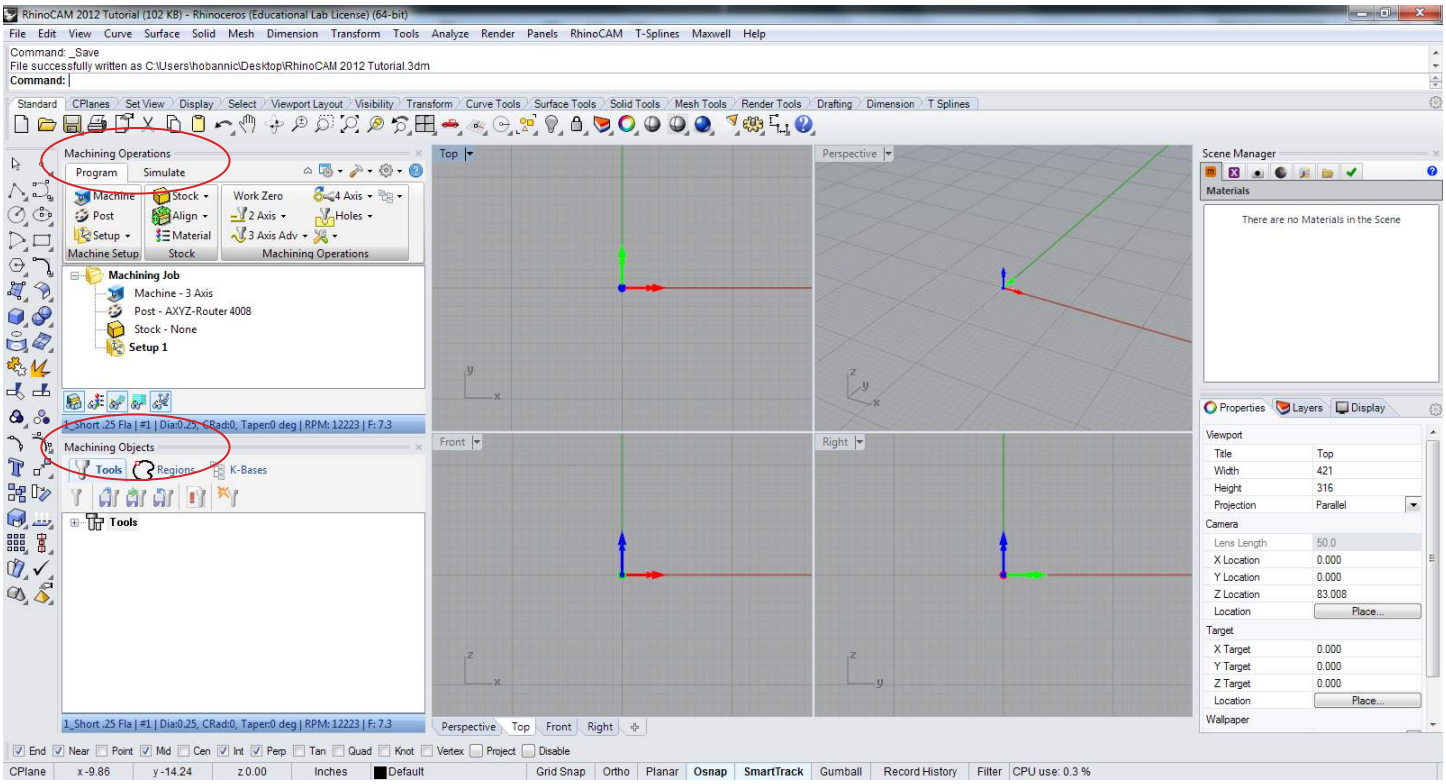
4 AXIS MACHINING OPERATIONS

Equipment P_ 75
File Preparation P_ 76
4 Axis Milling P_77 - 93

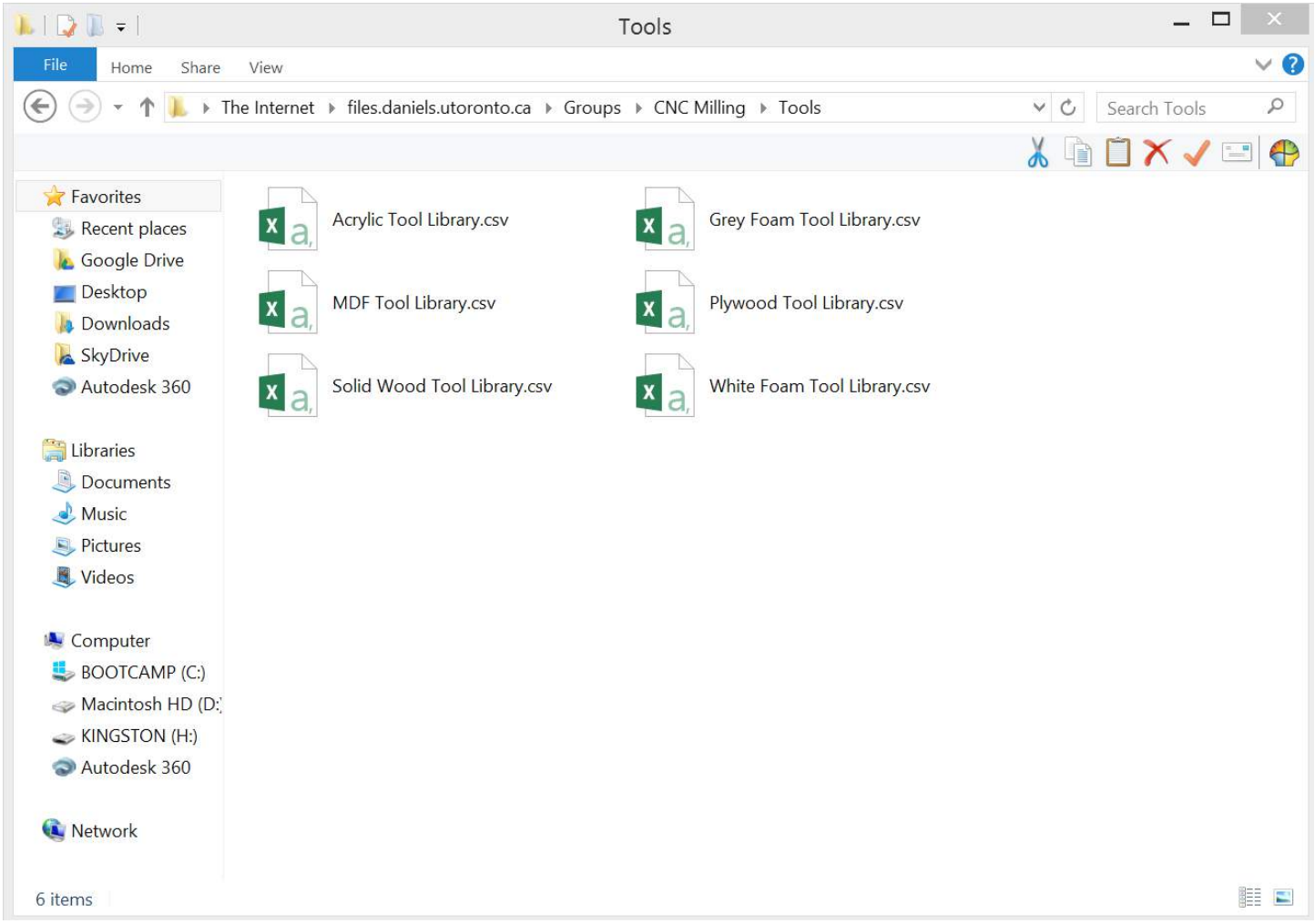
Rhino_CAM Loading the Operations/Objects Menu



To operate RhinoCAM launch the Machining Operations 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.



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.



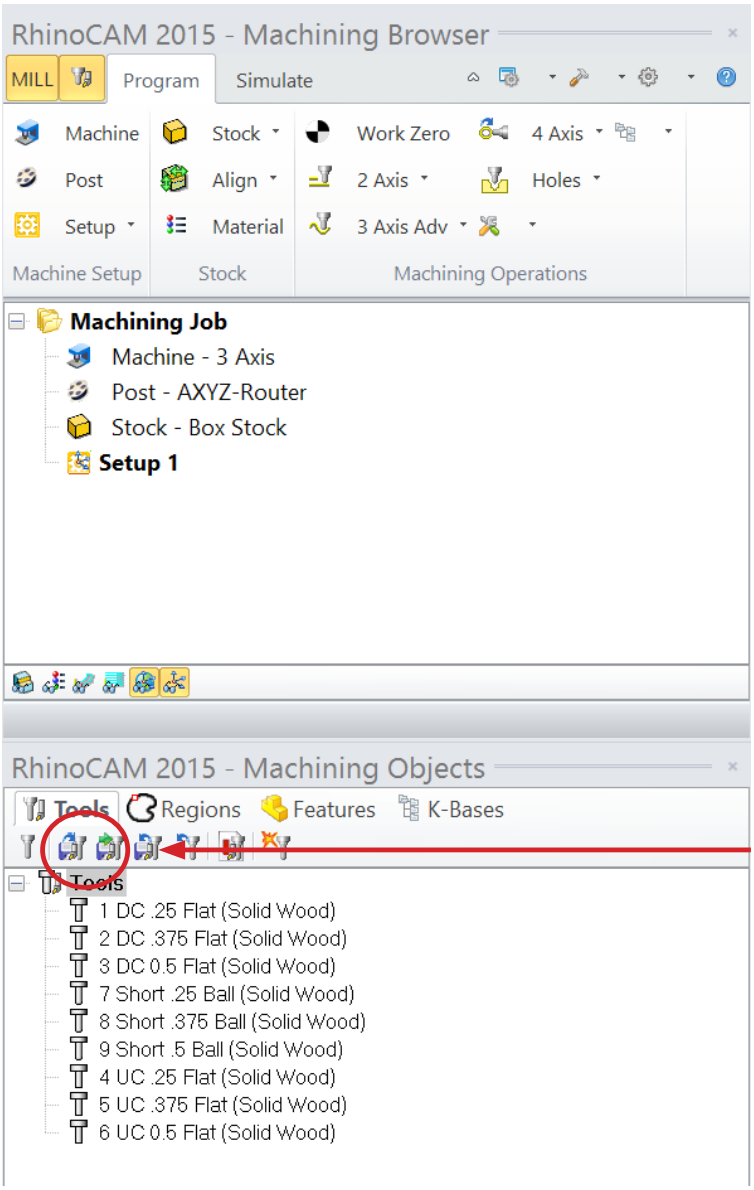
To find the Daniels Tool Libraries first go to:

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 endmills 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.

Rhino_CAM Loading the Daniels Tool Library



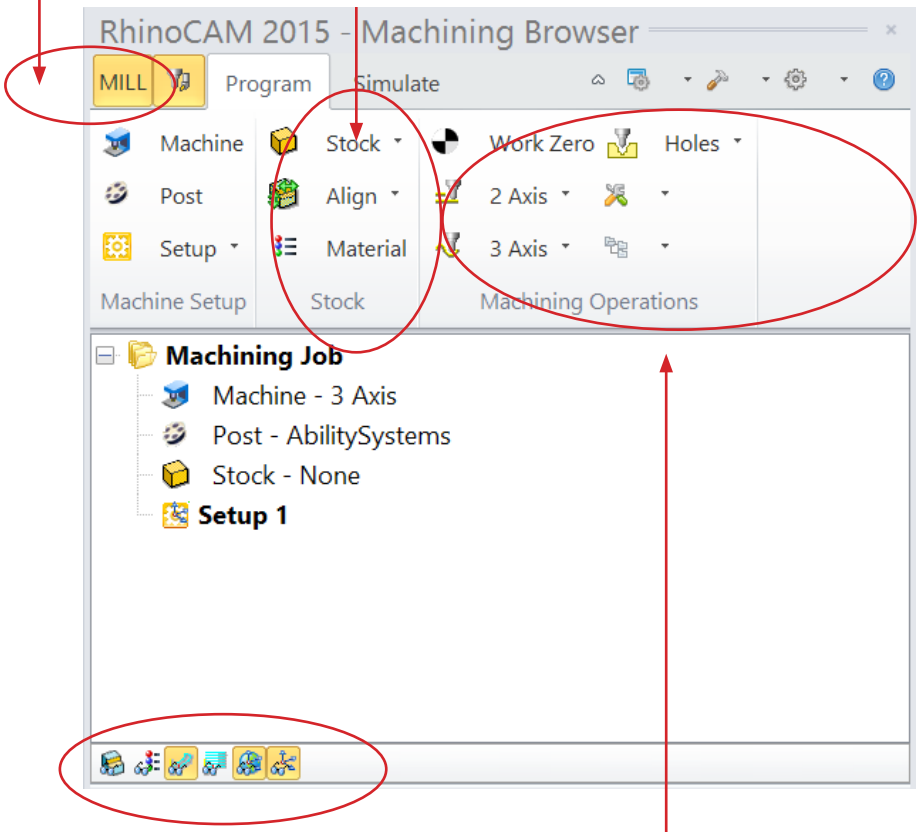
To load the tool library go to the Machining Objects Menu while under the tools tab click the load tool library button. Only load the tools specific to your material.

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

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.

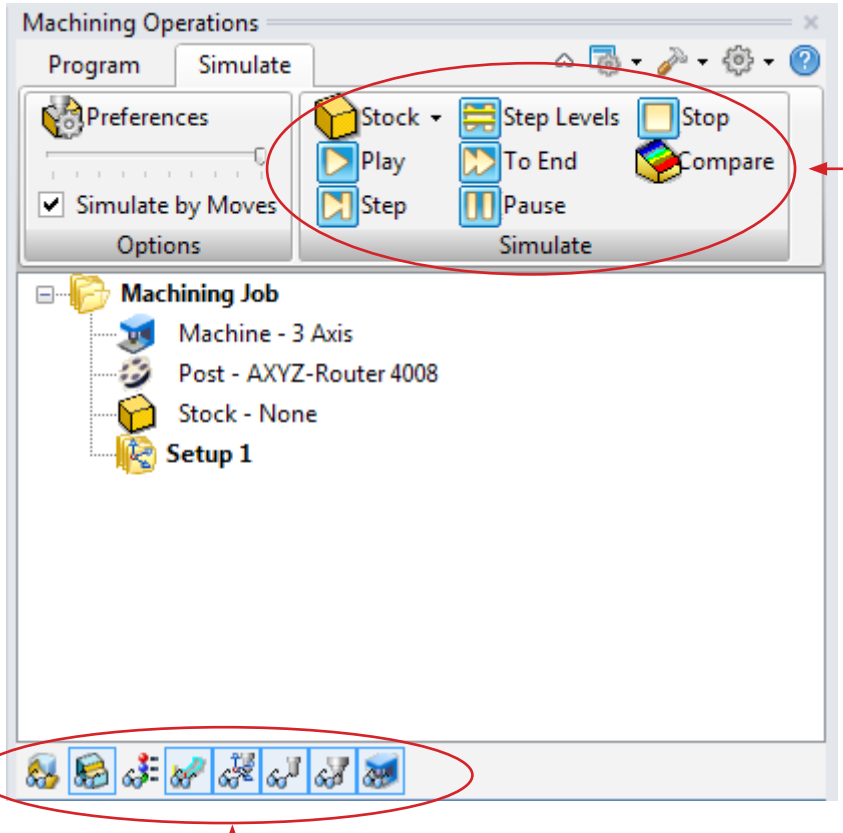
On the far left hand side of the display tabs, make sure MILL is displayed and highlighted. Clicking here will toggle between MILL and TURN, which is not a function available on the labs CNC Router.

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 6 along with setting material texture.



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.

The machining operations contain both sets of 2 axis and 3 axis adv cutting operations, these two drop down menus contain all necessary functions to create milling paths for 2d and 3d cutting.

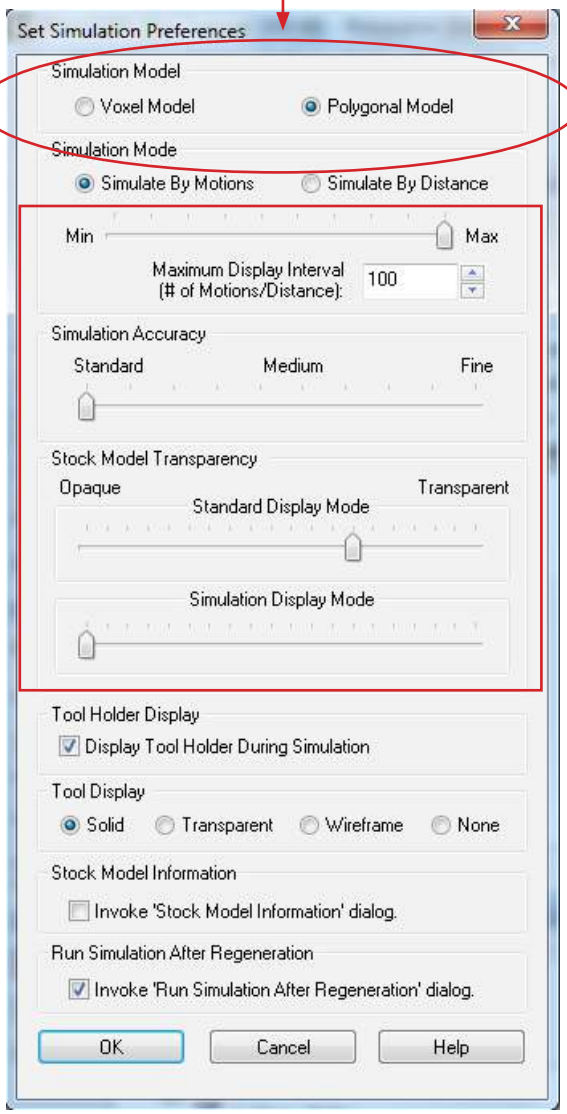


The Simulate Menu allows simulation and playback of the established milling operations. You can process by playing the Milling operations or stepping through the set incrementally with the step button. Pausing will allow you to fast forward to the end with the 'To End' button.

The 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

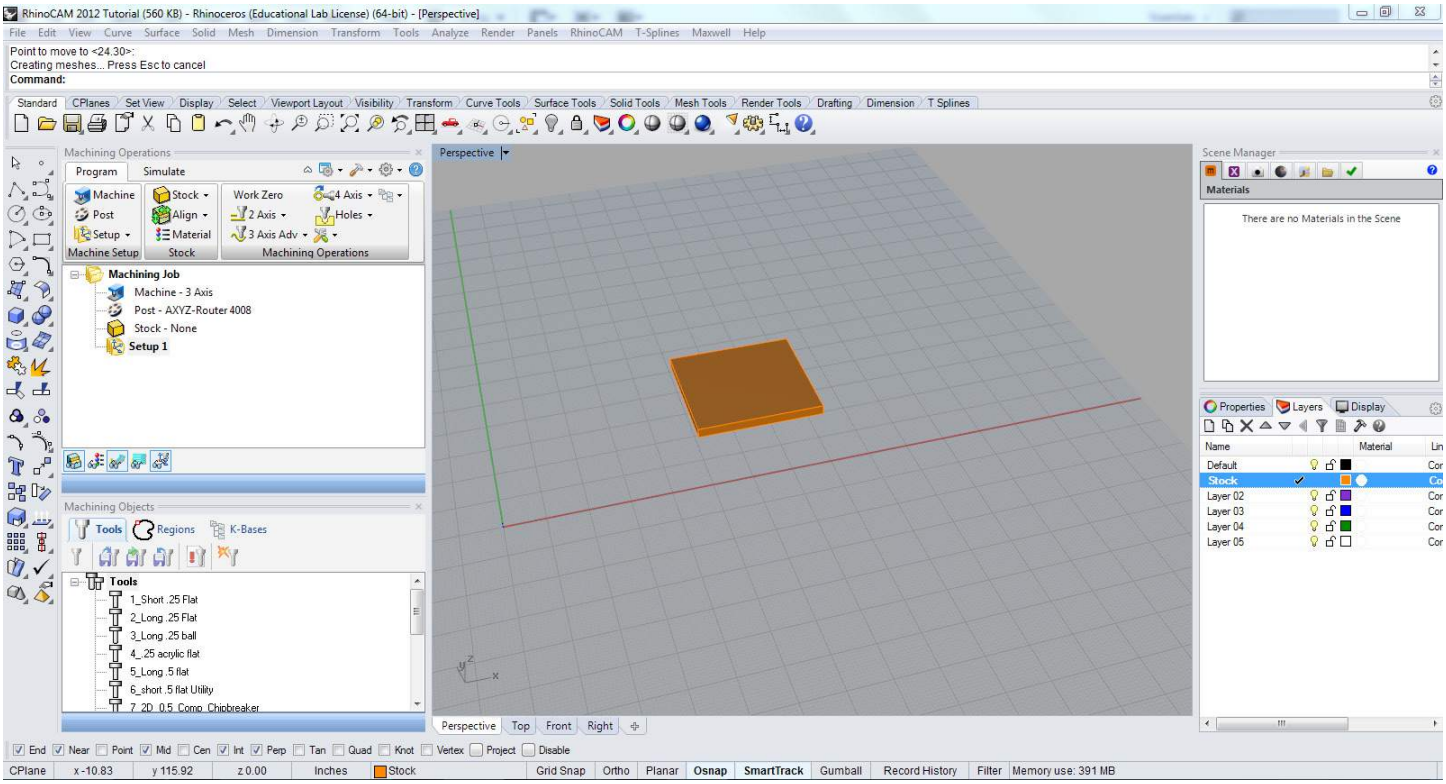
The visibility displays are located on the bottom left of the Simulation menu. The buttons allow to toggle on part, stock, material, toolpaths, Machine CSYS, Tool, Holder and Machine Visibility.

The simulation 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.

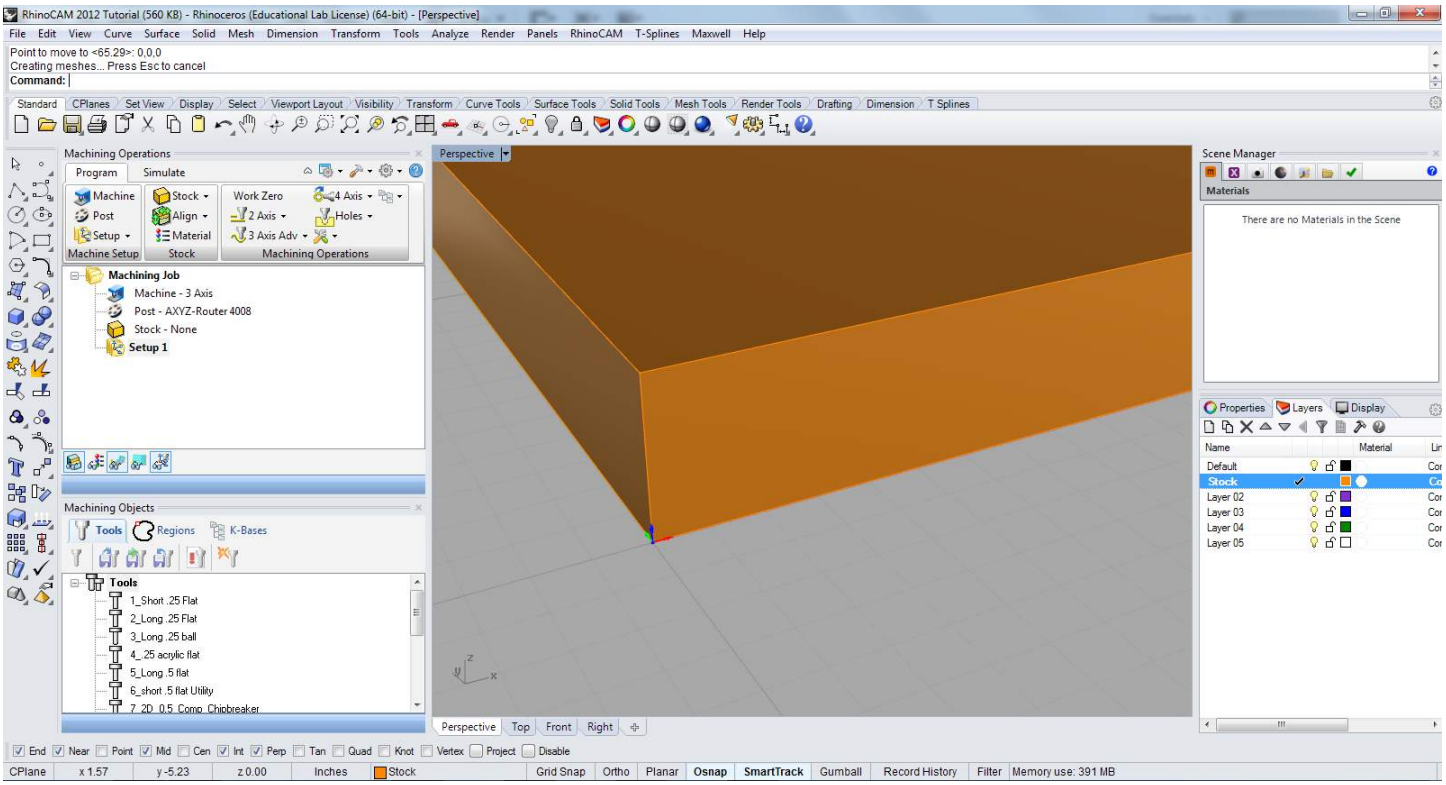


Rhino_CAM Stock Set Up

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.

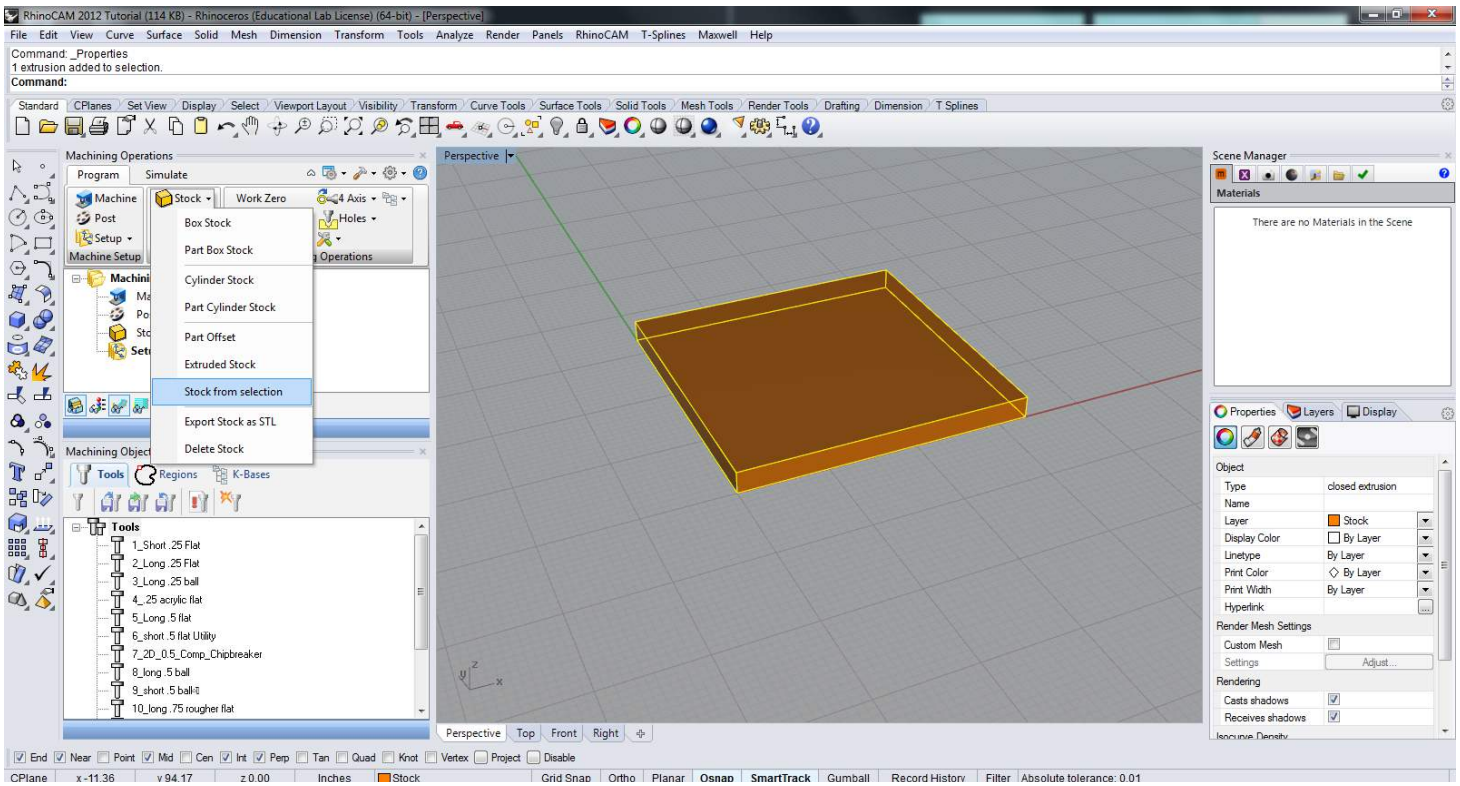


Create a stock layer, then create the stock size required to match the physical material stock. Create the stock in Rhino.

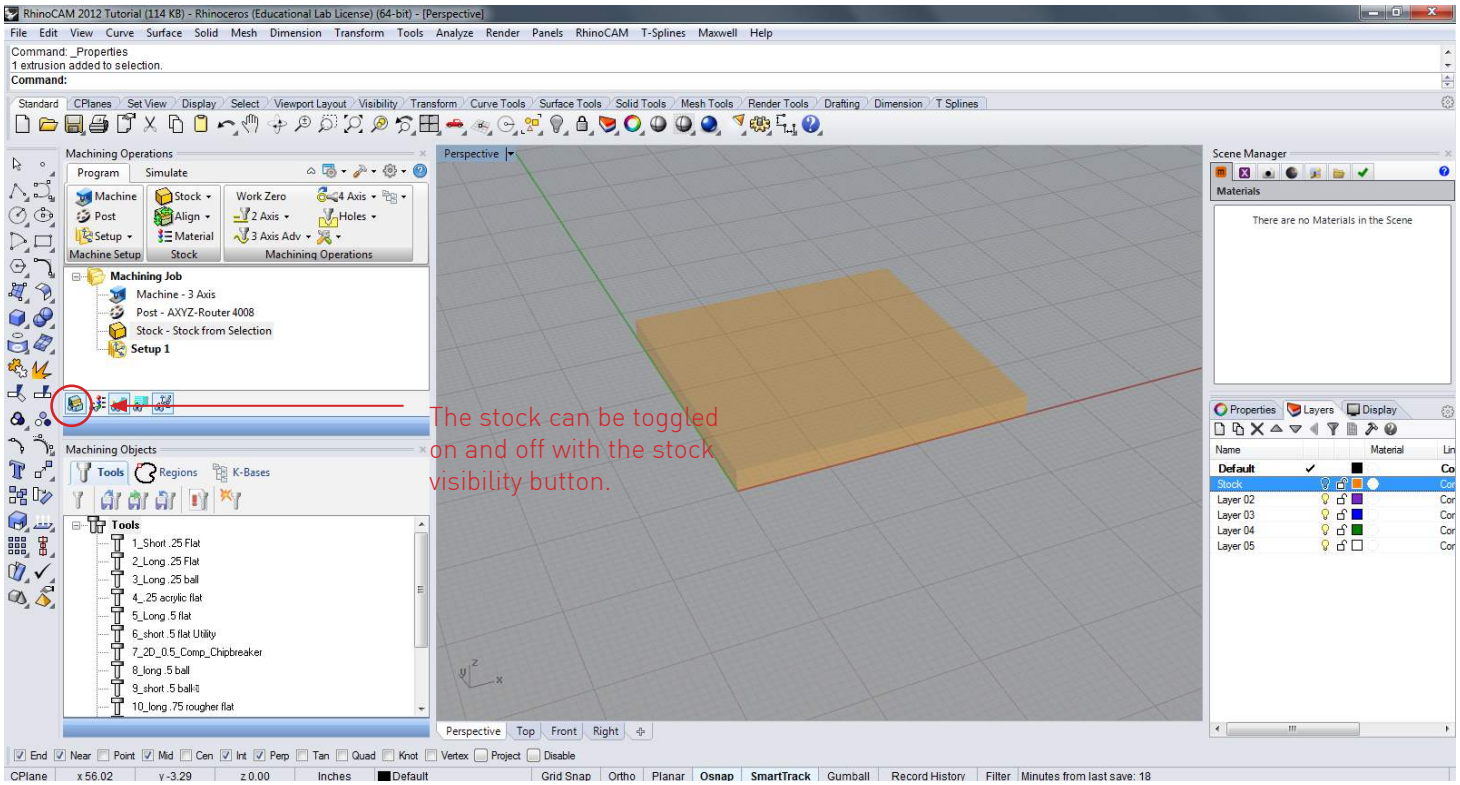


Move the stock to the origin in rhino (0,0,0) this corresponds to the origin of the CNC. The Red, Green and Blue arrows indicate the X,Y and Z direction of the CNC bed.

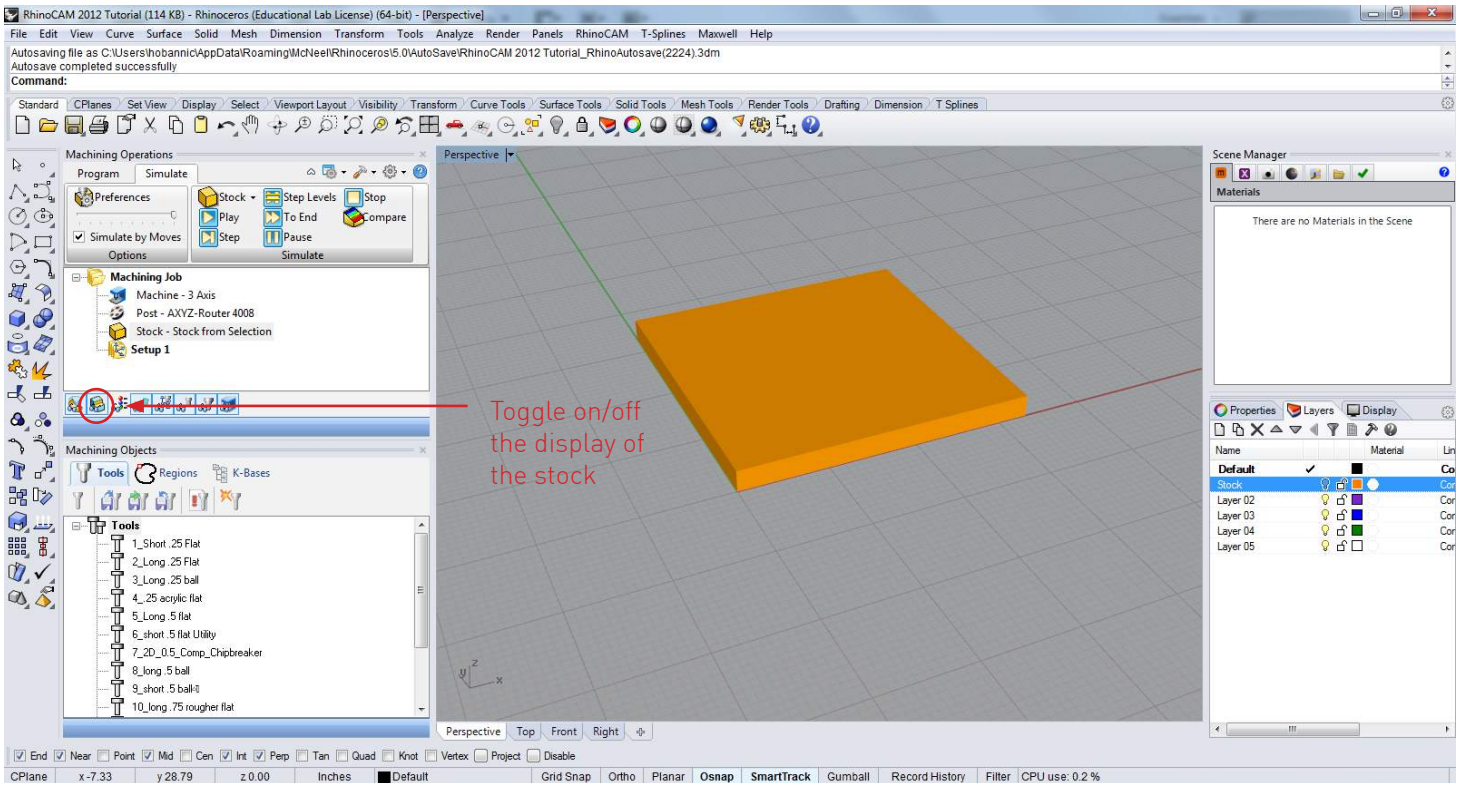
Highlight the created stock, from the machining operations menu select the stock drop down menu, then select stock from selection. This will create a stock from the object.



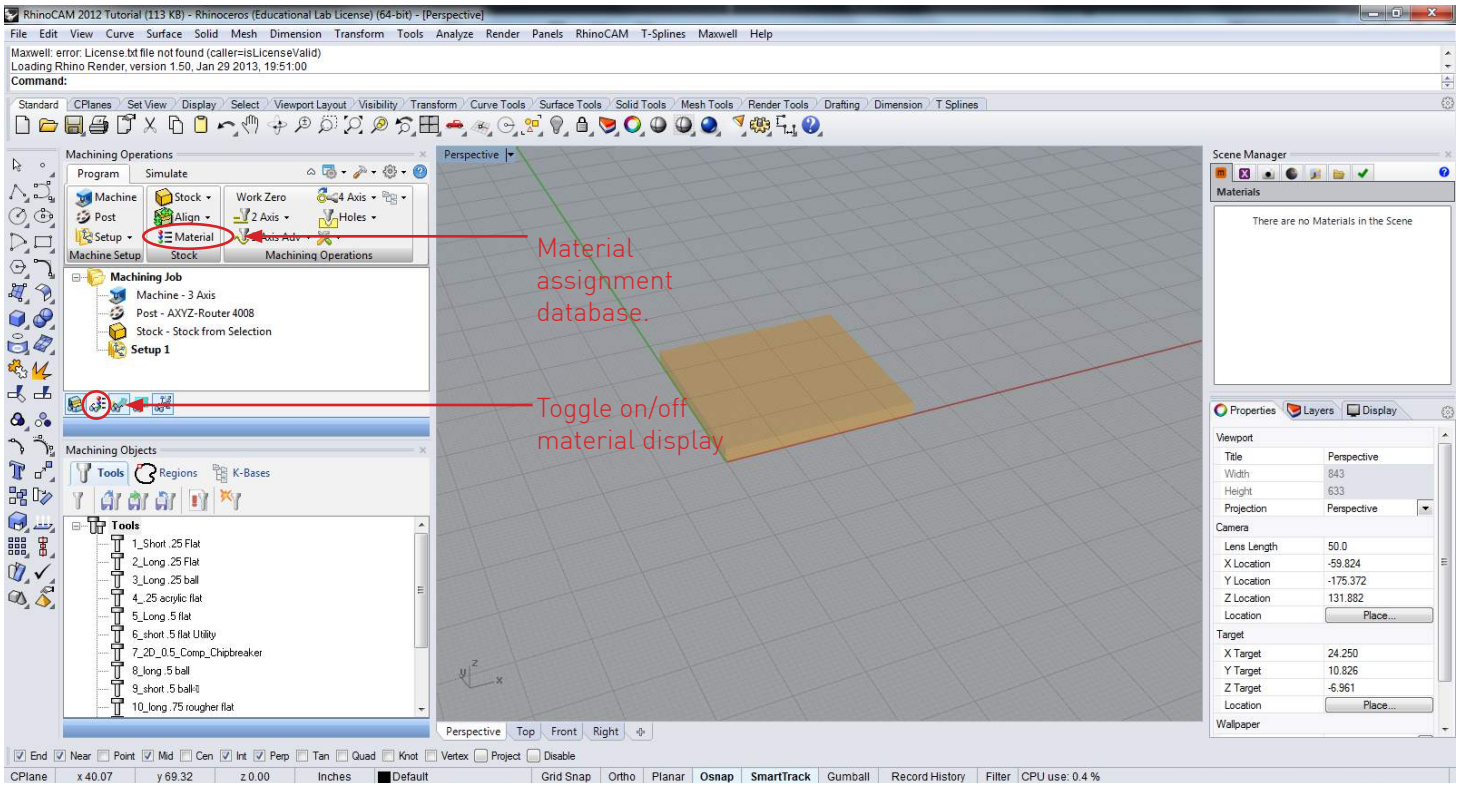
Once the stock has been created you can turn off the stock layer, the ghosted stock you see in the program menu is the RhinoCAM stock. All milling paths and operations will be applied to this stock.



Switching to the Simulate tab will show the stock as a solid rendered object. You can toggle on and off the display with the stock visibility button in the simulate mode.

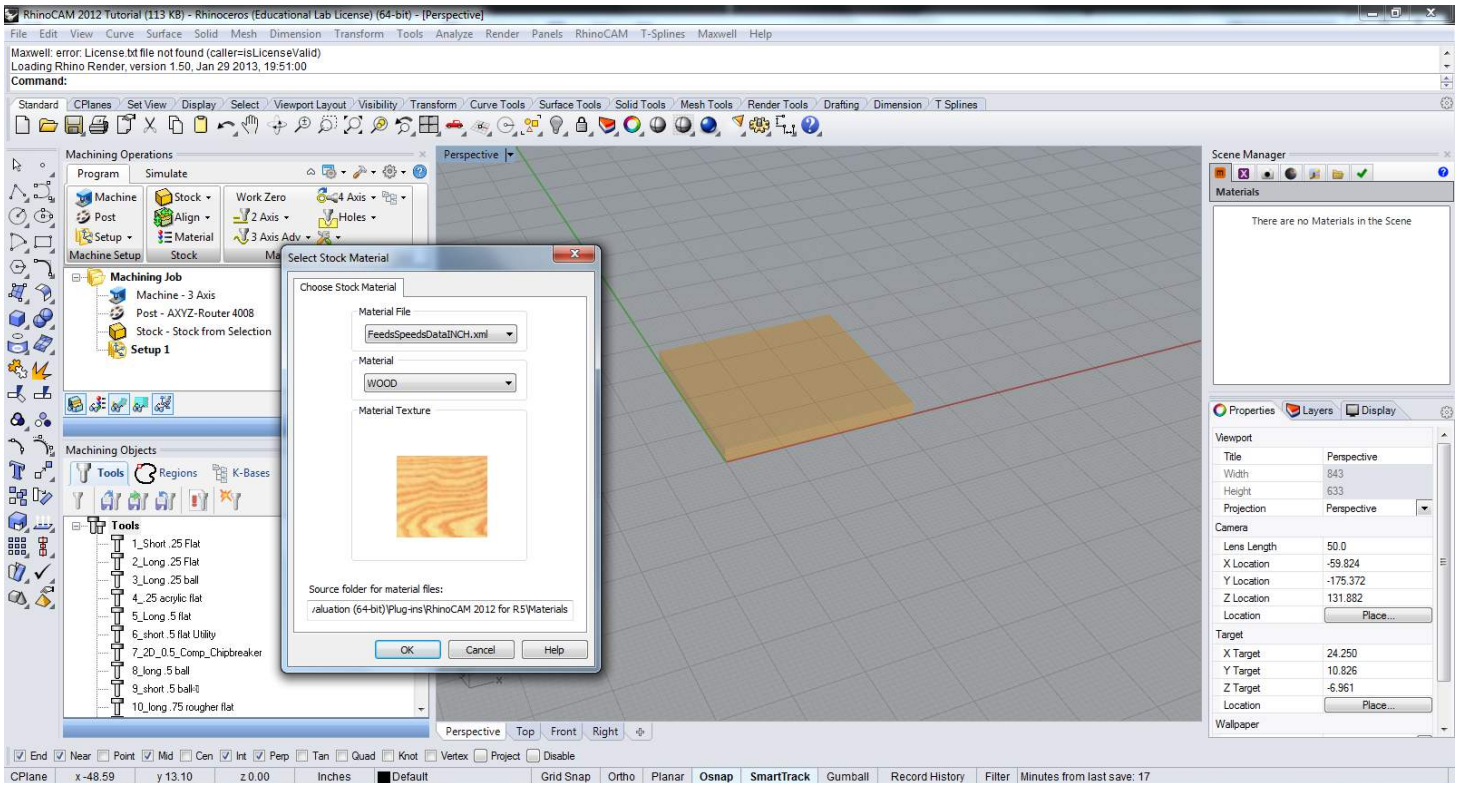


Material type can be assigned to the stock on the Program Tab, press the material button to access the stock material database in RhinoCAM.

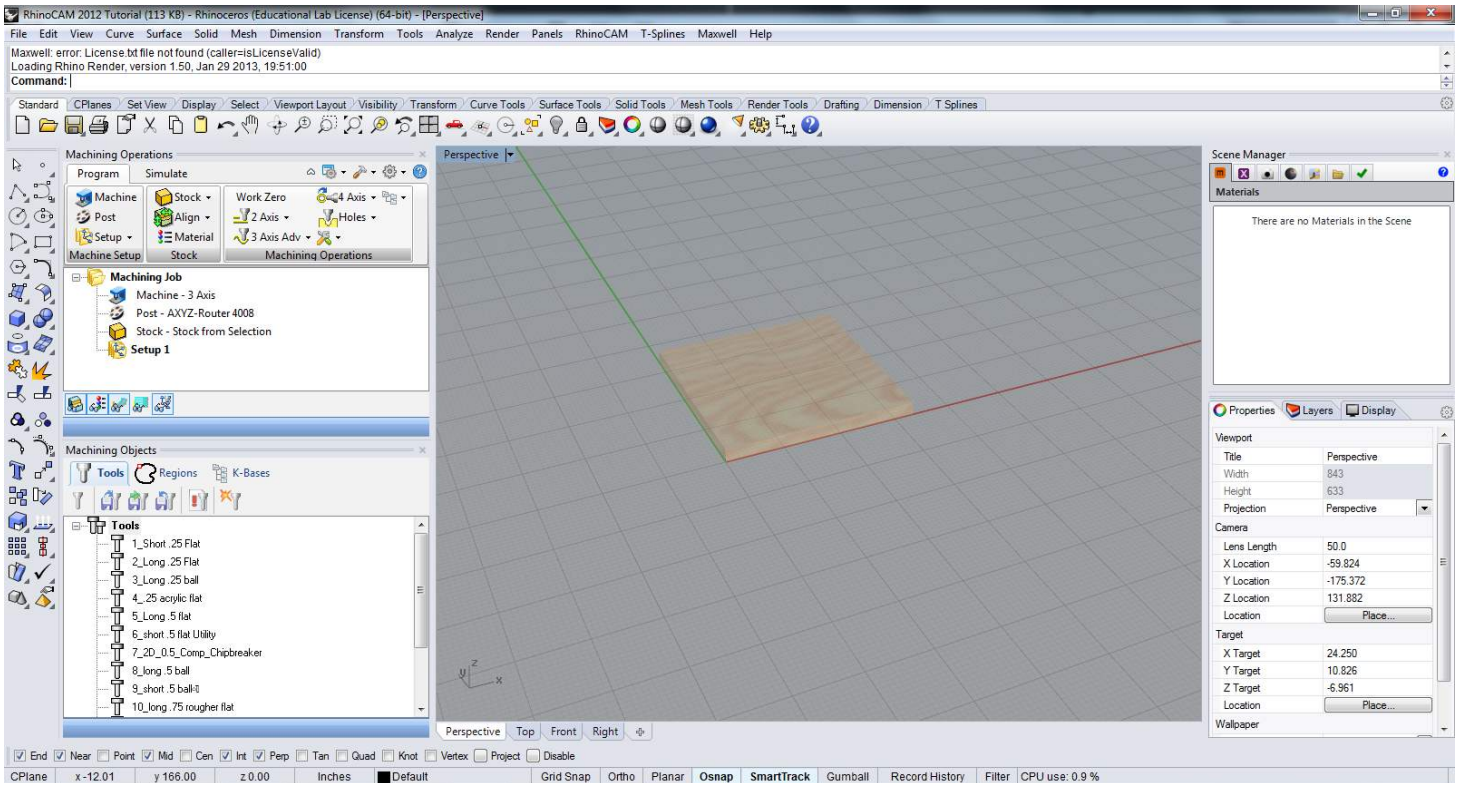


Rhino_CAM Stock Material Assignment

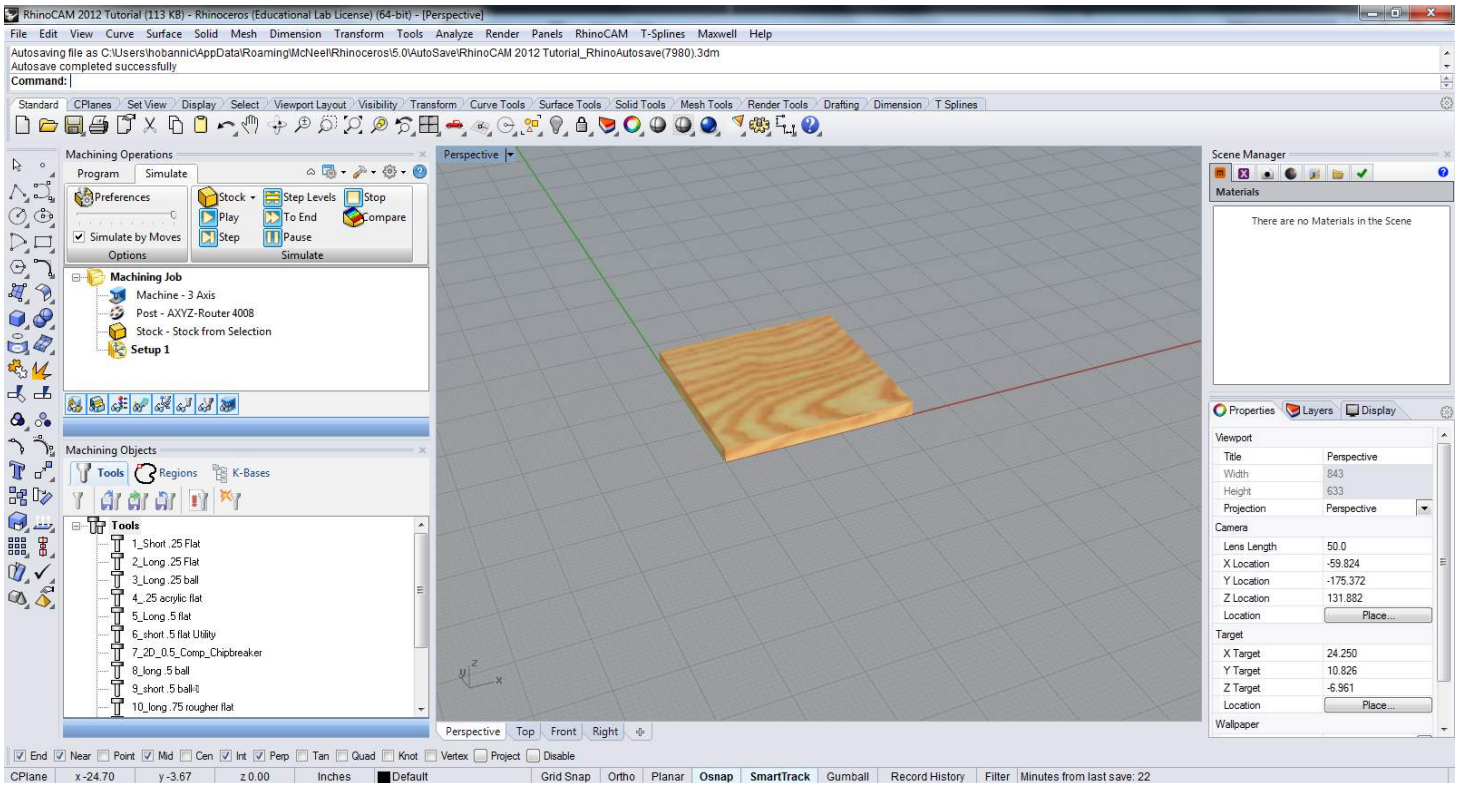
The pop up dialog allows for different material assignments to see the effects of the later assigned milling paths on the stock material.



While in program tab when the ‚material visibility is turned on the stock appears ghosted with assigned material type.

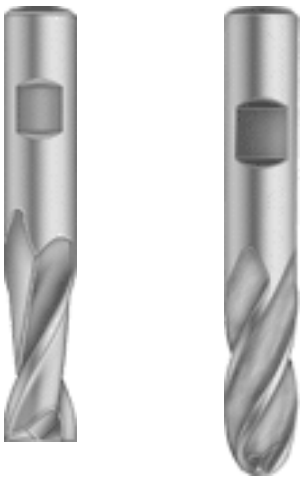


Under the simulate tab the material will appear fully rendered.



To begin setting up your RhinoCAM file have either your 2D geometry (Profiles) or 3D geometry (Surfaces) 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.

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.

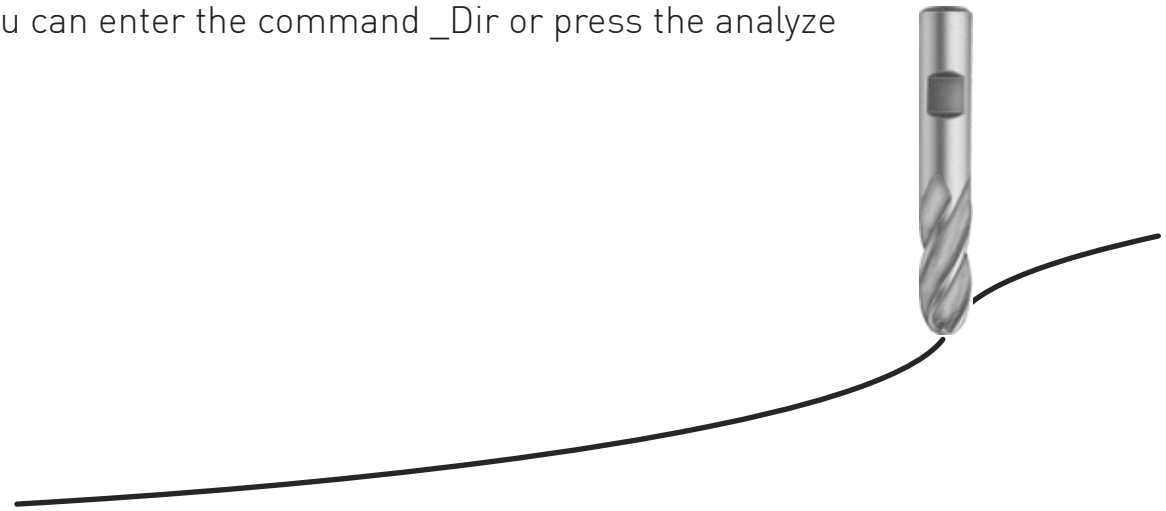


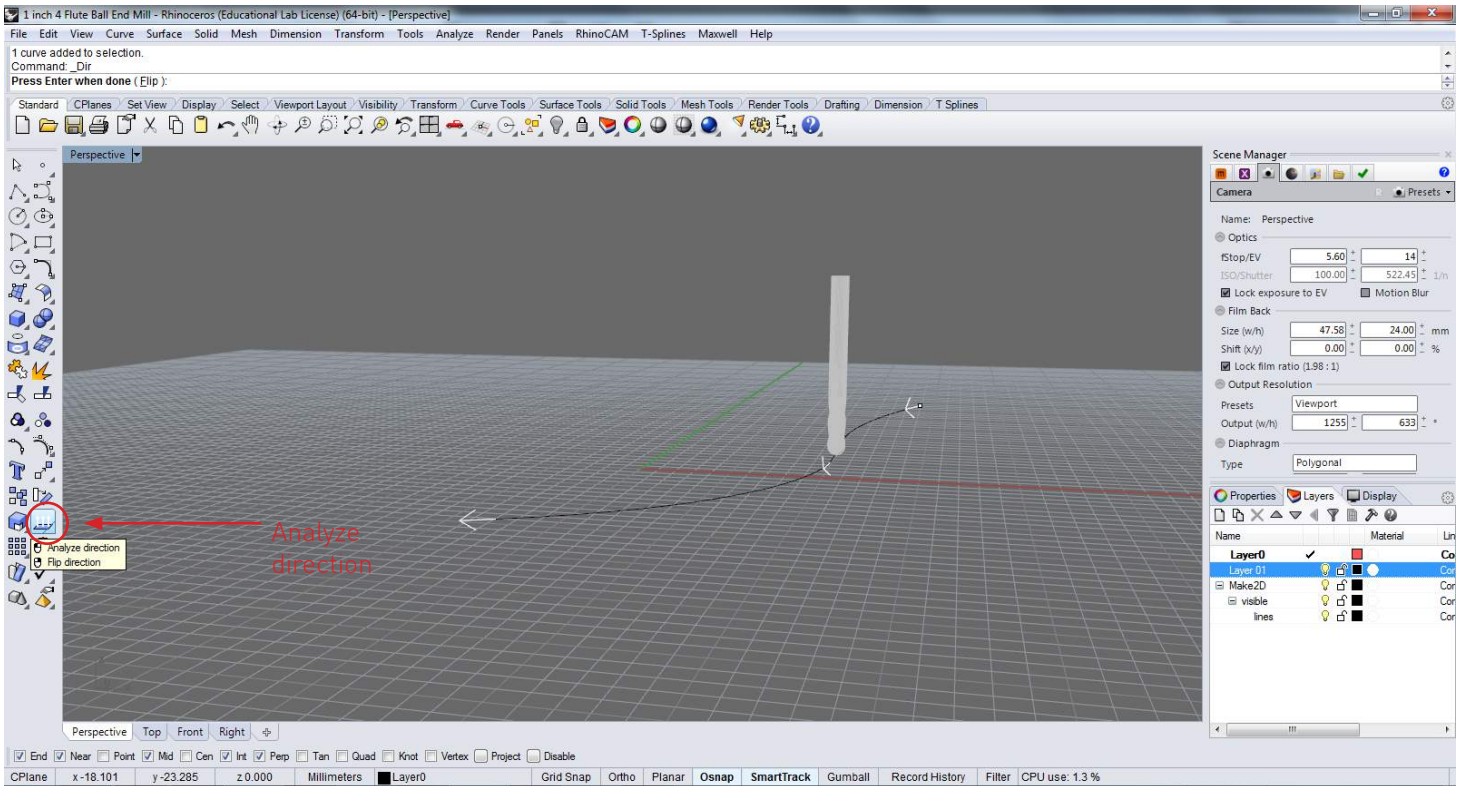
End Mills come in different lengths and diameters, the daniels tool library (covered previously in how to load the tool library) has a range of end mills from 1/4'' to 3/4'' in flat end mills and ball end mills. The end mills have a overall length, cutting depth and both shank diameter and mill diameter.

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.

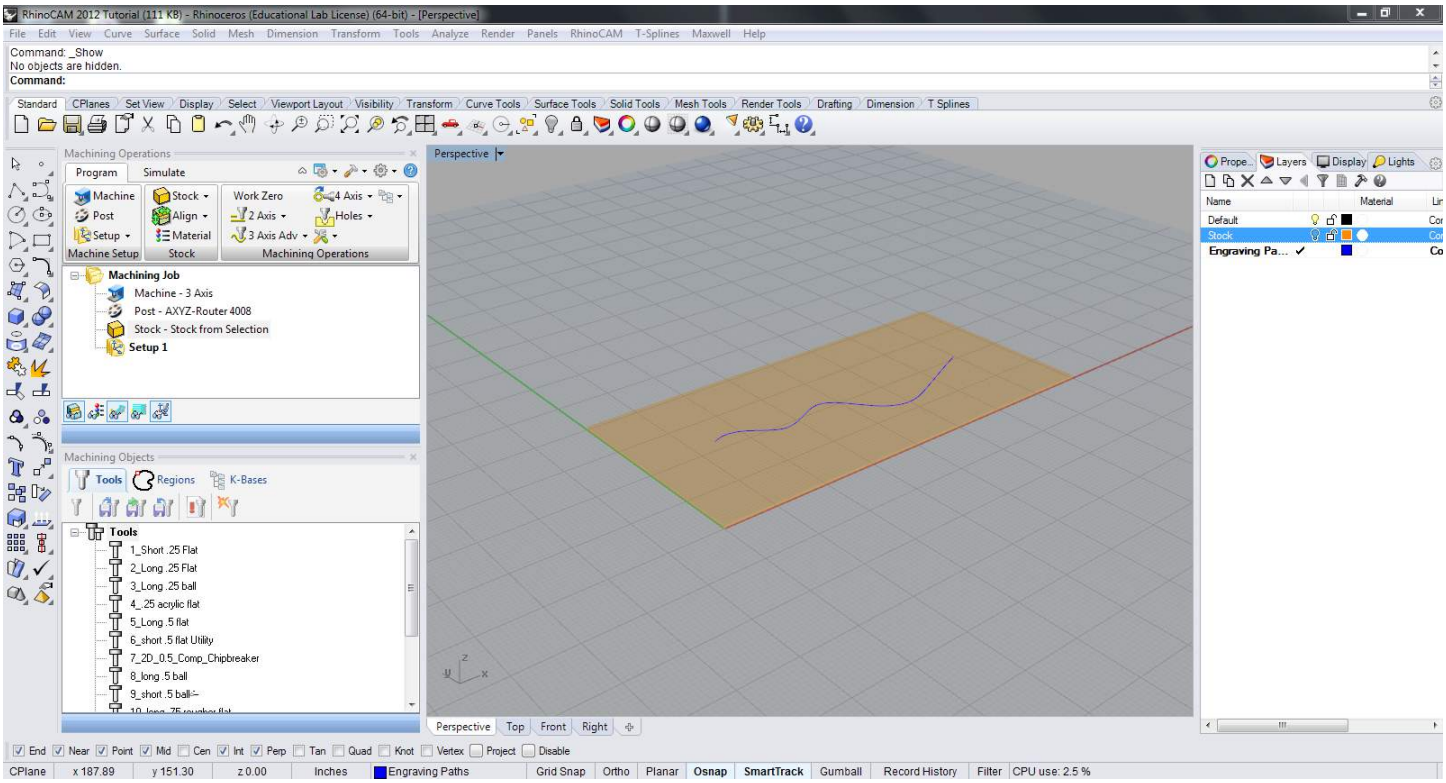
Once you have your 2D geometry ready for 2 axis milling, see the section on 2 Axis Machining Operations. If you have 3D surfacing see the section covering 3 Axis Machining Operations.

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.

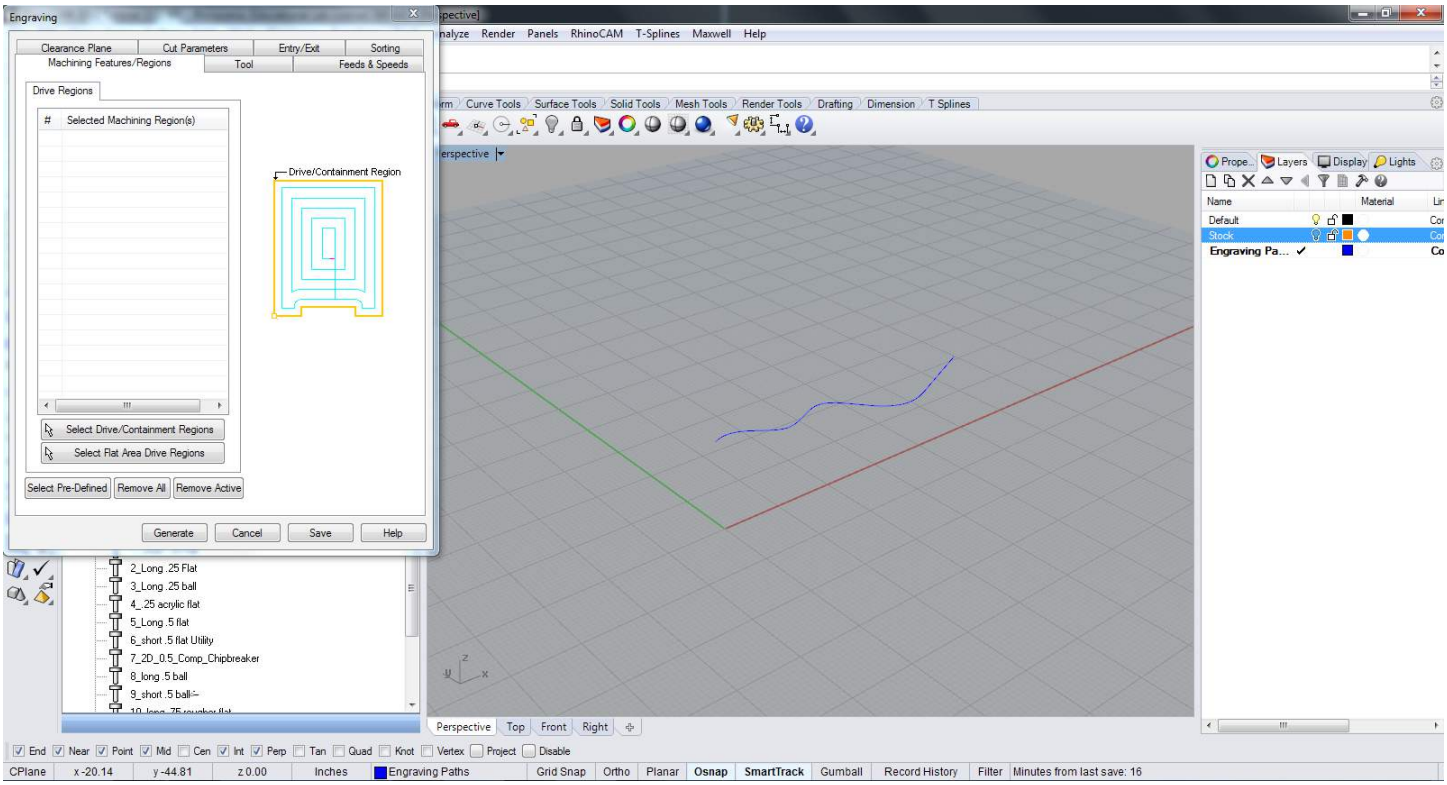




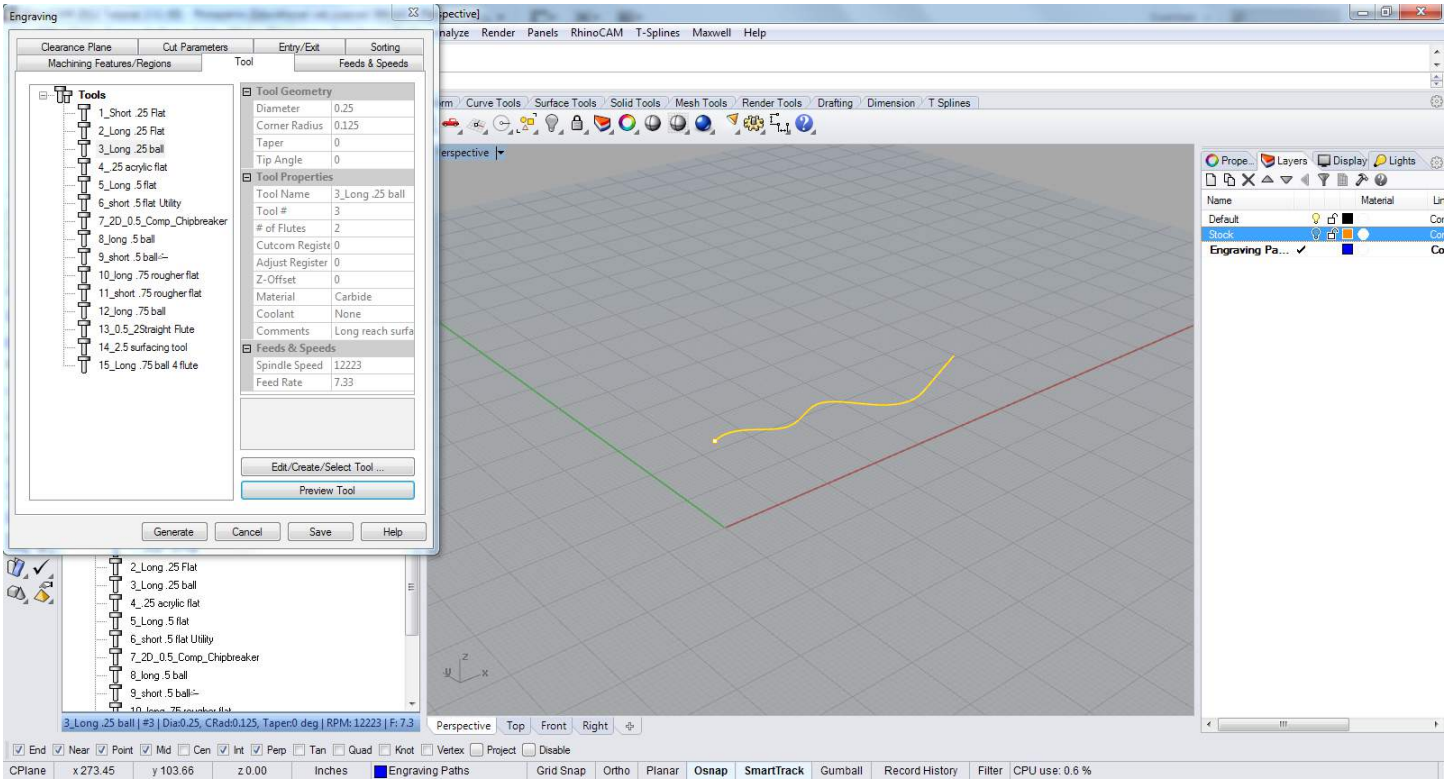
The direction of the curve can be flipped while in the Analyze direction command. These curvature directions will be important in assigning 2D milling operations as the line/ curve direction will dictate the cutting direction in your milling operations.



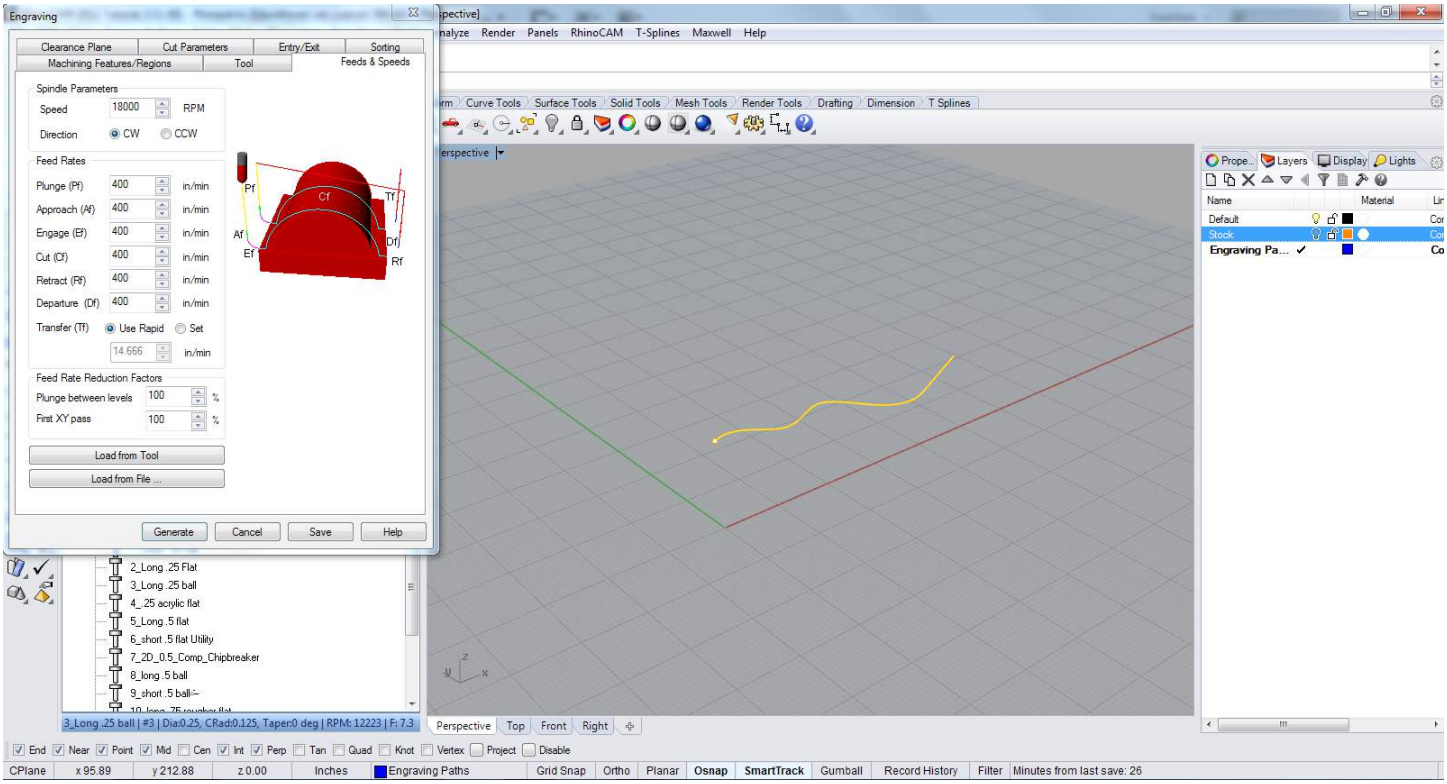
The established engraving path is aligned to the top surface fo the stock material. In the input parameters in the machining operations for engraving the cutting depth can be established.



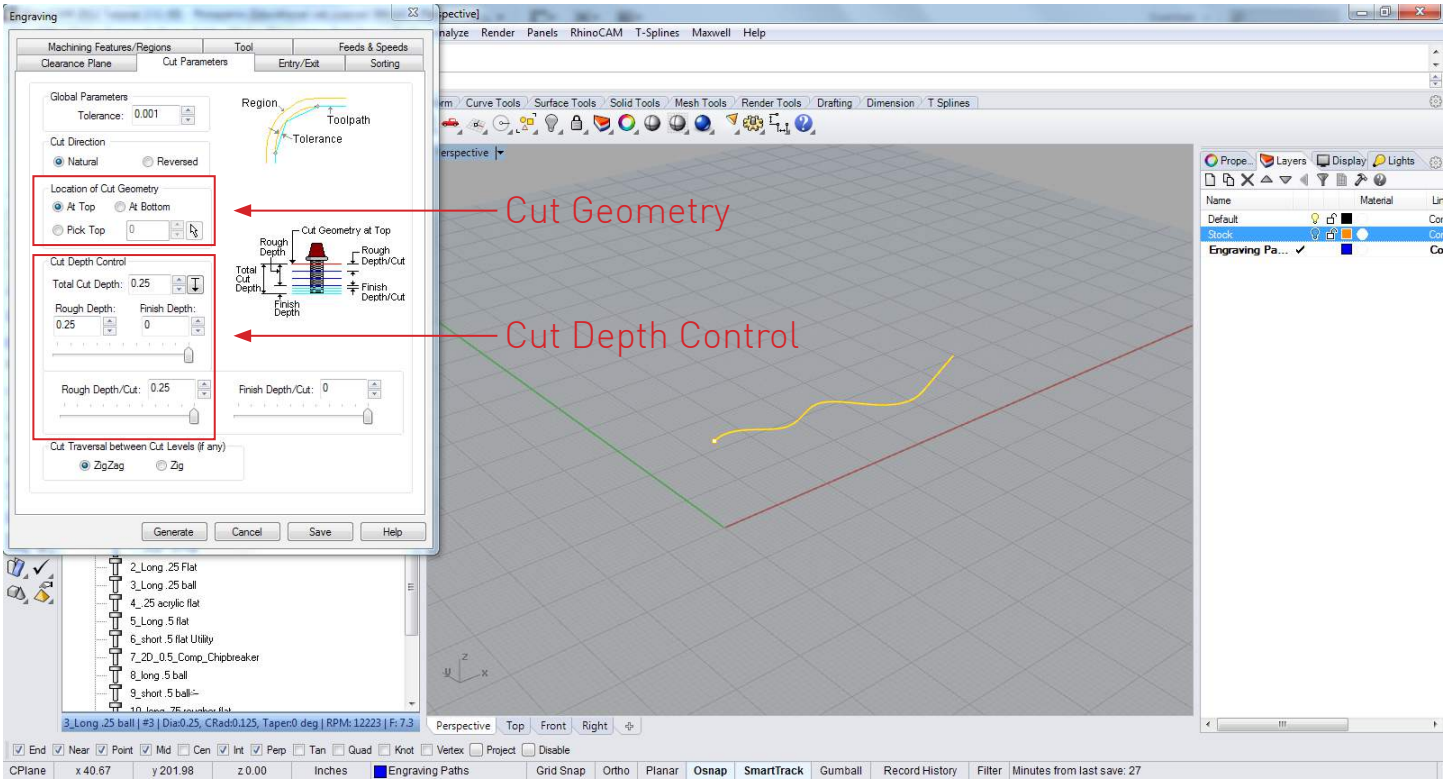
Select Engraving from the 2D milling operations tab, press select drive/containment region to select the engraving milling path(s) established.



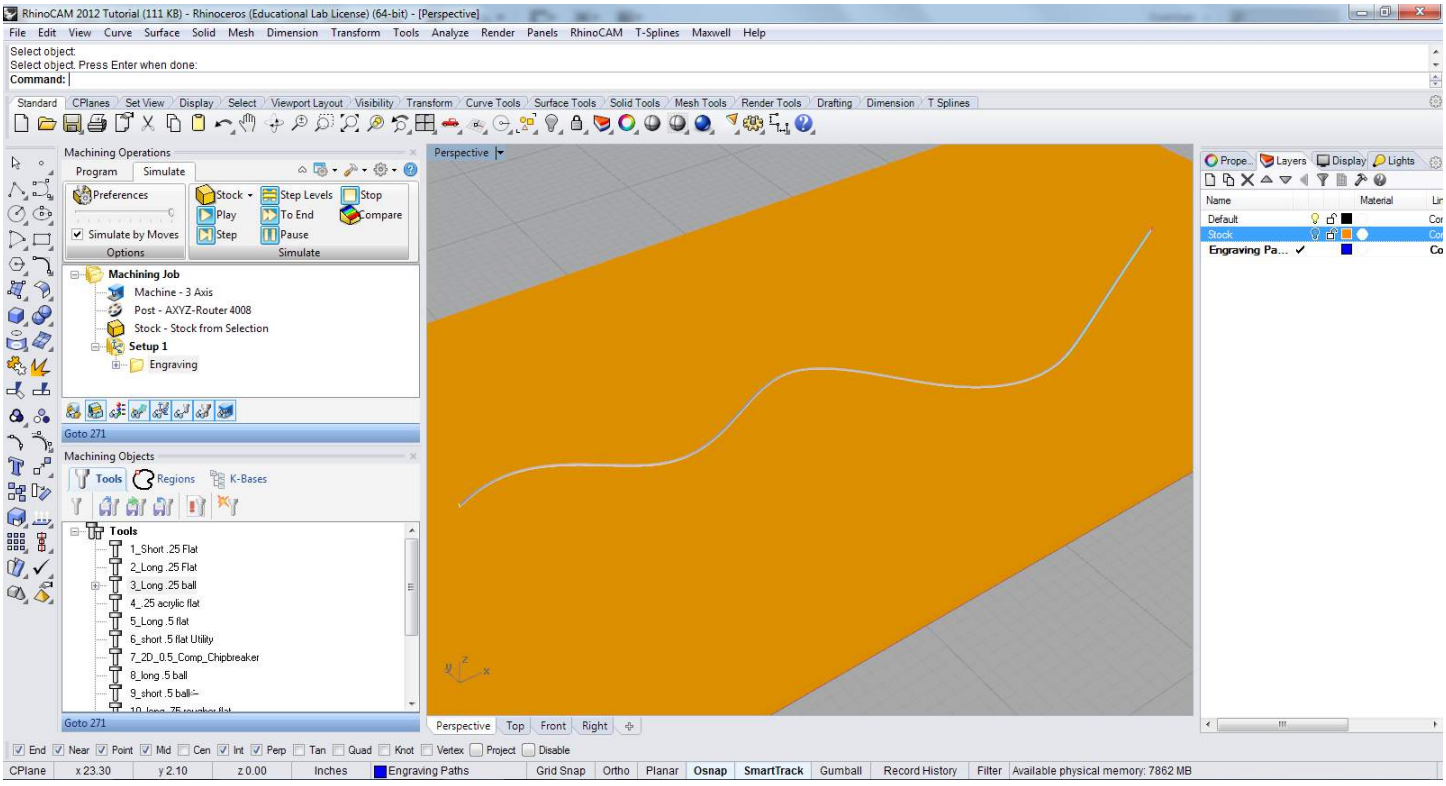
The endmill diameter selected will determine the width of the engraving line, as the endmill center point follows the selected engraving line the tooling path with be the endmill diameter from the center of the line.



The feeds and speeds will be determined by the material being milled and the type of endmill selected. Please see the feeds and speeds section and consult the Fab Lab.

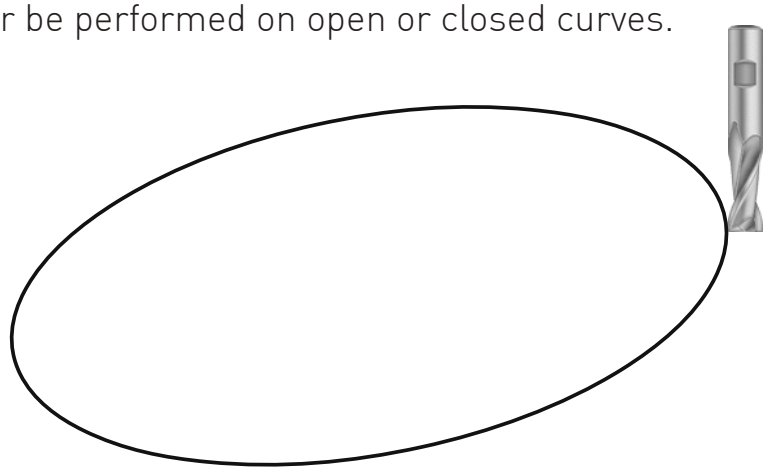


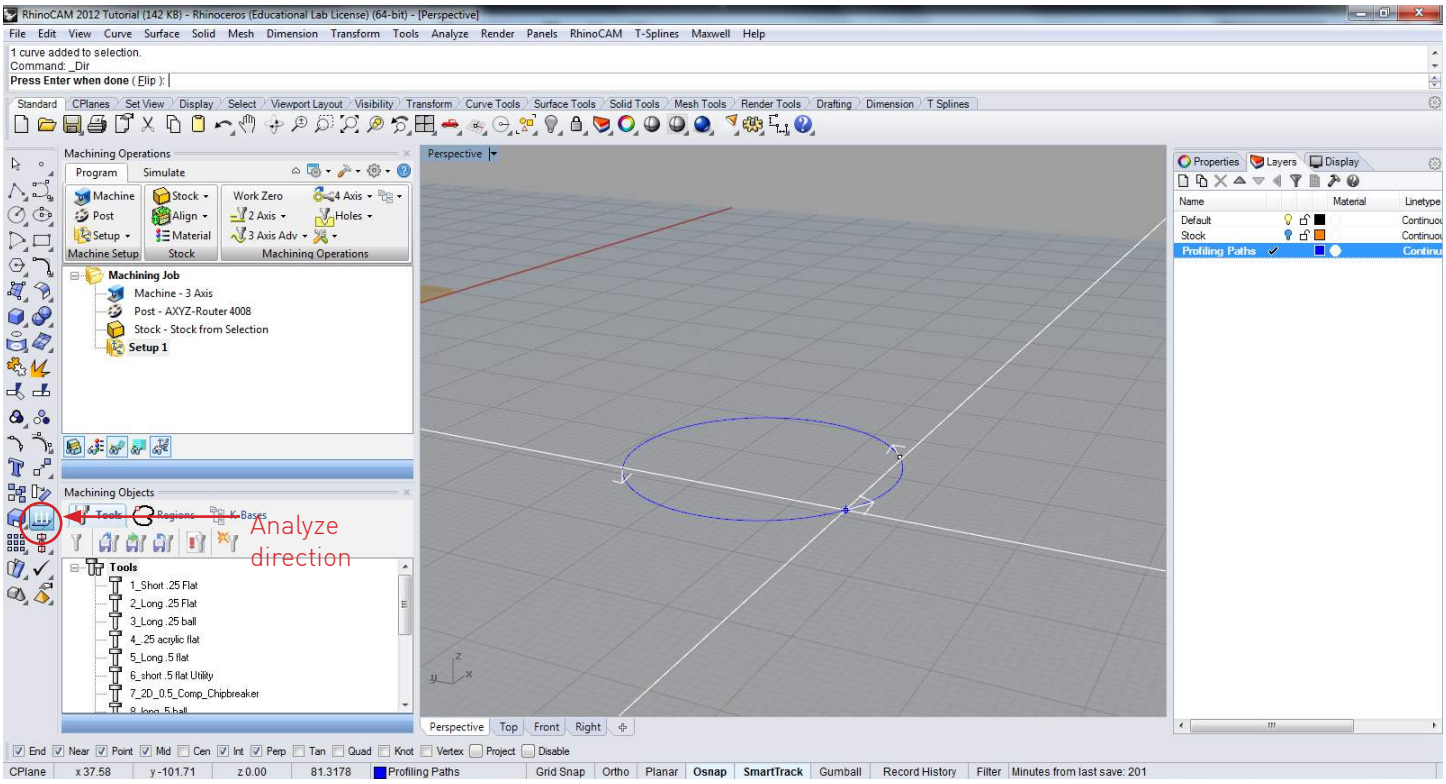
Under the Cut Parameters tab, the cutting direction, depth and geometry location is chosen. The geometry currently in the example is located at the top, the total cut depth is 1/4'', this is done in one pass, for deep milling the depth can be done in multiple passes by setting the rough cutting depth and the finish cut depth.



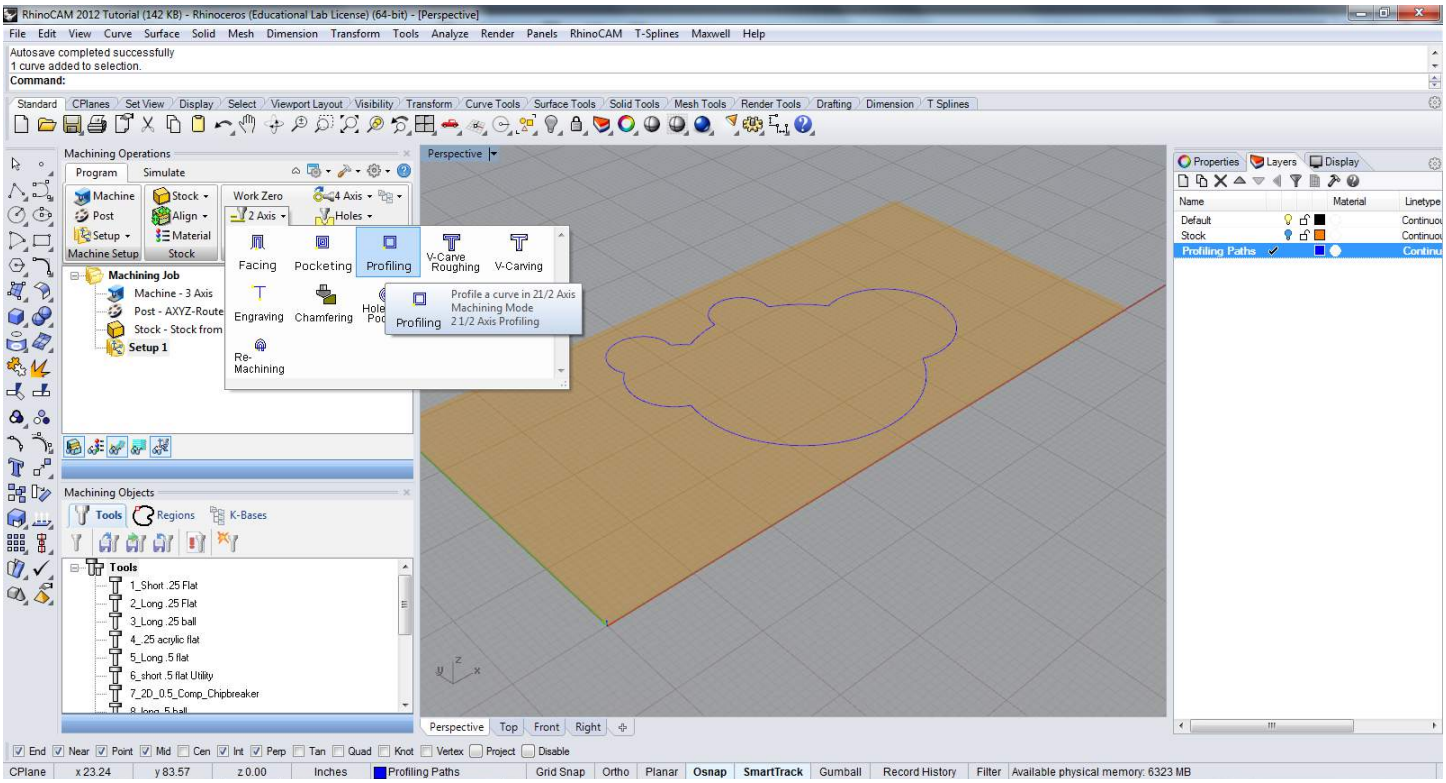
Once the milling paths have been generated select the simulate tab and run the engraving paths to see the toolpath result.

Profiling Milling paths follow curves along the directional normal of the curve, the endmill will always assign itself to the right side of the directional normal. 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.

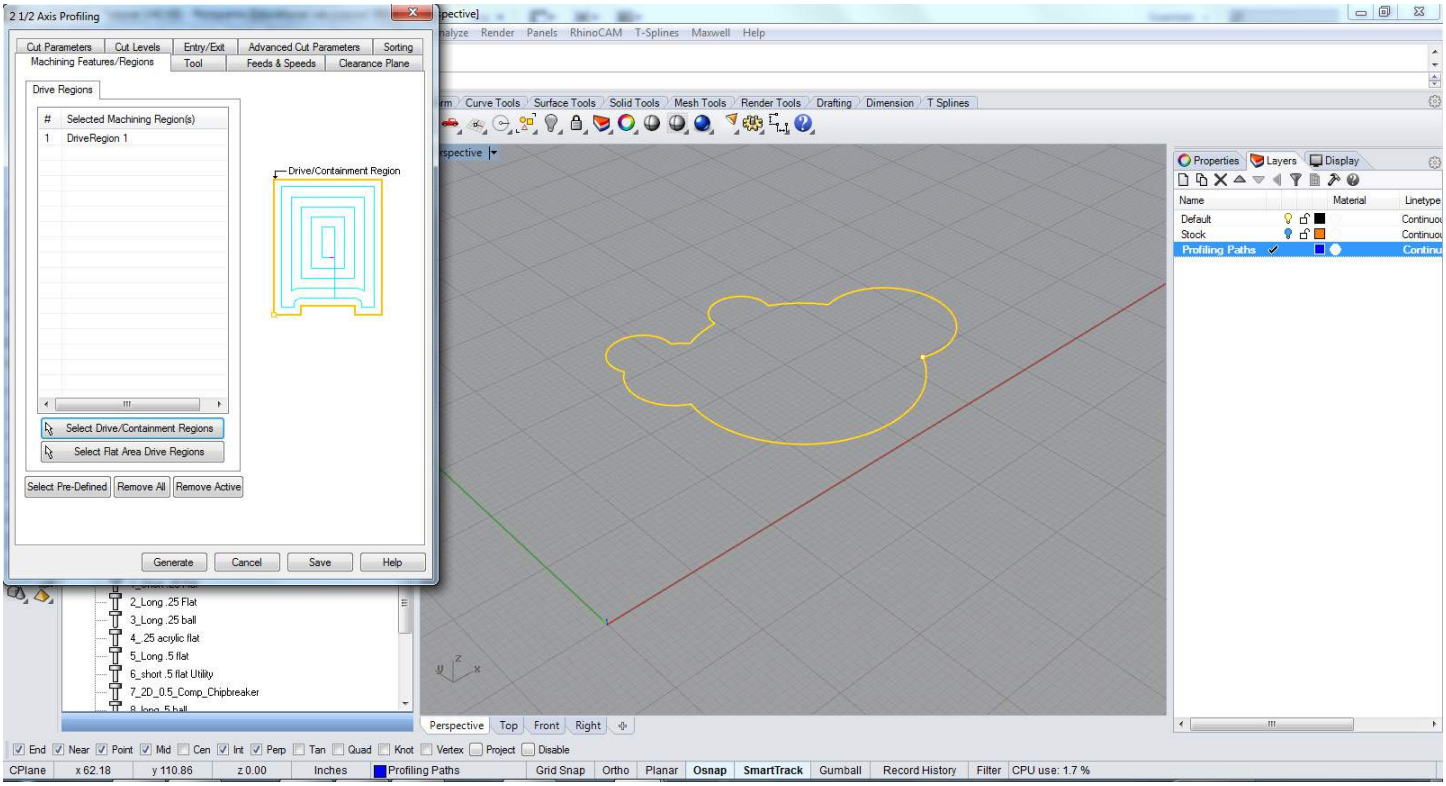


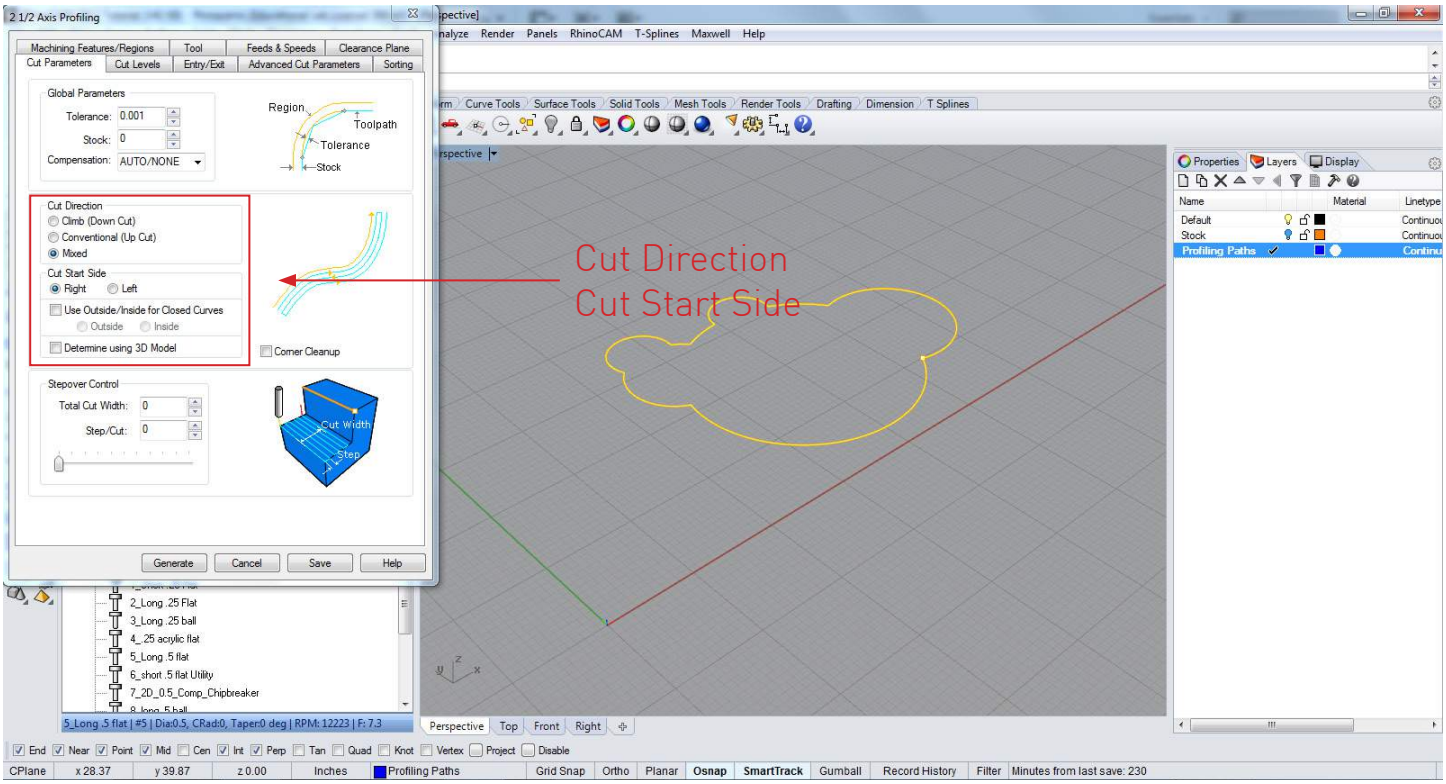


The direction of the curve can be flipped while in the Analyze direction command. These curvature directions will be important in assigning 2D milling operations as the line/ curve direction will dictate the cutting direction in your milling operations.

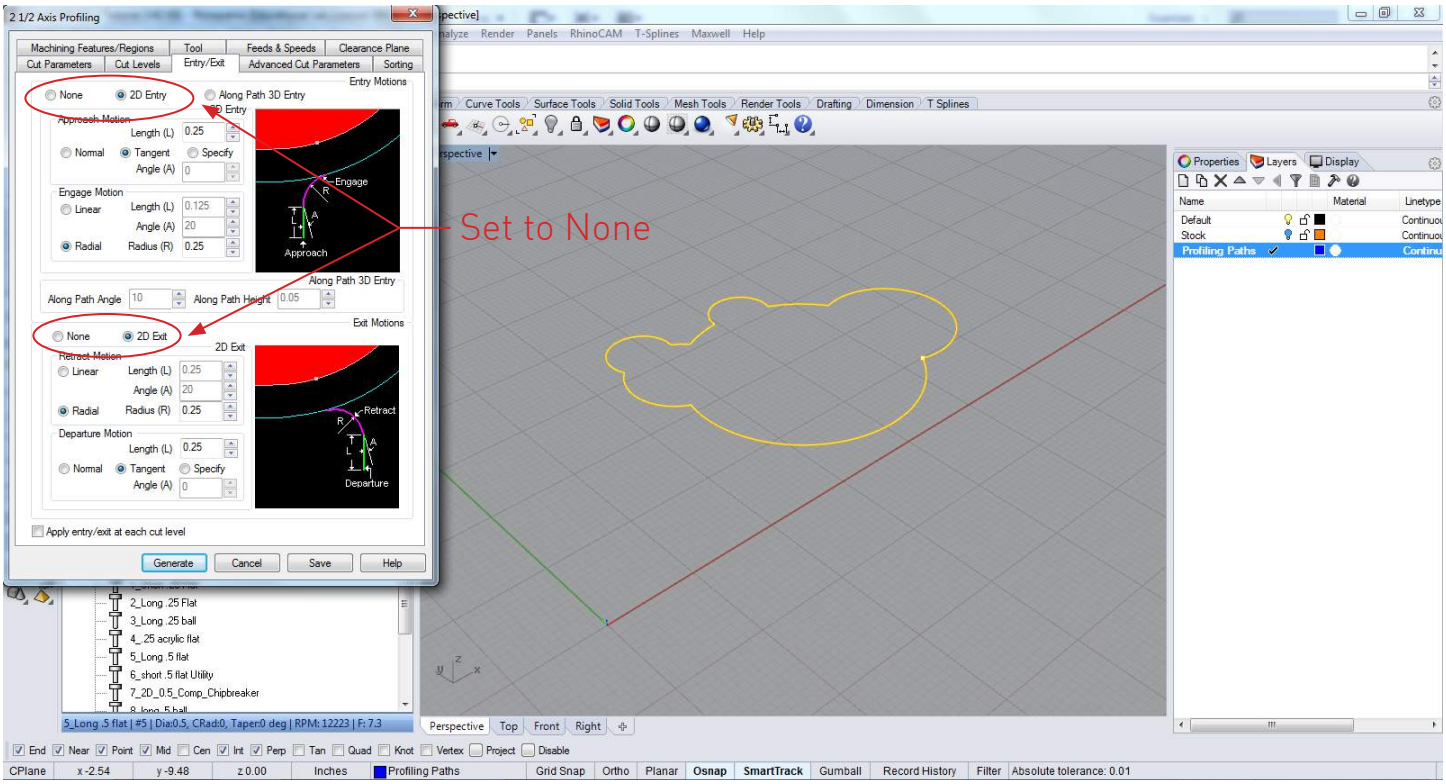


The Profiling tab is located under the 2 Axis machining functions in the program tab in RhinoCAM.

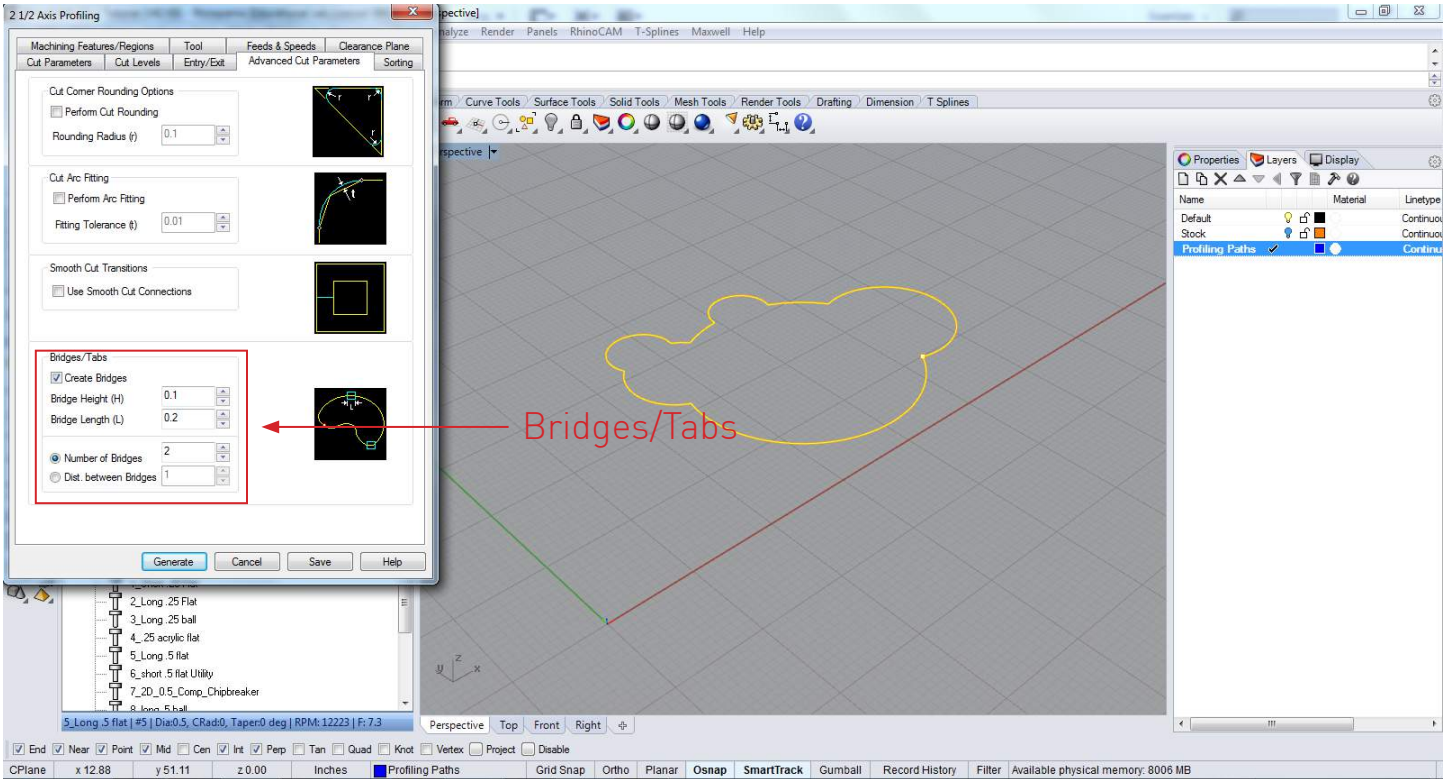




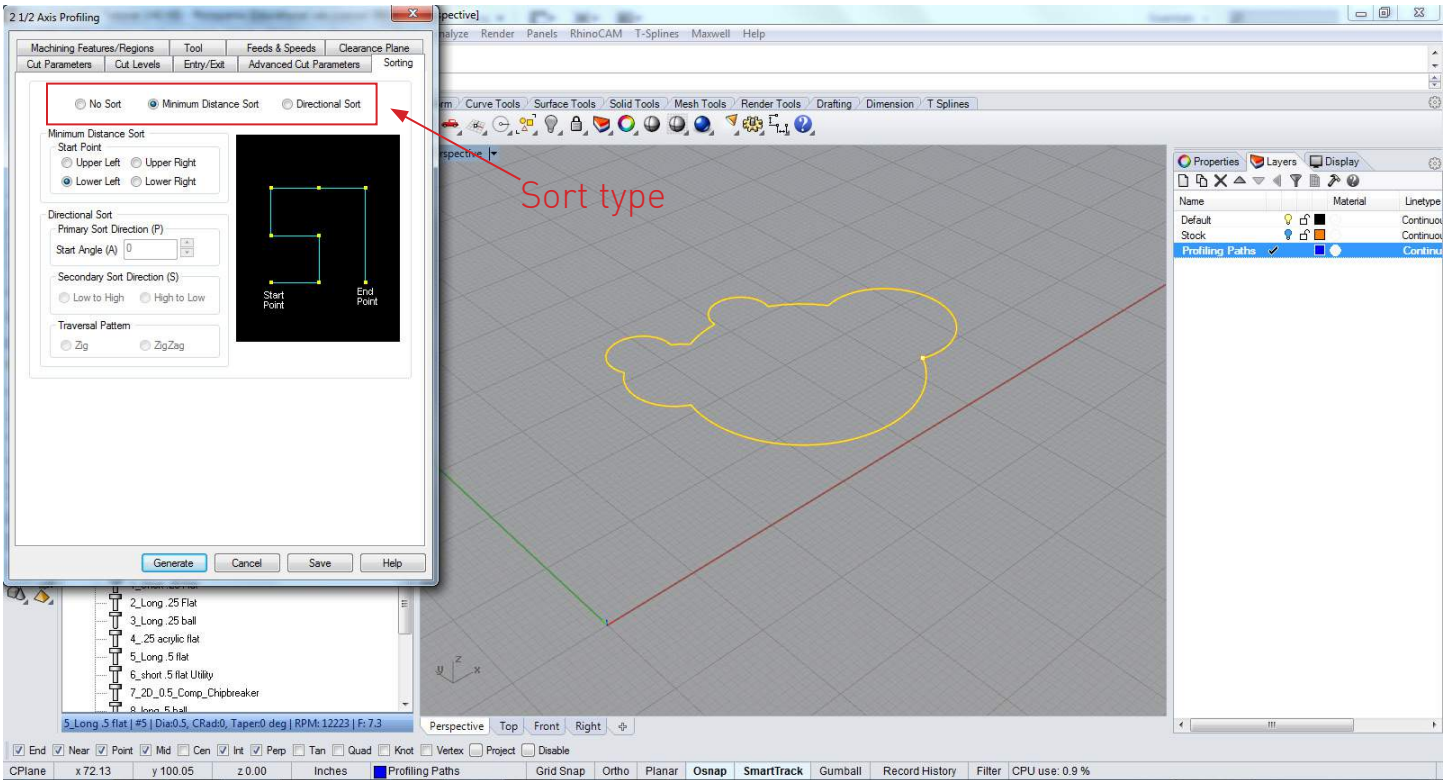
In the Cut Parameters tab the cut direction can be changed and for closed curvature either the inside or the outside of a closed curve can be chosen to cut. A offset of the profile can be setup with Stepover Control, the cut width and the step over to the desired width can be set up here. As a general rule never cut deeper or step over more then half of the diameter of the chosen endmill, for denser materials this should decrease.



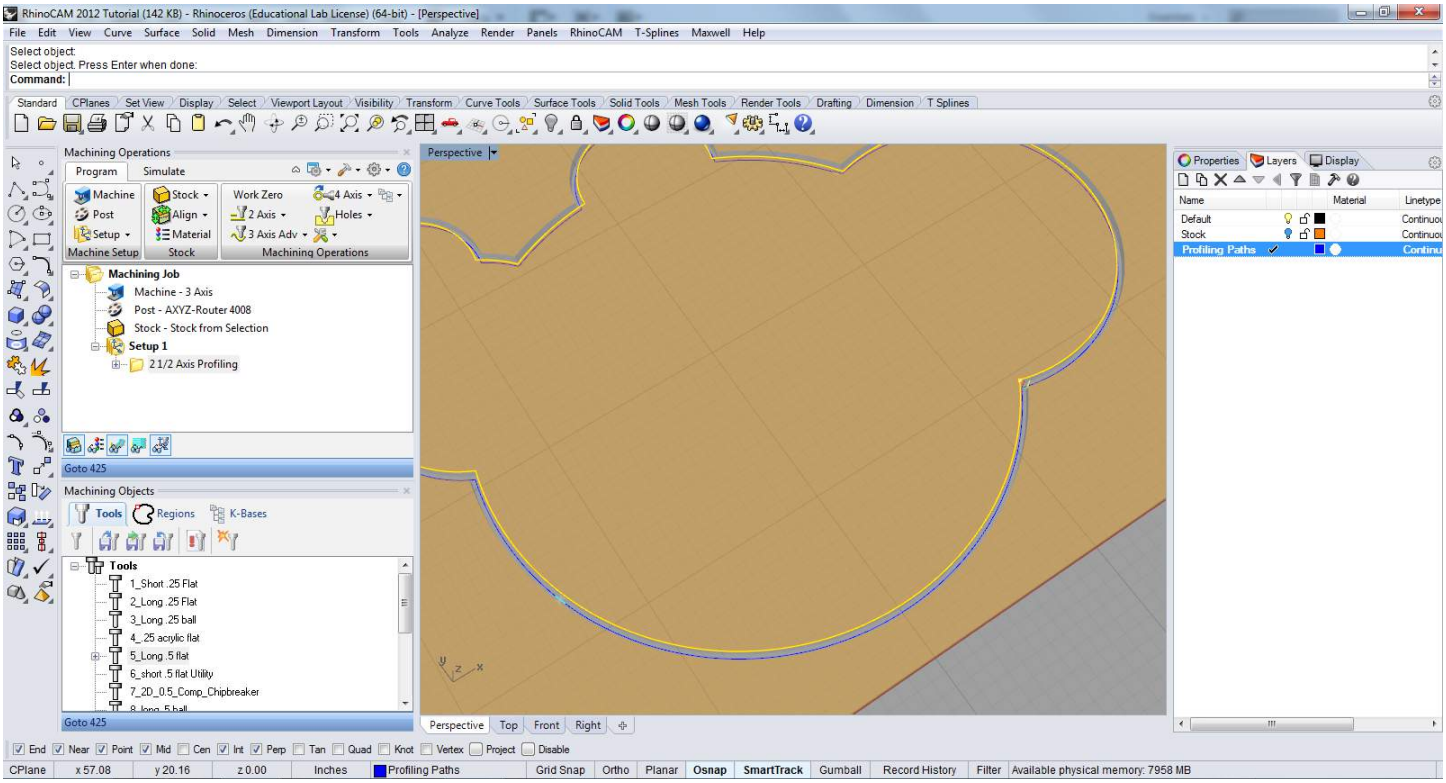
Under the exit / entry tab set the entry and exit motions to none, the 2d exit will create a turn out cut that can come close to other profiles if there are tightly nested.



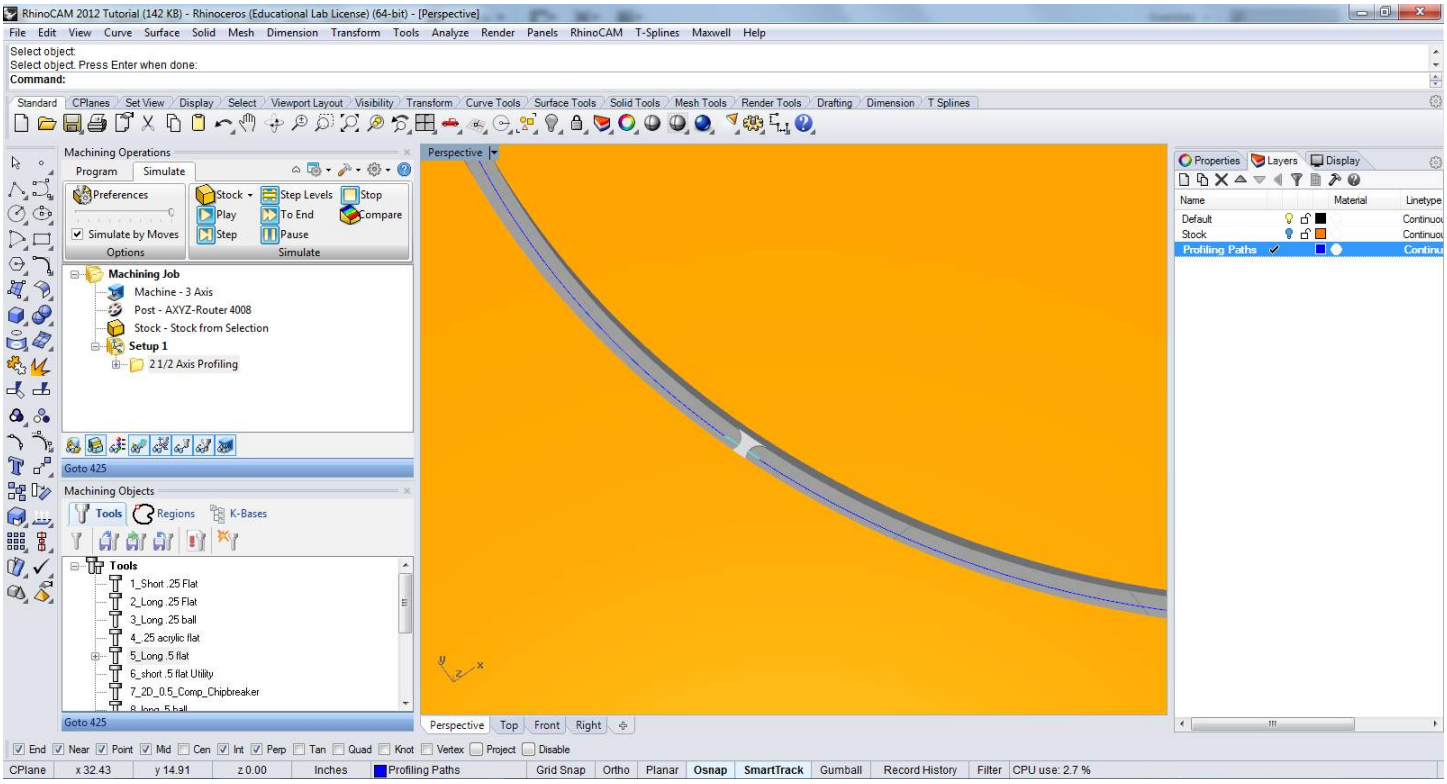
When cutting a profile out of a larger surrounding material the inner cut can lose suction to the CNC vacuum bed once cut. This can cause it to shift and bump into the endmill. To prevent this bridges can be created, these create small tabs that link the cut piece to the larger material. The height,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



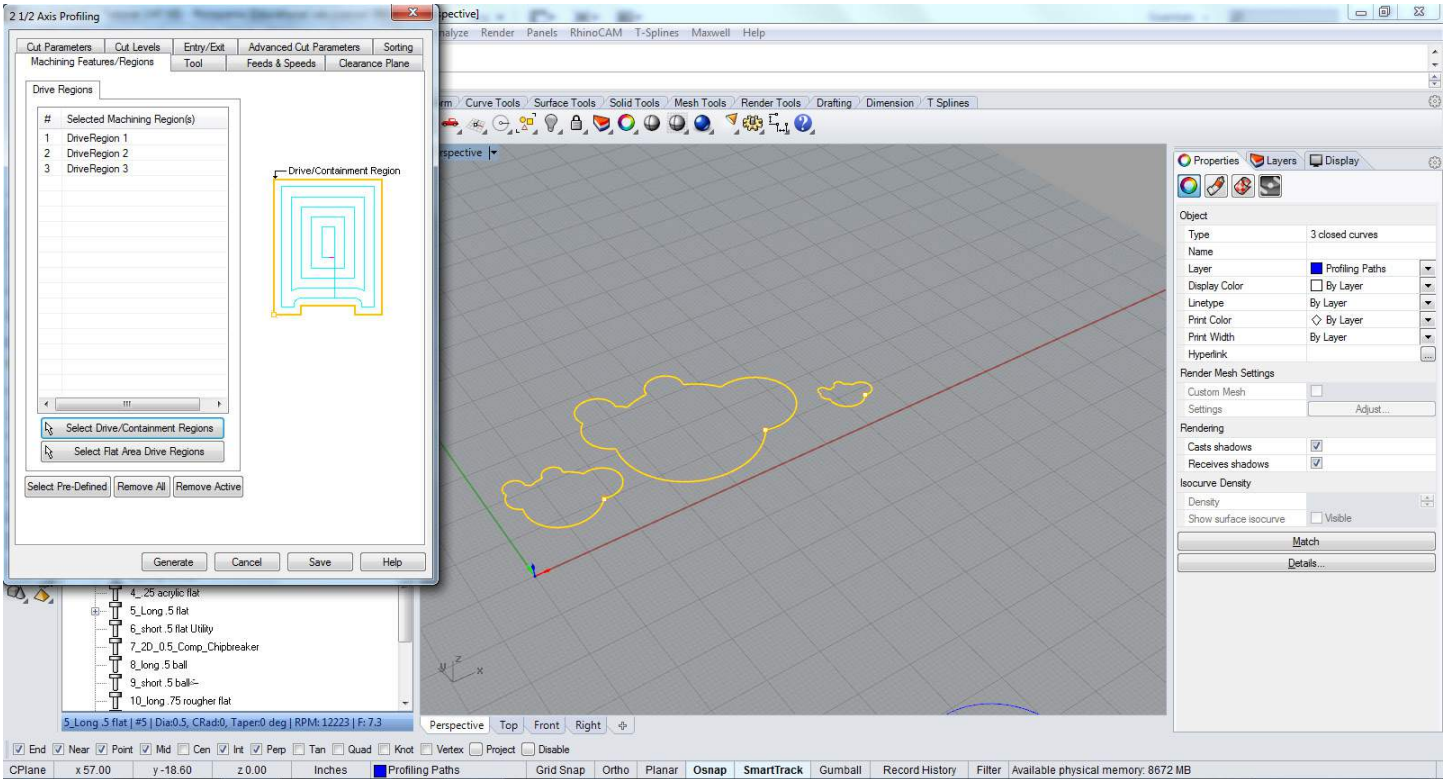
If multiple profiles are being cut in a single sheet the cutting order can be set up under the sorting tab, this allows the shortest path generated from a starting point.



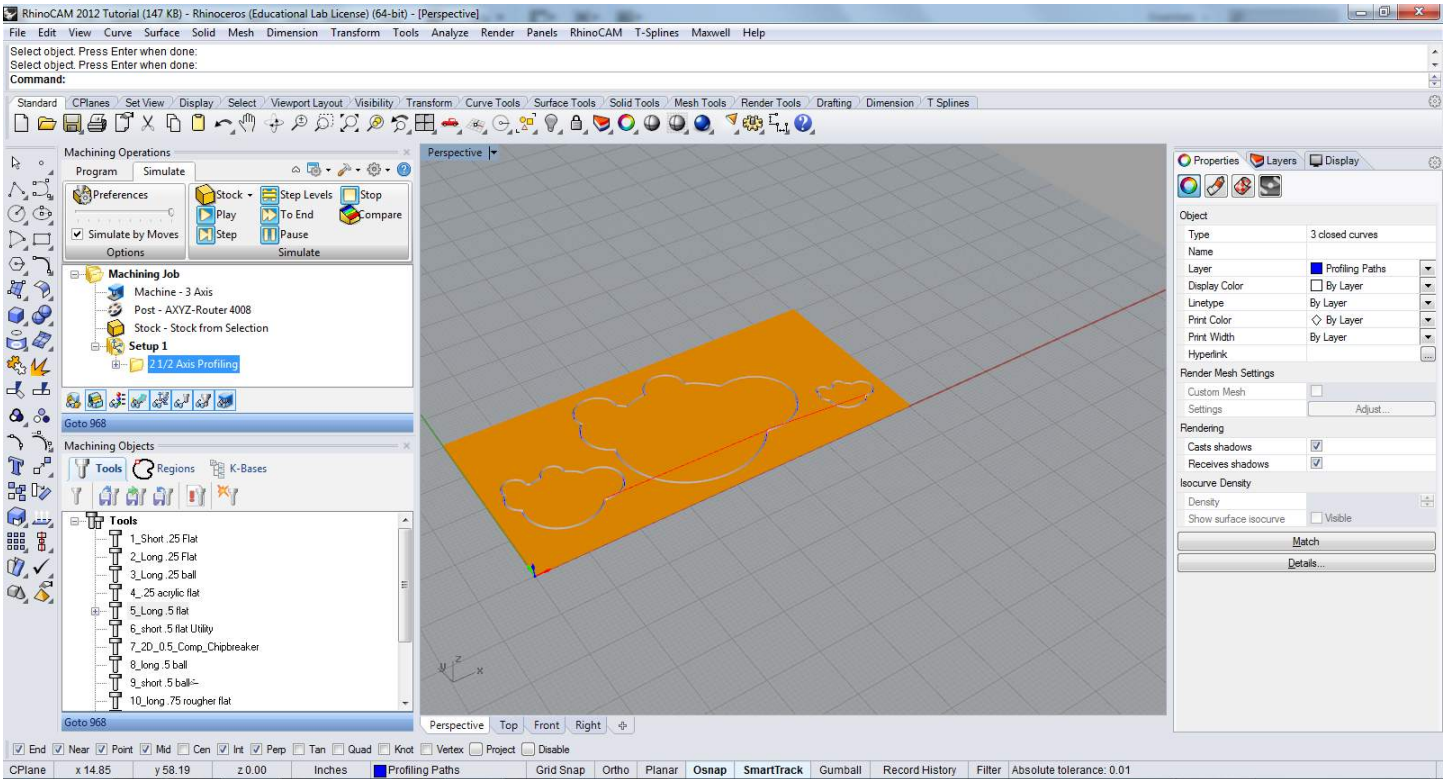
Once generated the profiling path will appear in the Program Tab view, the light blue lines are the bridges and the dark blue is the profile tooling path.



Under the Simulate tab once the simulation has been run the bridges are visible linking the profile cut back to the main sheet material.

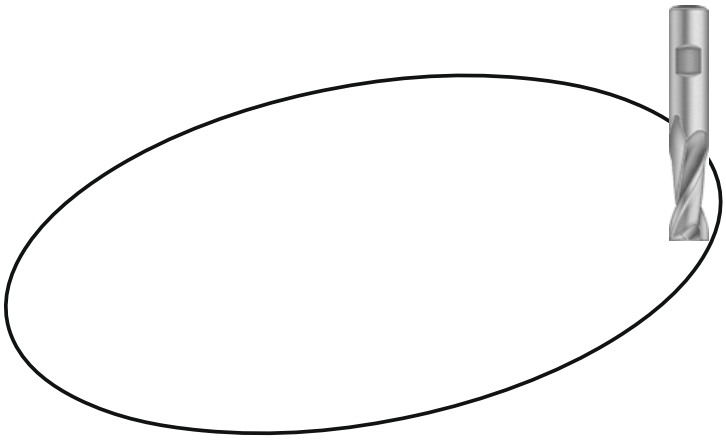


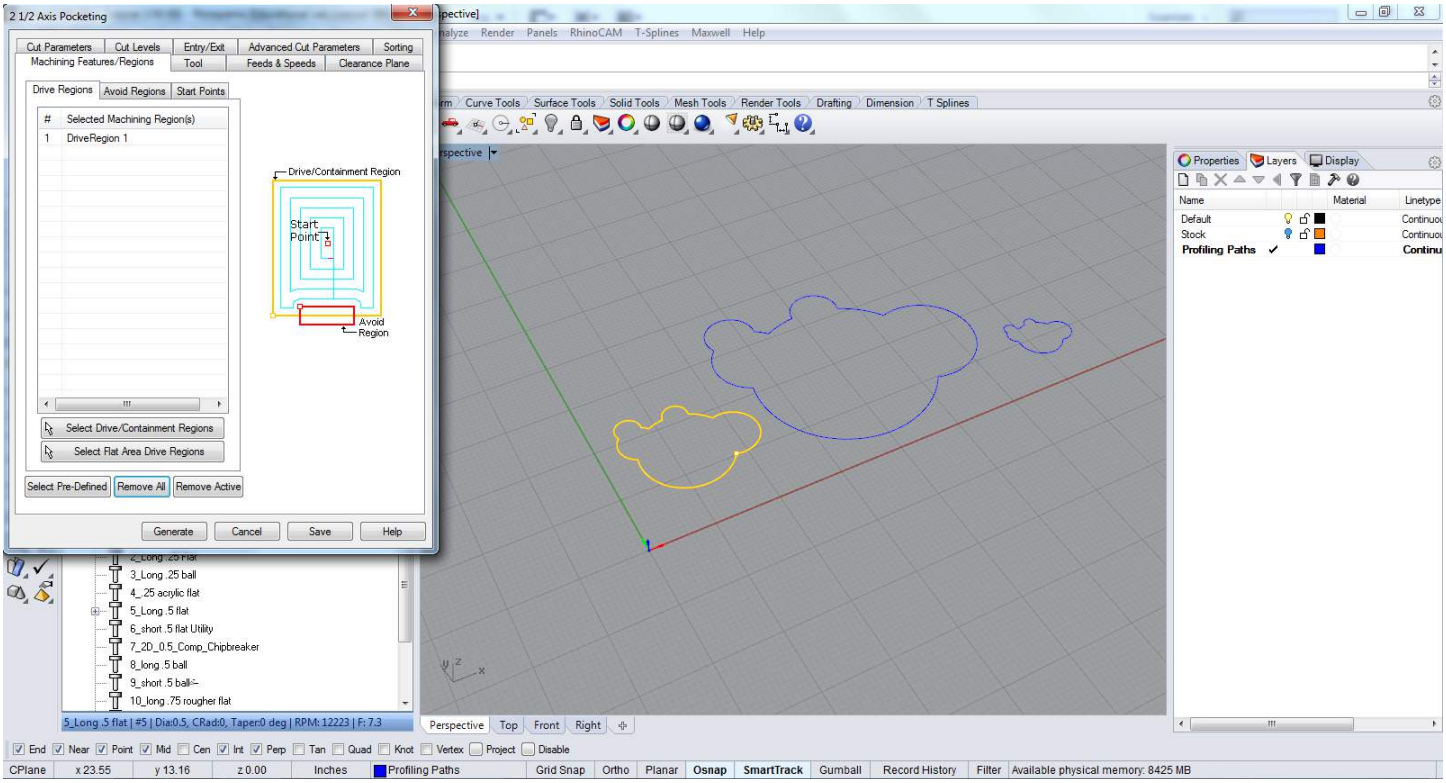
For multiple profiles, select all the drive regions as long as the cutting operation performed remains the same. Under the Sort tab select minimum sorting distance.



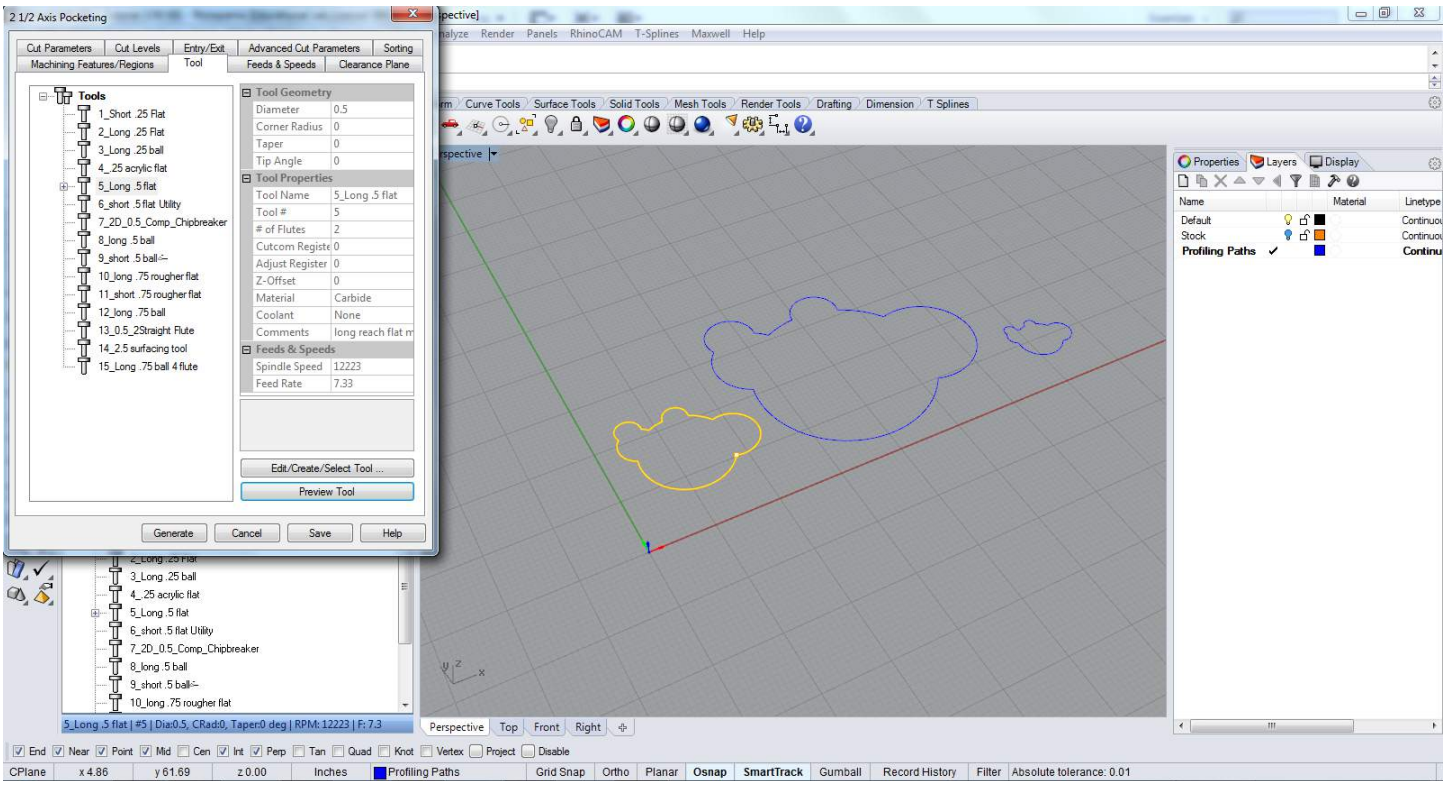
The tooling paths generated will be sequential from the start regions selected from the sort tab, this will speed up operations when multiple profile cuts are performed.

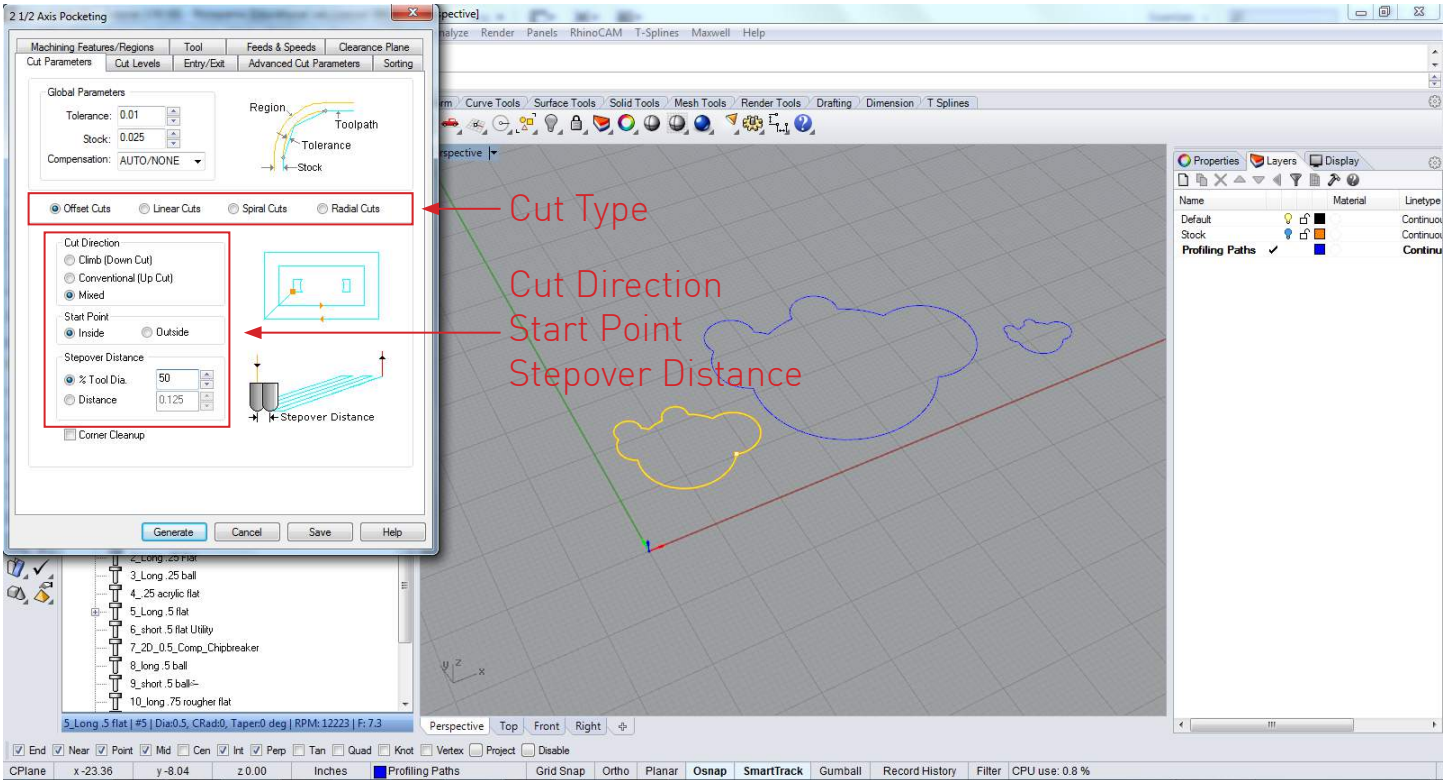
Pocketing milling operation will mill out a selected region, either within a bounding regions our between two bounding areas. The generated paths will populate the region with milling paths to remove the desired material to a depth assigned in the cutting depth operation.



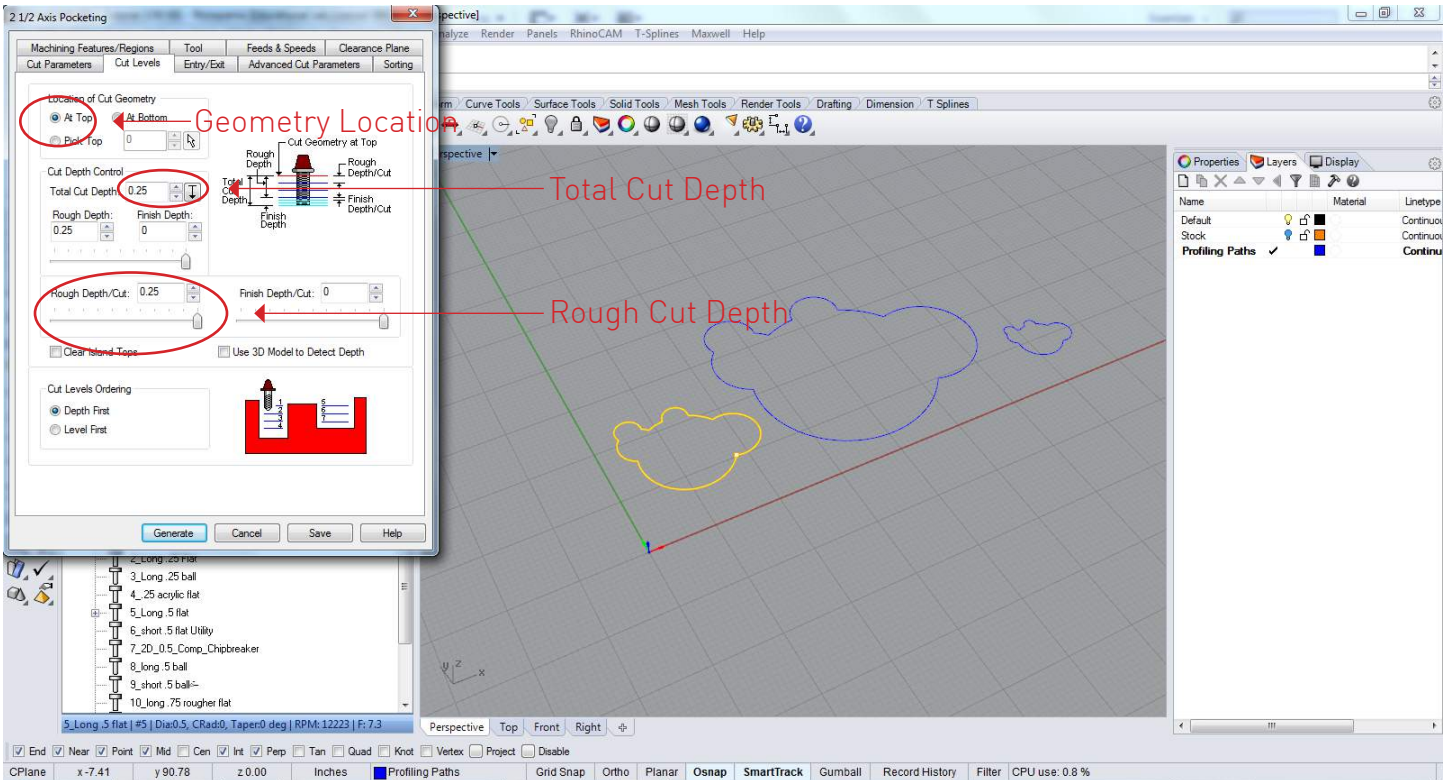


Using the same geometry as in the profiling path, select the desired drive/containment region. For a closed region the profiling path will be generated inside of the contained region

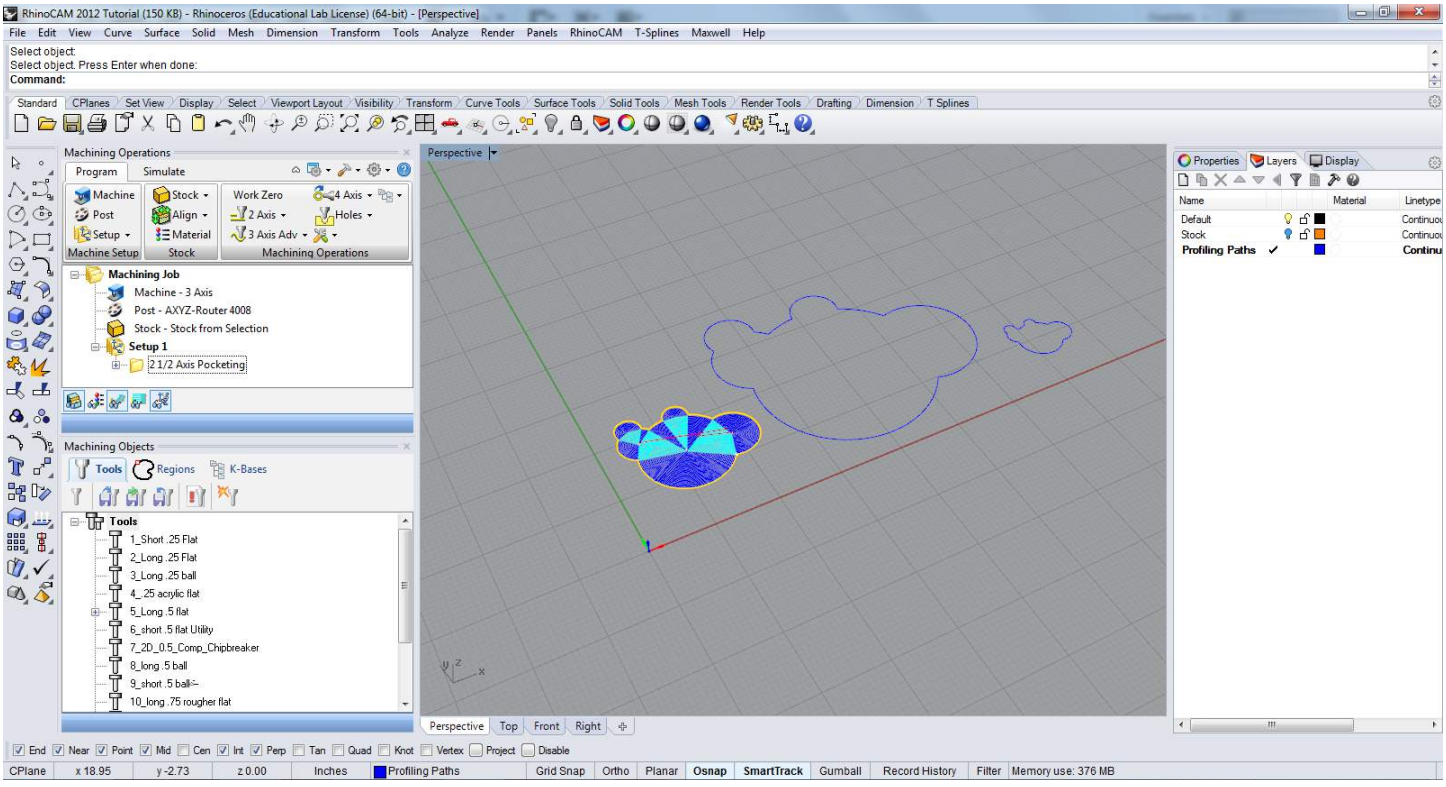




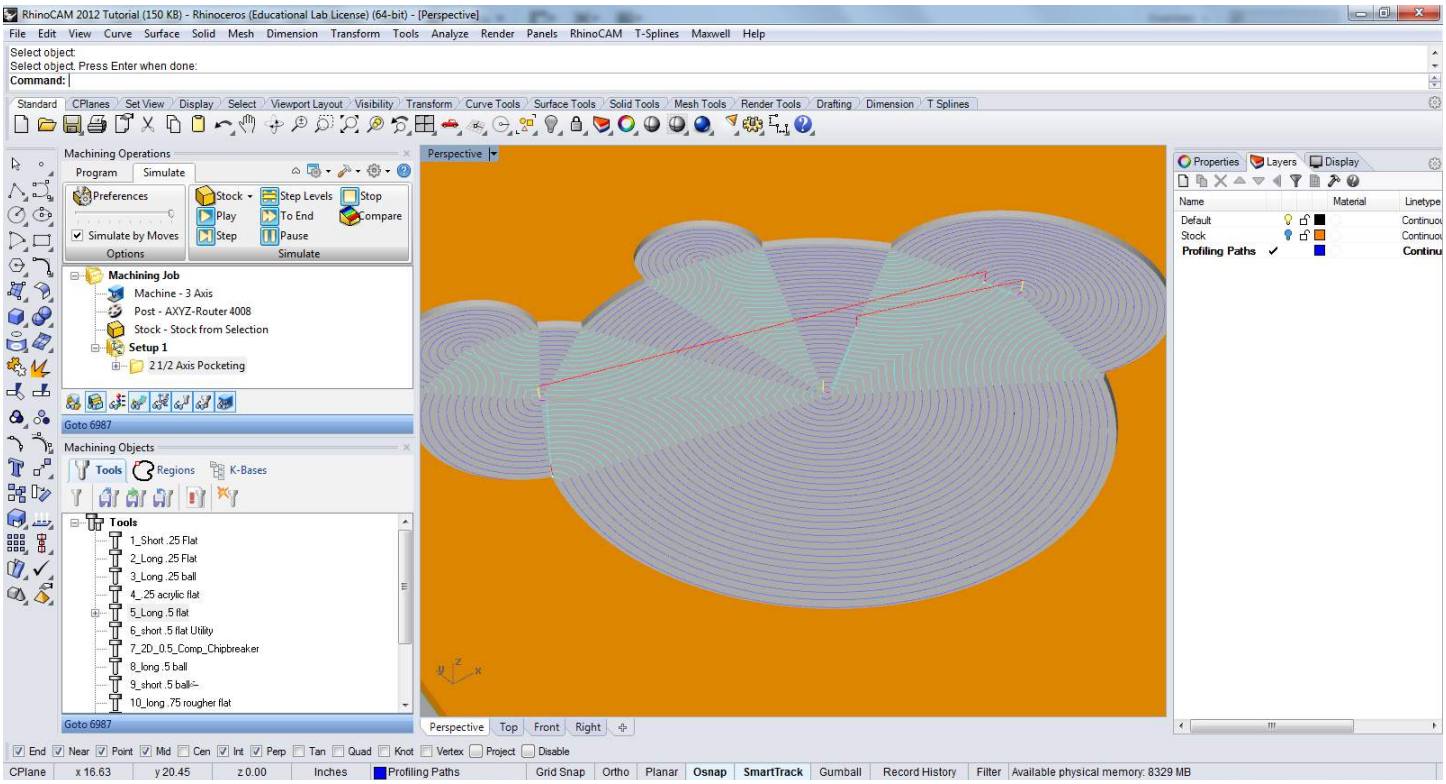
Under Cut Parameters the pocket can be cut using either Offset, Linear, Spiral or Radial cuts. The choice of operation will be dependent of the geometry to be pocketed, for the curvature shown the offset cut will be used starting from the inside to perifery of the geometry, The stepover is set to 50%.



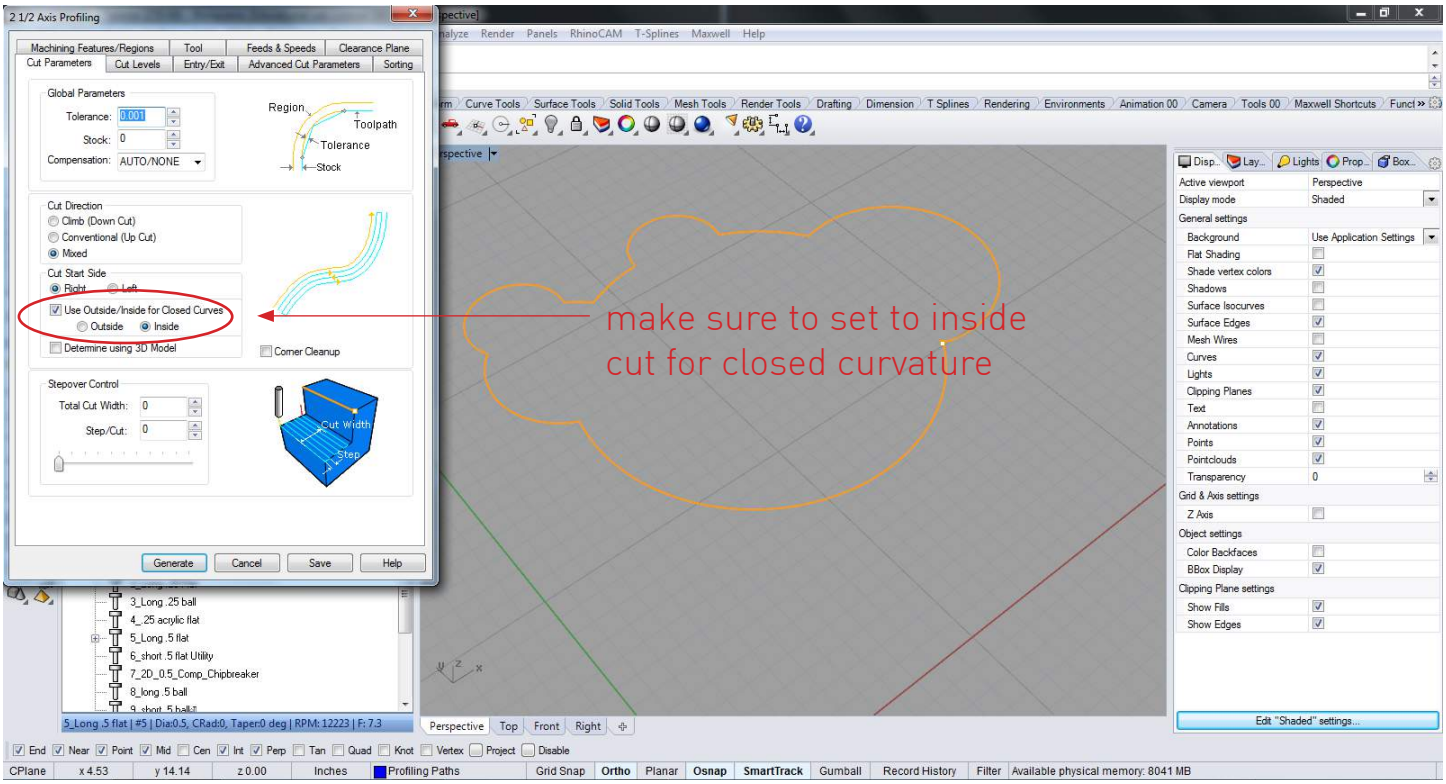
Under Cut Levels the depth and step down of the cut is defined, chose the geometry location, the curves in the example are located at he top, the rough cut depth is set to match the total depth as the cut is only .25'' if great the rough cut depth is set to step down, this is done through the slider to set the steps to the total cut depth



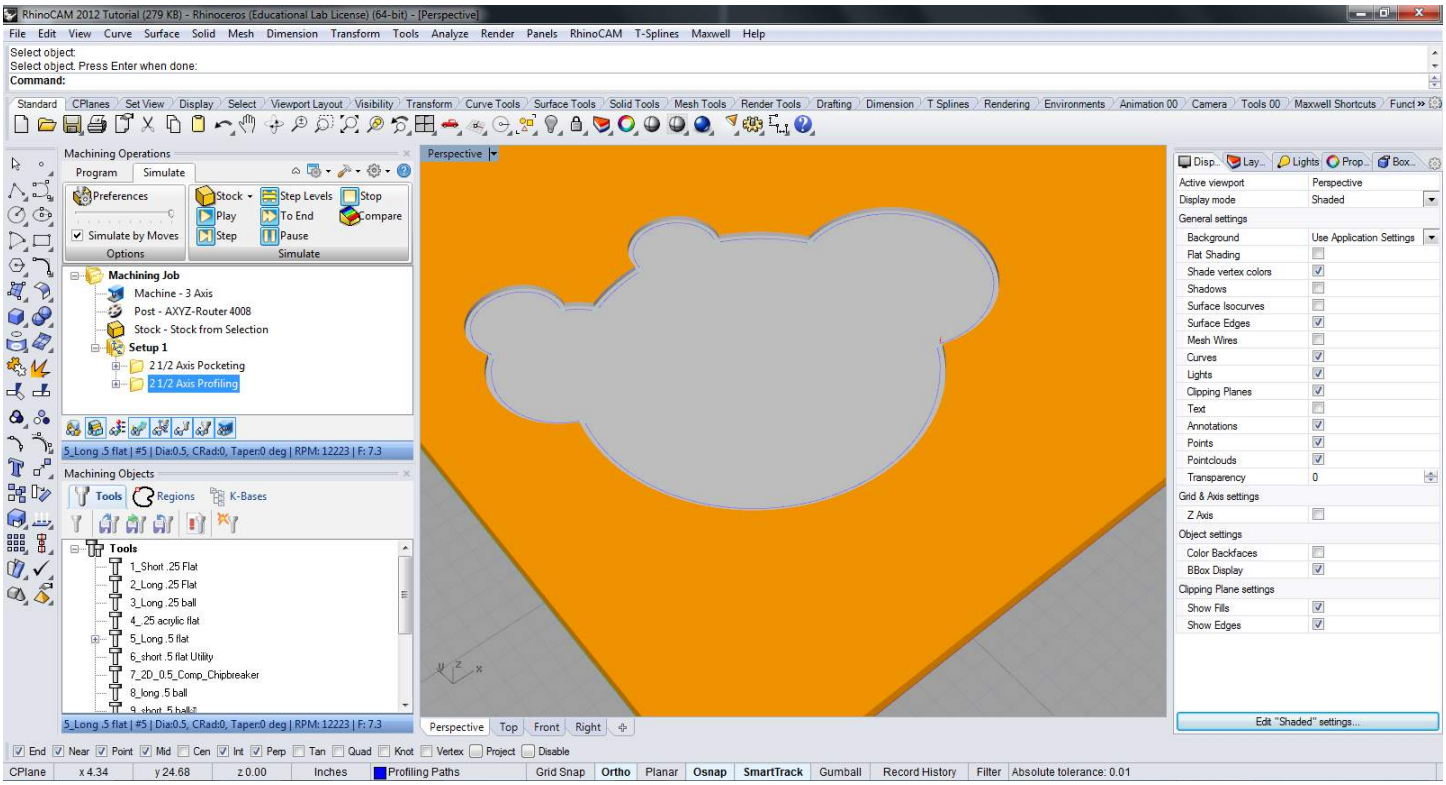
Once all the parameters have been set press generate, the tooling paths will appear in a offset orientation starting from the inside working outwards to the curvature.



Go to the simulate tab to see the generated toolpaths run, with pocketing passes sometimes the pocket geometry will be slightly inset from the original curve, a profile pass can be run with the pocket to insure a tight alignment to the original curvature.



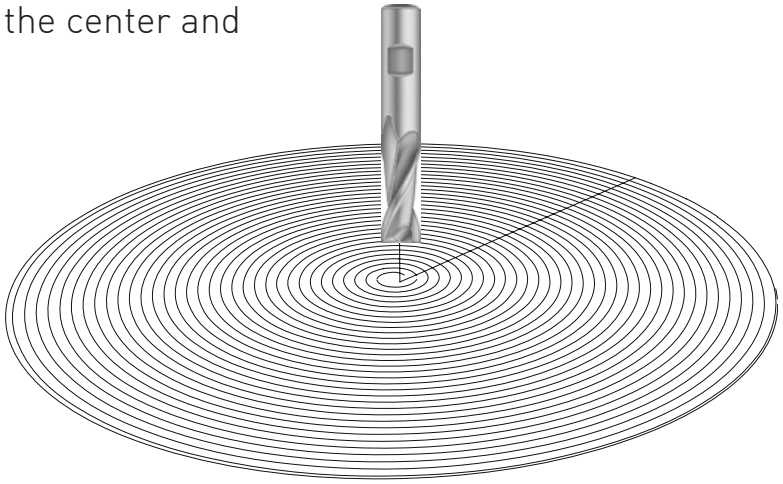
Select the same curvature used for the pocketing pass, in Cut parameters make sure the use outside/inside curves is selected and set to inside. set the depth the same as the pocketing pass.

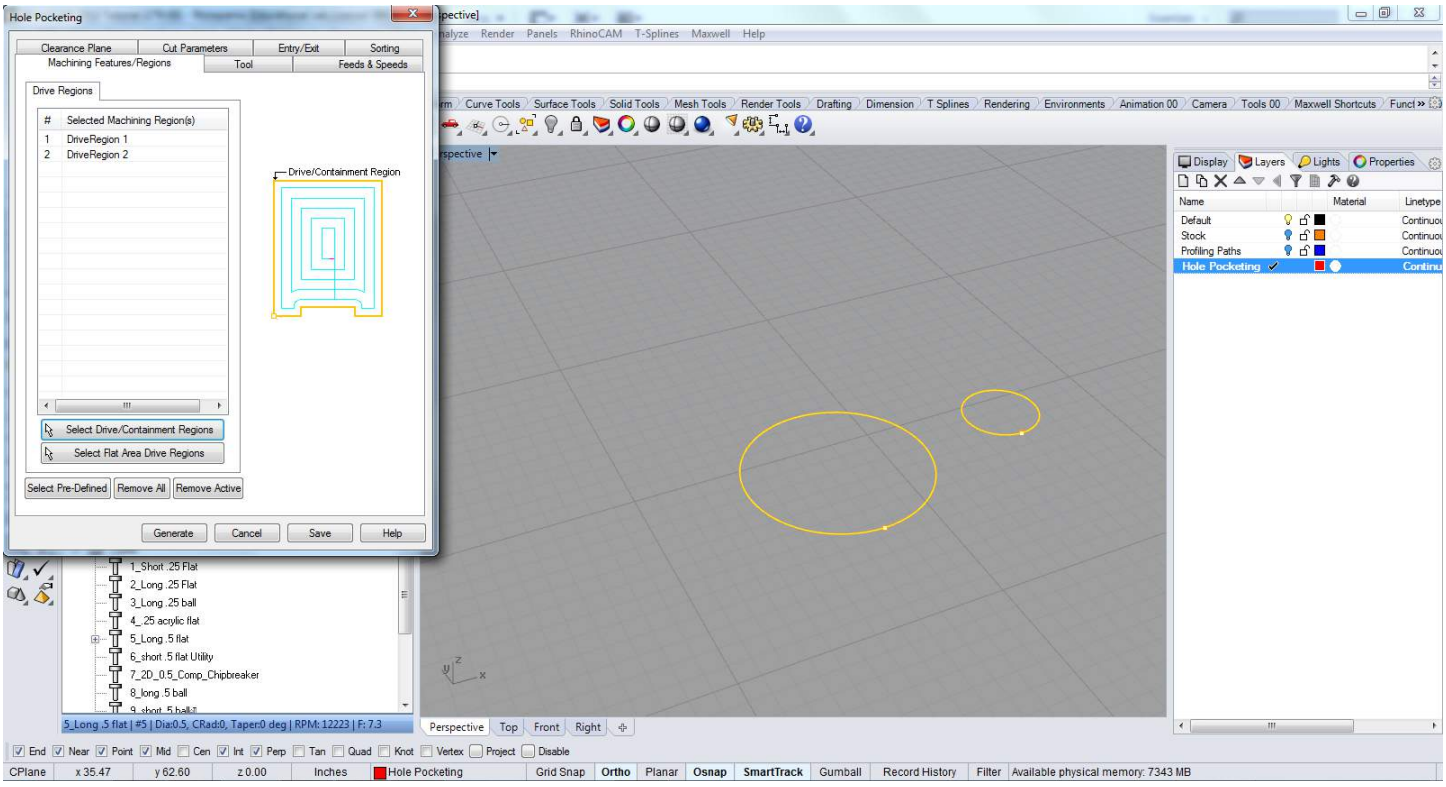


Once the profiling pass is simulated you will see a small stepover from the pocketing pass to the profiling pass, the new cut profile is tight to the original curvature.

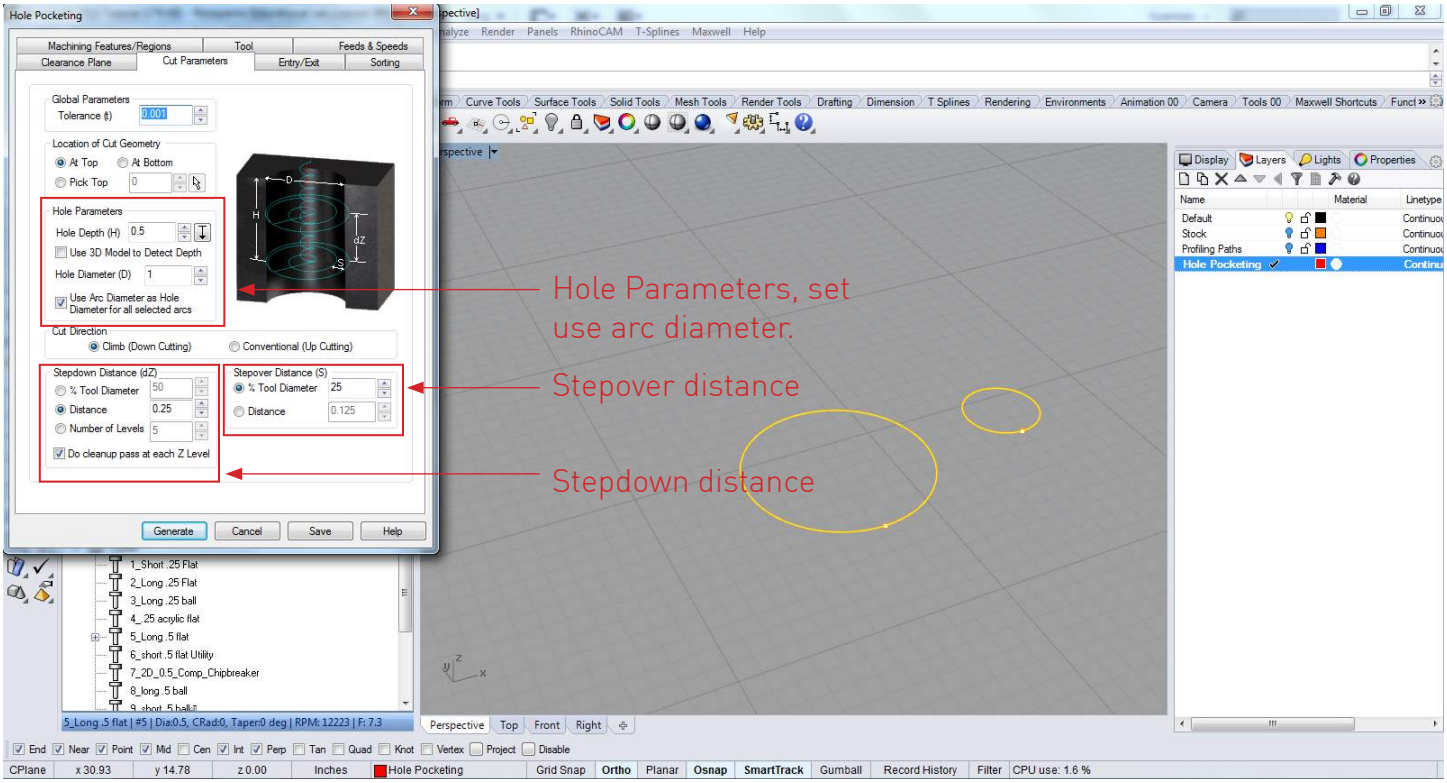
Rhino_CAM 2 Axis Machining Operations_ Hole_Pocketing

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.

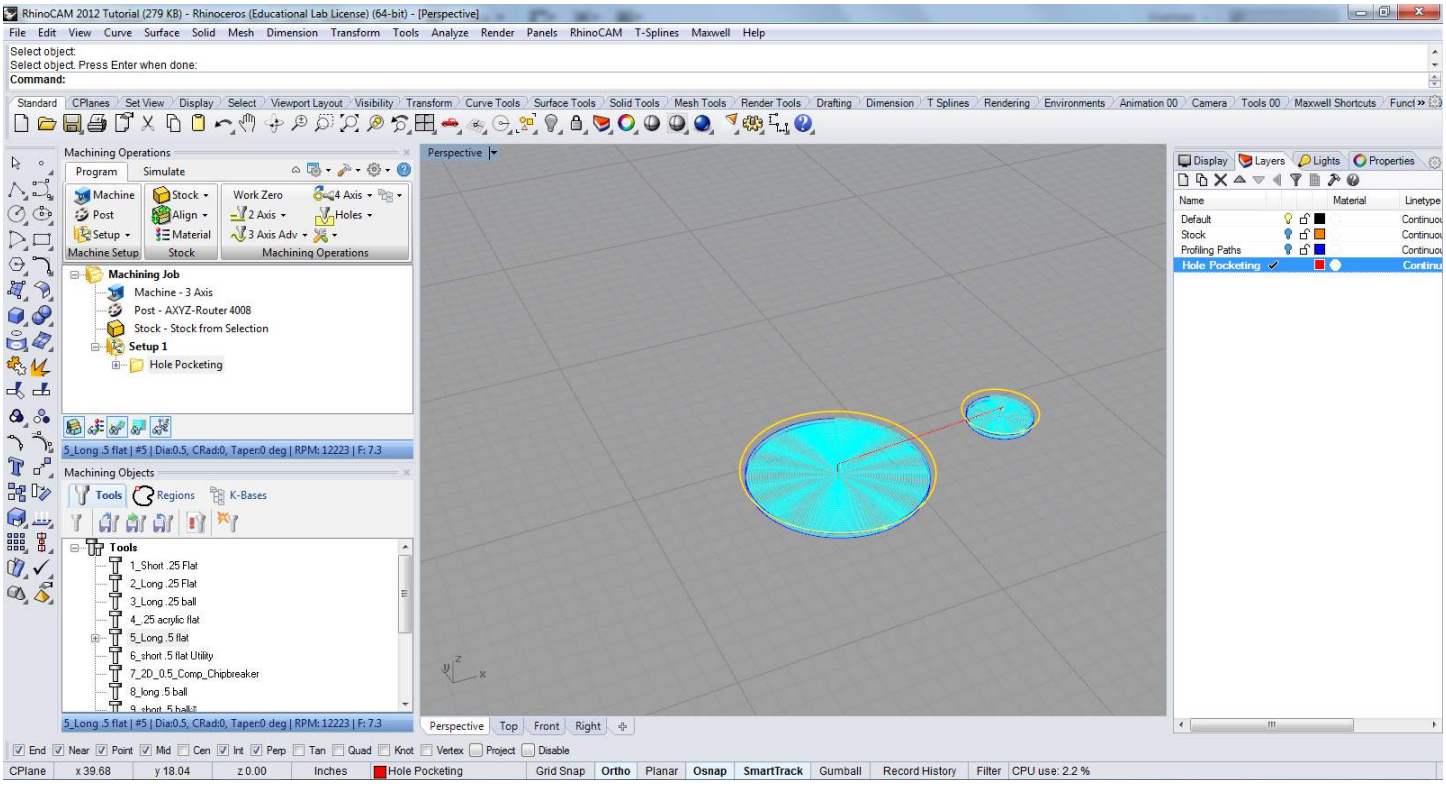




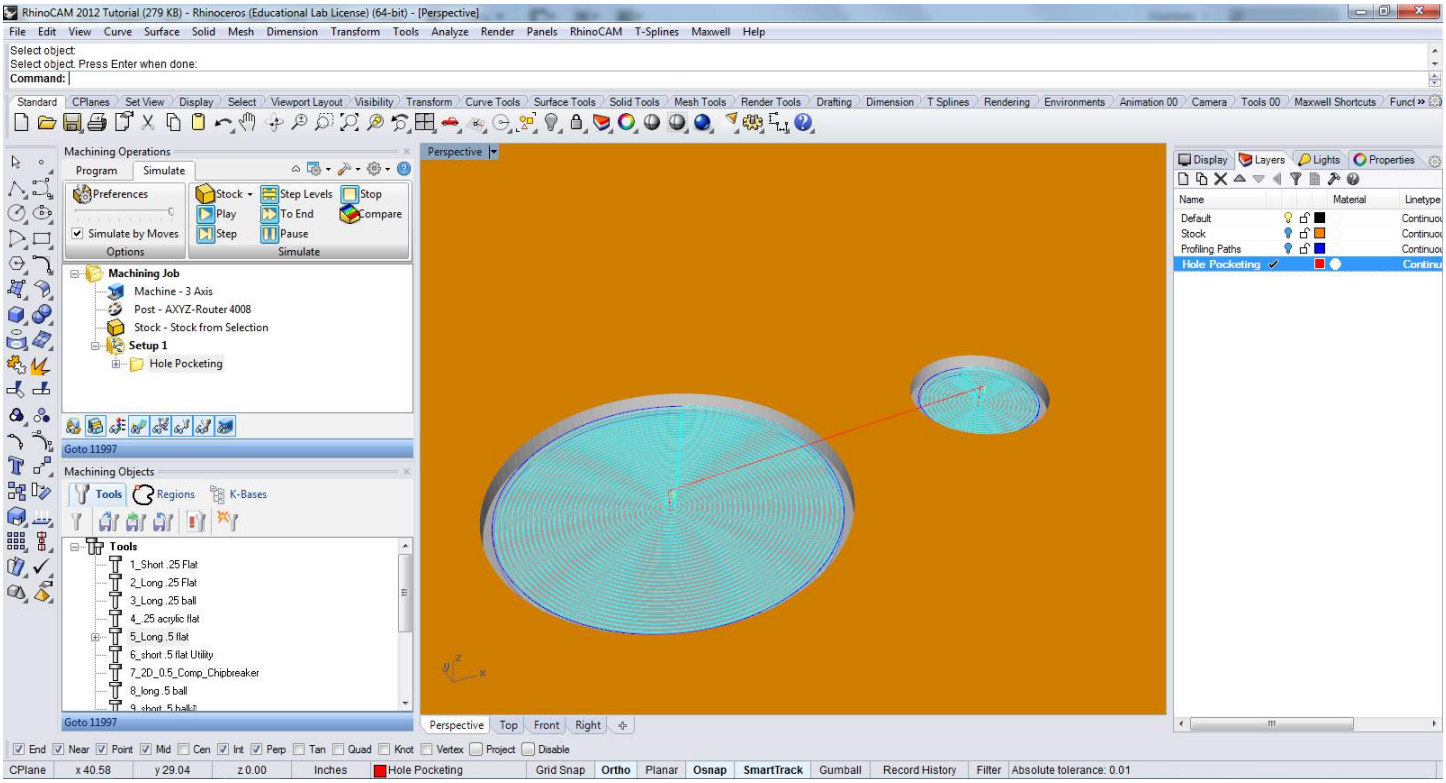
Select hole pocketing from the 2 Axis drop down menu, select the two circles to pocket. Make sure the geometry is closed, if not RhinoCAM will give errors in trying to compute the tooling paths.



Once the regions have been set, set the chosen endmill and the feeds and speeds required for the material to be milled. Under the Cut Parameters chose the location of the geometry, next set the hole depth and check "Use Arc Diameter as Hole Diameter for all selected arcs". Chose the stepdown distance and stepover distance based on the endmill chosen and the material being milled.

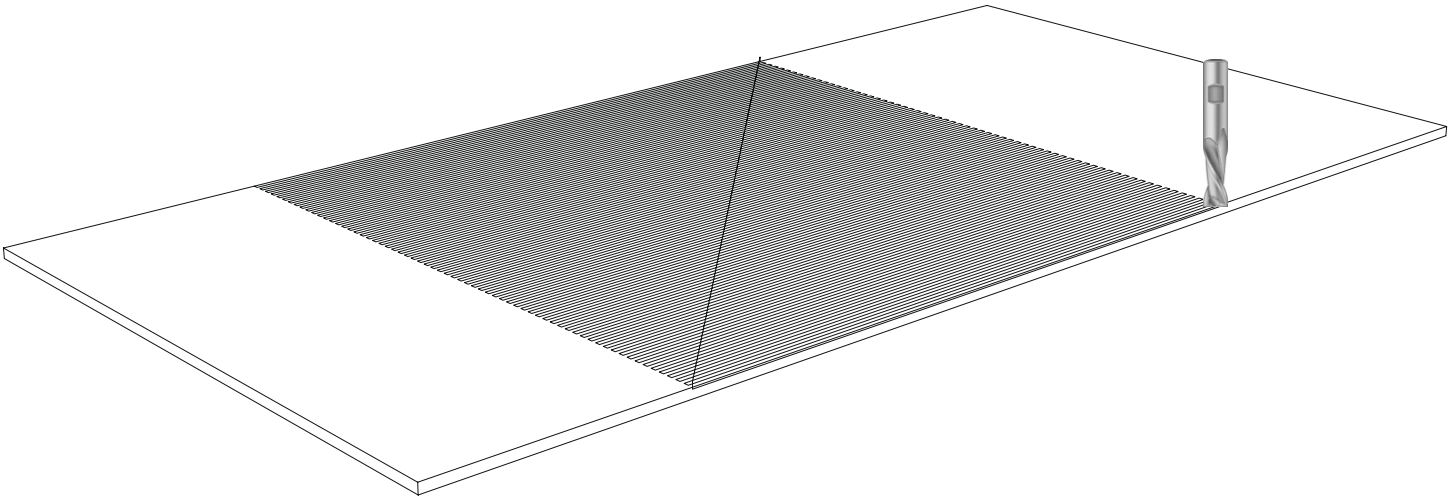


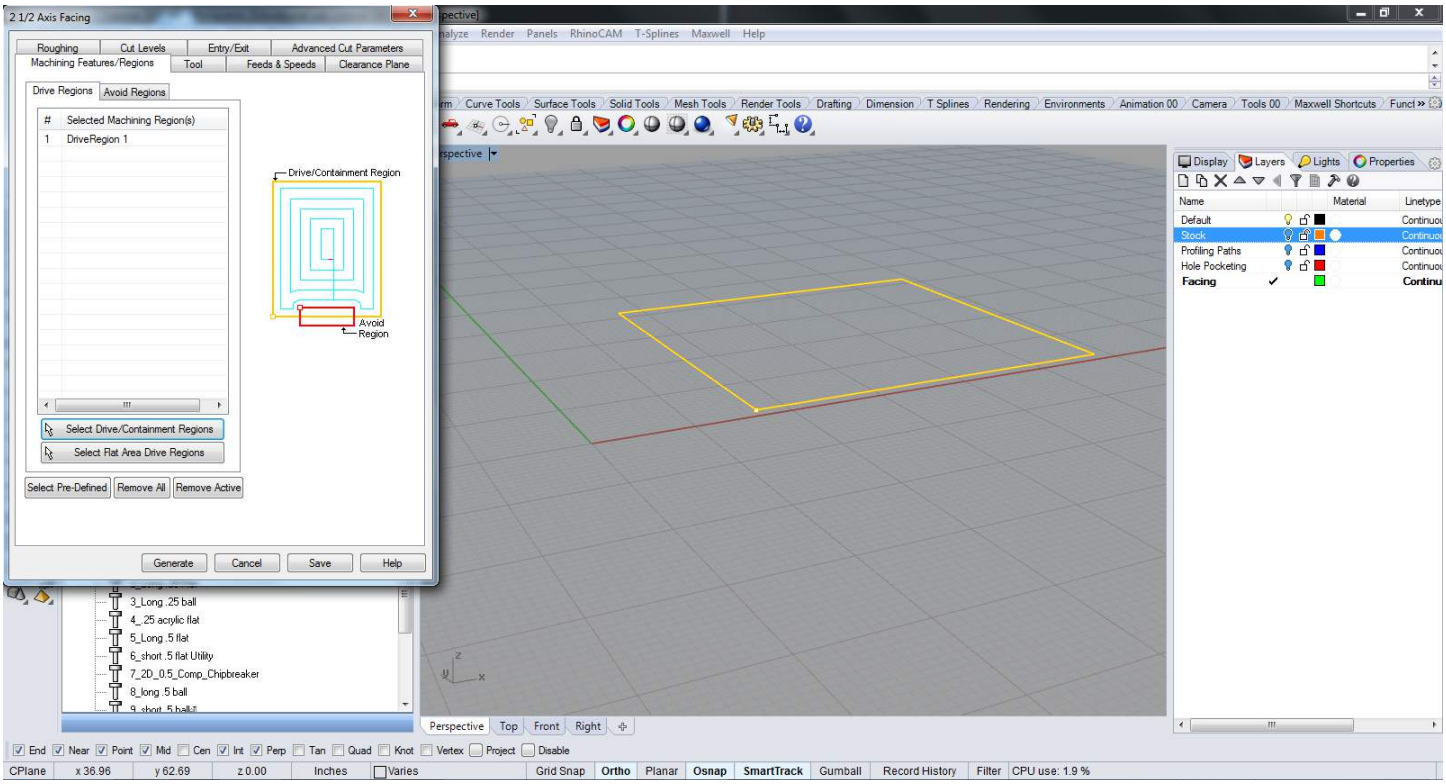
Once the tooling paths have been generated the circles will be populated with spiral offset milling paths.



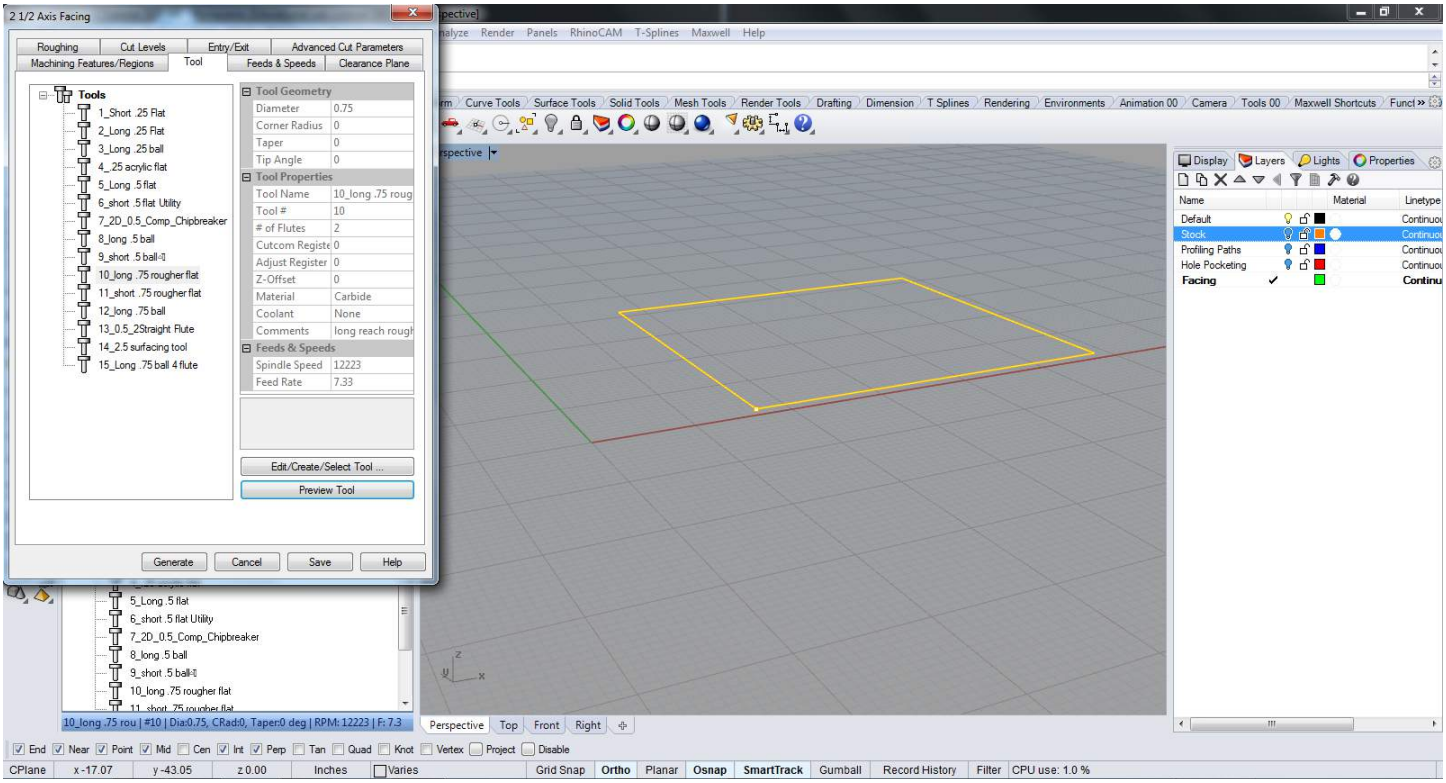
Run the simulation to see the milling paths in process, always run the simulation to make sure there are no errors or collisions with the collet or head of the CNC.

The facing operation is used for large areas to plane down the material in a setup stepdown and stepover assigned in the cutting parameters. The facing operation is useful for planing materials to become planar and for removing larger amounts of materials in pre-set areas.

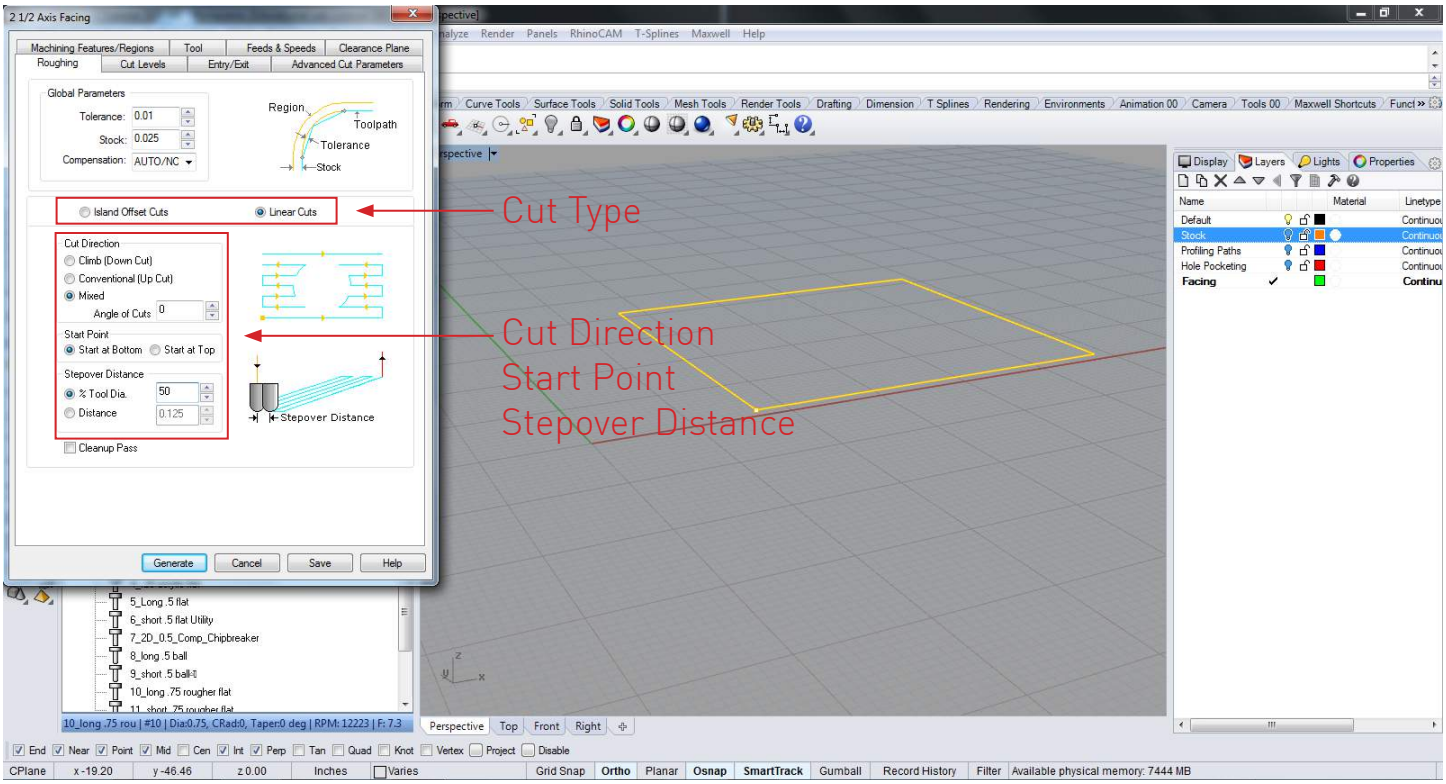




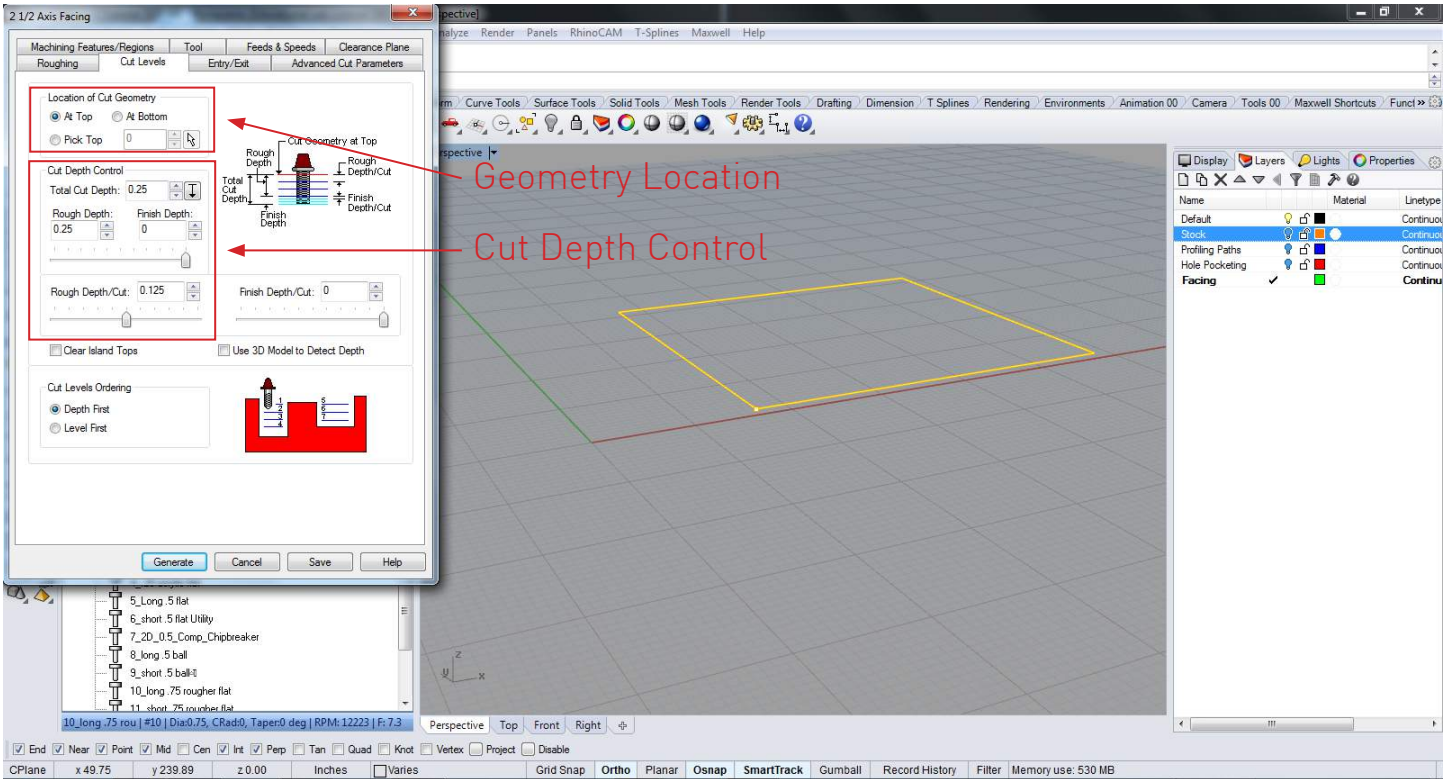
Select the 2 1/2 Axis Facing from the 2Axis drop down tab, select the region that is to be faced on the set up stock.



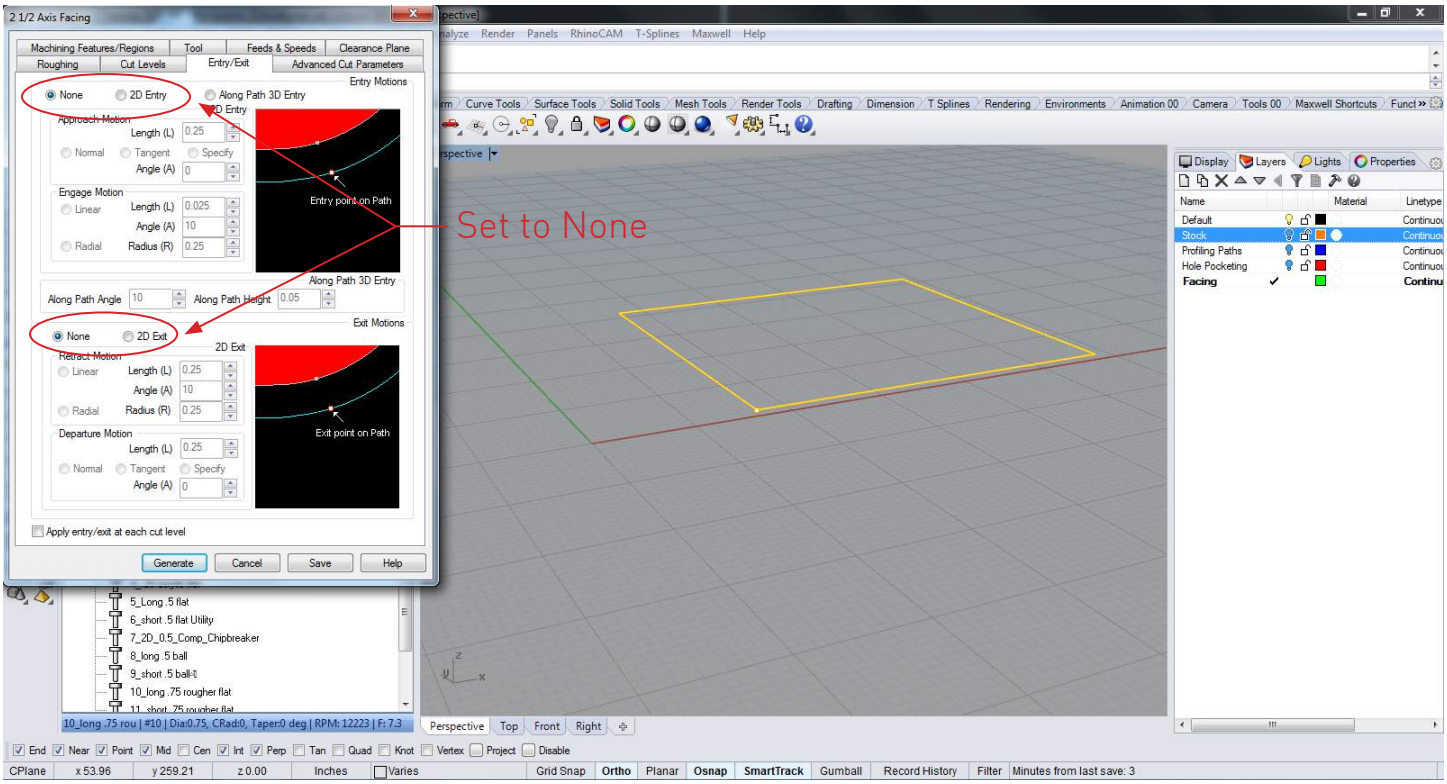
Select the endmill depending on the amount of material to be removed, for a larger area with geometry that is simple use a large endmill for faster facing. If the area is a smaller geometry select a smaller endmill to fit accordingly.



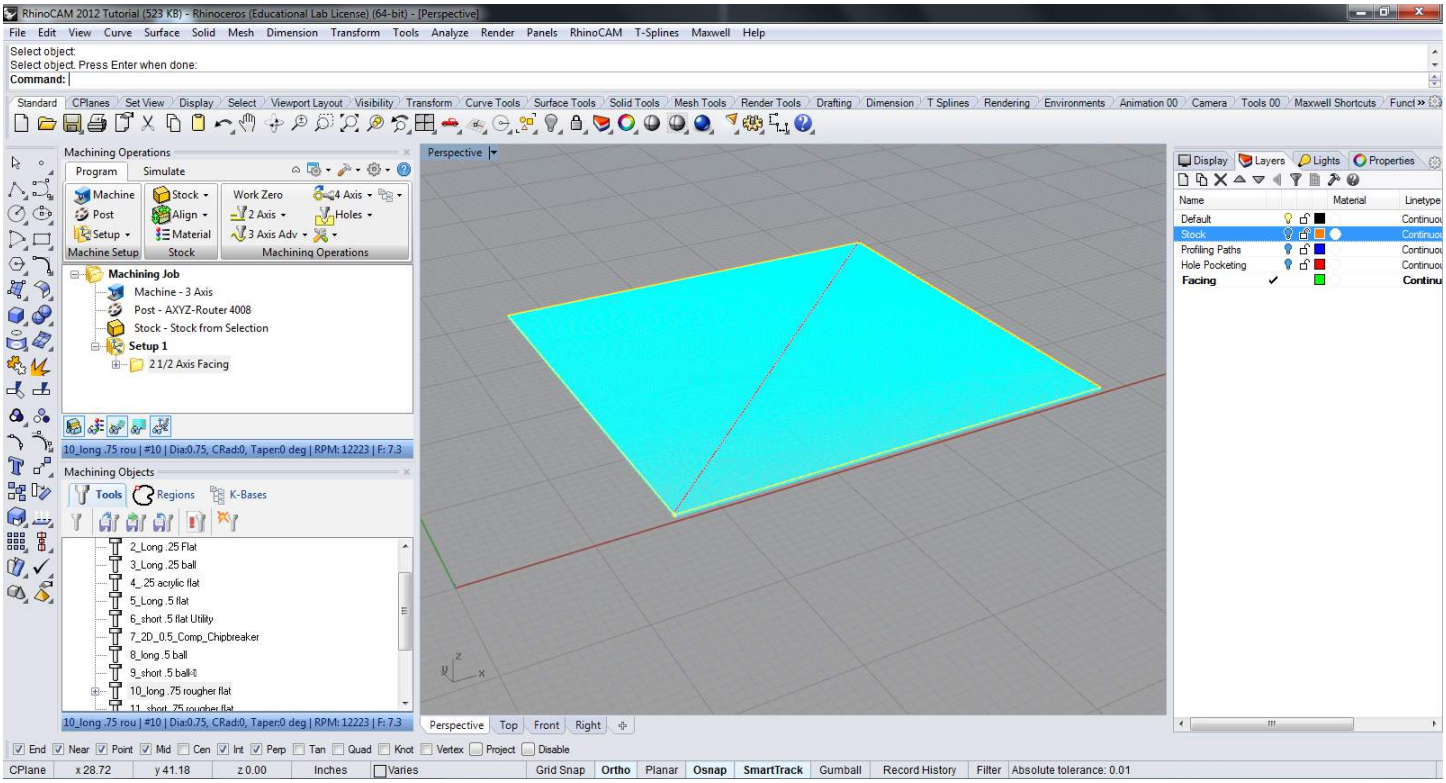
Under the Roughing tab set the type of cuts to linear, the cut direction can be set if milling hardwoods, next set up the Stepover distance.



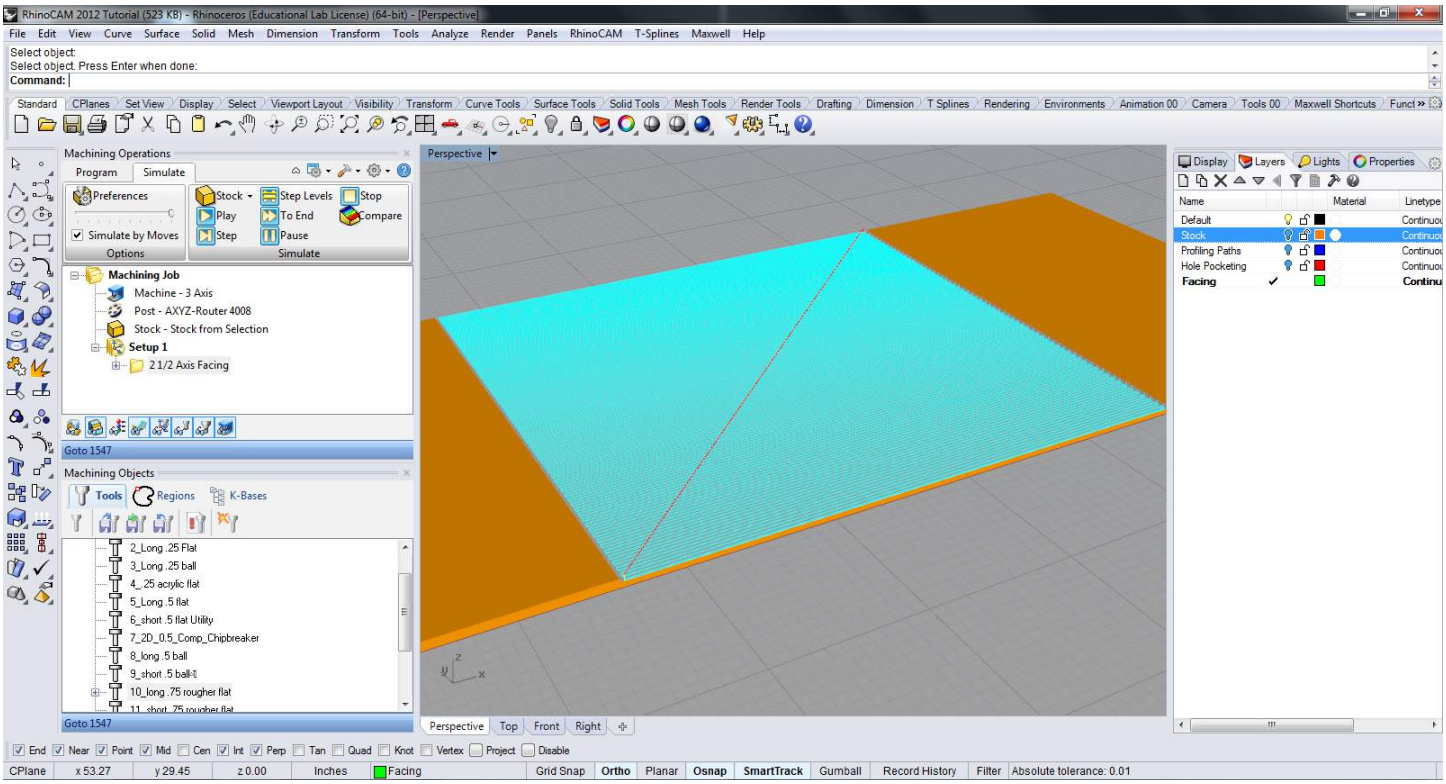
Under the Cut Levels tab set the location of the input geometry, the total cut depth and the number of stepdowns in the rough depth cut, typically never exceed half the diameter of the chosen endmill in hardwoods.



Under the Exit/Entry tab make sure they are set to nine for a straight z axis entry or along 3d path for a sloping z axis entry.



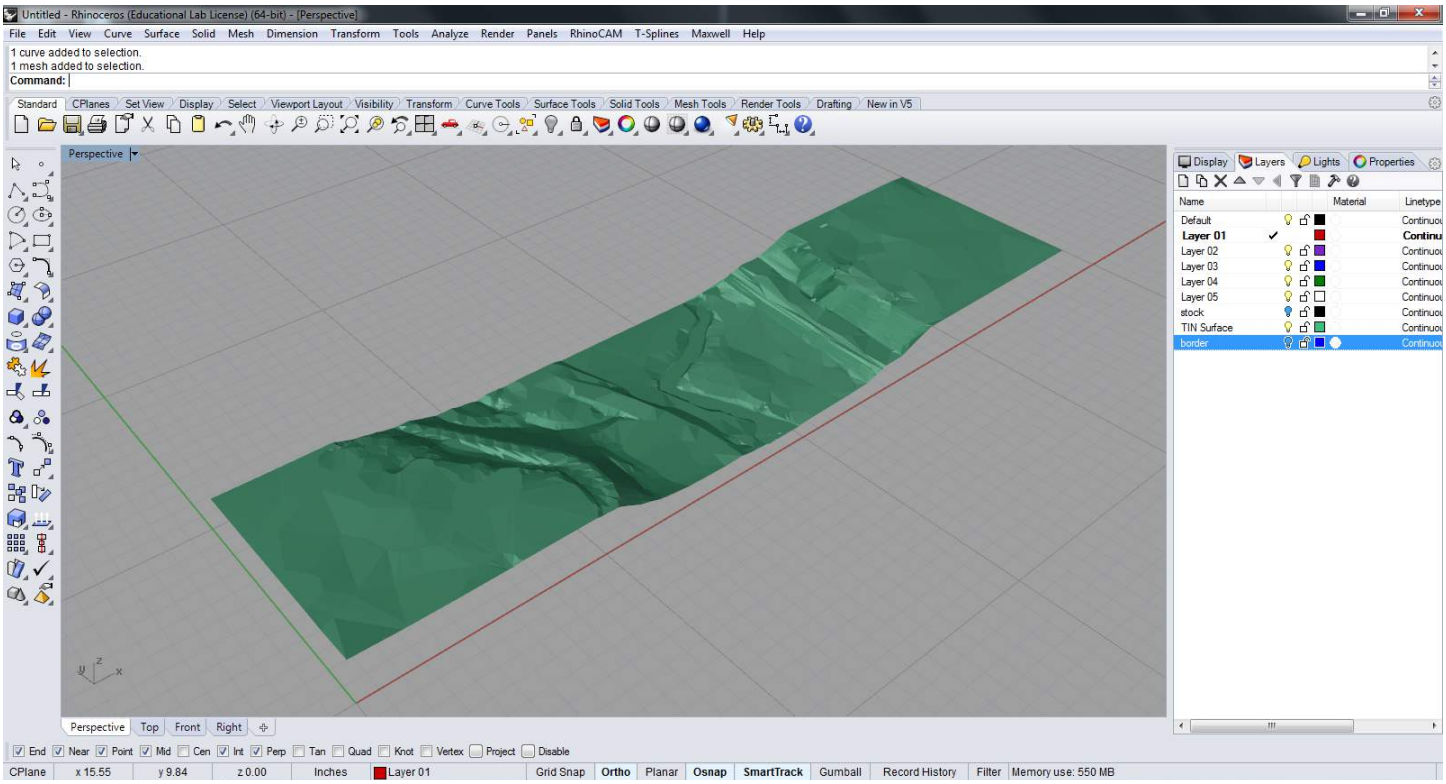
Press generate once the setup is done and the tooling paths will be generated.



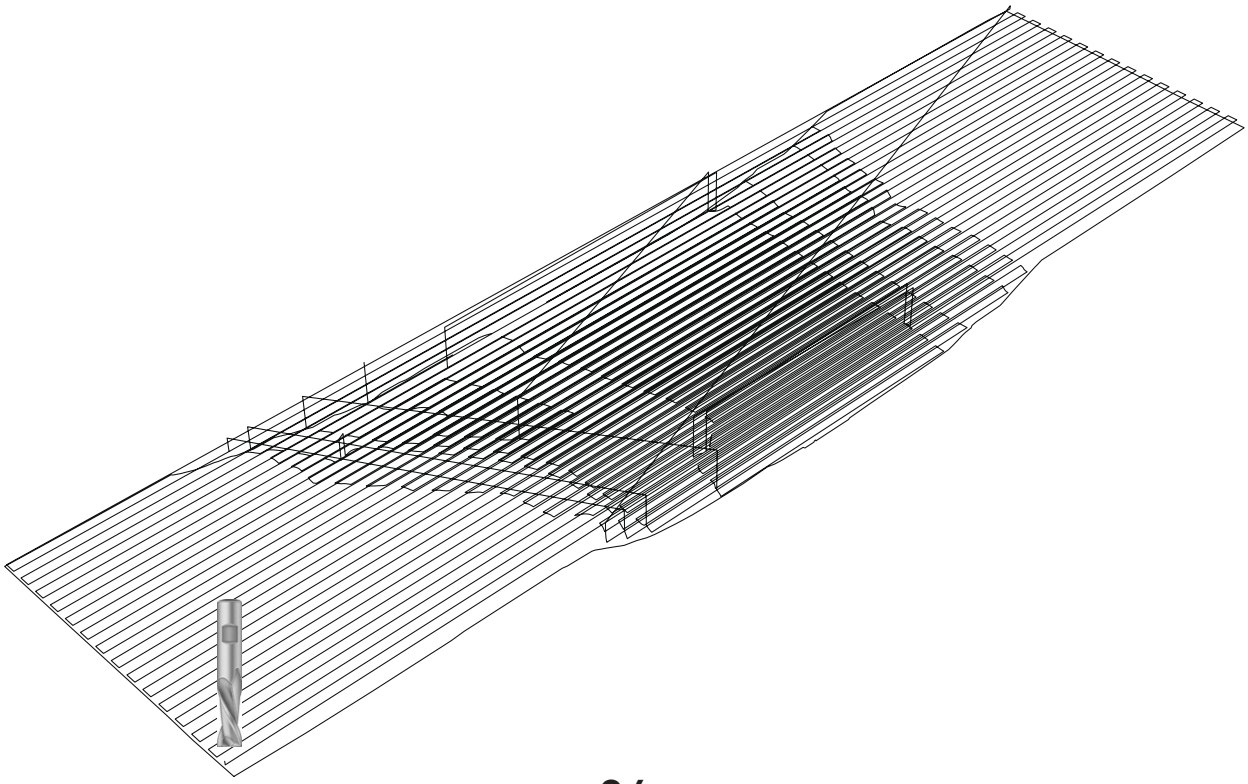
Once the tooling paths have been generated run the simultaion under the simulate tab.

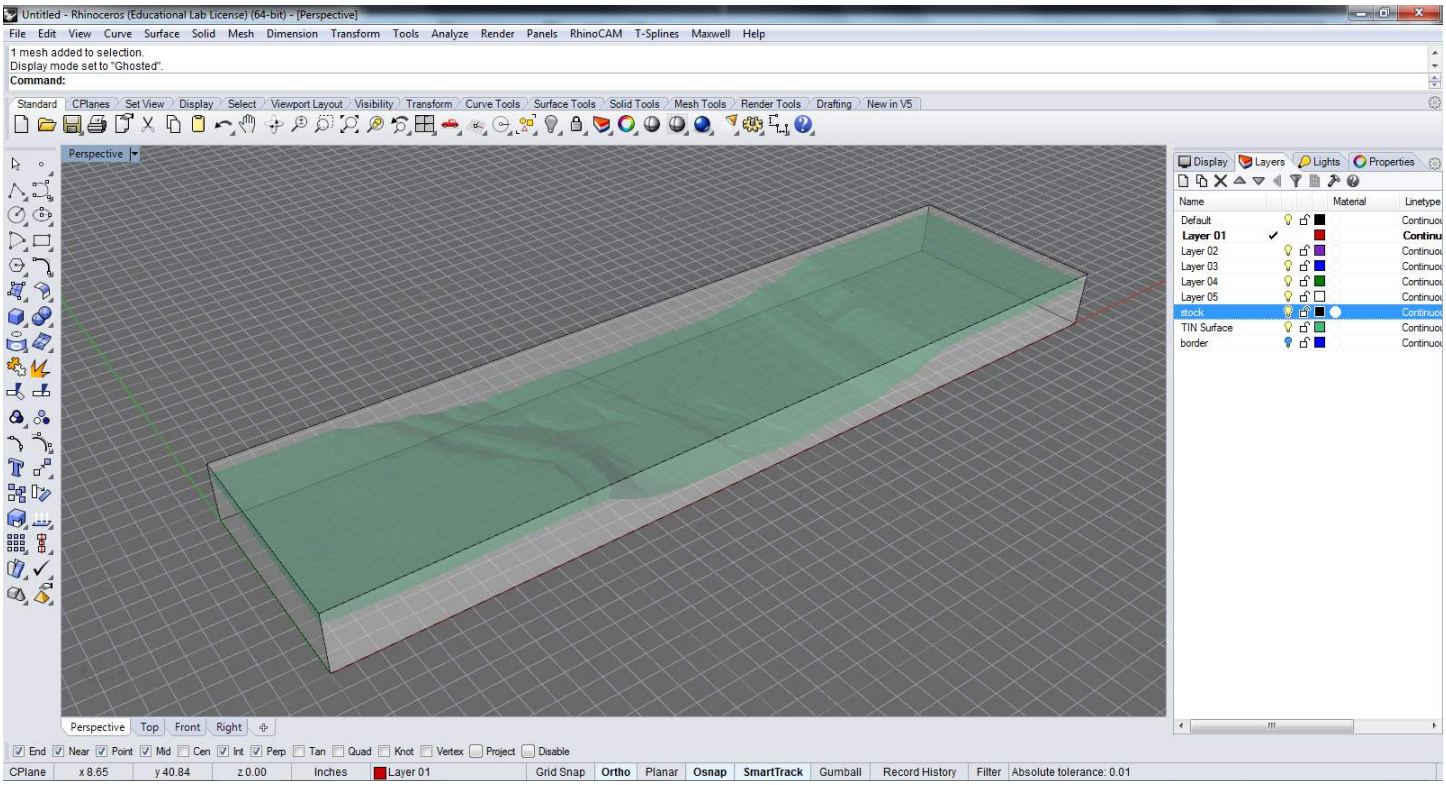
Rhino_CAM 3 Axis Machining Operations_ Horizontal_Roughing

Horizontal Roughing is located in the 3 Axis milling operations, the function is used to define a regions for a surface and milling roughly to the surface. This allows for the rough removal of material before a finishing pass is run to create the final surface.

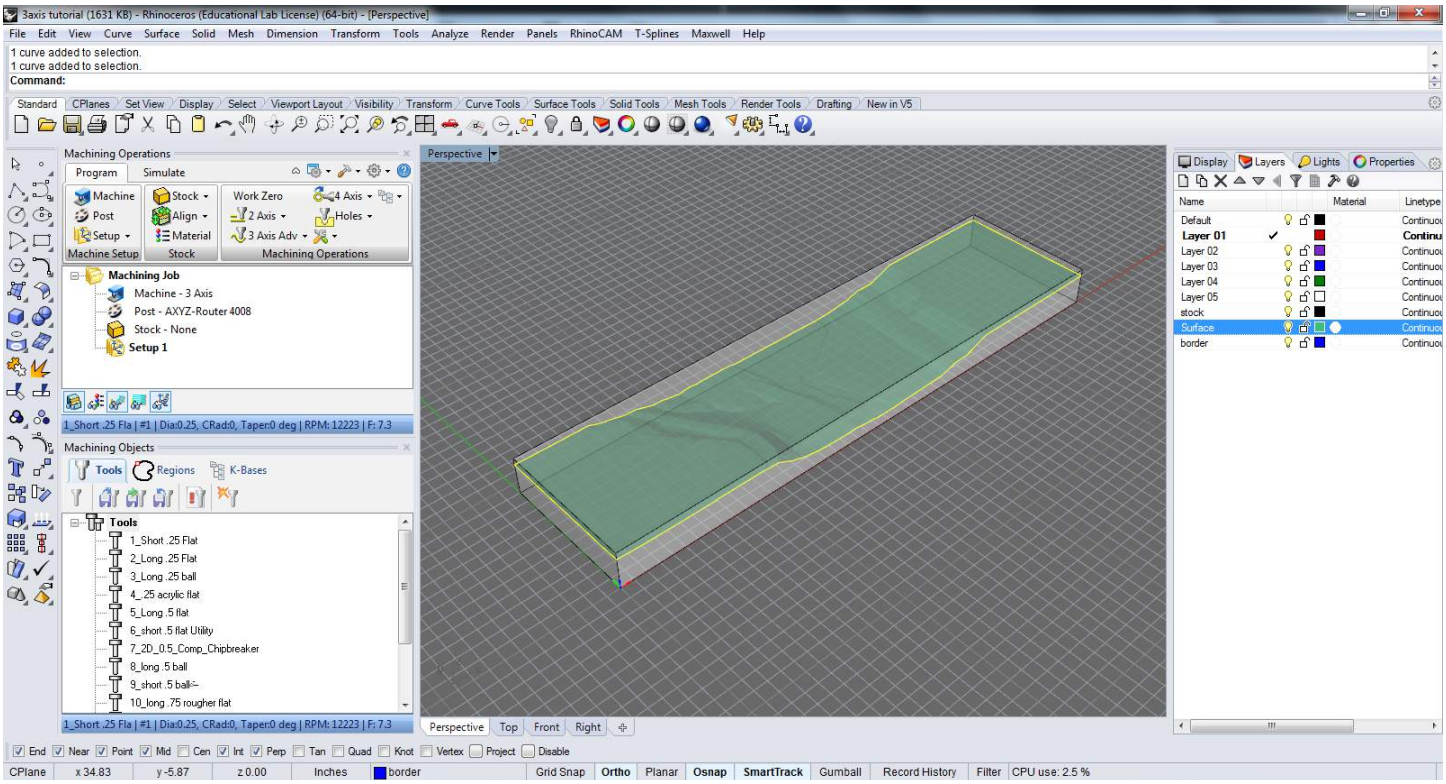


In the sample above a terrain file has been imported into Rhino, the mesh was generated from contour data. The surface will be placed in a piece of stock that will match the physical stock to be milled. Both meshes and nurbs surfaces can be used for 3d milling.

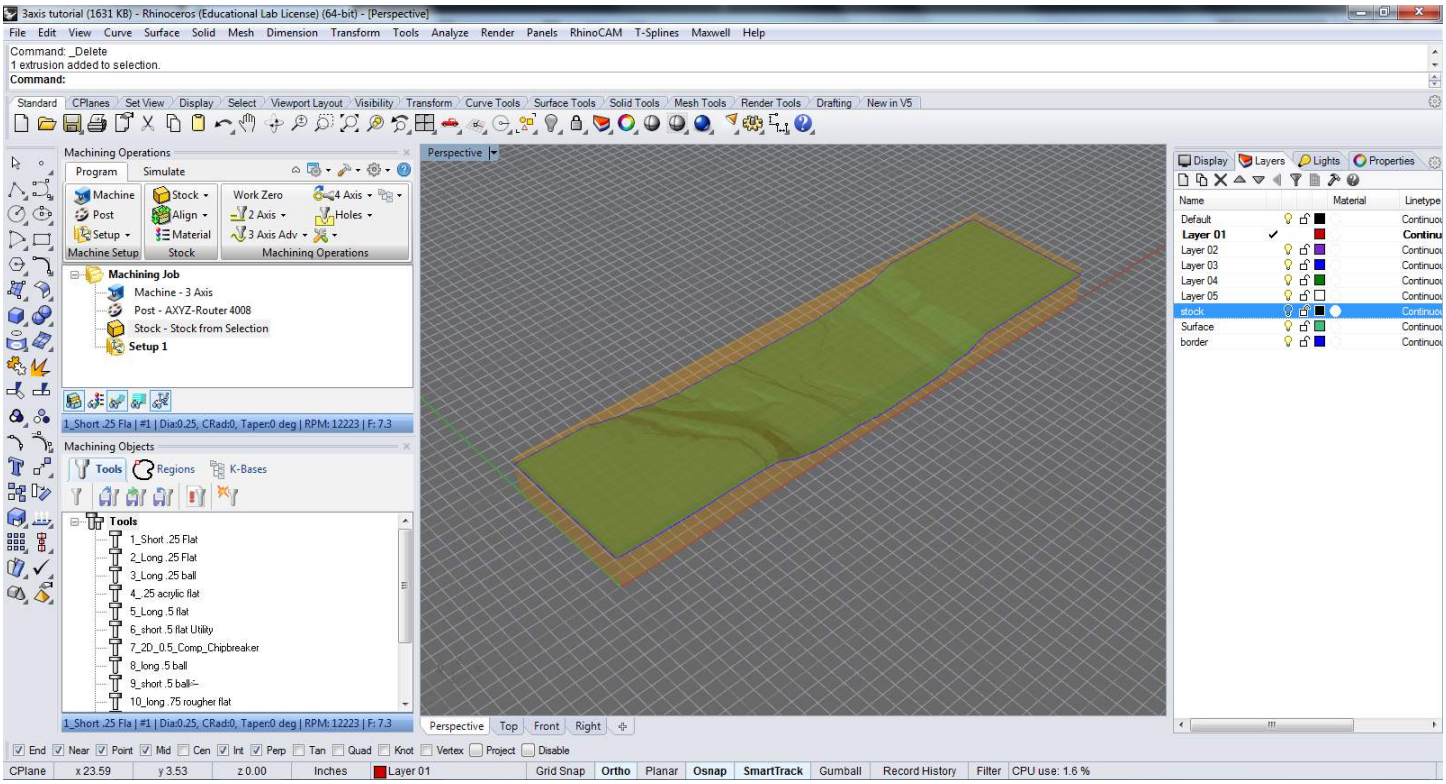




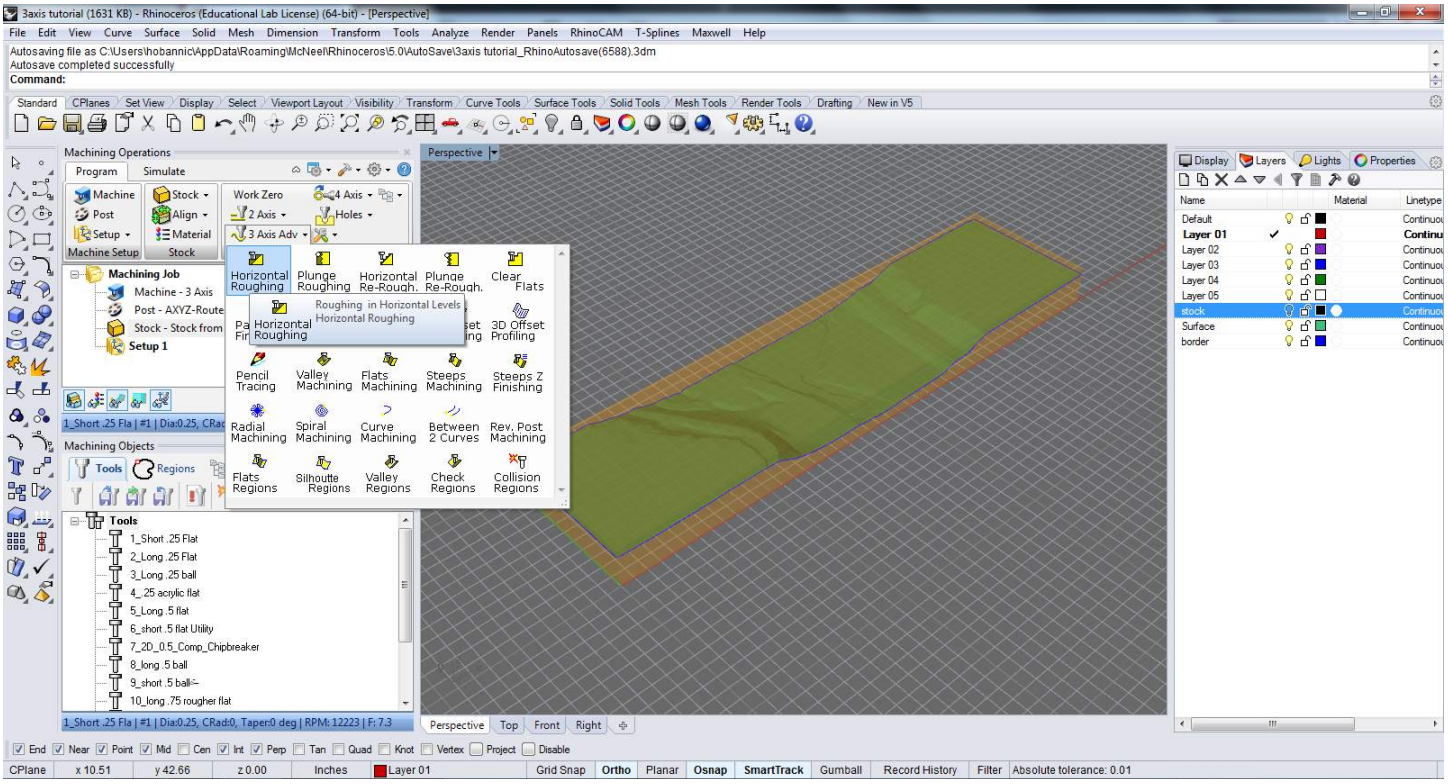
Create a stock to match the physical milling stock, place the terrain within the stock, try to allow as little as possible distance between the top of the stock and the terrain. This will allow for reduced milling time. The deep the terrain is inset into the stock the more material there is to remove for the finished topography.



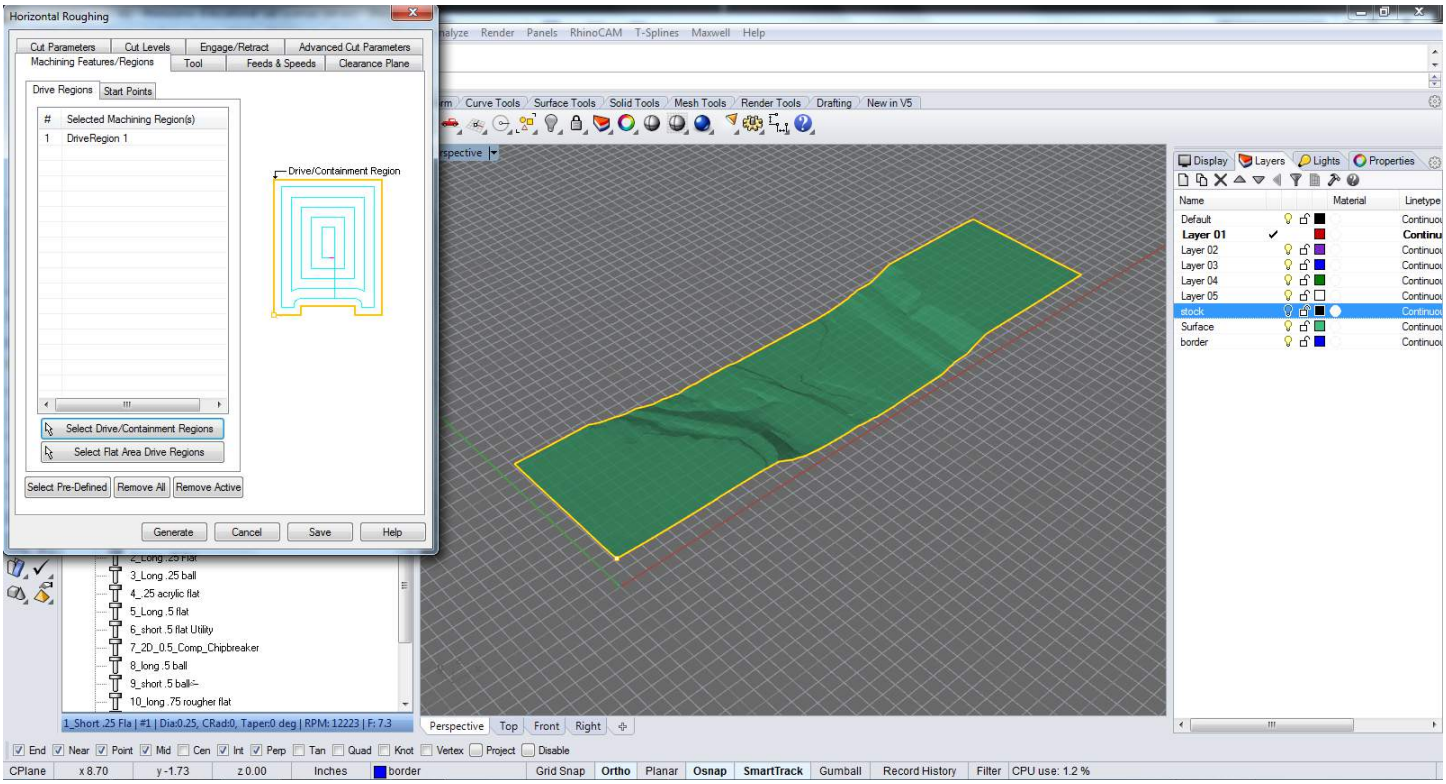
Create a bounding region around the terrain to be milled, this can be a flat region or following the terrain edge. This region will assign the surface to be milled.



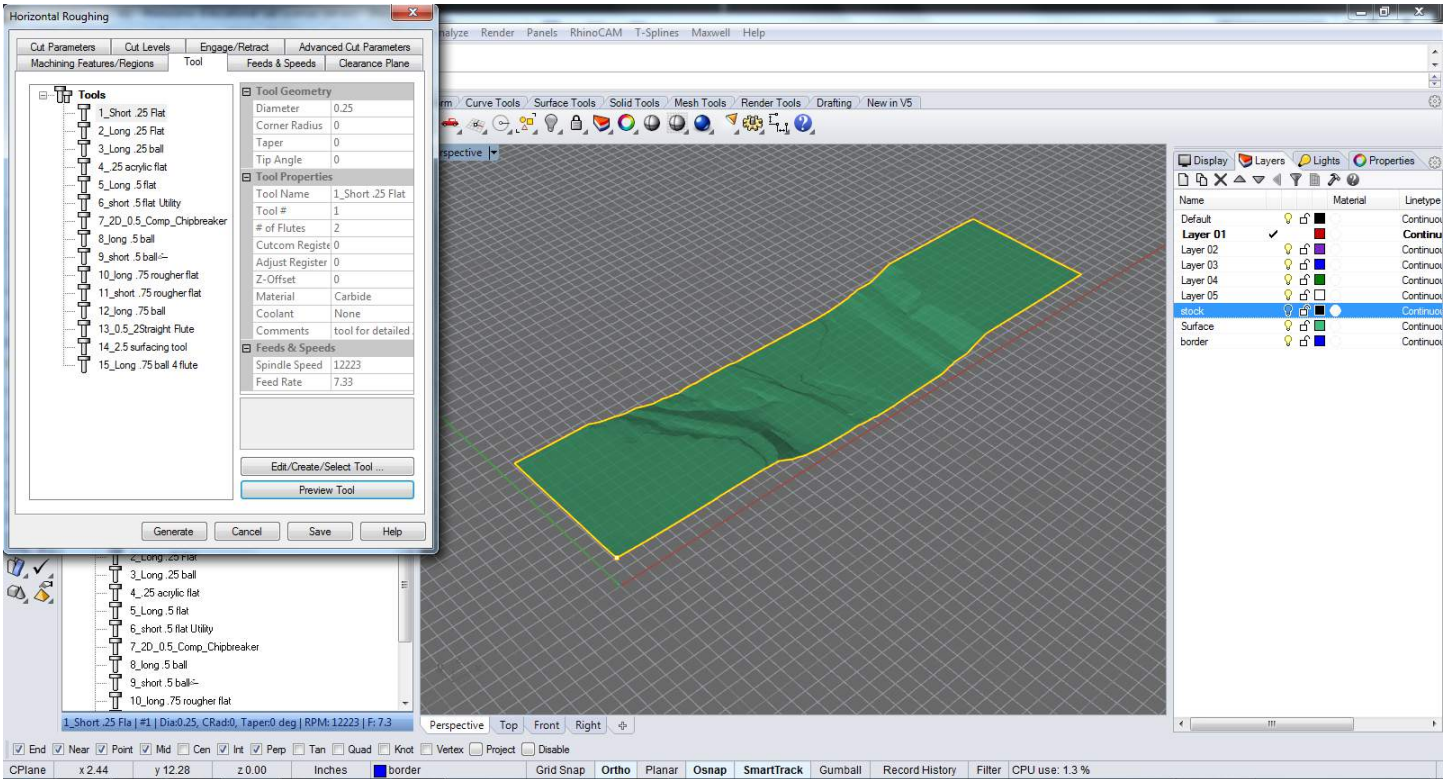
Highlight the stock and from the program tab, choose stock and then stock from selection. This will create the RhinoCAM stock to work within.



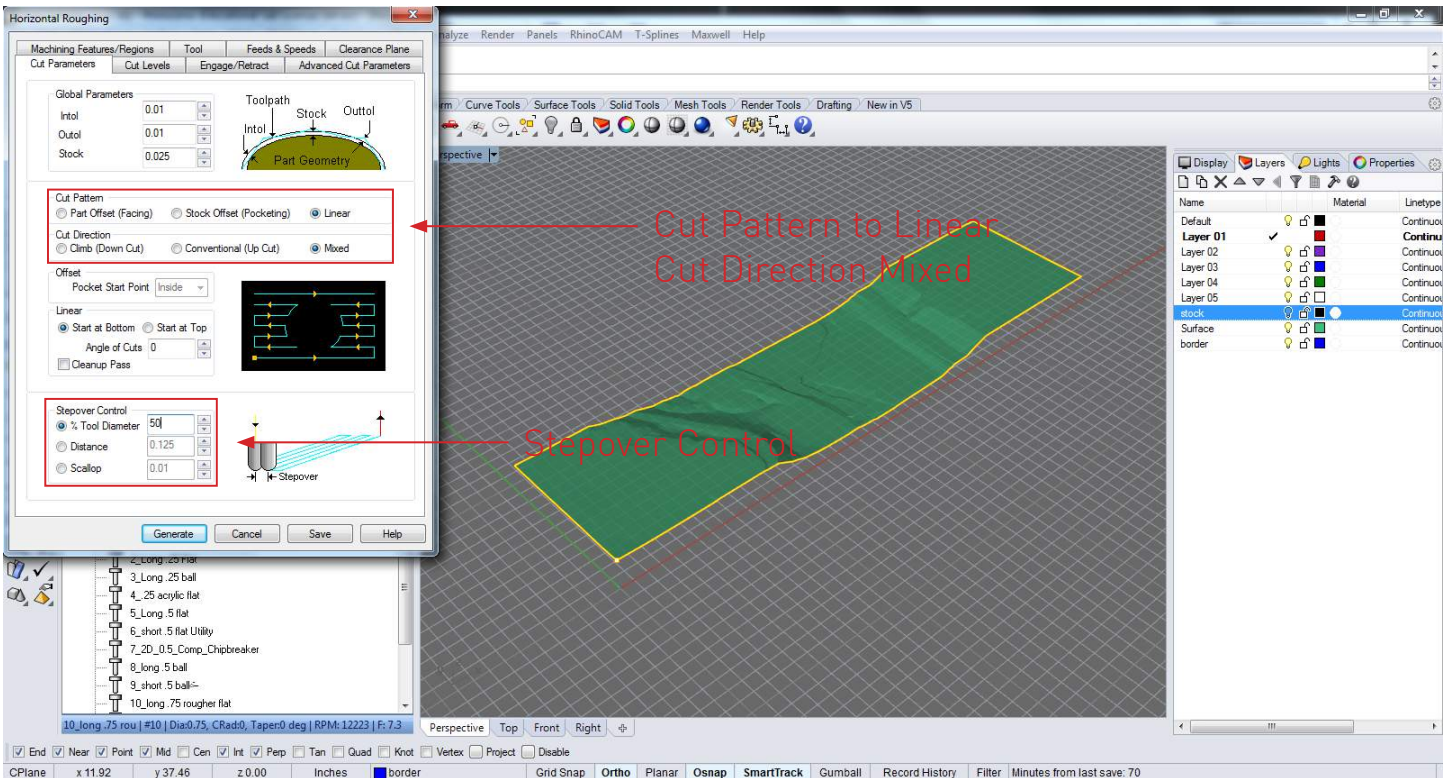
Under the 3 Axis Adv tab, select Horizontal roughing.



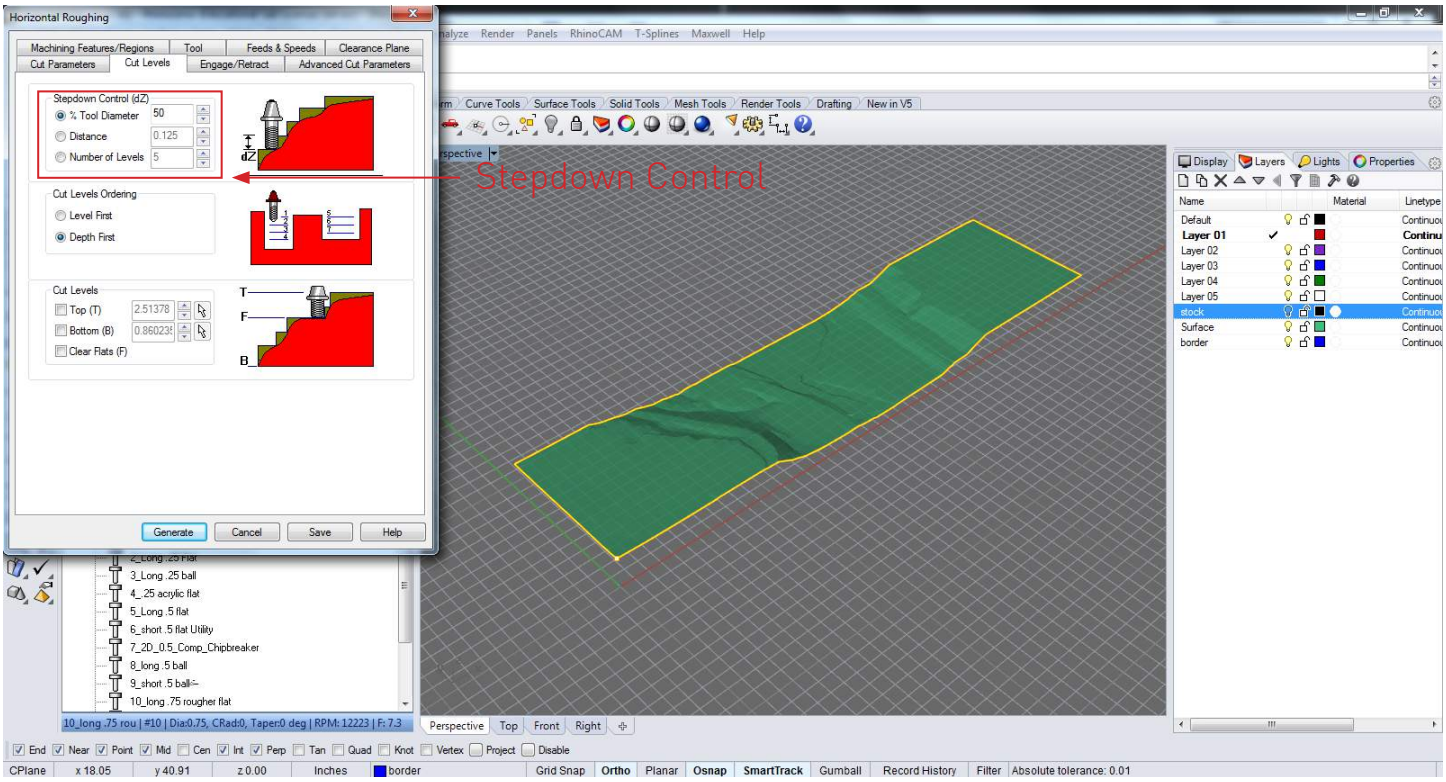
Select the created bounding region to define the area to be milled. Use Select/Drive Containment regions as in the 2 axis milling operations.



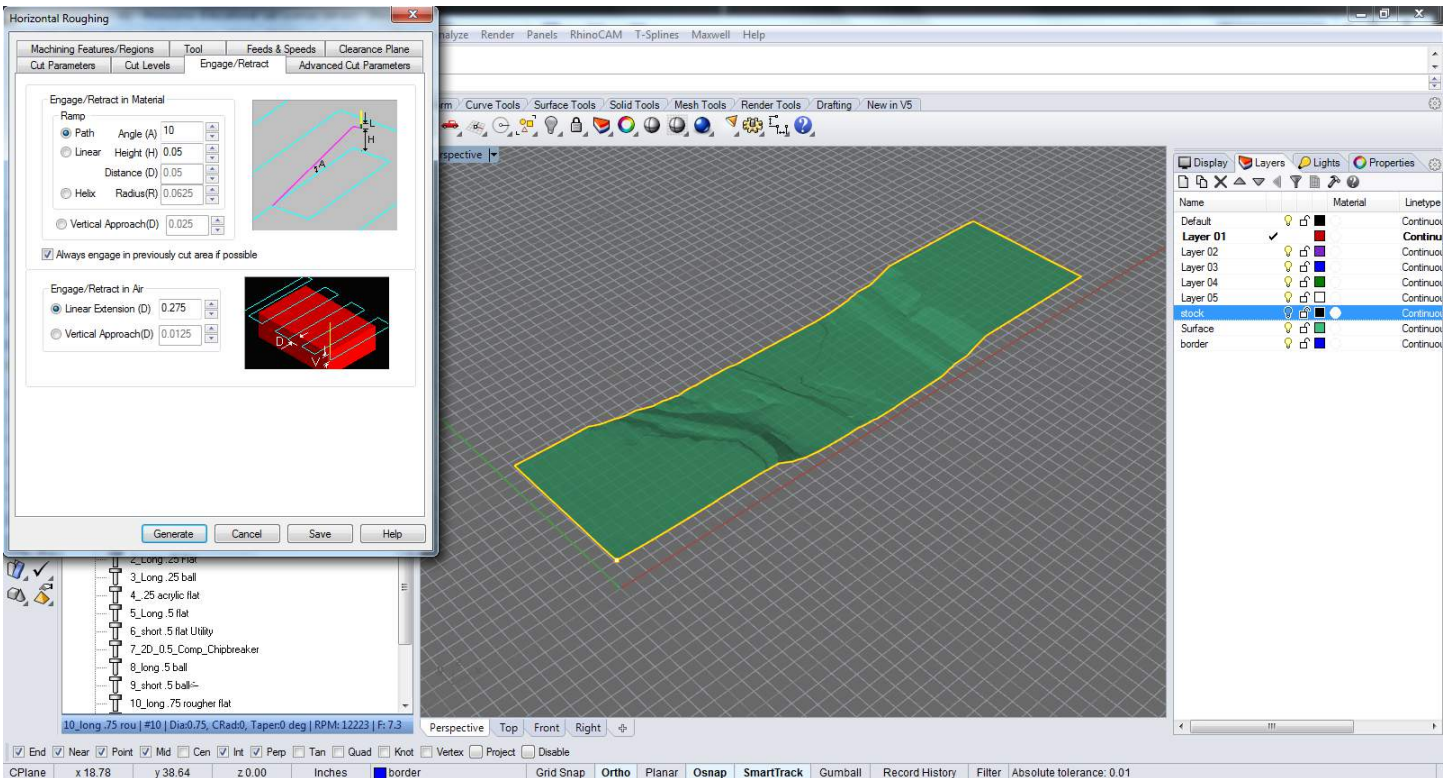
Select the endmill to be used for the roughing operation, typically a larger diameter flat endmill will be used to remove the material to approximate the rough surface.



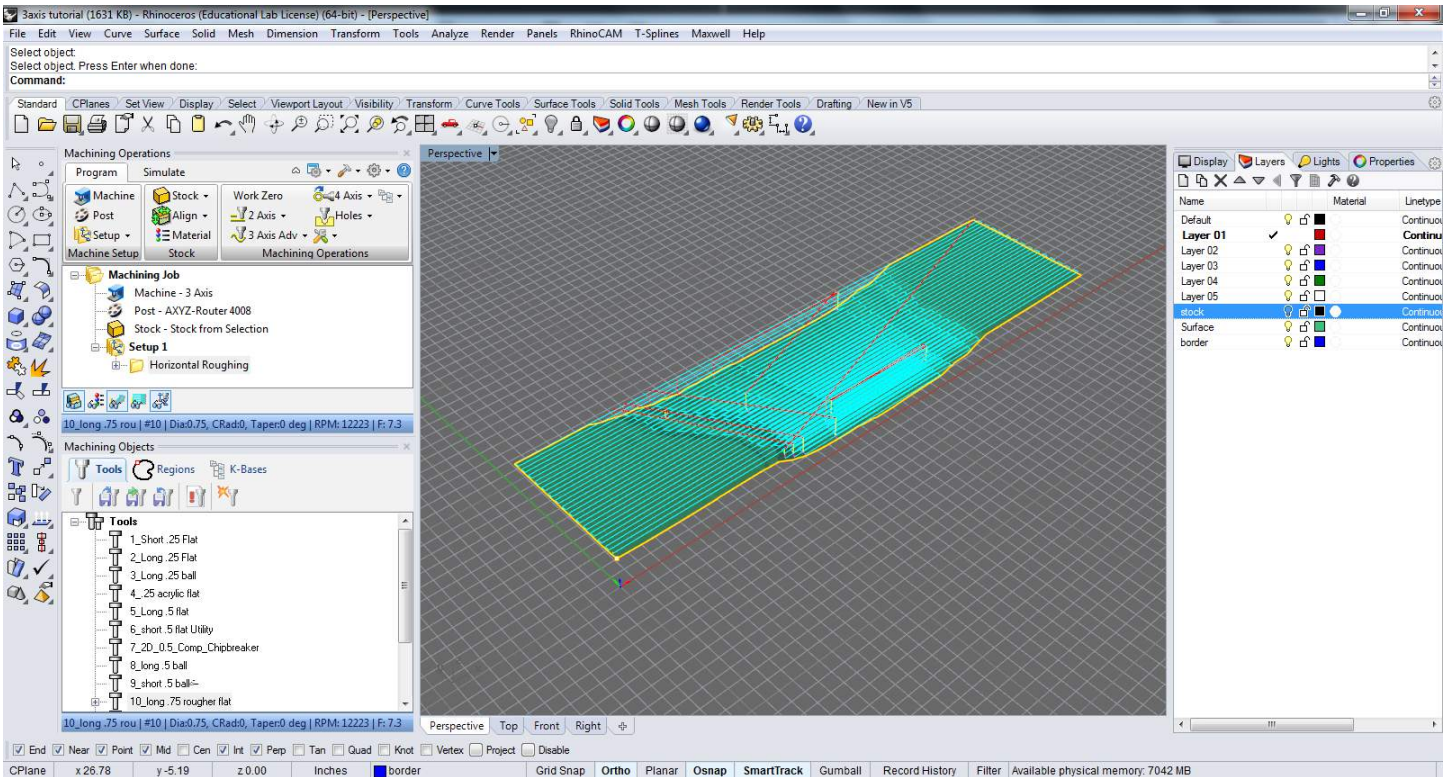
Under Cut Parameters set the cut pattern to linear and the cut direction to mixed for materials that cut direction isn't necessary (ie Foam). For stepover control set to half the endmill diameter for dense materials and for foam this can be increased to 75-100.



Under Cut Levels the stepdown control can be set, either the distance or the tool diameter can be set to determine the levels for the Horizontal Roughing.

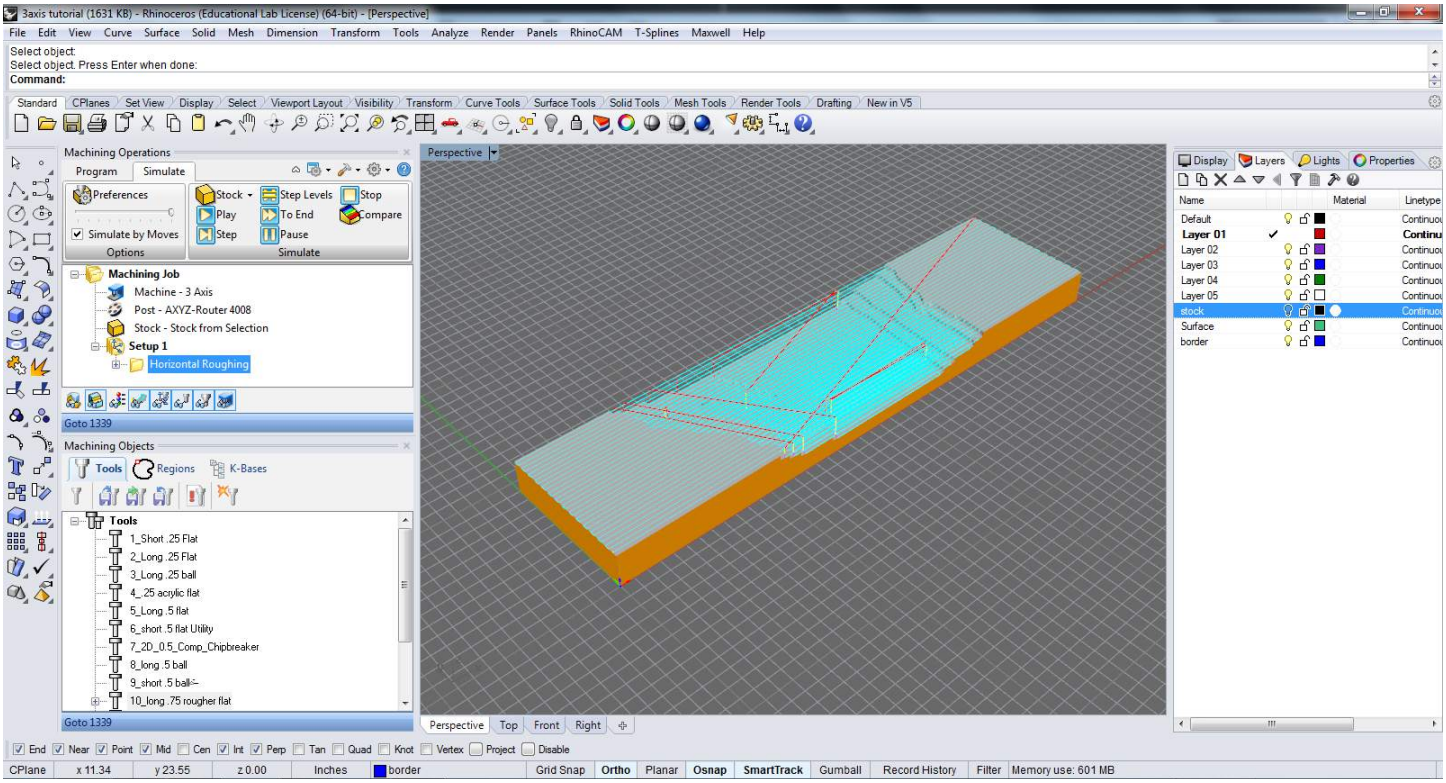


Under the engage/retract tab set engage/retract in material to path, the endmill will follow a path into the beginning tooling path.

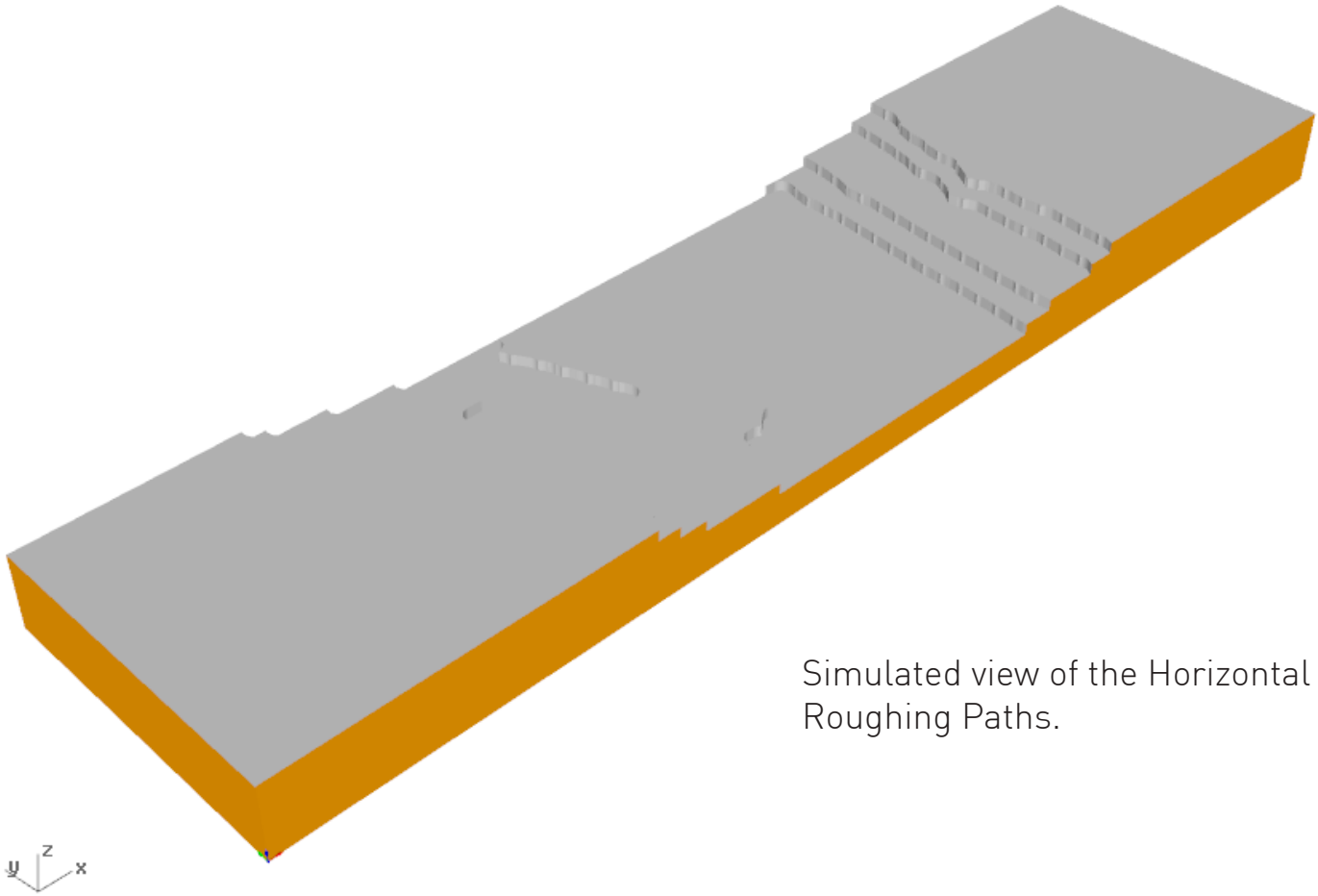


Press generate once all the setup has been completed, the tooling paths with levels will populate the surface.

Rhino_CAM 3 Axis Machining Operations_ Horizontal_Roughing



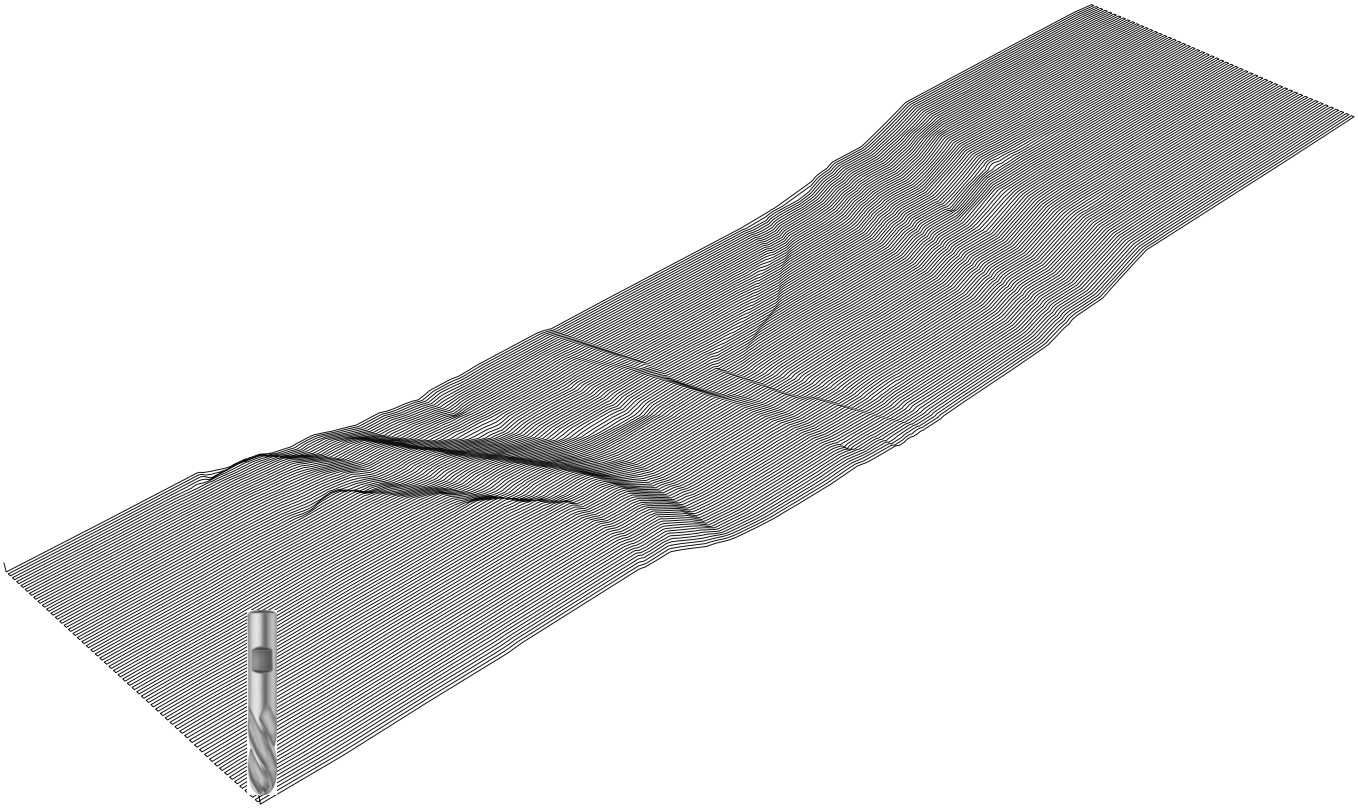
Run the simulation under the simulate tab to make sre there are no collisions or errors in the tooling path setup.

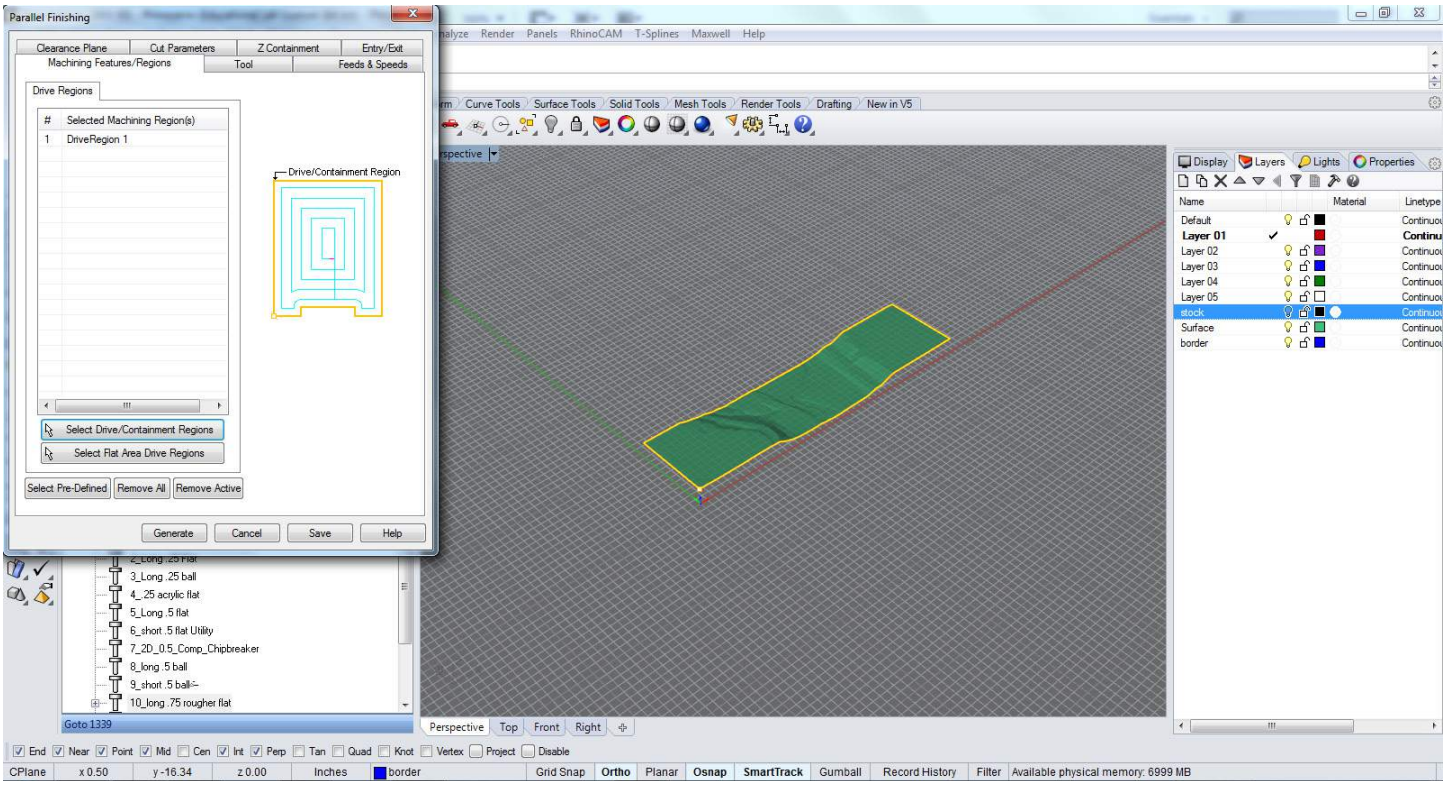


Simulated view of the Horizontal Roughing Paths.

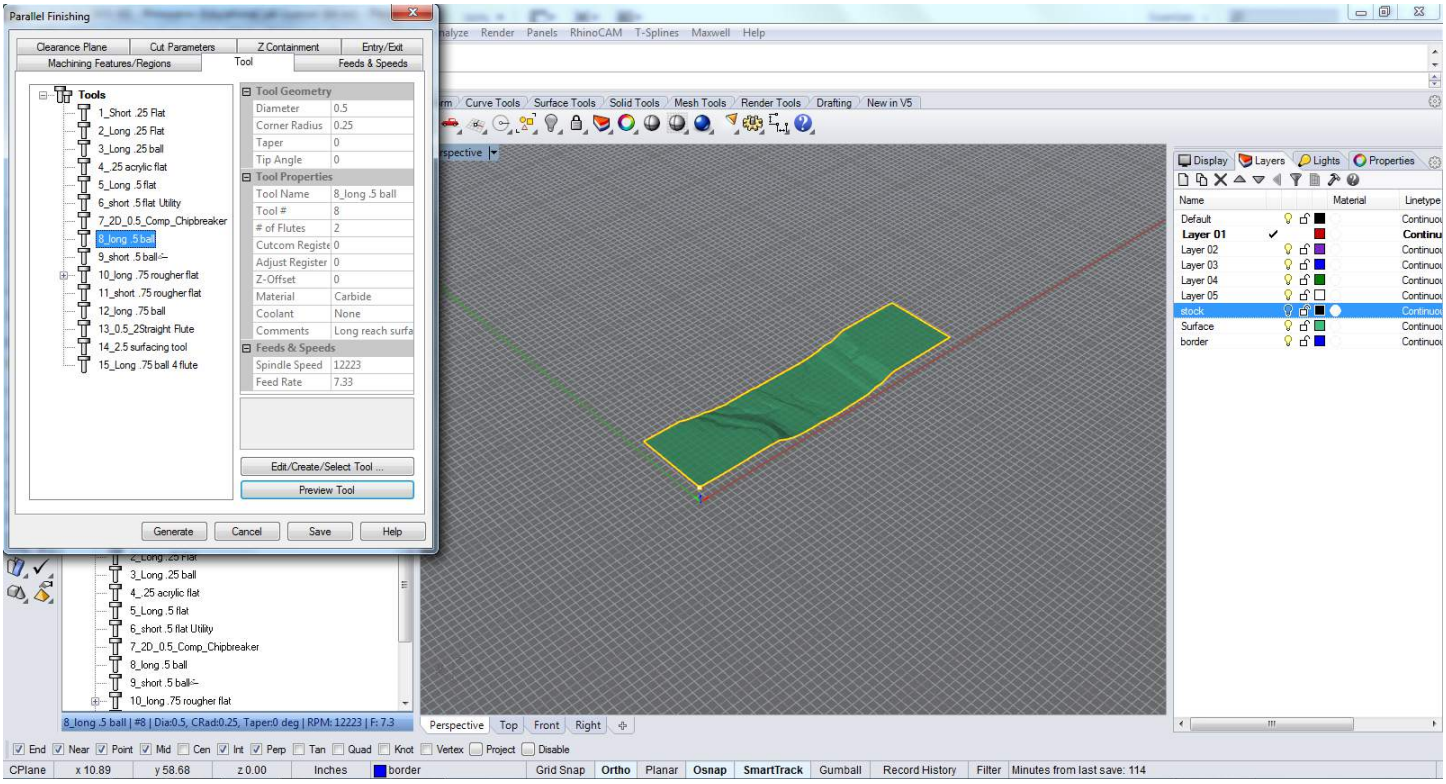
Rhino_CAM 3 Axis Machining Operations_ Parallel_Finishing

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. (ie a ravine in a topography model)



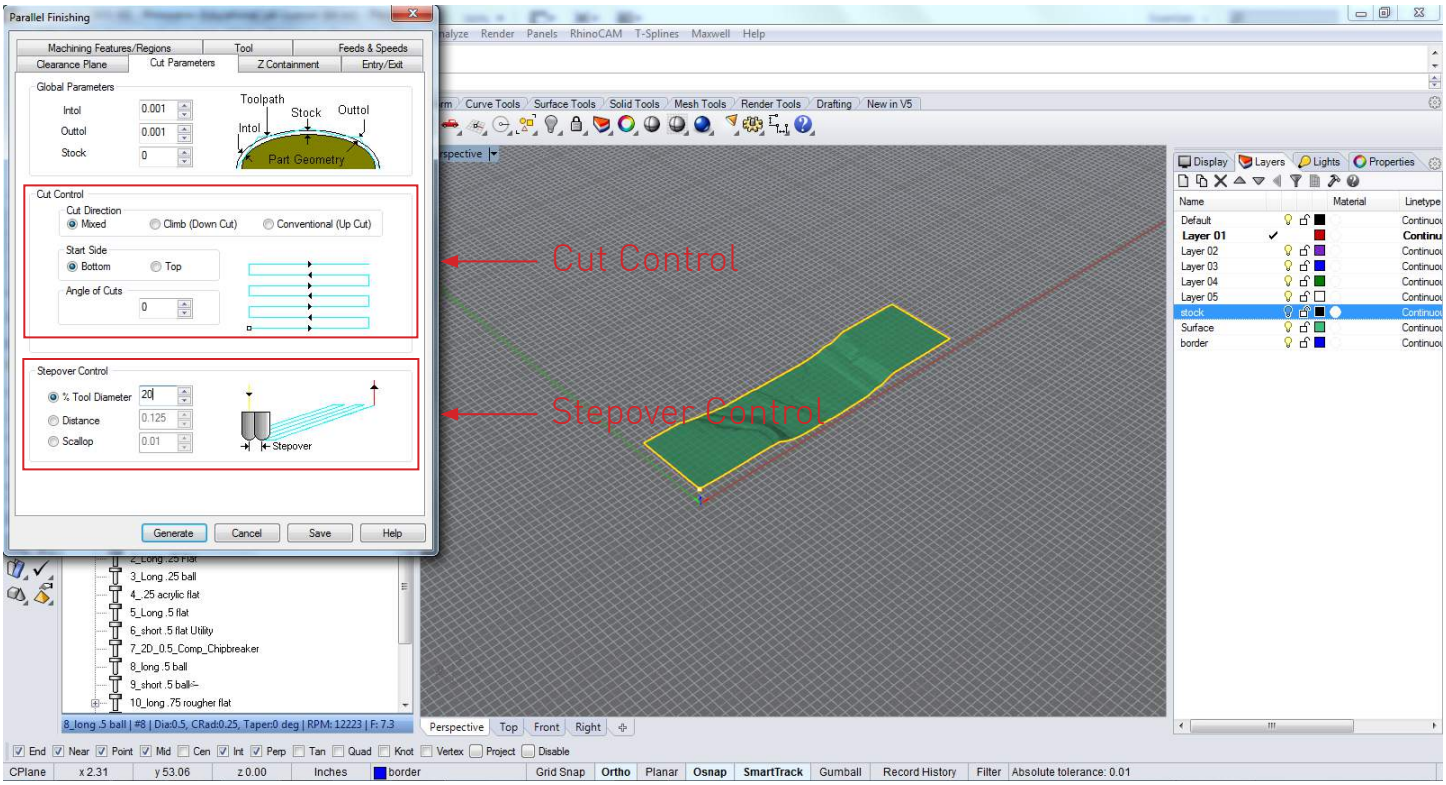


Select Parallel Finishing from the 3 Axis Adv drop down menu, select the drive region to be milled. This will be the same region used in the Horizontal Roughing pass.

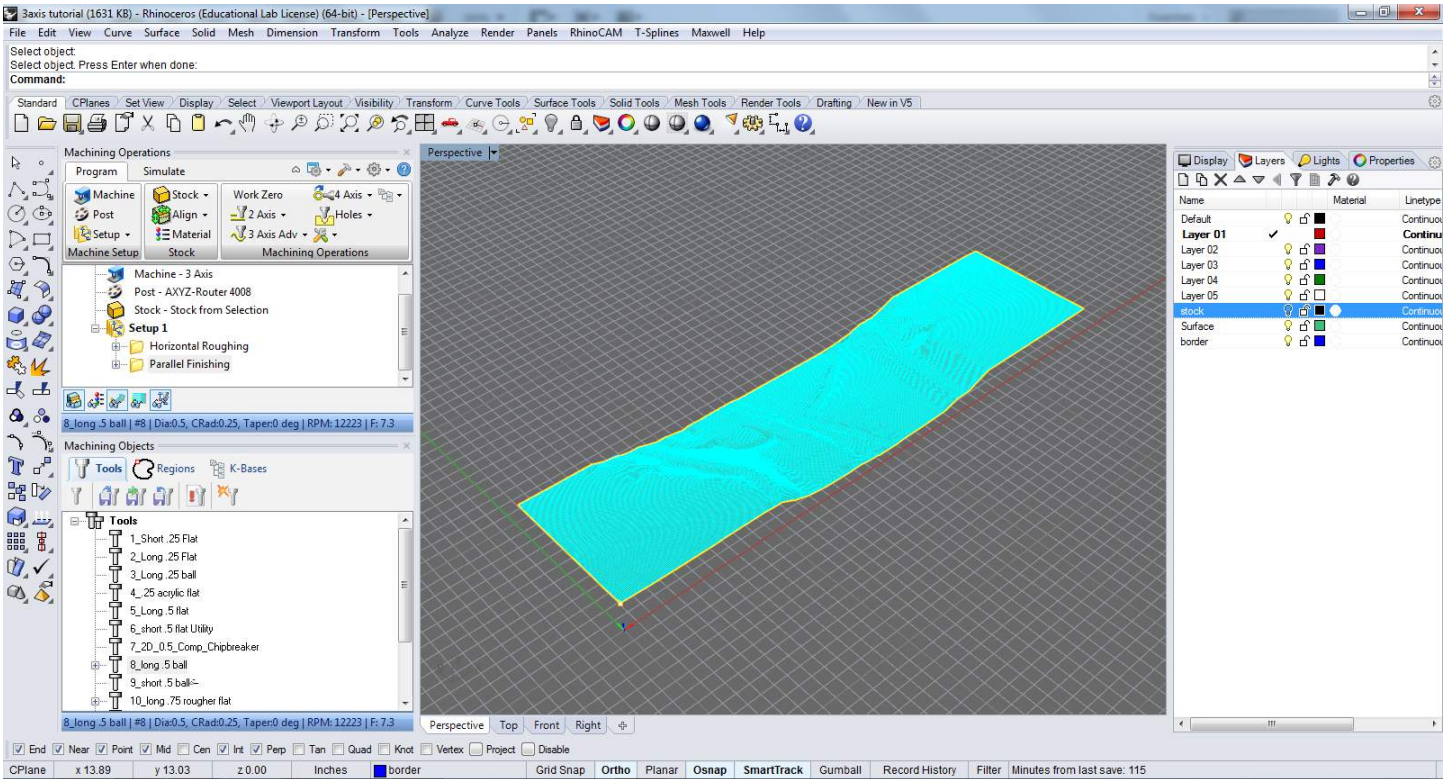


Under the Tool selection tab, select the tool used for the final finished surface mill. For the topography a .5 ball endmill is selected to give a smooth finish with a slight scallop from the stepover of the ball endmill.

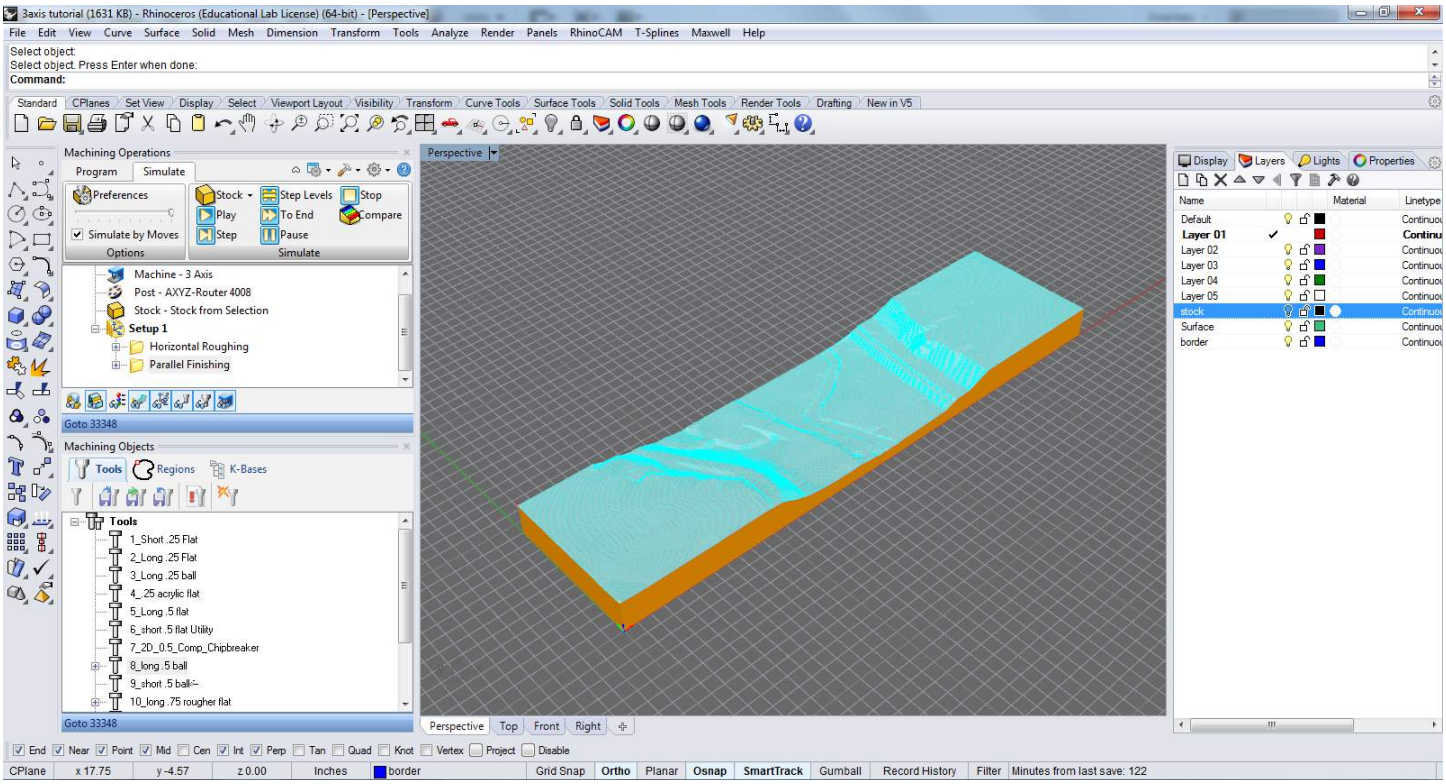
Rhino_CAM 3 Axis Machining Operations_ Parallel_Finishing



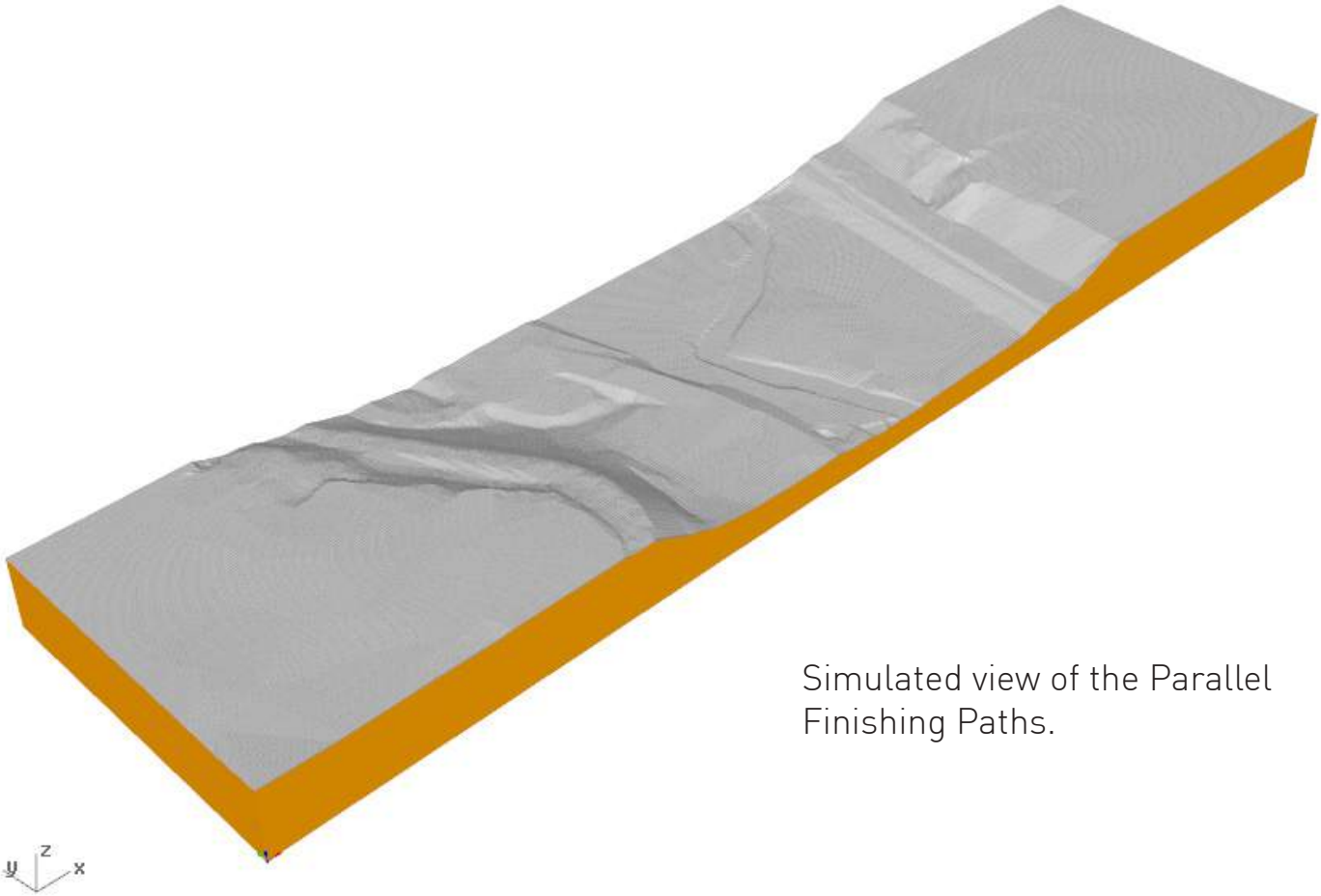
Under Cut Parameters set up the Cut Control, the direction is set to mixed and the angle is set to zero, if angled passes are desired set the angle of cuts to the chosen angle. The stepover control is set to 20%, the tighter the stepover the longer the milling time.



Once all the parameters have been set up press generate and the surface will be populated with milling paths following the finish surface of the topography.



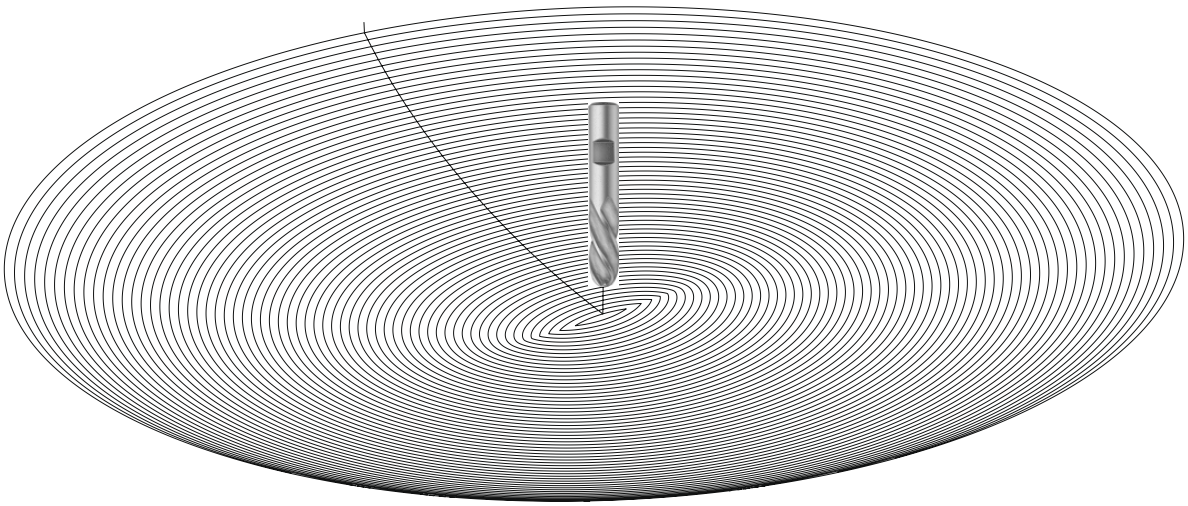
Under the simulate tab run the simulation to ensure that there are no errors or collisions in the tooling paths.

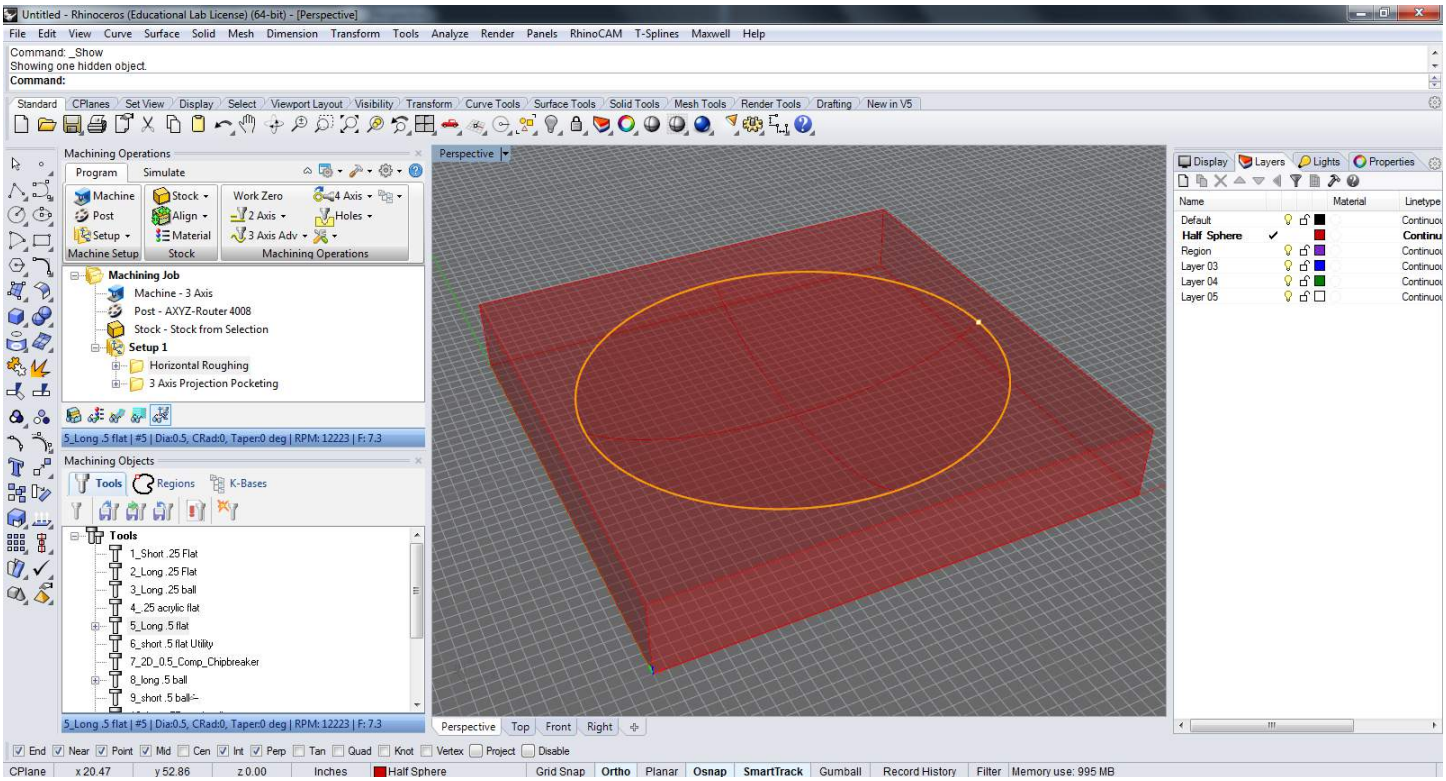


Simulated view of the Parallel Finishing Paths.

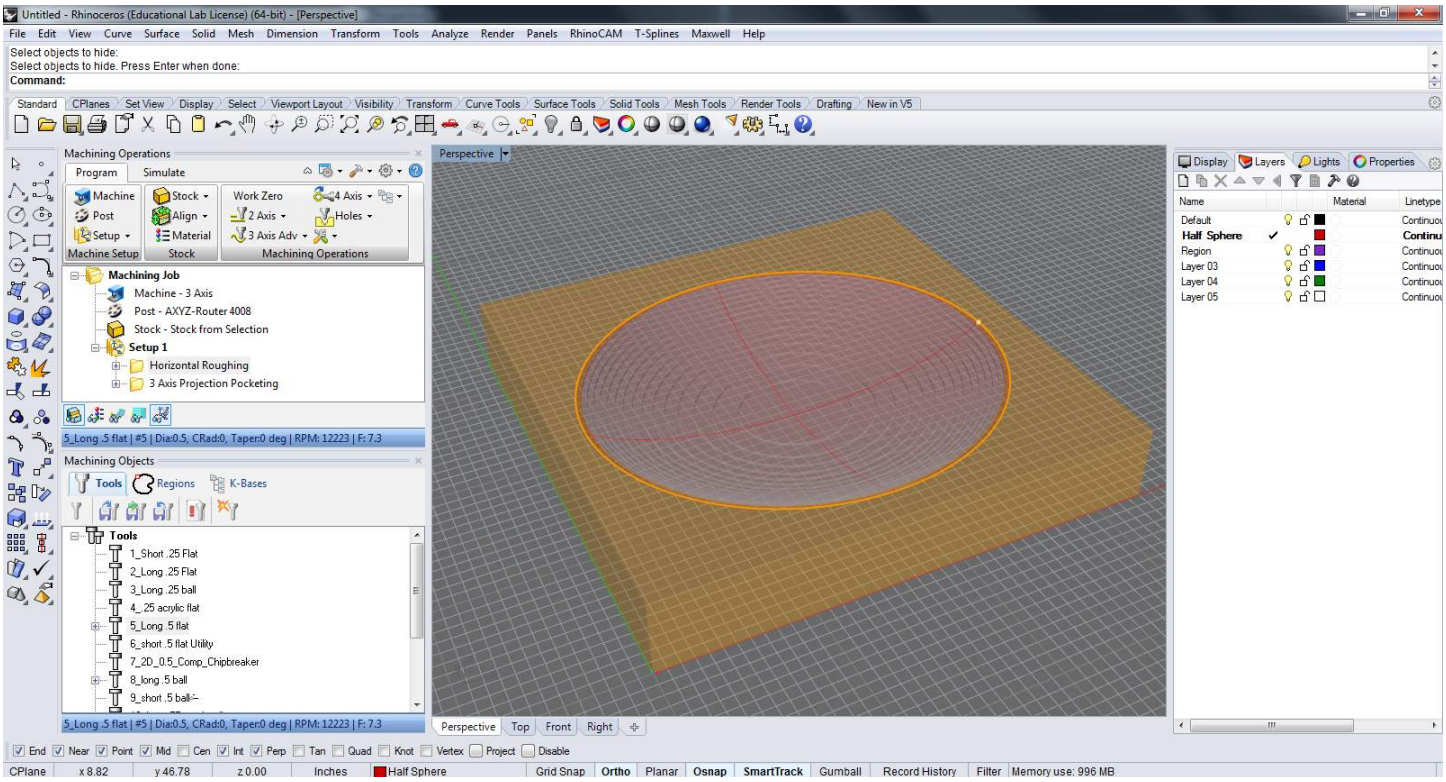
Rhino_CAM 3 Axis Machining Operations_ Projection_Pocketing

Projection pocketing operation can be used for Pre finishing or fine finishing surface bottoms in closed regions. In this method, the toolpath is generated in 2D and then projected down to the surfaces below. Machining regions are necessary to be active for this cut method to work. There is no limitation on the number of regions or the number of nesting of the region.

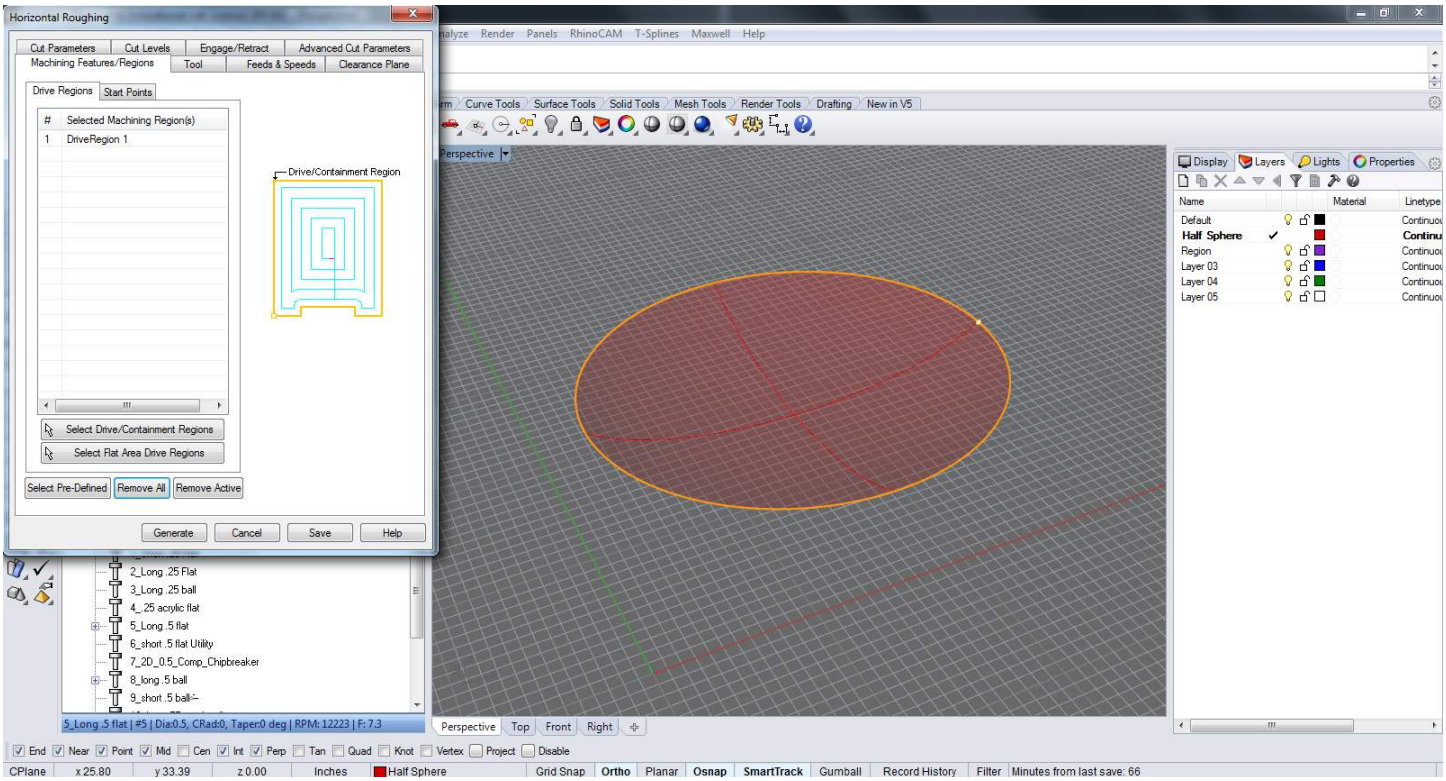




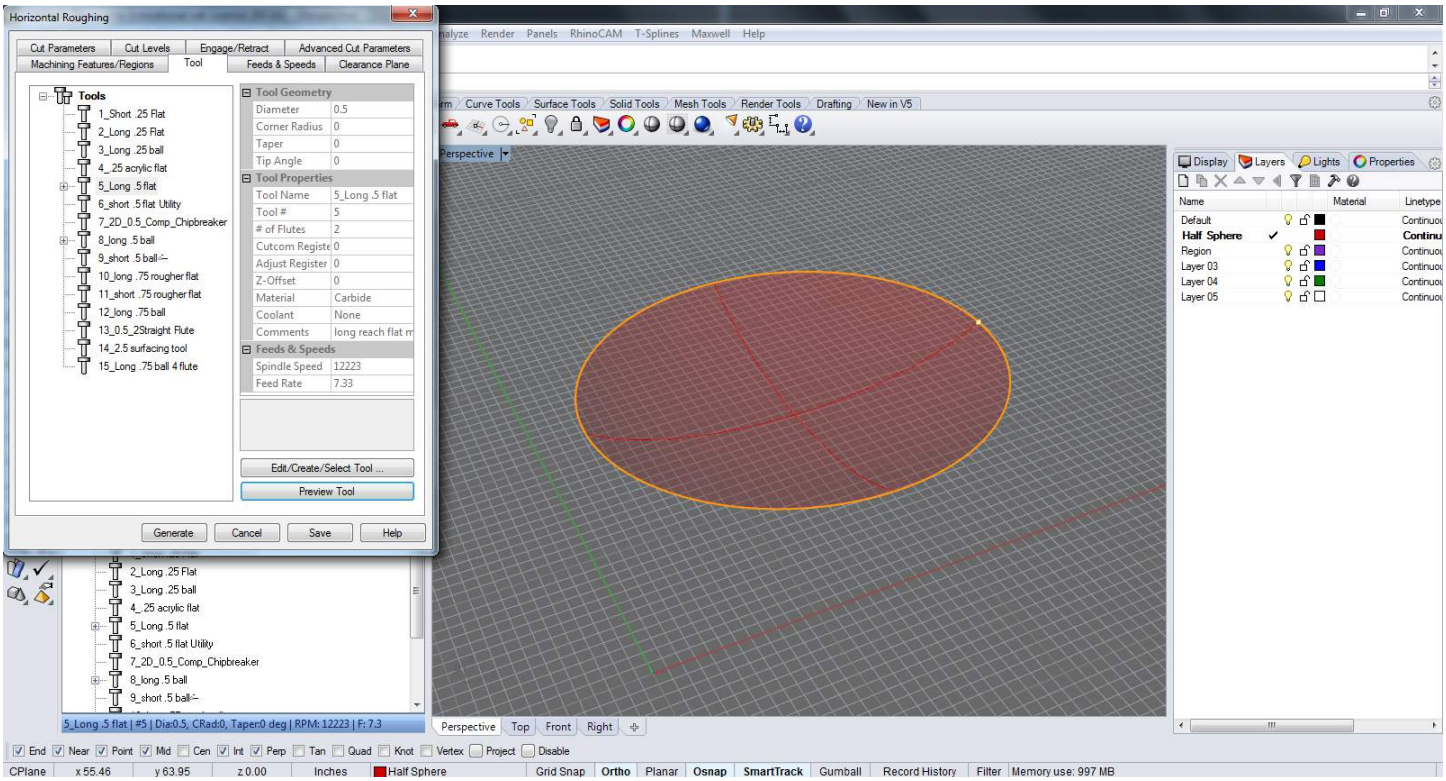
For the pocket projection we will start off with a horizontal roughing pass to clear out the material from the inside of the half sphere before the finishing pass with the projection pocketing.



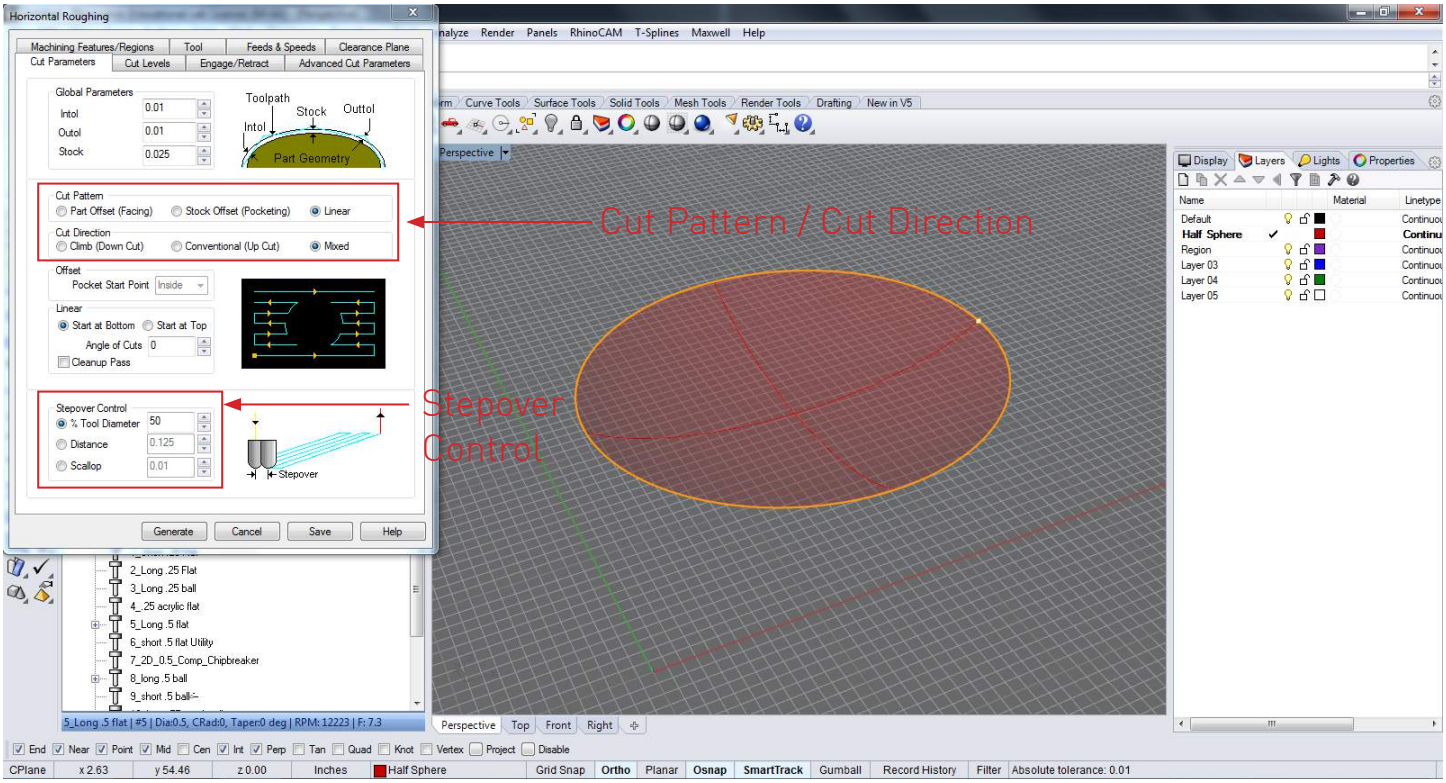
Create a stock from set to the stock size of your physical stock, create a bounding regions for the horizontal roughing pass.



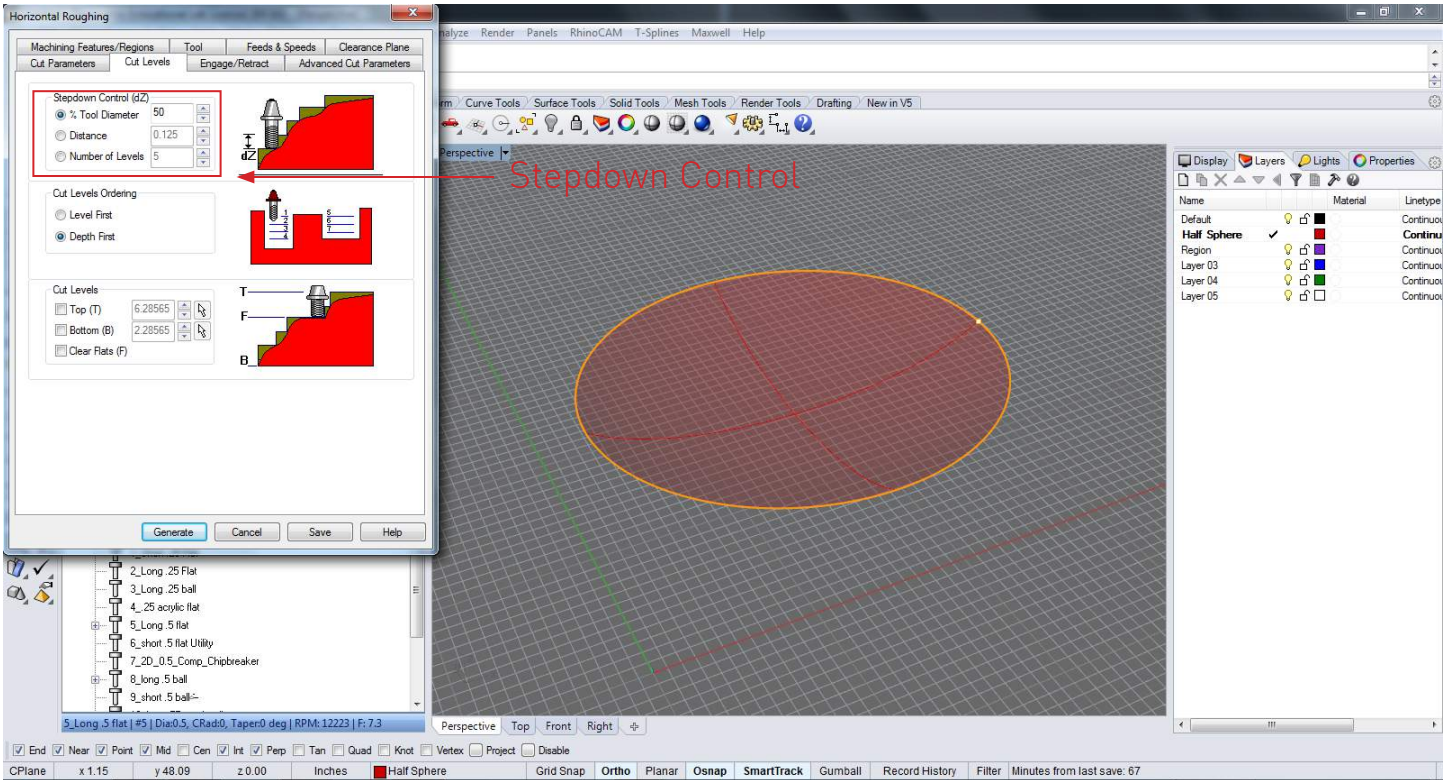
Select Horizontal roughing from the 3 axis adv drop down menu, select the drive region, this is the bounding region just created for the roughing pass.



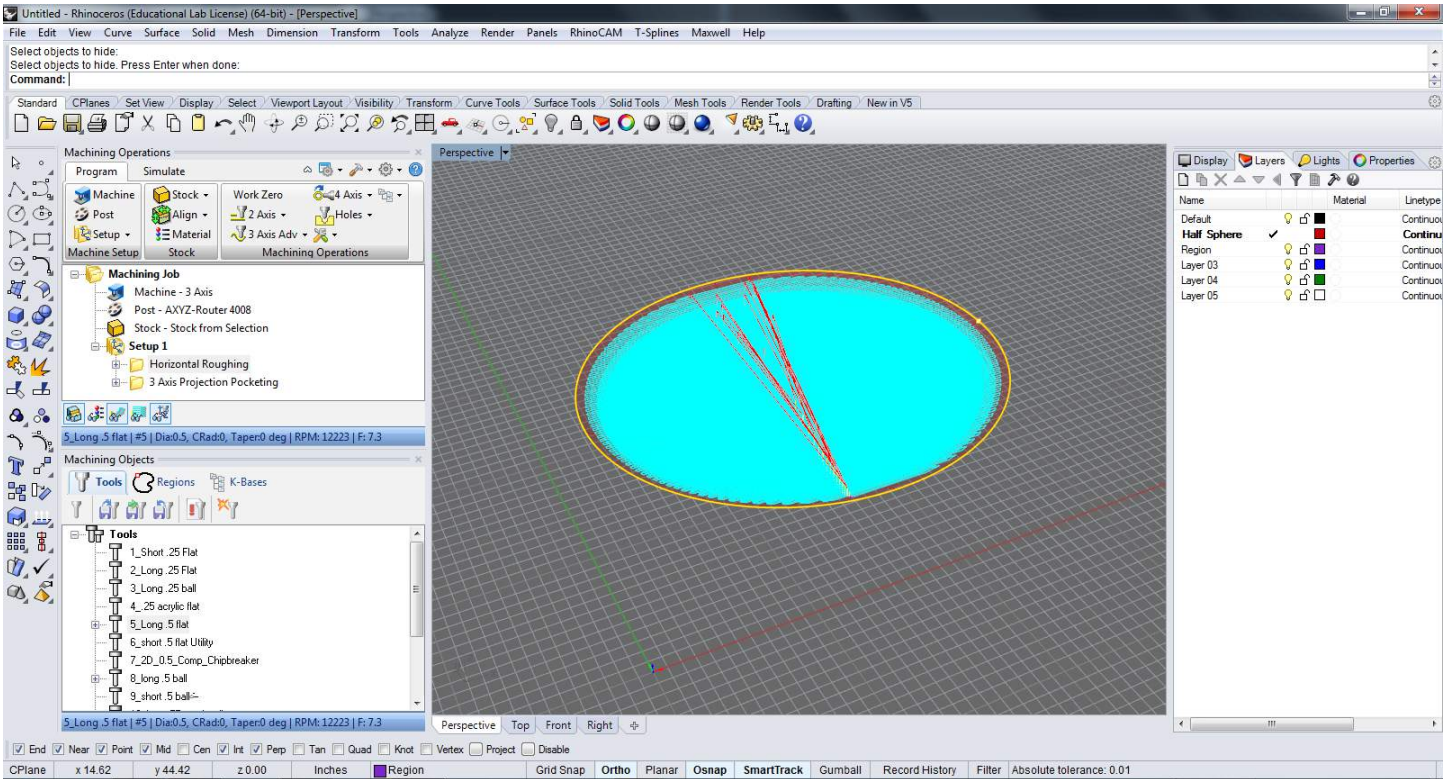
Select the proper endmill, dependent on the material being milled and the cutting depth required, in this example the long flat 1/2'' endmill is chosen from the Daniels tool library.



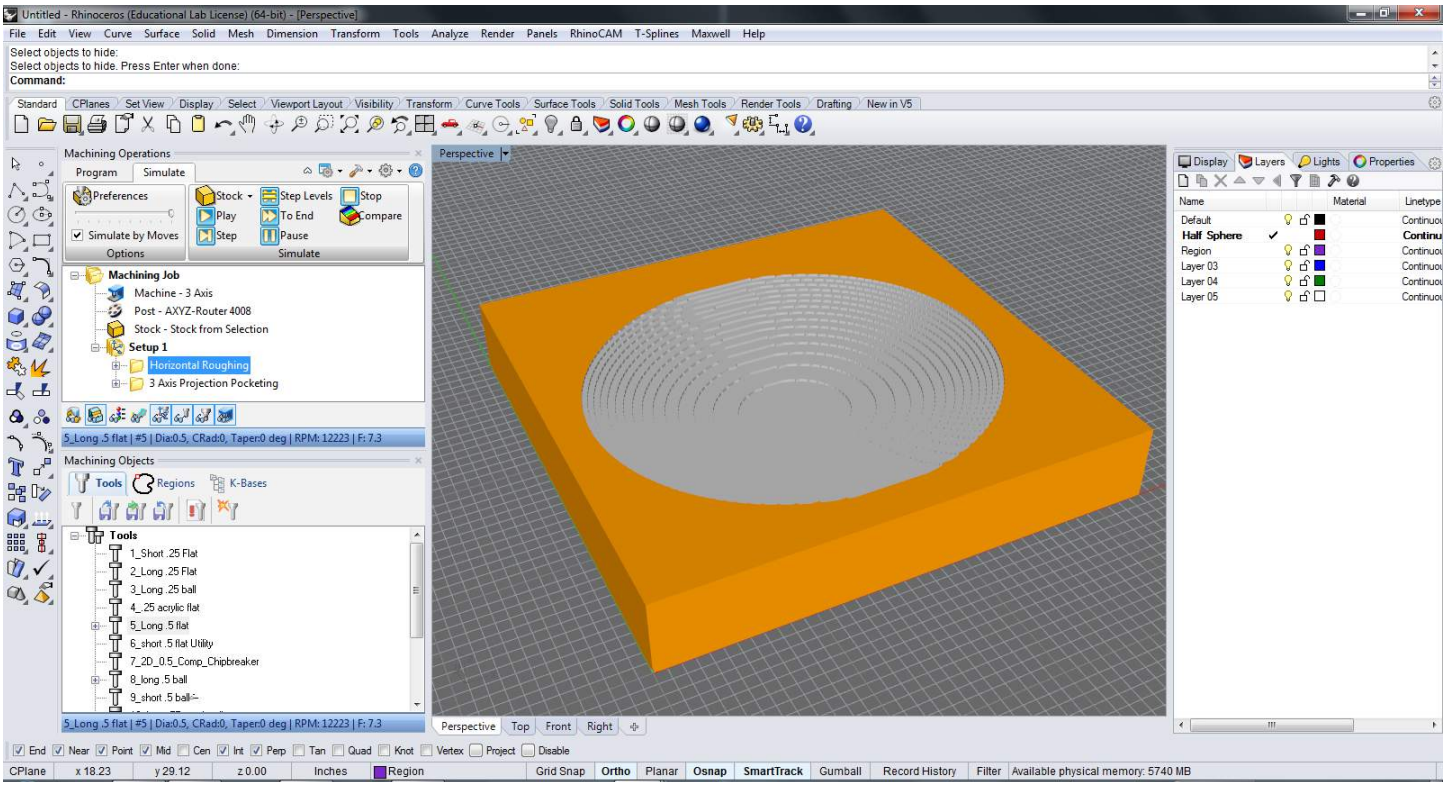
Under the Cut Parameters tab set the cut pattern to Linear and the cut direction to mixed, the stepover should be set accordingly to the material being milled.



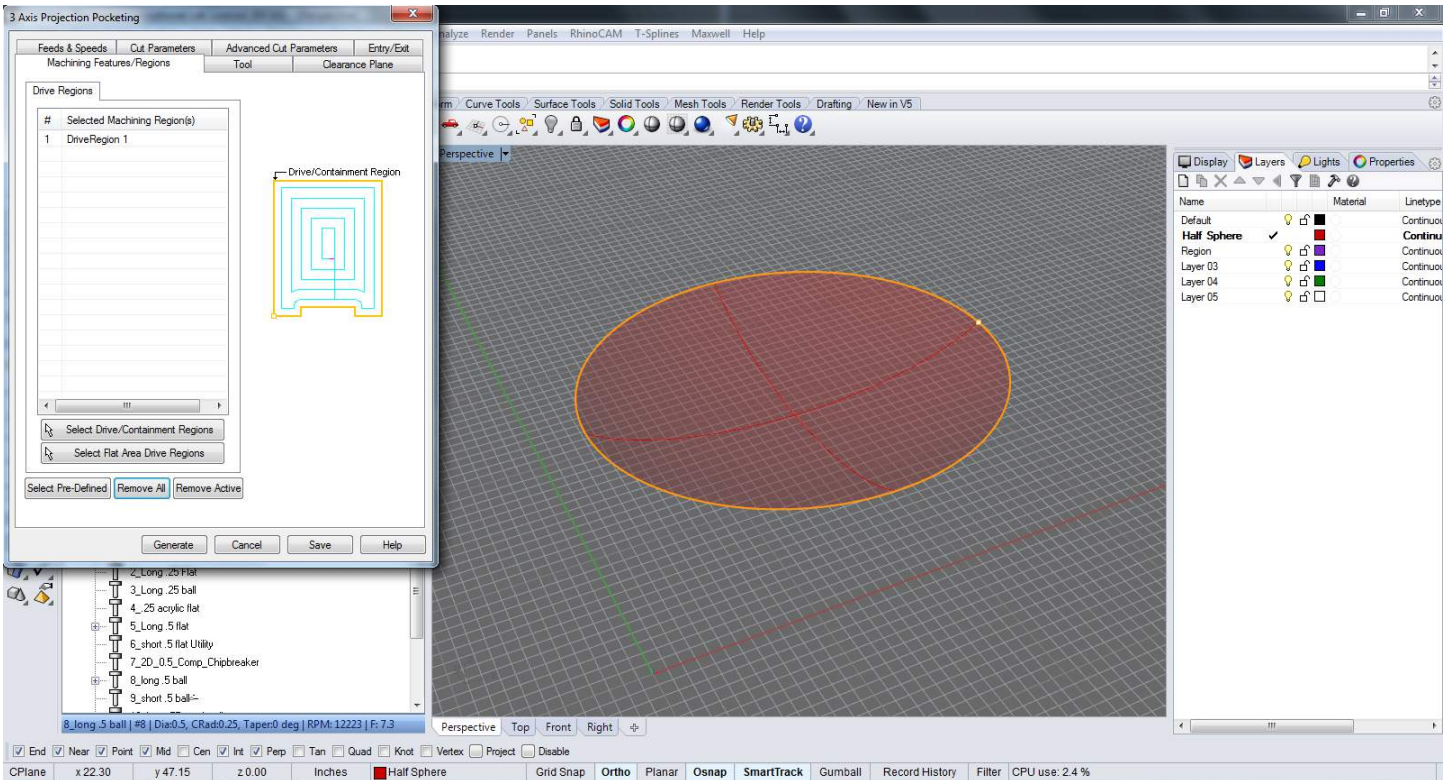
Under cut levels set the stepdown control to 50 percent of the tool diameter, this percentage can be higher for less dense material (ie foam).



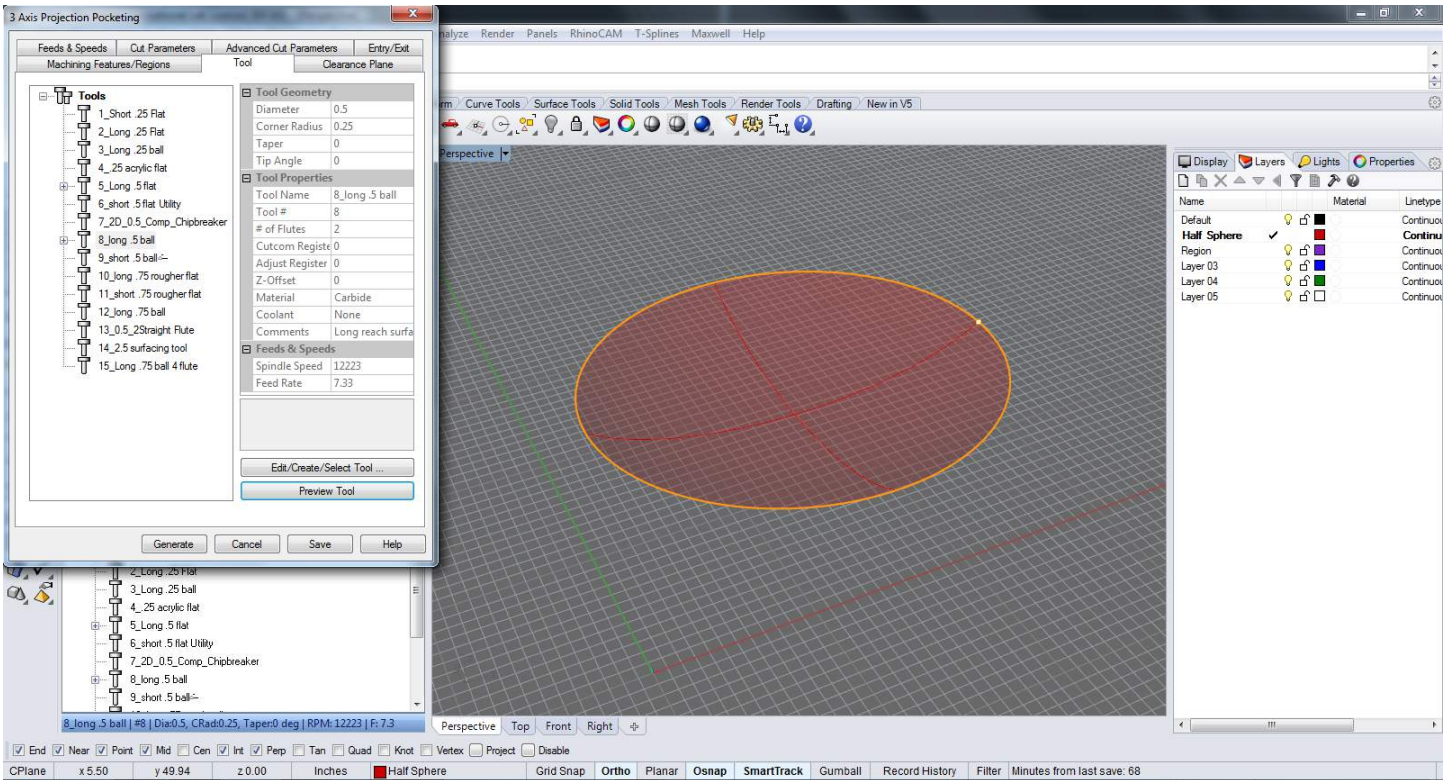
Once all the settings have been completed press generate to view the tooling paths created by the input parameters.



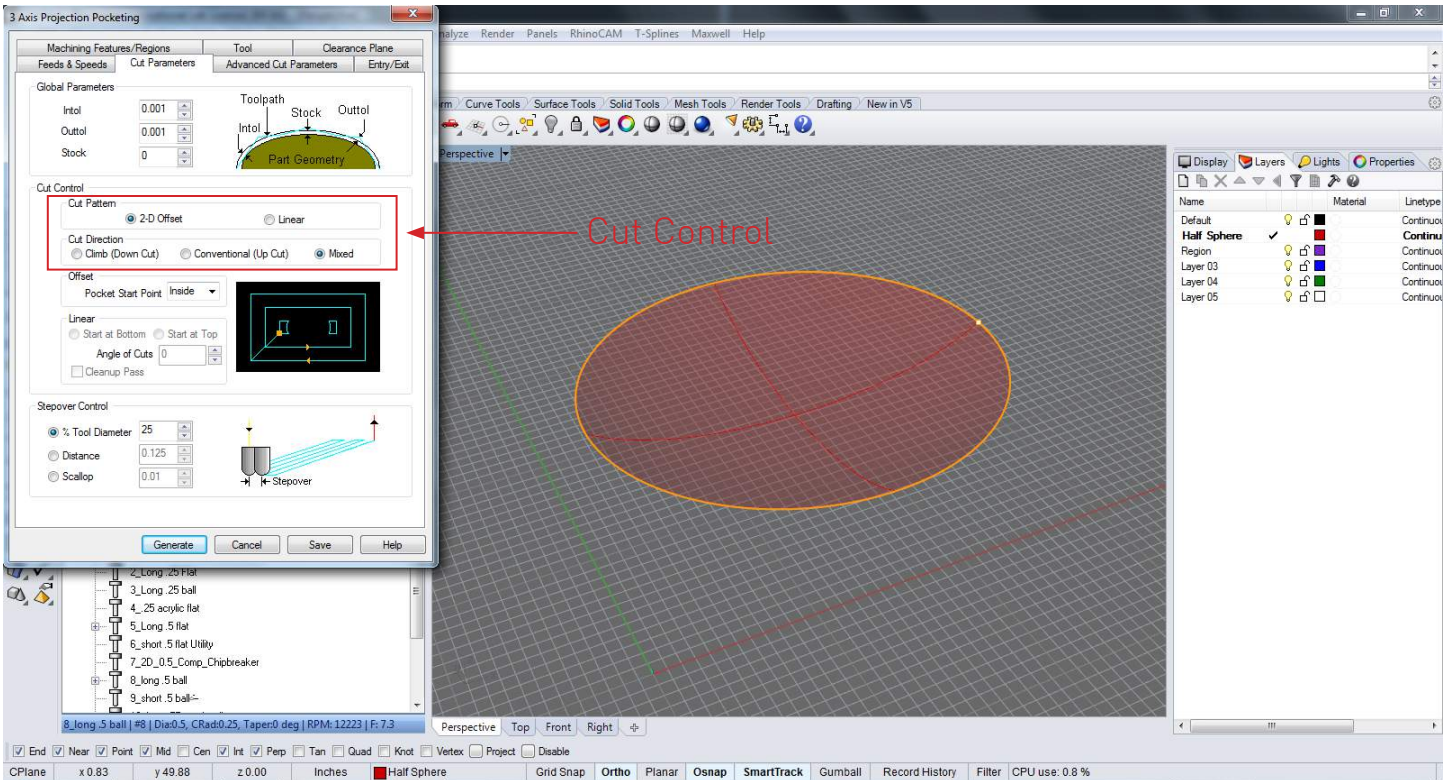
Go to the Simulate Tab and run the simulation to make sure there are no collisions or errors in the tooling path setup.



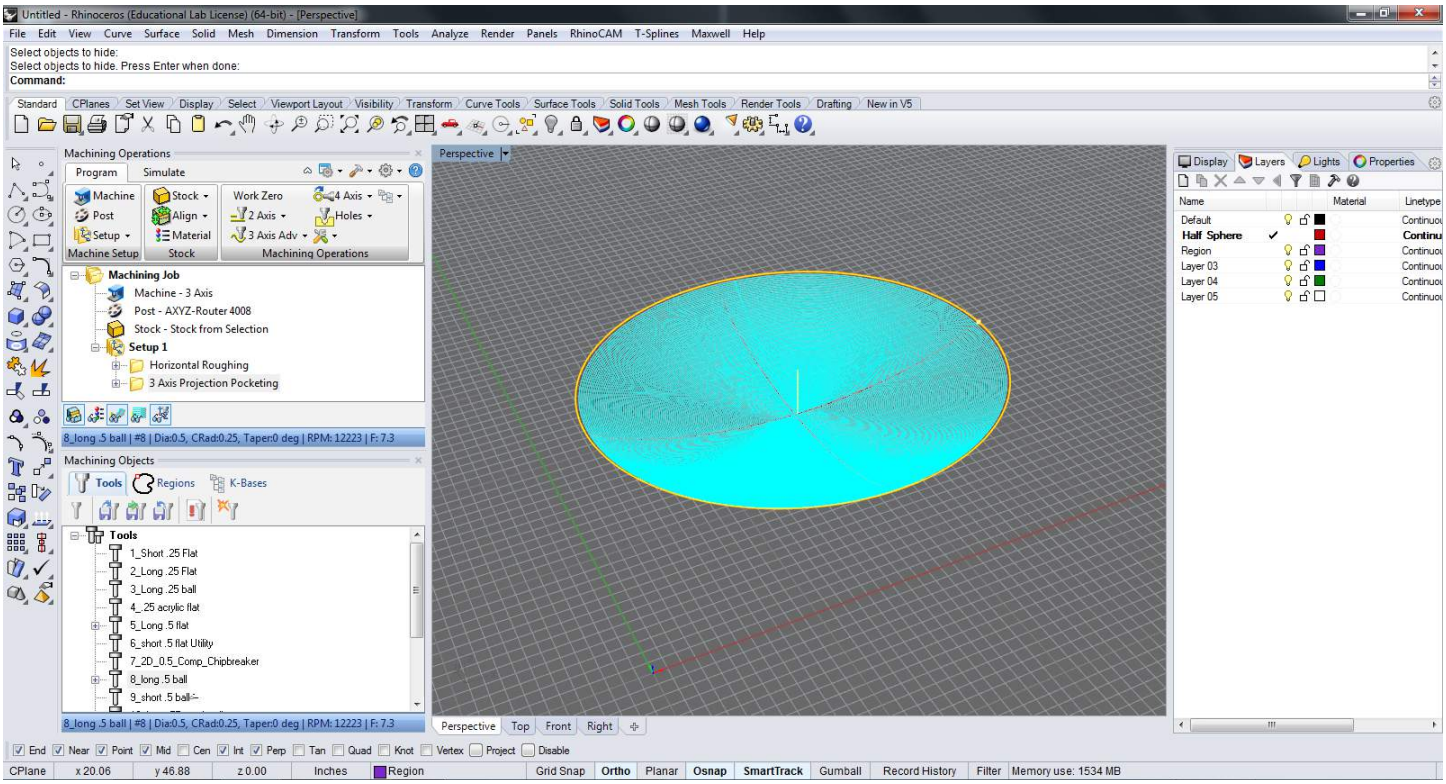
Select Projection Pocketing from the 3 axis adv menu, select the same drive region that was just used the in the previous horizontal roughing pass.



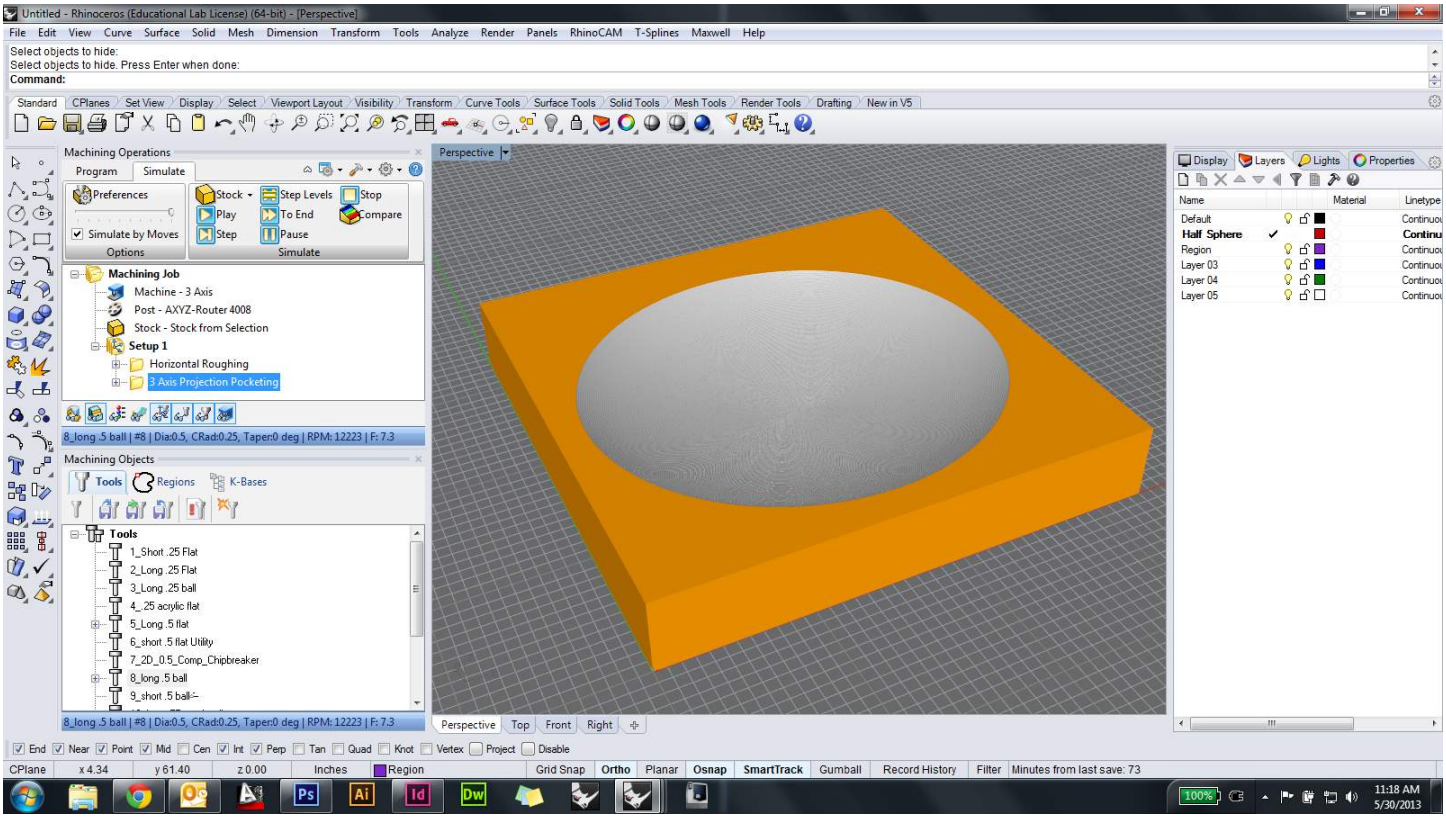
Under Tool select a ball endmill to create a smoother finishing pass, in the example the Daniels long 1/2'' ball endmill is selected.



Under Cutting Parameters set the cut direction to mixed, the cut control is set to 2D Offset, linear can be selected depending on the resultant tooling paths desired. The stepover is set to 25% for a smoother final pass.



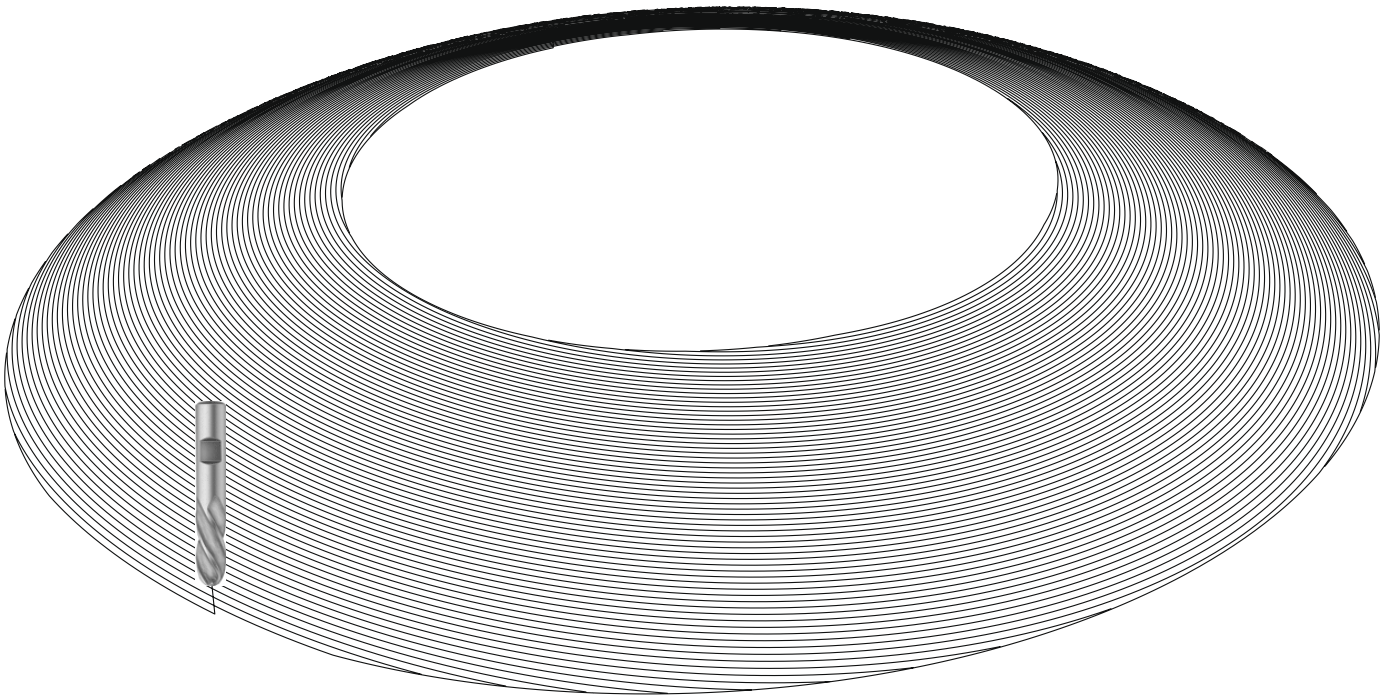
Once all the parameters have been setup press generate to create the tooling paths, parameters can be re-adjusted if tighter / looser milling paths are required.

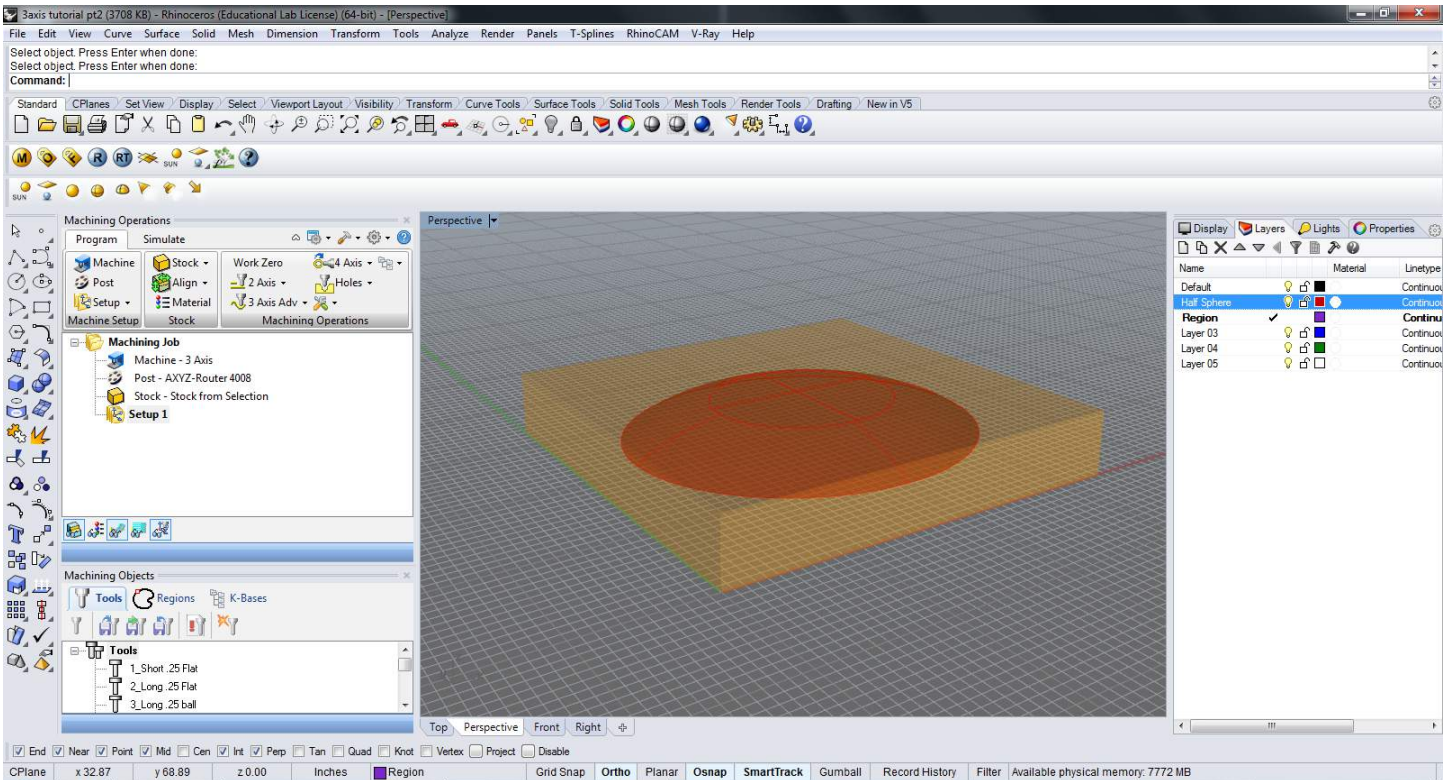


Under the Simulate tab run the simulation to see the final surface with the Projection Pocketing. Check to make sure there are no collisions or errors in the milling paths.

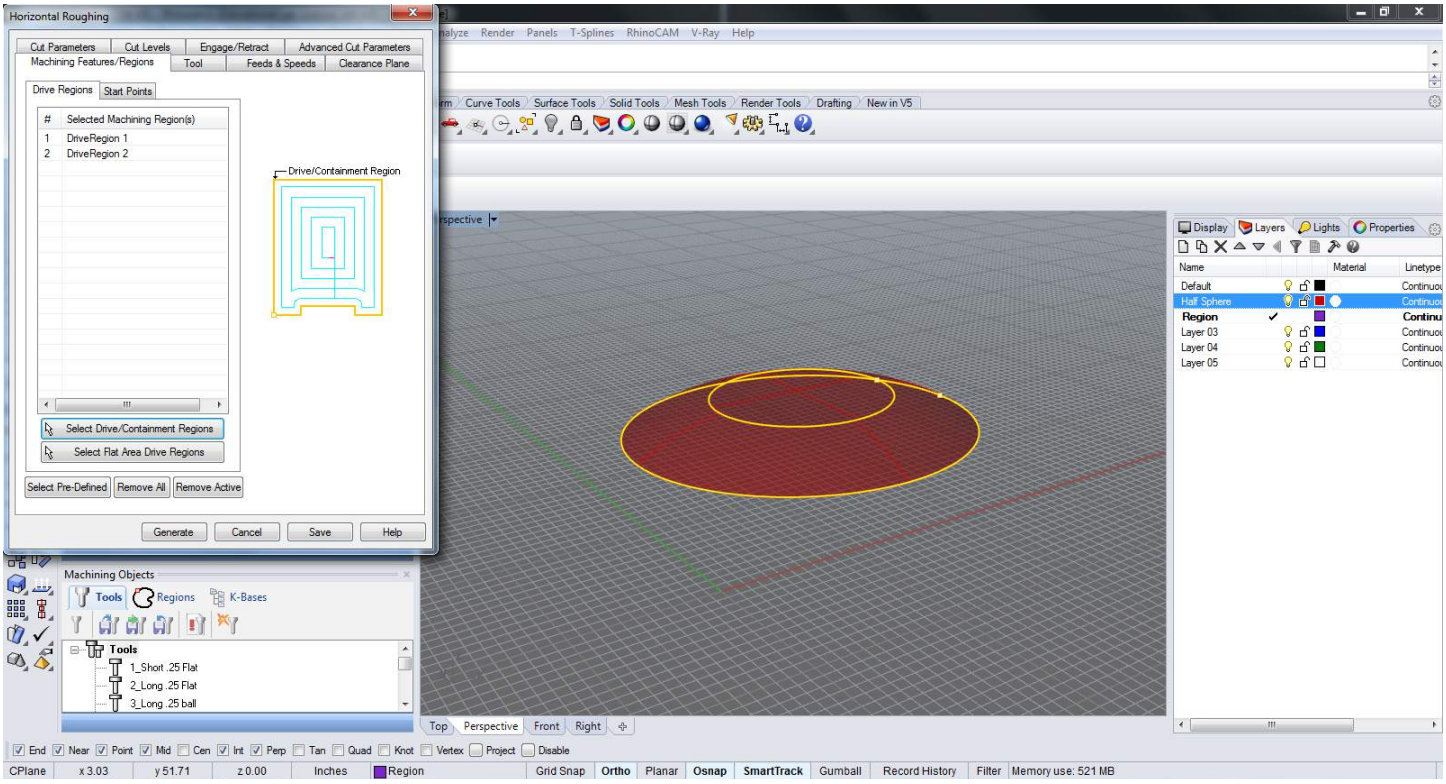
Rhino_CAM 3 Axis Machining Operations_ Spiral_Machining

Spiral machining is a method of generating a spiral toolpath. It can be used efficiently for circular regions. Single/multiple regions must be selected and activated to generate the spiral toolpath. The toolpath will be generated only within the activated regions.

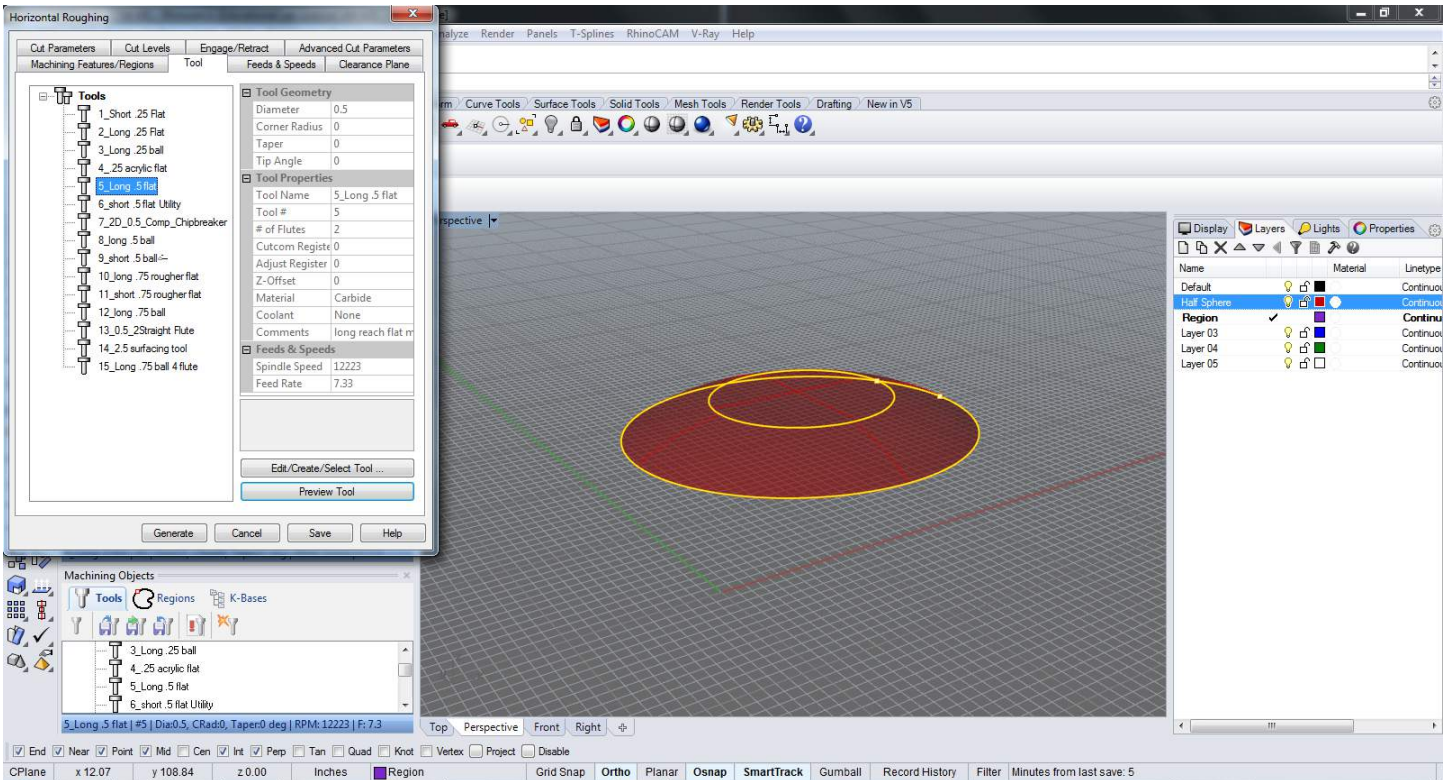




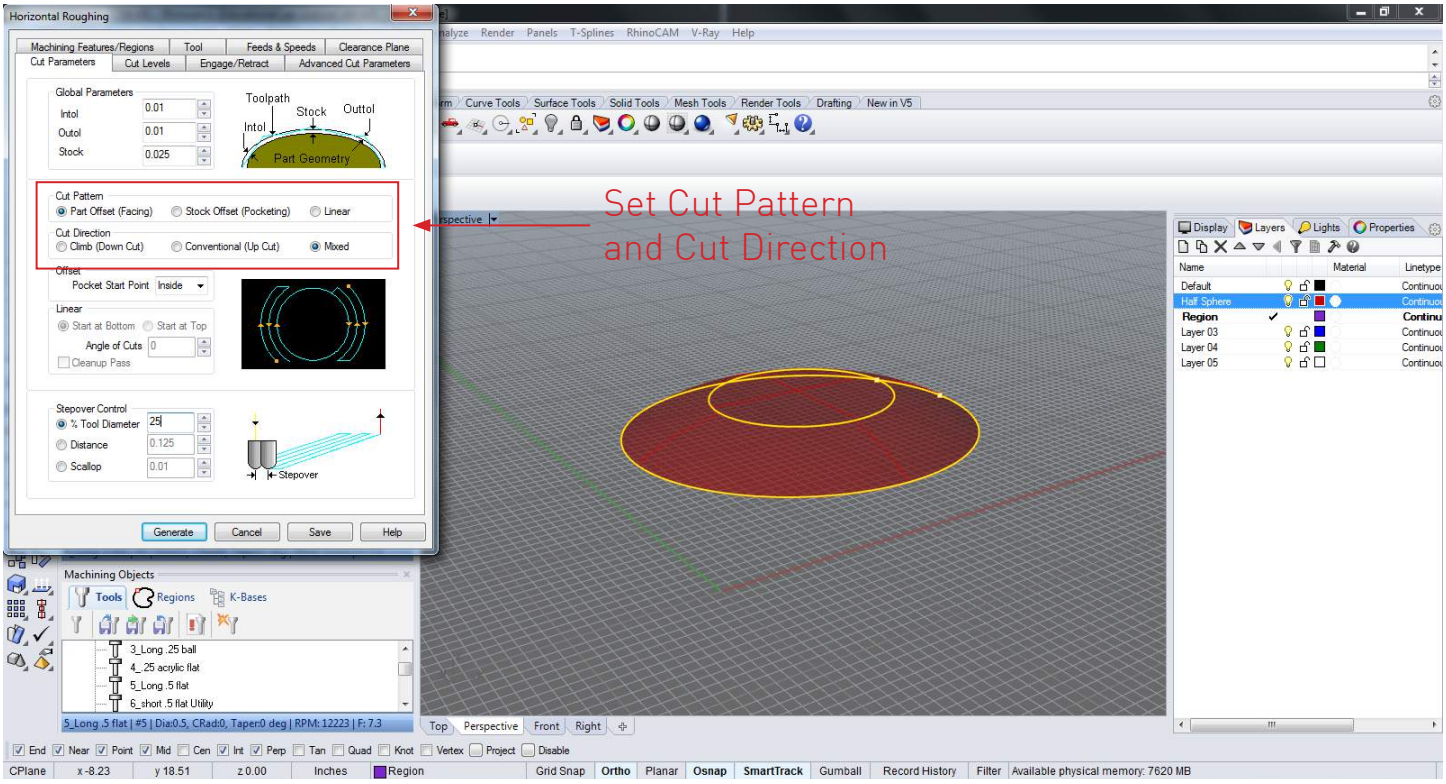
For Spiral Machining the same process will be repeated from the projection pocketing example. Begin with placing the poly object in the stock and then use Horizontal Roughing to remove the material before the final milling pass with Spiral Machining.



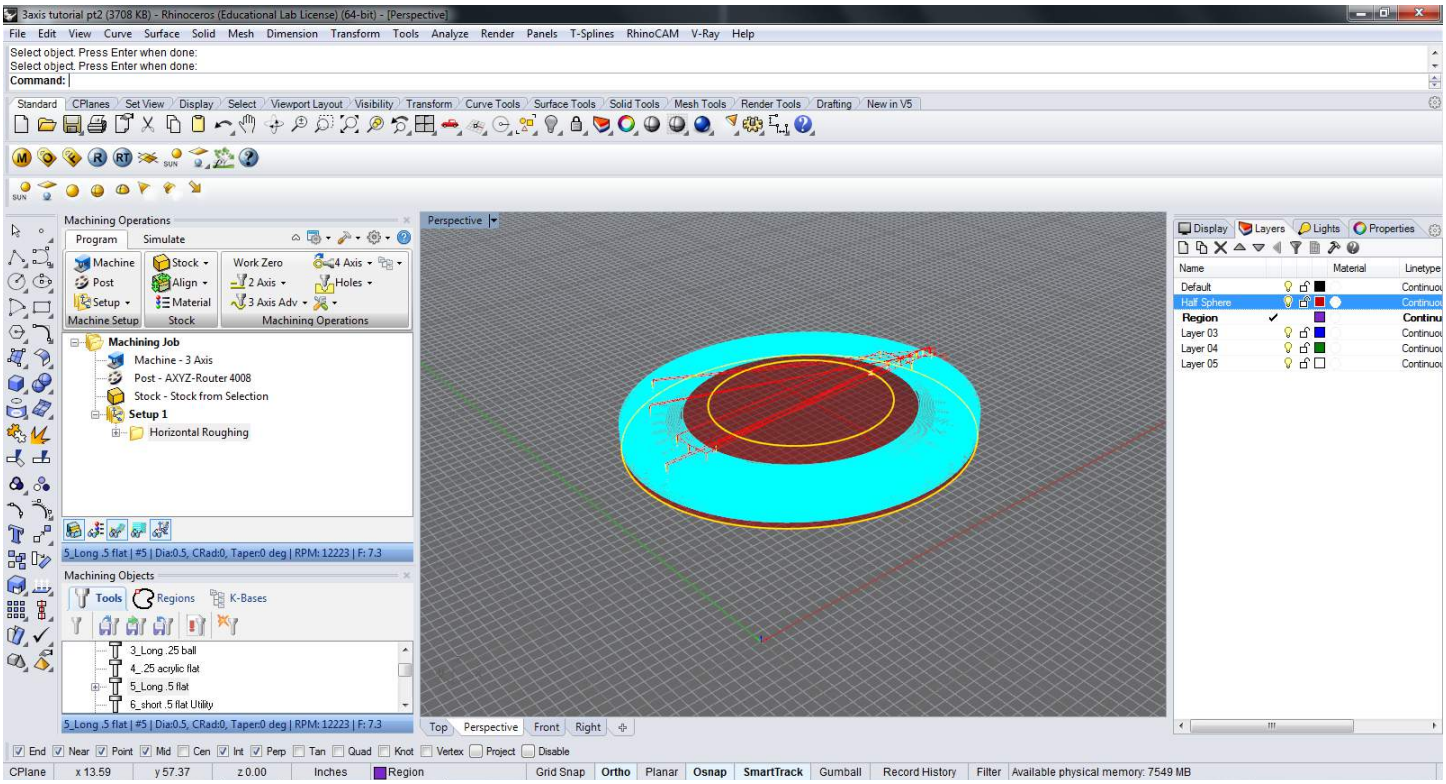
For the Horizontal Roughing create to drive regions to contain the Horizontal Roughing pass, the pass will run between this two defined regions.



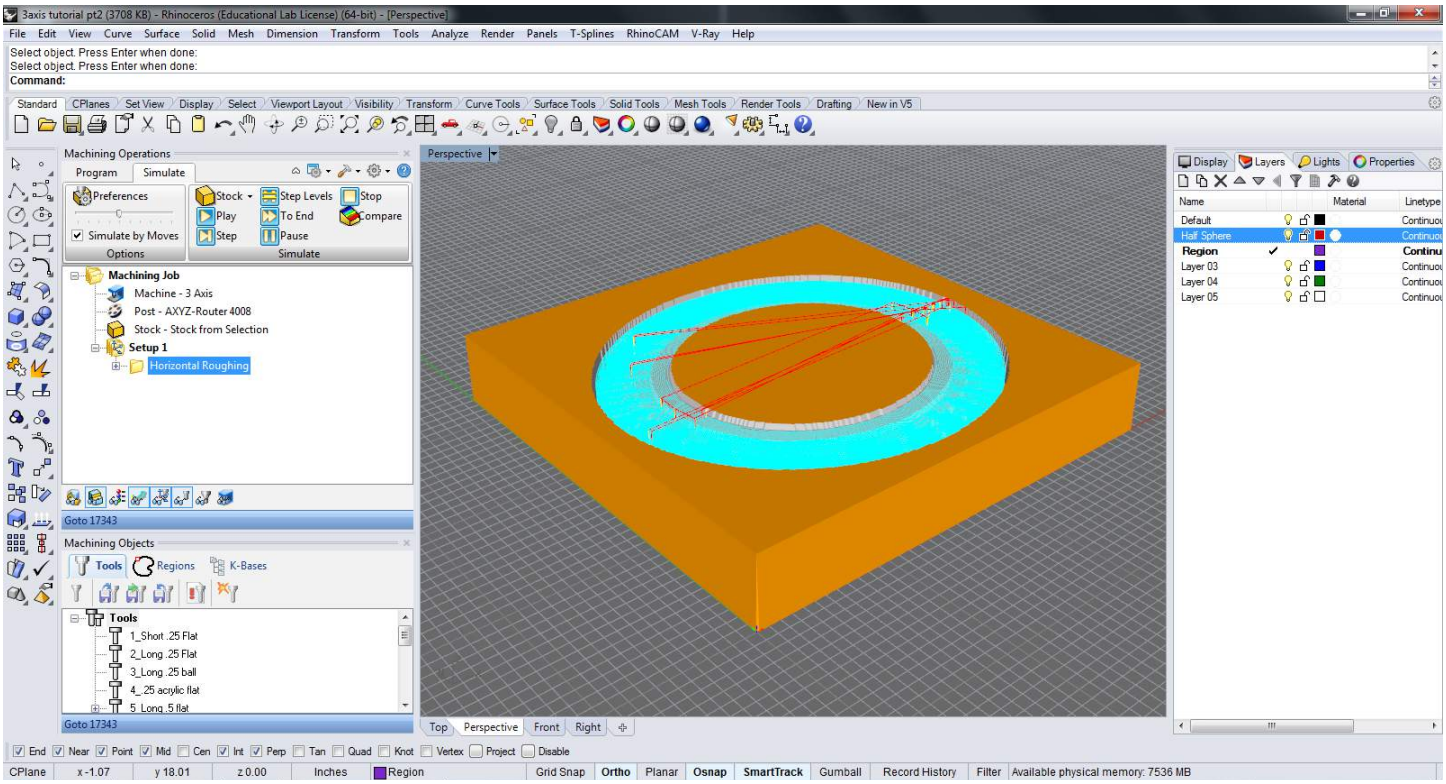
Under Tool for the Roughing pass select the .5" long Flat endmill.



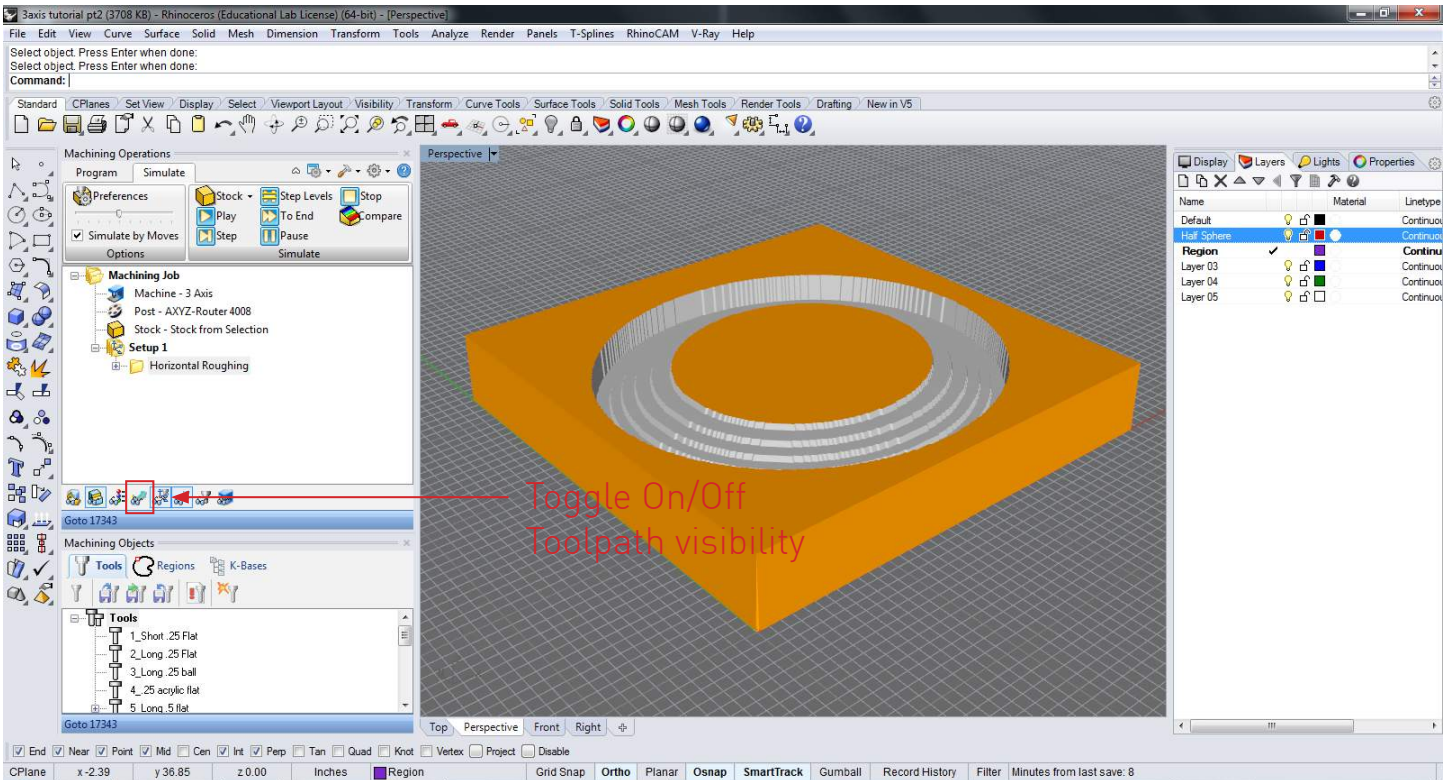
Under the Cut Parameters set the Cut Pattern to Part Offset and the Cut Direction to mixed. The tool diameter is set to 25% in this example, for less dense material it can be set higher to speed up the Roughing Pass.



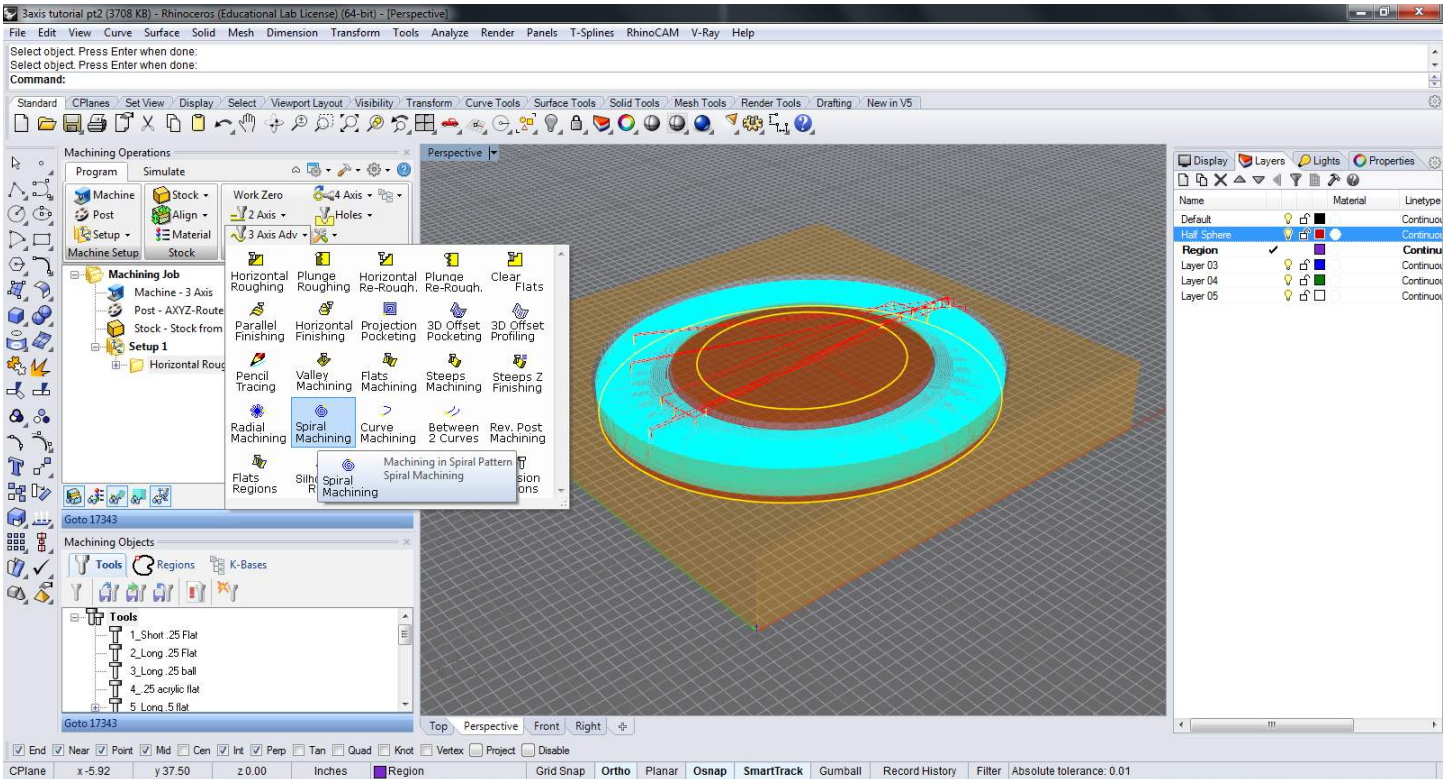
Once all the parameters have been set up press generate to create the milling paths.



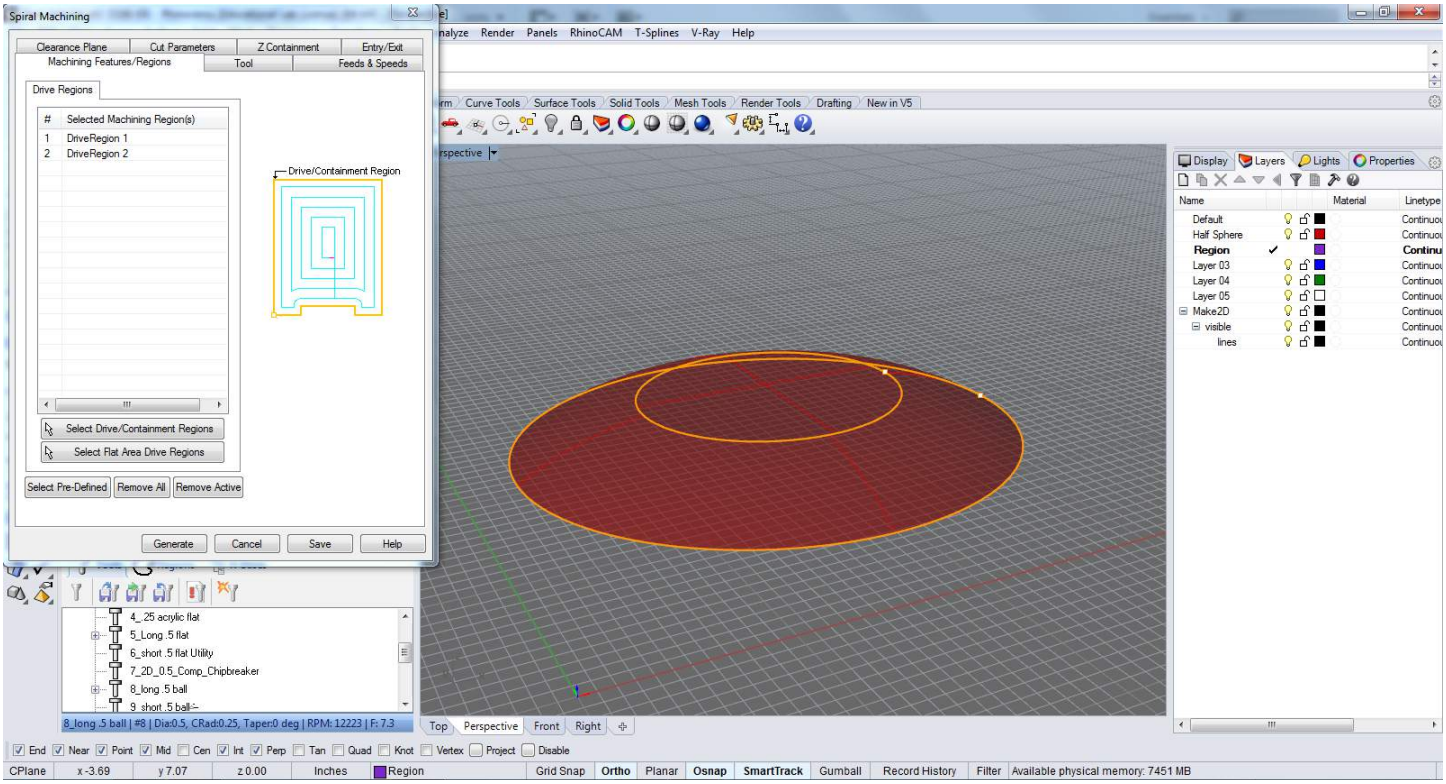
Under the Simulate tab run the simulation to check for any collisions with the head and make sure the milling paths are the desired ones.



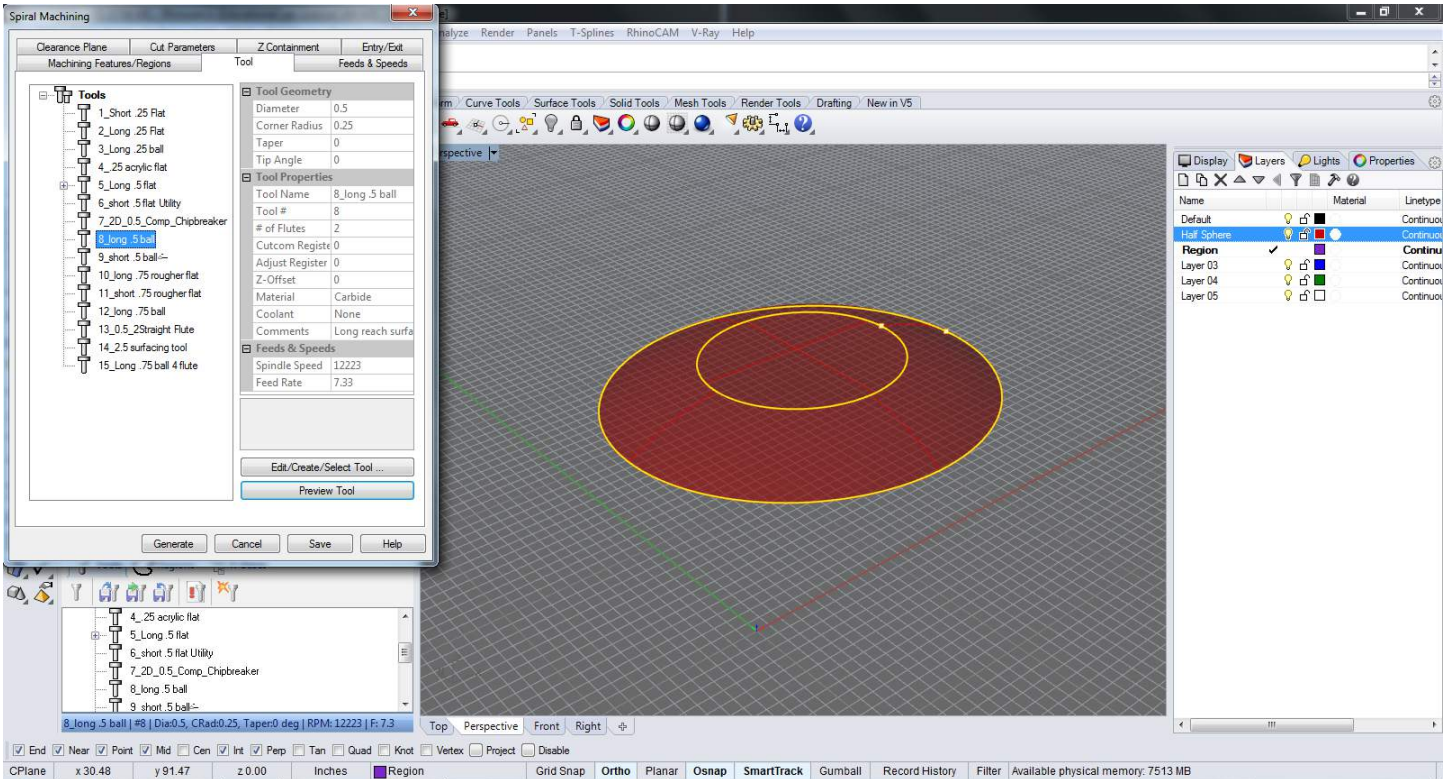
Toggle off the tooling paths to view the Horizontal Roughing path surface.



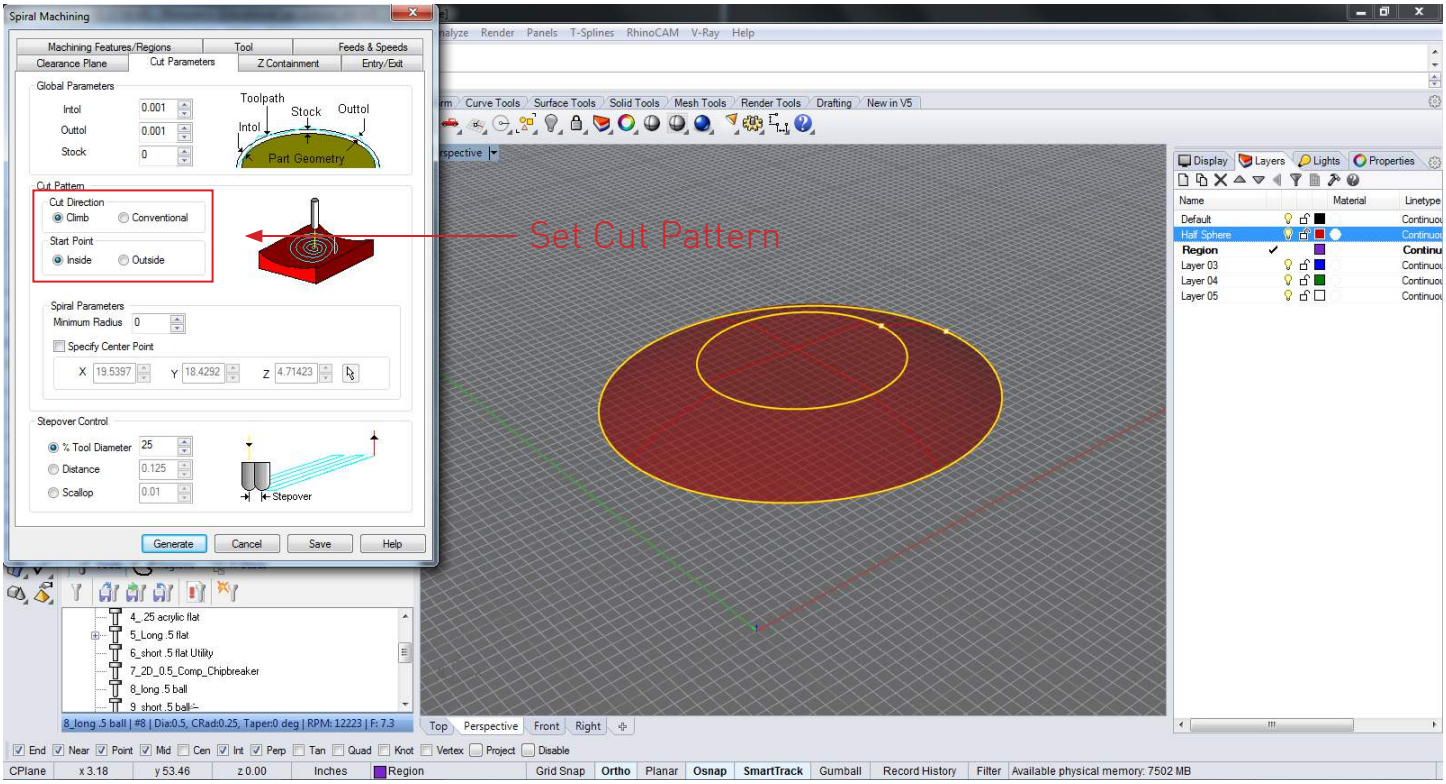
Under the 3 Axis Adv select the Spiral Machining tab.



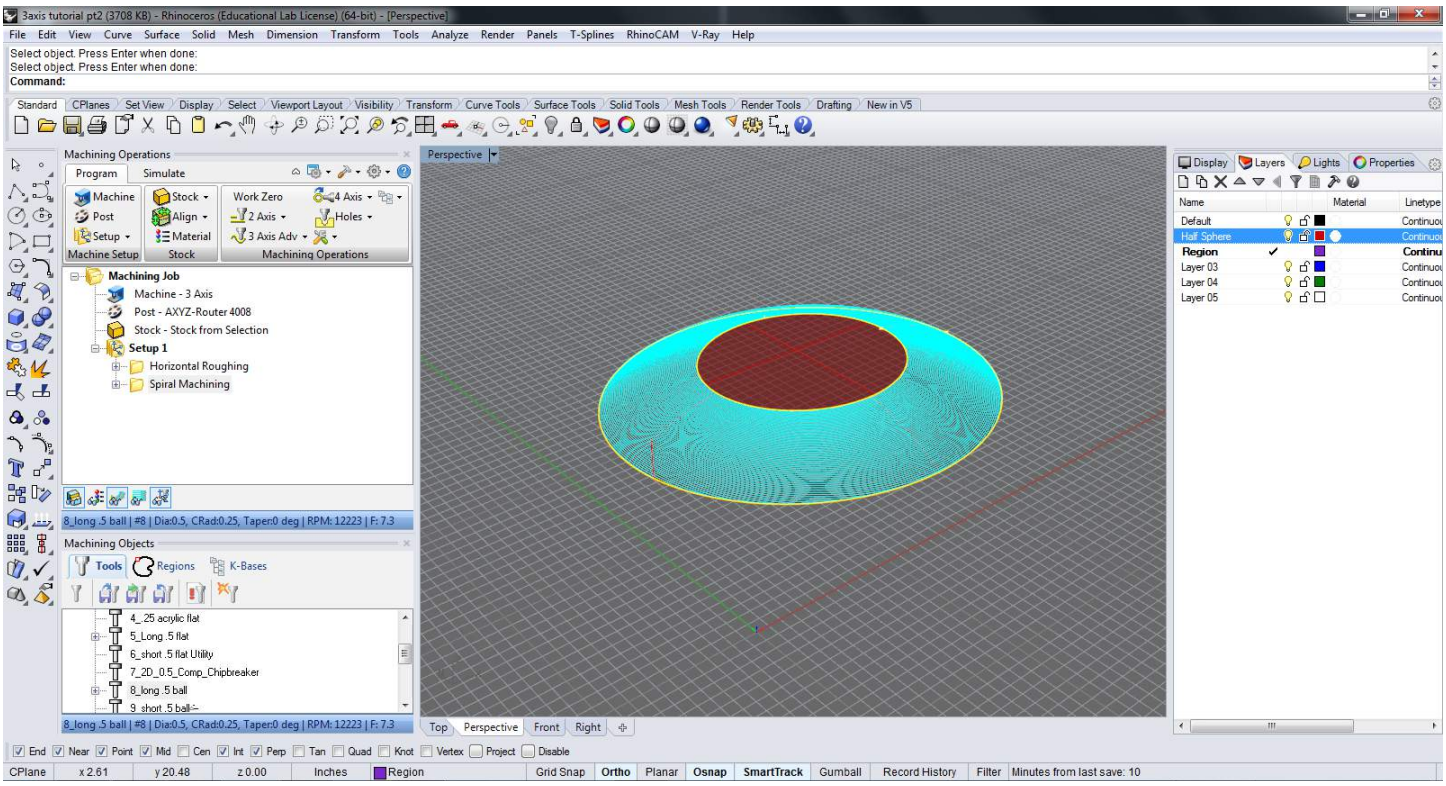
Under Machining Features/Regions select the same two drive regions from the Horizontal Roughing Pass.



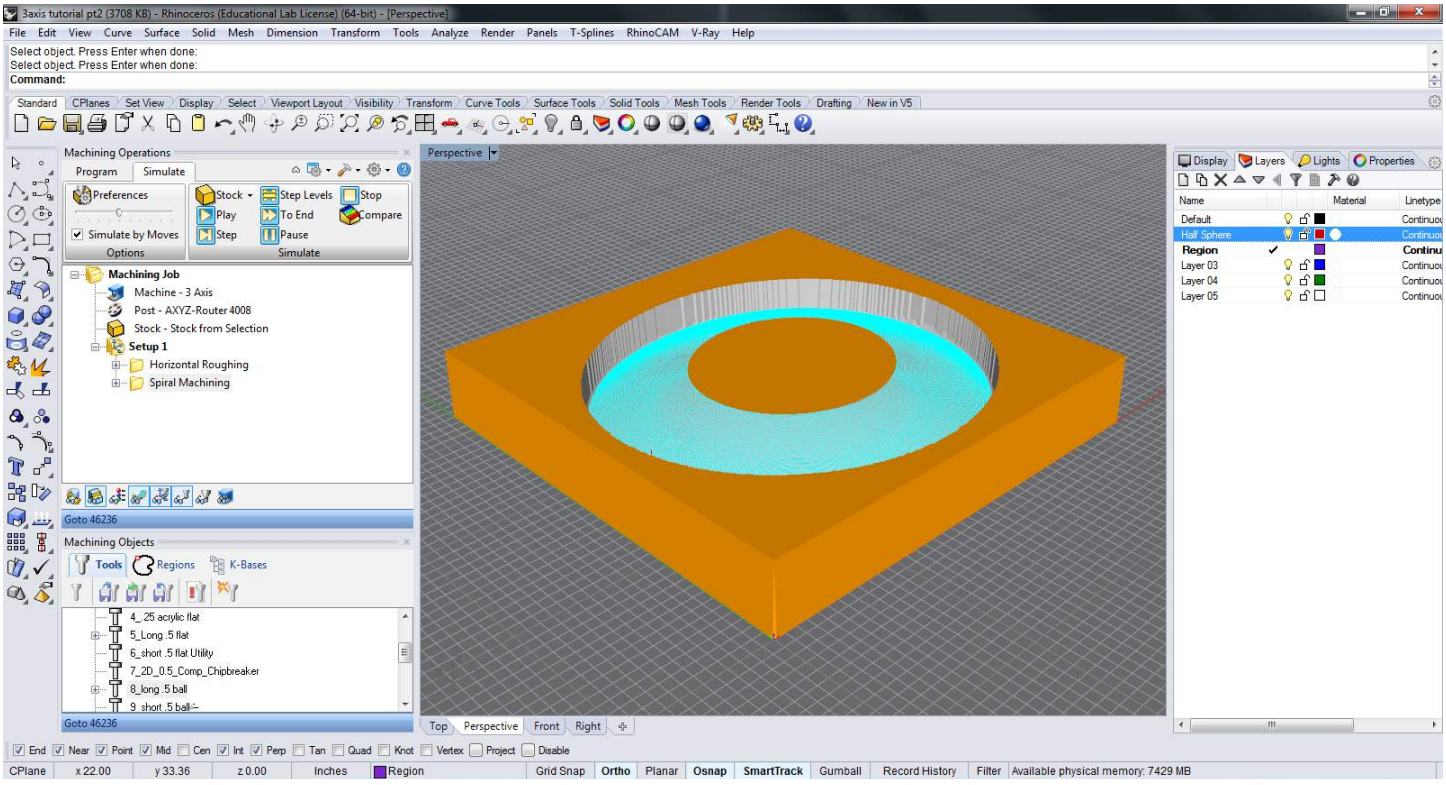
Under the Tool tab select the endmill for the finished surface, the long .5'' ball is used in the example.



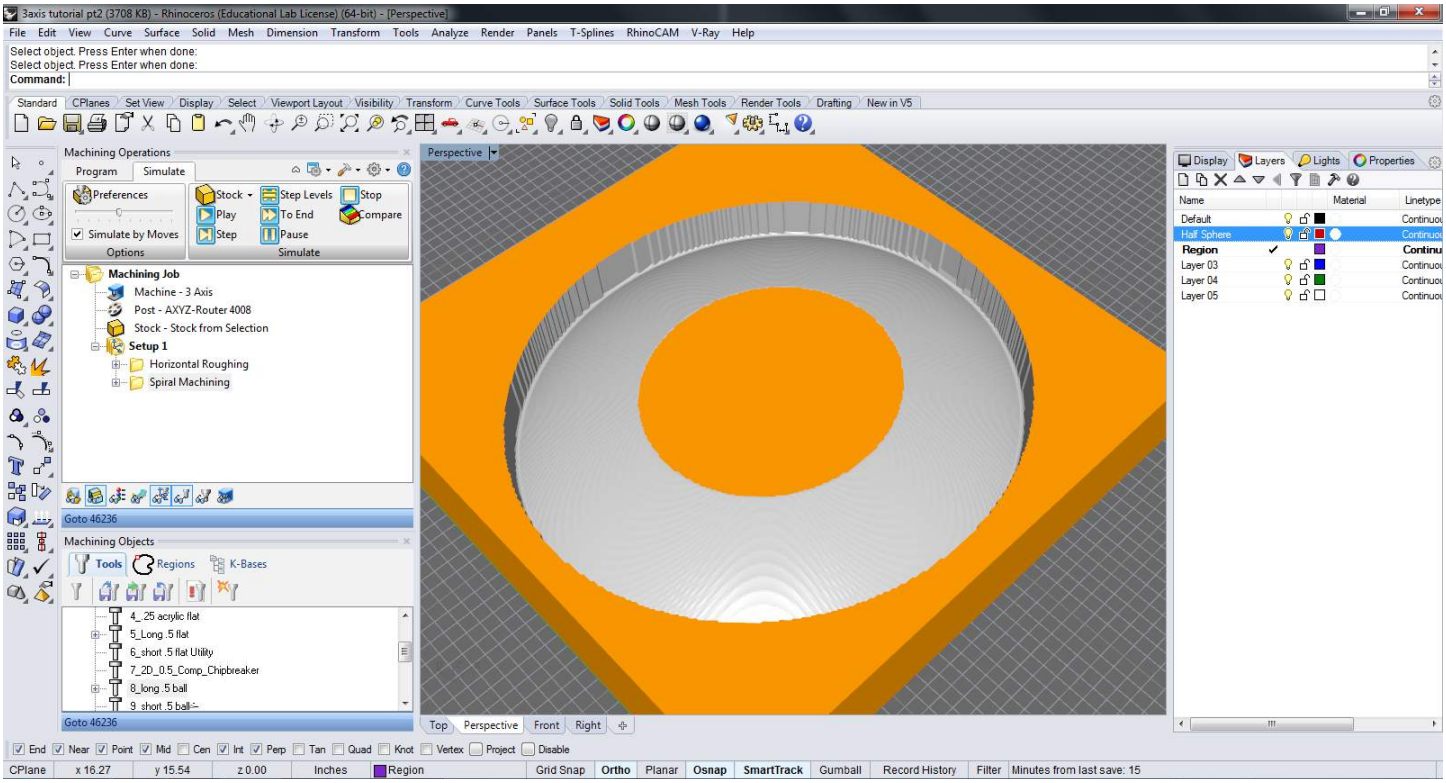
Under Cut Parameters The Cut Direction is set to Climb, the start point is set from the inside and the Stepover Control is set to 25%.



Once all the parameters have been set up press generate to create the milling paths.



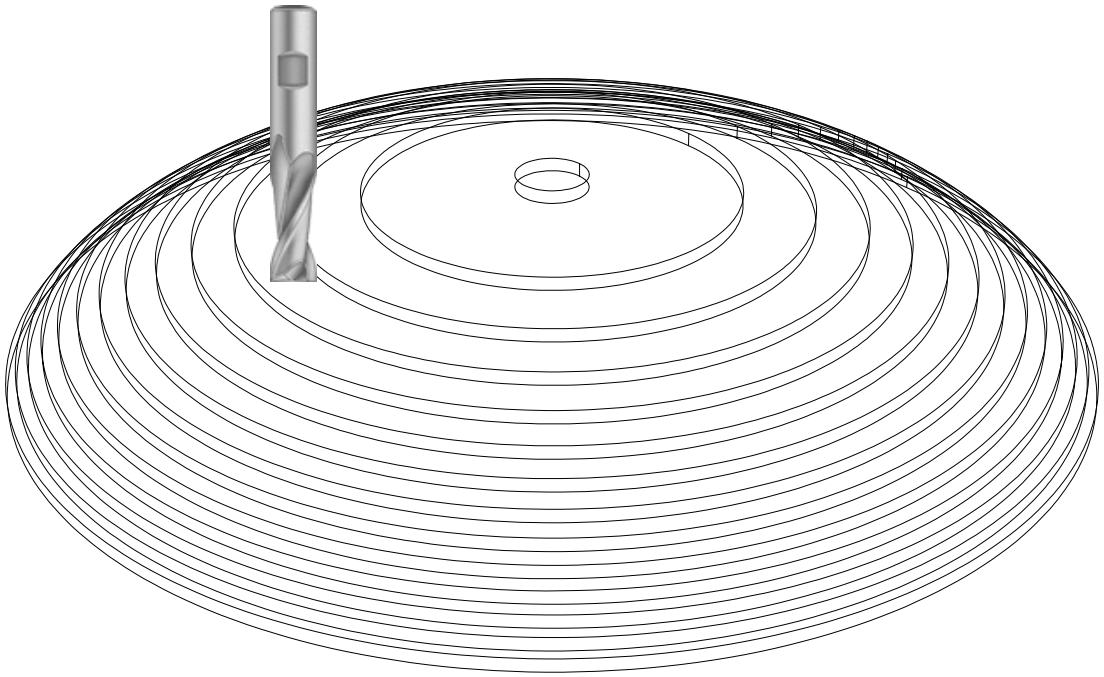
Under the Simulate Tab run the tool path simulation to make sure there are no collisions and that the tooling paths generated are the desired ones.

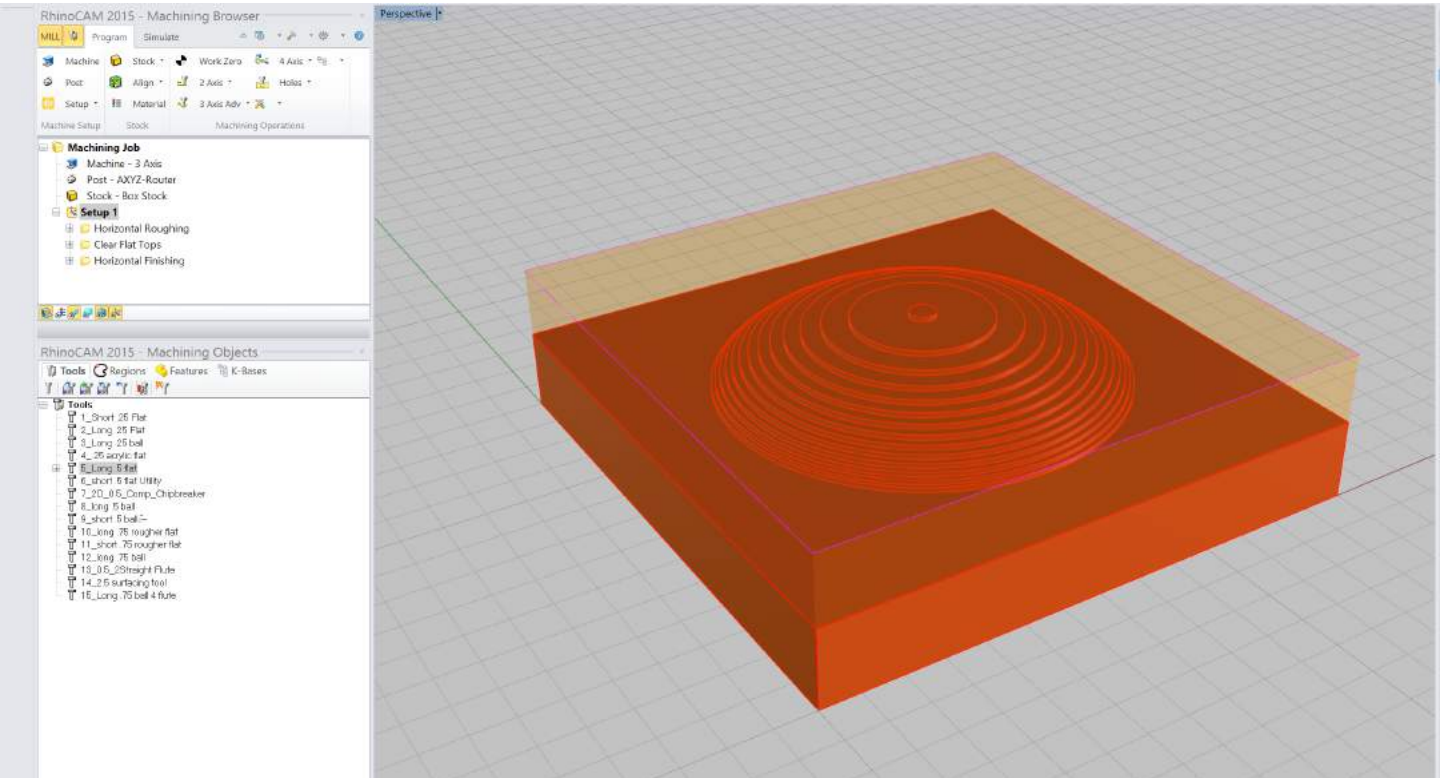


Once again you can turn off the tooling path visibility to see the simulated version of the final milled surface.

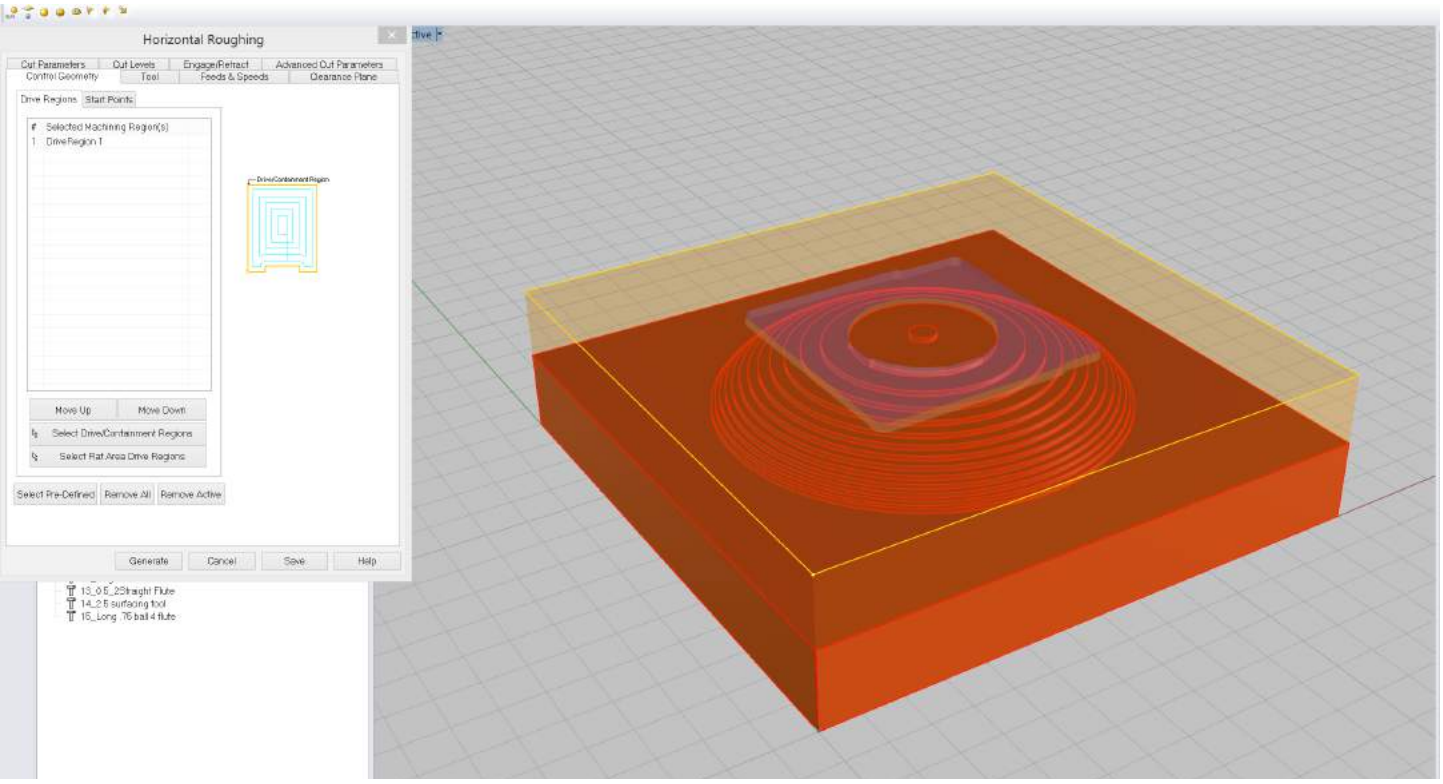
Rhino_CAM 3 Axis Machining Operations_ Horizontal Finishing

Horizontal Finishing enables the detection of vertical faces within a 3-Dimensional area, and allows the endmill to follow these faces as if they were curves (similar to a profile pass). This allows for stepped contour models to be programmed with a single function, rather than creating a machining operation for each individual contour.

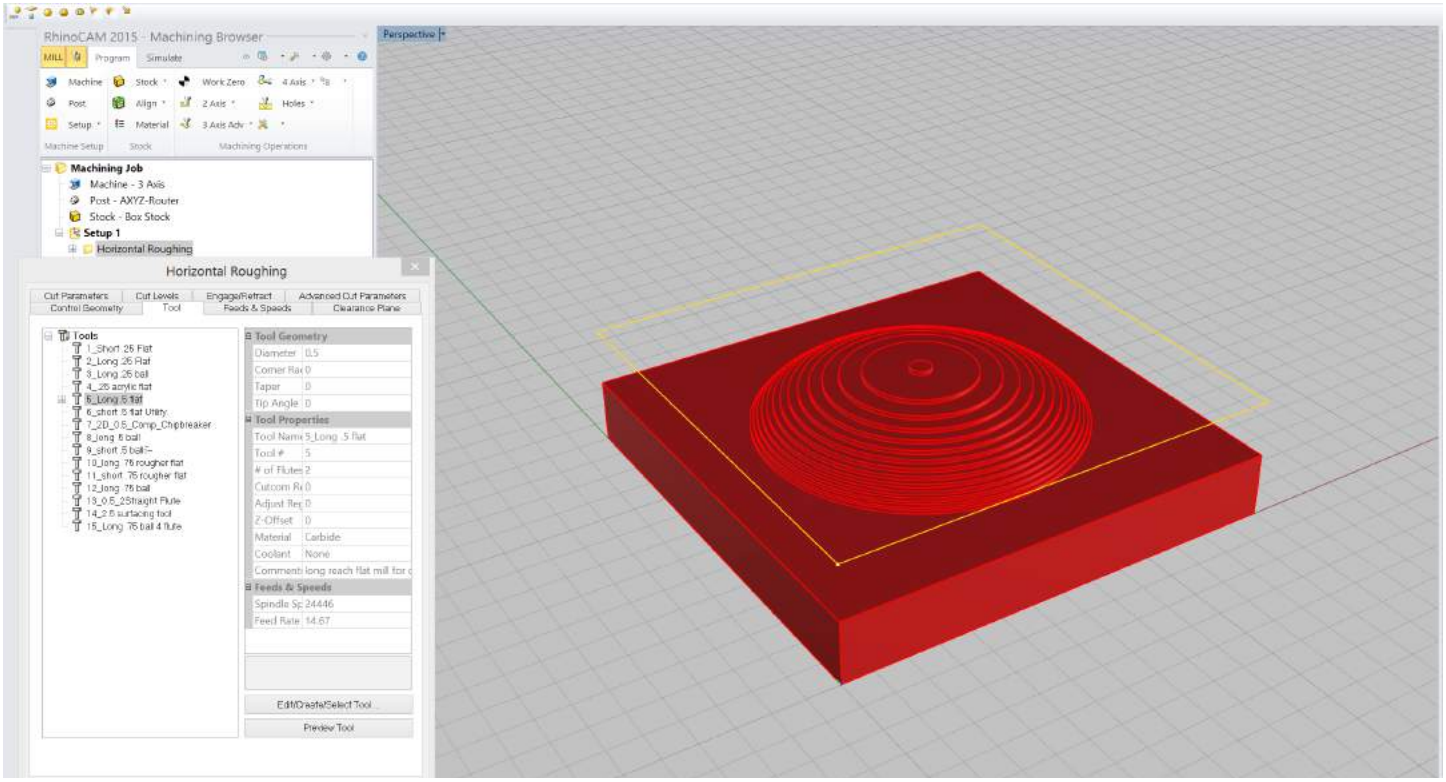




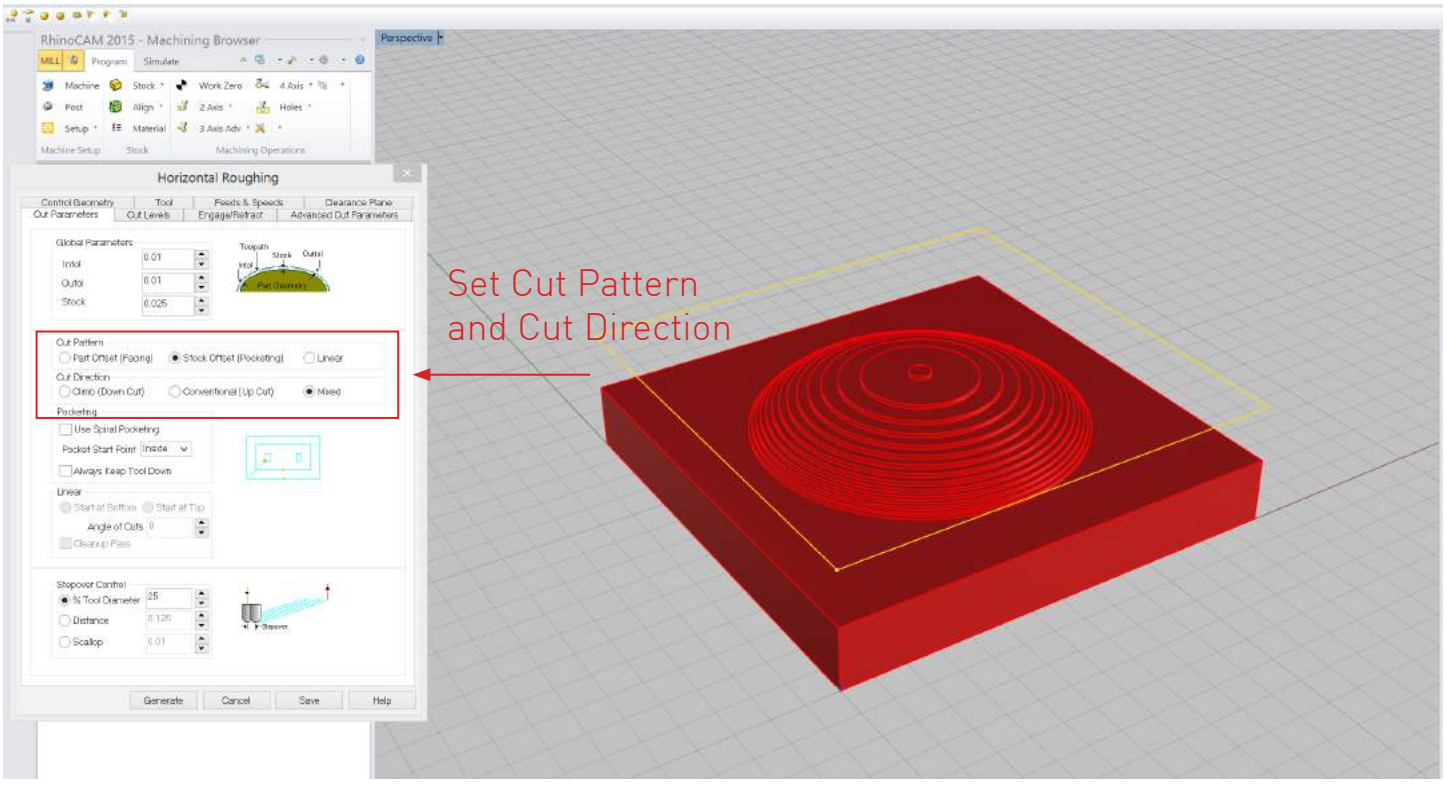
For Horizontal Finishing the same process will be repeated from the projection pocketing example. Begin with placing the poly object in the stock and then use Horizontal Roughing to remove the material before the next pass.



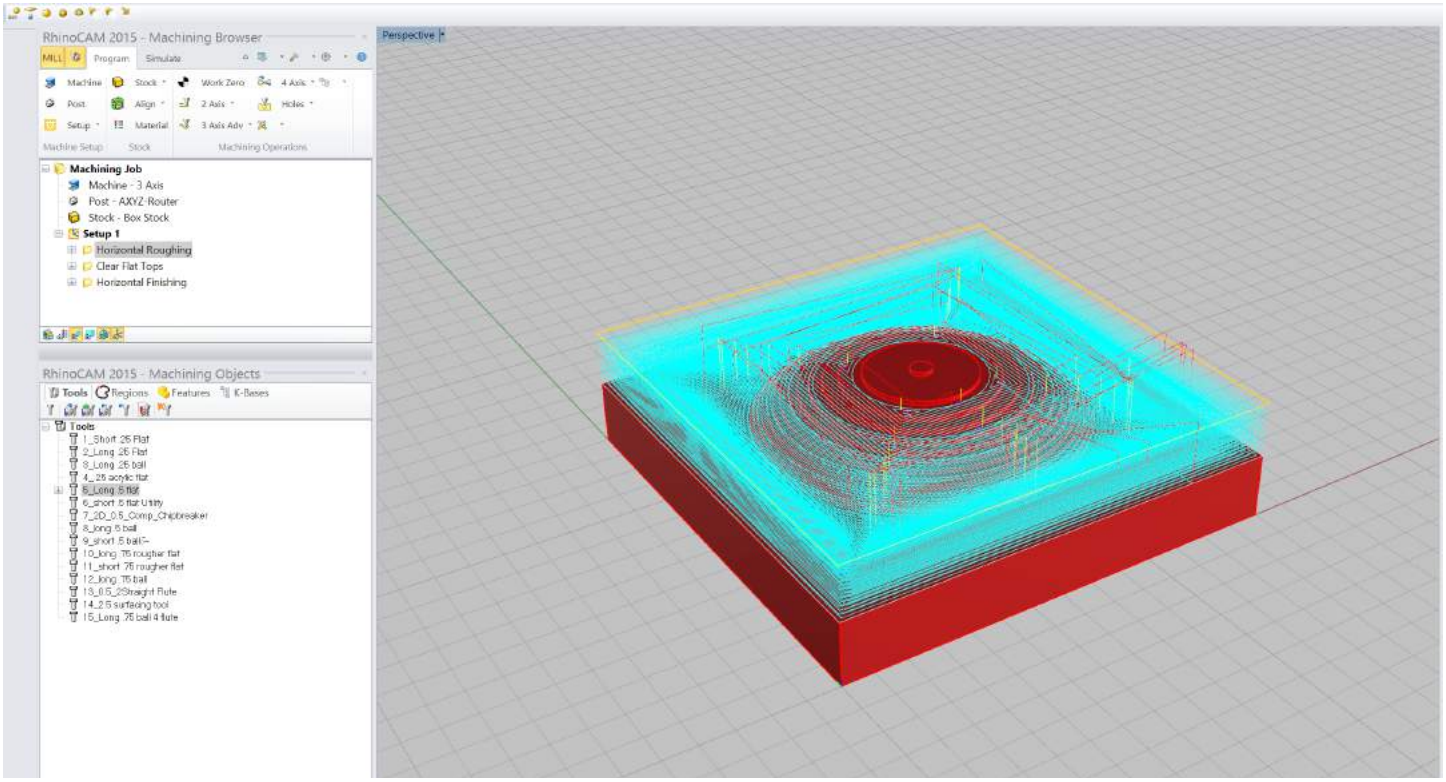
For the Horizontal Roughing create a drive region to contain the Horizontal Roughing pass. In this case the region is defined by the square profile of our stock material.



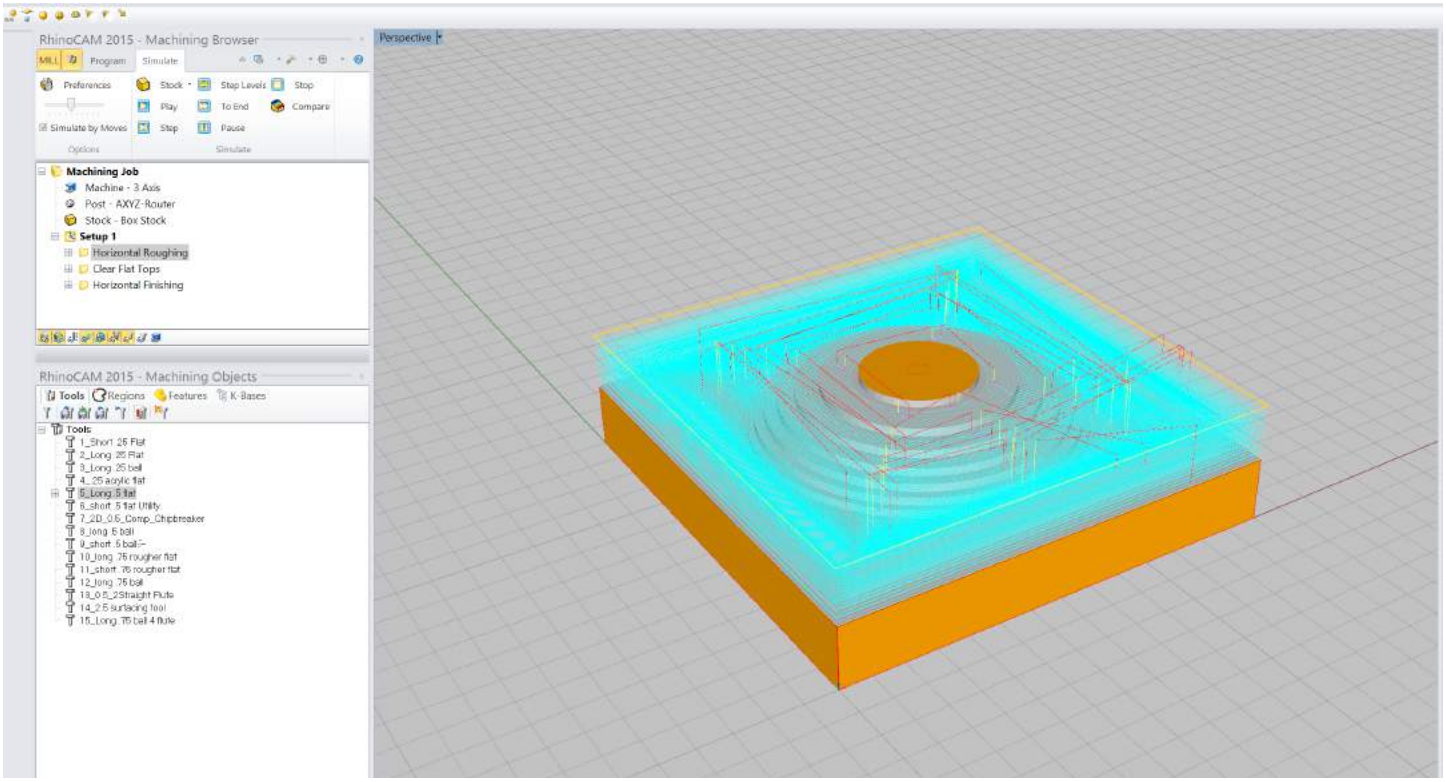
Under Tool for the Roughing pass select the .5" long Flat endmill.

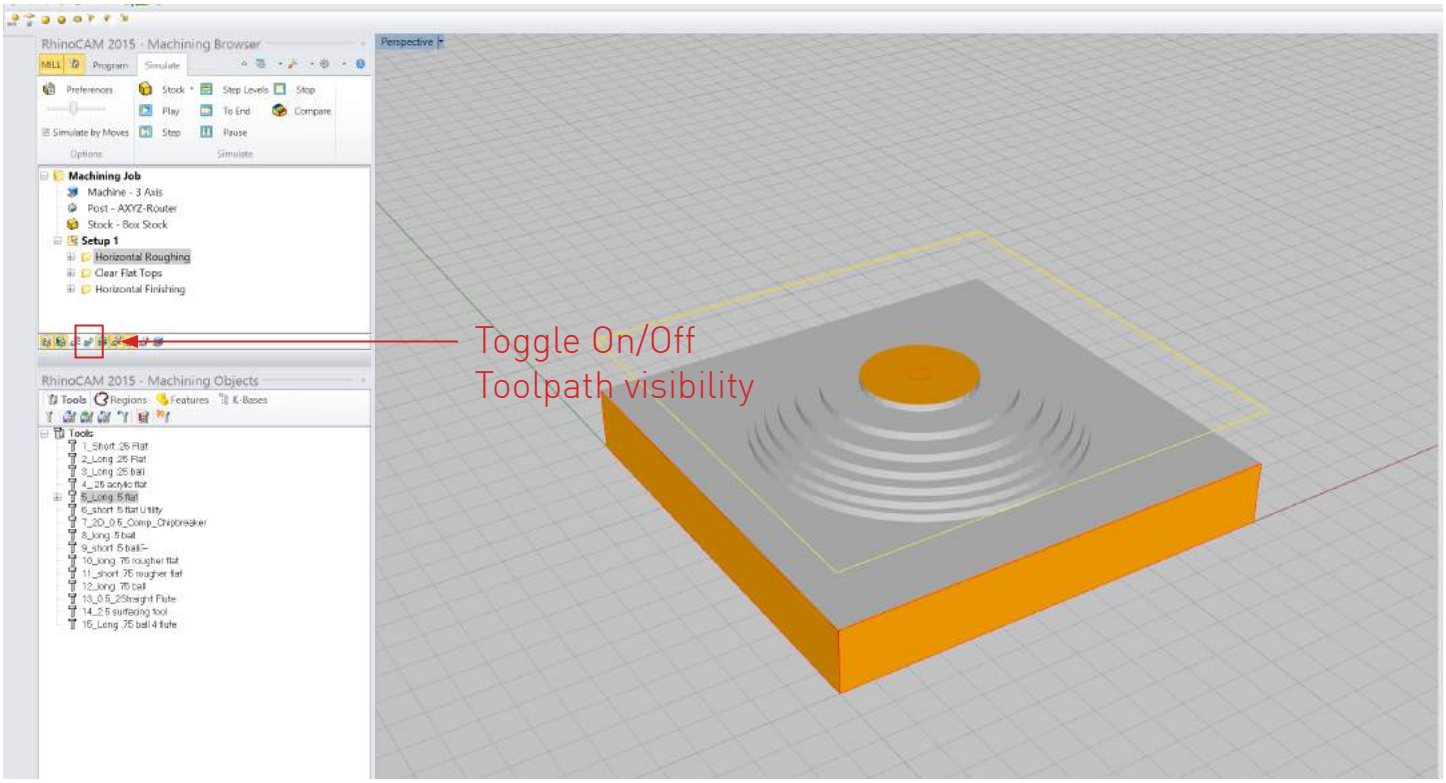


Under the Cut Parameters set the Cut Pattern to Part Offset and the Cut Direction to mixed. The tool diameter is set to 25% in this example, for less dense material it can be set higher to speed up the Roughing Pass.

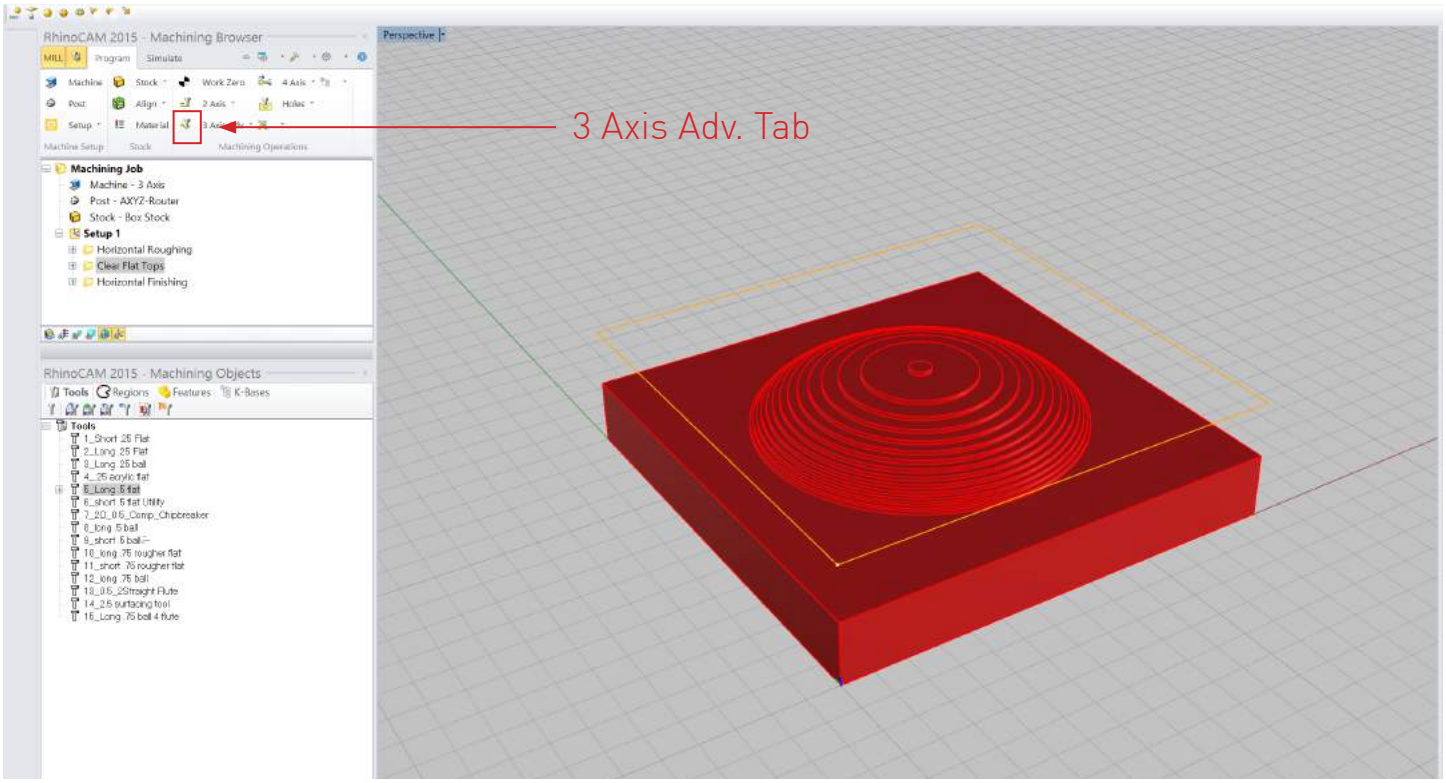


Once all the parameters have been set up press generate to create the milling paths.

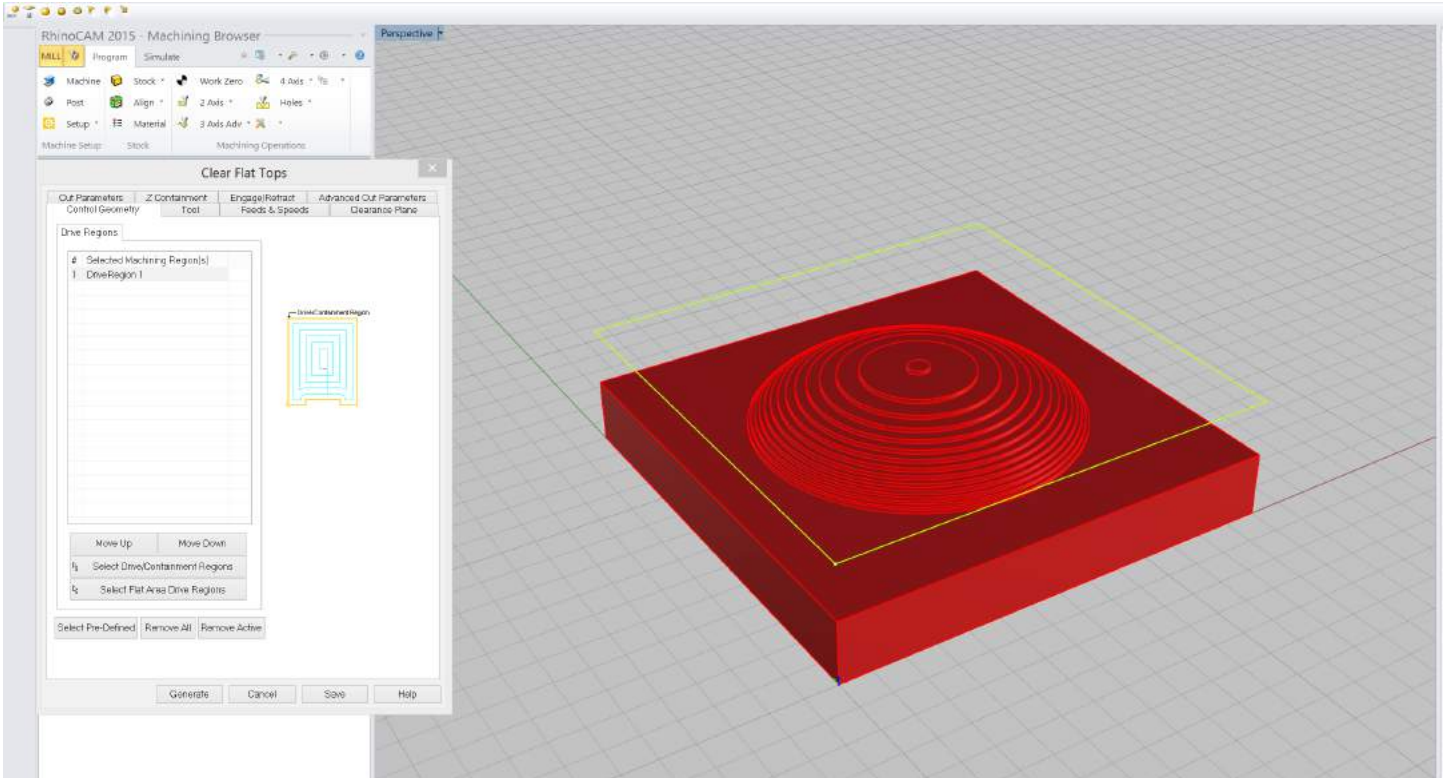




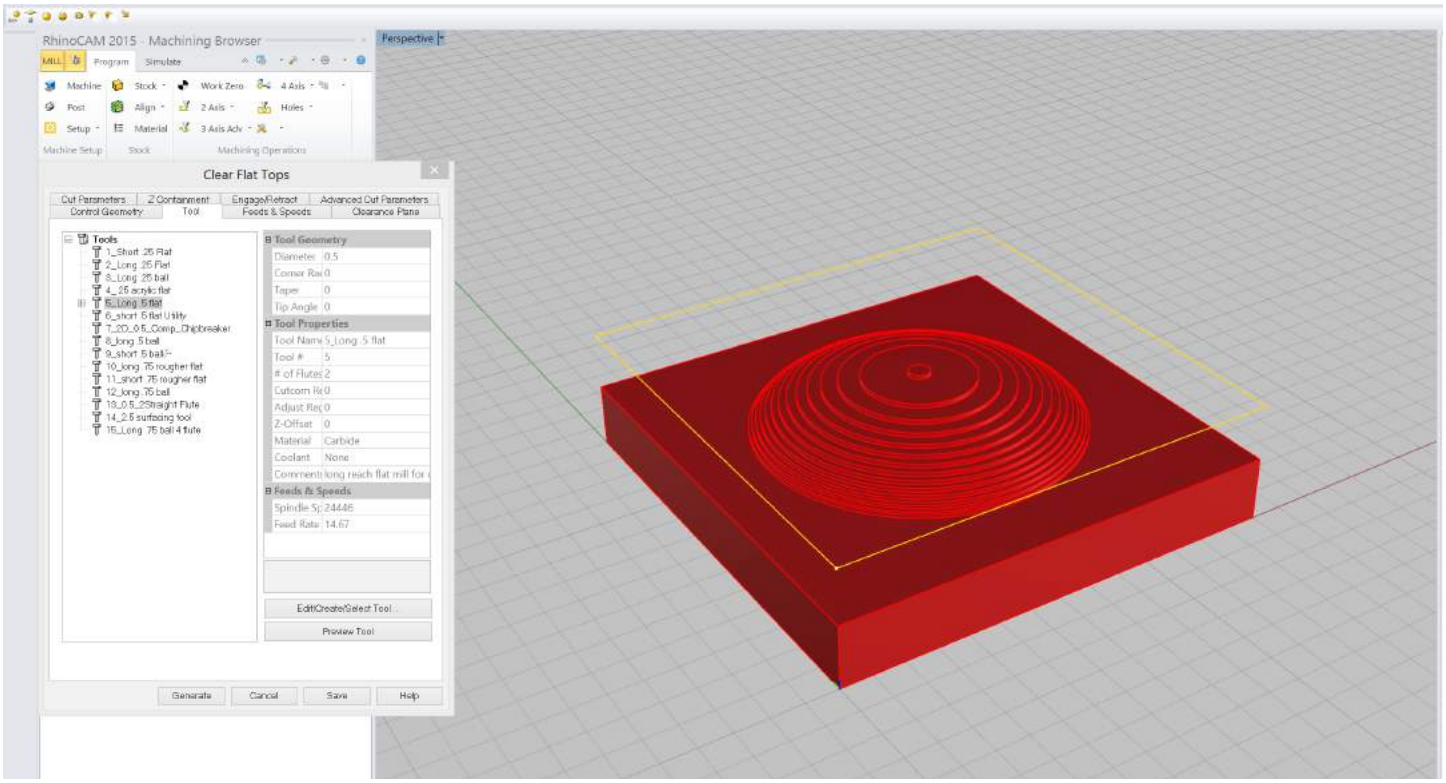
Toggle off the tooling paths to view the Horizontal Roughing path surface.



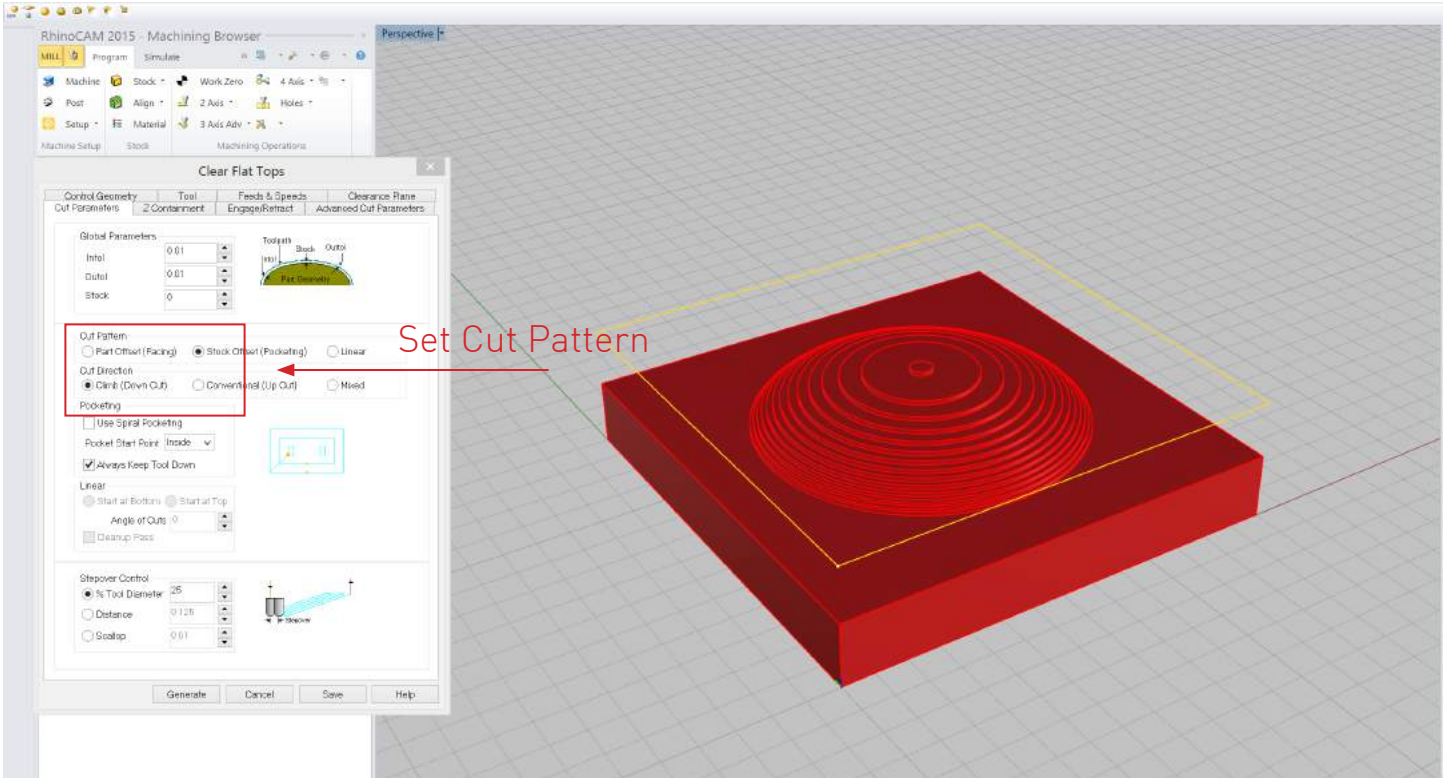
Under the 3 Axis Adv select the Clear Flats tab. This operation is useful if the horizontal distance between your contours is larger than the diameter of the endmill, and will clear excess material before your final finishing pass.



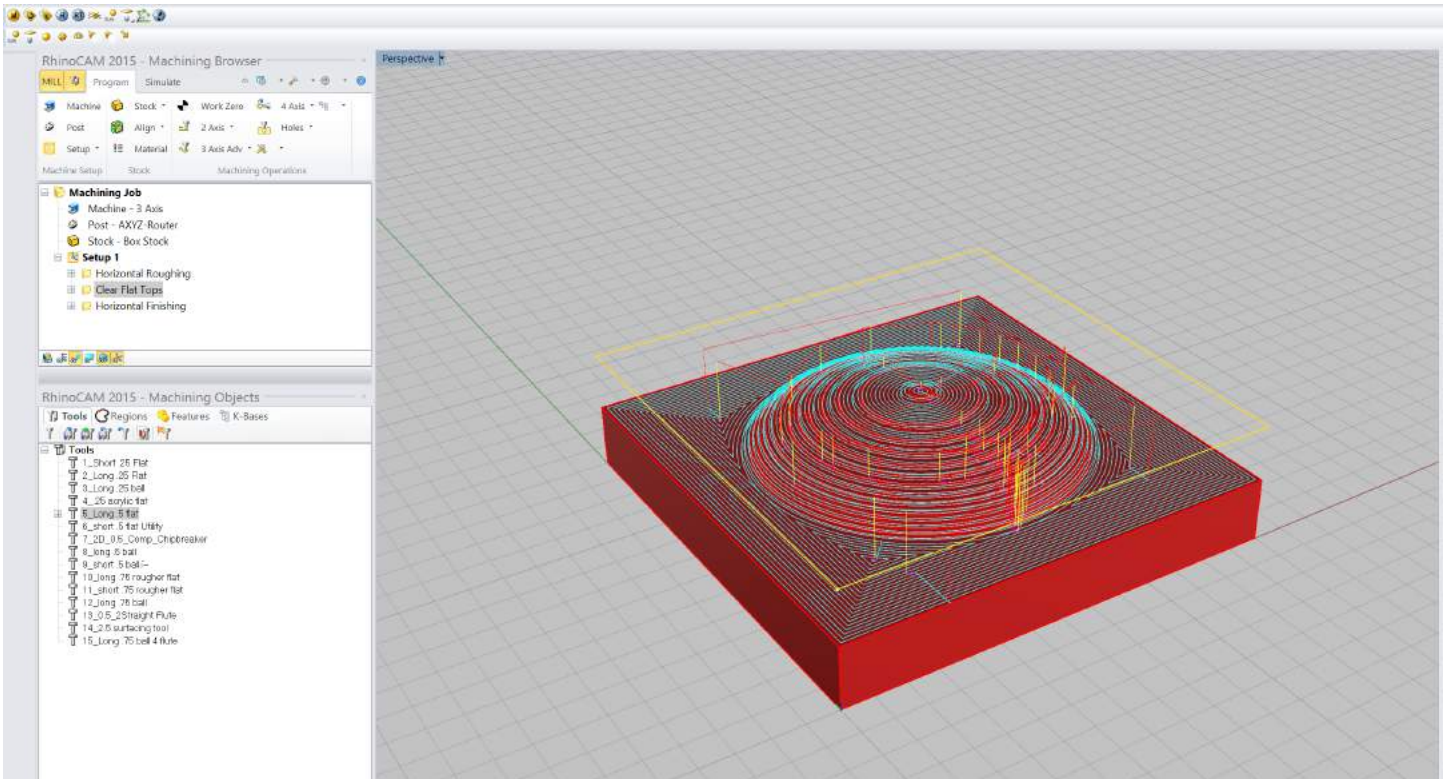
Under Machining Features/Regions select the same drive region from the Horizontal Roughing Pass.



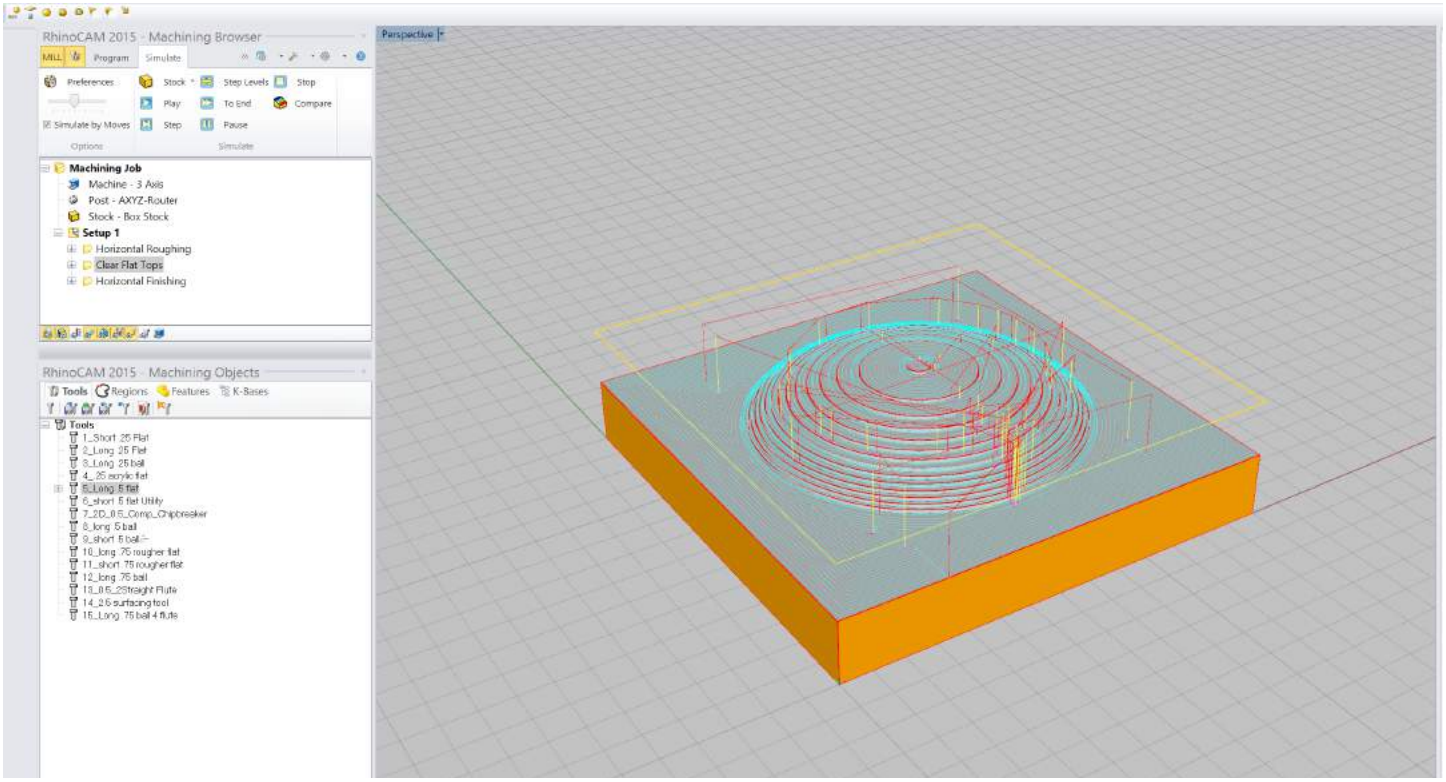
Under the Tool tab select the endmill for the finished surface, the long .5'' flat is used in this example.



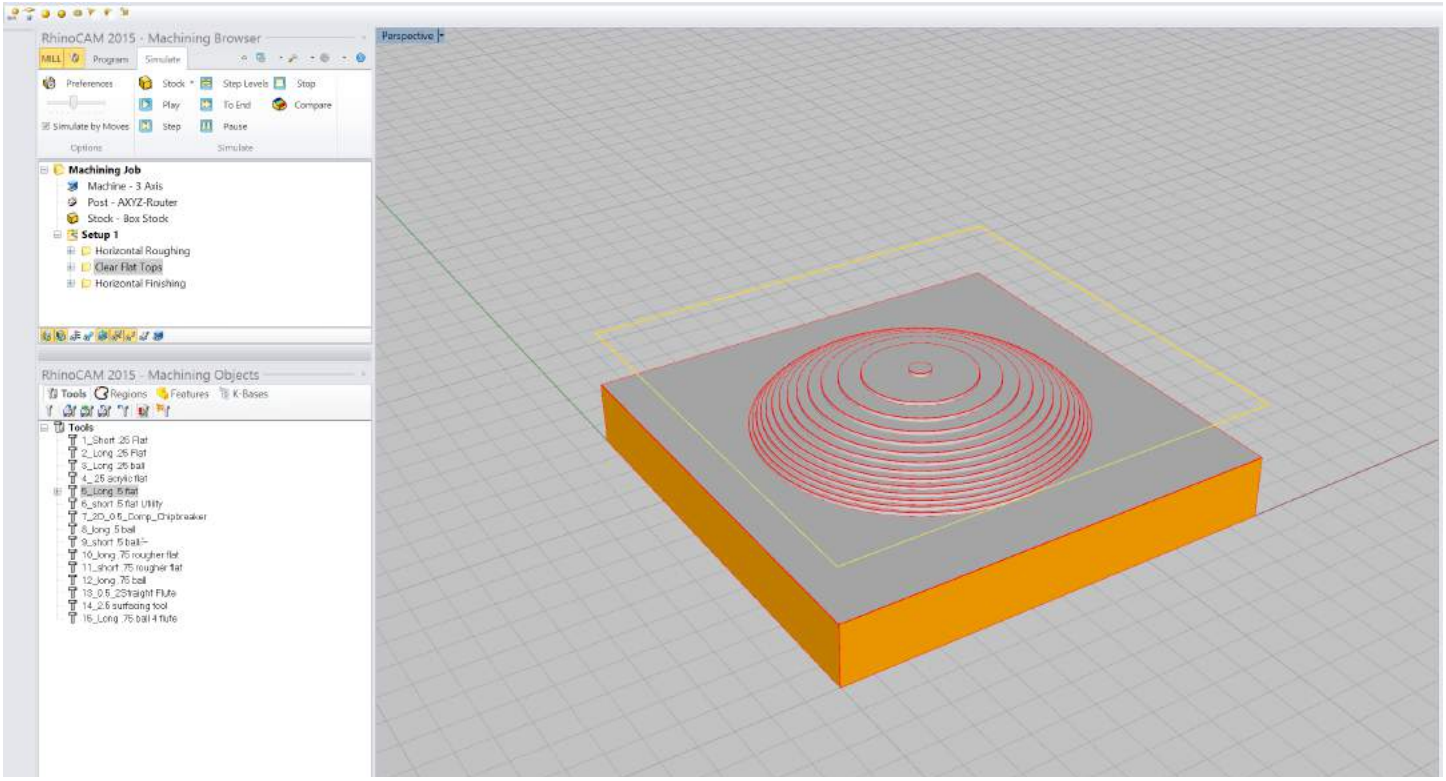
Under Cut Parameters The Cut Direction is set to Climb and the Stepover Control is set to 25%.



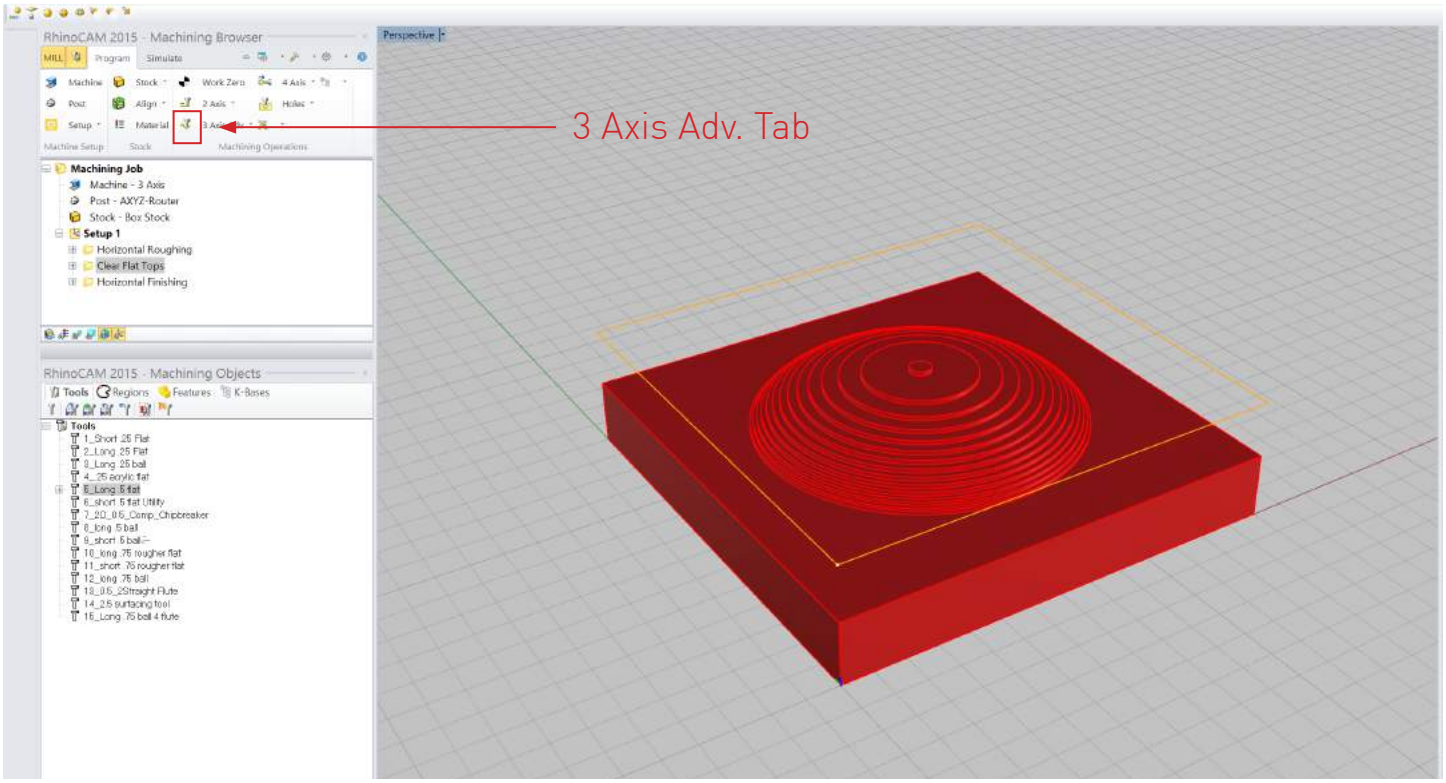
Once all the parameters have been set up press generate to create the milling paths.



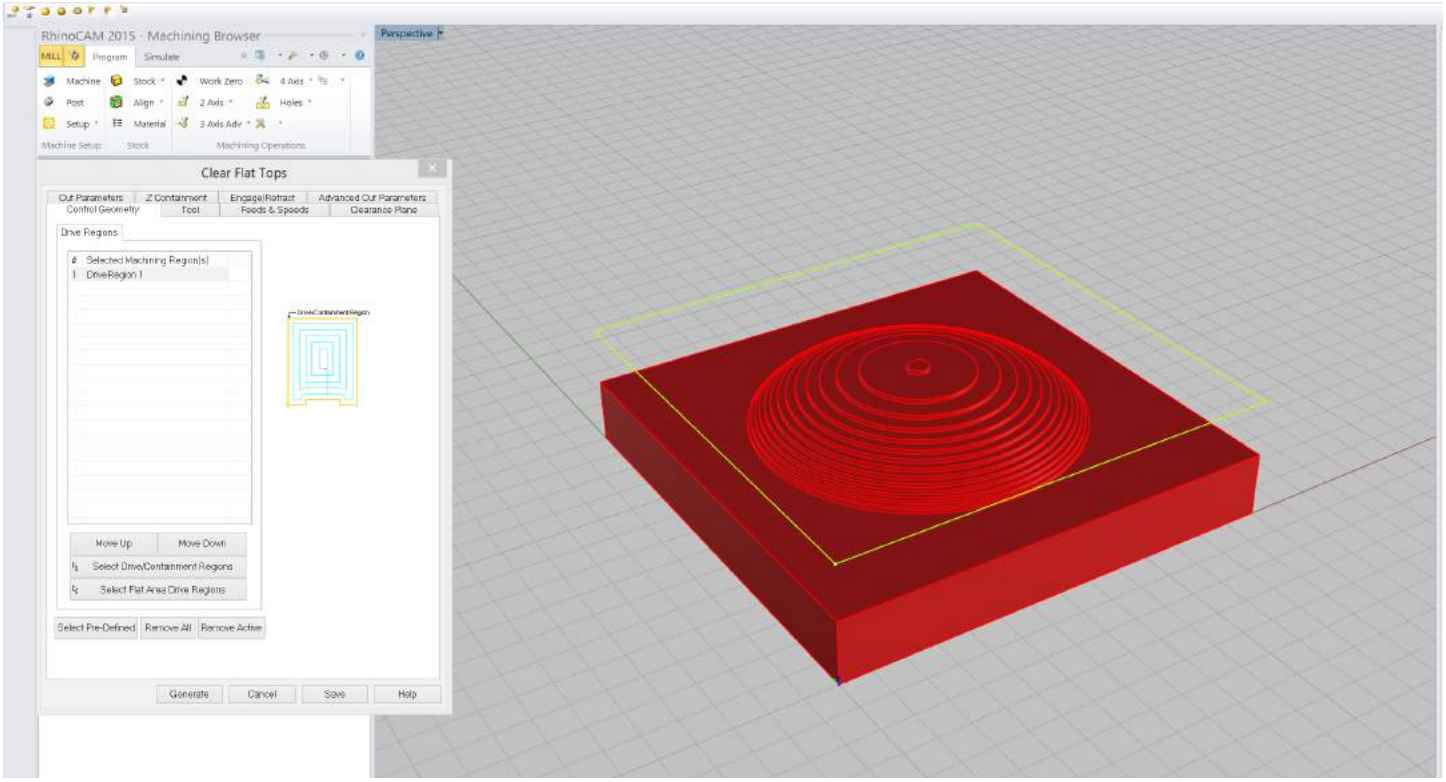
Under the Simulate Tab run the tool path simulation to make sure there are no collisions and that the tooling paths generated are the desired ones.



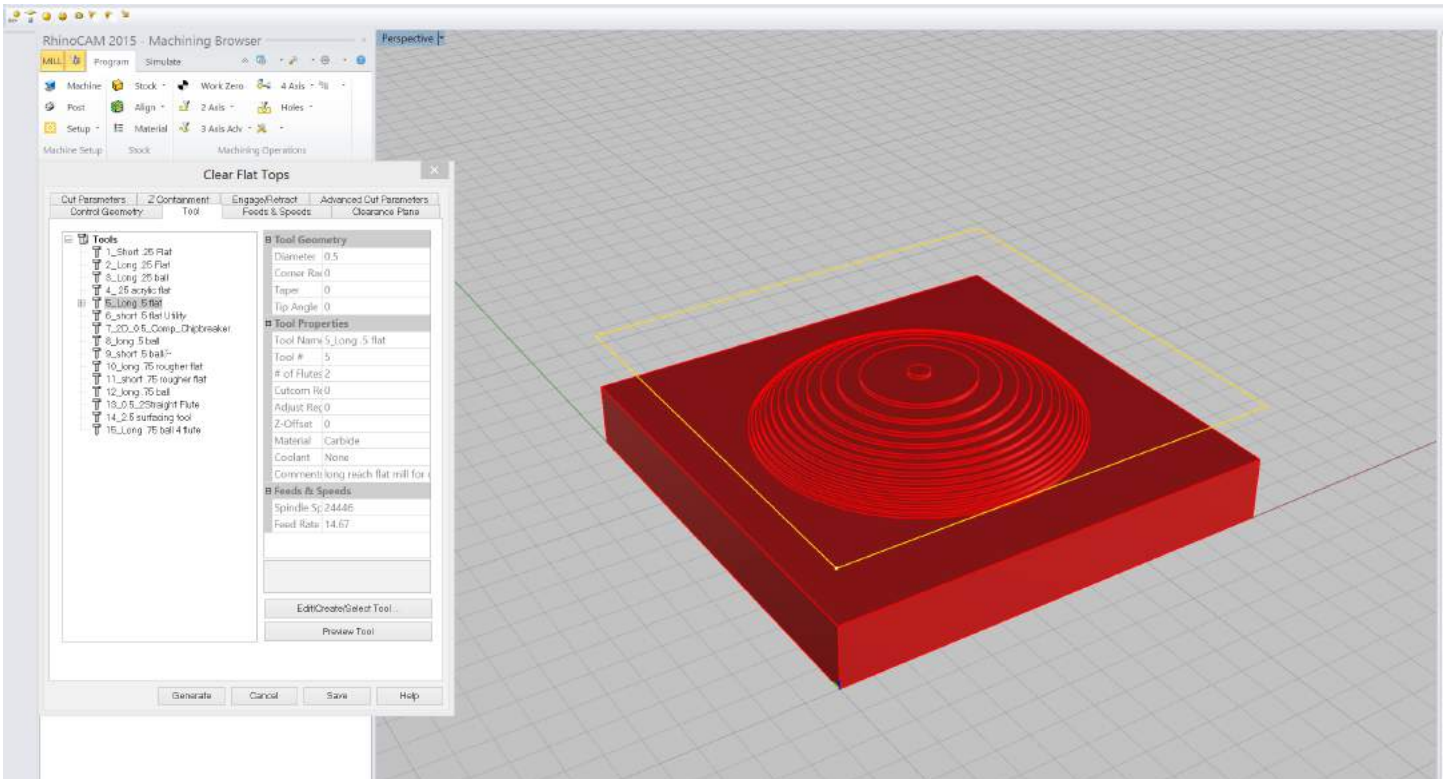
Once again you can turn off the tooling path visibility to see the simulated version of the final milled surface.



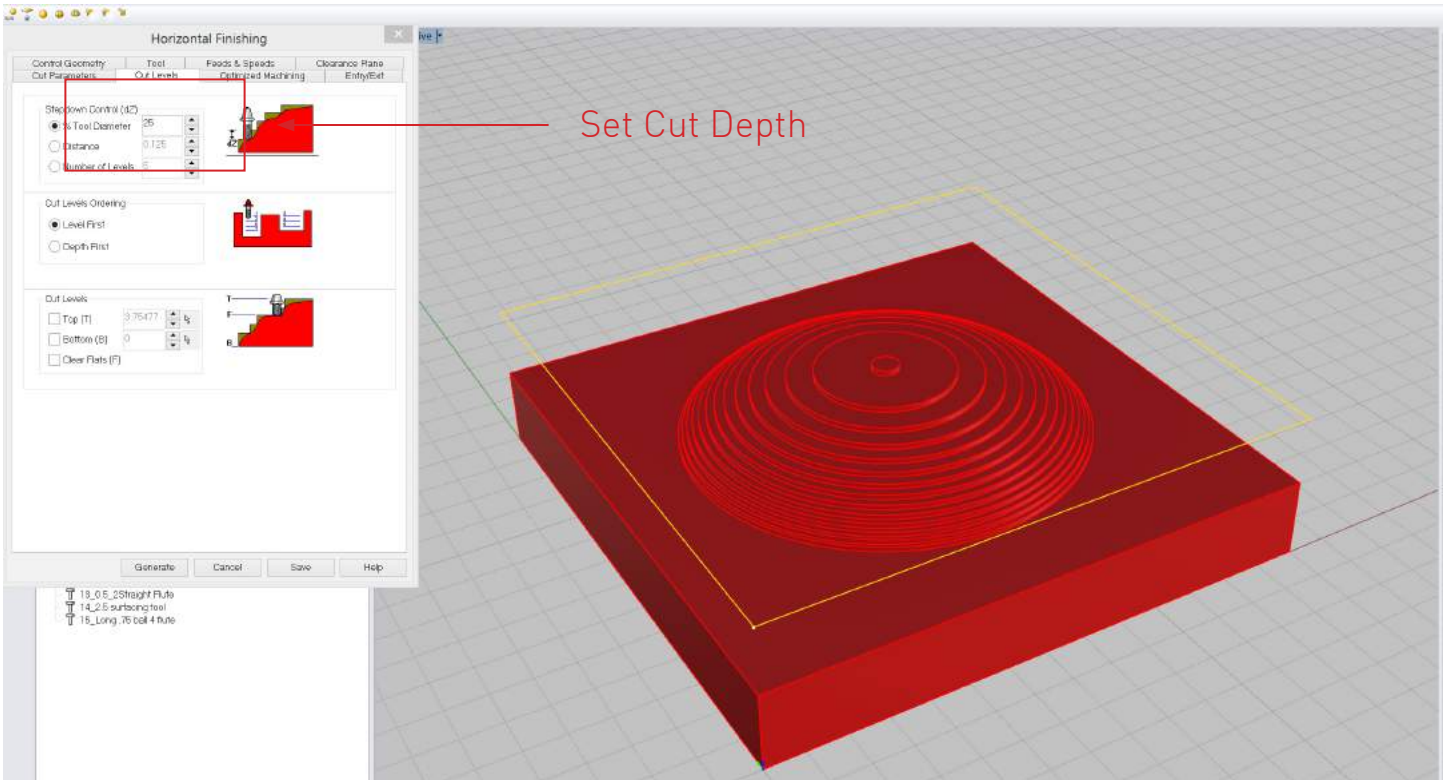
Under the 3 Axis Adv select the Horizontal Finishing tab. This operation will detect the vertical faces in your model and treat them as if they were profile curves. This should give you a nice clean finished contour edge.



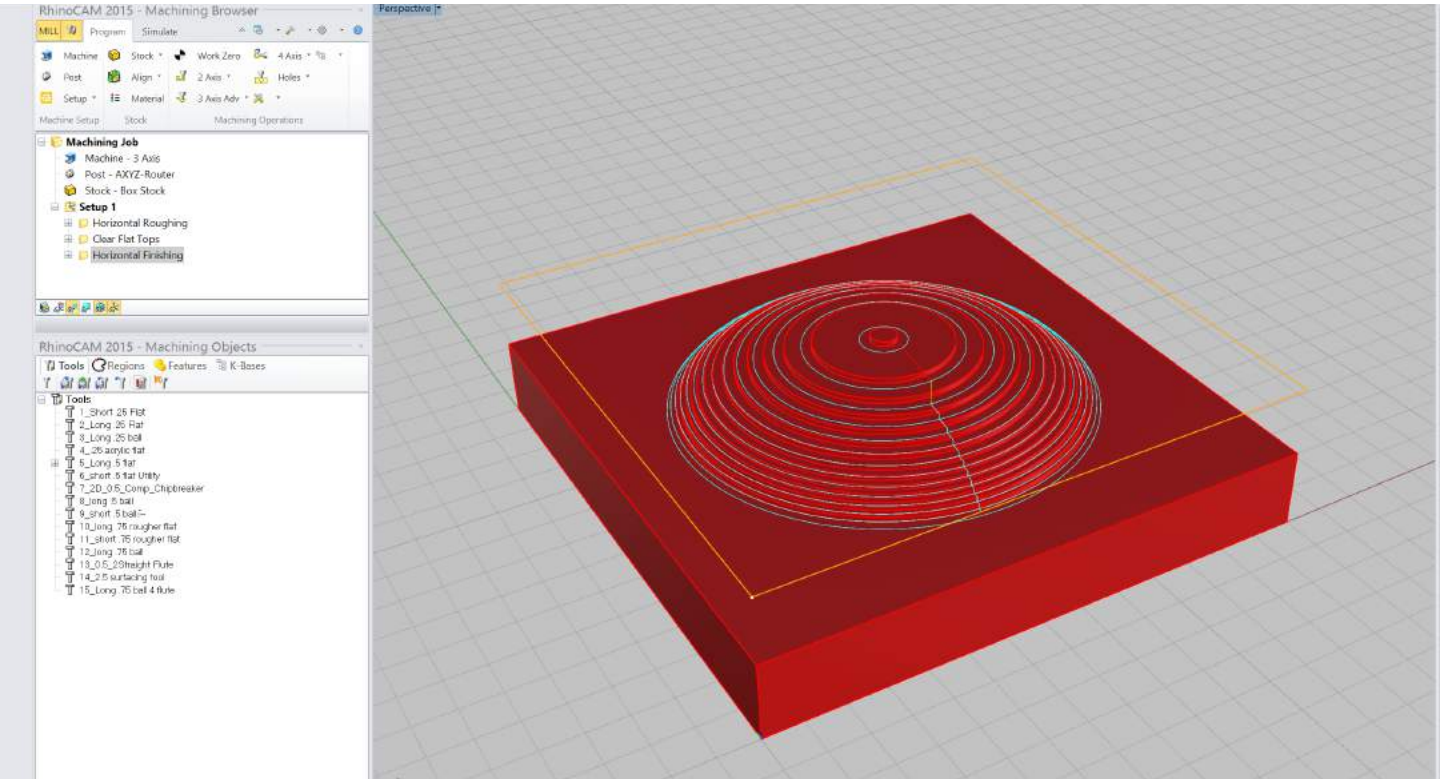
Under Machining Features/Regions select the same drive region from the Horizontal Roughing Pass.



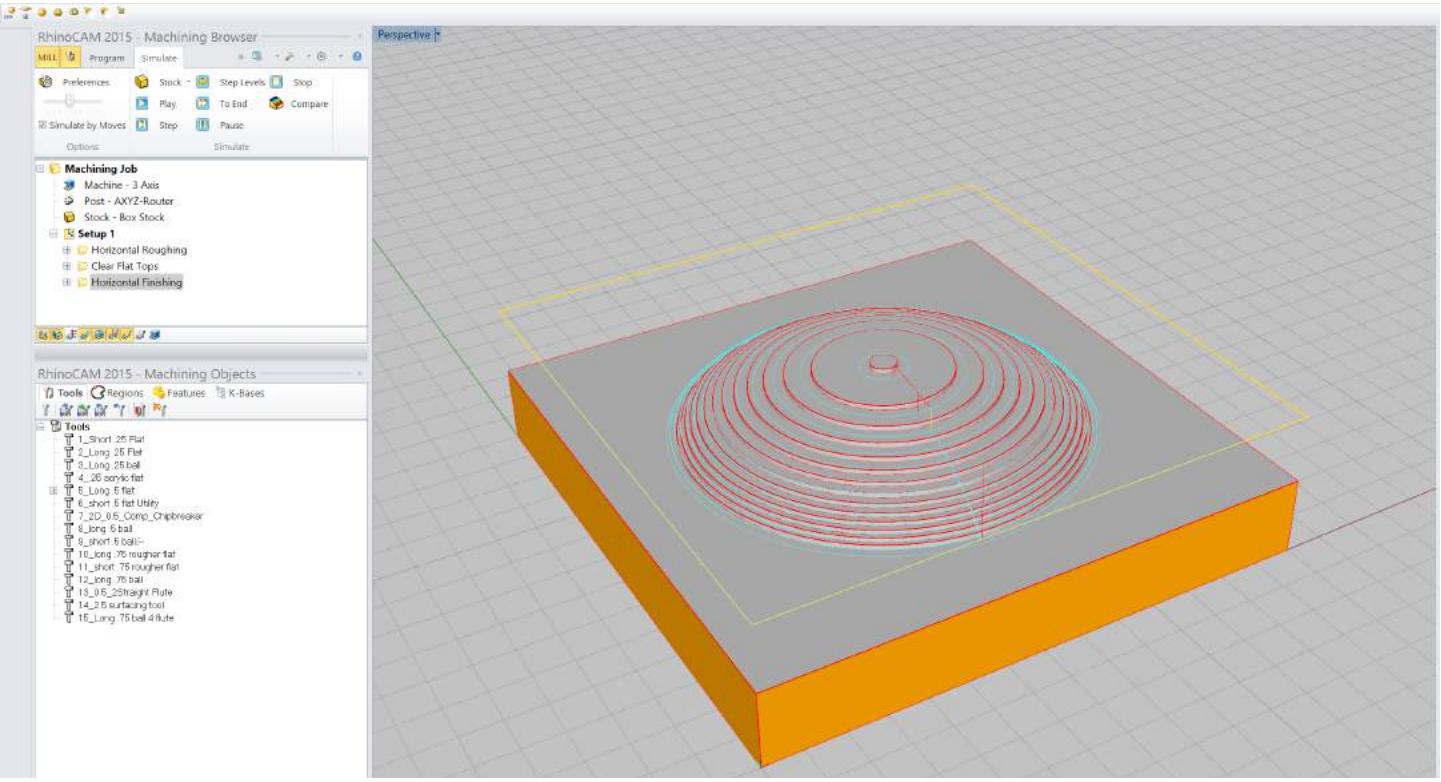
Under the Tool tab select the endmill for the finished surface, the long .5'' flat is used in this example.



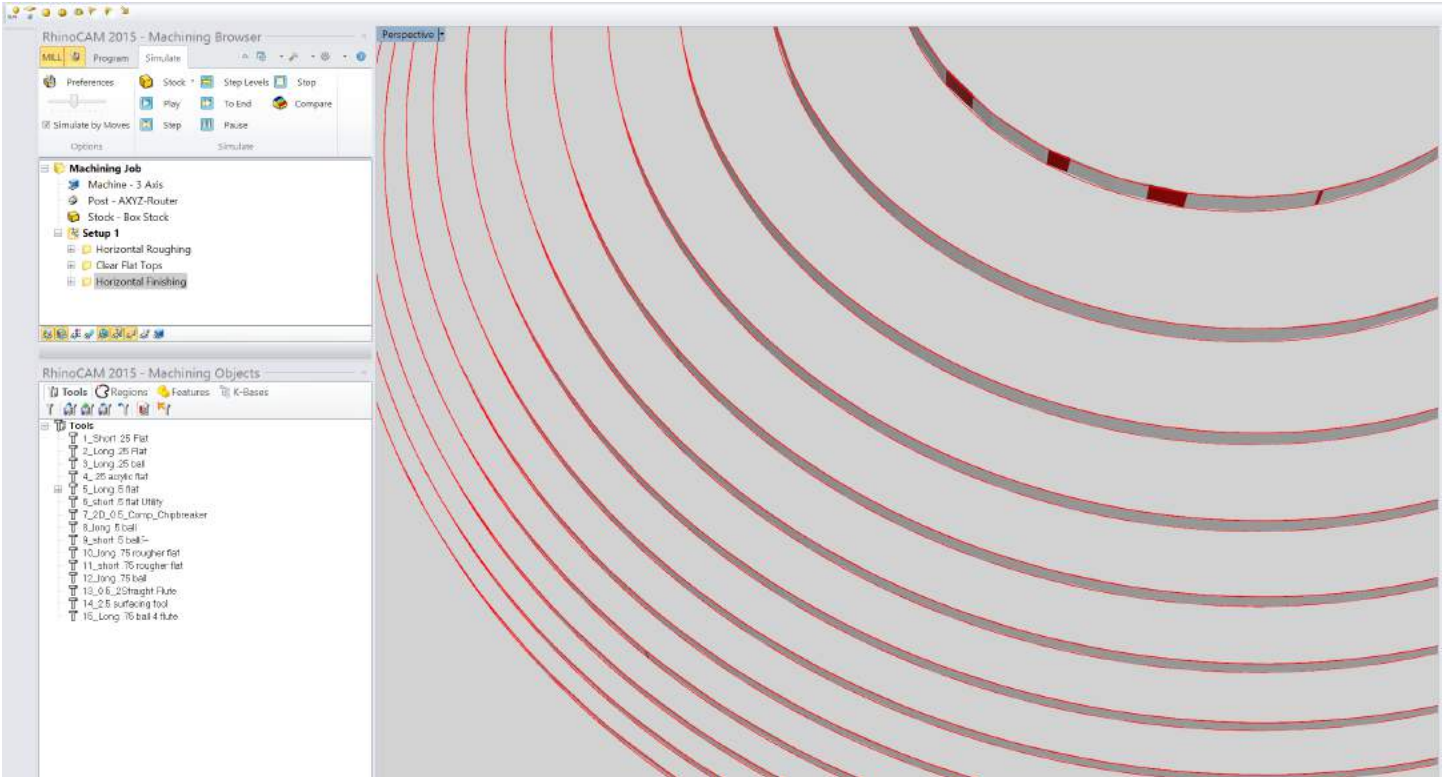
Under Cut Levels, set the cut depth to match the vertical spacing between the contours in your model. In this example, each contour is 1/8", so the cut level is 25% of the tool diameter. 25% of the 1/2" endmill = 1/8", so we will catch every contour in our model. If the cut level does not match your contour heights, the software may skip levels.



Once all the parameters have been set up press generate to create the milling paths.



Under the Simulate Tab run the tool path simulation to make sure there are no collisions and that the tooling paths generated are the desired ones.



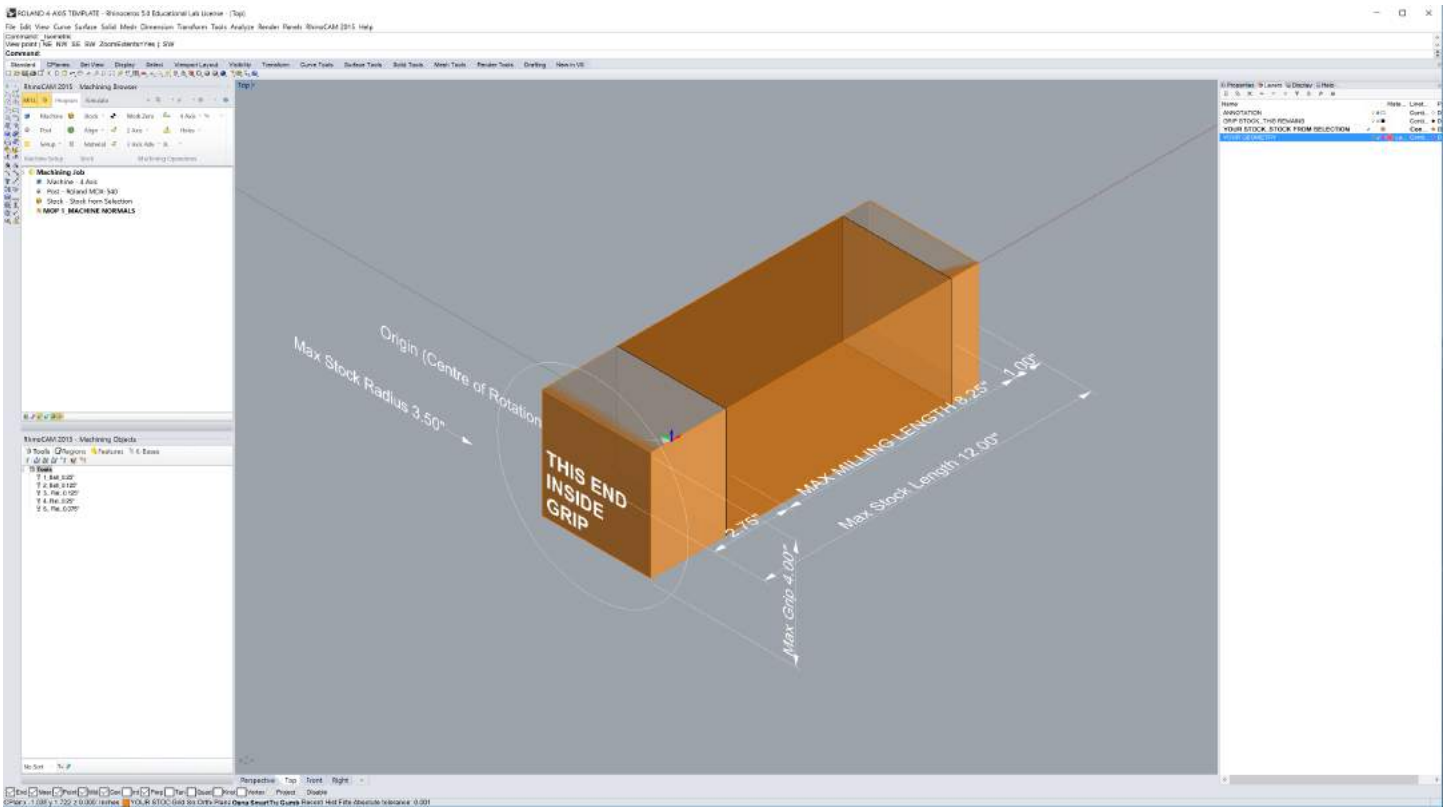
Once again you can turn off the tooling path visibility to see the simulated version of the final milled surface. You can see that the simulated results (grey) match our desired part (red).

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 endmill to gain acess 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.



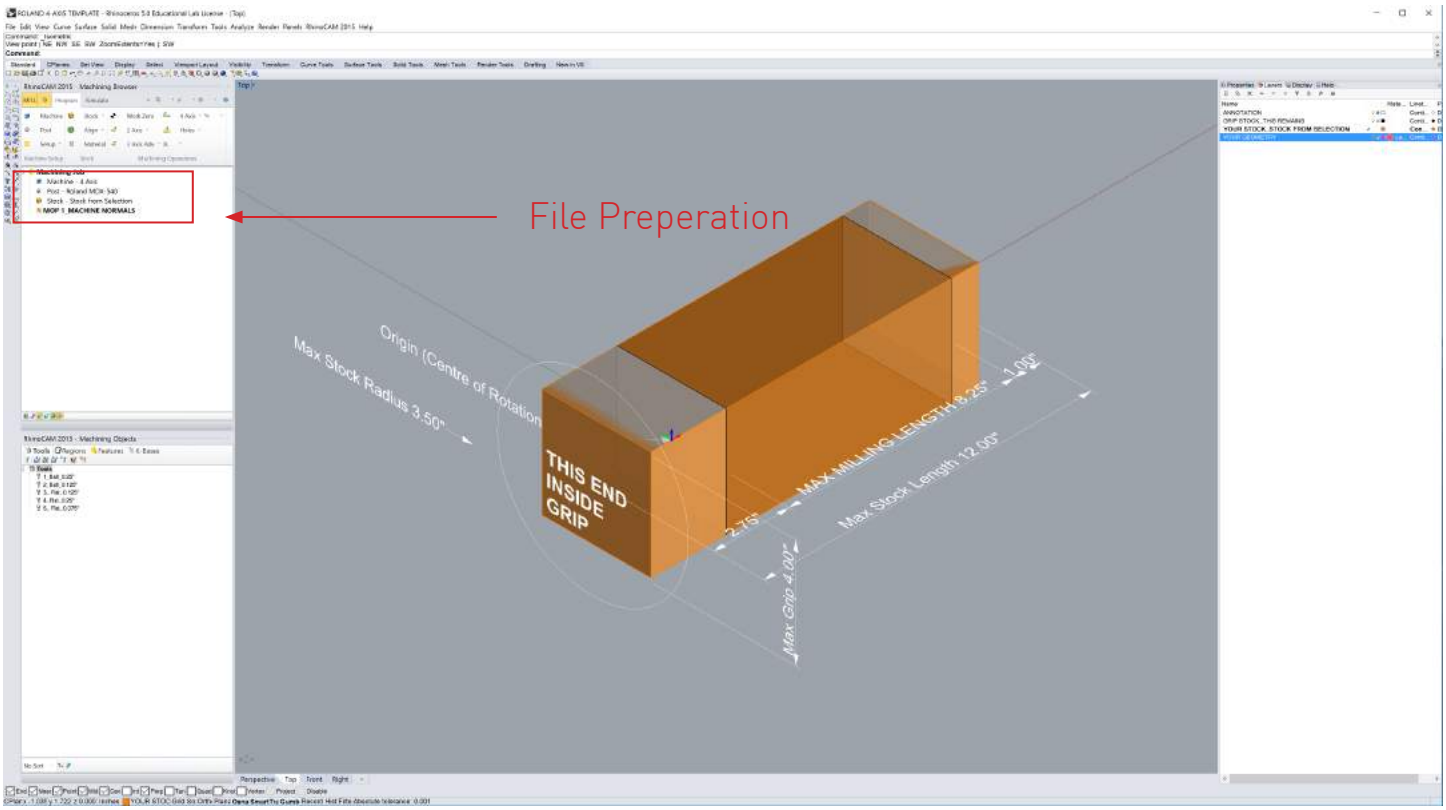
A template file for the Roland MDX 540 can be found on the ftp under Groups < CNC Milling < Submissions < ROLAND MDX 540. Please copy the template file and the tool library directly onto your computer drive.

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 endmill cannot travel here.

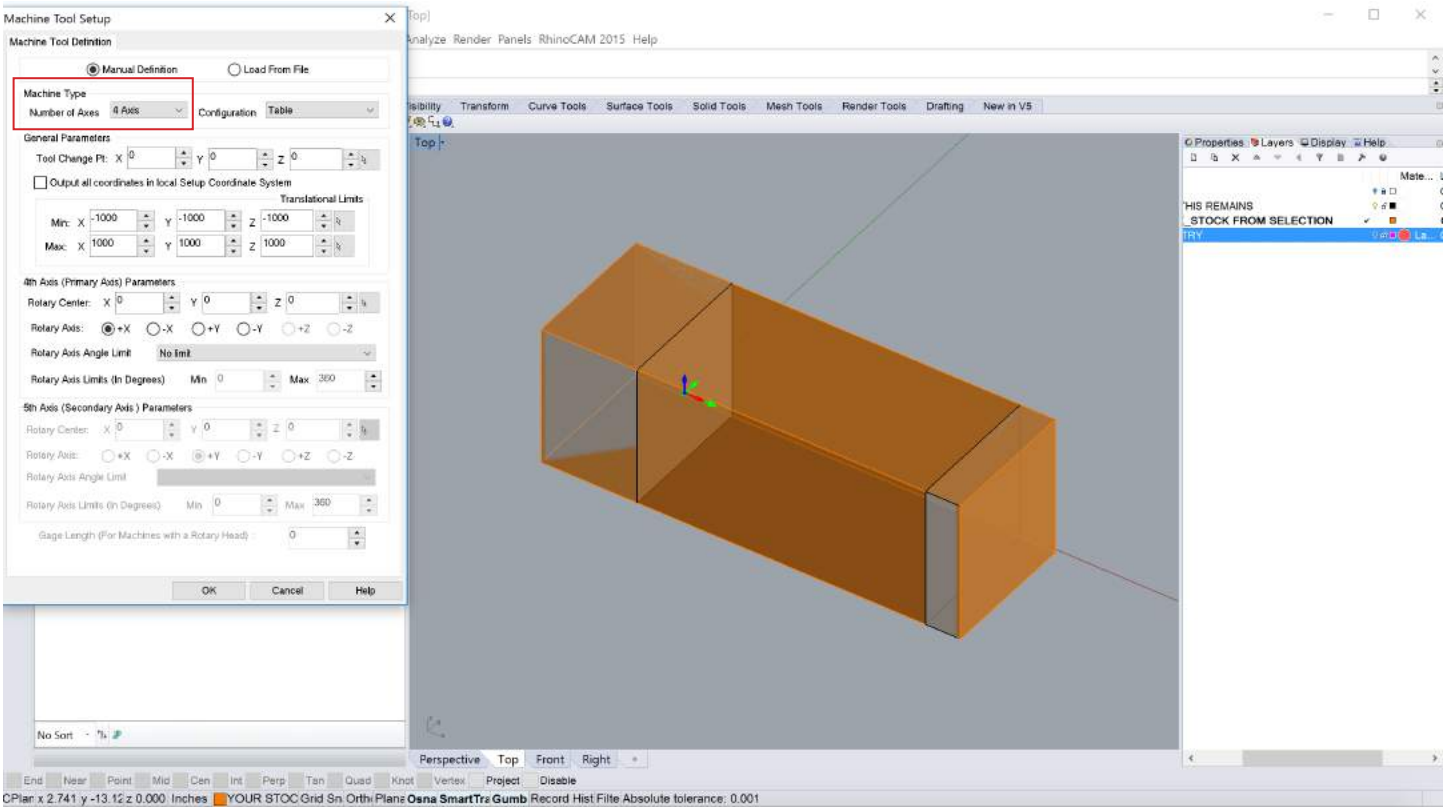
The template file has annotations to guide you. It is important to note these key dimensions:

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

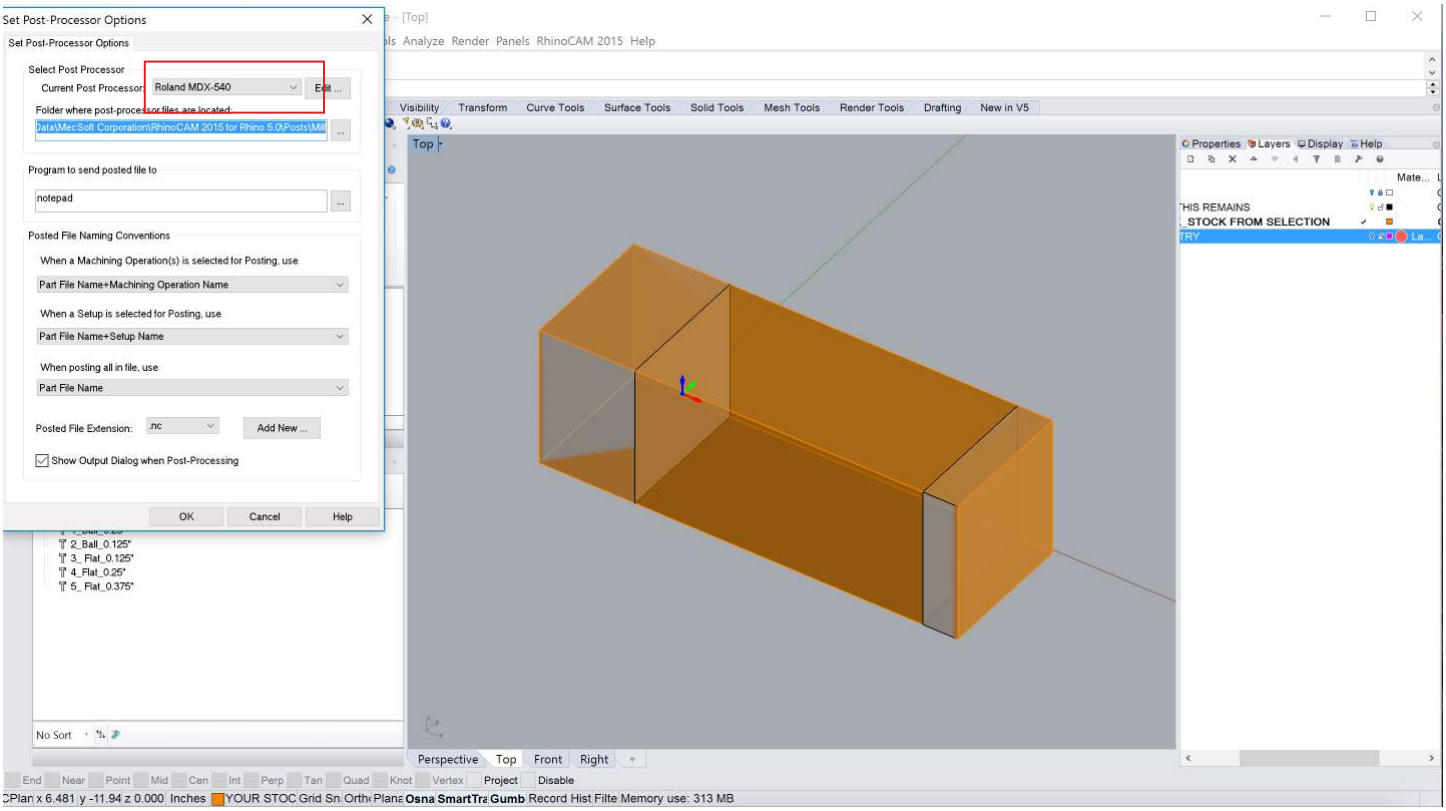
All stock must fit within a 7" diameter rotation***



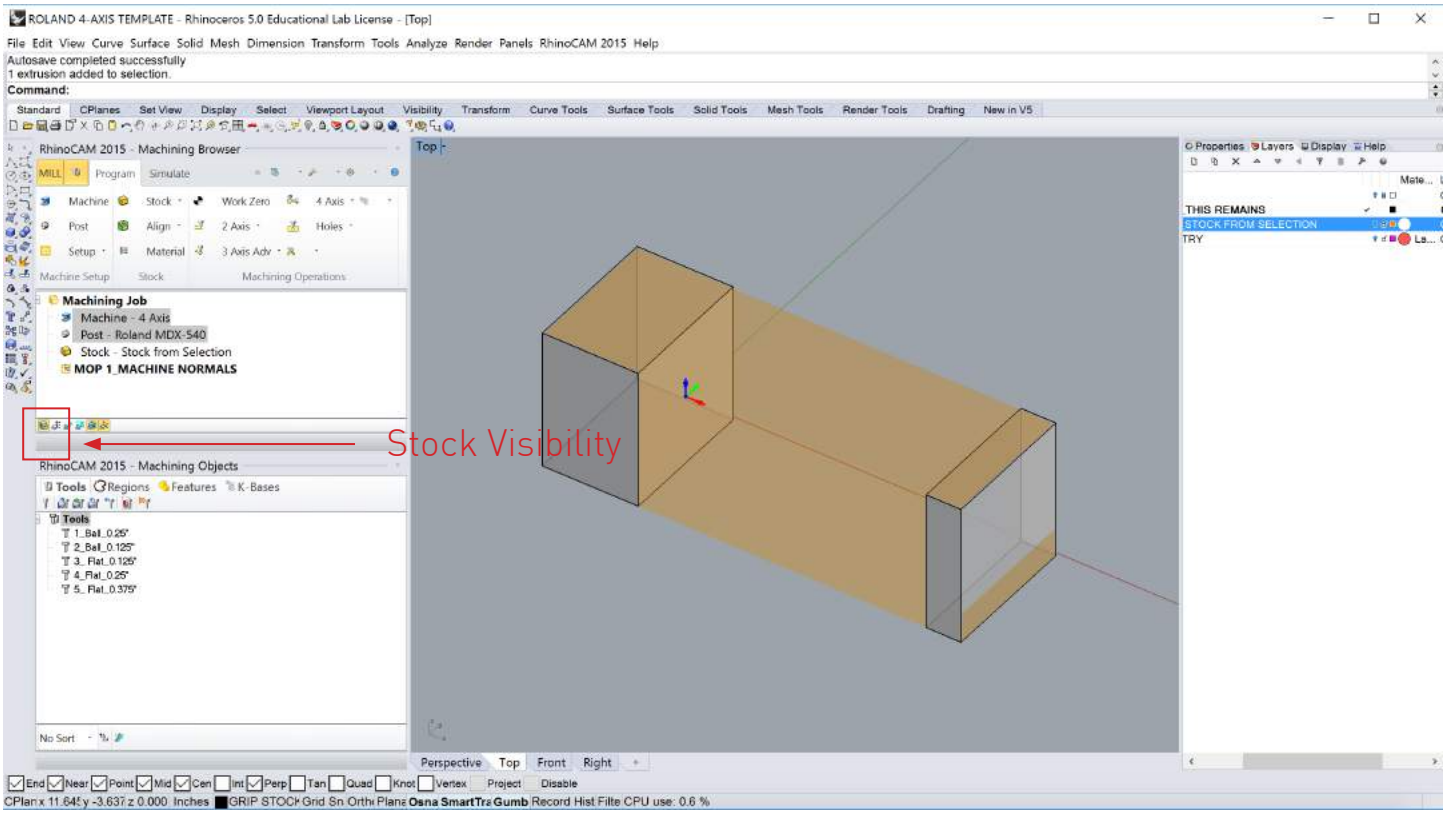
File Preparation : here, you will change the Machine from 3 axis to 4 axis, and the Post Processor to the Roland MDX 540. You will also set your 'stock from selection' as you would in any other 2 or 3 axis milling.



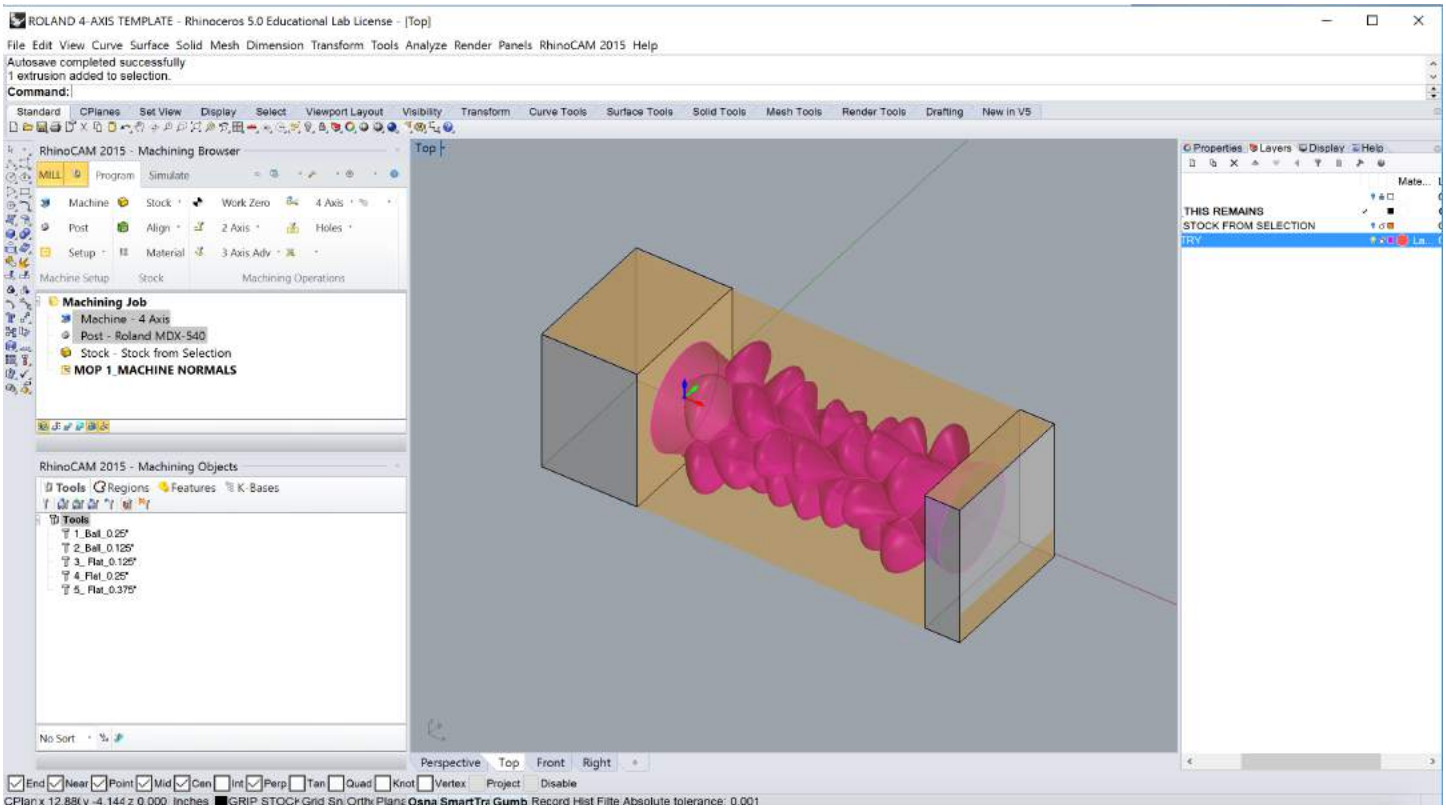
Right click on 'Machine', change # of axis to 4



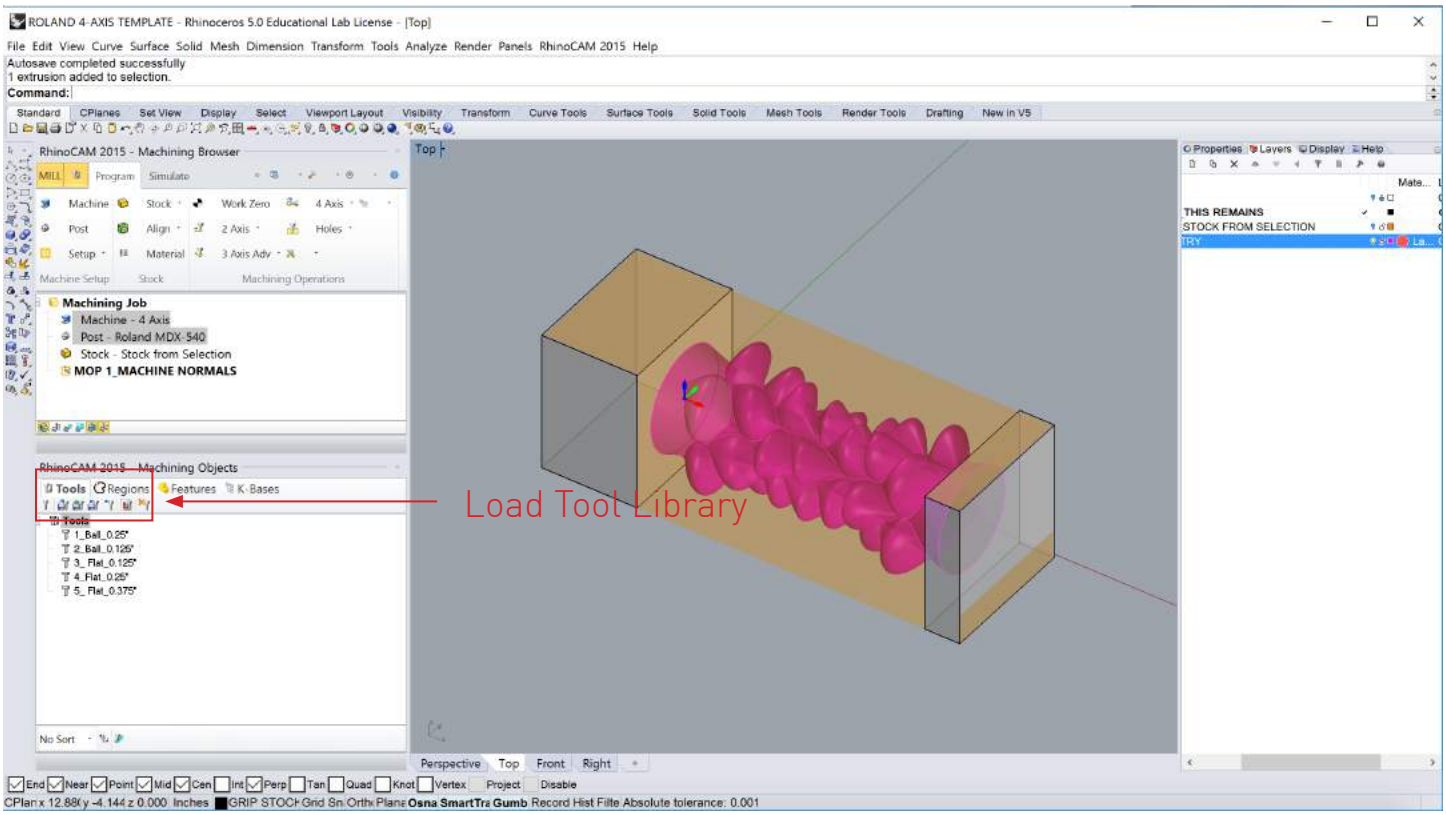
Right click on 'Post', find Roland MDX 540 on the drop down menu



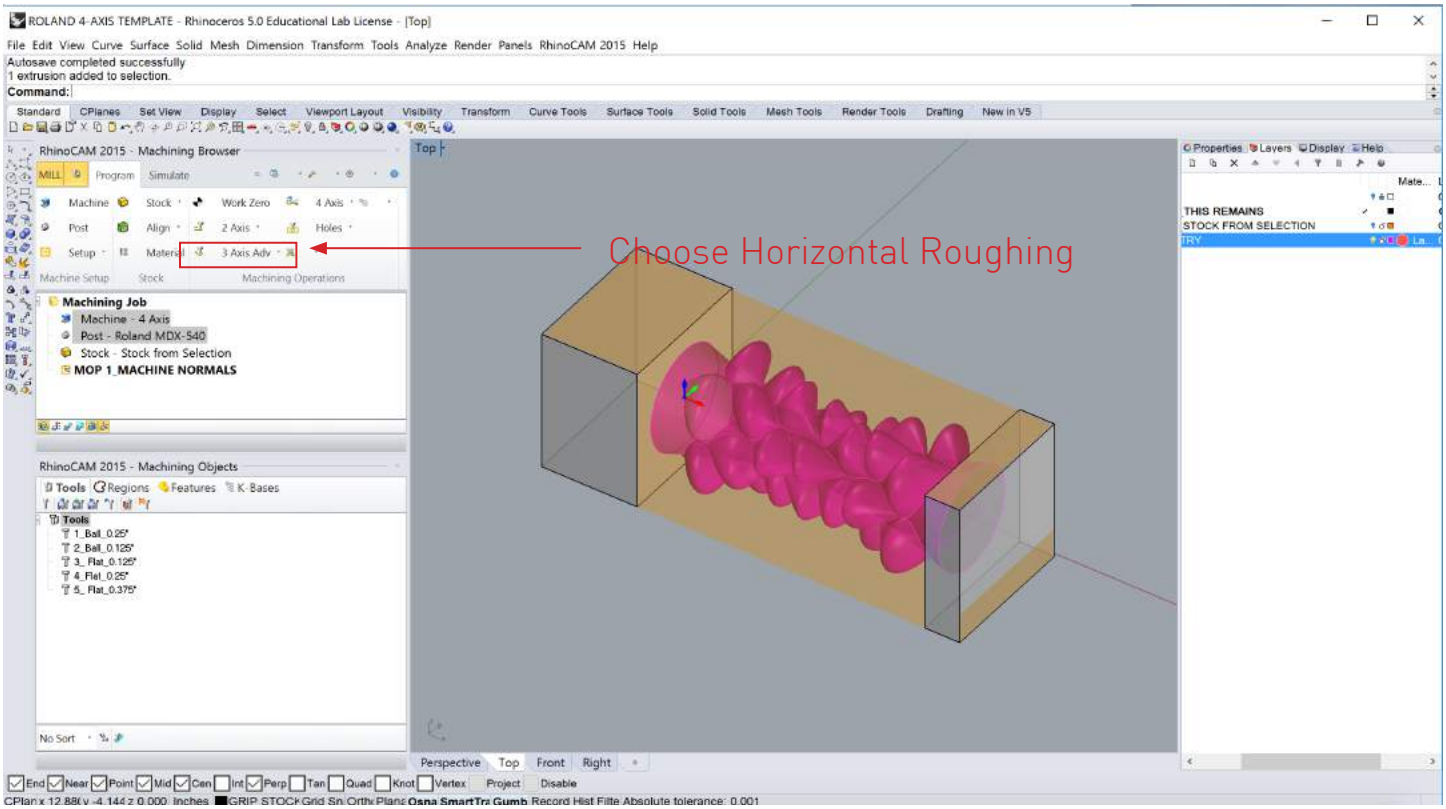
Select your stock material, right click 'Stock' and choose 'stock from selection'. You should now see your stock ghosted in orange when you toggle stock visibility on and off.



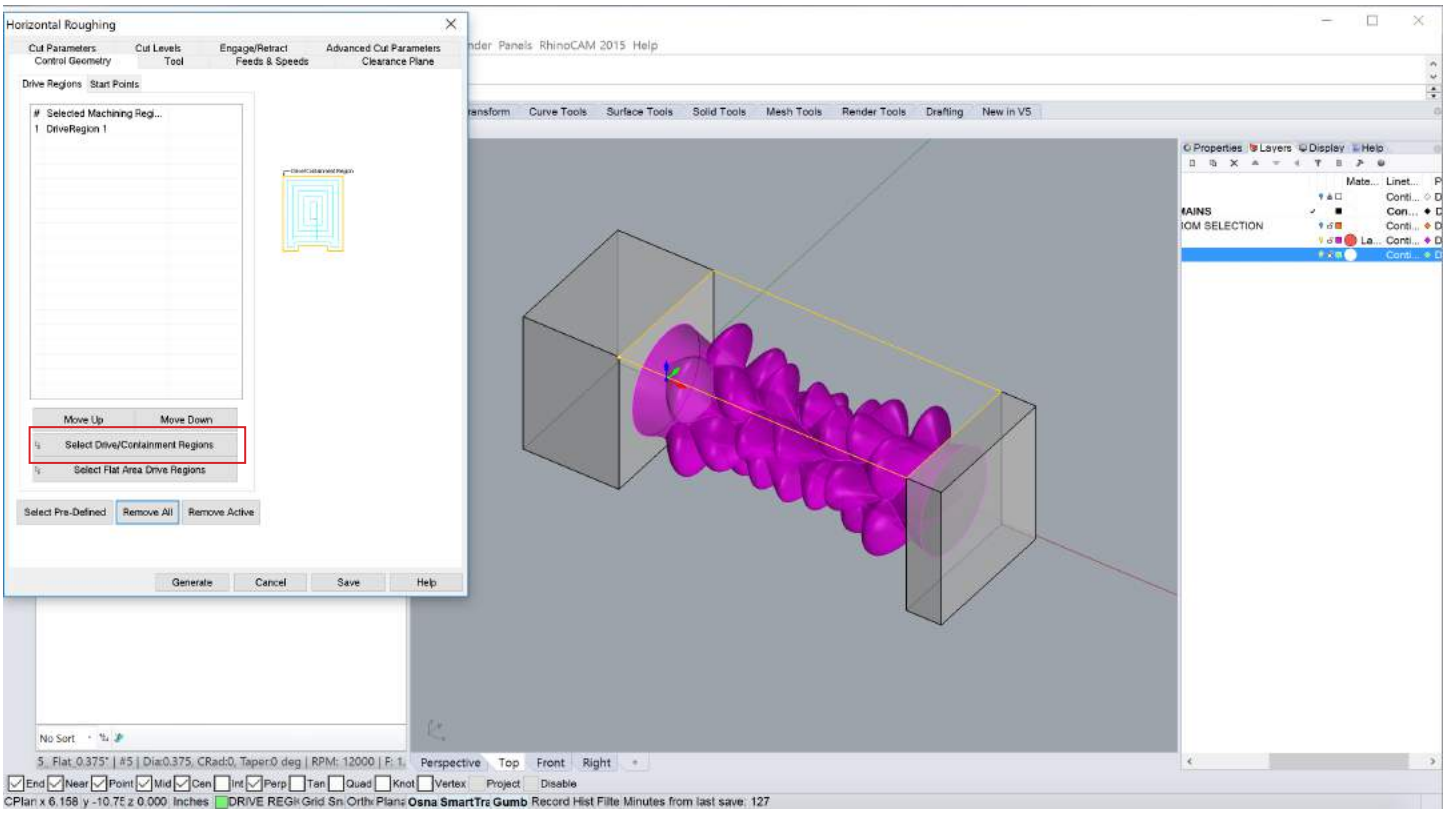
Position your desired geometry within the millable region, centered on the x axis. Your geometry should be fully encased within your stock material, and attach to the leftover fixturing material at either end. Geometry cannot simply float within the bounding volume



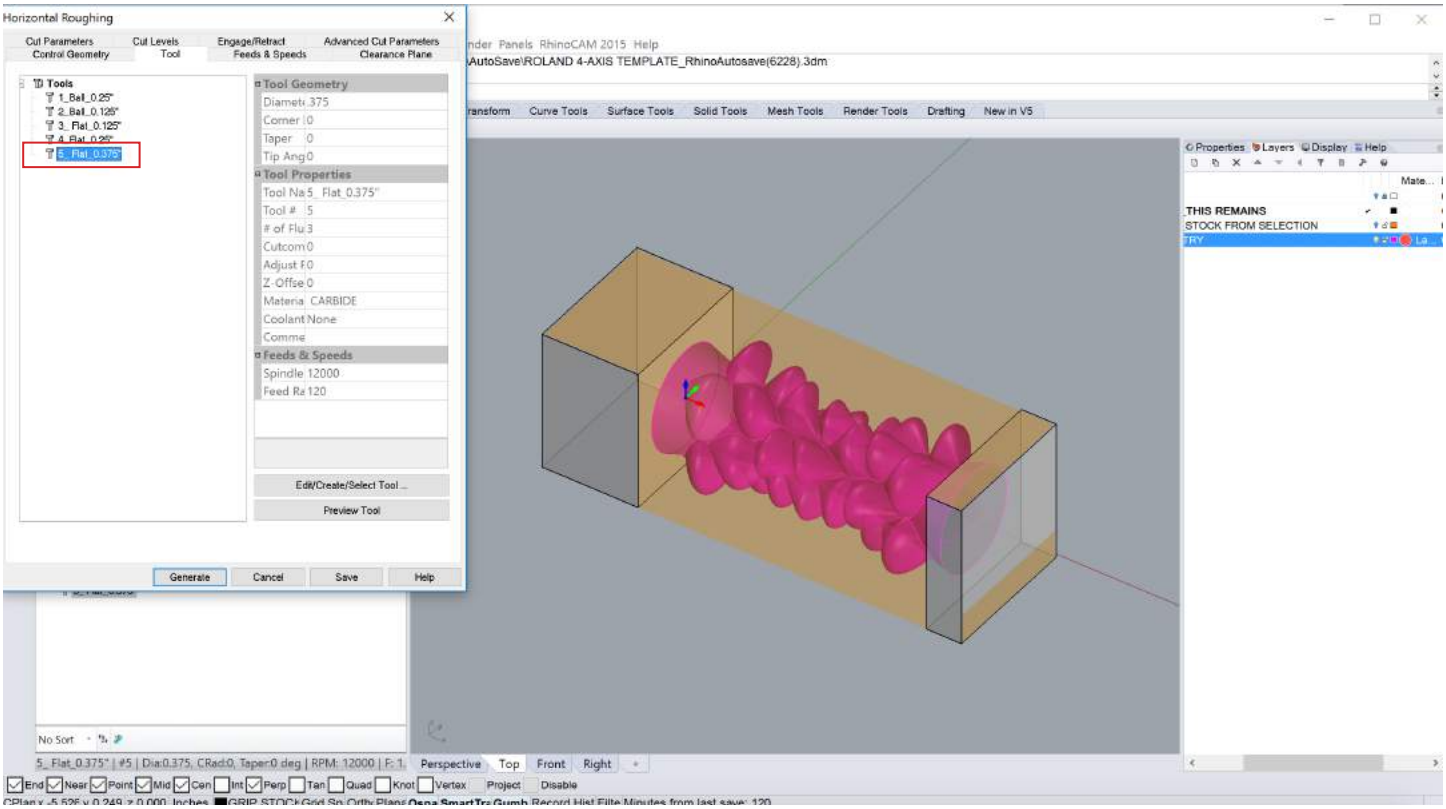
Load the Roland Tool Library you downloaded from the ftp



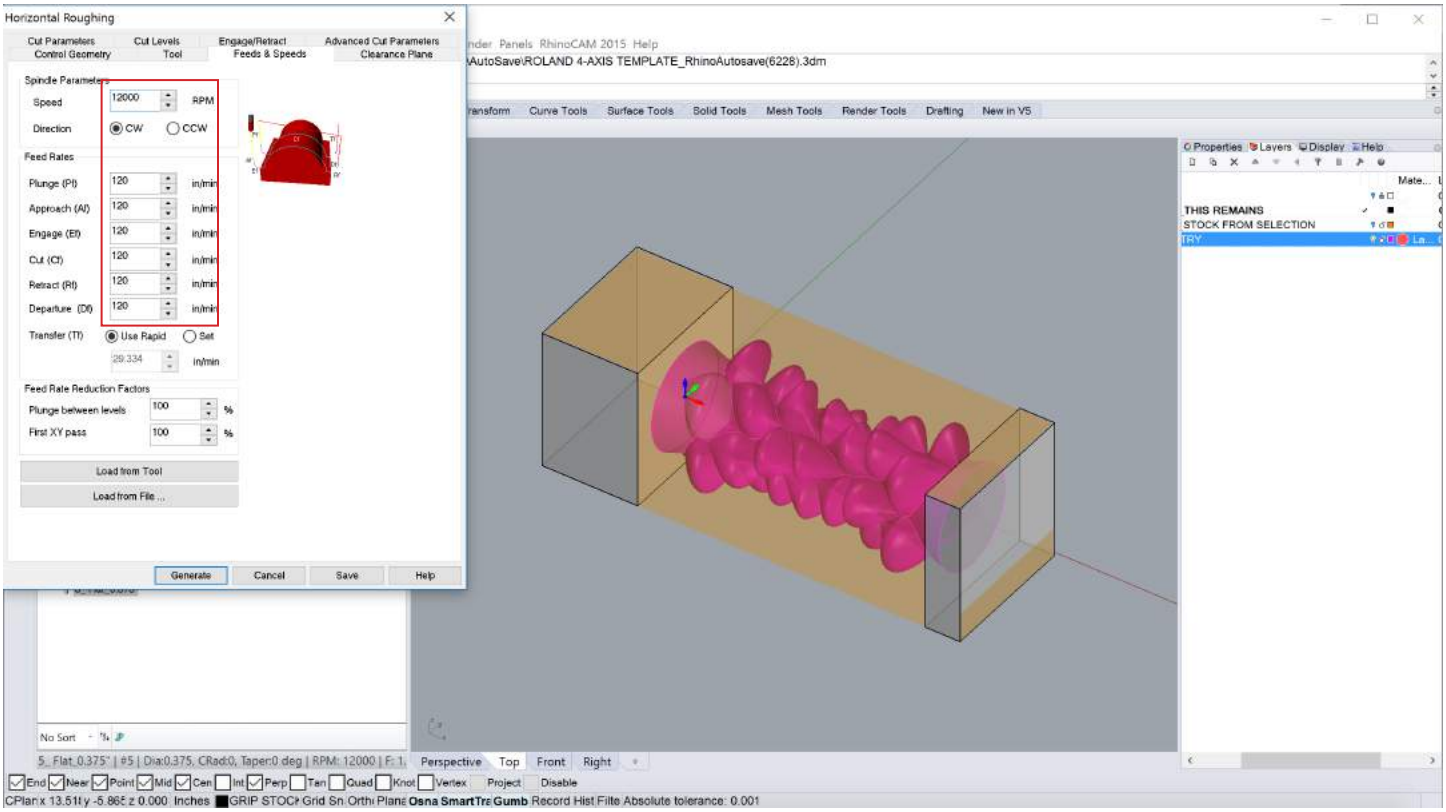
We will rely on 3 axis milling operations to perform a roughing pass from both top and bottom in order to clear out stock material before finishing passes can be run. Select 3 Axis Adv. operations from the Machining Browser, and choose Horizontal Roughing.



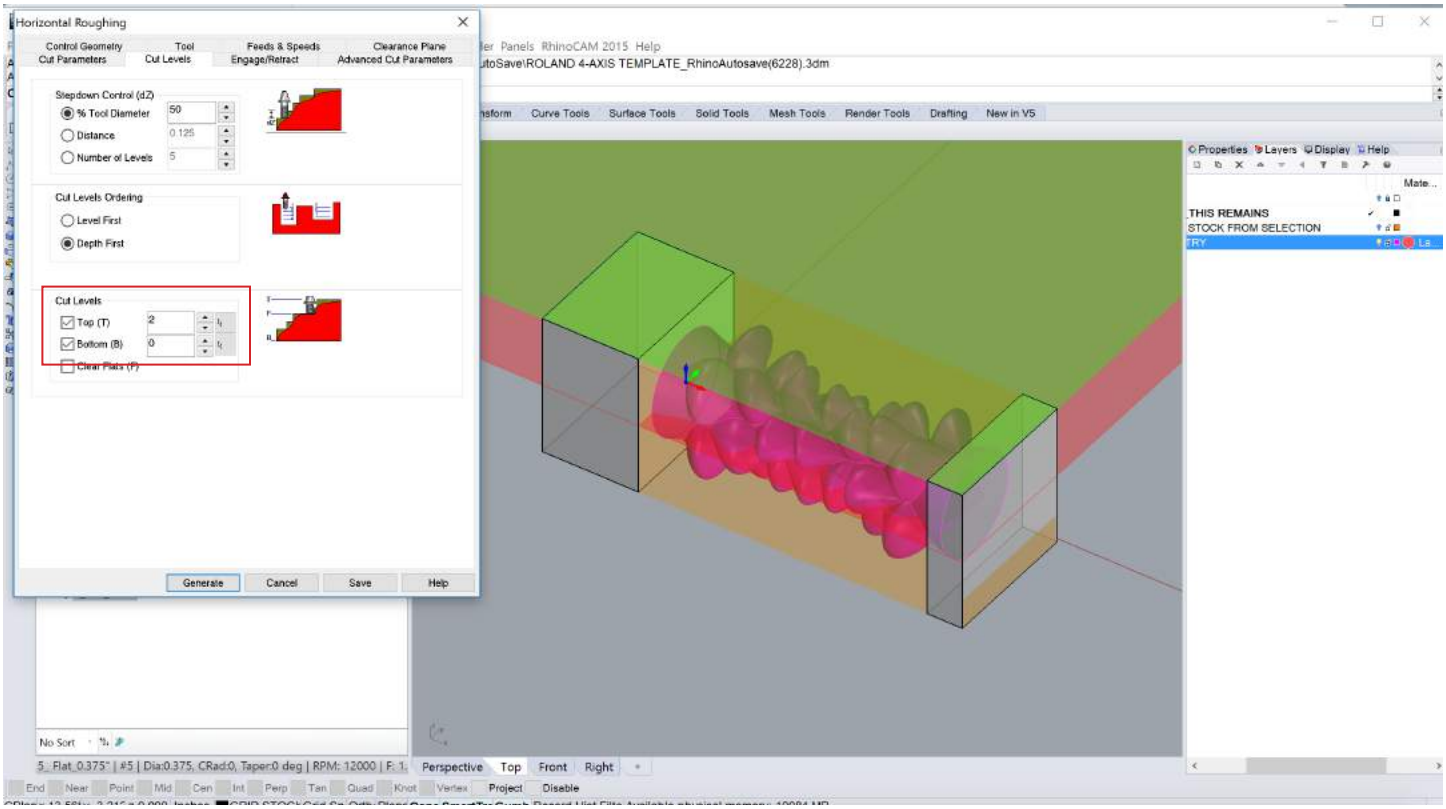
Select a drive region (closed curve) that represents the perimeter of the machinable area.



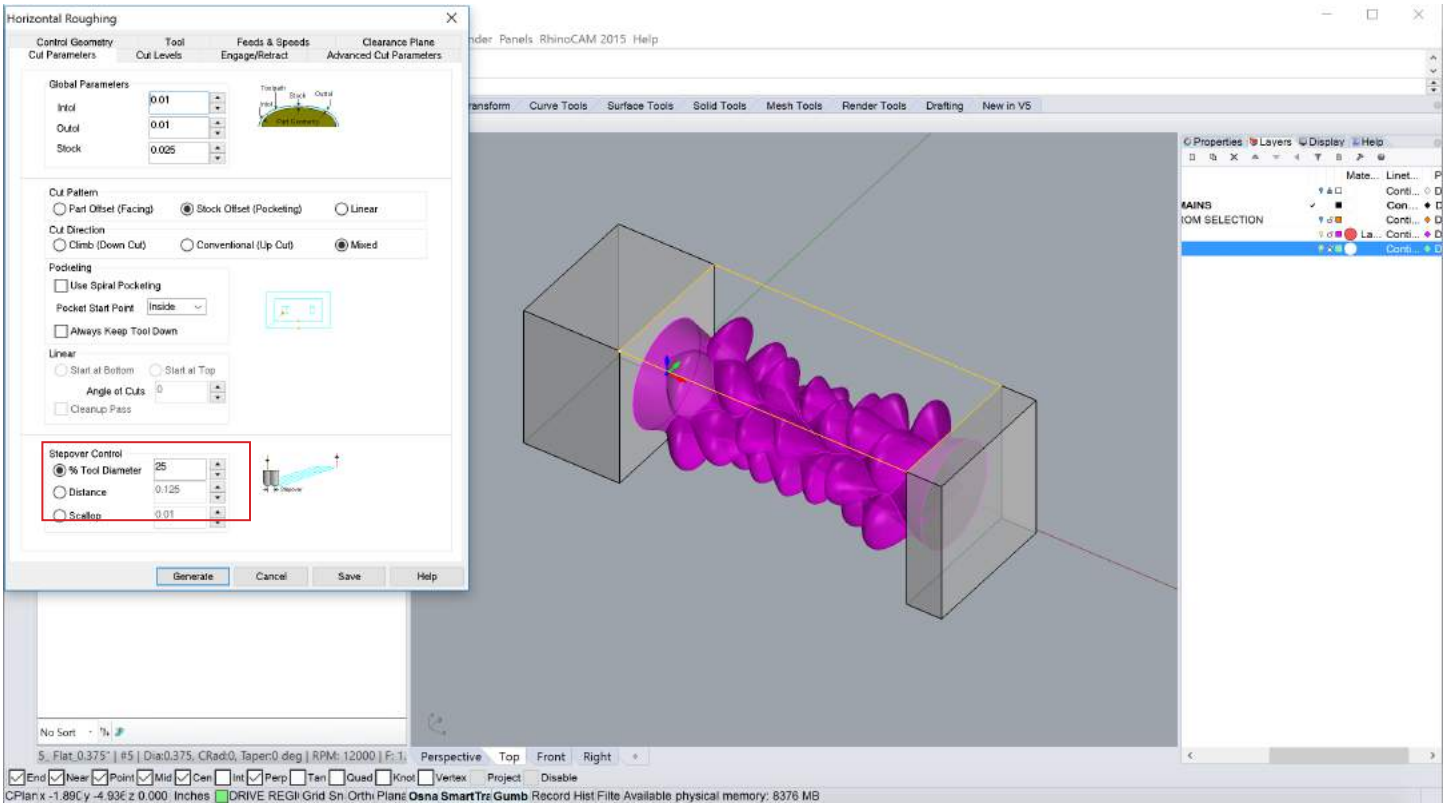
Select the tool you wish to use. In this case, using the largest tool at our disposal will speed up milling rates and allow us to remove more material more quickly.



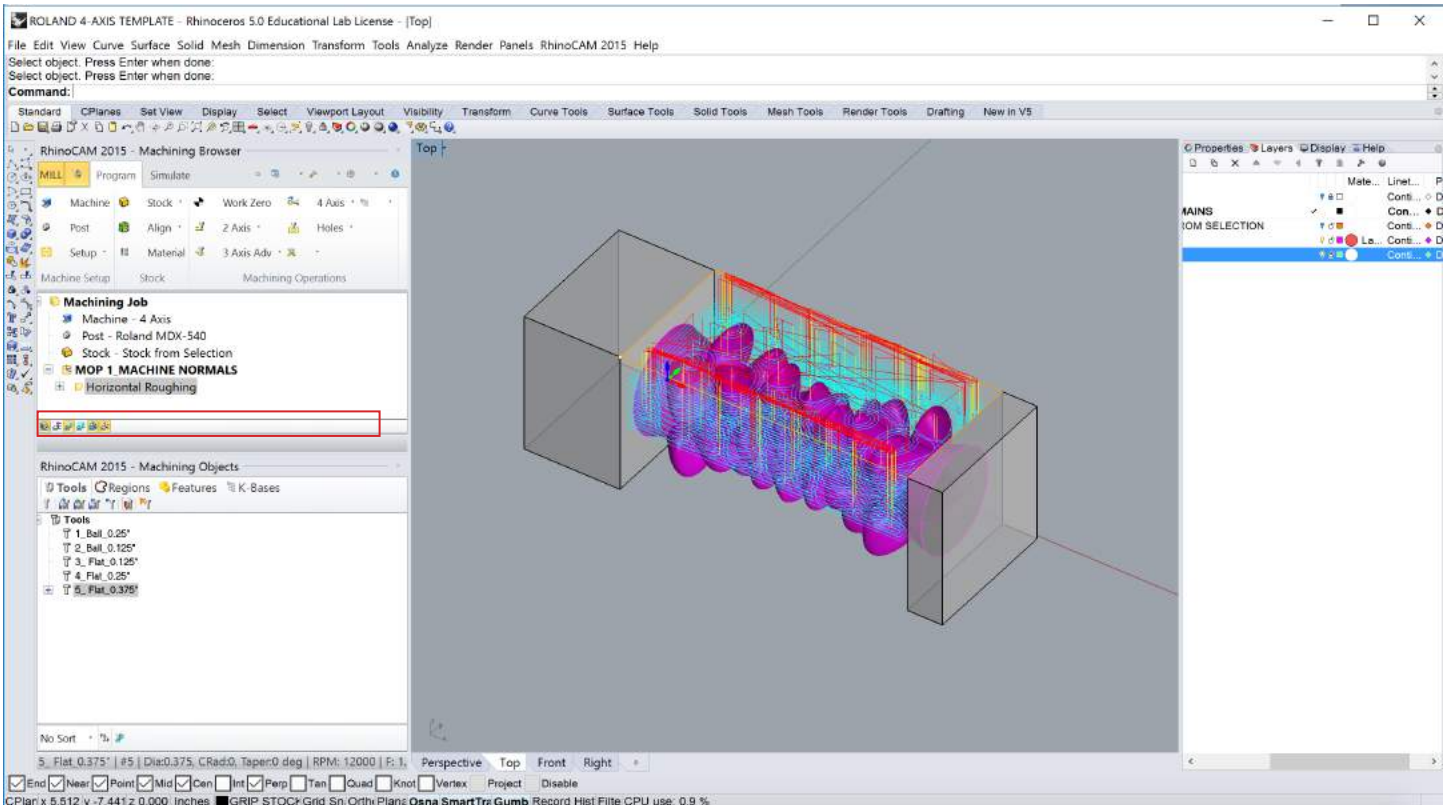
If you have loaded the Roland Tool Library correctly, these values should fill in automatically. A milling TA will review these with you prior to milling, as these values will change for each material. Note that the Roland MDX 540 has a maximum spindle RPM of 12 000 vs the 18 000 of our AXYZ 4008.



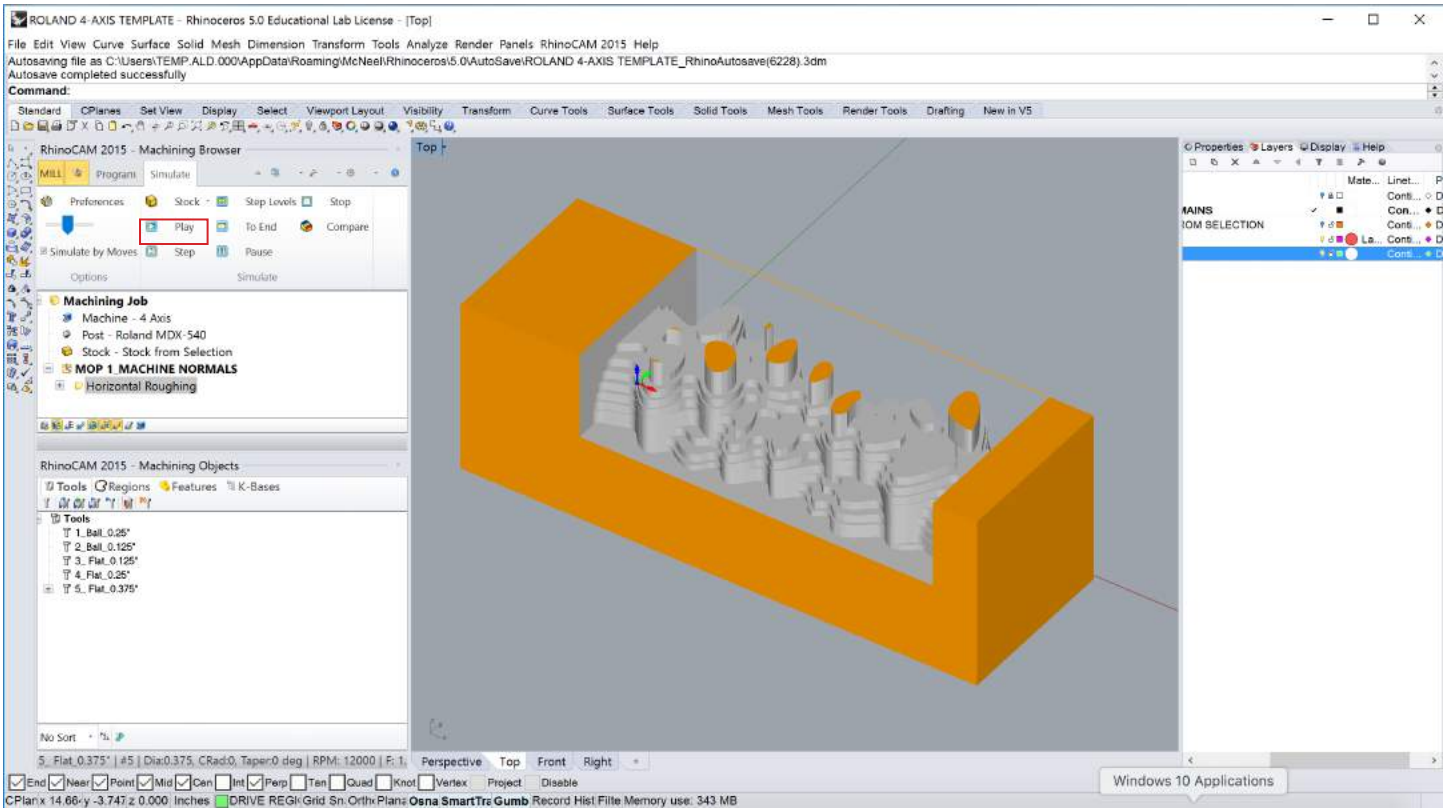
The automatic Clearance Plane, Advanced Cut Parameters, and Engage/Retract values can remain at their defaults. In the Cut Levels Tab, we need to set a manual Top and Bottom values to our cut. We are going to rough from both top and bottom to avoid undercuts, so setting the bottom to zero will have both operations meet in the middle.



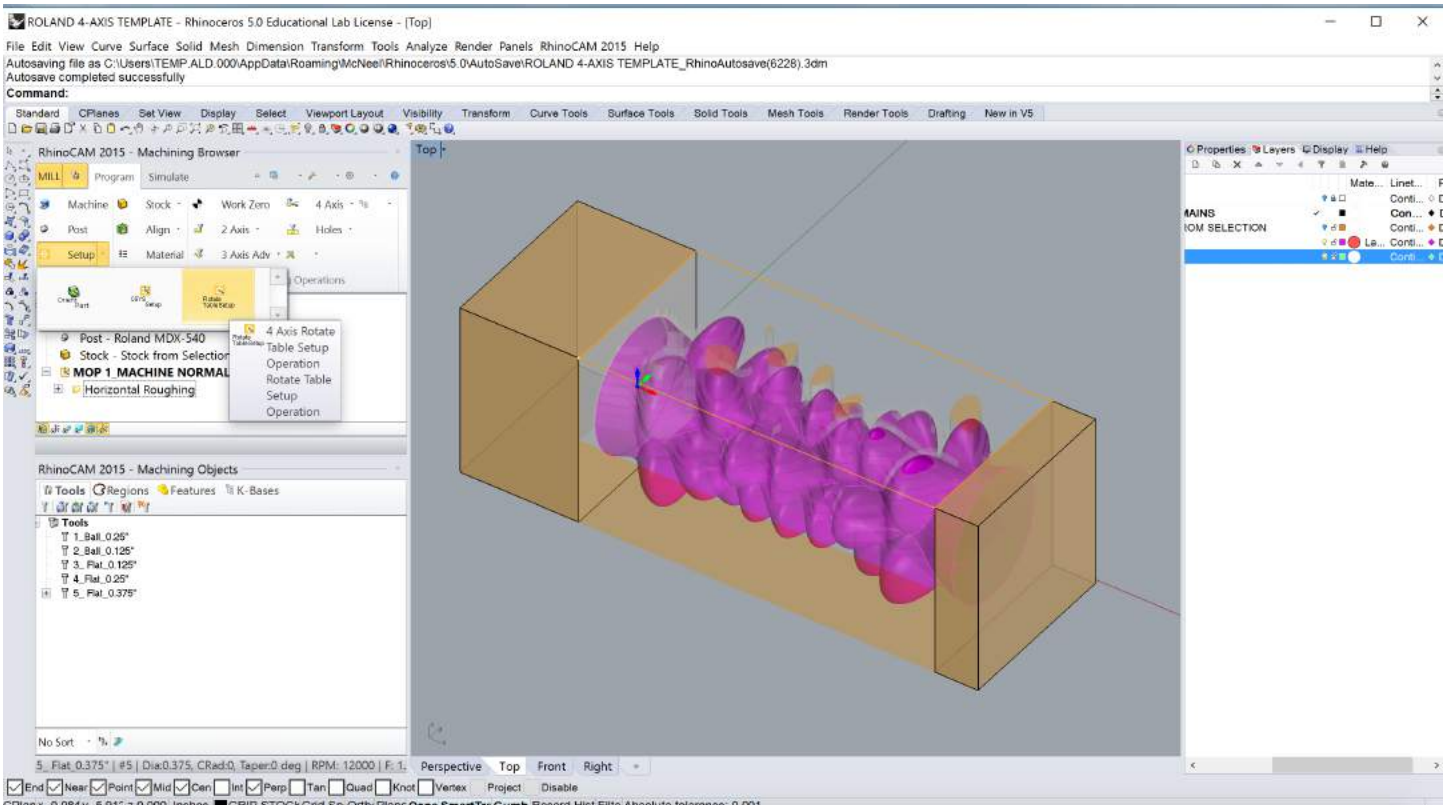
In this case, the default stepover of 25% is desirable. This could be adjusted based on material choice, review this with a milling TA prior to starting your job.



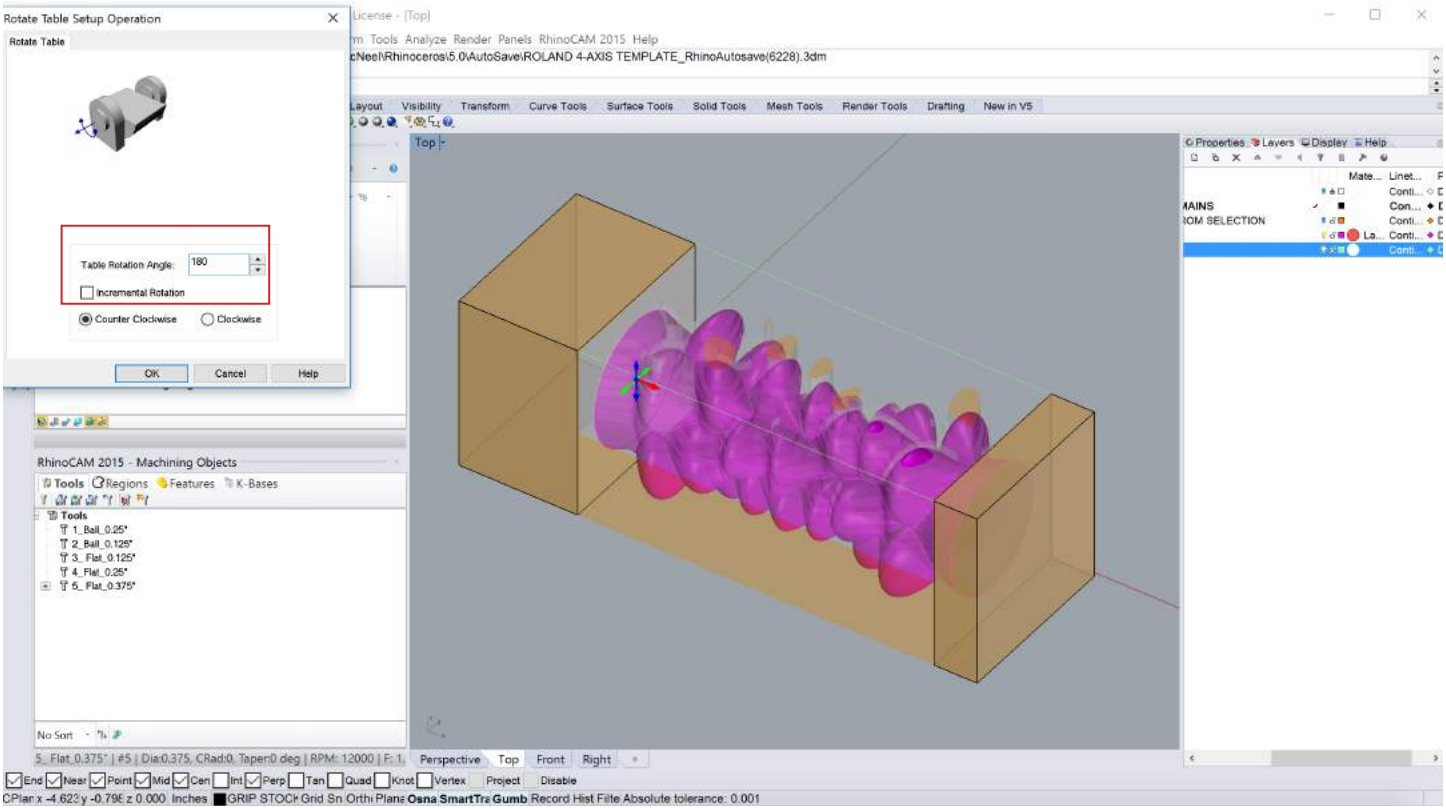
Generate this operation. Toolpaths will be visible as you toggle “ToolPath Visibility” on and off



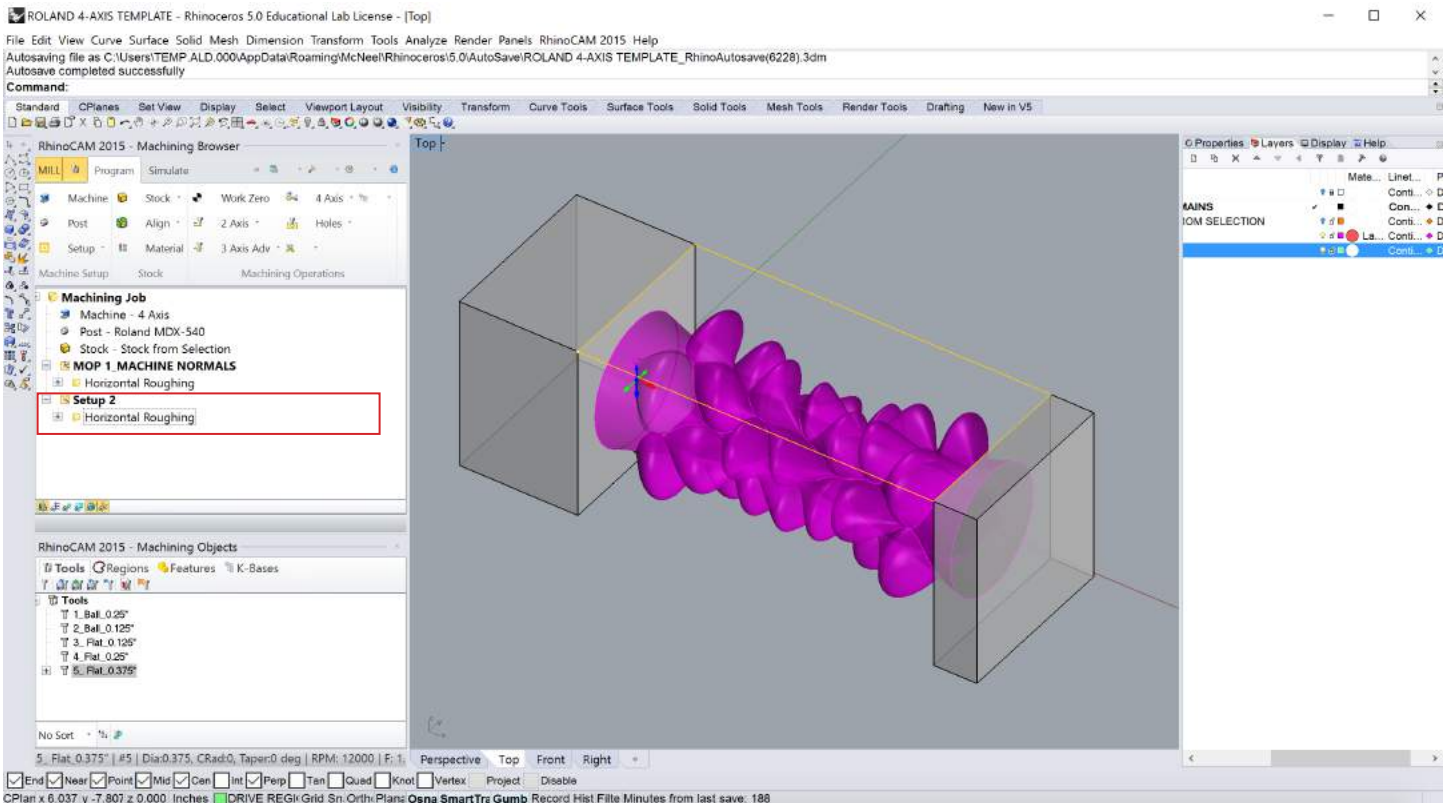
Highlight the Horizontal Roughing operation and click ‘play’ in the Simulations tab. Toggling the toolpath visibility on and off will show you the results of our roughing operations so far. Notice that we have only gone to 0, and the bottom side remains.



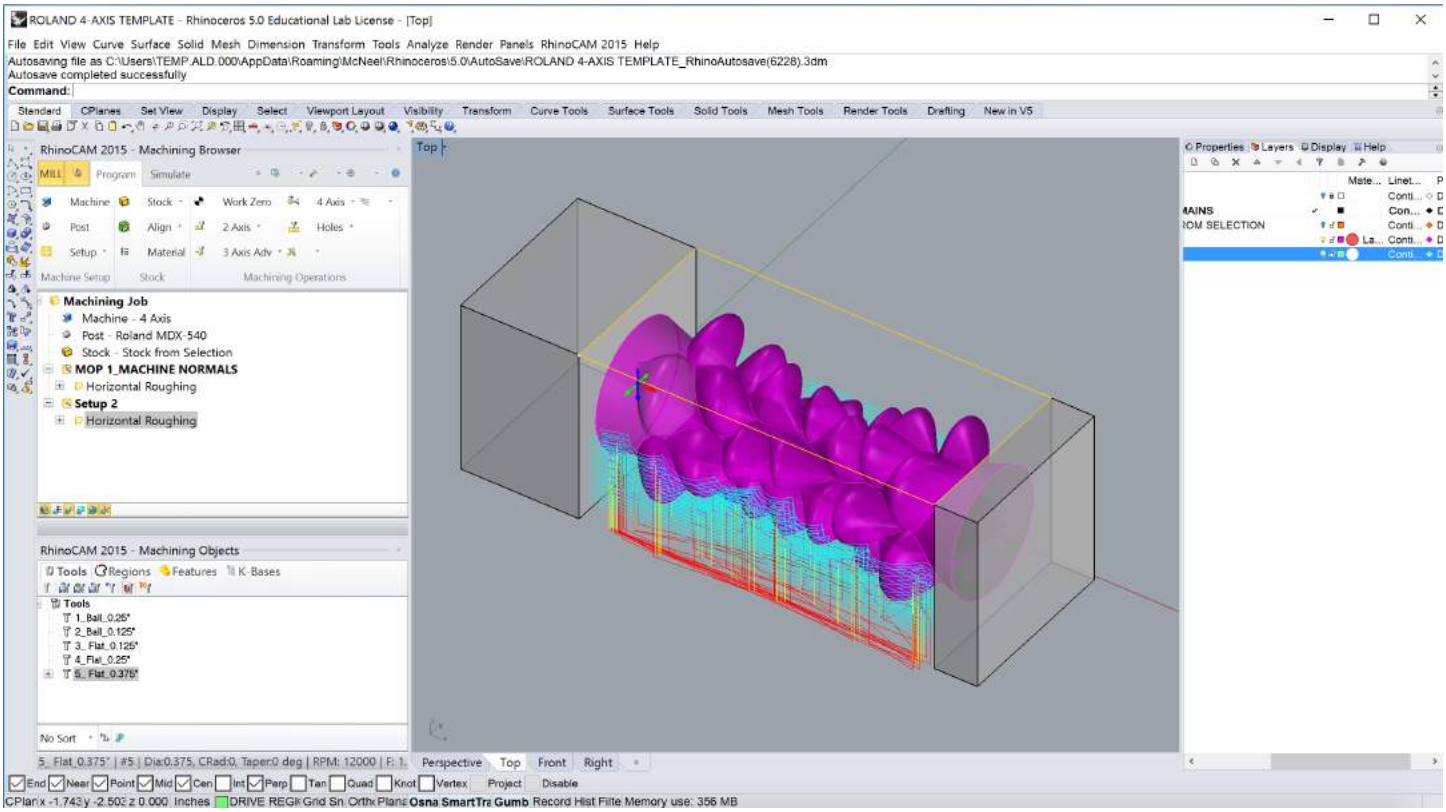
The Machine Operation Set (CSYS Setup) we used for the first roughing pass uses the Rhino axis' to determine part orientation, ie. Rhino Z is 'up'. This is the default setup type. Now, we need a Machine Operation Set that will change the orientation by 180 degrees in order to mill from the bottom. Click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



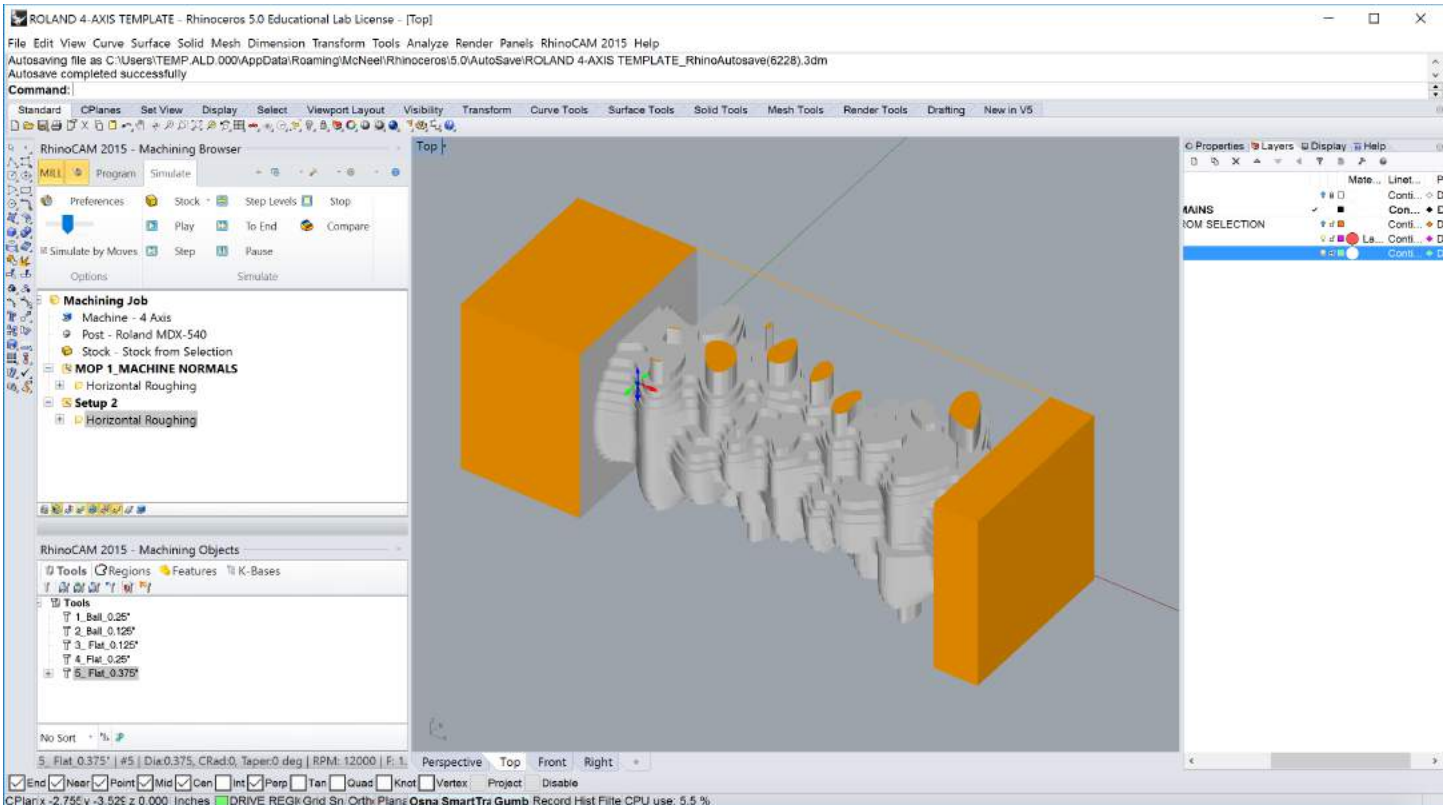
Change the rotation value to 180 degrees. Notice in the Rhino window that the axis arrows at the origin have rotated to match.



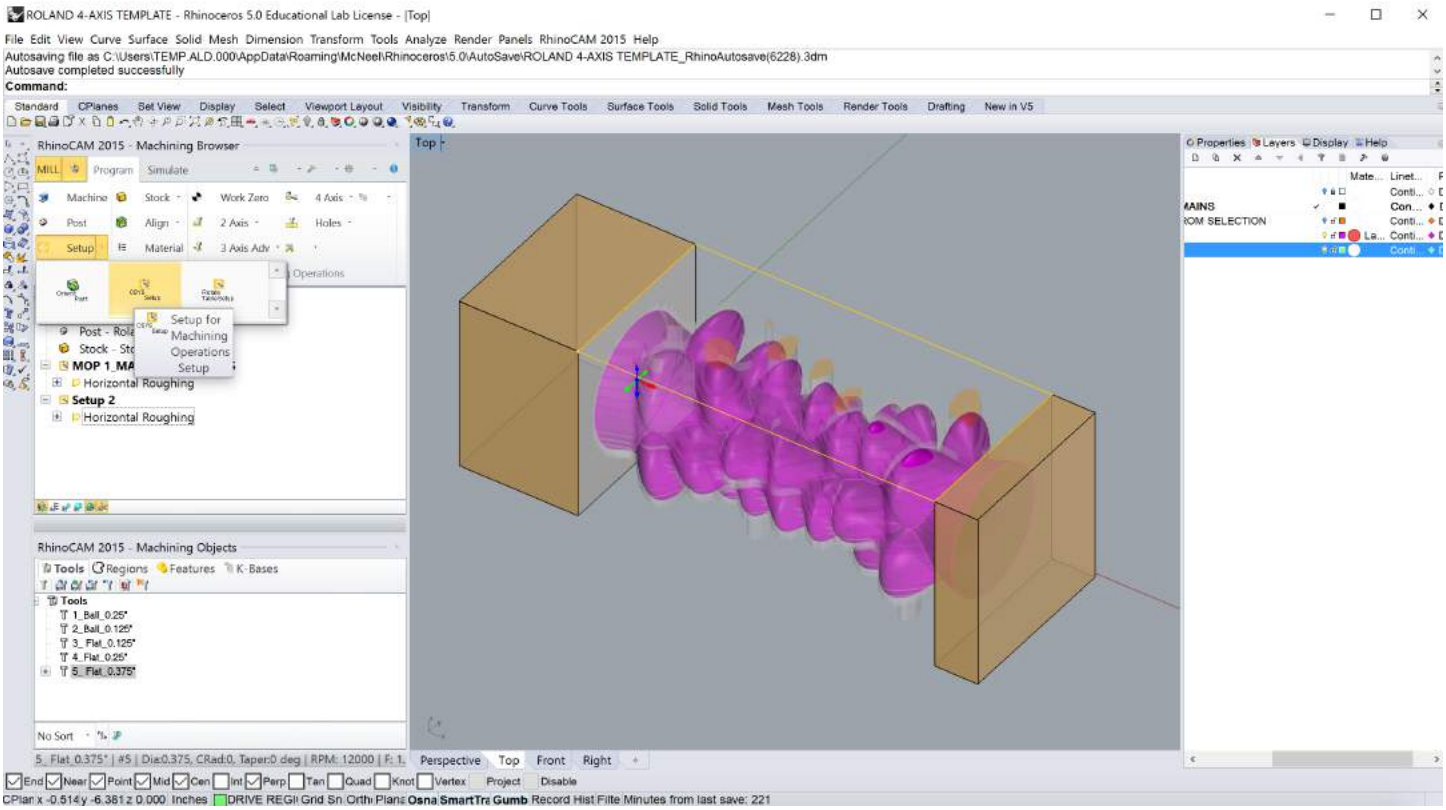
Copy and paste the original Horizontal Roughing operation into this new Setup. Regenerate this operation.



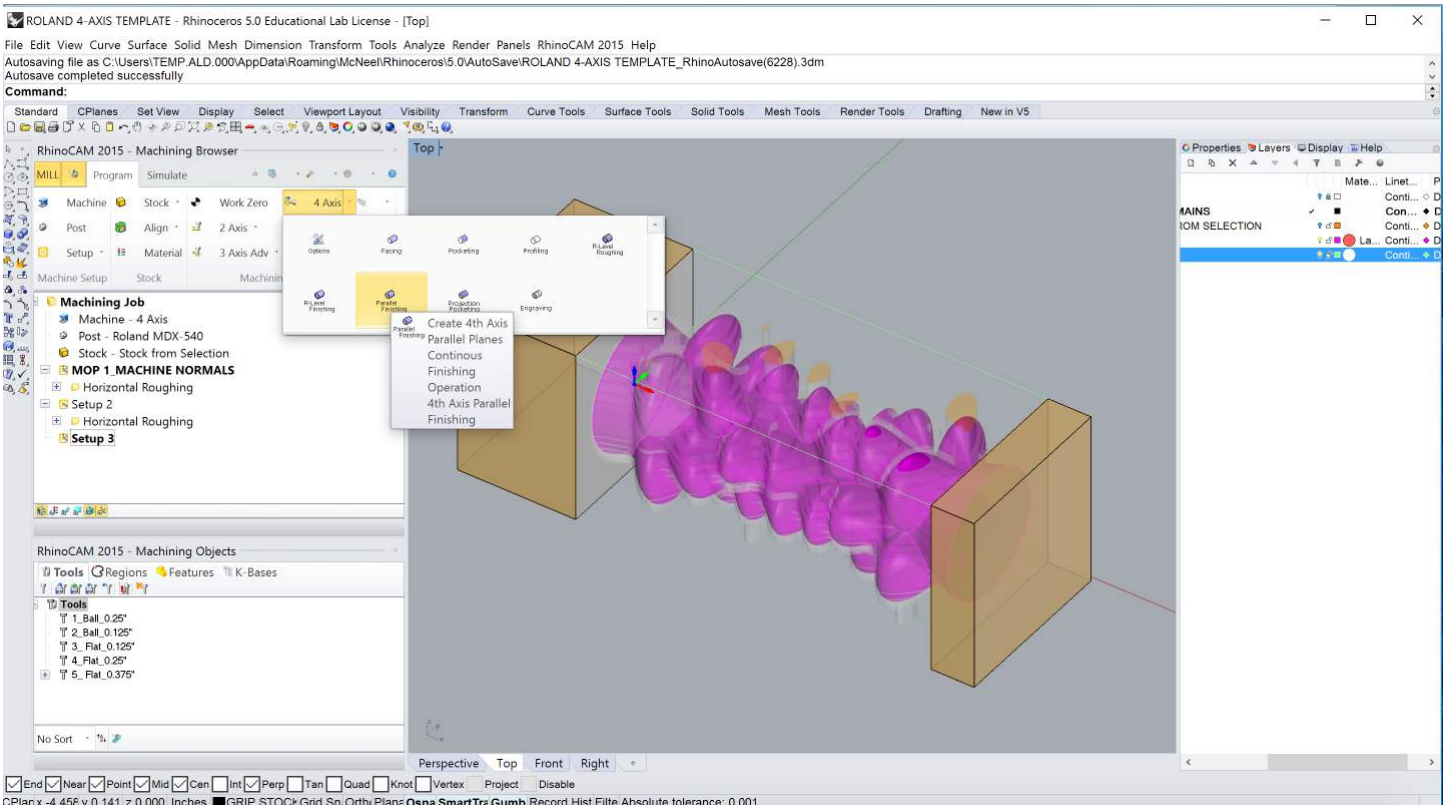
Your toolpaths should now be generated from the underside of the stock. Note that this is just a Rhino quirk, in reality it will be the piece that is rotating, while the tool always approaches from above.



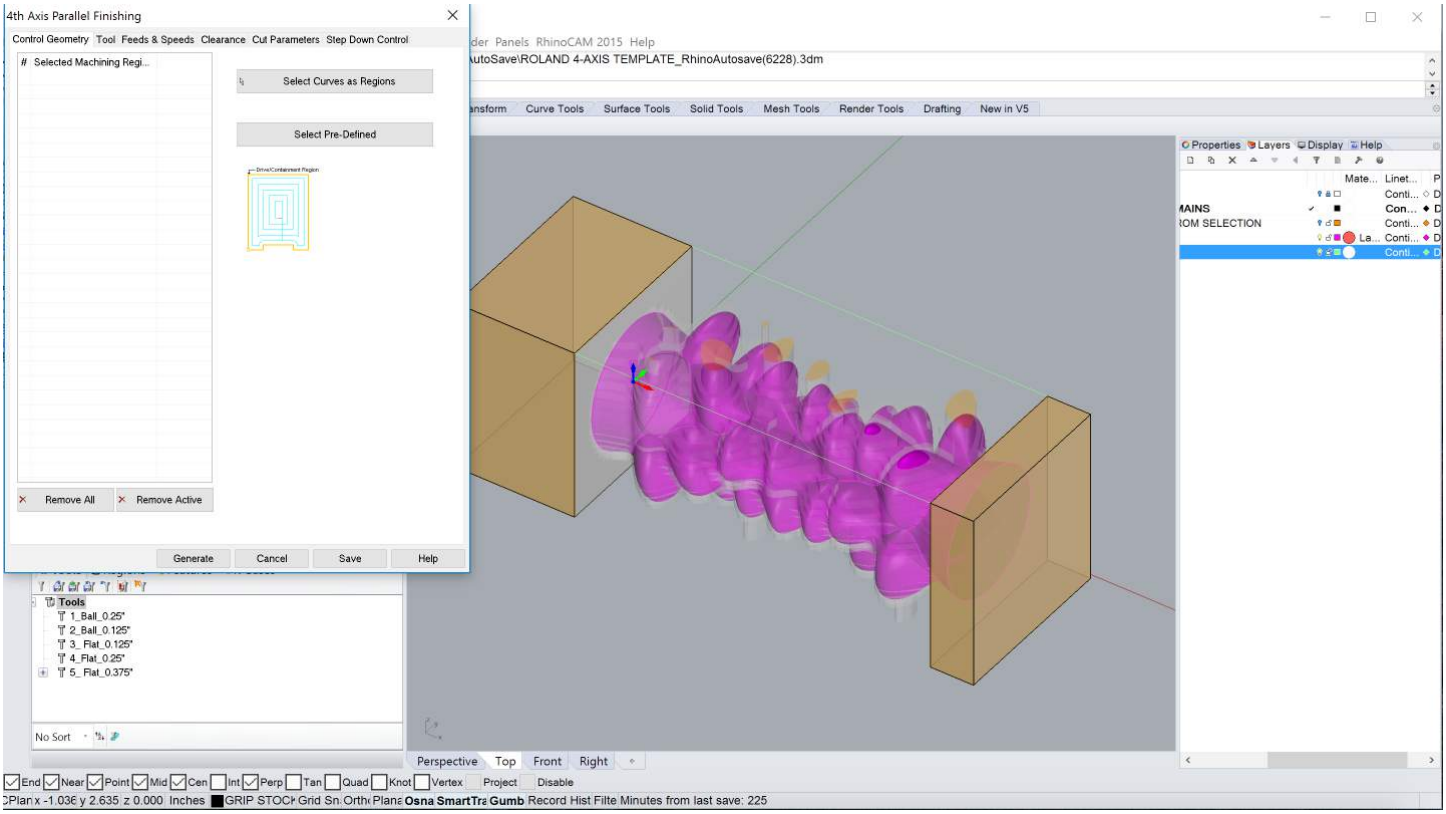
Simulating both operations should show you a roughed out geometry from both top and bottom.



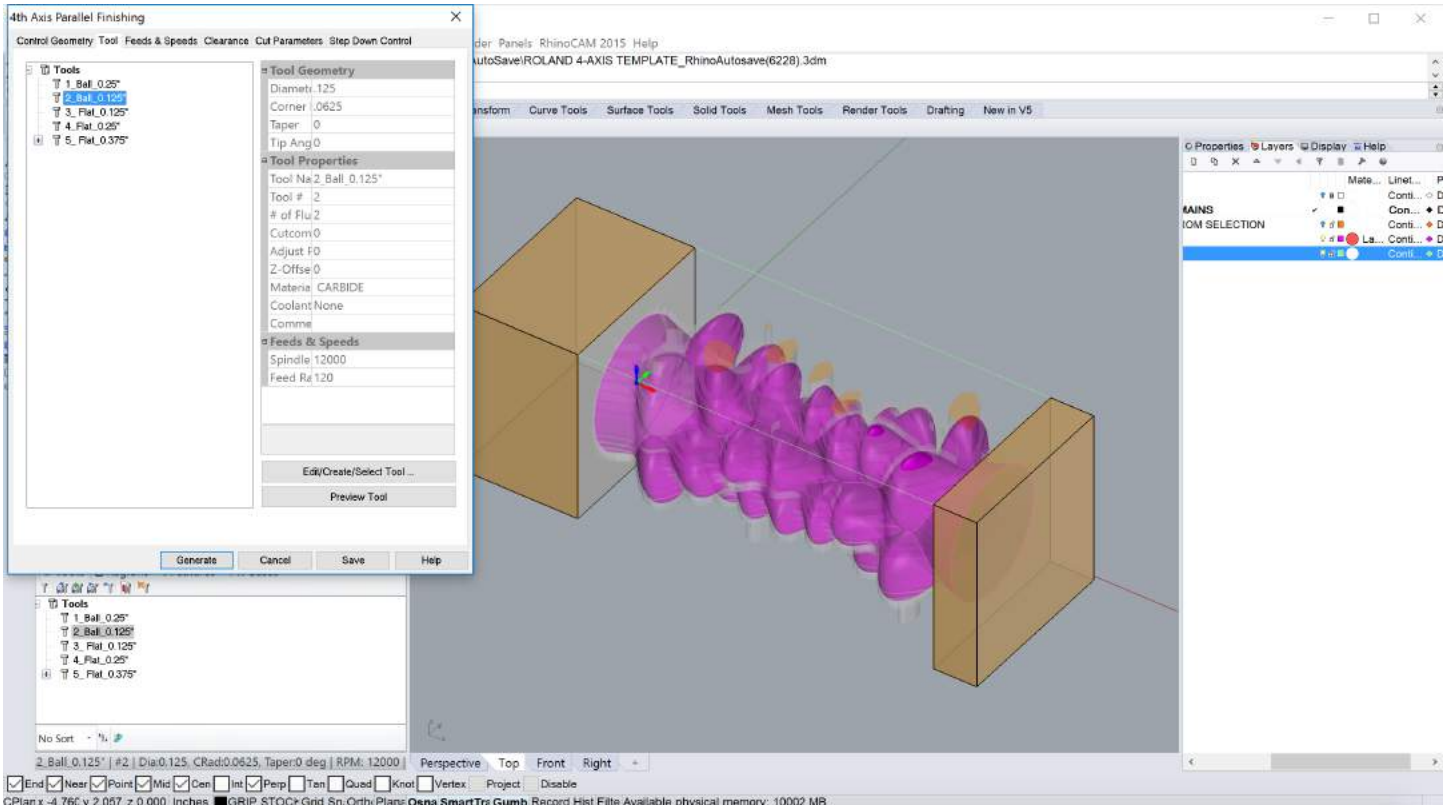
Now, create a new CSYS setup to return to the original orientation.



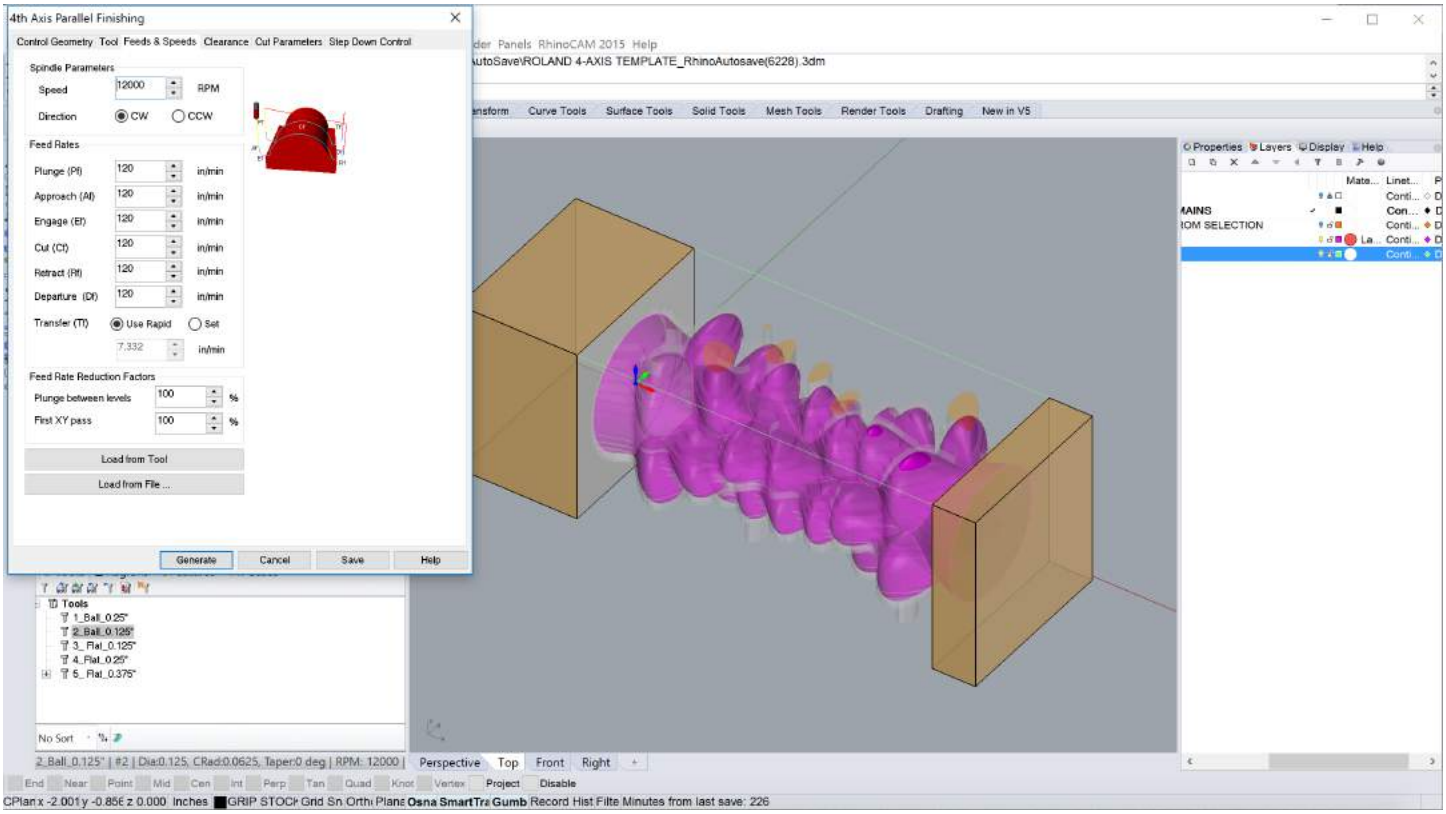
Click the 4 Axis tab and create a 4 Axis Parallel Finishing operation



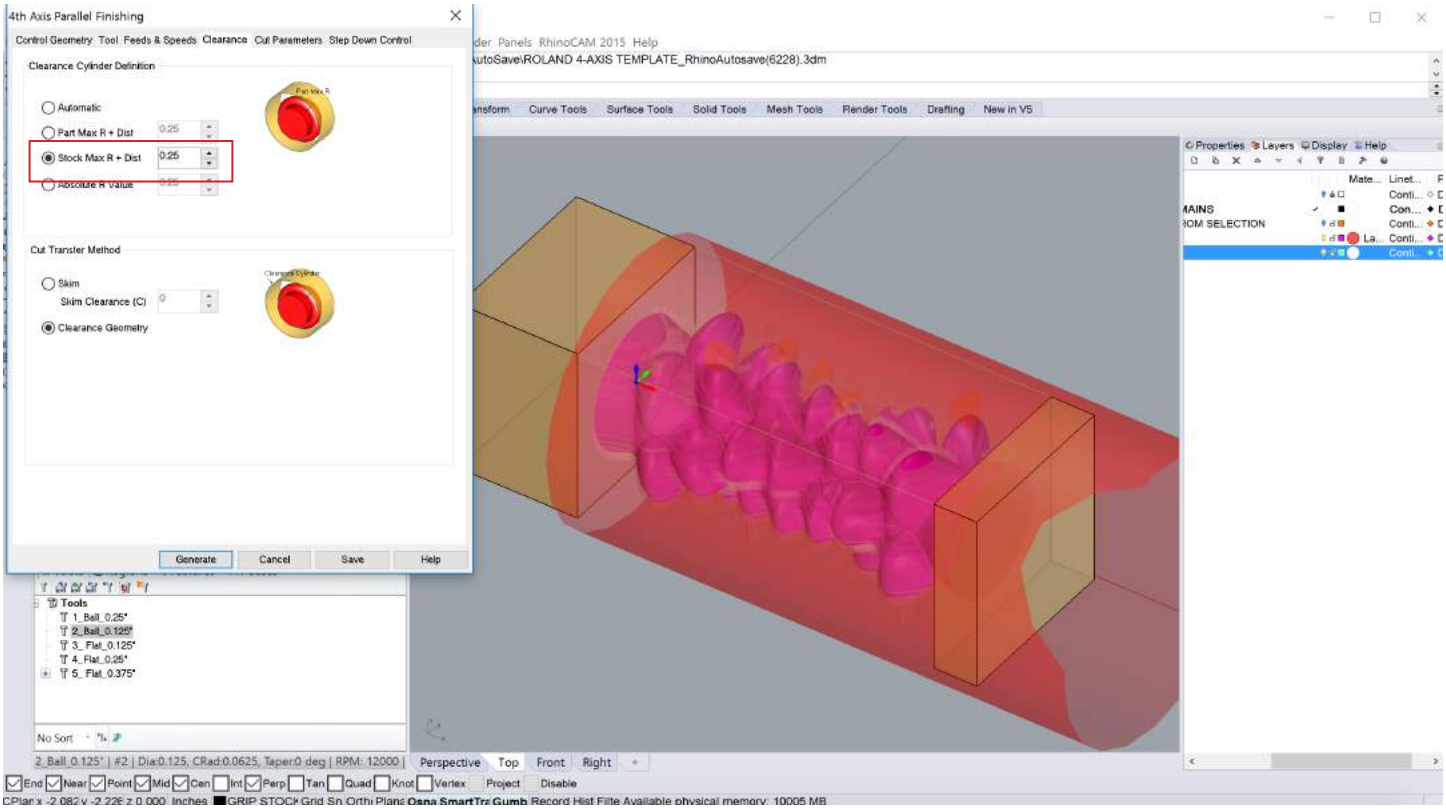
This type of operation does not require a drive region. We will define its bounding conditions later.



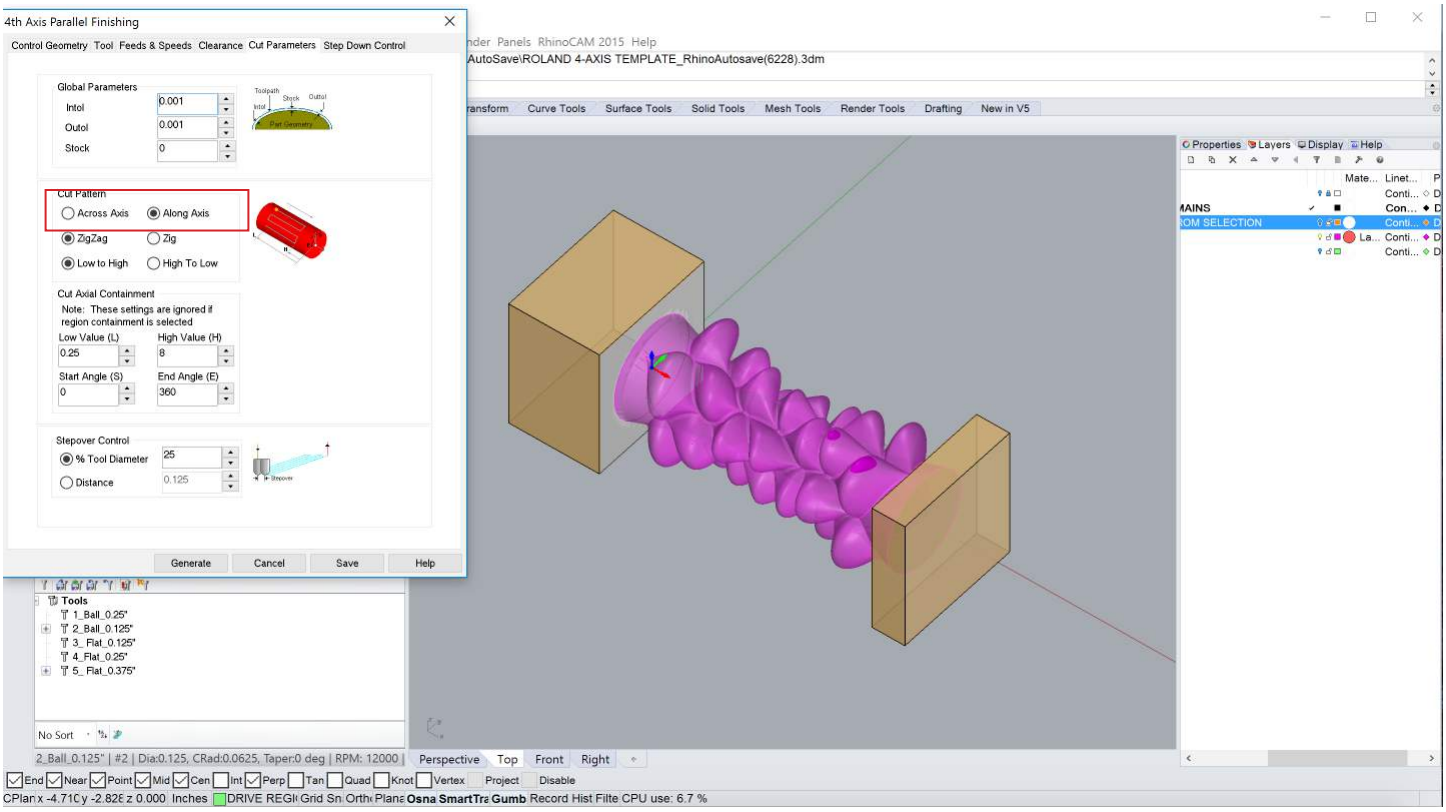
Select the tool you wish to use. In this case, a 1/8" ball endmill will give us ideal resolution.



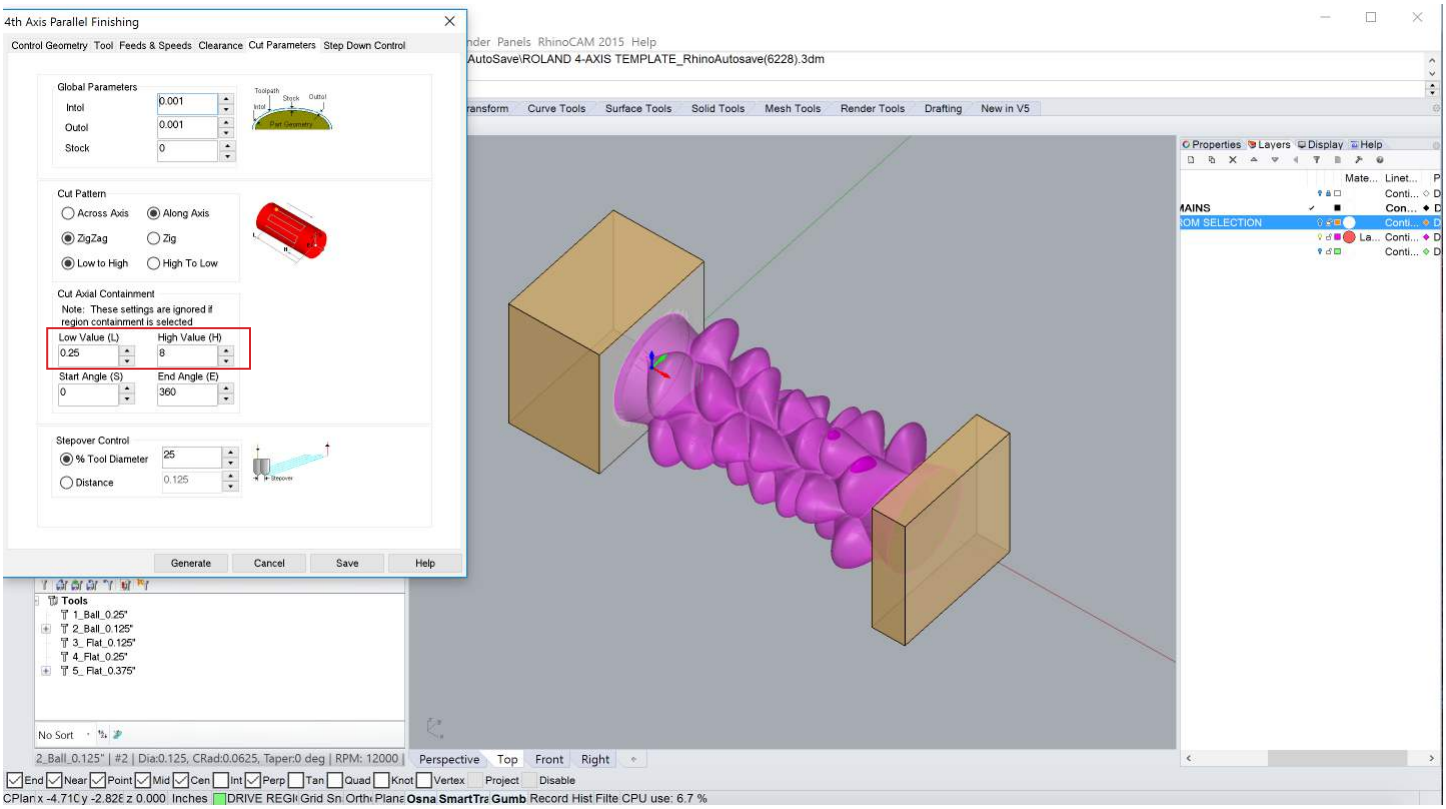
Again, feeds and speeds load automatically from the Roland Tool Library. Review these with a milling TA prior to starting your job.



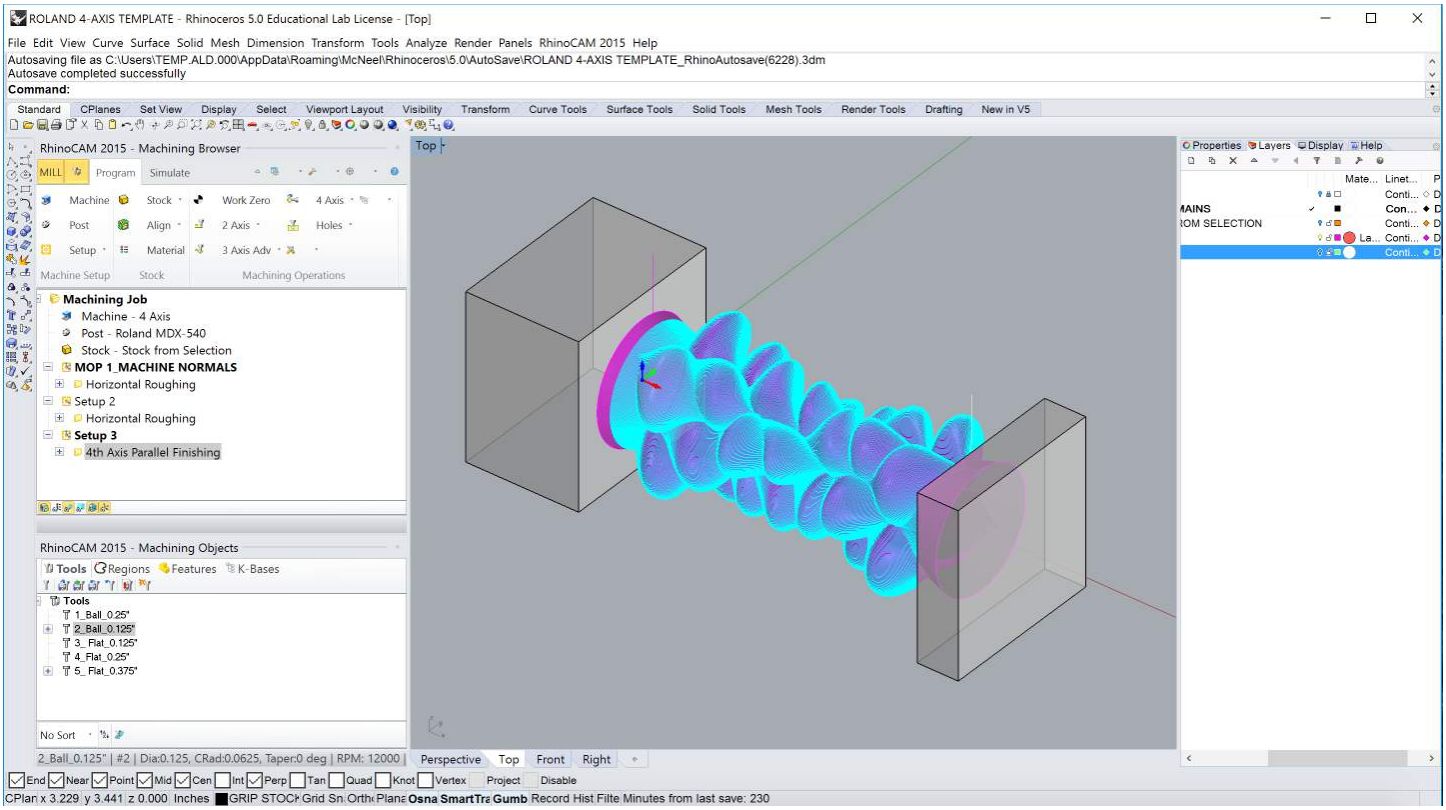
In the clearance plane tab, select stock max + 0.25". You should see that the clearance plane is represented by a cyclinder here rather than a flat plane.



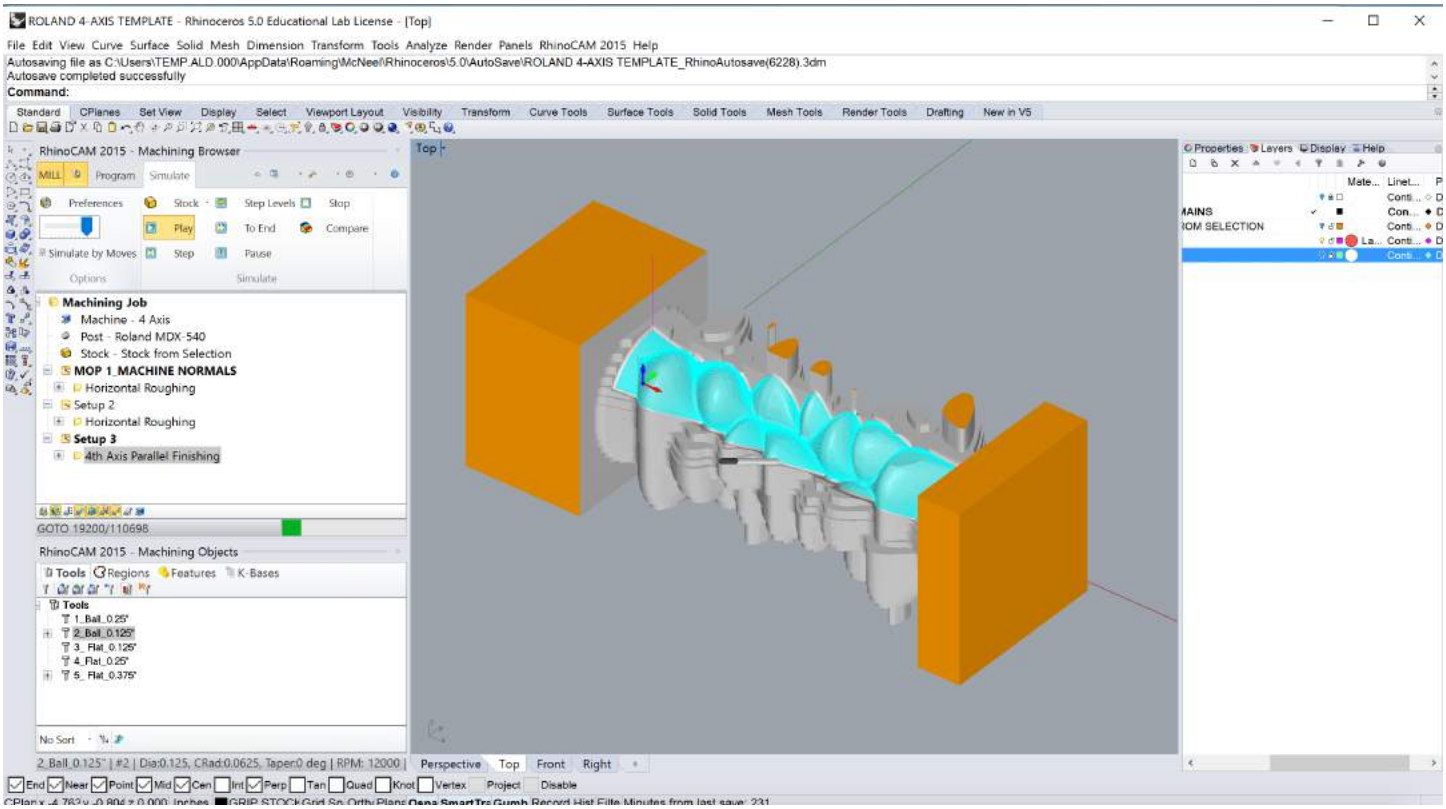
In the cut parameters tab, we need to define a few variables. First, whether we want to cut along the axis vs across. This is a choice you can make depending on your geometry, for now the along axis works fine.



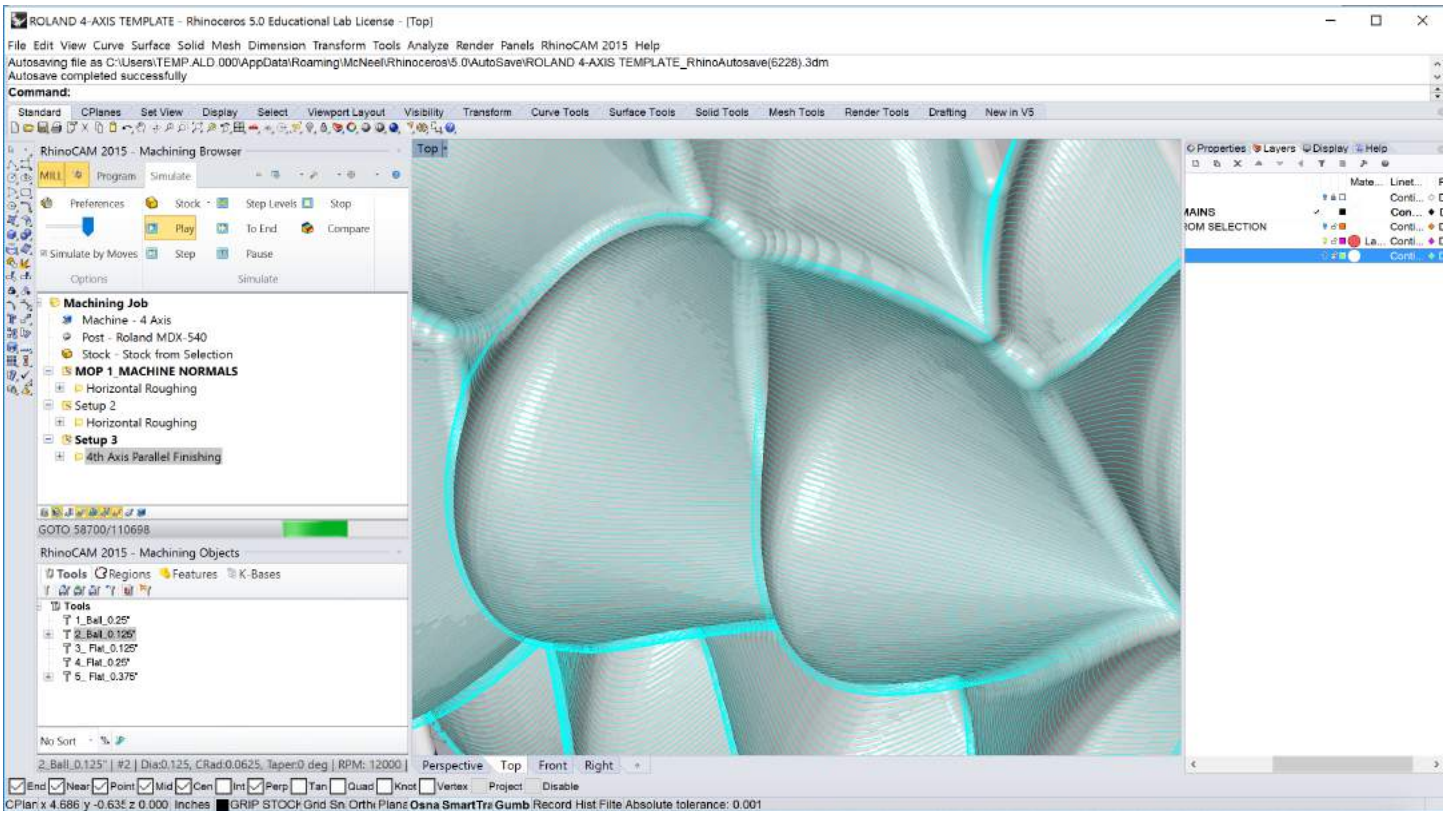
Next, we define axial containment. We want to keep the endmill off of those harsh vertical faces to reduce cutting strain, so we tell it to only cut along the x axis between 0.25" and 8" (our actual geometry runs 0" to 8.25"). The default 25% stepover is fine for now.



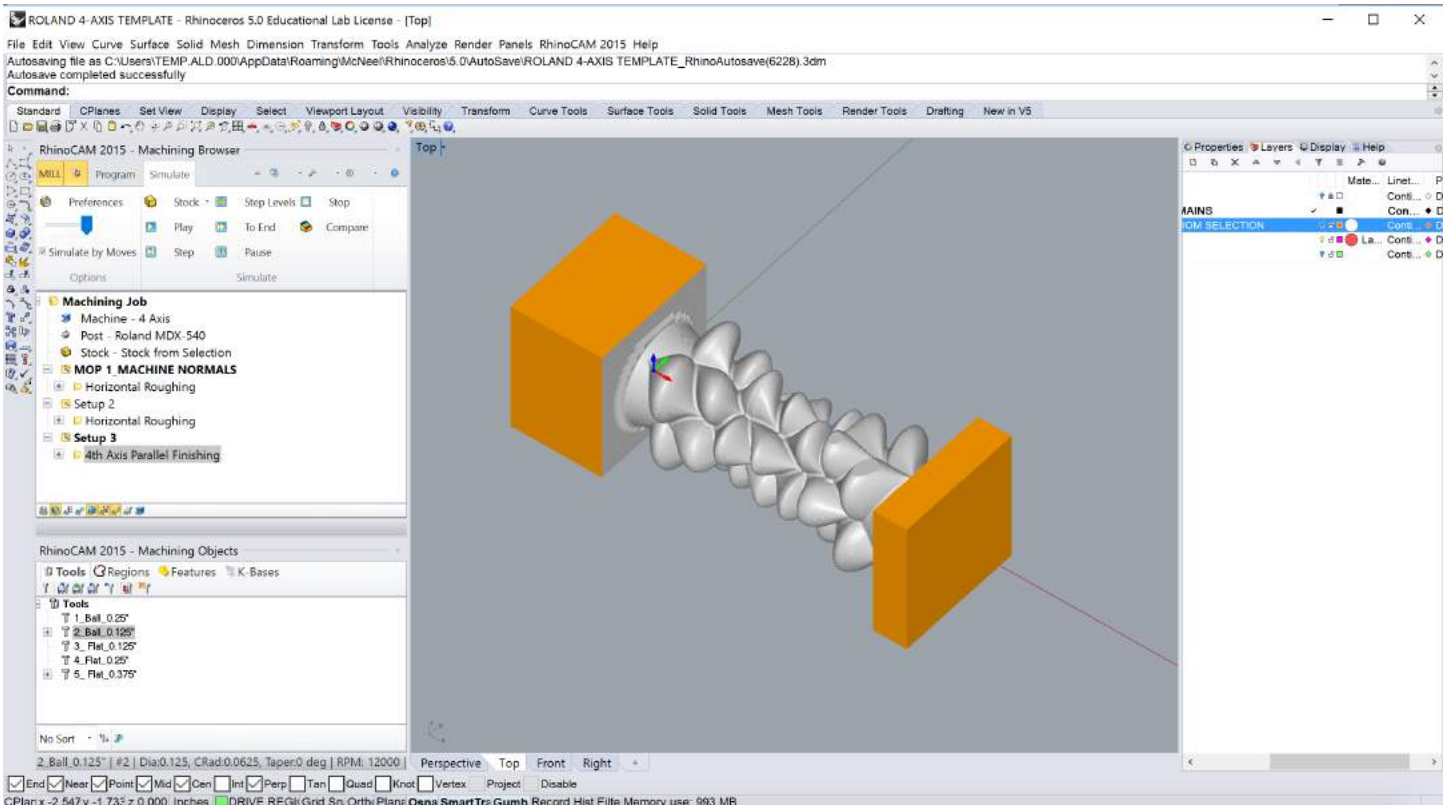
Generate this operation to view its toolpaths. Notice how they stop short of the vertical faces of our fixturing geometry.



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



Zoom in and check out your toolpaths. These types of paths would be impossible to realize without the simultaneous movement of the 4th axis component.



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