

**RhinoCAM<sup>®</sup>**



**RhinoCAM 2012\_for Rhino 5**

## CONTENTS\_

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

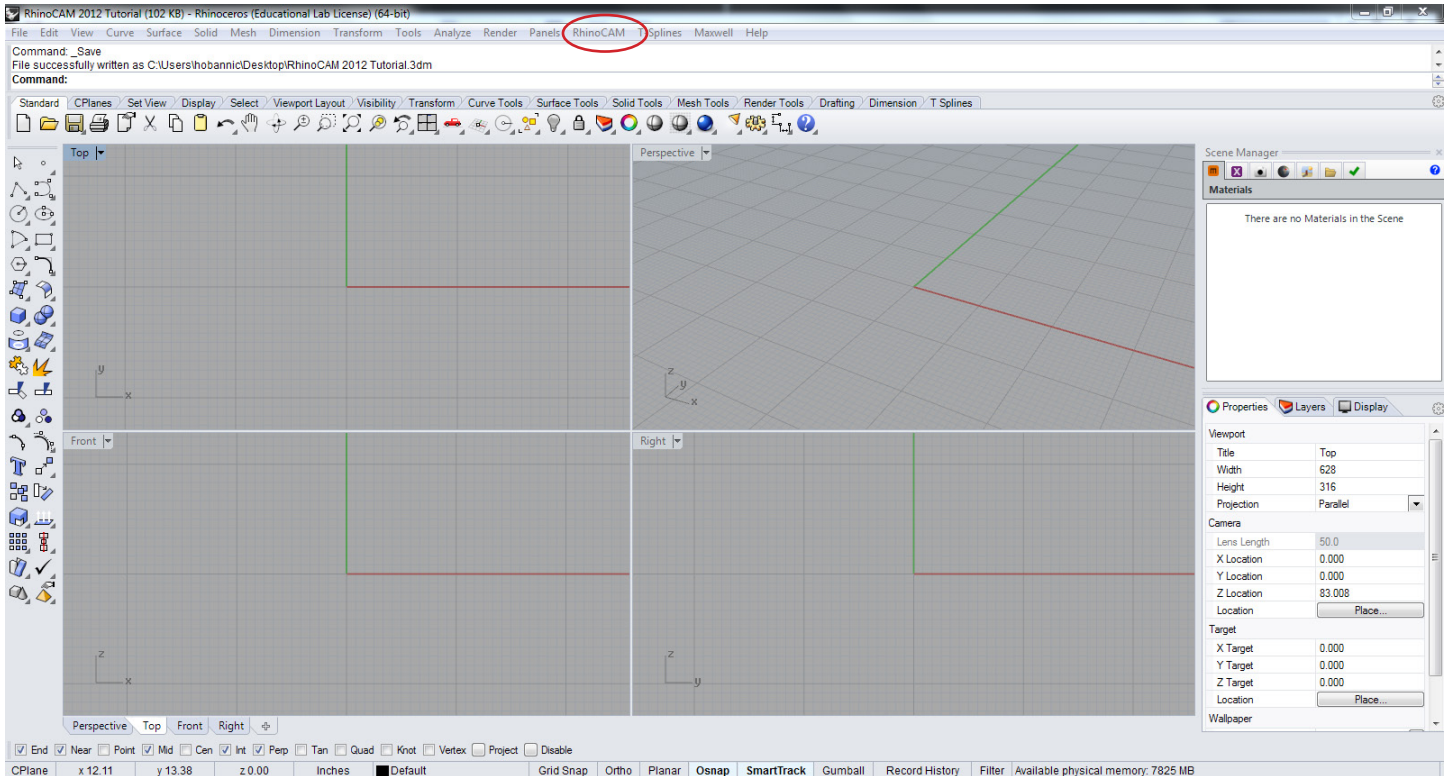
## 2\_AXIS MACHINING OPERATIONS

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

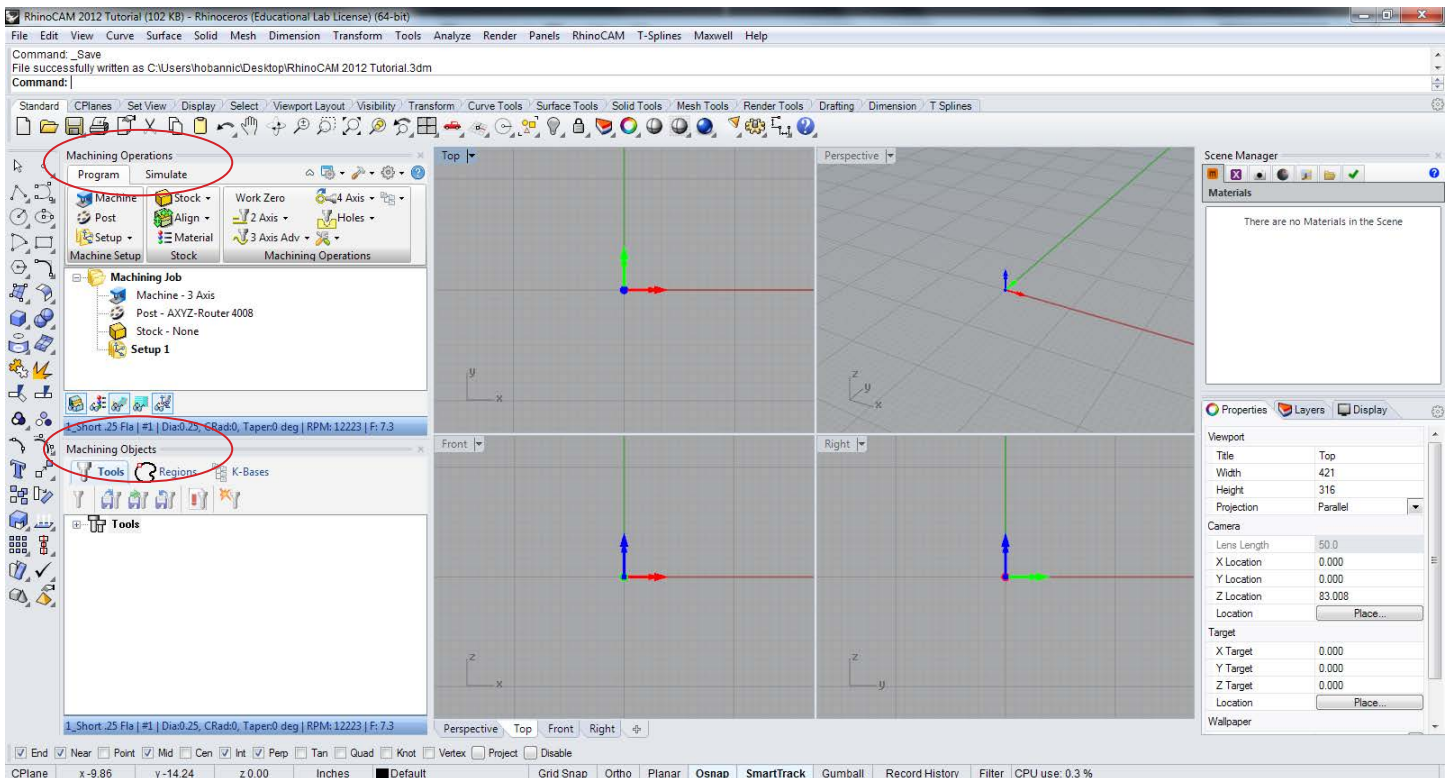
## 3 AXIS MACHINING OPERATIONS

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

# Rhino\_CAM Loading the Operations/Objects Menu

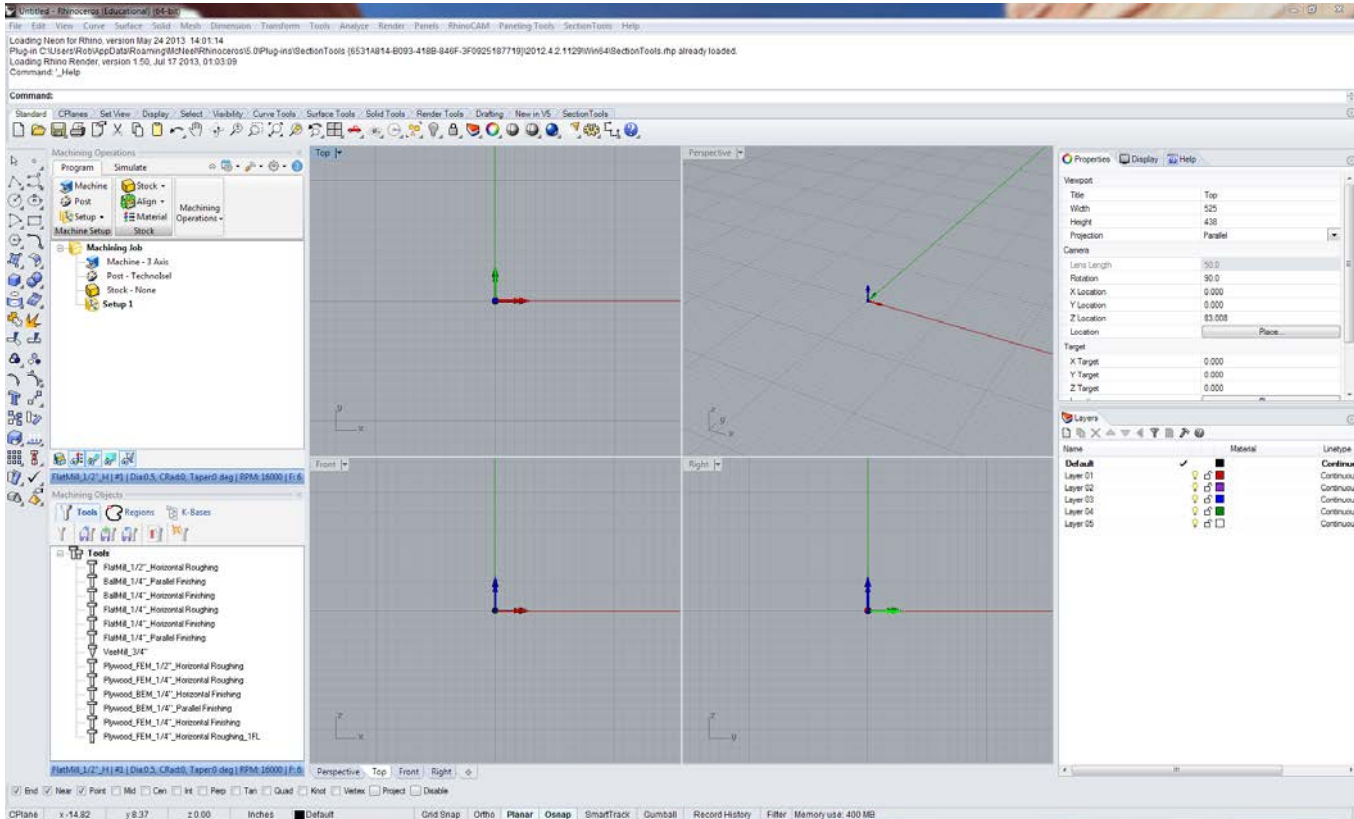


To operate RhinoCAM launch the Machining Operations Tab and the Machining Objects Tab. These can both be found in the RhinoCAM Drop down menu located along the top of the Rhino menu.

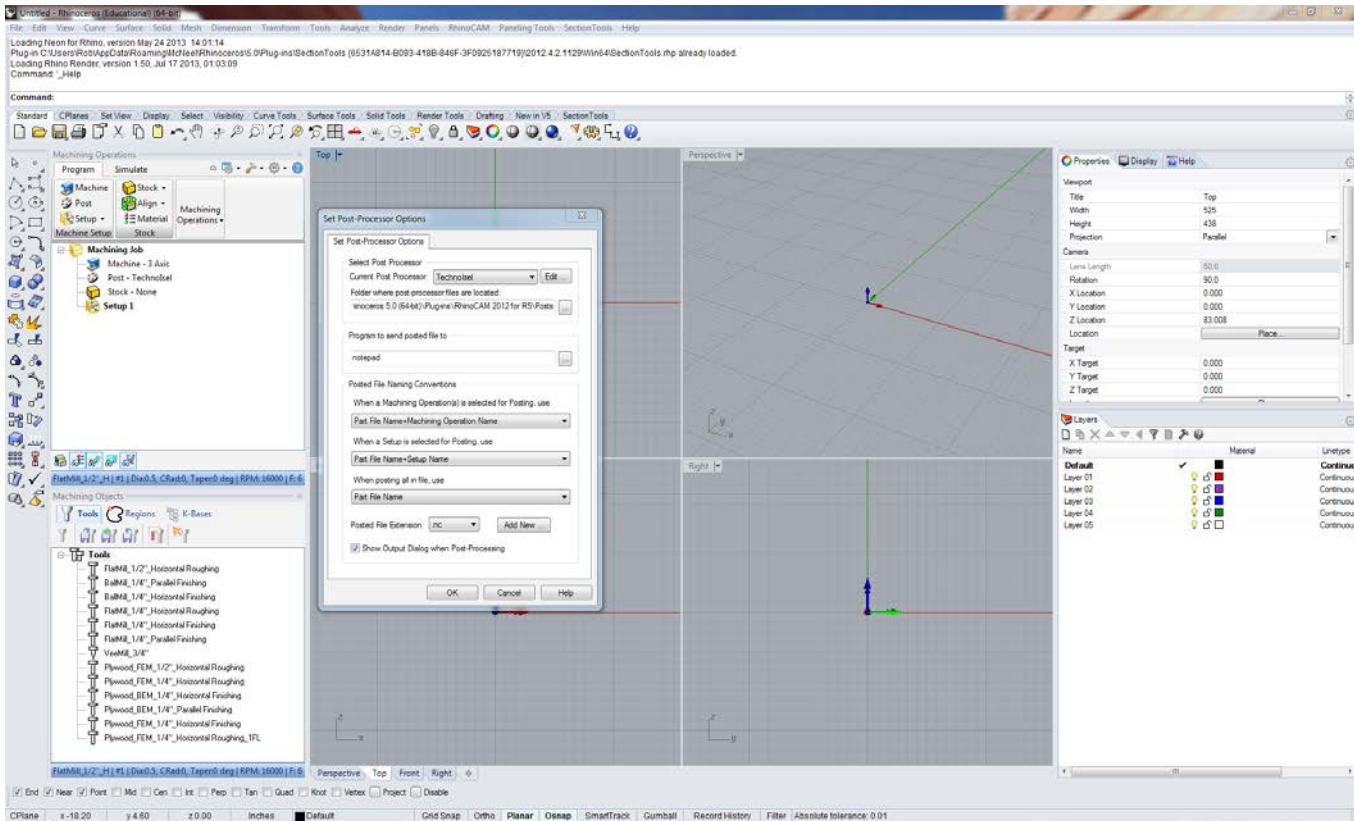


Once launched you can dock the two menus on the left side of the screen, both menus are necessary for setting up and running milling operations.

# Rhino\_CAM Loading the Post Processor

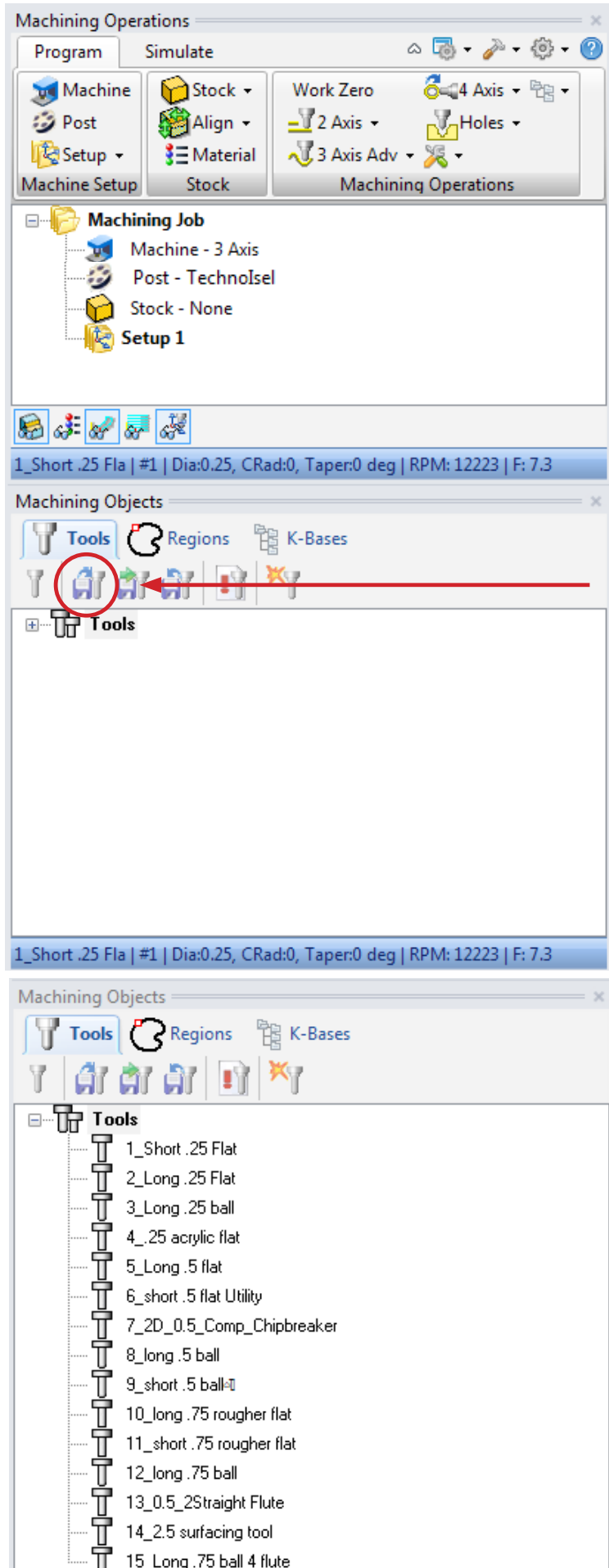


To begin first you should set the Post Processor to TechnoSel (our 3-axis router). To set the Post-Processor, click on the “Post” button. This will bring up a pop-up window allowing you to change the machine post language.



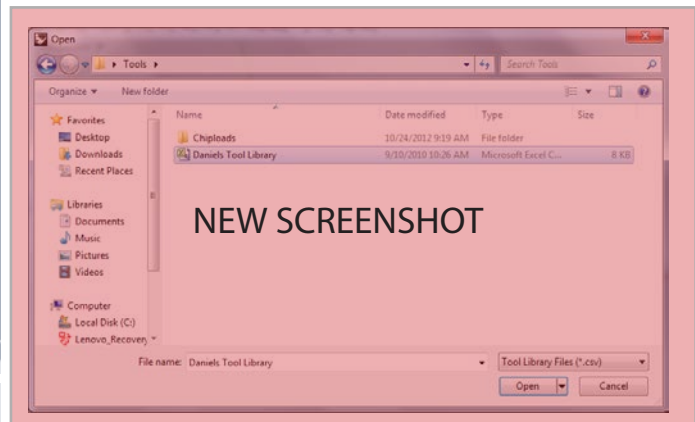
Once you have set the Post Processor to TechnoSel, click “OK”.

# Rhino\_CAM Loading the UMass A+D Tool Library



To load the UMass A+D Tool Library, copy the CSV file you were given during orientation to your computer into the folder Program Files>RhinoCeros 5.0>Plugins>RhinoCAM 2012 for R5>Data. The tool library is an established tool set of the end mills we have available in the Router Lab. Later in the tutorial you will learn how to create and edit your own end mills in RhinoCAM.

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

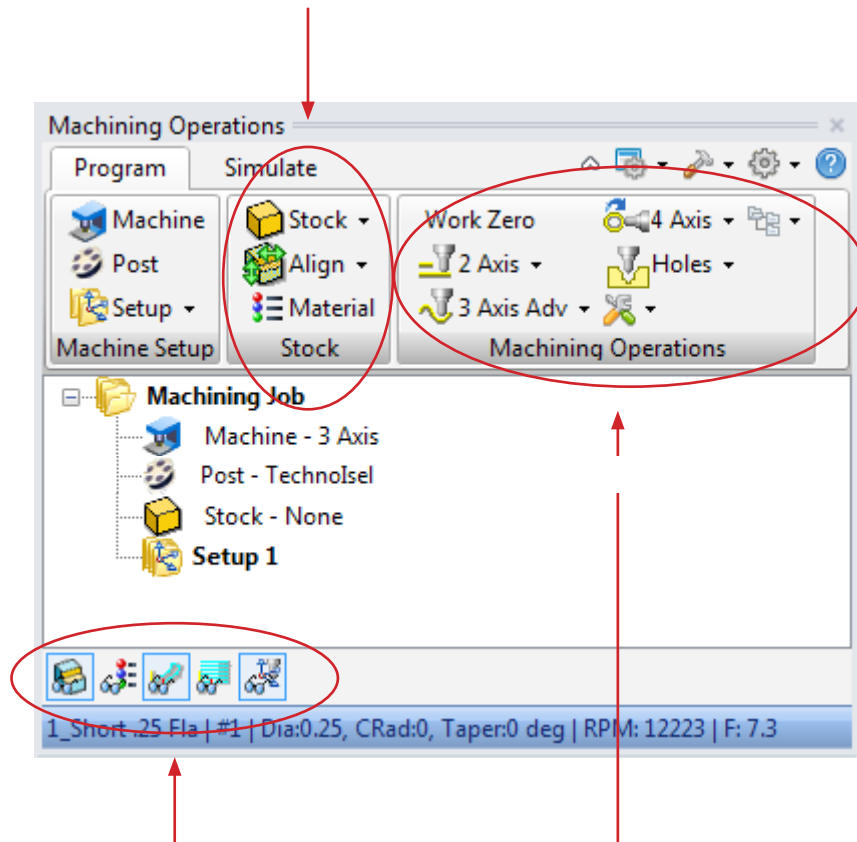


Once loaded, the library will display all the tools available in the lab.

# Rhino\_CAM 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.

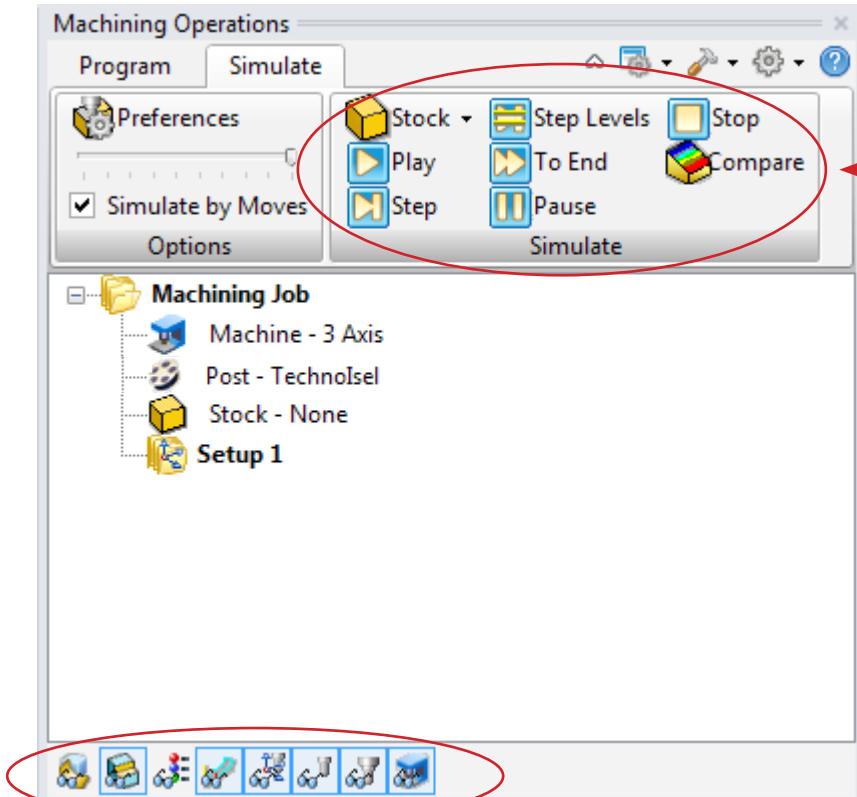
The stock align and material menu allow for the creation of stock, alignment to the working surface and the material texture of the stock. The stock dropdown menu and creation of stock is covered on page 6 along with setting material texture.



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.

# Rhino\_CAM Interface and Layout



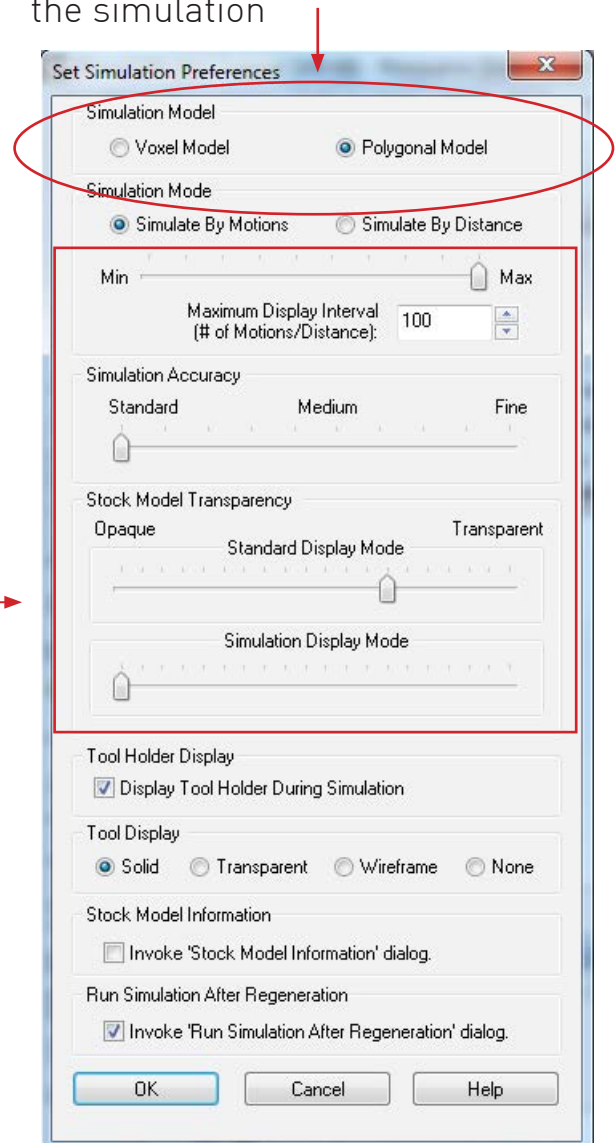
The Simulate Menu allows simulation and playback of the established milling operations. You can process by playing the Milling operations or stepping through the set incrementally with the step button. Pausing will allow you to fast forward to the end with the 'To End' button.

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 you to toggle the part, stock, material, toolpaths, Machine CSYS, Tool, Holder and Machine Visibility, respectively.

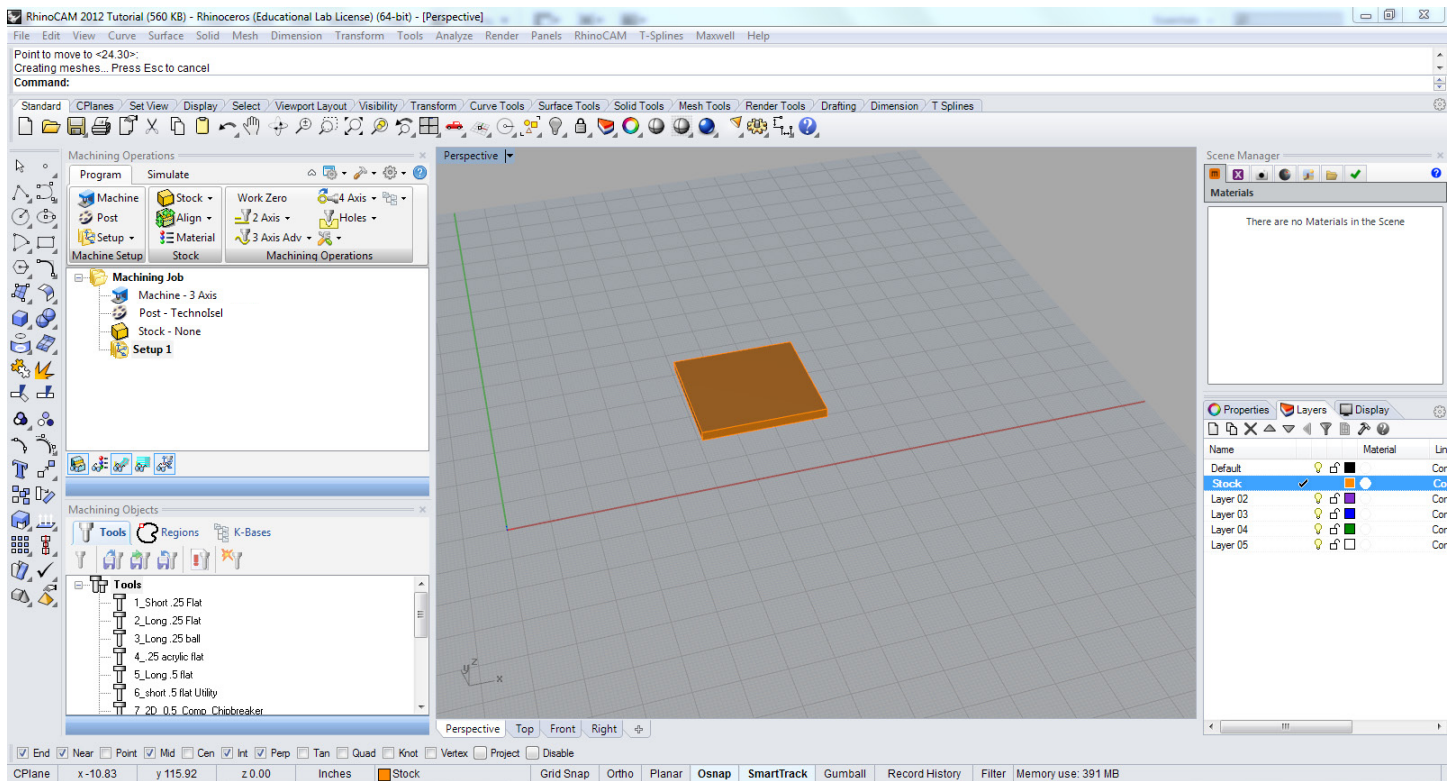
The simulation accuracy can be shifted between standard and fine. This will increase resolution on the simulated milling paths. On a large model with multiple complex milling paths, switching to a voxel model with standard accuracy and reducing the display intervals will speed up the simulation where greater accuracy will give too slow a simulation.

In general, it is useful to keep the stock partially transparent to keep track of its location relative to the parts.

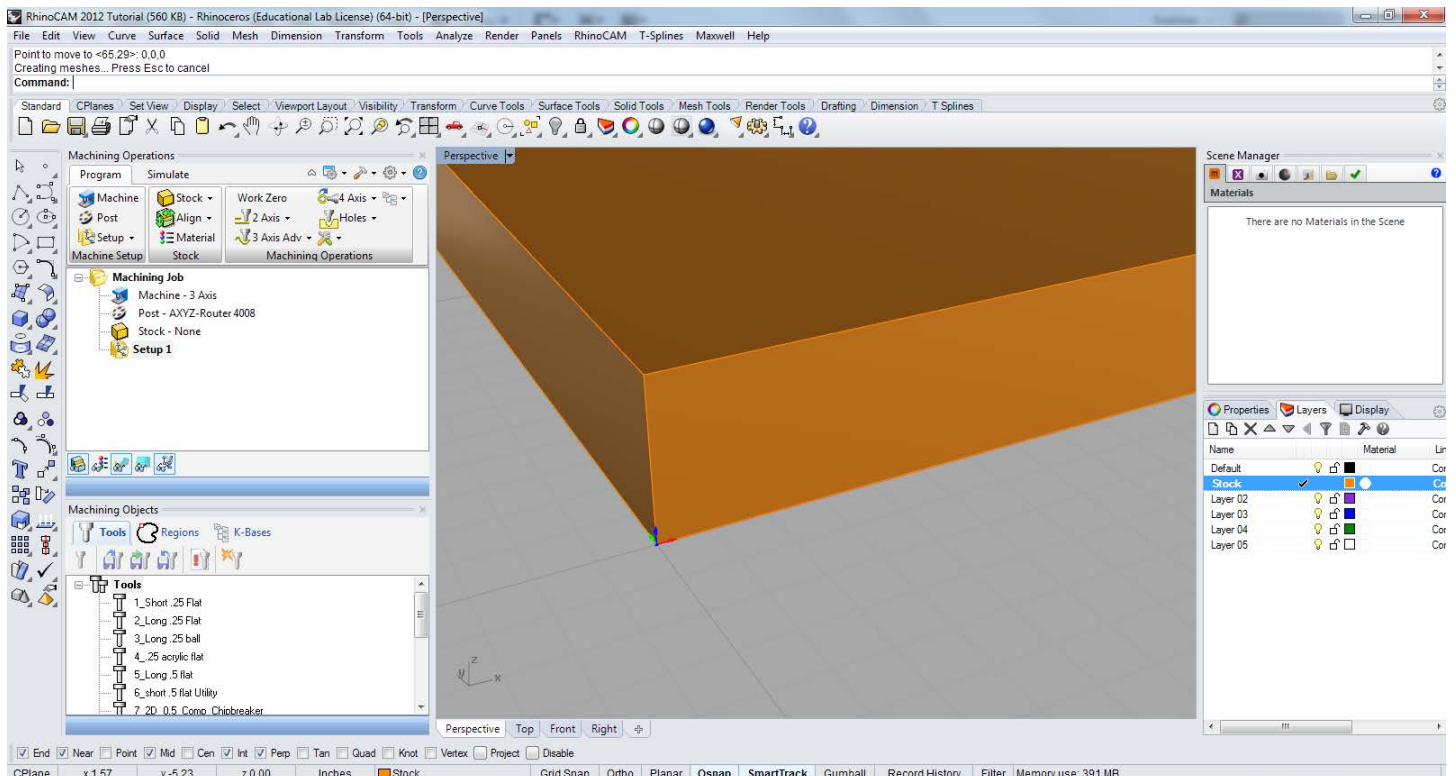


# Rhino\_CAM Stock Set Up

Creating stock in RhinoCAM is done by creating a solid polyobject (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.

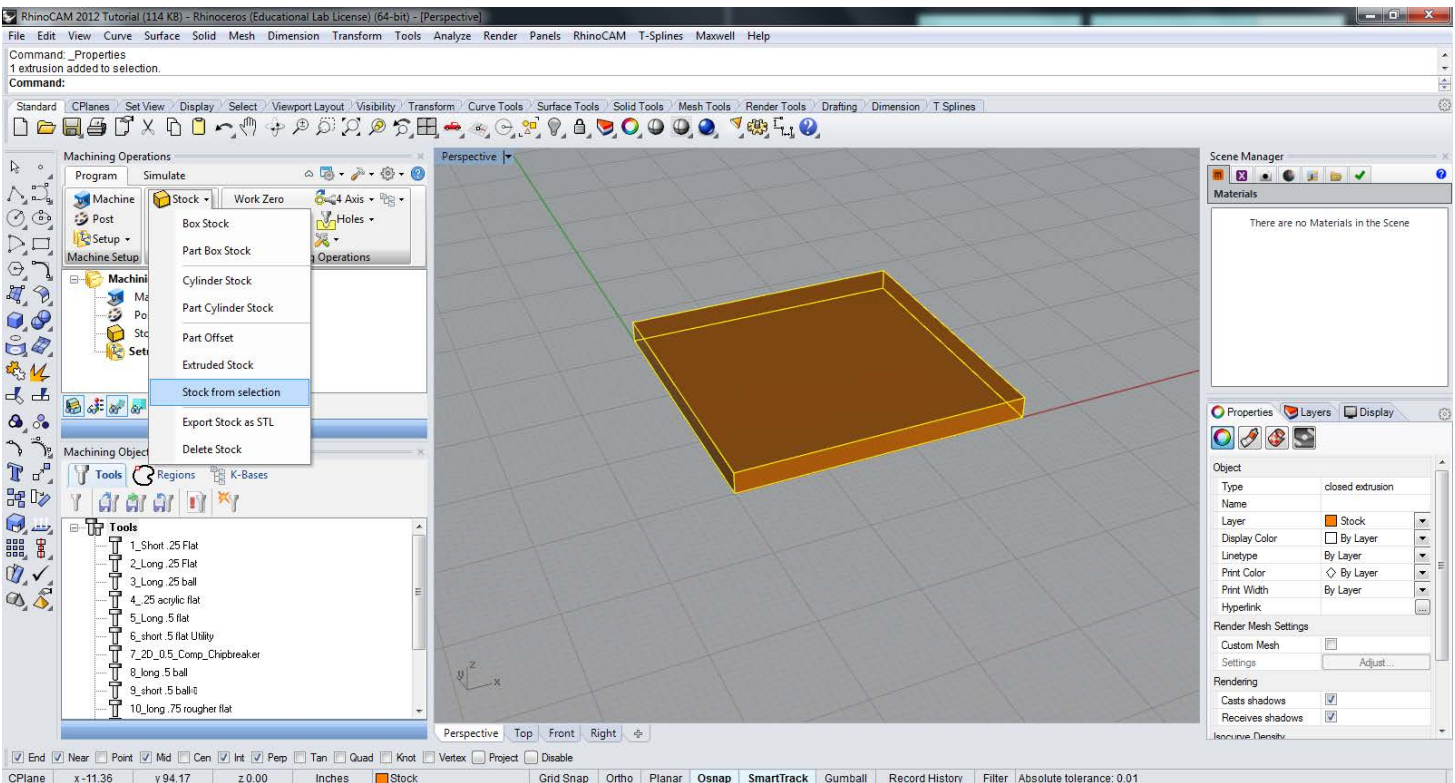


Move the stock to the origin in rhino (0,0,0) this corresponds to the origin of the CNC. The Red, Green and Blue arrows indicate the X,Y and Z direction of the CNC bed.

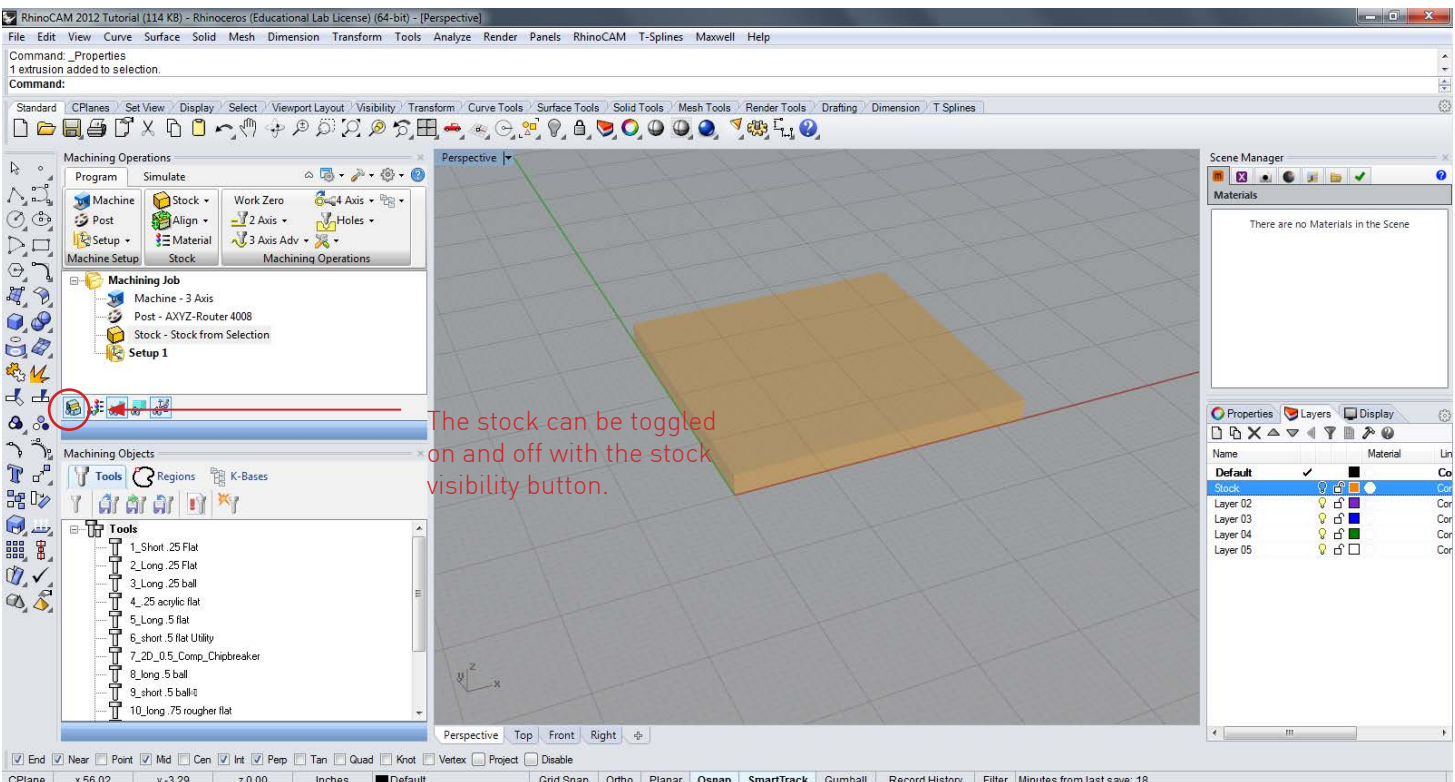


# Rhino\_CAM Stock Set Up

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.

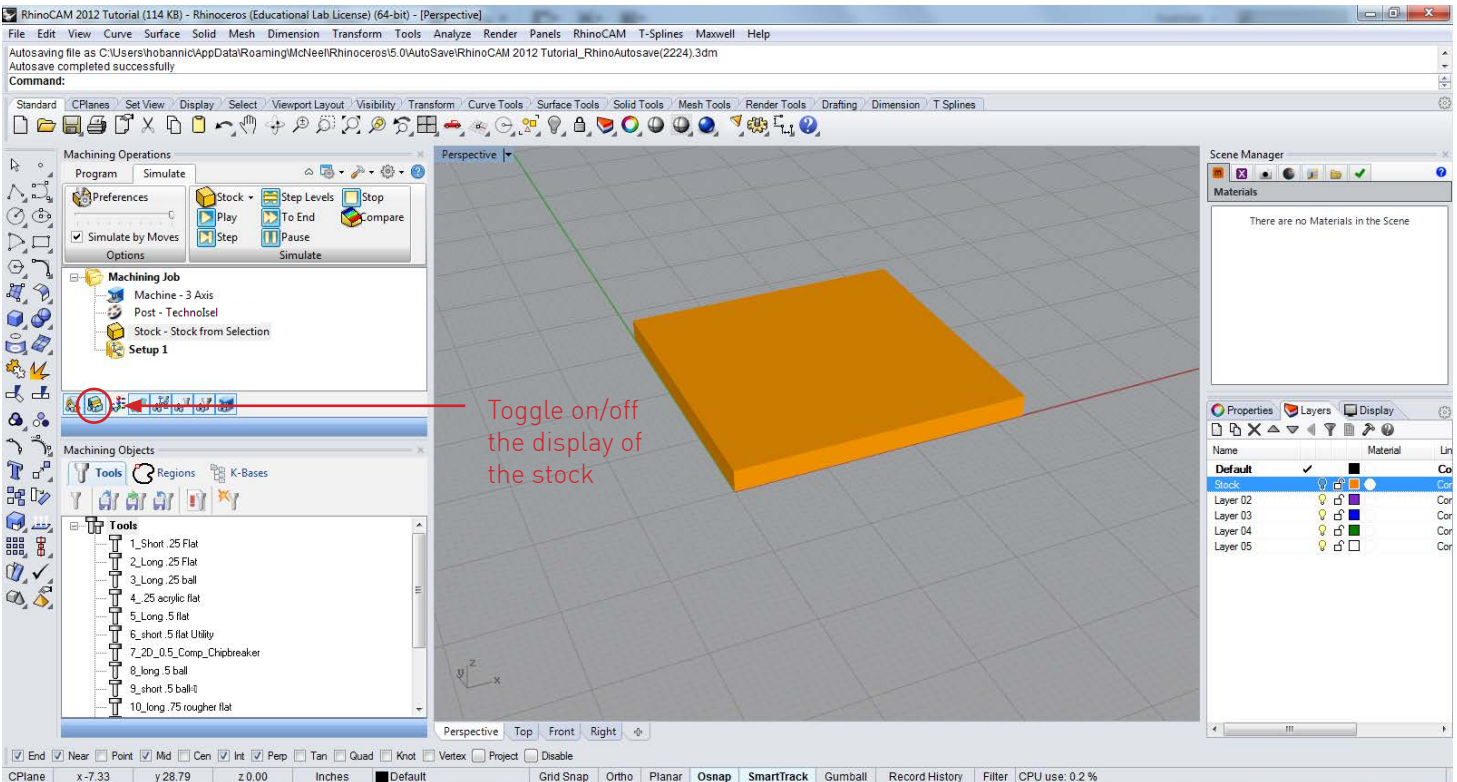


Once the stock has been created you can turn off the stock layer, the ghosted stock you see in the program menu is the RhinoCAM stock. All milling paths and operations will be applied to this stock.

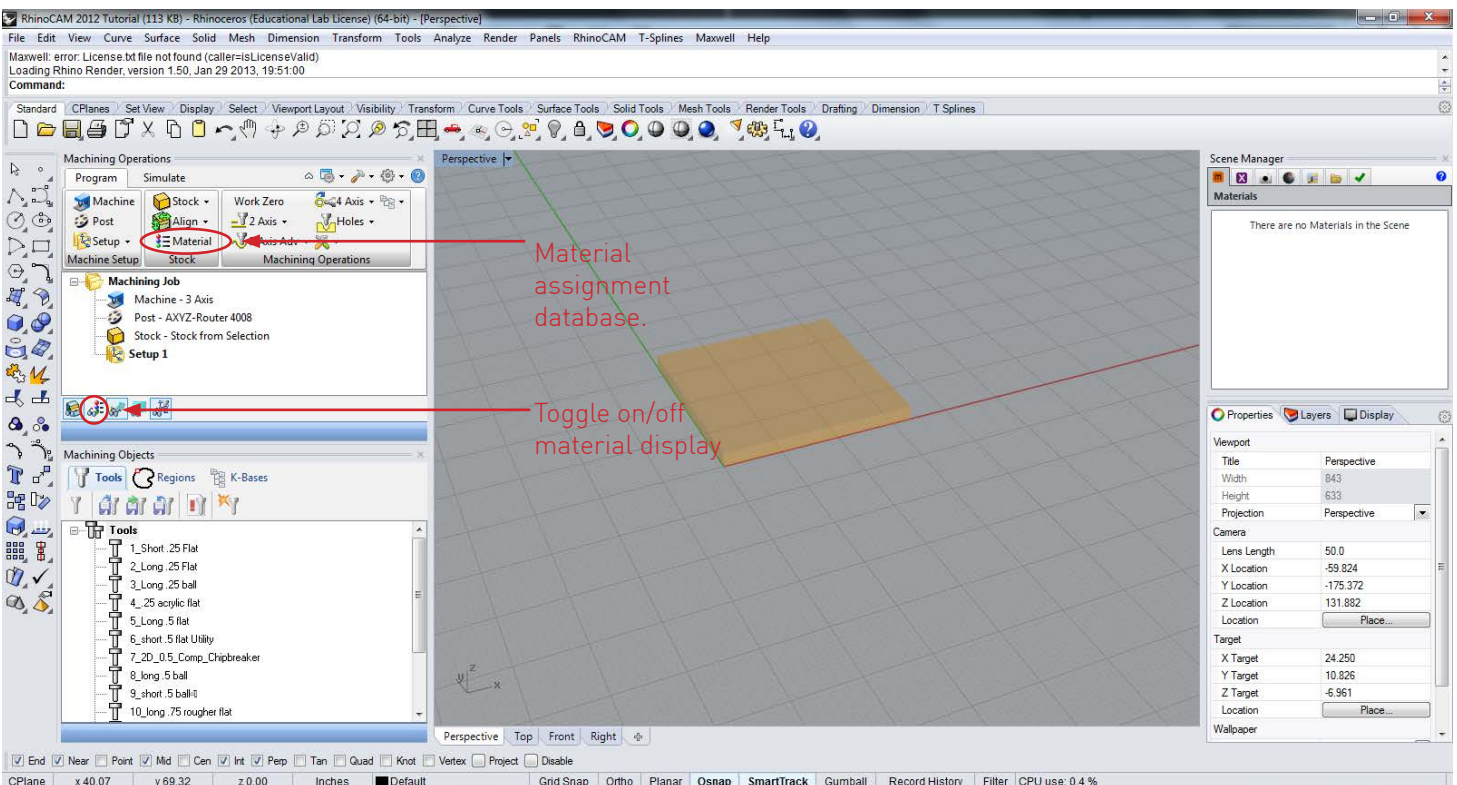


# Rhino\_CAM Stock Set Up

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.

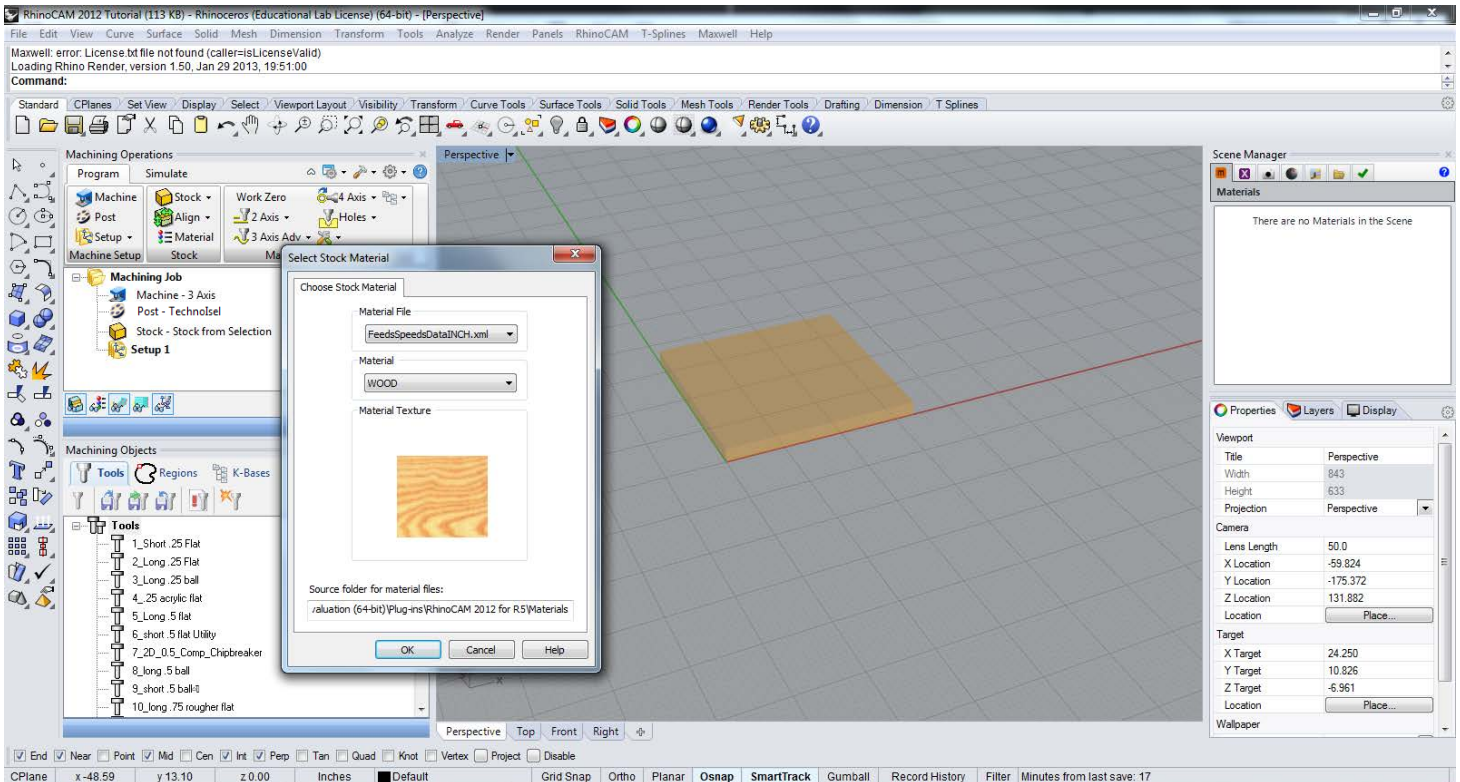


Material type can be assigned to the stock on the Program Tab, press the material button to access the stock material database in RhinoCAM.

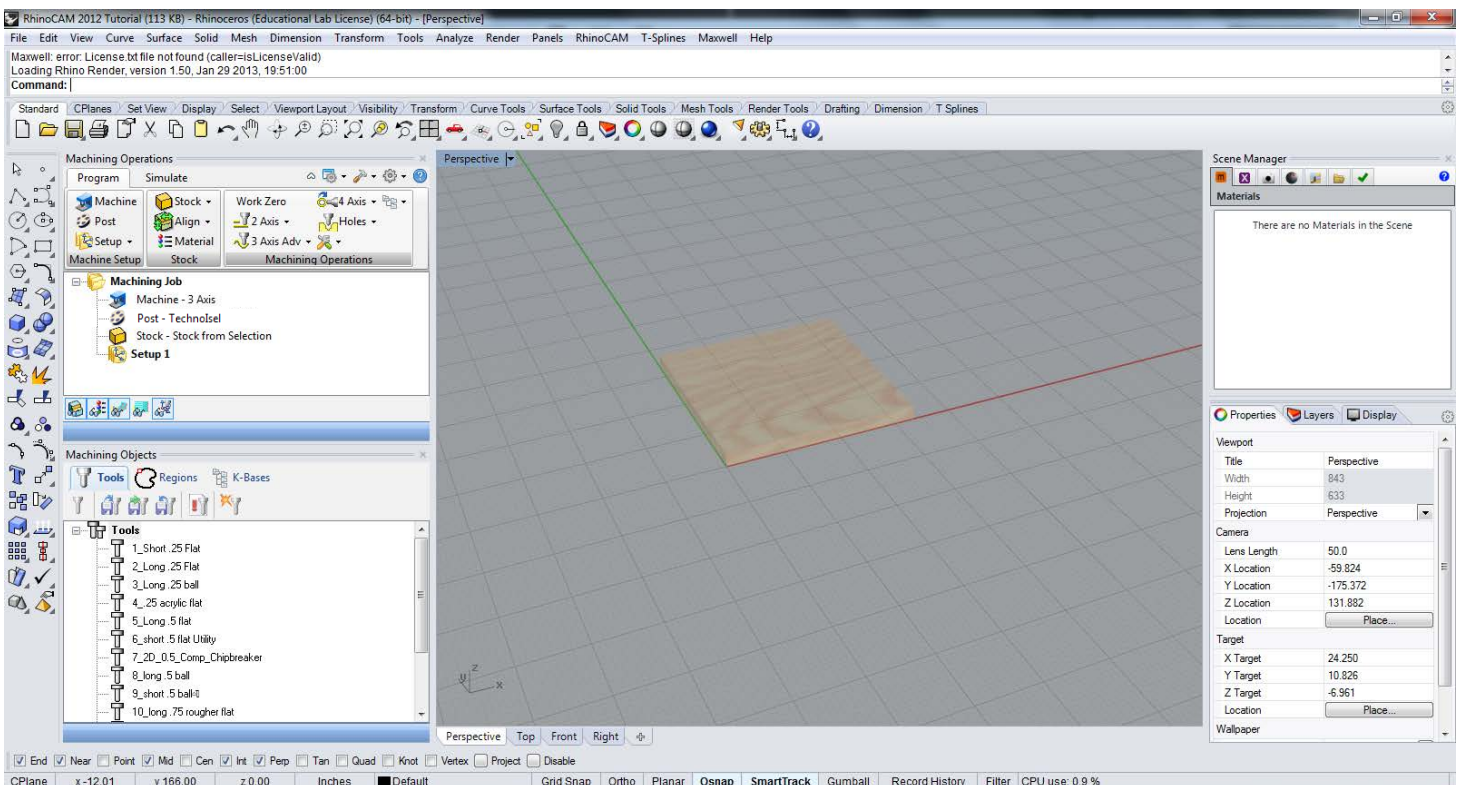


# Rhino\_CAM Stock Material Assignment

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

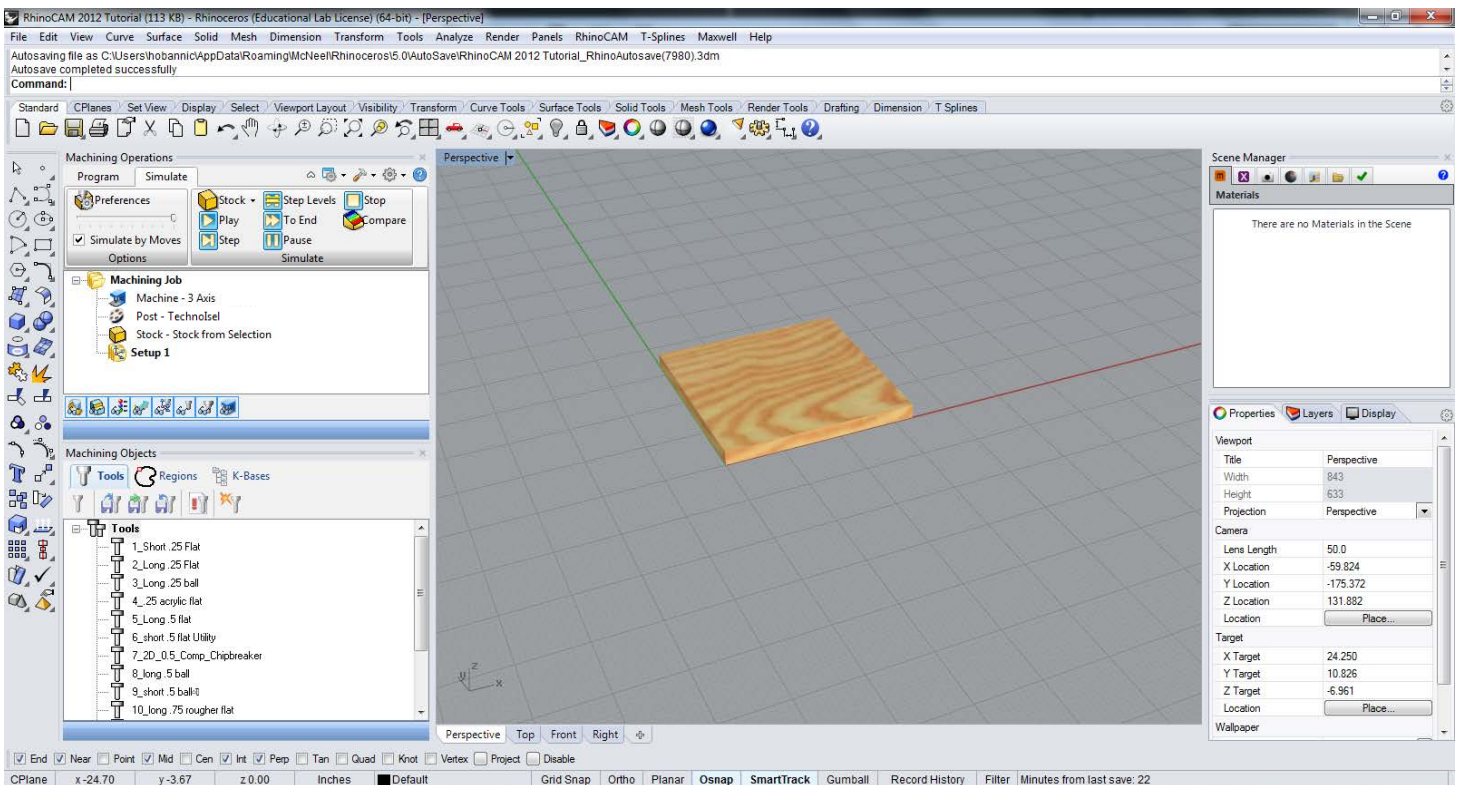


While in program tab when the material visibility is turned on the stock appears ghosted with assigned material type.



# Rhino\_CAM Stock Material Assignment

Under the simulate tab the material will appear fully rendered.



## Rhino\_CAM Setting up a Milling File

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.

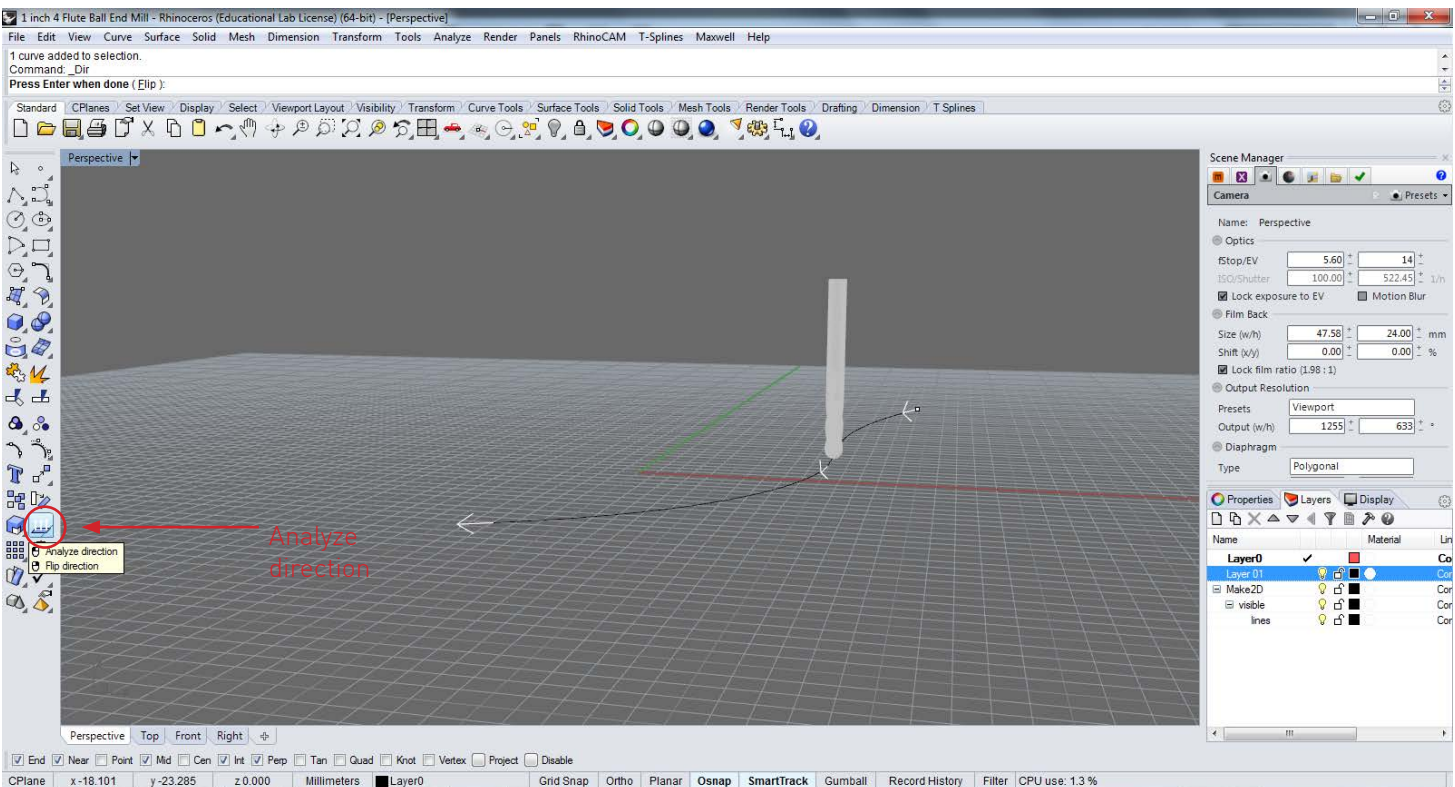
Different end mill sizes and types will leave different tooling effects on the material surface. RhinoCAM will simulate these paths and effects based on selection of end mill and tooling paths.

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.

# Rhino\_CAM 2 Axis Machining Operations\_ Engraving

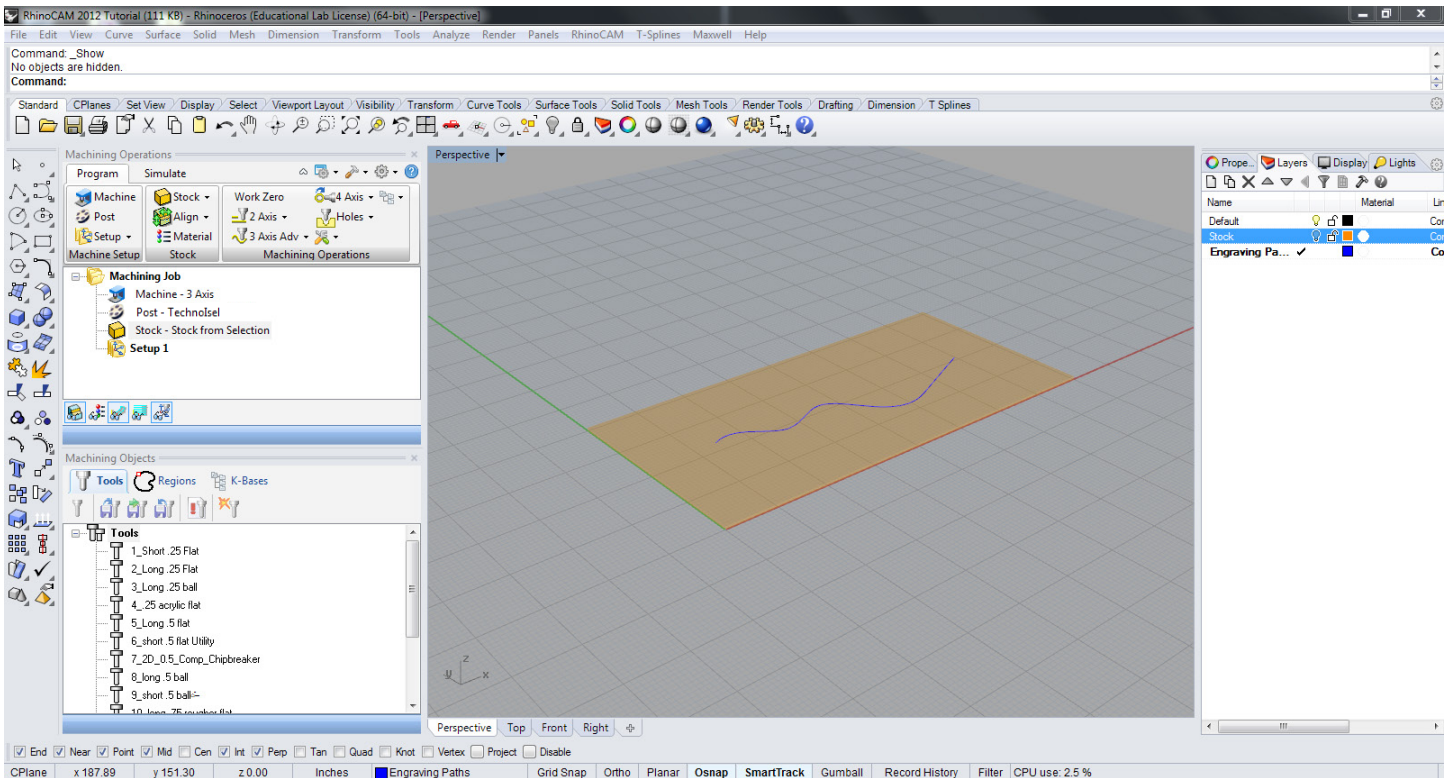
Engraving Milling paths follow curves along the center of the end mill to the assigned curve or line. The end mill cutting direction will follow the direction of the curve or line, to find the direction of the curve in Rhino you can enter the command `_Dir` or press the analyze direction button.

Tip: If the desired toolpaths were generated in Rhino or Grasshopper, use the Engraving Operations to override the RhinoCAM algorithms (See <http://vimeo.com/61239086> for demonstration).

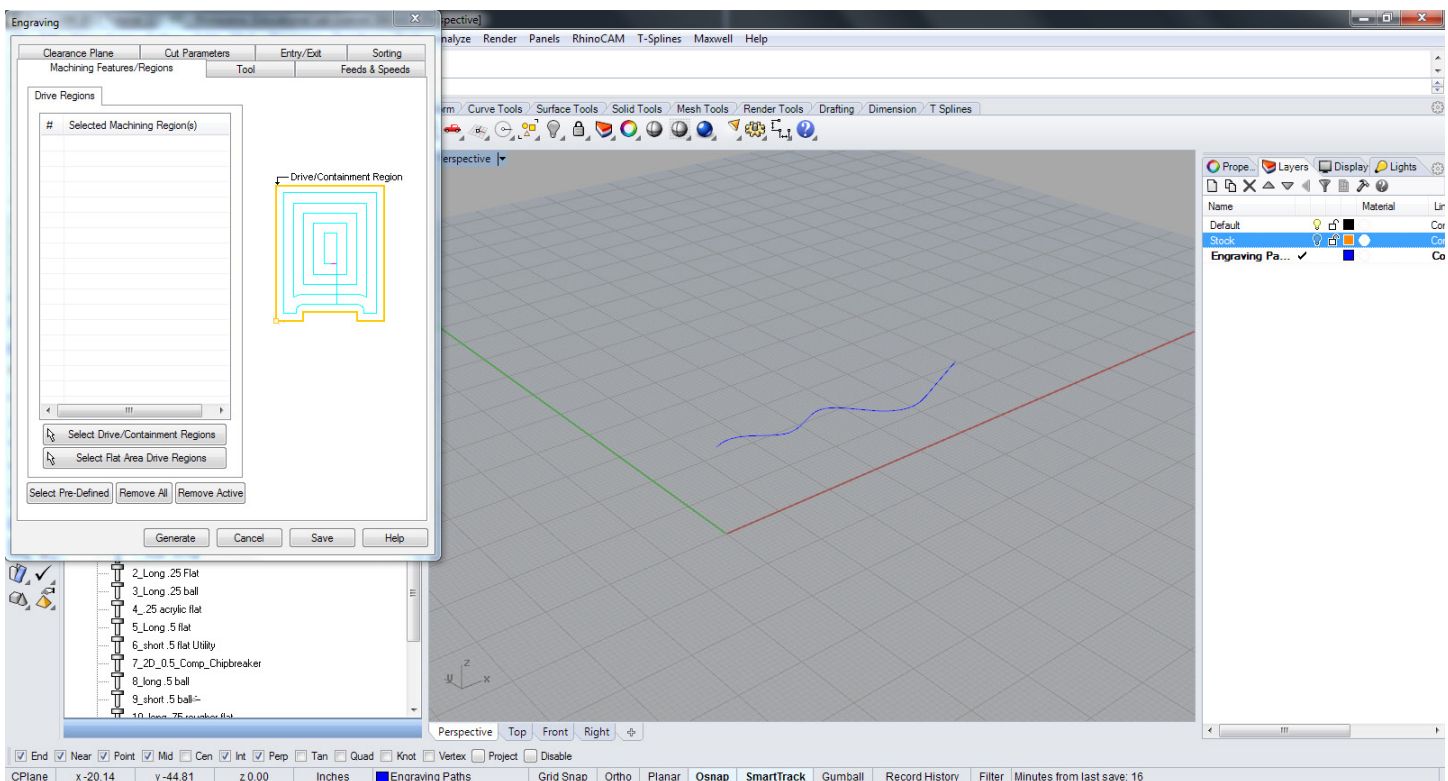


The direction of the curve can be flipped while in the Analyze direction command. These curvature directions will be important in assigning 2D milling operations as the line/curve direction will dictate the cutting direction in your milling operations.

# Rhino\_CAM 2 Axis Machining Operations\_ Engraving

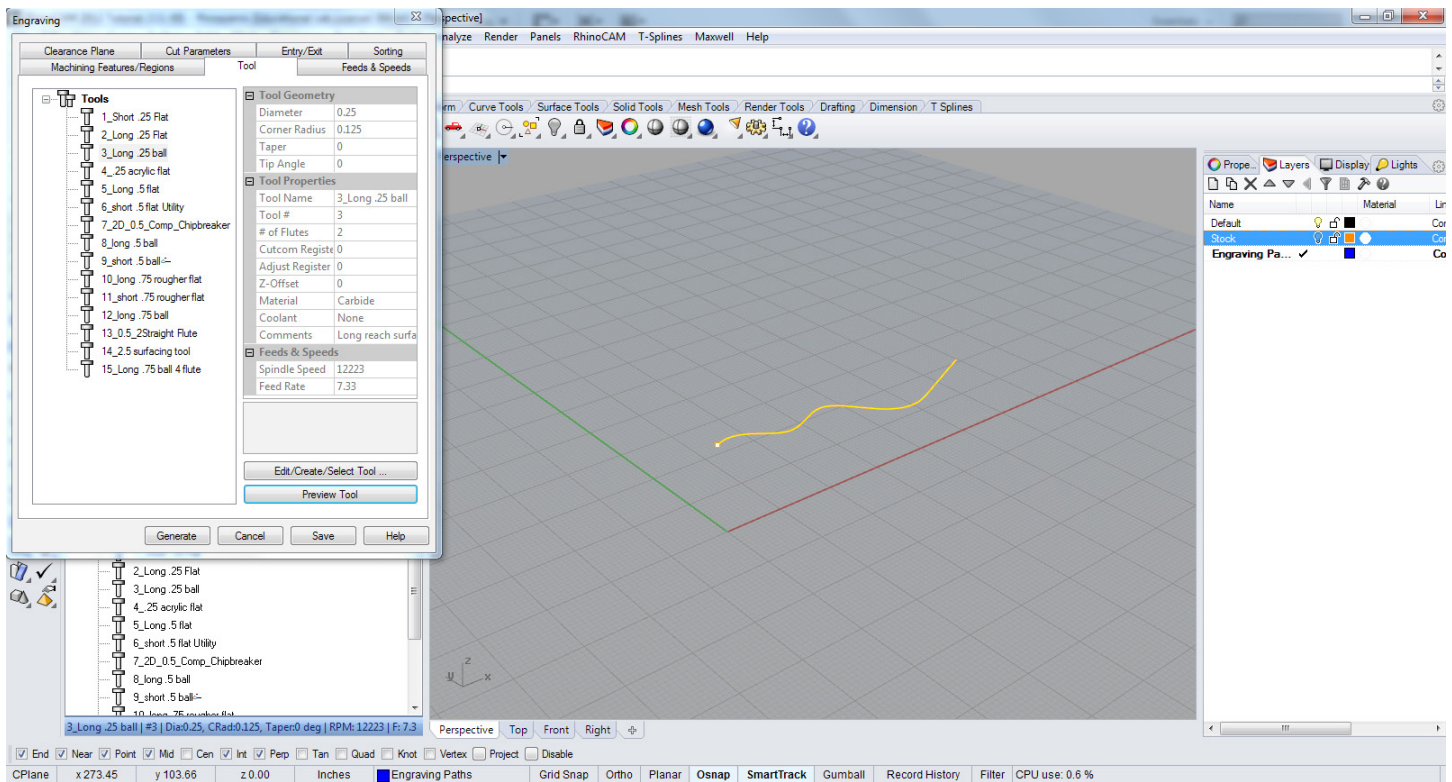


The established engraving path is aligned to the top surface for the stock material. In the input parameters in the machining operations for engraving the cutting depth can be established.

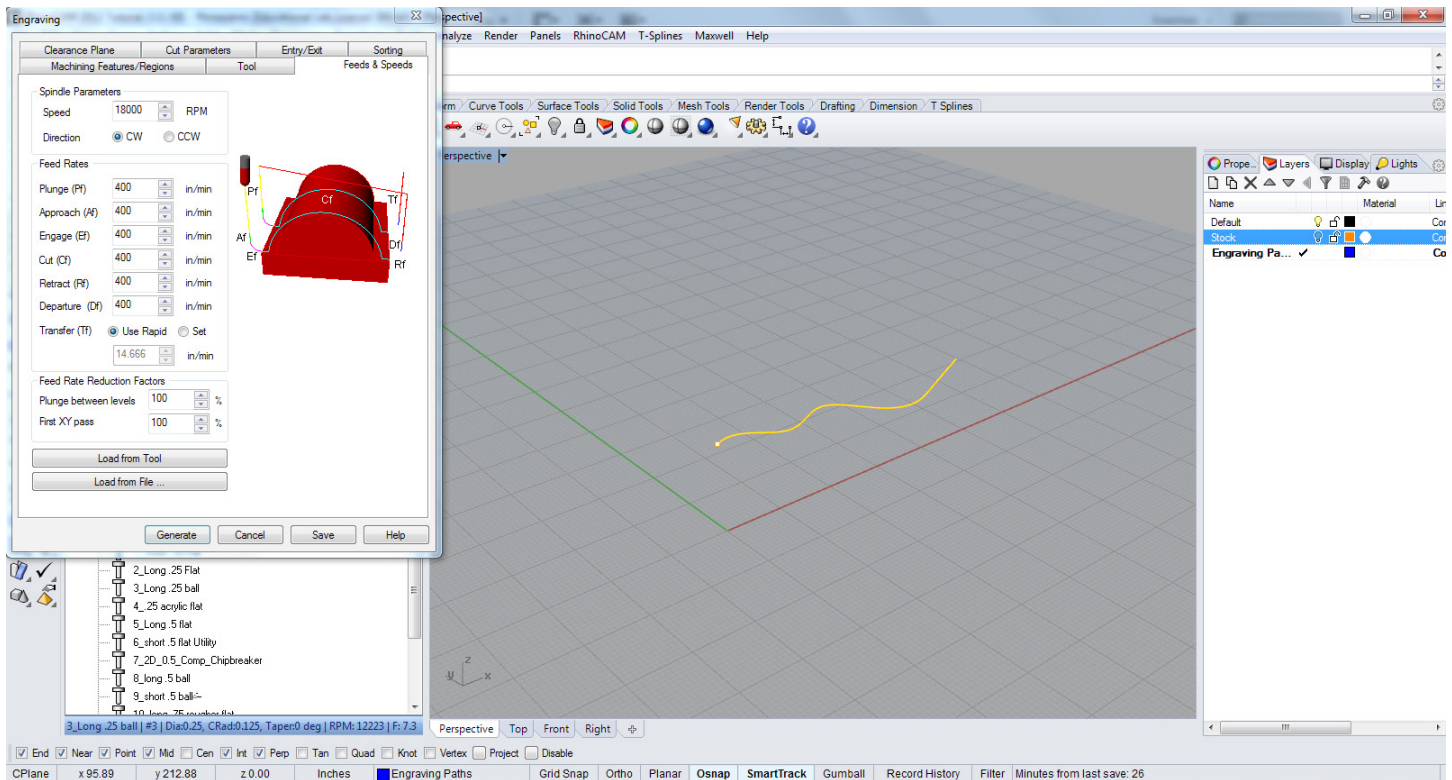


Select Engraving from the 2D milling operations tab, press select drive/containment region to select the engraving milling path(s) established.

# Rhino\_CAM 2 Axis Machining Operations\_ Engraving



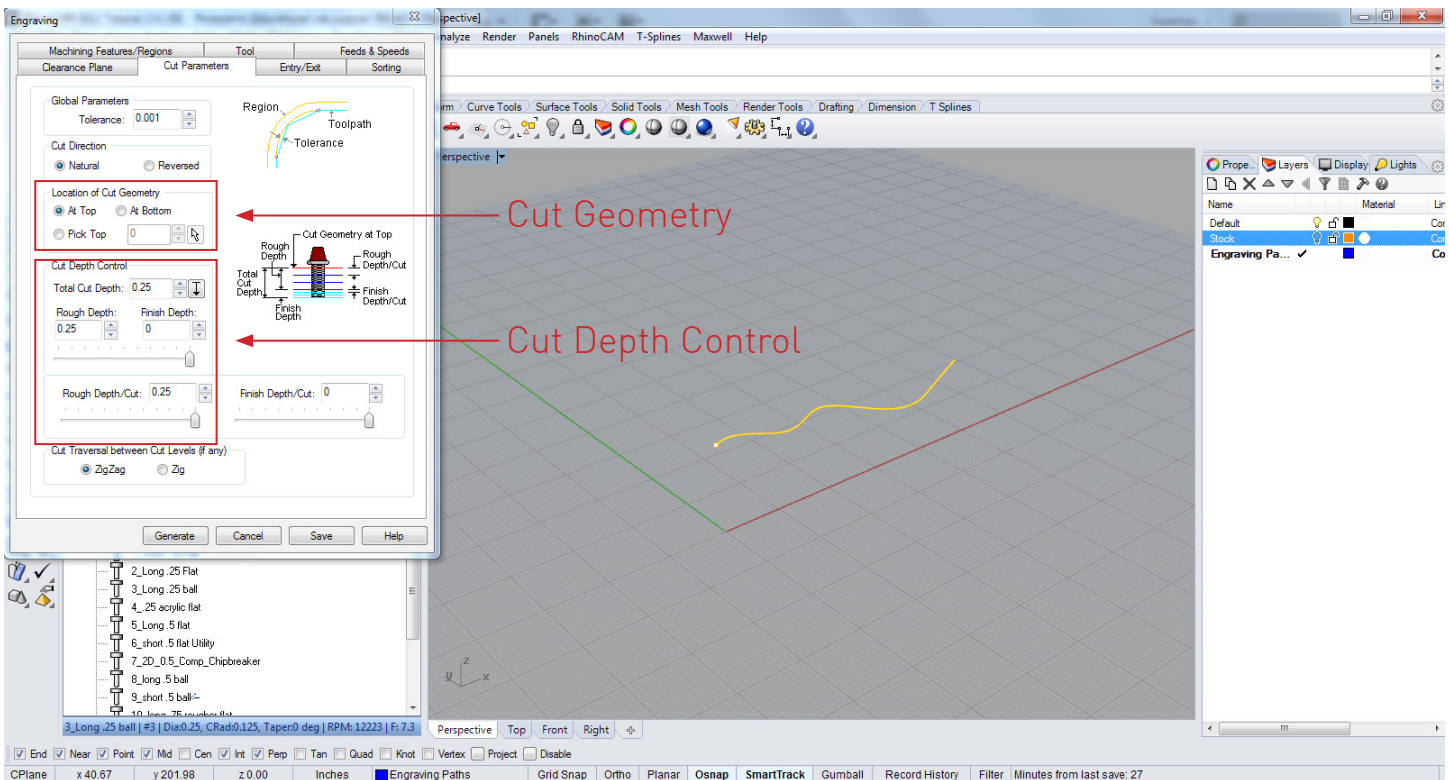
The endmill diameter selected will determine the width of the engraving line, as the endmill center point follows the selected engraving line the tooling path will be the endmill diameter from the center of the line.



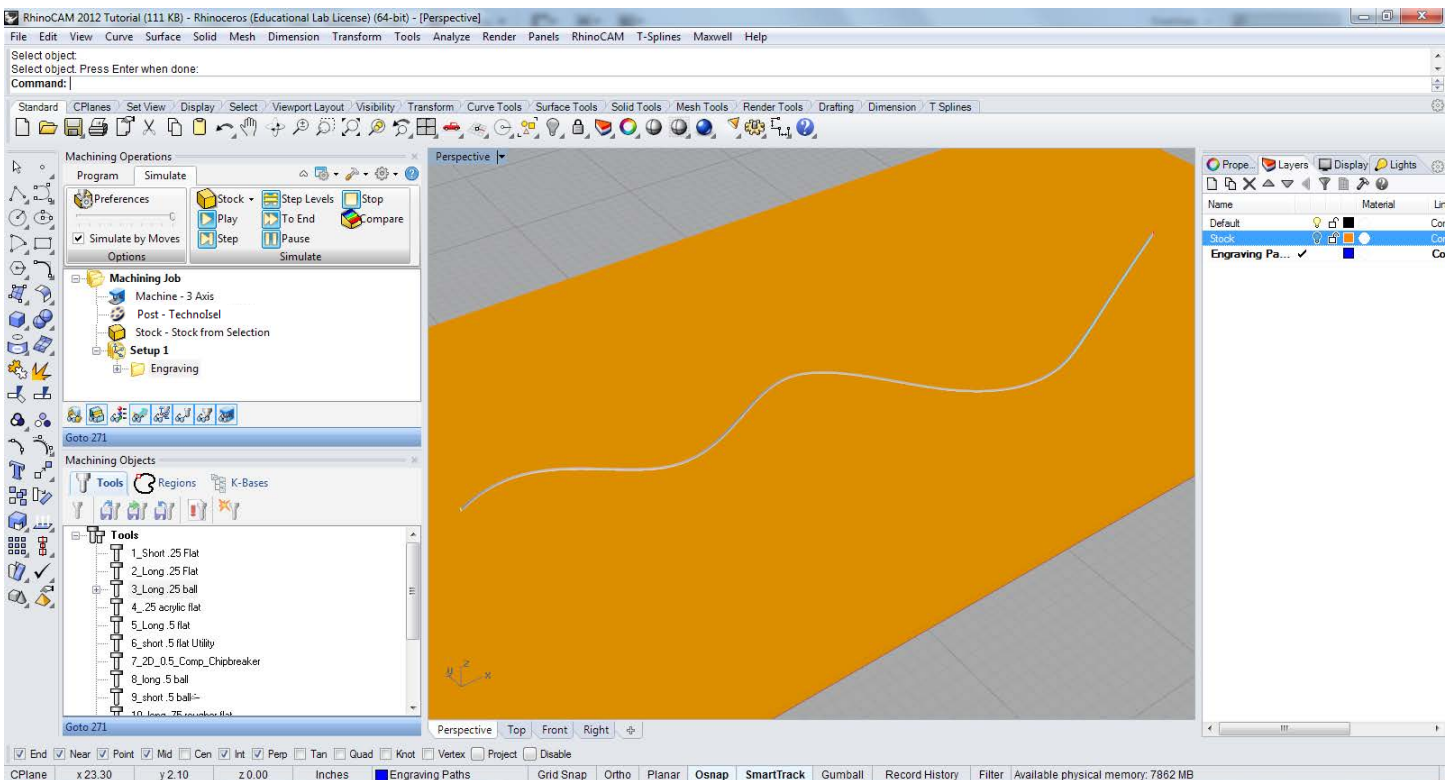
The feeds and speeds will be determined by the material being milled and the type of endmill selected. Please see the feeds and speeds section and consult the Fab Lab.



# Rhino\_CAM 2 Axis Machining Operations\_ Engraving



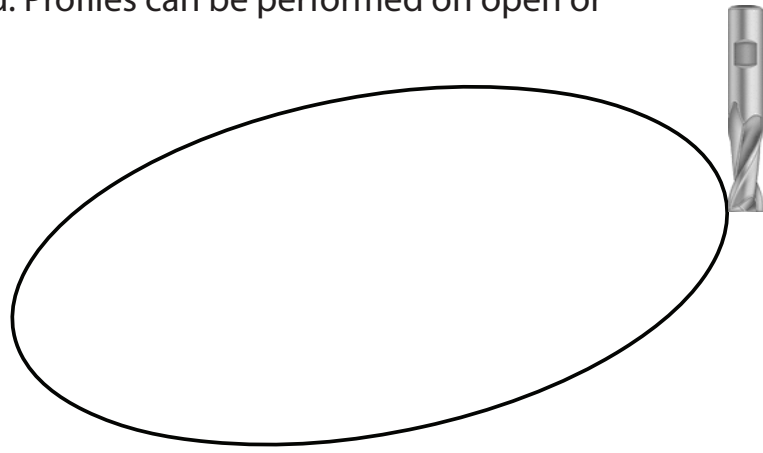
Under the Cut Parameters tab, the cutting direction, depth and geometry location is chosen. The geometry currently in the example is located at the top, the total cut depth is 1/4", this is done in one pass, for deep milling the depth can be done in multiple passes by setting the rough cutting depth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



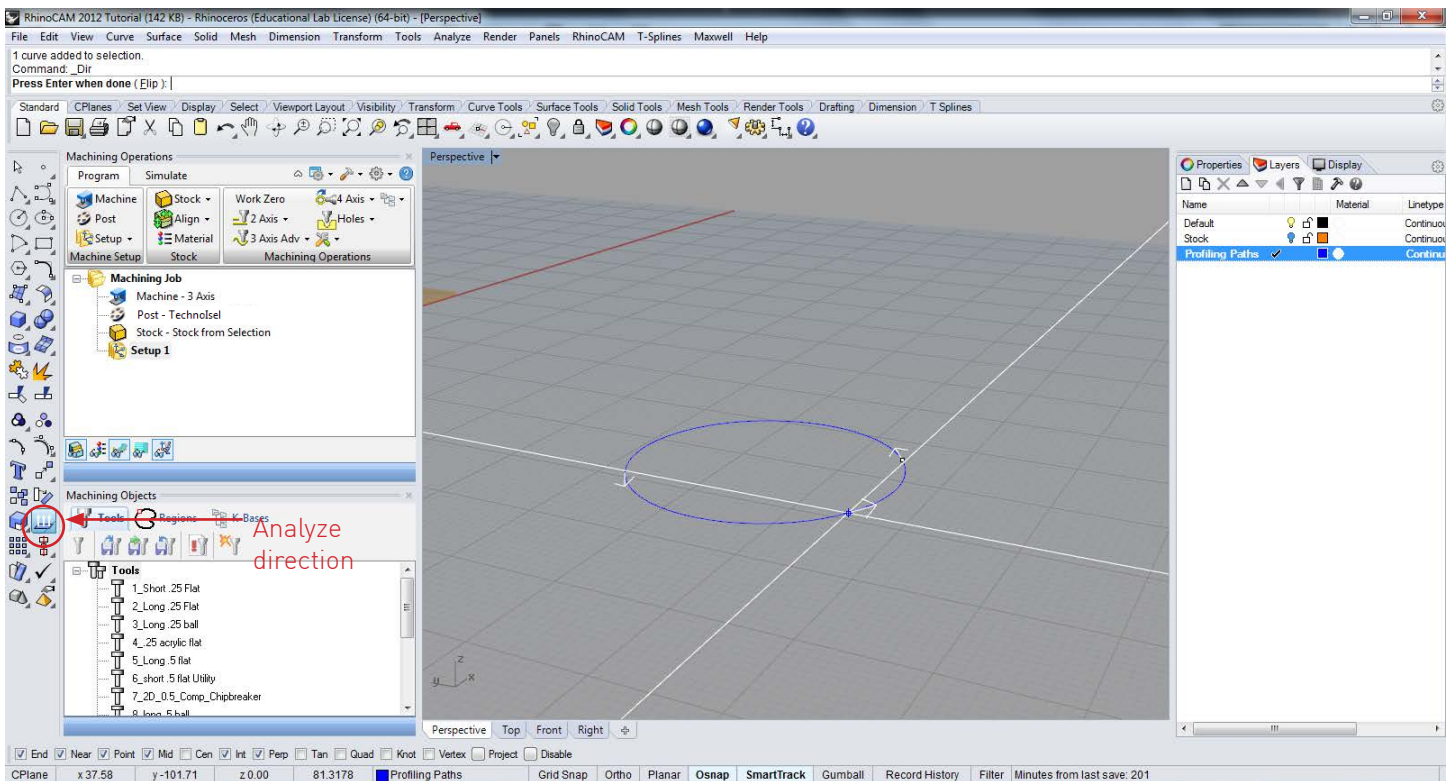
Once the milling paths have been generated select the simulate tab and run the engraving paths to see the toolpath result.

# Rhino\_CAM 2 Axis Machining Operations\_ Profiling

Profiling Milling paths follow curves along the direction normal to the curve. The endmill will by default always assign itself to the right side of the directional normal. To change the cutting direction of the profile the curve direction can be flipped in Rhinoceros or the setup parameters in the profiling pass dialog can be changed. Profiles can be performed on open or closed curves.

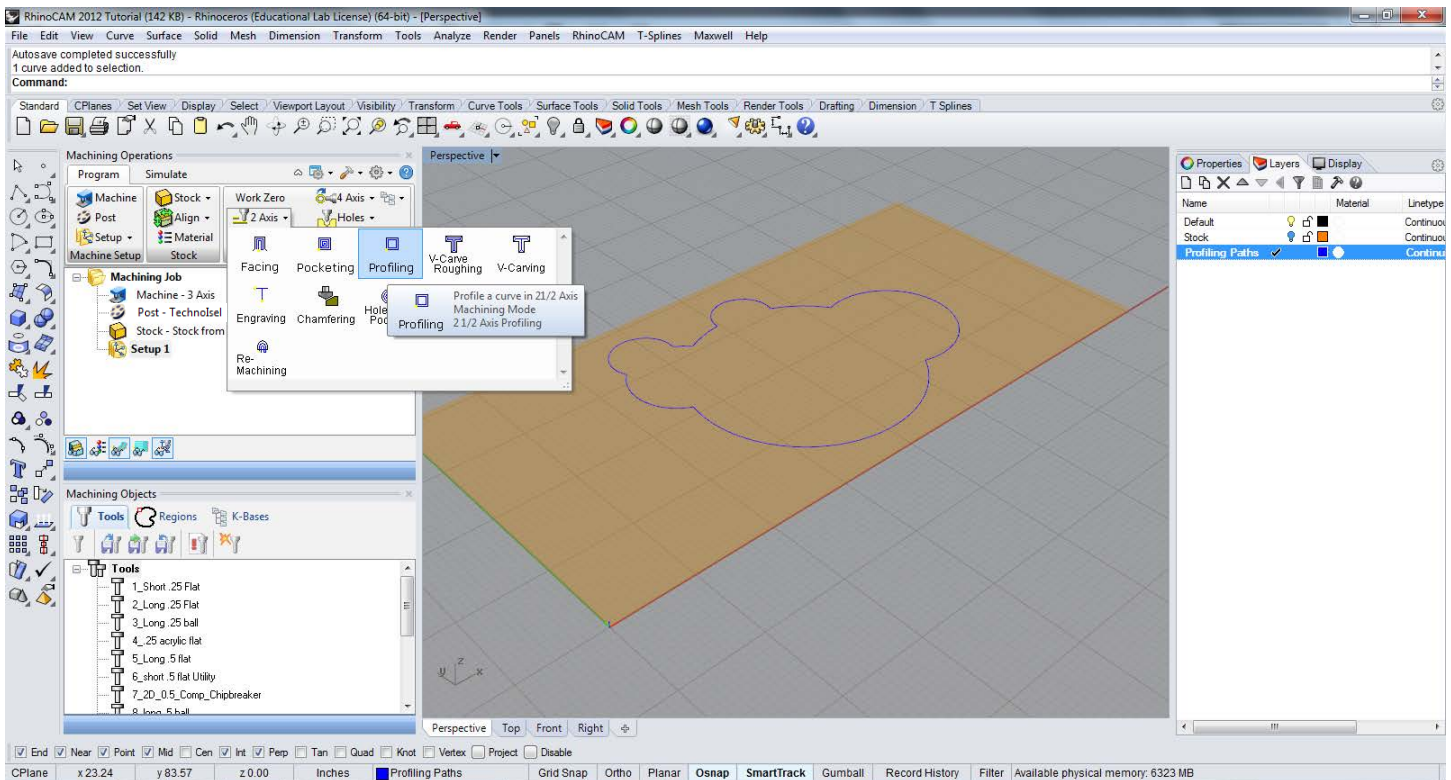


Tip: Profiling toolpaths are optimal for cutting out two-dimensional shapes on the router.

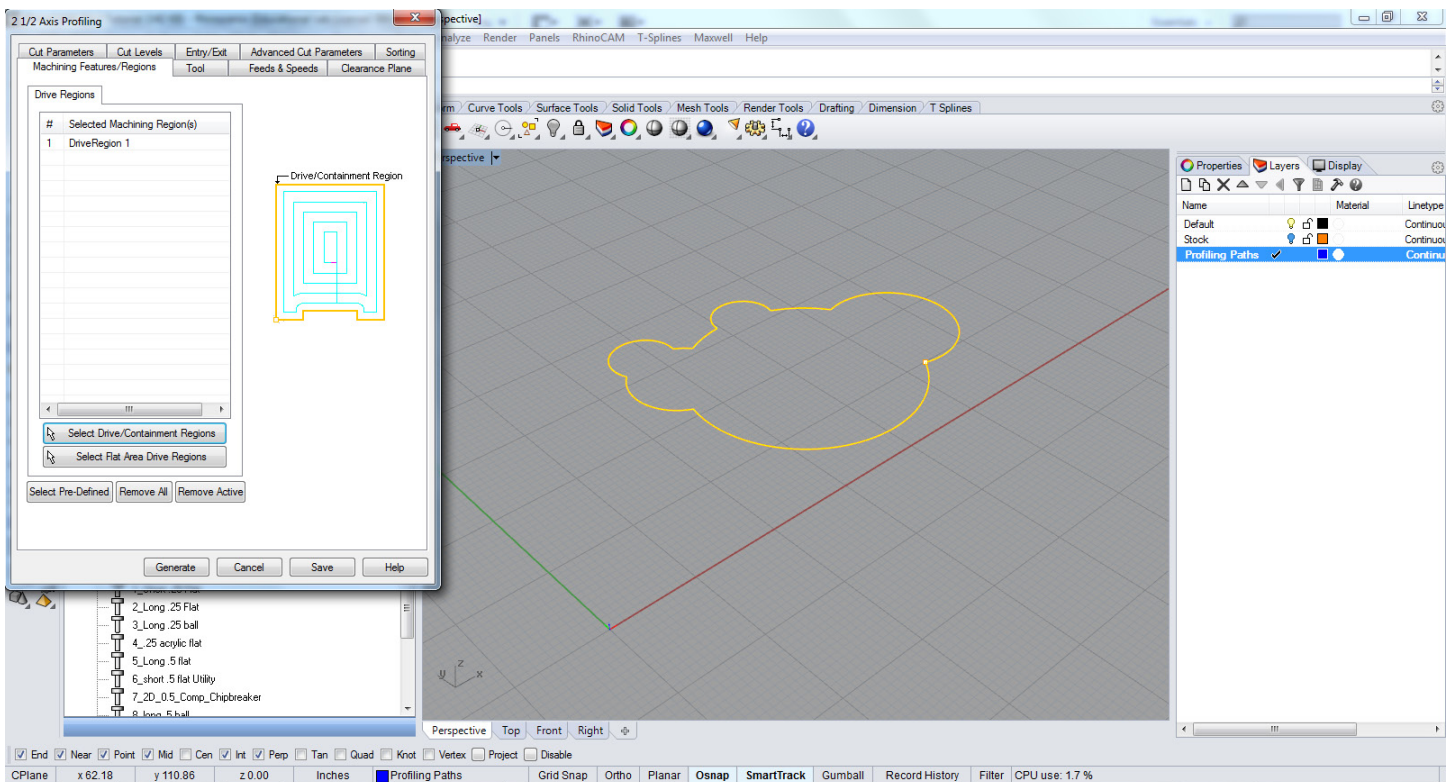


The direction of the curve can be flipped while in the Analyze direction command. These curvature directions will be important in assigning 2D milling operations as the line/curve direction will dictate the cutting direction in your milling operations.

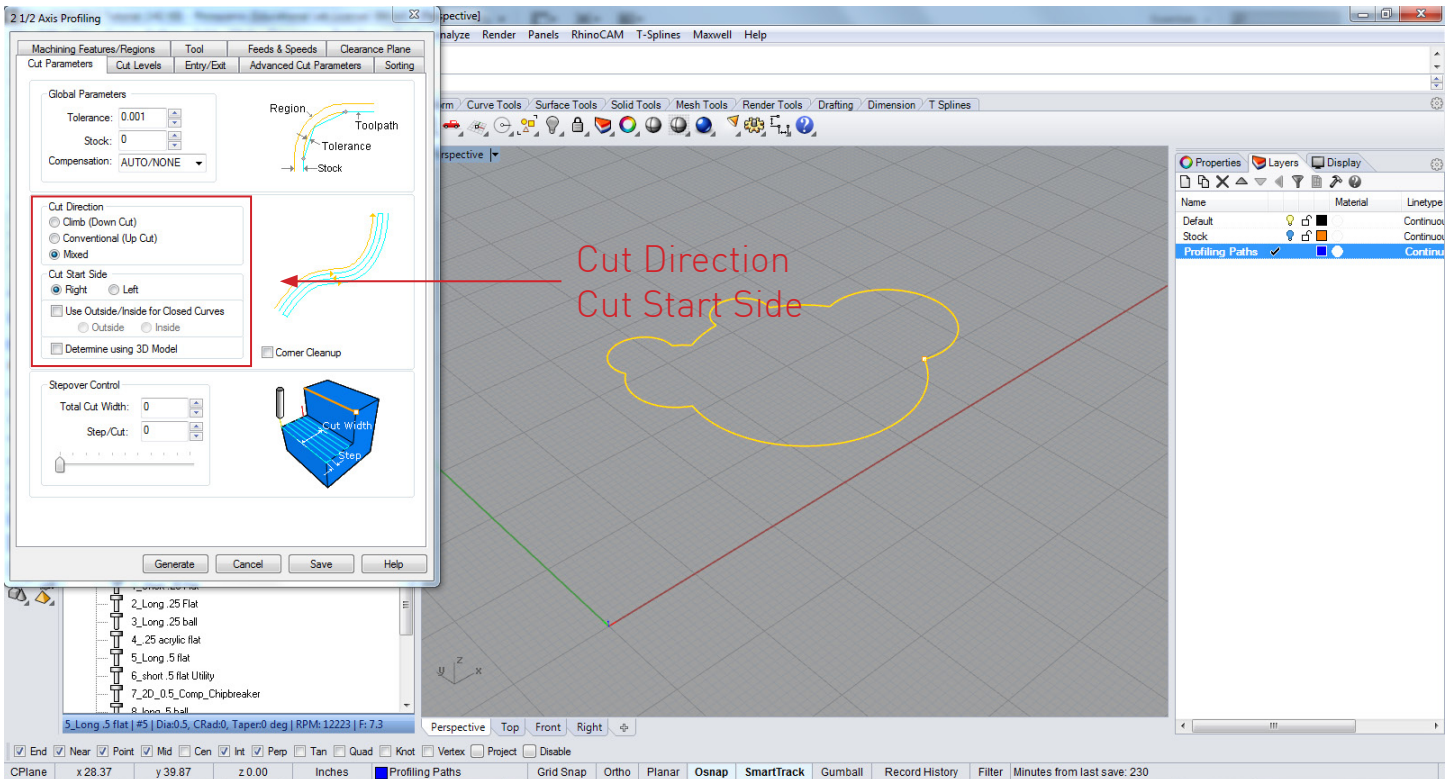
# Rhino\_CAM 2 Axis Machining Operations\_ Profiling



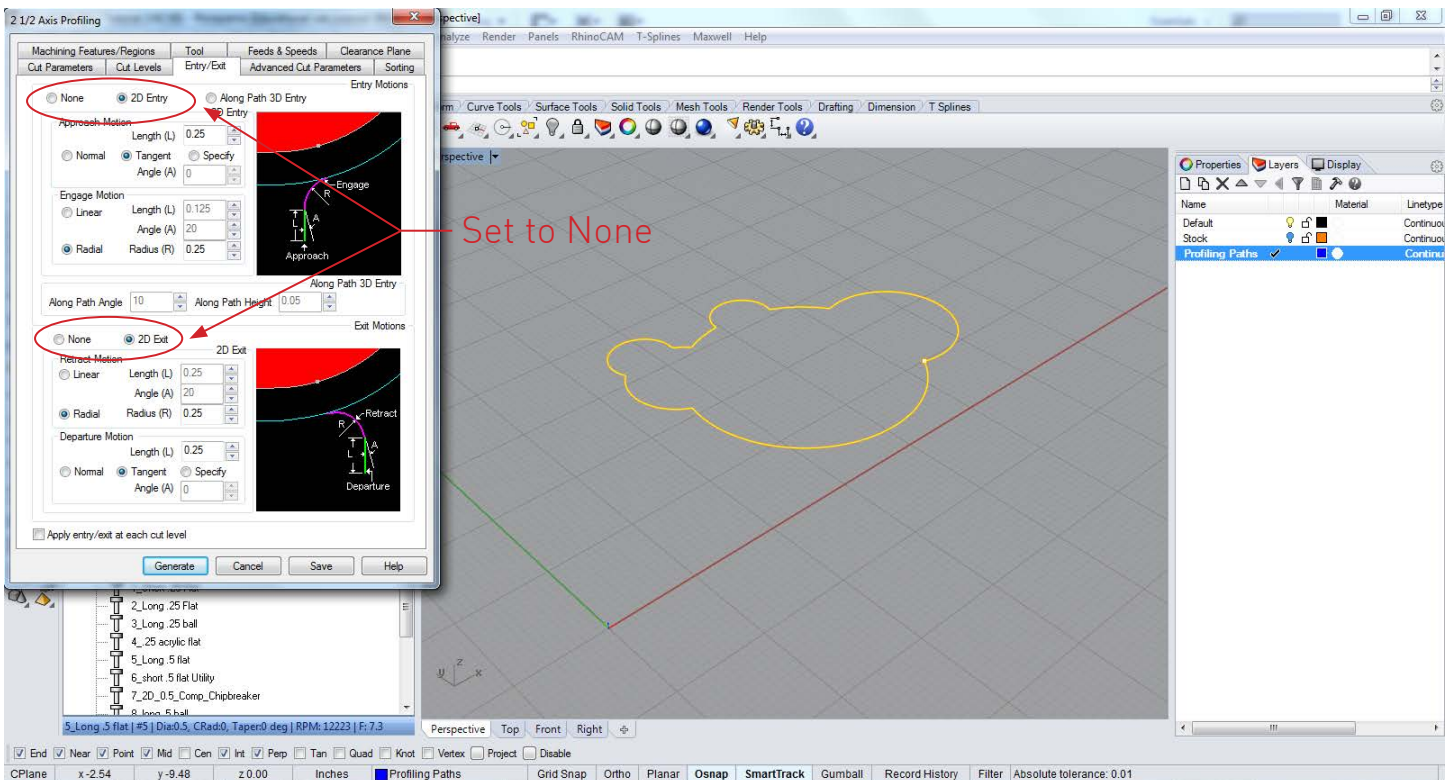
The Profiling tab is located under the 2 Axis machining functions in the program tab in RhinoCAM.



# Rhino\_CAM 2 Axis Machining Operations\_ Profiling

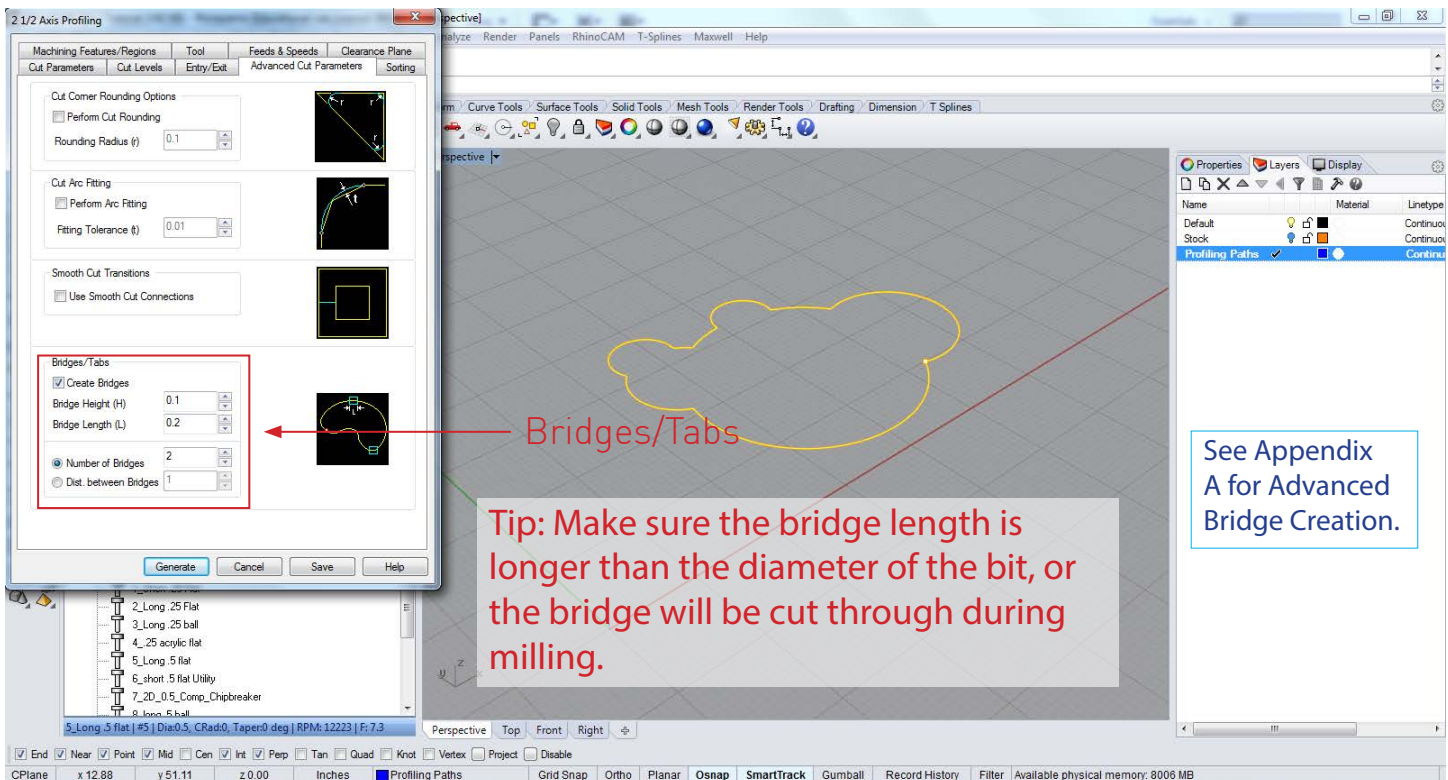


In the Cut Parameters tab the cut direction can be changed and for closed curvature either the inside or the outside of a closed curve can be chosen to cut. An offset of the profile can be setup with Stepover Control, the cut width and the step over to the desired width can be set up here. As a general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

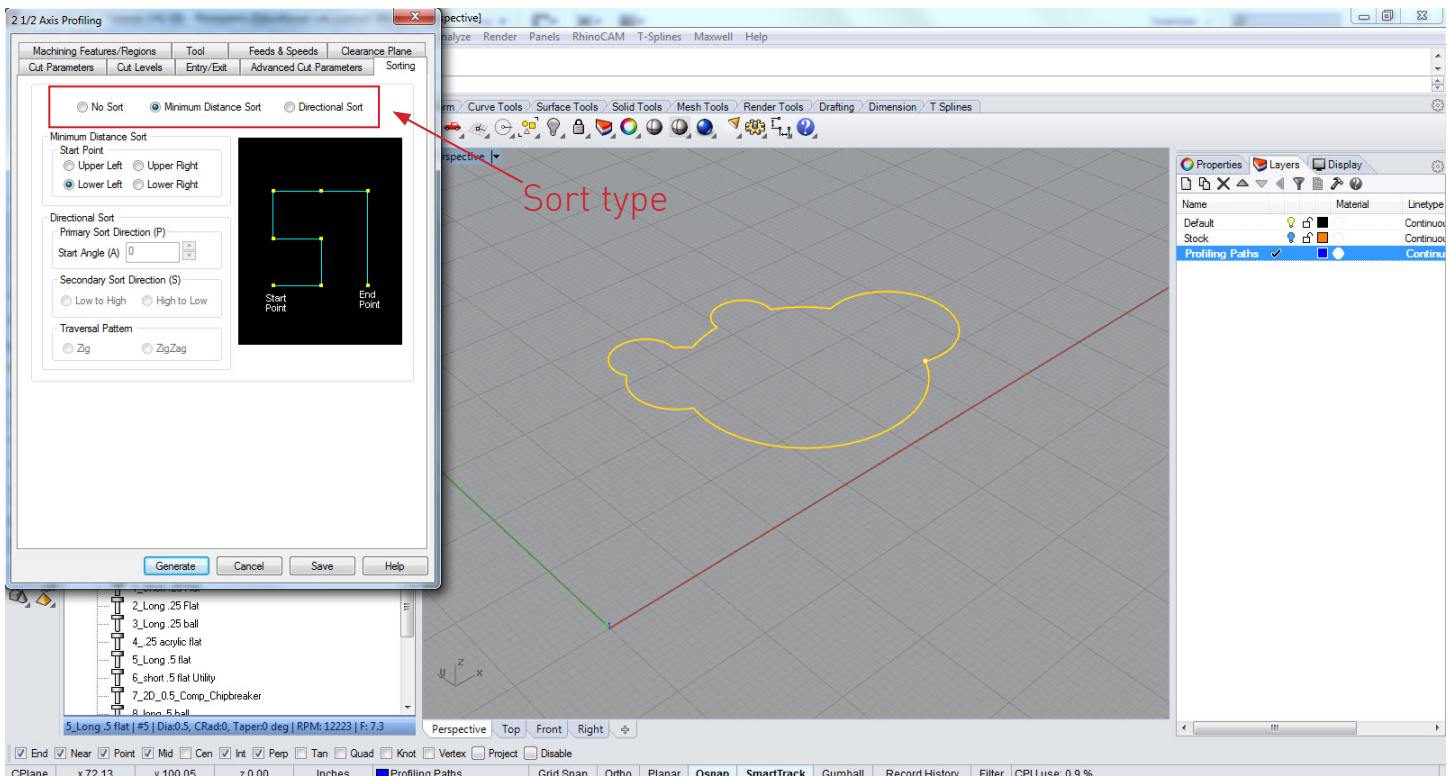


Under the exit / entry tab set the entry and exit motions to none, the 2d exit will create a turn out cut that can come close to other profiles if they are tightly nested.

# Rhino\_CAM 2 Axis Machining Operations\_ Profiling

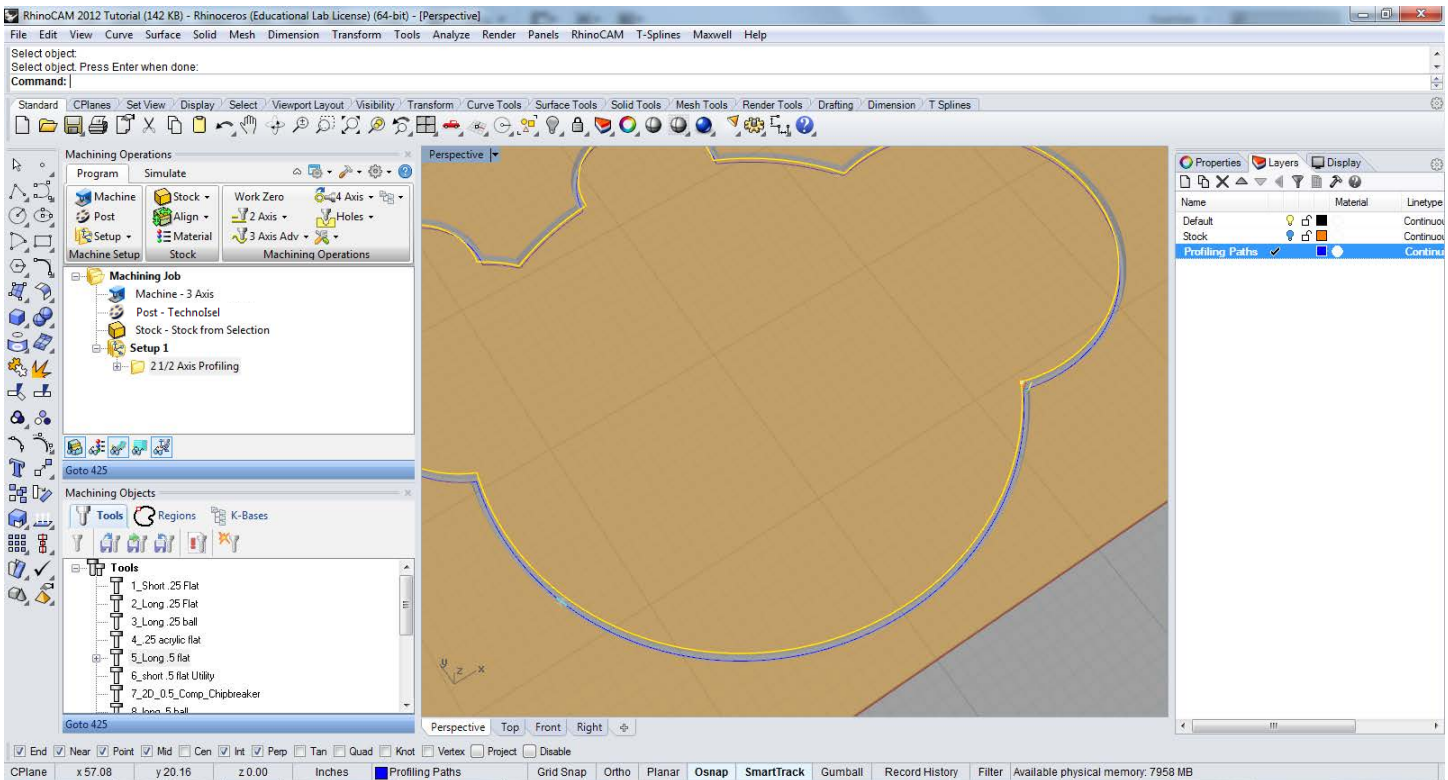


When cutting a profile out of a larger surrounding material the inner cut can lose suction to the CNC vacuum bed once cut. This can cause it to shift and bump into the endmill. To prevent this bridges can be created, these create small tabs that link the cut piece to the larger material. The height,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

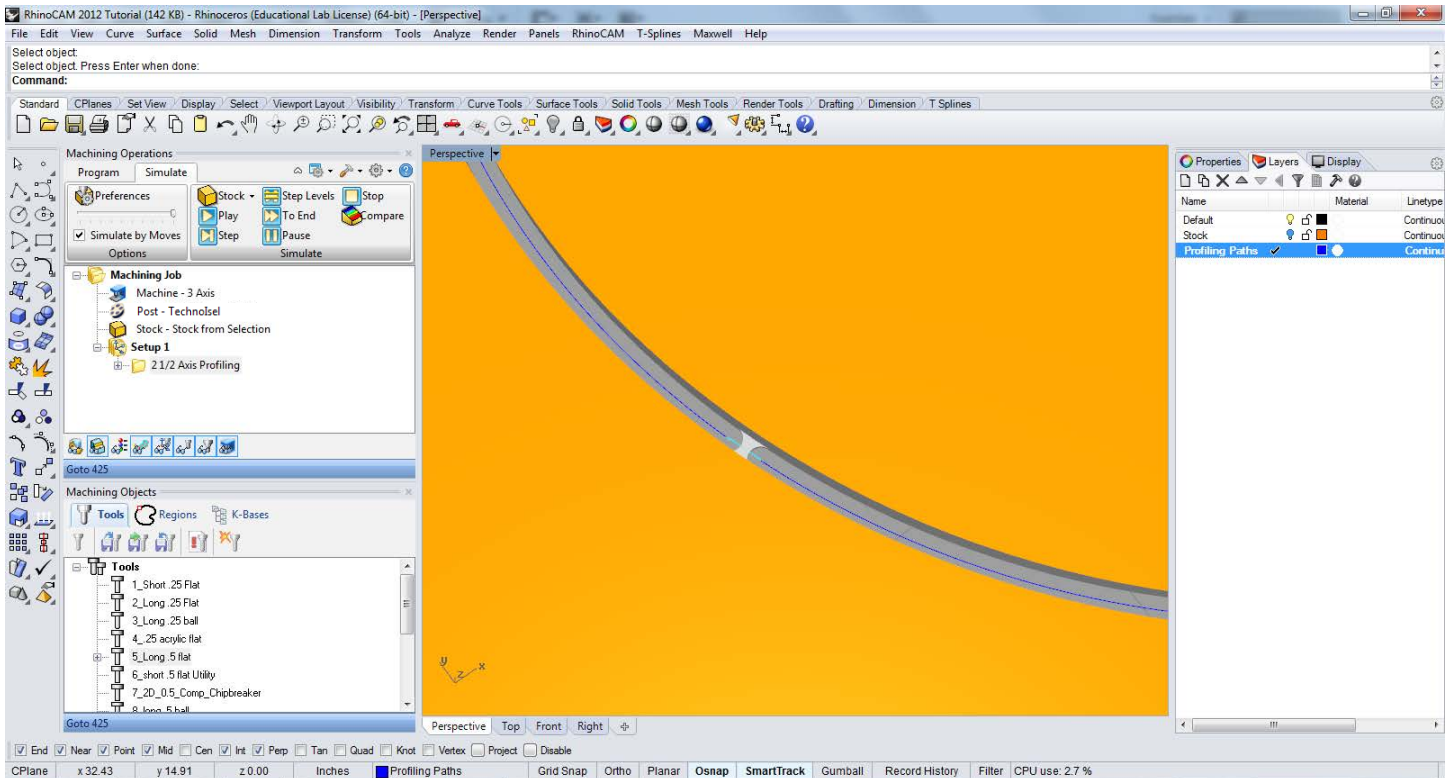


If multiple profiles are being cut in a single sheet the cutting order can be set up under the sorting tab, this allows the shortest path generated from a starting point.

# Rhino\_CAM 2 Axis Machining Operations\_ Profiling

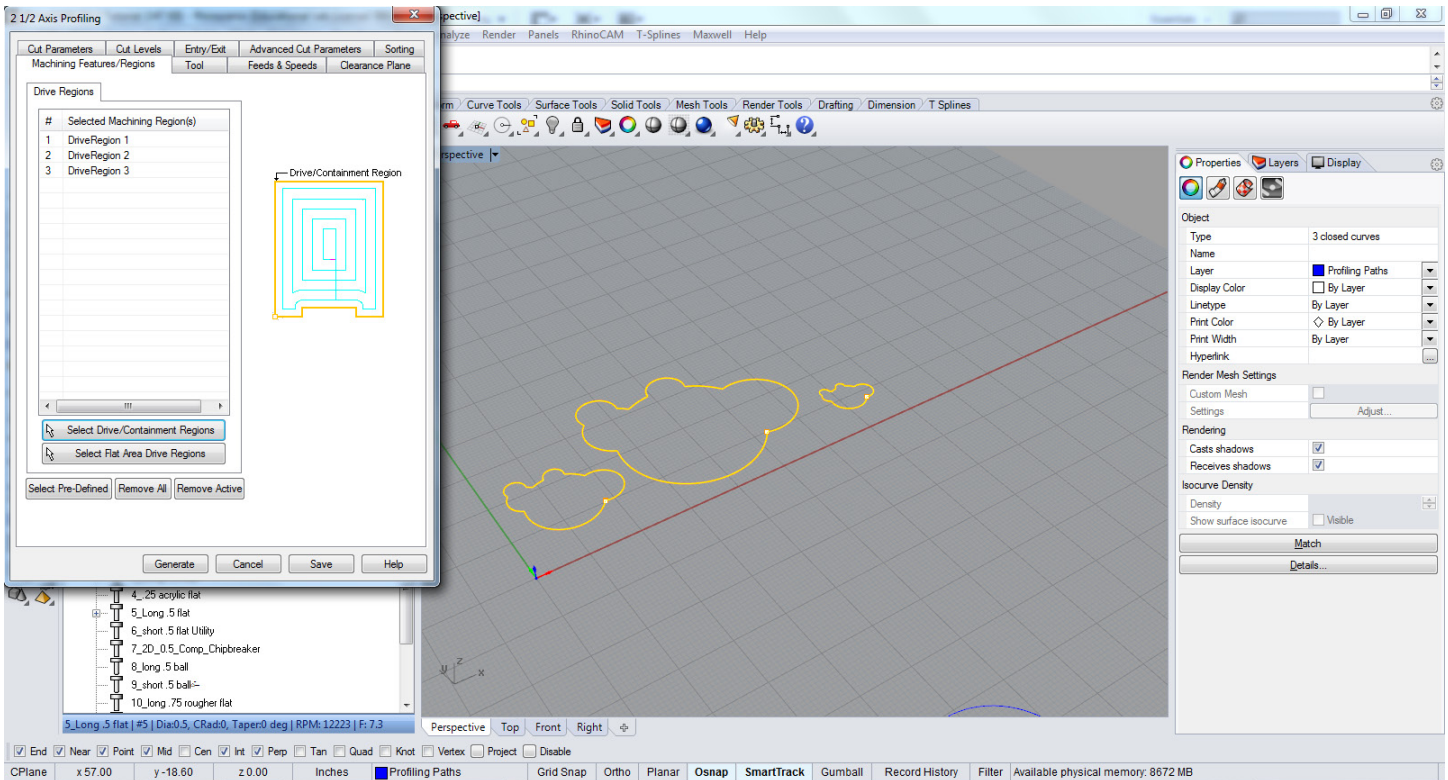


Once generated the profiling path will appear in the Program Tab view, the light blue lines are the bridges and the dark blue is the profile tooling path.

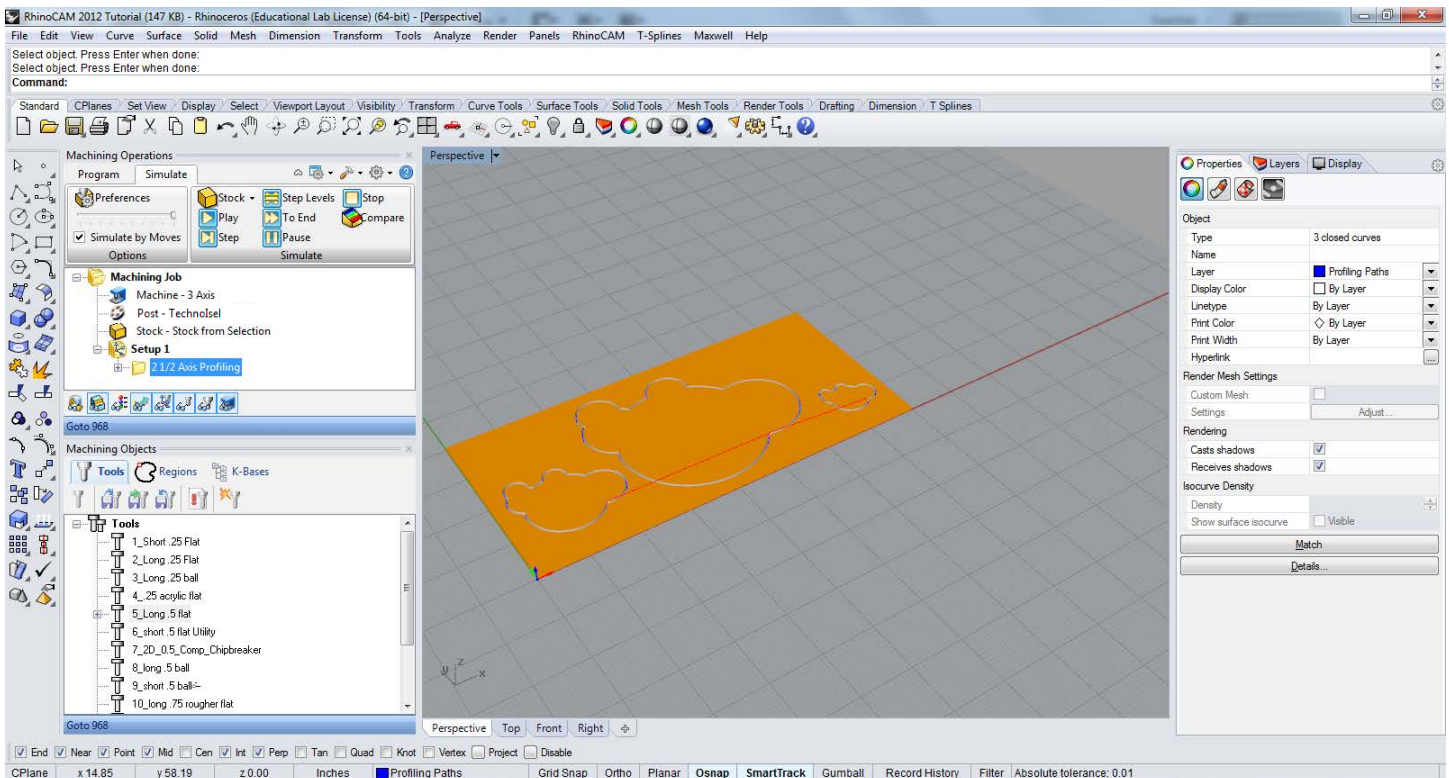


Under the Simulate tab once the simulation has been run the bridges are visible linking the profile cut back to the main sheet material.

# Rhino\_CAM 2 Axis Machining Operations\_Profiling



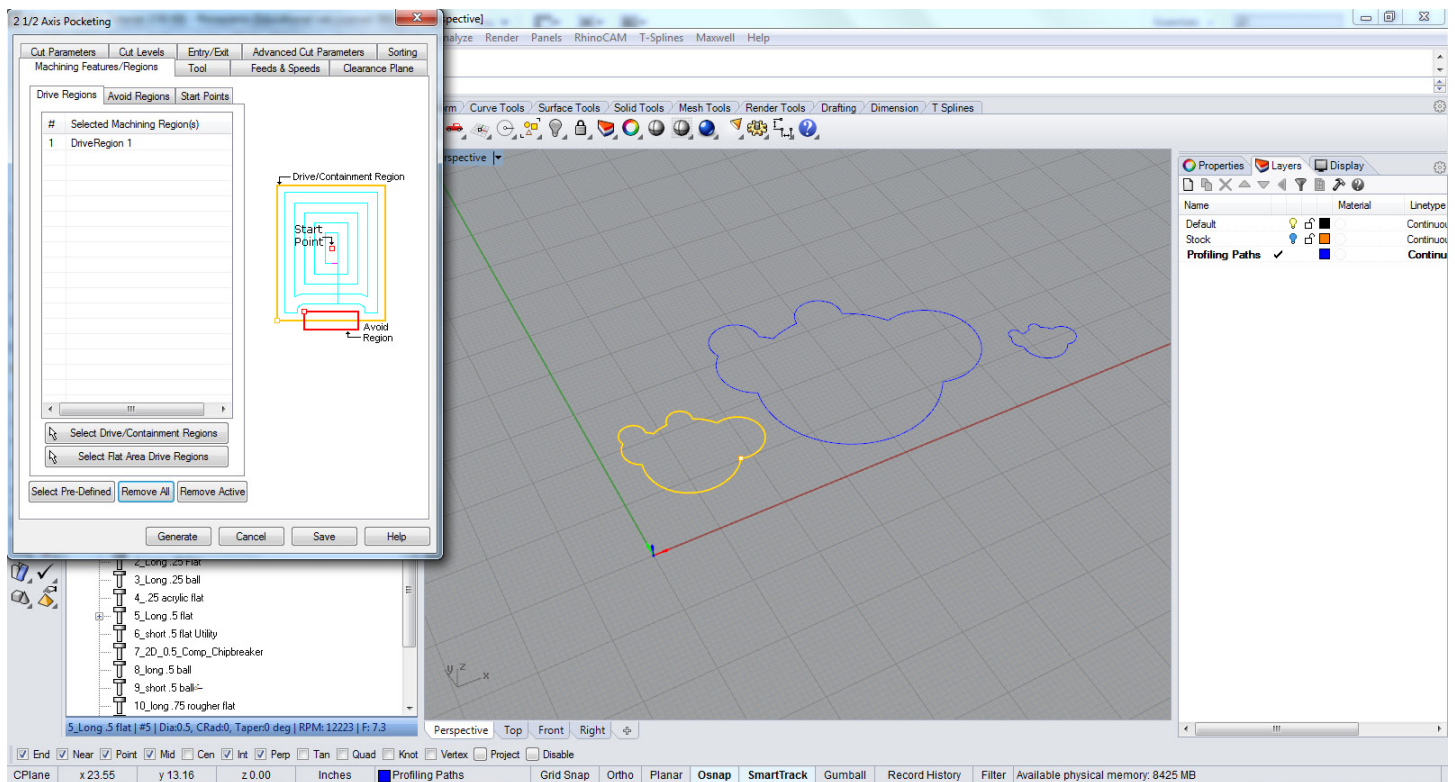
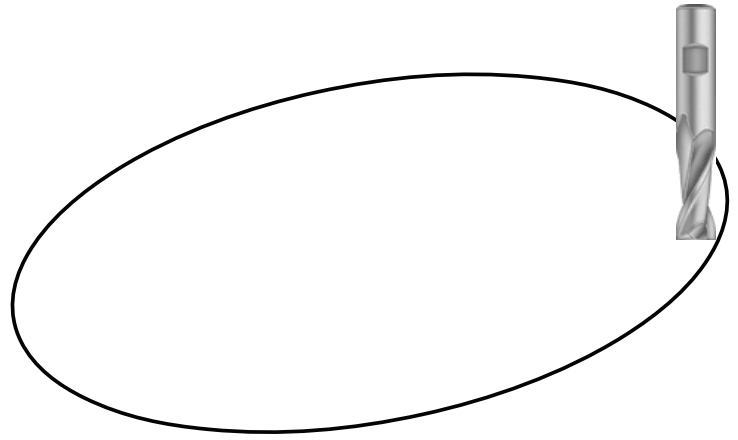
For multiple profiles, select all the drive regions as long as the cutting operation performed remains the same. Under the Sort tab select minimum sorting distance.



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.

# Rhino\_CAM 2 Axis Machining Operations\_ Pocketing

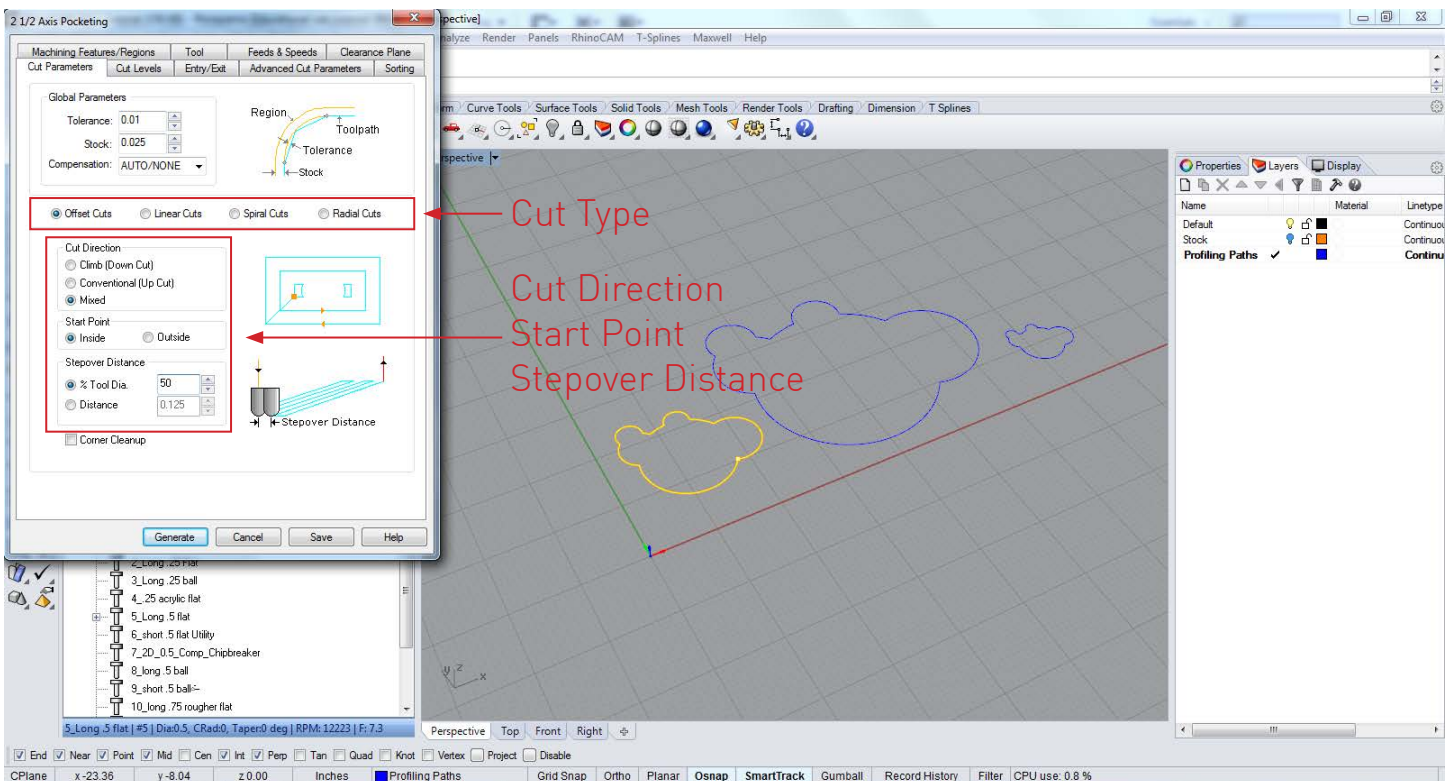
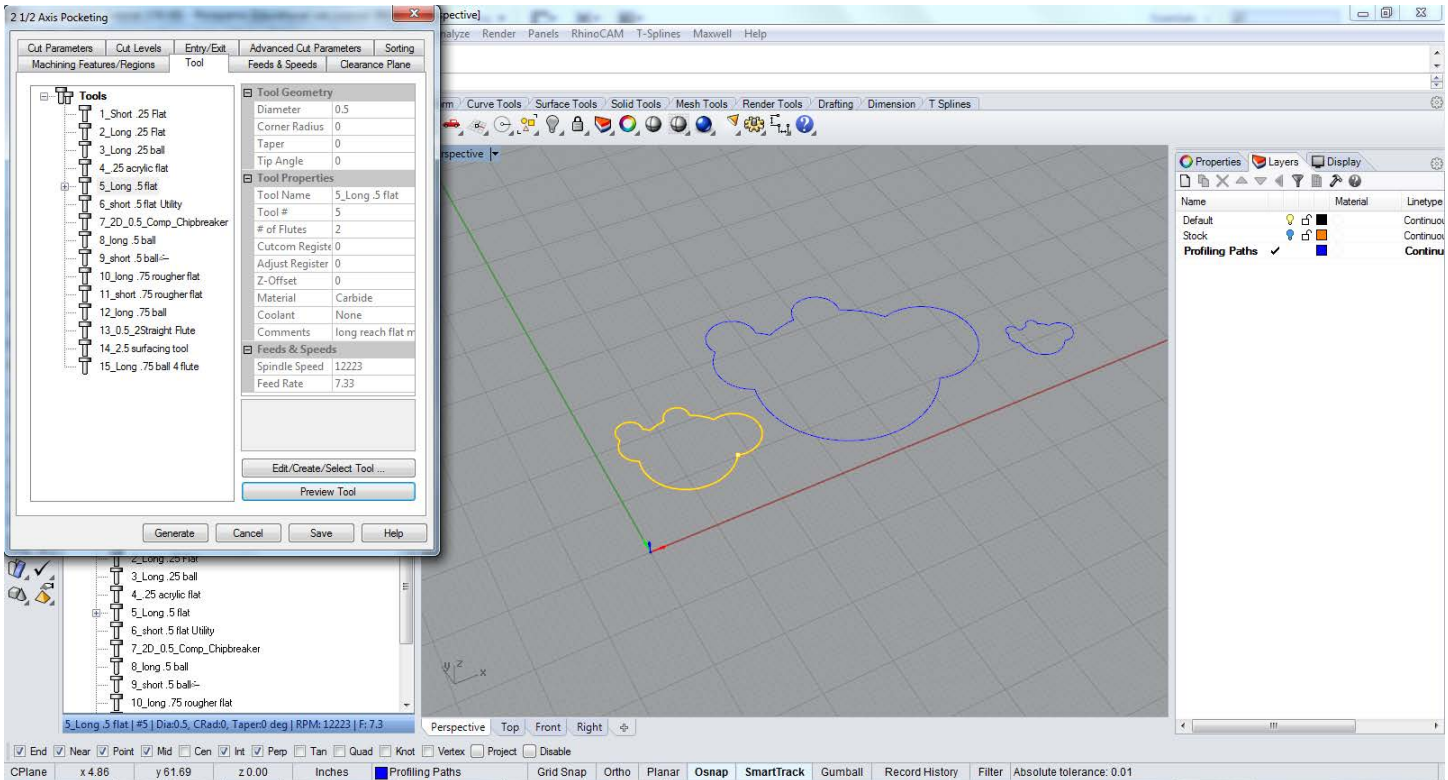
Pocketing milling operations will mill out a selected region - either within a bounded region or between two regions. The operation will populate the region with milling paths to remove the desired material to a depth assigned in the 'Cut Levels' dialog.



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

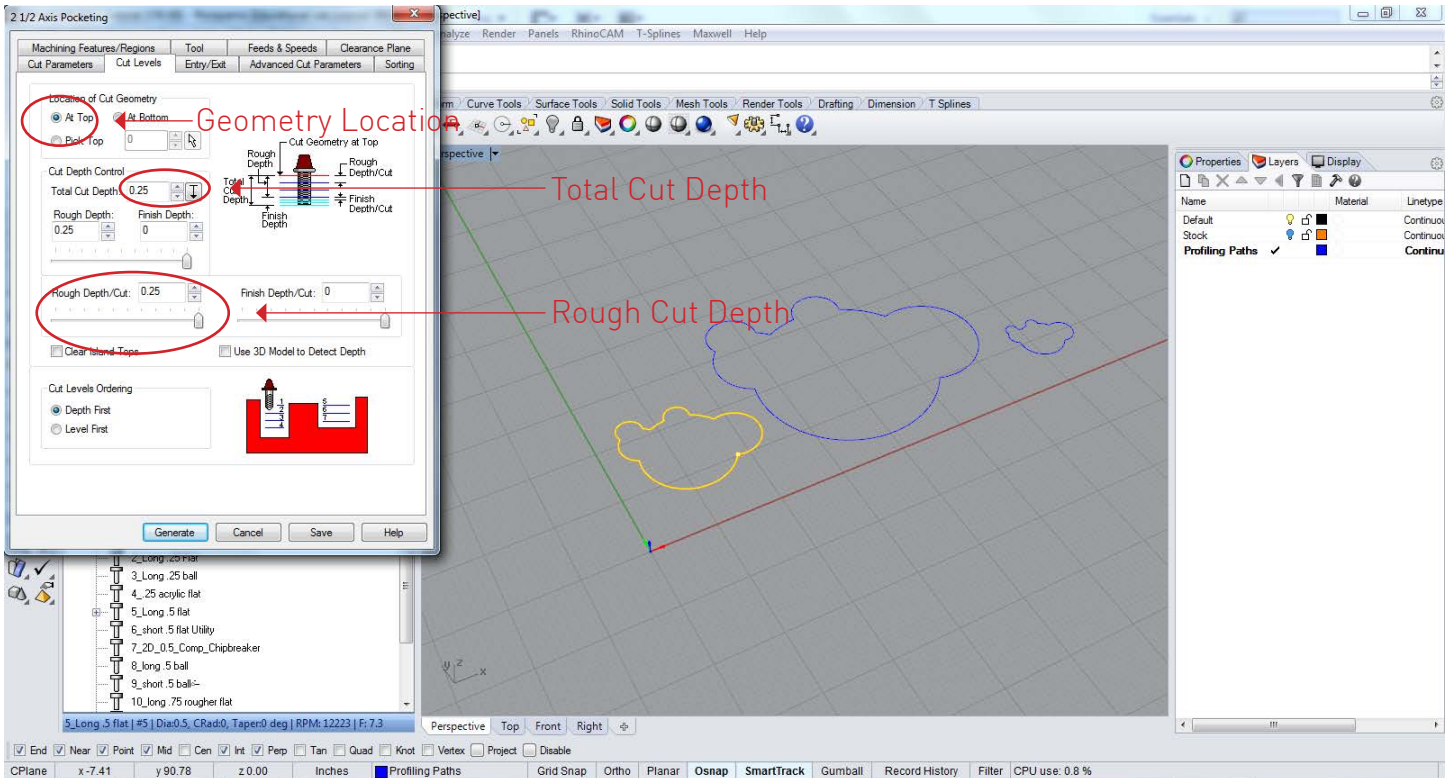


# Rhino\_CAM 2 Axis Machining Operations\_ Pocketing

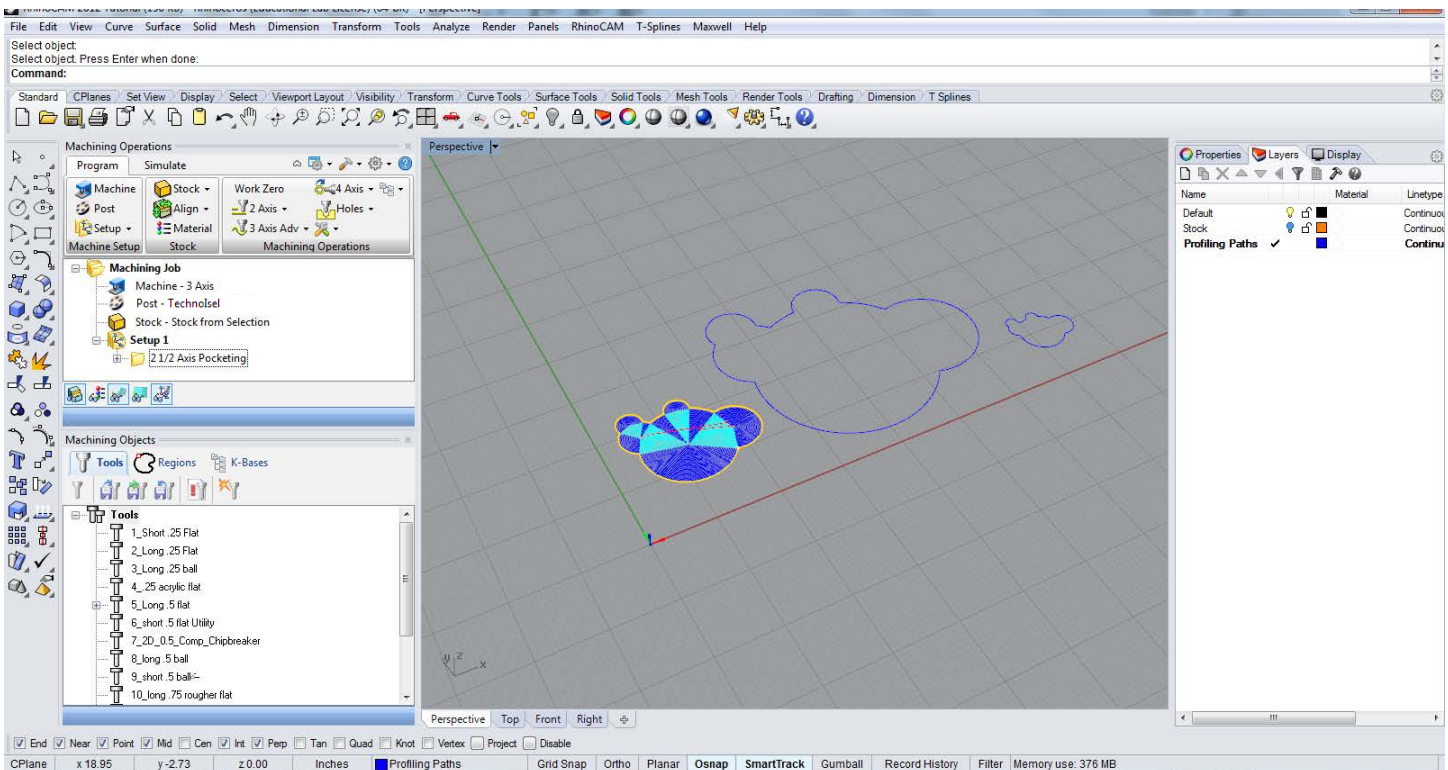


Under Cut Parameters the pocket can be cut using either Offset, Linear, Spiral or Radial cuts. The choice of operation will be dependent of the geometry to be pocketed, for the curvature shown the offset cut will be used starting from the inside to periphery of the geometry, The stepover is set to 50%.

# Rhino\_CAM 2 Axis Machining Operations\_ Pocketing

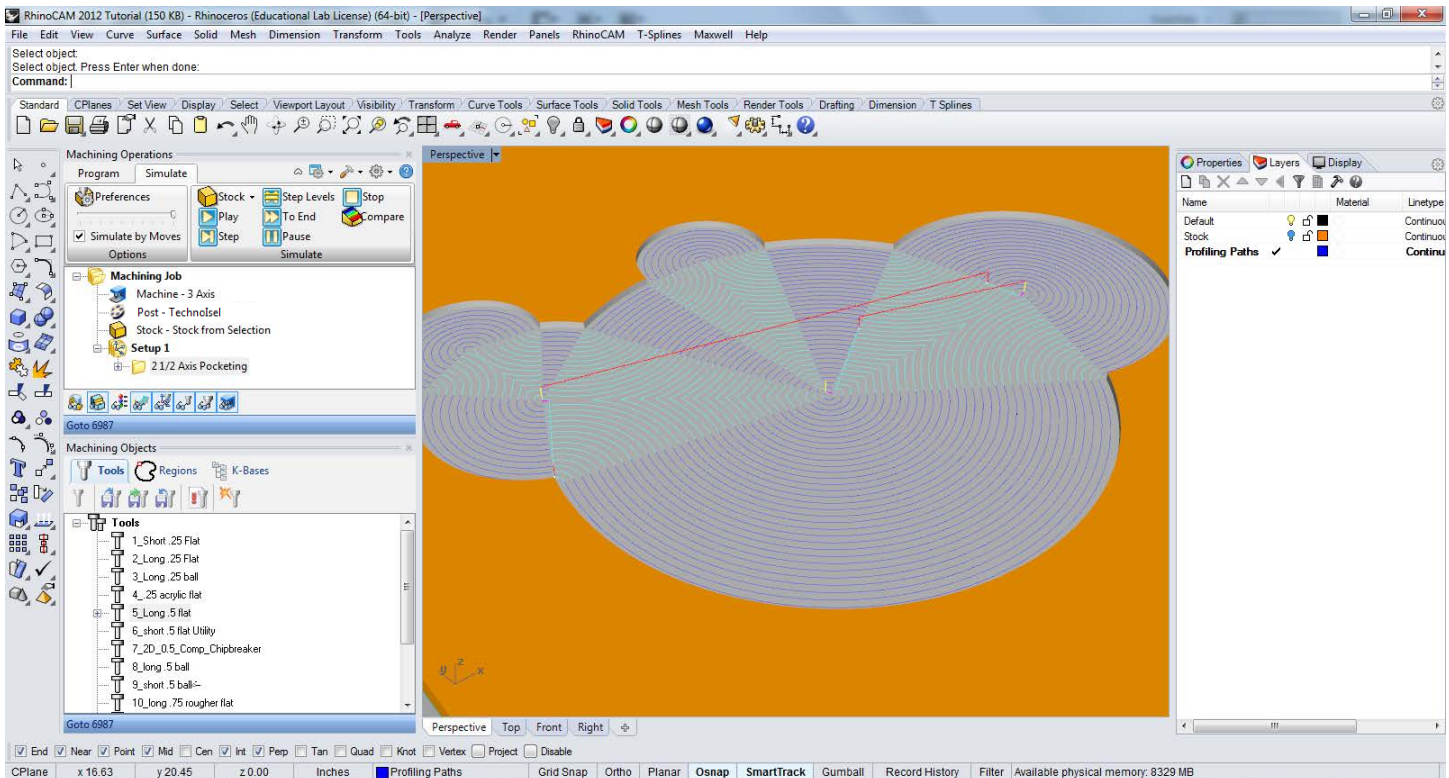


Under 'Cut Levels,' the depth of the cut and the number and height of the stepdowns is defined. First, choose the geometry location where your curve is located relative to the stock. In the example, the curves are located at the top. The rough cut depth is set to match the total depth, as the cut is only 0.25". If the total depth is substantial, use a series of stepdowns to remove the material. The stepdowns are set using the sliders.

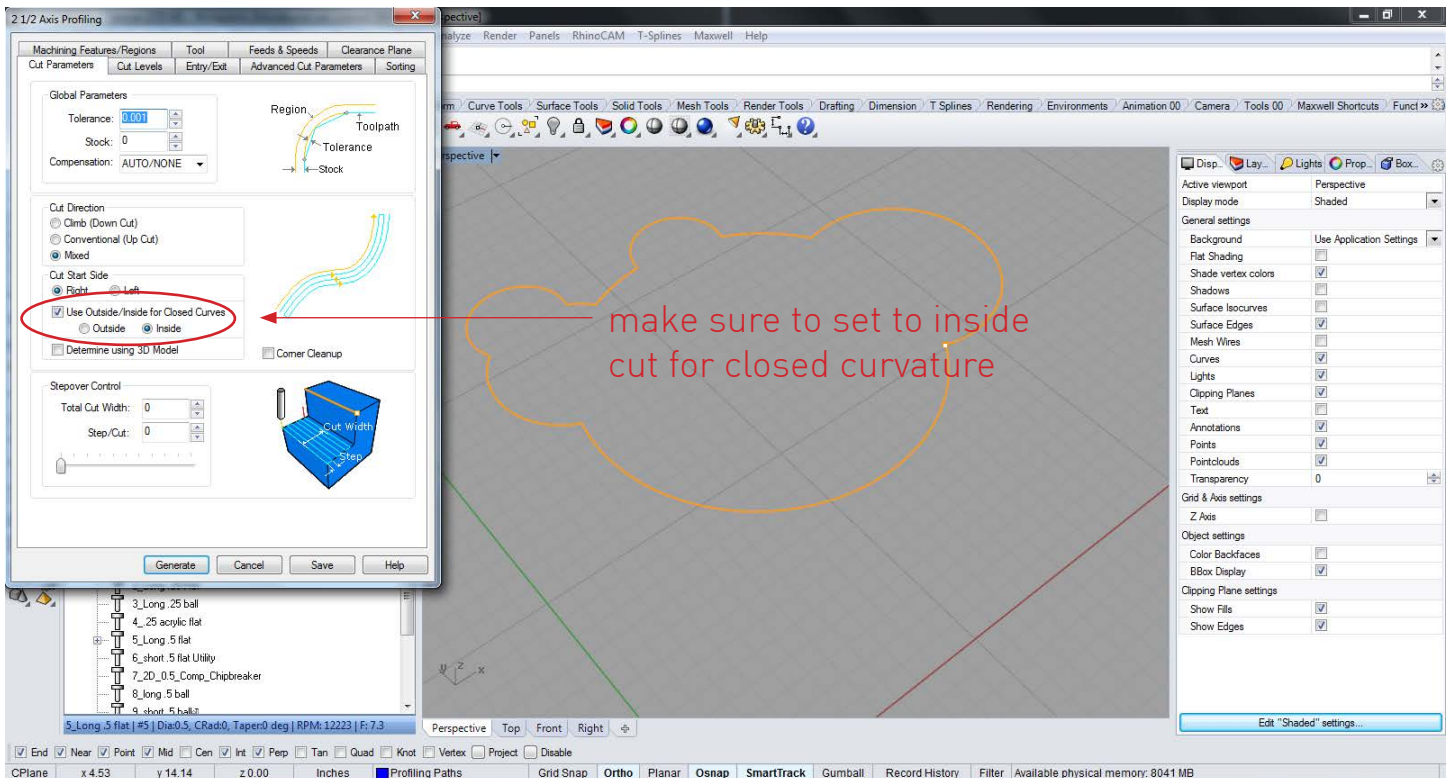


Once all of the parameters have been set, press 'Generate.' The toolpaths will appear in the region in concentric offsets starting from the inside and working outwards towards the region boundary.

# Rhino\_CAM 2 Axis Machining Operations\_ Pocketing

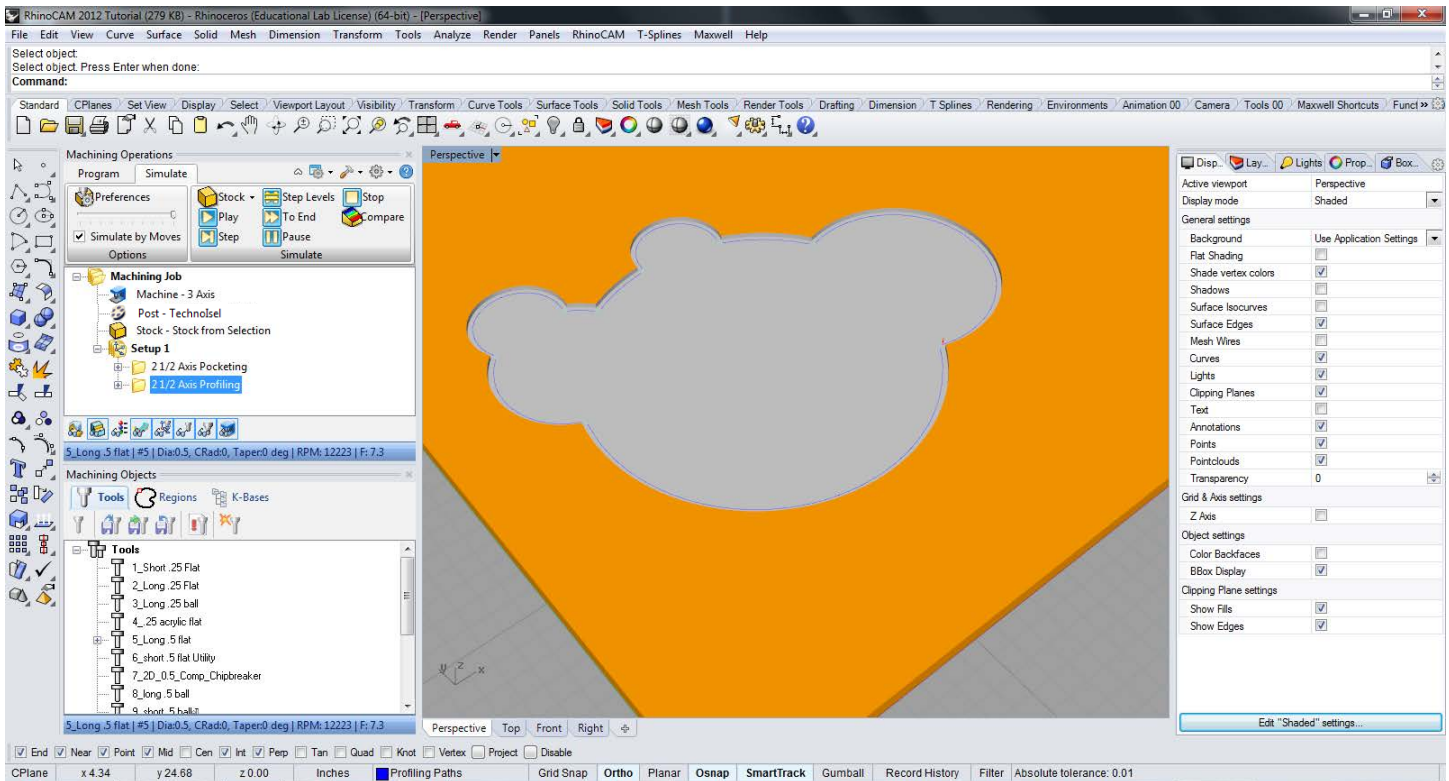


Go to the simulate tab to see the generated toolpaths run. With pocketing passes, sometimes the pocket geometry will be slightly inset from the original curve. A profile pass can be run with the pocket to insure a tight alignment to the original region boundary.



Select the same curve geometry used for the pocketing operation. Under Cut parameters, make sure 'Use Inside/Outside for Closed Curves' setting is selected and set to 'Inside.' Set the depth the same as for the pocketing pass.

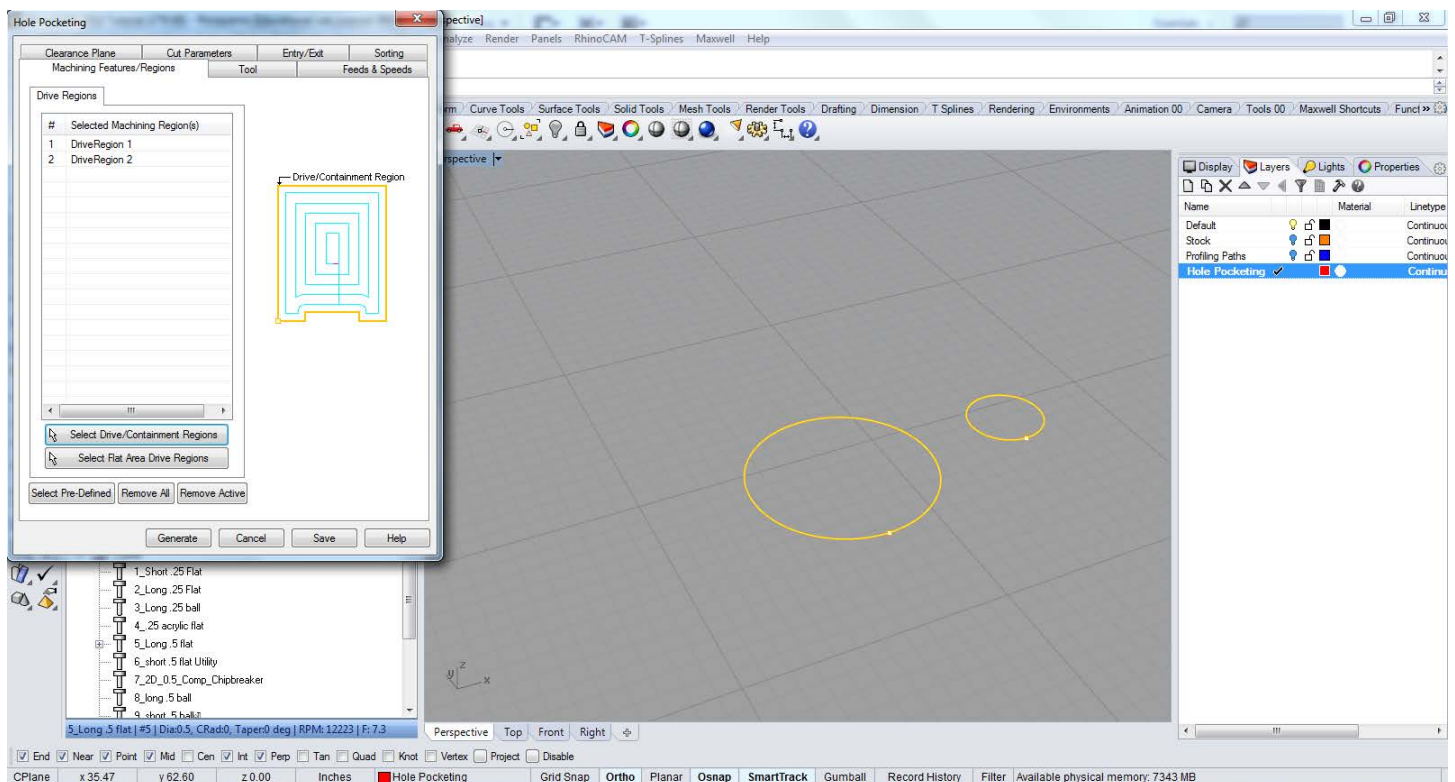
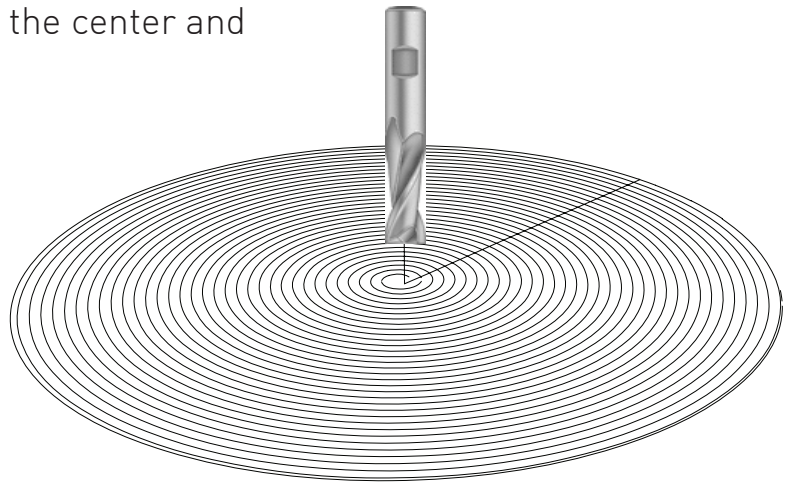
# Rhino\_CAM 2 Axis Machining Operations\_ Pocketing



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 curve geometry.

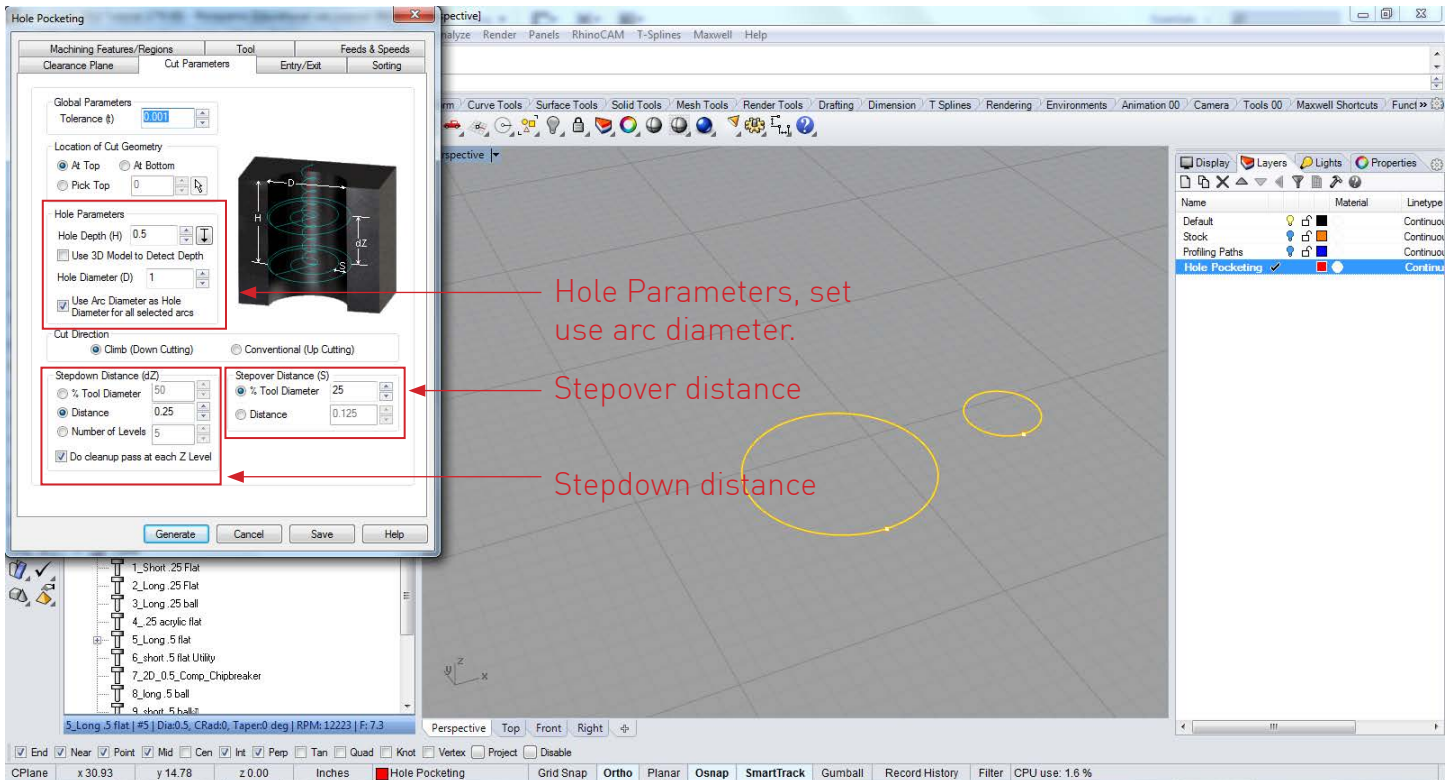
# Rhino\_CAM 2 Axis Machining Operations\_ Hole\_Pocketing

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.

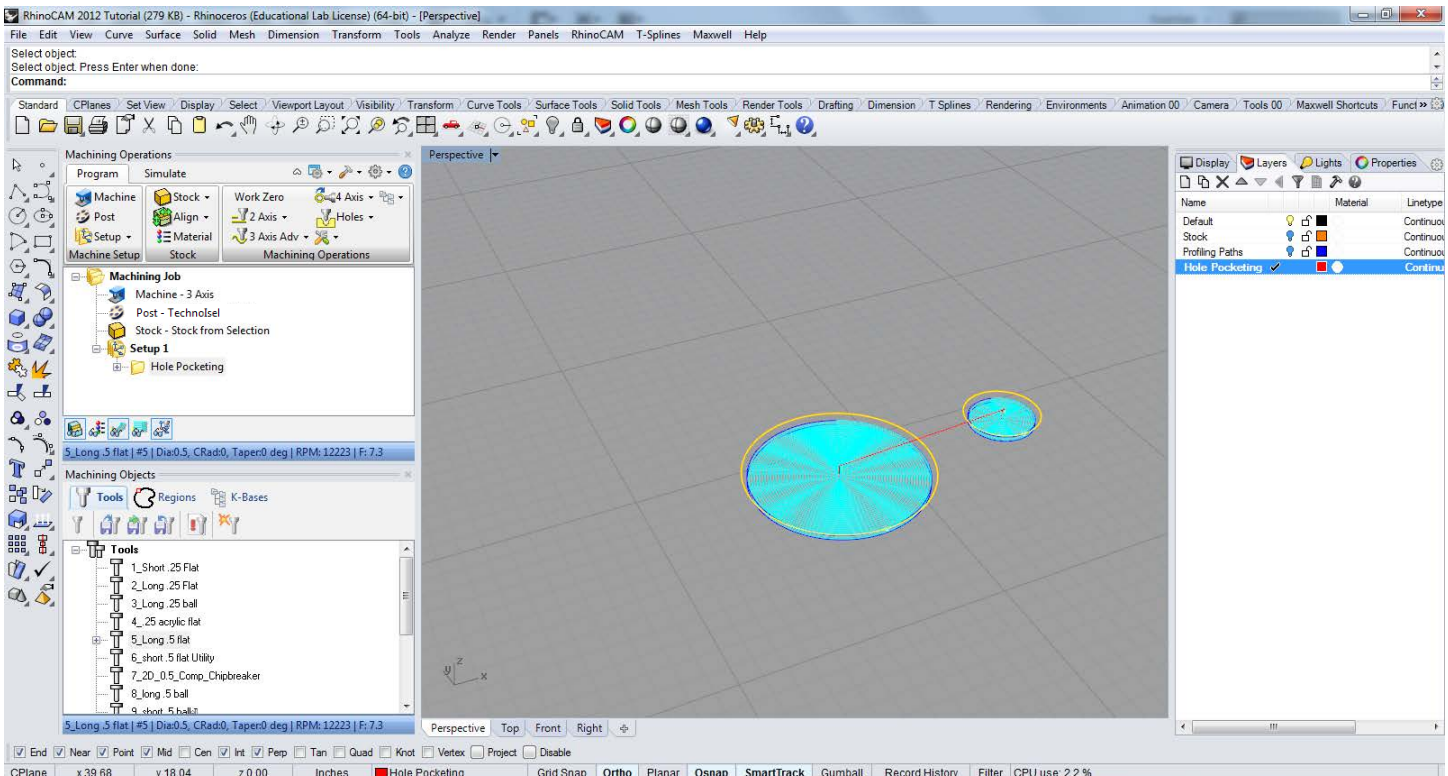


Select 'Hole Pocketing' from the 2-Axis dropdown menu, and draw and select the circles to pocket. Make sure the geometry is closed. If it isn't, RhinoCAM will return error messages when it tries to compute the tooling paths.

# Rhino\_CAM 2 Axis Machining Operations\_ Hole\_Pocketing

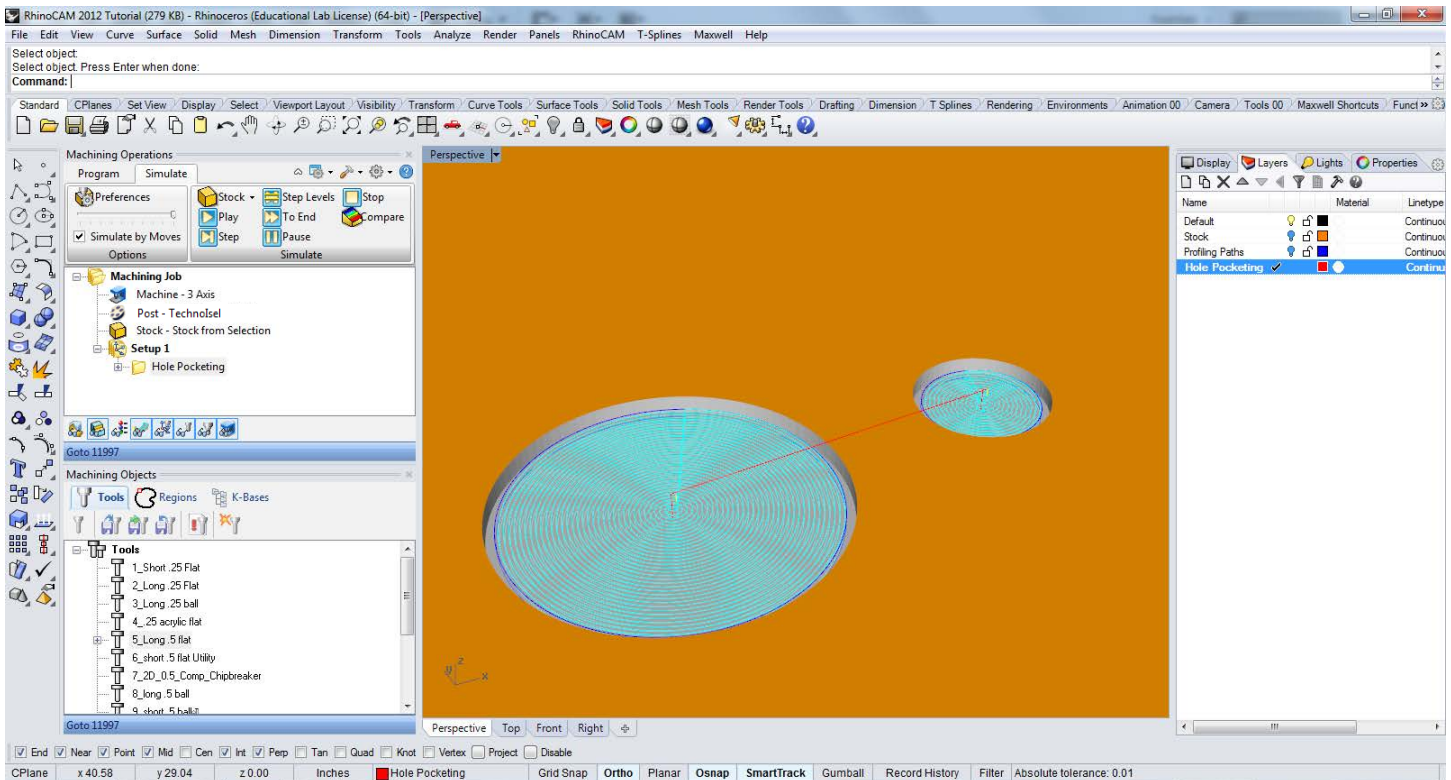


Once the regions have been set, set the chosen endmill and the feeds and speeds required for the material to be milled. Under 'Cut Parameters,' choose the location of the geometry, set the hole depth, and check 'Use Arc Diameter as Hole Diameter for all selected arcs.' Choose the stepdown distance and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



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

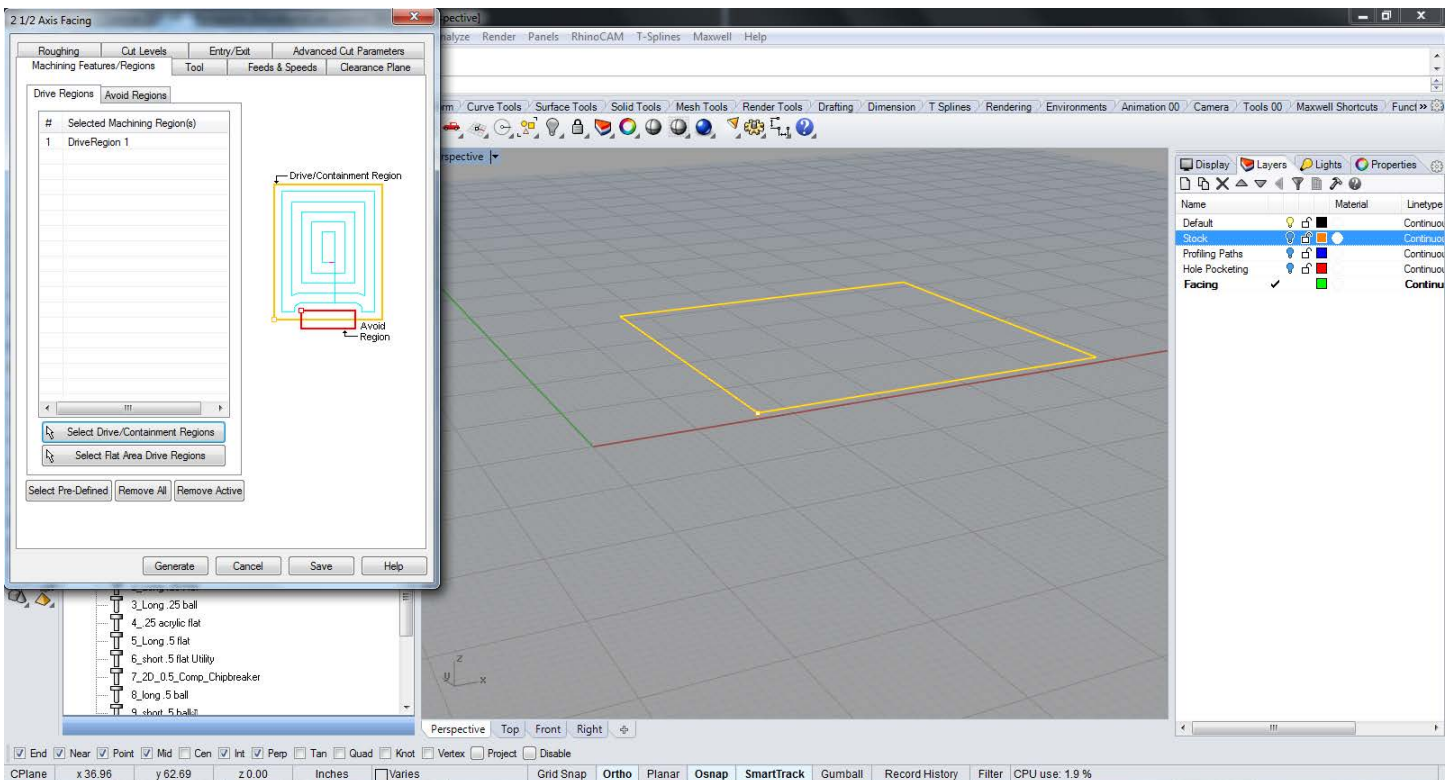
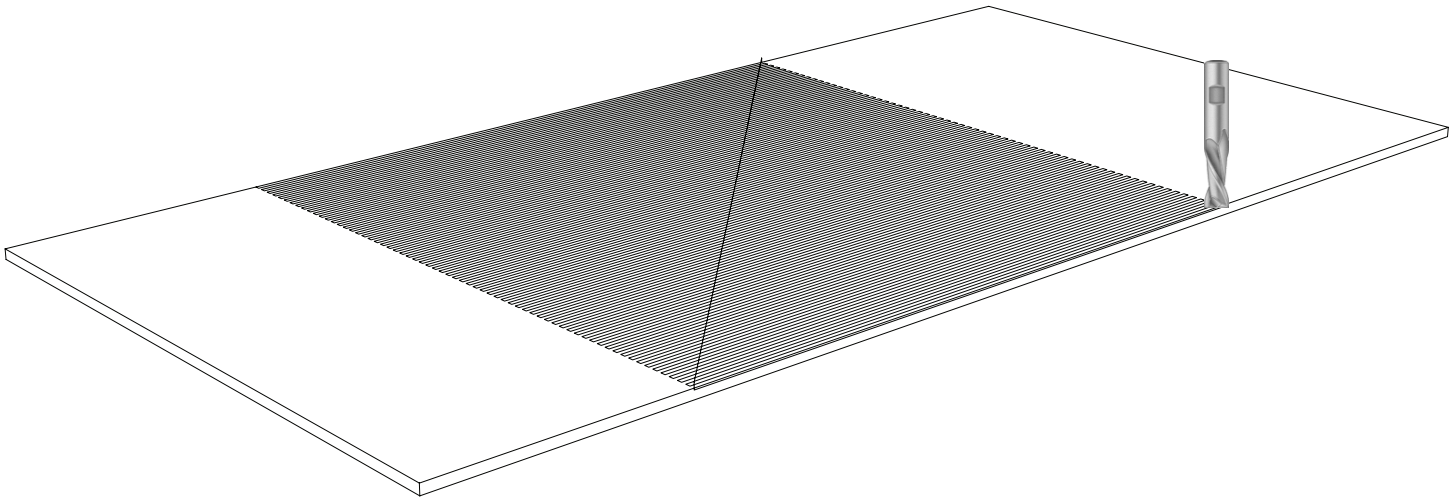
# Rhino\_CAM 2 Axis Machining Operations\_ Hole\_Pocketing



Run the simulation to see the milling paths in in progress. Always run the simulation to make sure there are no errors or collisions with the collet or head of the CNC router.

# Rhino\_CAM 2 Axis Machining Operations\_ Facing

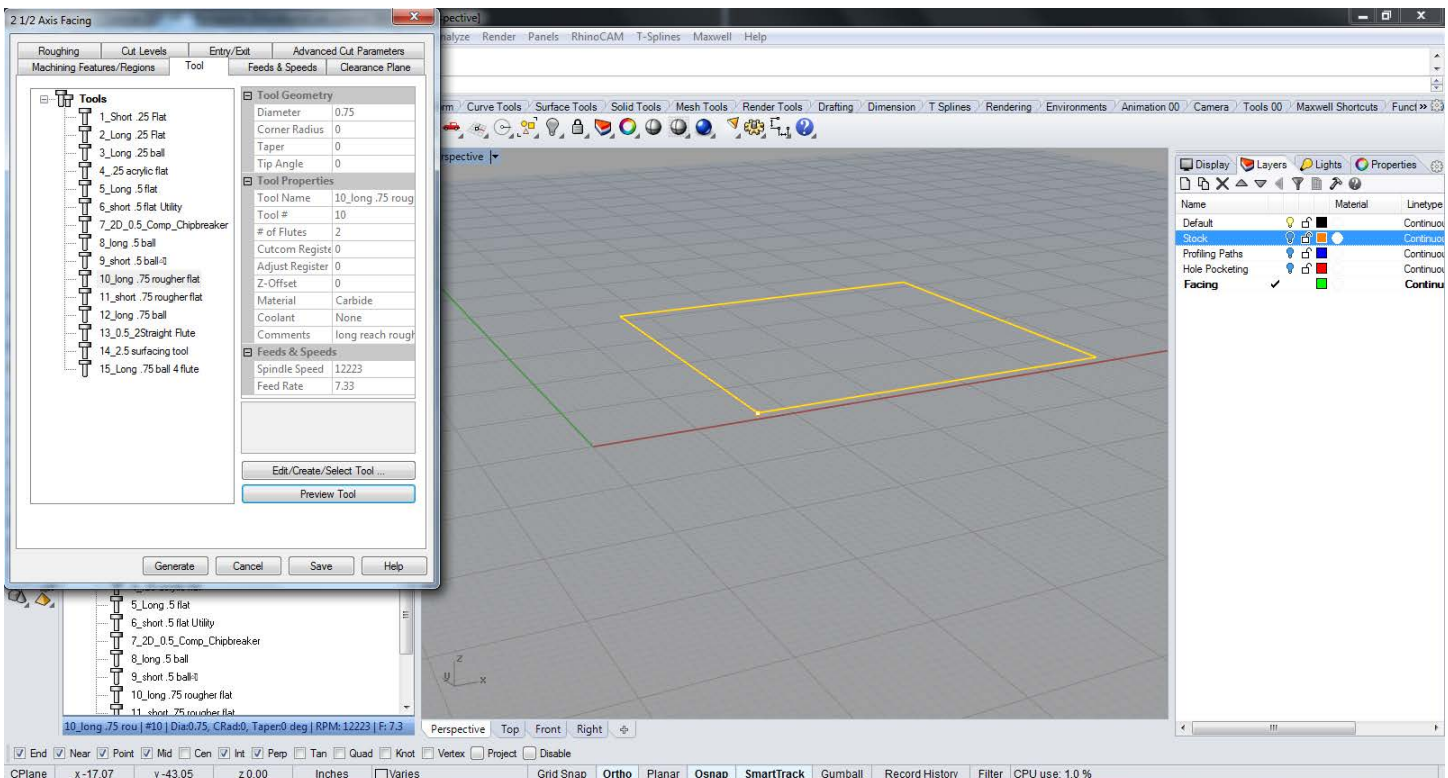
The facing operation is used for large areas to plane down the material in a setup stepdown and stepover assigned in the cutting parameters. The facing operation is useful for planing materials to become planar and for removing larger amounts of materials in pre set areas.



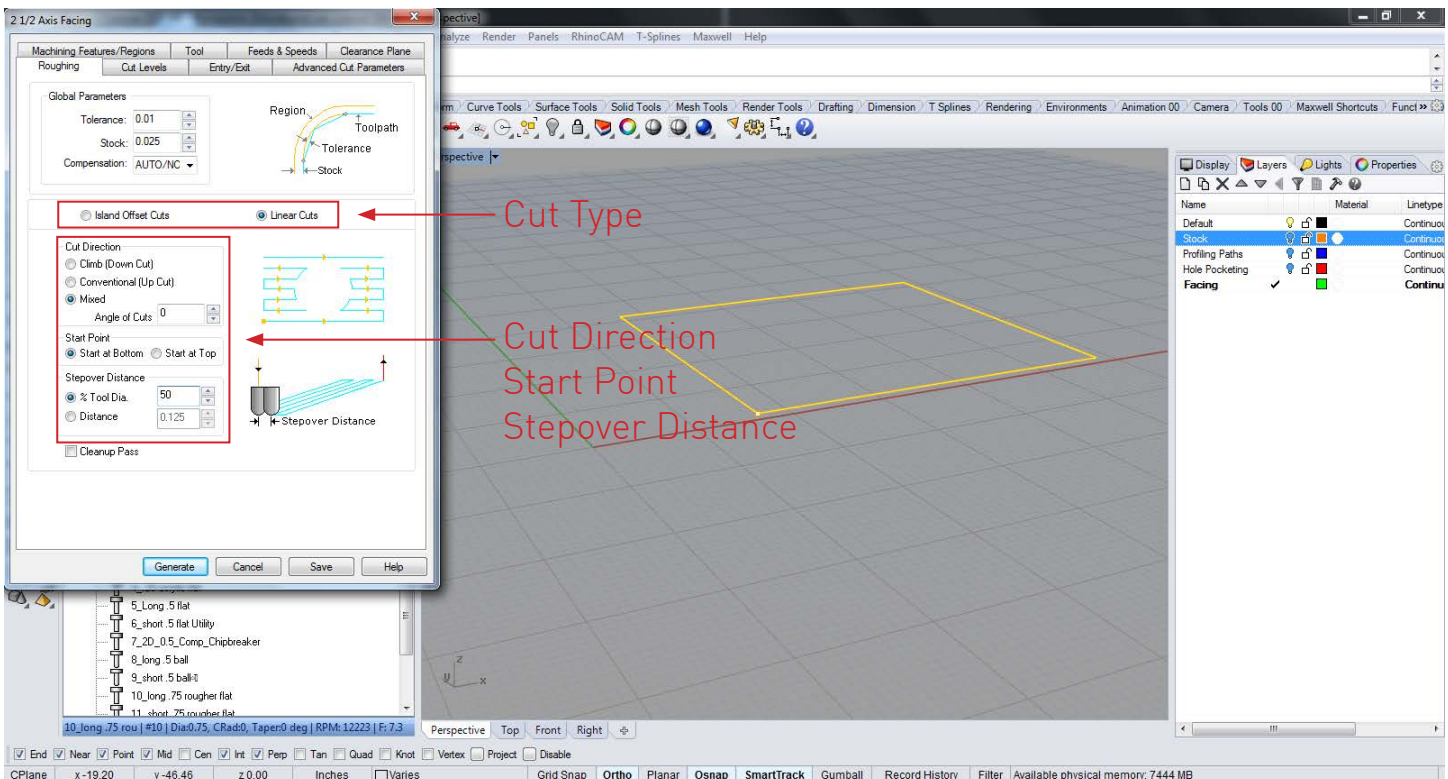
Select the 2 1/2 Axis Facing from the 2Axis drop down tab, select the region that is to be faced on the set up stock.



# Rhino\_CAM 2 Axis Machining Operations\_ Facing

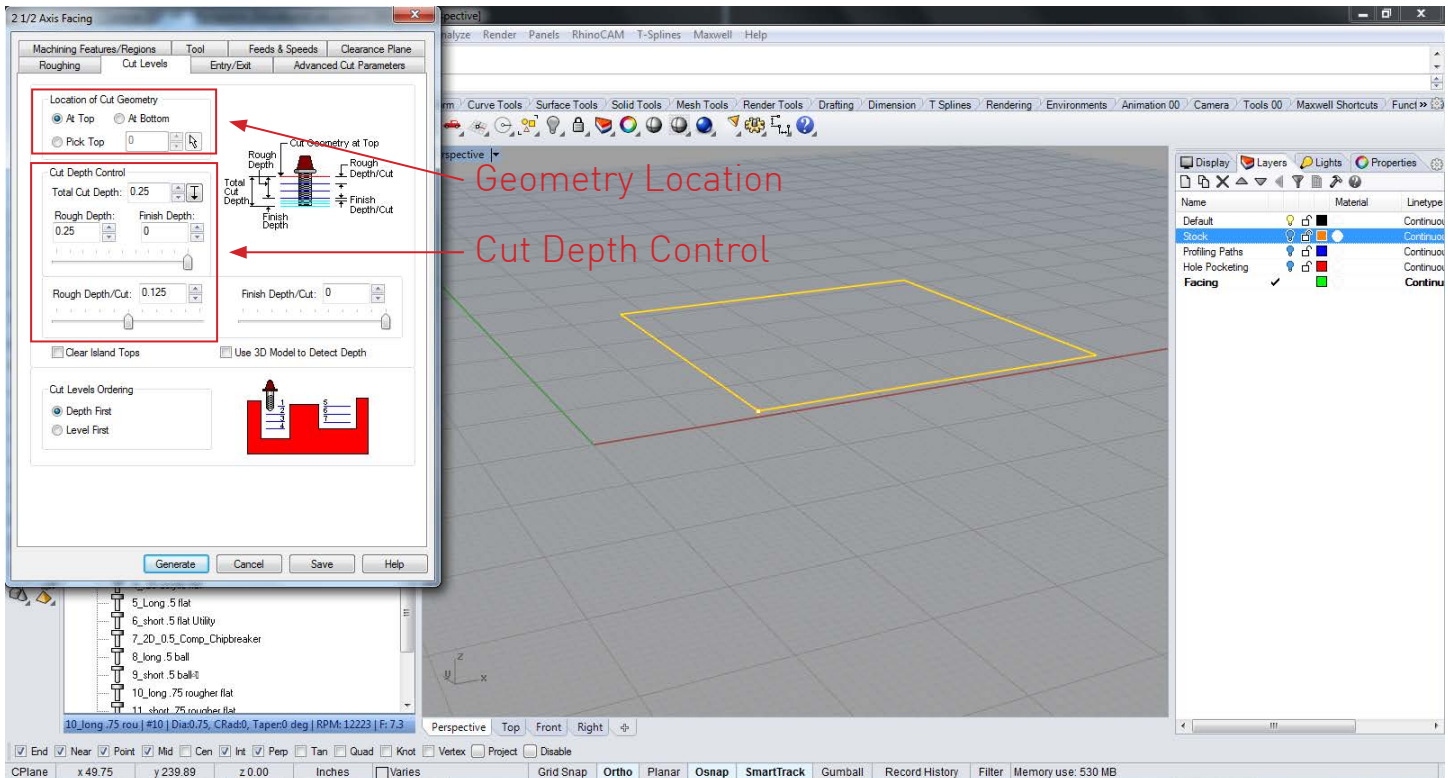


Select the endmill depending on the amount of material to be removed, for a larger area with geometry that is simple use a large endmill for faster facing. If the area is a smaller geometry select a smaller endmill to fit accordingly.

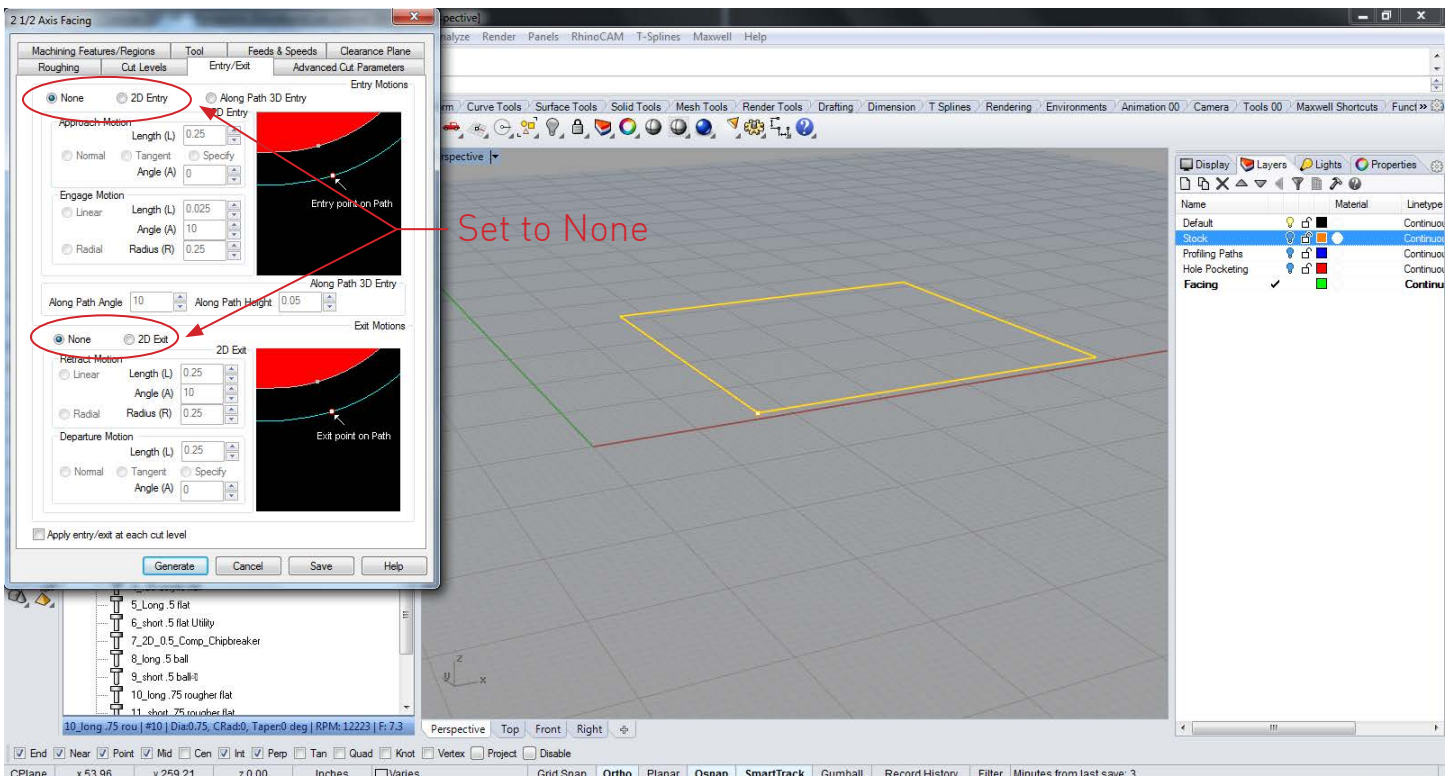


Under the Roughing tab set the type of cuts to linear, the cut direction can be set if milling hardwoods, next set up the Stepover distance.

# Rhino\_CAM 2 Axis Machining Operations\_ Facing

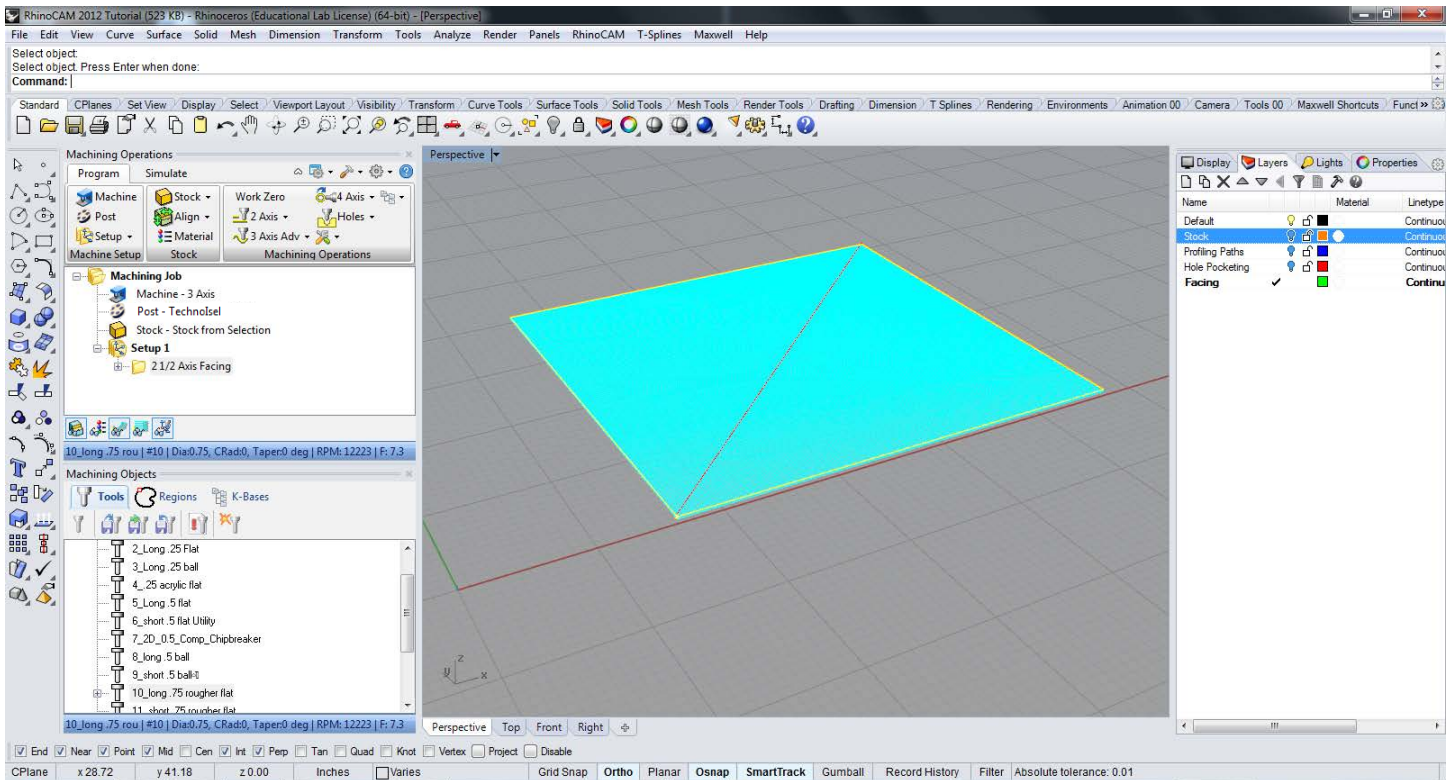


Under the 'Cut Levels' tab set the location of the input geometry, the total cut depth and the number of stepdowns in the rough depth cut. Typically, never exceed the radius of the chosen endmill in plywood or hardwoods.

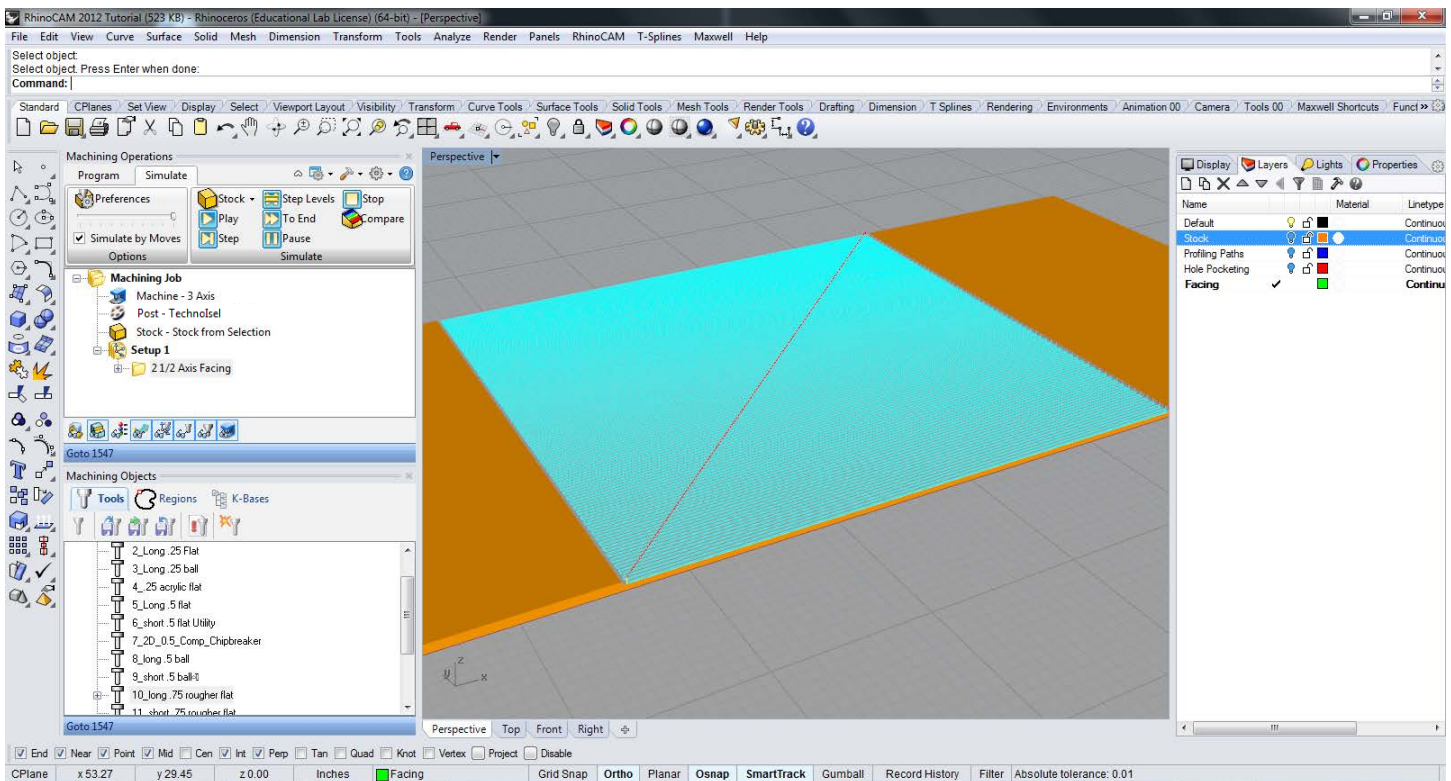


Under the Exit/Entry tab make sure they are set to **none** for a straight z axis entry or along 3d path for a sloping z axis entry.

# Rhino\_CAM 2 Axis Machining Operations\_ Facing



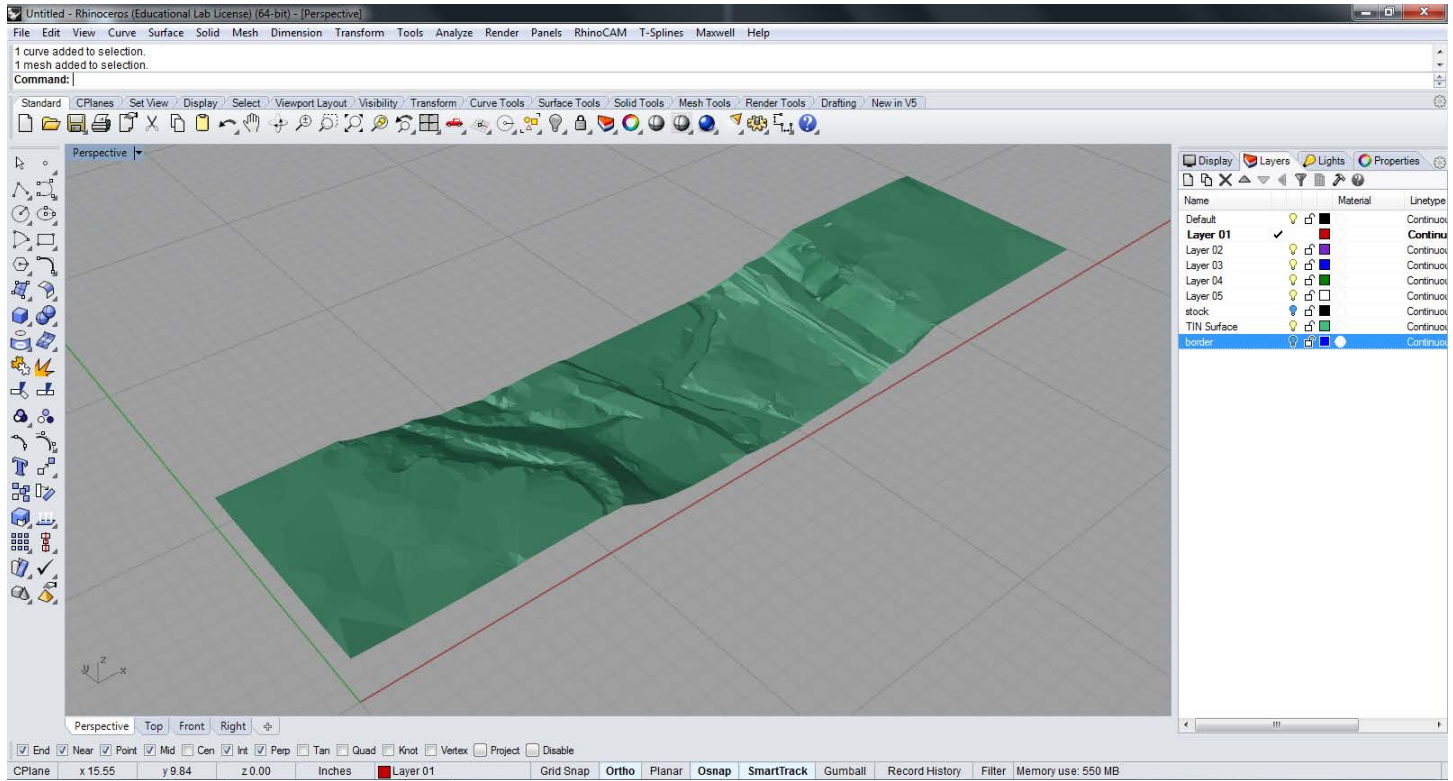
Press generate once the setup is done and the tooling paths will be generated.



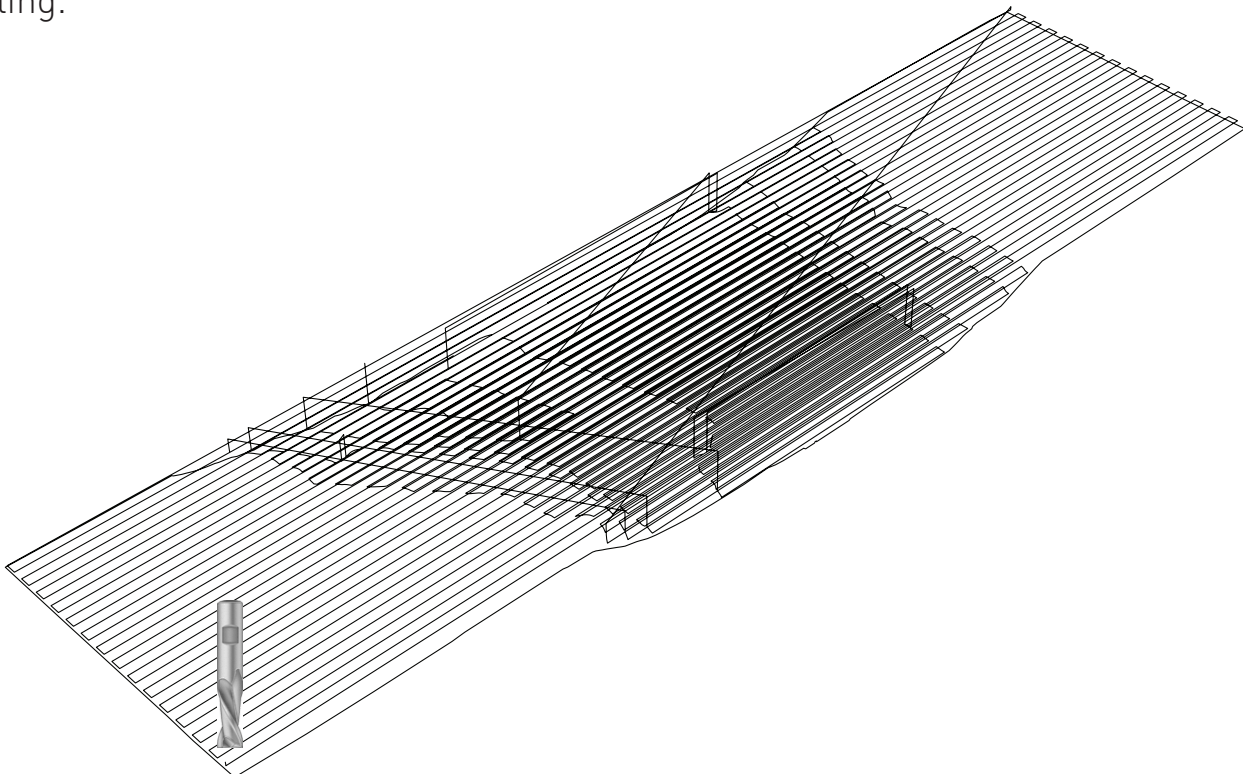
Once the tooling paths have been generated run the simulation under the simulate tab.

# Rhino\_CAM 3 Axis Machining Operations\_ Horizontal\_Roughing

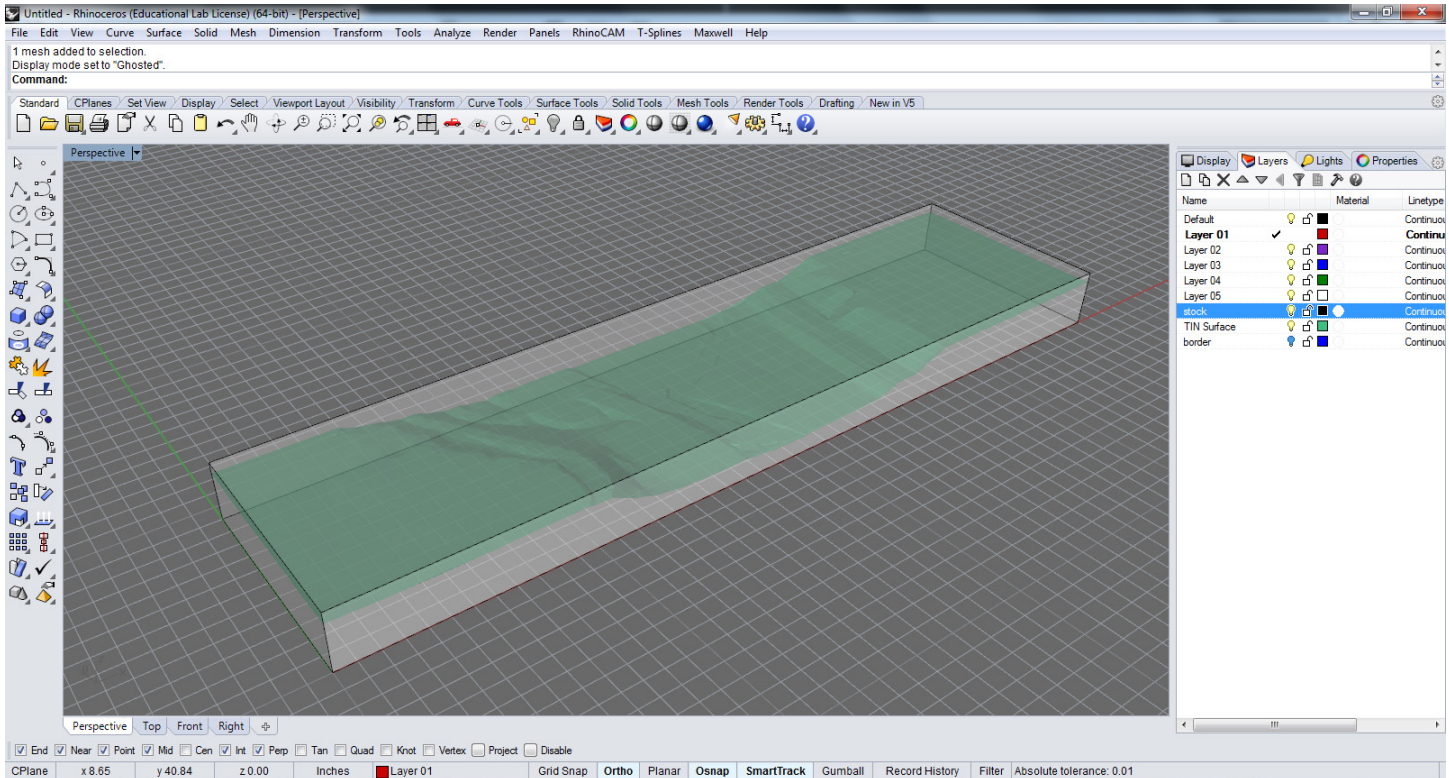
Horizontal Roughing is located in the 3 Axis milling operations, the function is used to define a regions for a surface and milling roughly to the surface. This allows for the rough removal of material before a finishing pass is run to create the final surface. This is done to save time in the finishing passes and to reduce wear on the cutting heads.



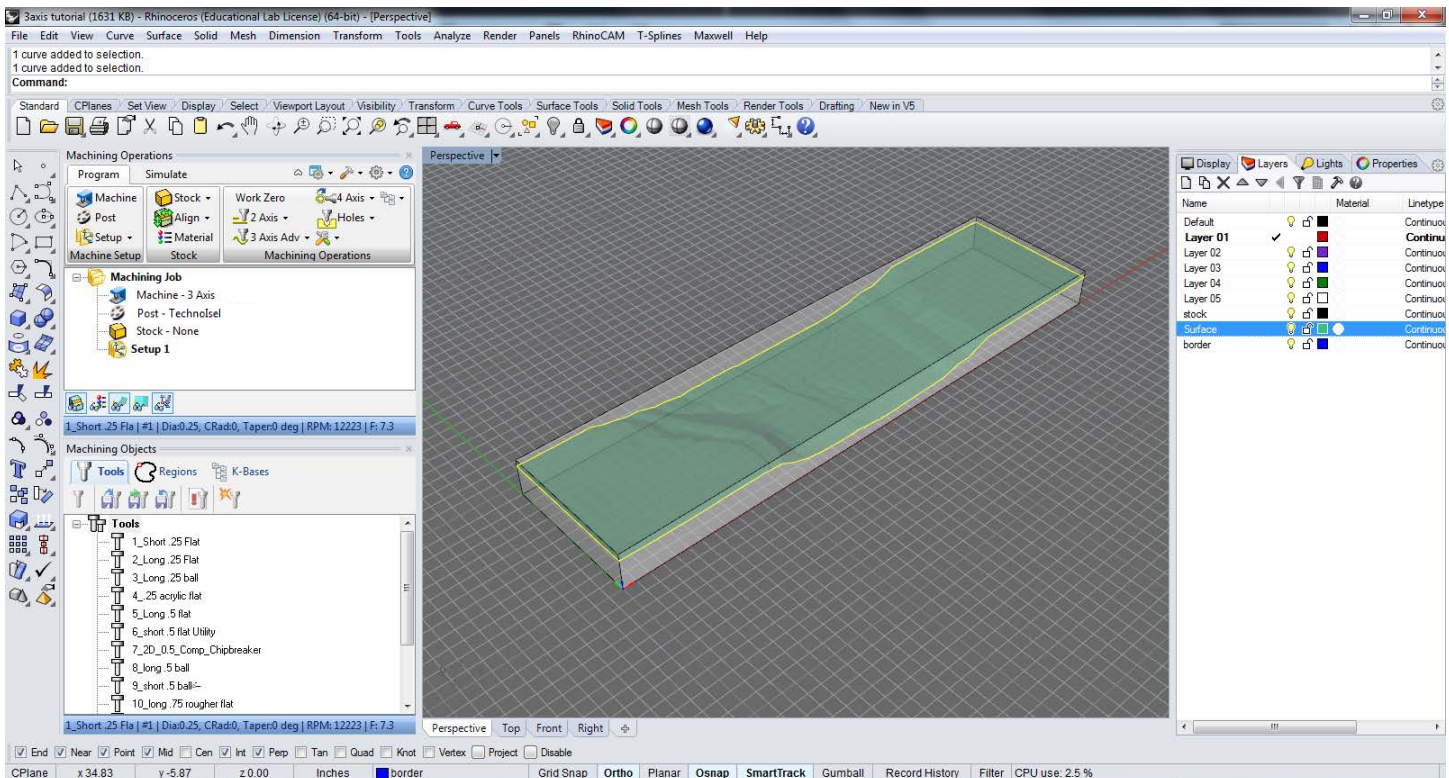
In the sample above a terrain file has been imported into Rhino, the mesh was generated from contour data. The surface will be placed in a piece of stock that will match the physical stock to be milled. Both meshes and nurbs surfaces can be used for 3d milling.



# Rhino\_CAM 3 Axis Machining Operations\_Horizontal\_Roughing

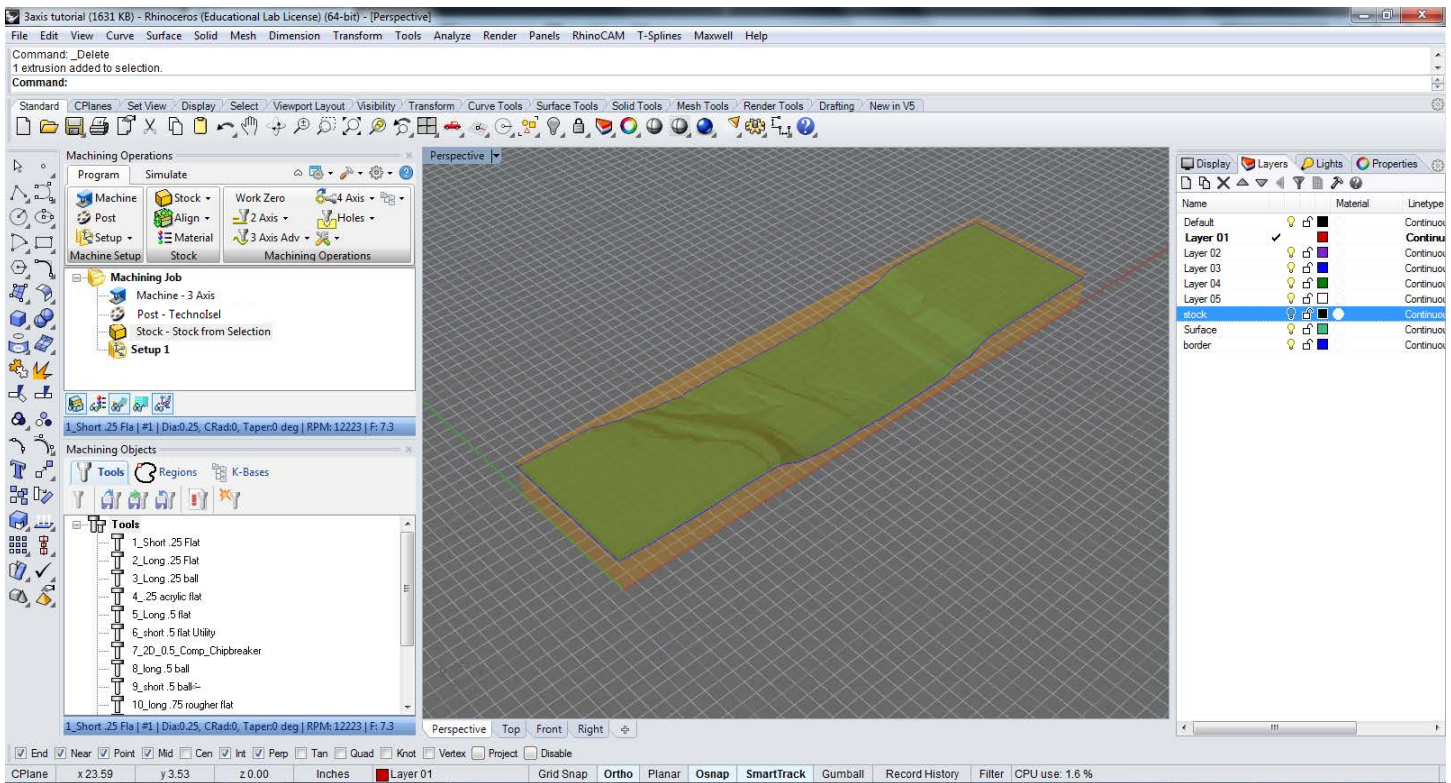


Create a stock to match the physical milling stock. Place the terrain within the stock, and try to allow as little distance as possible between the top of the stock and the terrain to minimize milling time. The deeper the terrain is inset into the stock, the more material needs to be removed for the finished topography.

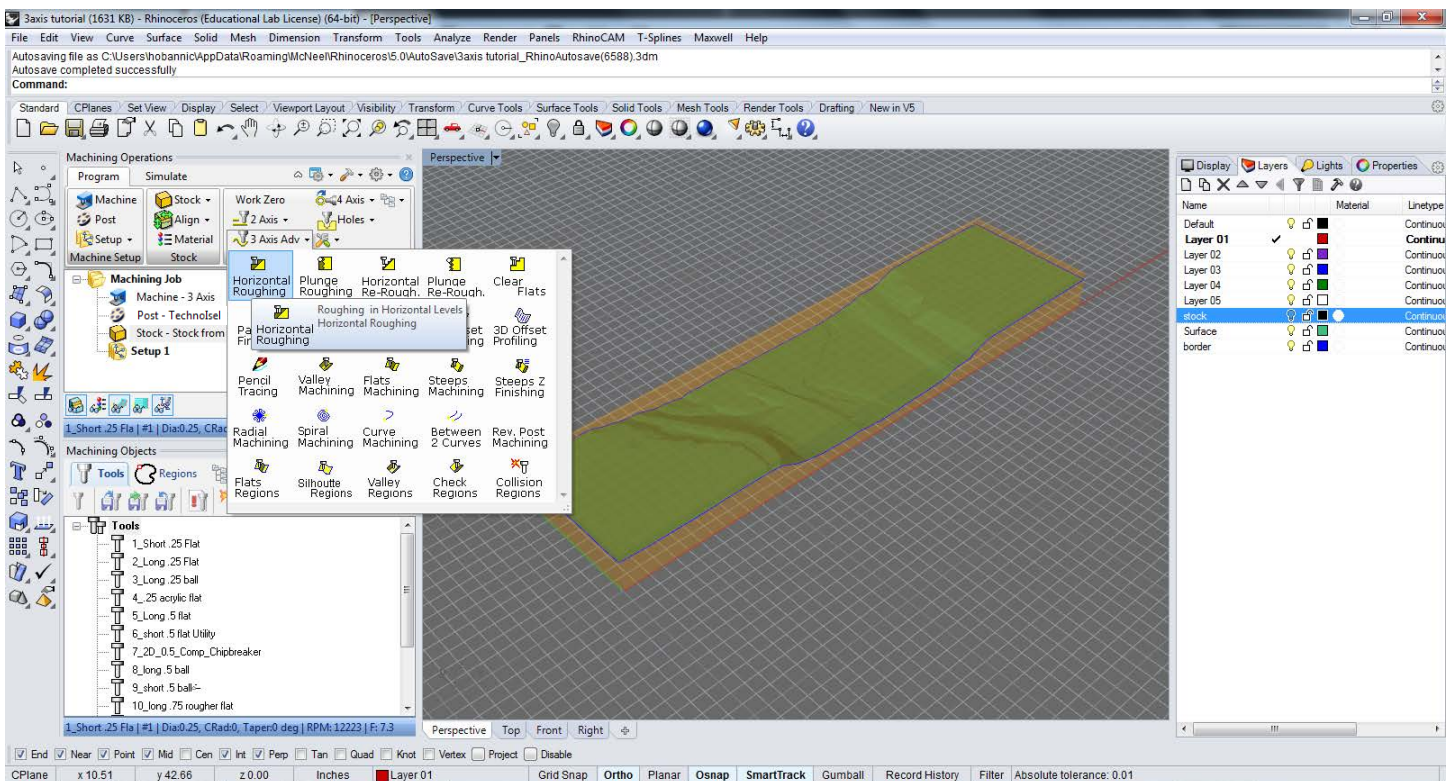


Create a bounding region around the terrain to be milled. This can be a flat region or it can follow the terrain edge. This region will assign the area to be milled.

# Rhino\_CAM 3 Axis Machining Operations\_Horizontal\_Roughing

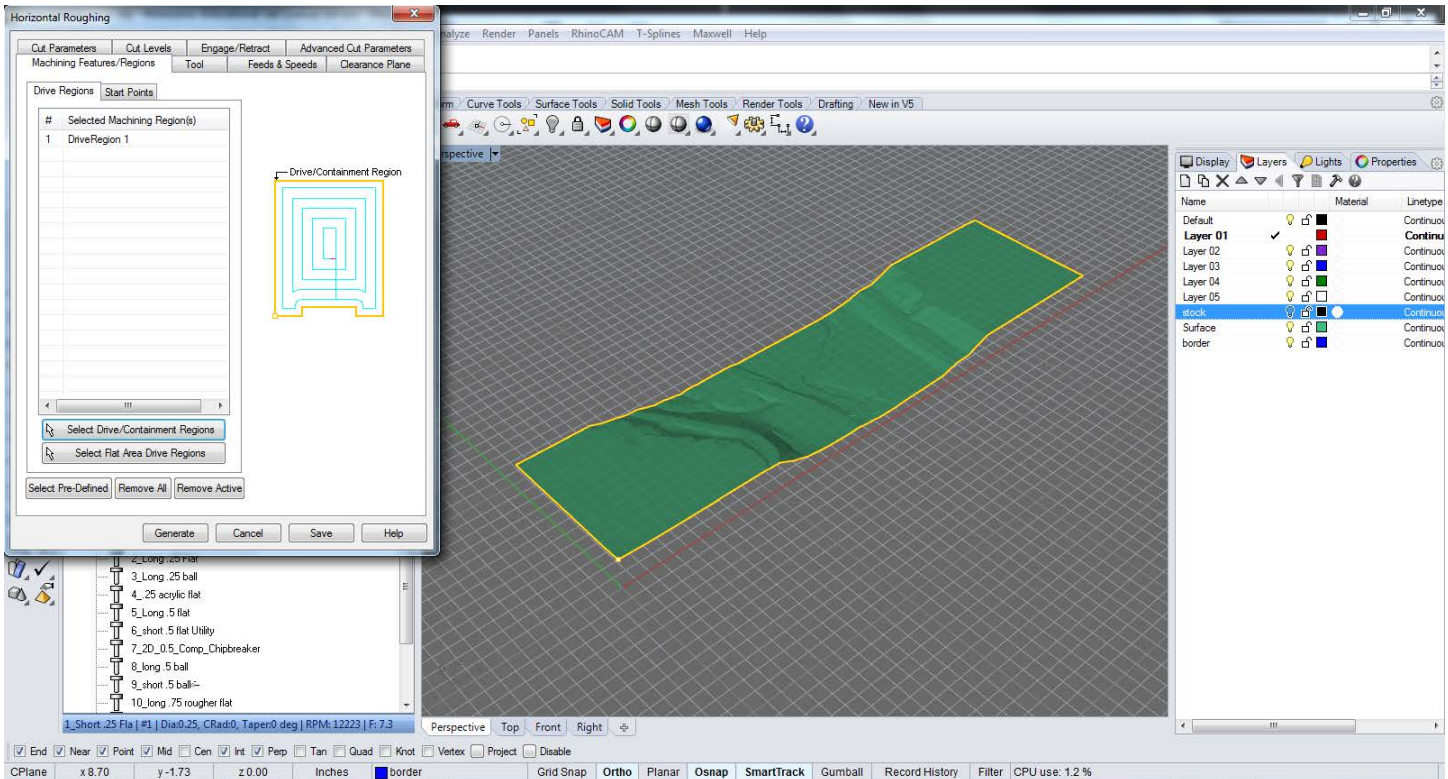


Highlight the stock and from the program tab, choose stock and then stock from selection. This will create the RhinoCAM stock to work within.

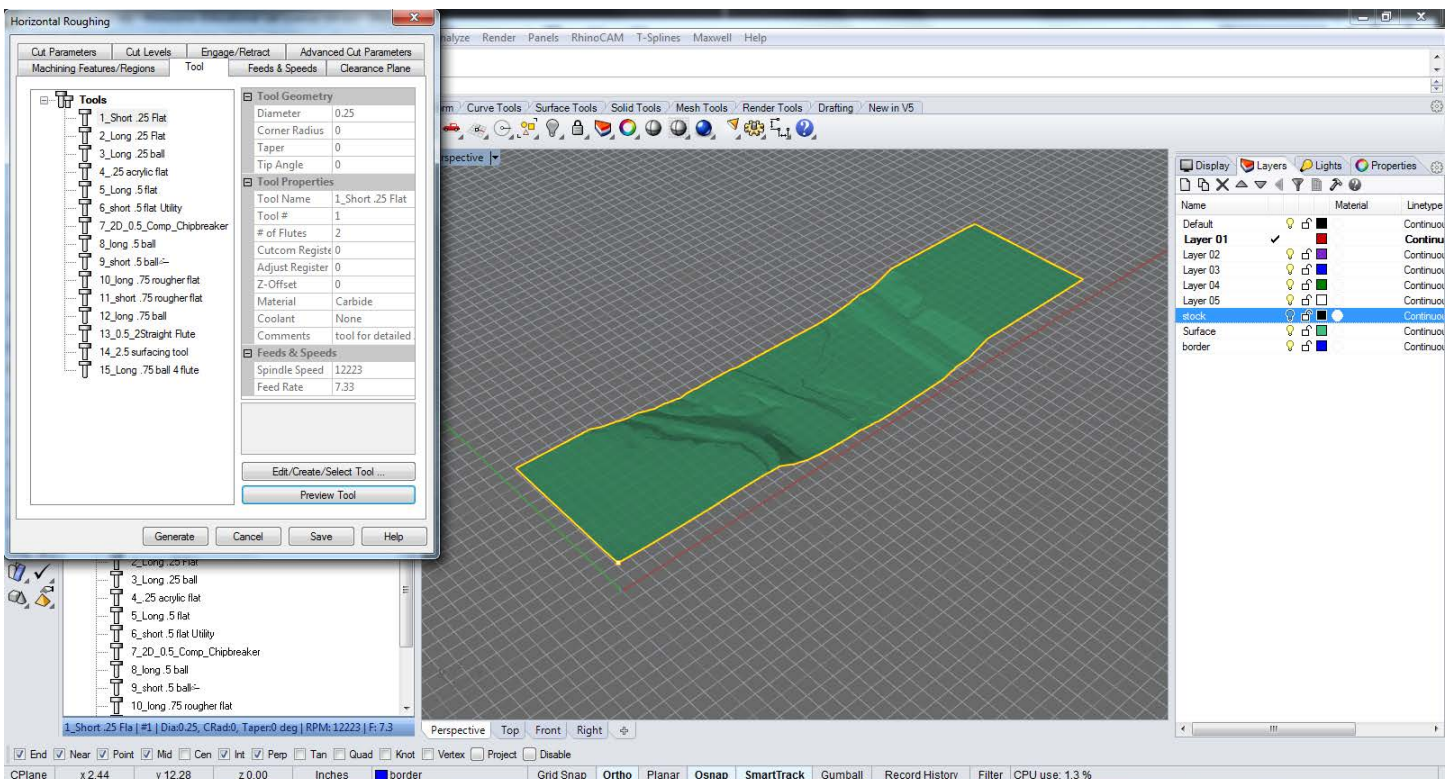


Under the 3 Axis Adv tab, select Horizontal roughing.

# Rhino\_CAM 3 Axis Machining Operations\_ Horizontal\_Roughing



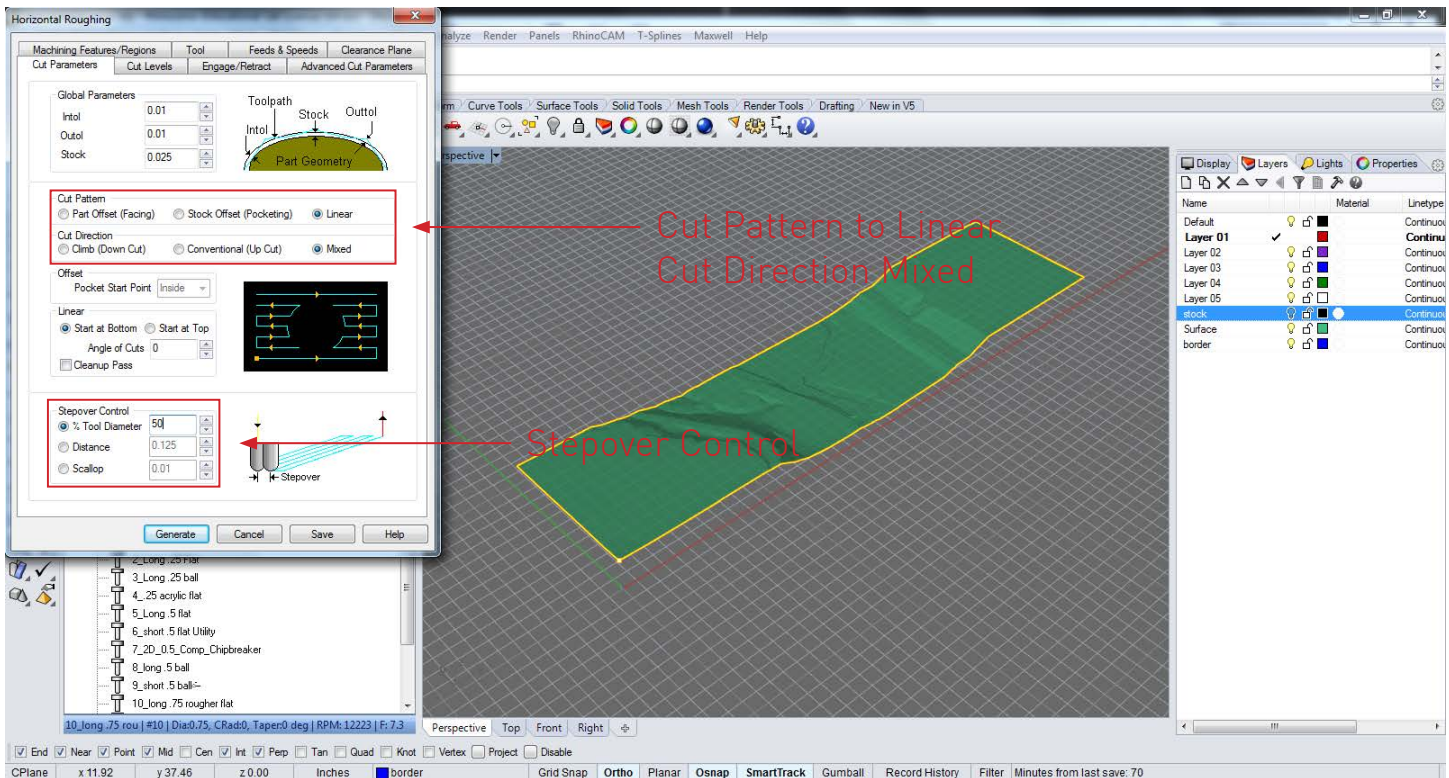
Select the created bounding region to define the area to be milled. Use Select/Drive Containment regions as in the 2 axis milling operations.



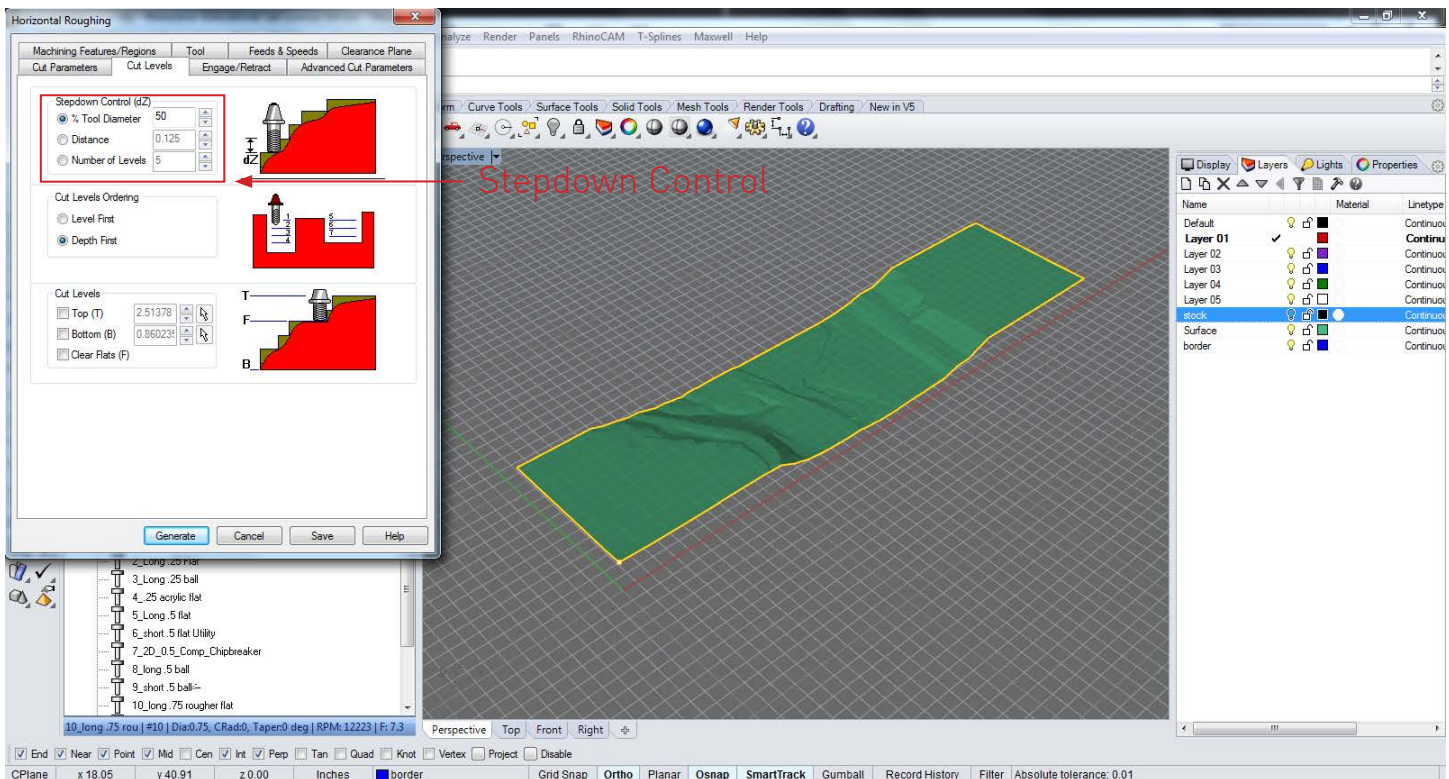
Select the endmill to be used for the roughing operation, typically a larger diameter flat endmill will be used to remove the material to approximate the rough surface.

**Tip:** In plywood, the best results are achieved using a straight, single fluted flat end mill.

# Rhino\_CAM 3 Axis Machining Operations\_ Horizontal\_Roughing



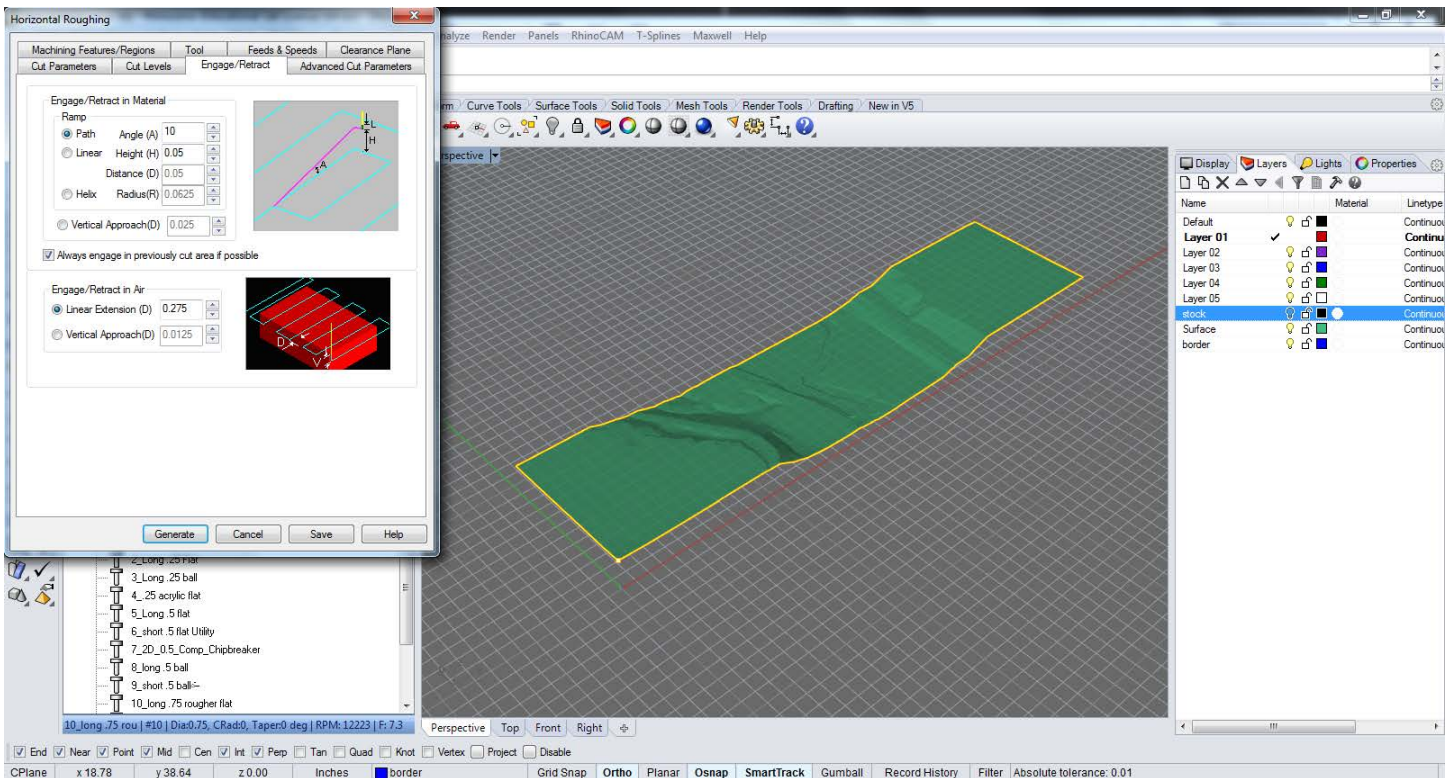
Under 'Cut Parameters,' set the cut pattern to 'linear' and the cut direction to 'Mixed' for materials where direction doesn't matter (e.g., Foam). For stepover control, set to half the endmill diameter for dense materials. For foam this can be increased up to 90%.



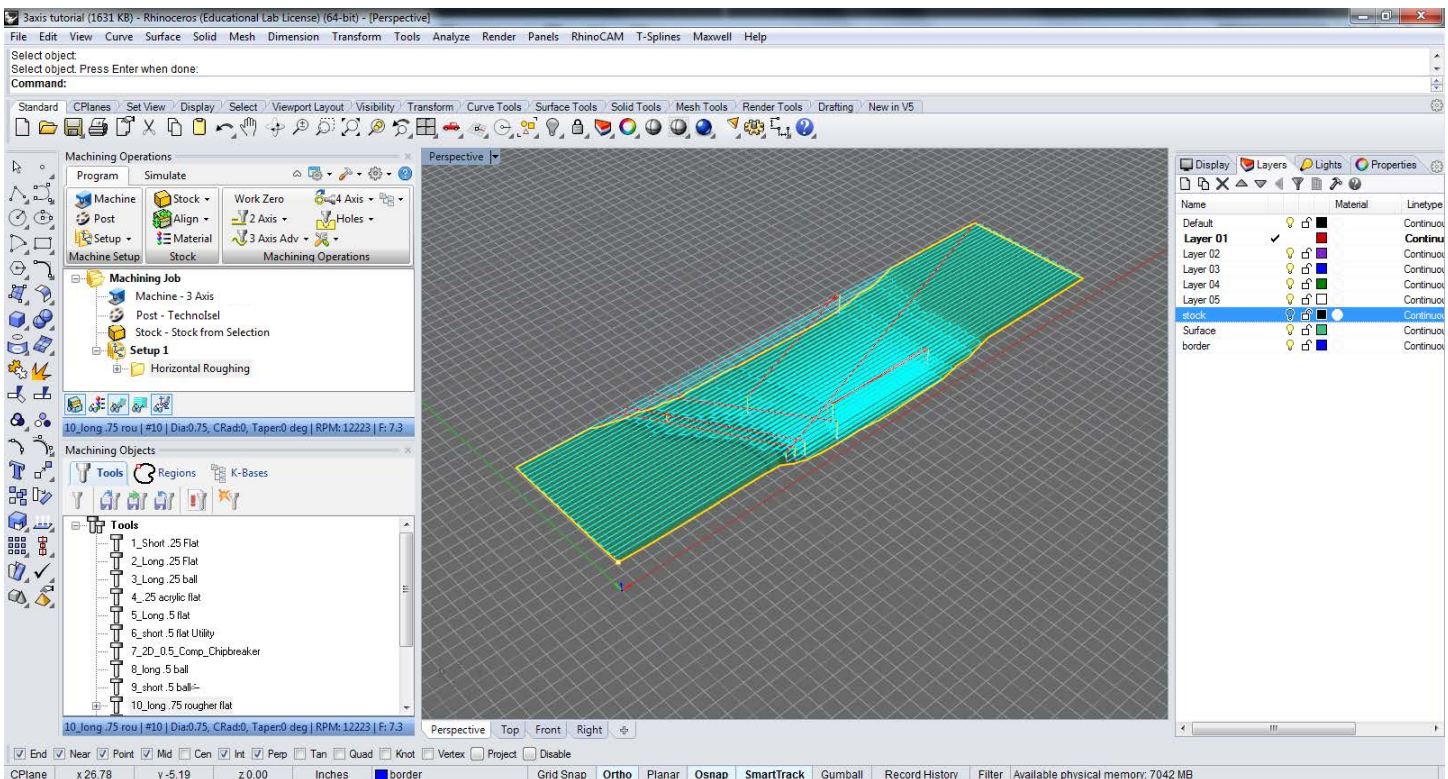
Under 'Cut Levels' the stepdown control can be set, either the distance or the tool diameter can be set to determine the levels for the Horizontal Roughing.



# Rhino\_CAM 3 Axis Machining Operations\_Horizontal\_Roughing

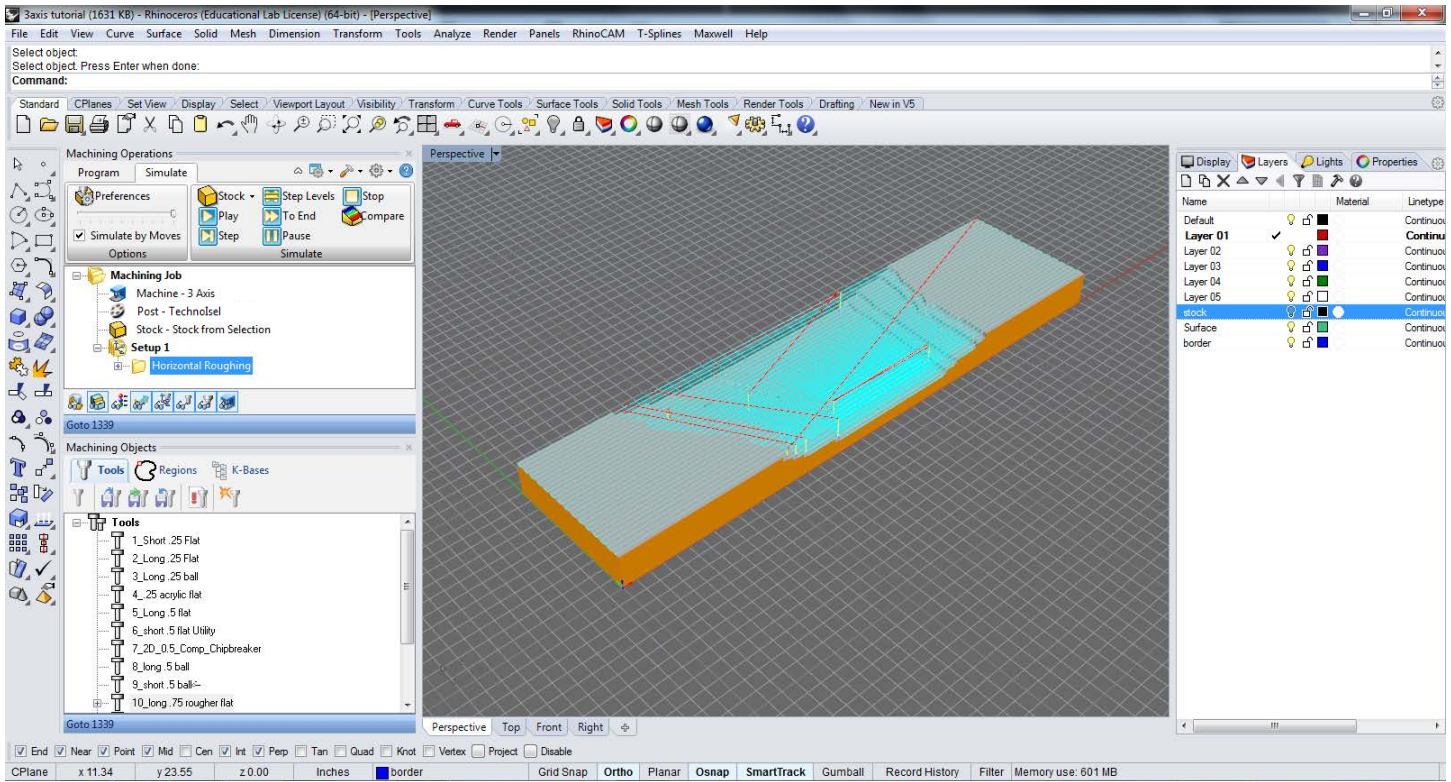


Under the engage/retract tab set engage/retract in material to path, the endmill will follow a path into the beginning tooling path.

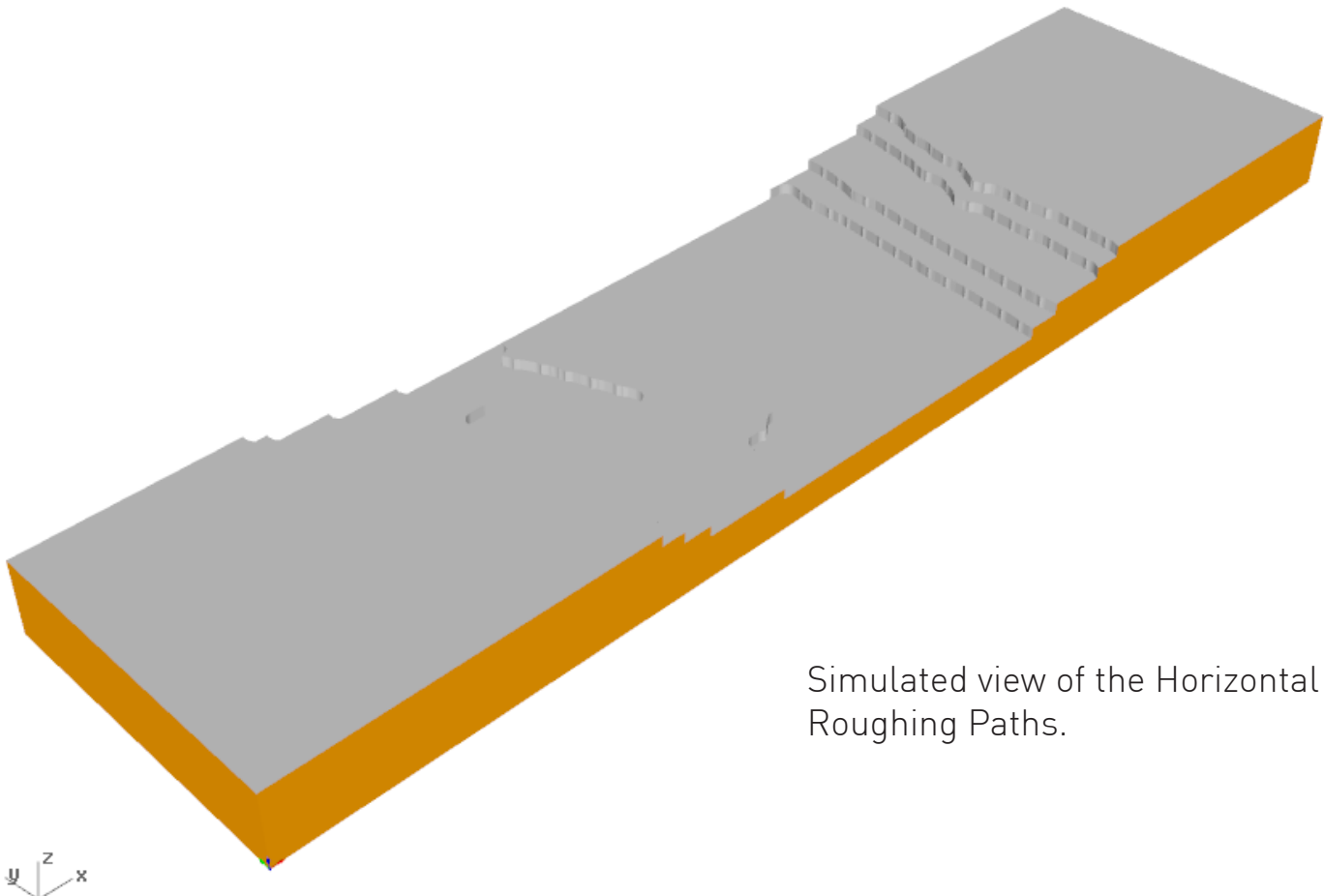


Press generate once all the setup has been completed, the tooling paths with levels will populate the surface.

# Rhino\_CAM 3 Axis Machining Operations\_Horizontal\_Roughing



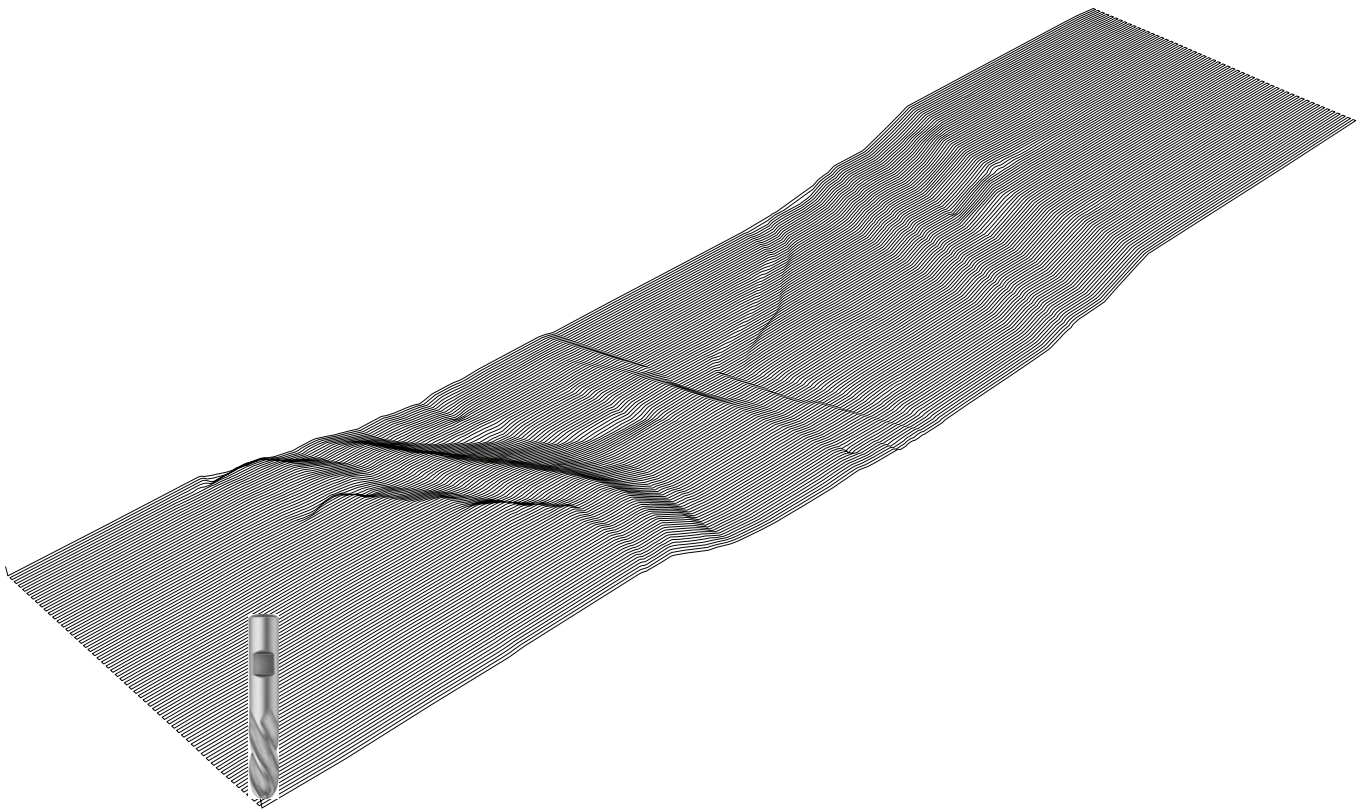
Run the simulation under the simulate tab to make sure there are no collisions or errors in the tooling path setup.



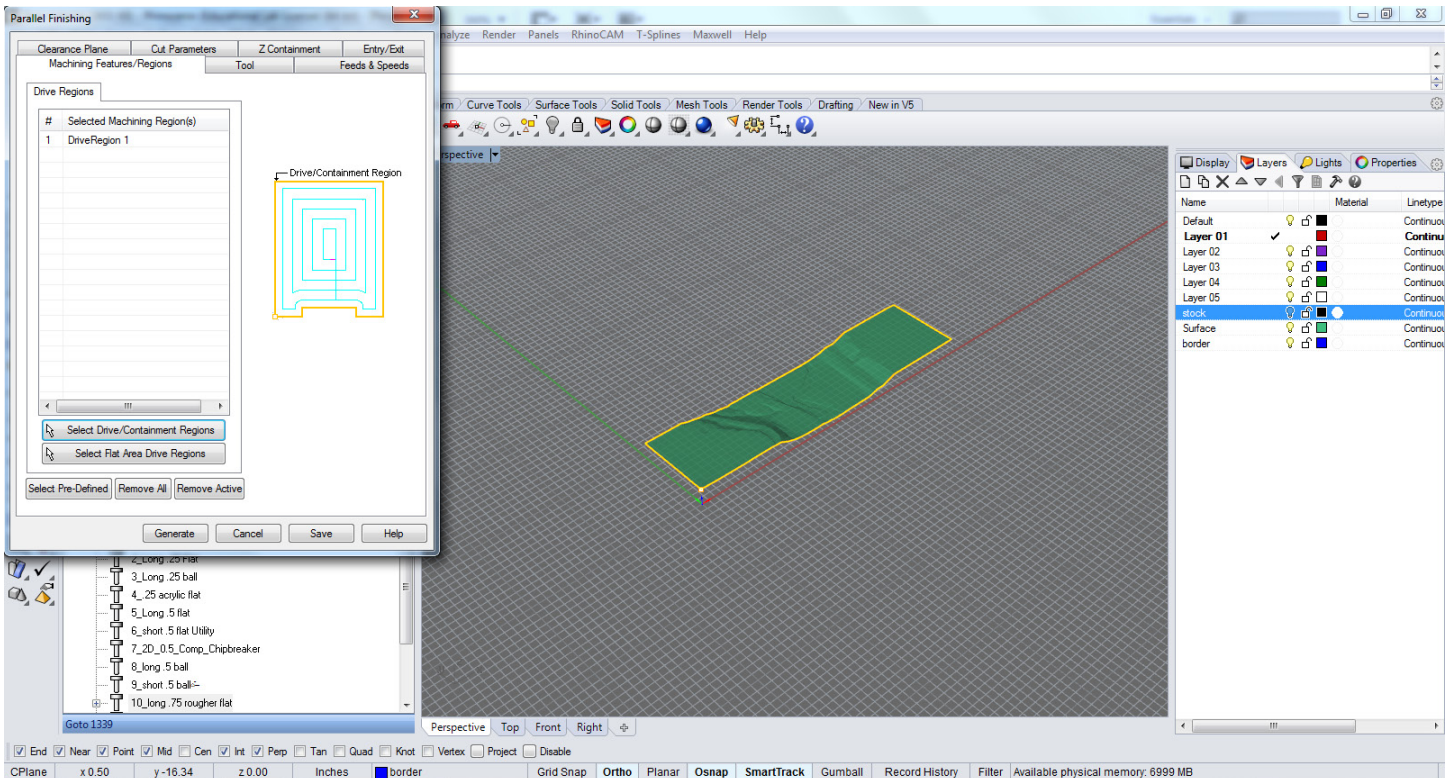
Simulated view of the Horizontal Roughing Paths.

## Rhino\_CAM 3 Axis Machining Operations\_ Parallel\_Finishing

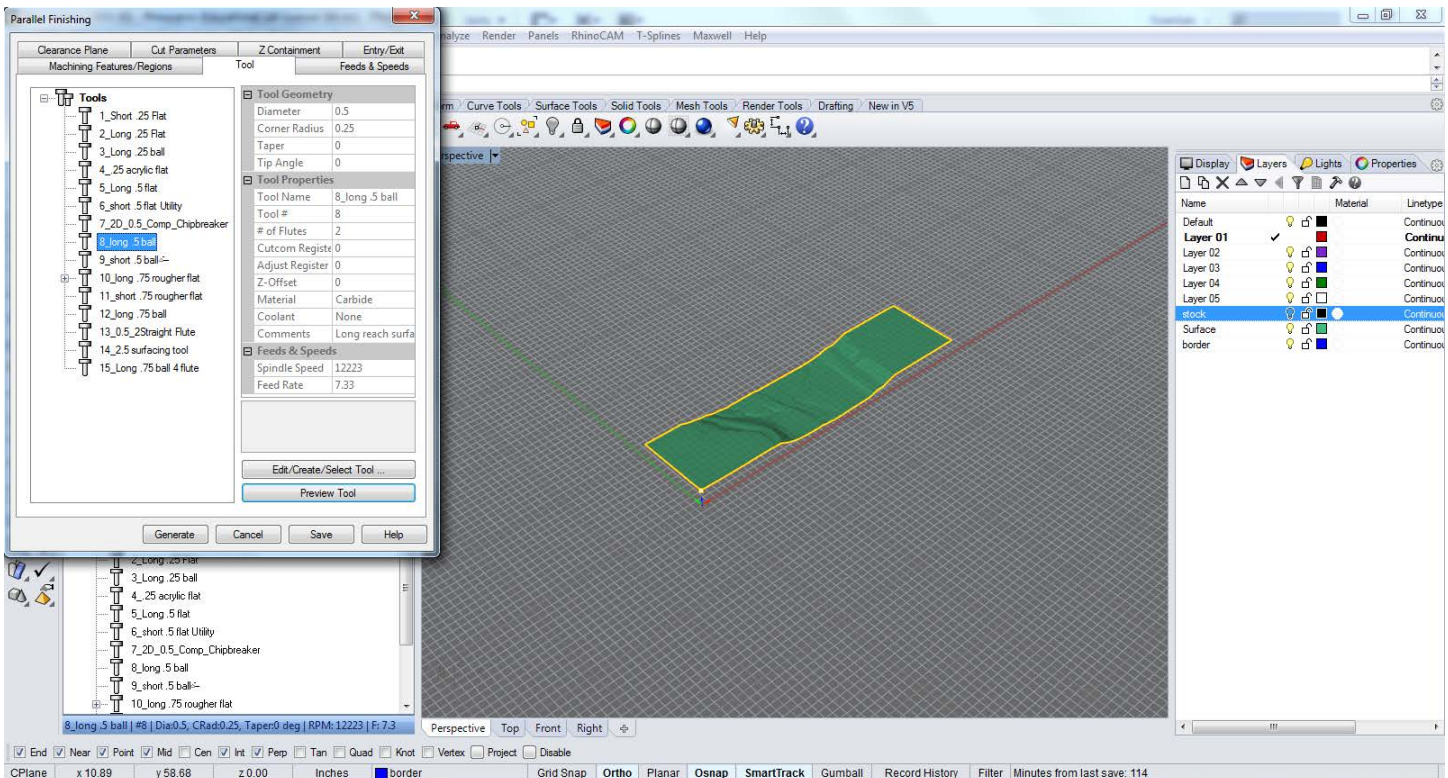
Parallel Finishing operation follows the finish surface with the chosen endmill, the tooling paths create different material effects depending on the endmill chosen. For tight geometry a small endmill is necessary to fit into the terrain or surface, a larger diameter endmill can be used but it will approximate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



# Rhino\_CAM 3 Axis Machining Operations\_ Parallel\_Finishing

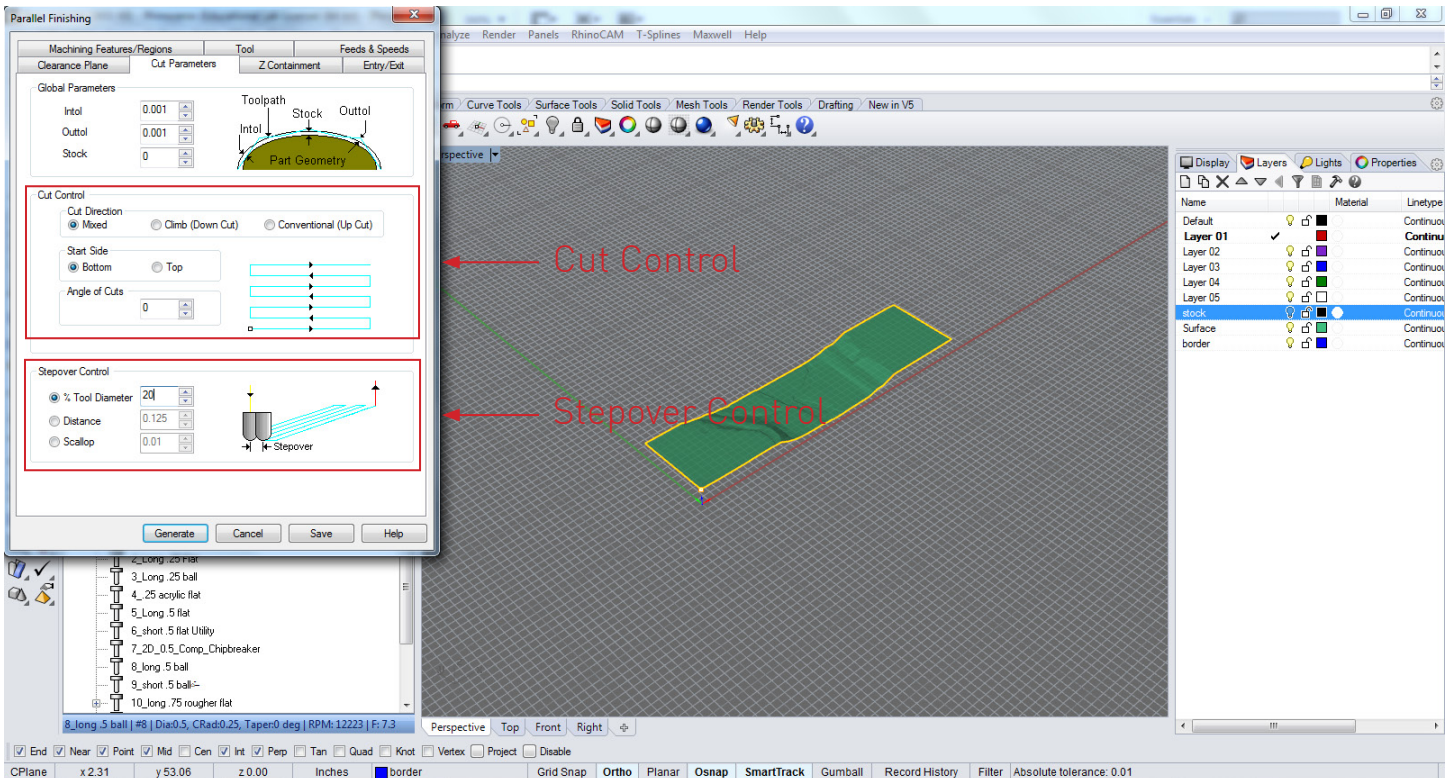


Select Parallel Finishing from the 3 Axis Adv drop down menu, select the drive region to be milled. This will be the same region used in the Horizontal Roughing pass.

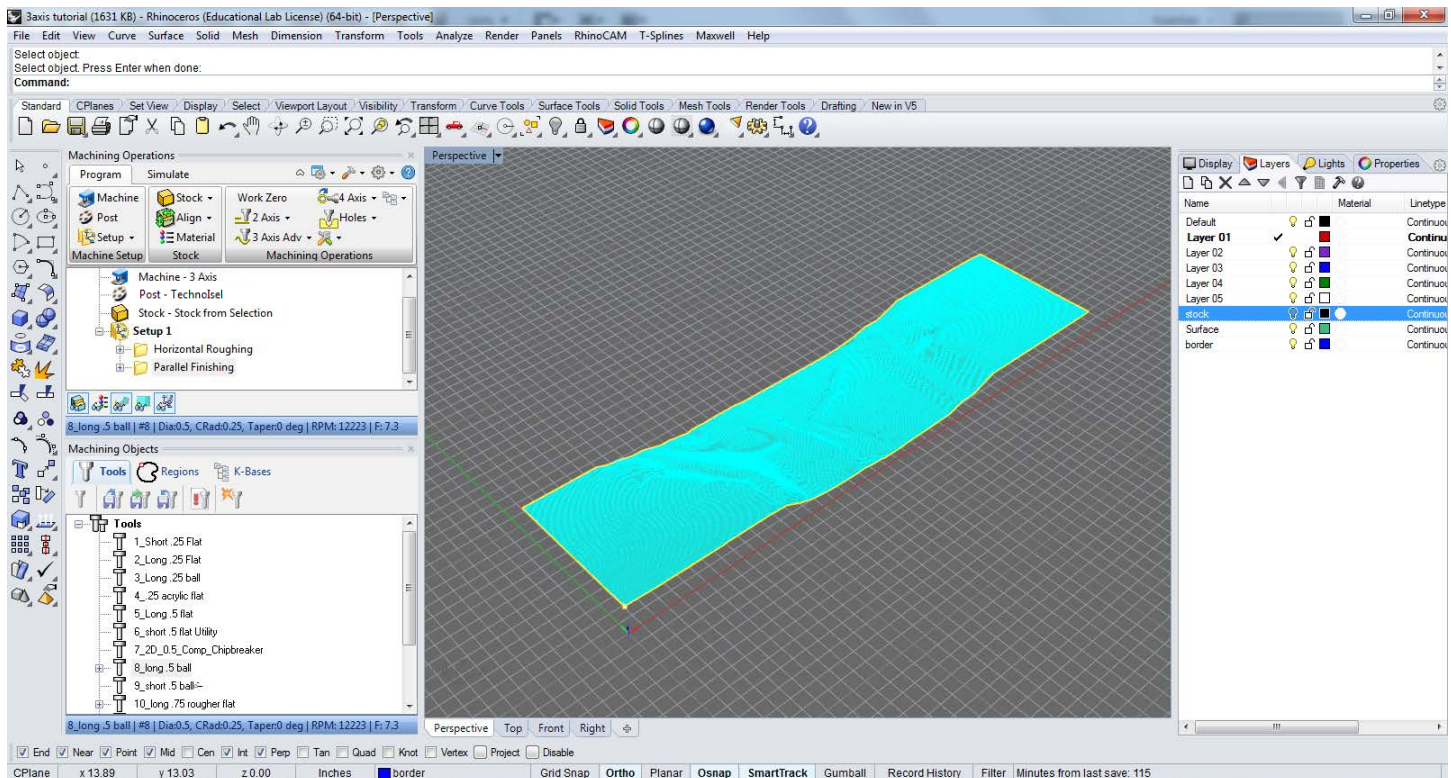


Under the Tool Selection tab, select the tool used for the final finished surface mill. For the topography a .5 ball endmill is selected to give a smooth finish with a slight scallop from the stepover of the ball endmill. Other tools will produce different results.

# Rhino\_CAM 3 Axis Machining Operations\_ Parallel\_Finishing

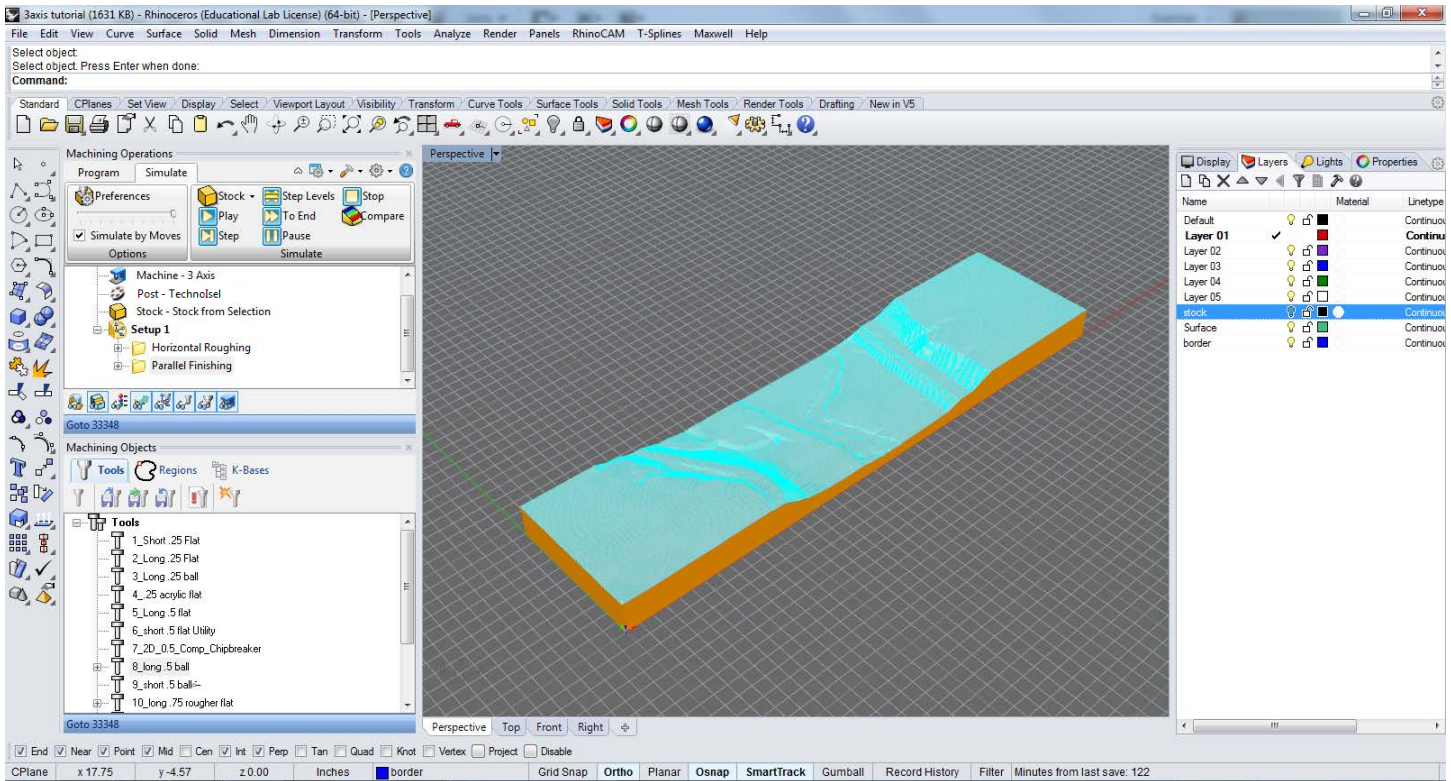


Under Cut Parameters set up the Cut Control, the direction is set to mixed and the angle is set to zero, if angled passes are desired set the angle of cuts to the chosen angle. The stepover control is set to 20%, the tighter the stepover the longer the milling time.

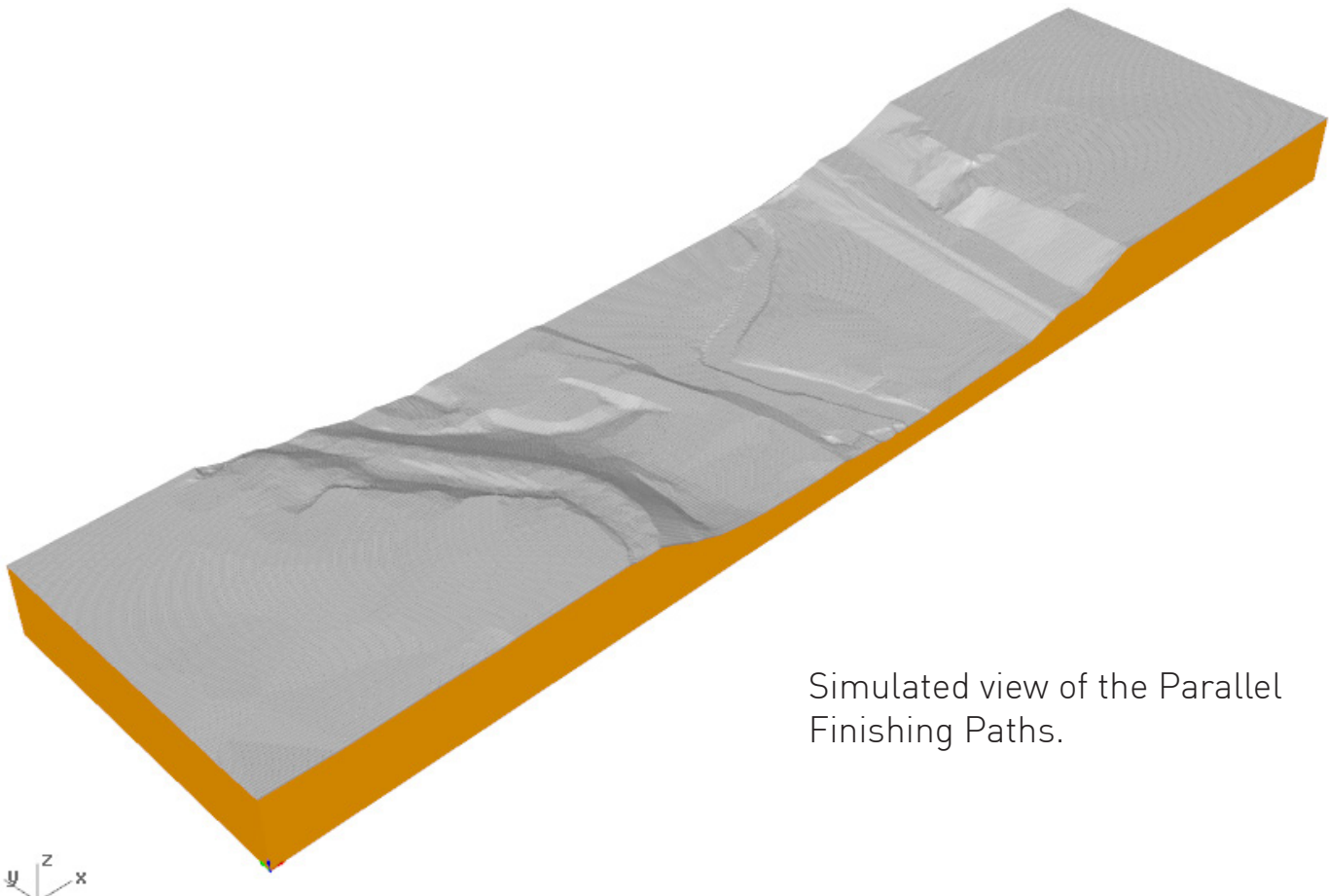


Once all the parameters have been set up press generate and the surface will be populated with milling paths following the finish surface of the topography.

# Rhino\_CAM 3 Axis Machining Operations\_ Parallel\_Finishing



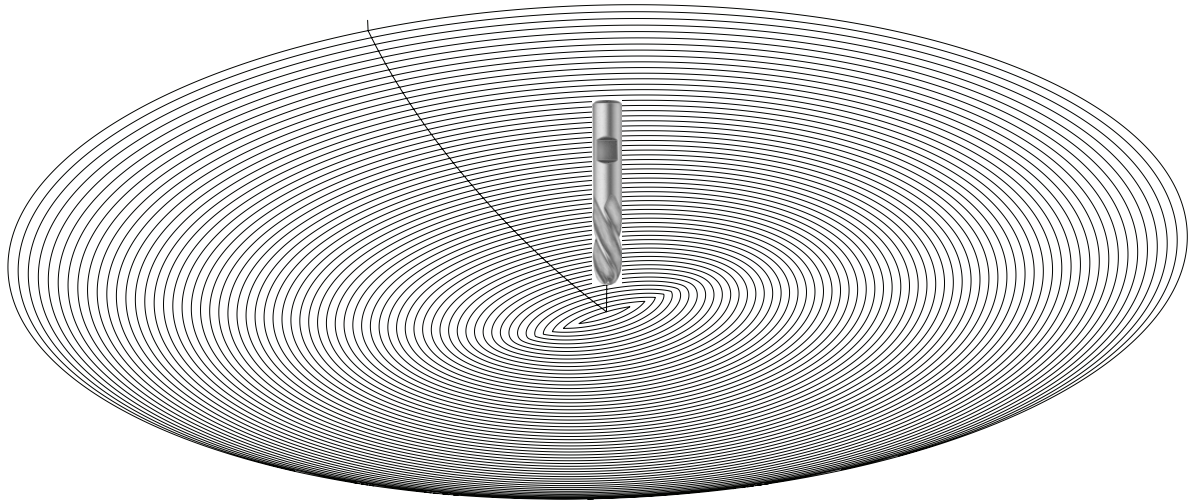
Under the simulate tab run the simulation to ensure that there are no errors or collisions in the tooling paths.



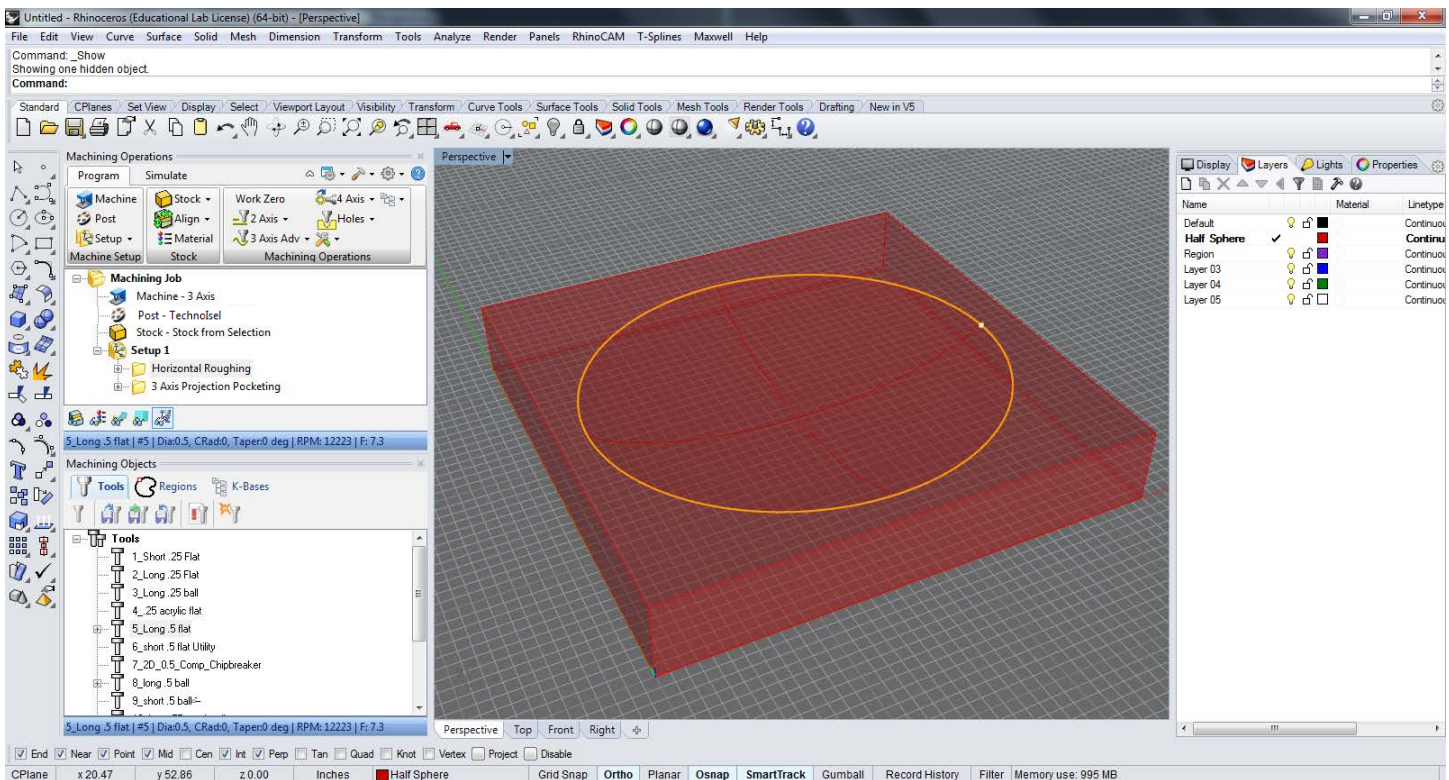
Simulated view of the Parallel Finishing Paths.

## Rhino\_CAM 3 Axis Machining Operations\_ Projection\_Pocketing

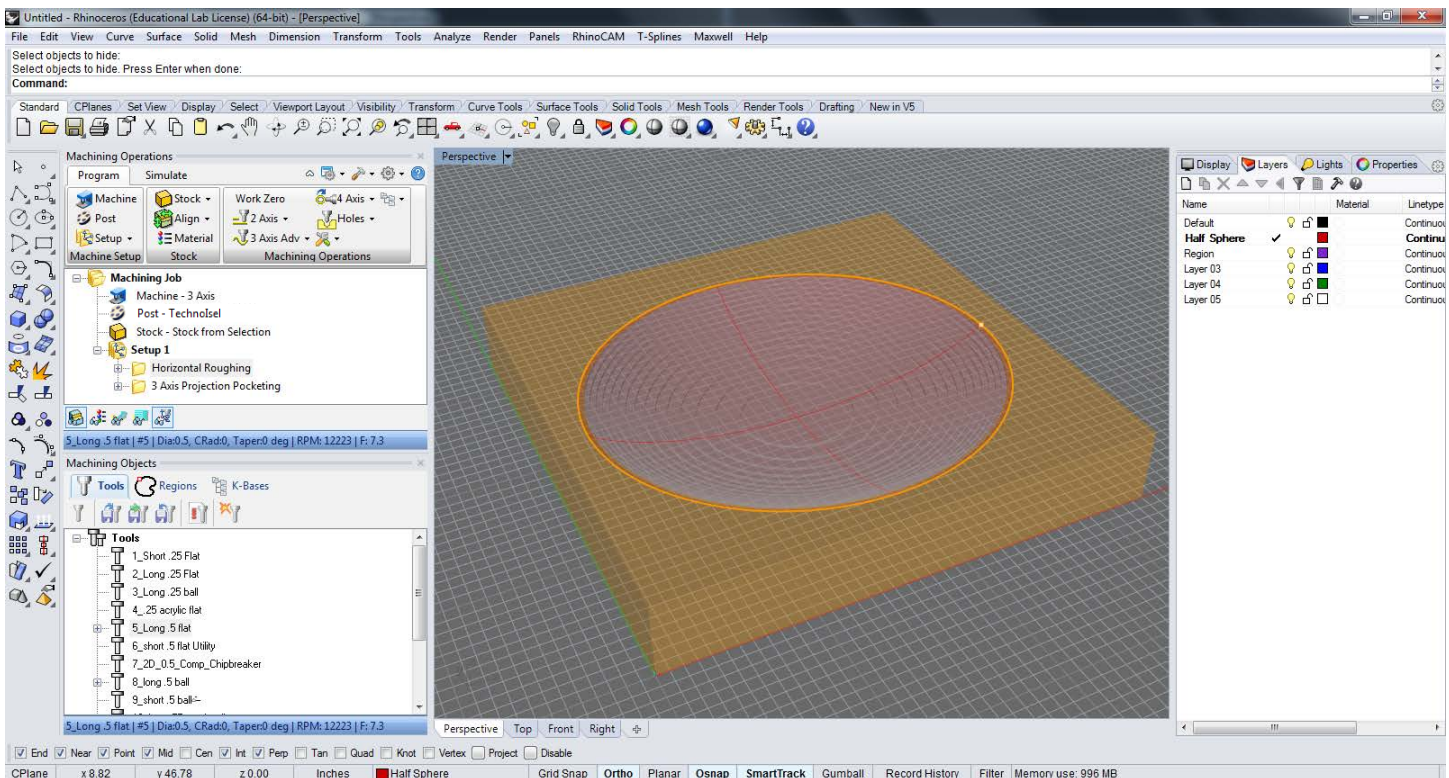
Projection pocketing operation can be used for Pre finishing or fine finishing surface bottoms in closed regions. In this method, the toolpath is generated in 2D and then projected down to the surfaces below. Machining regions are necessary to be active for this cut method to work. There is no limitation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



# Rhino\_CAM 3 Axis Machining Operations\_Projection\_Pocketing



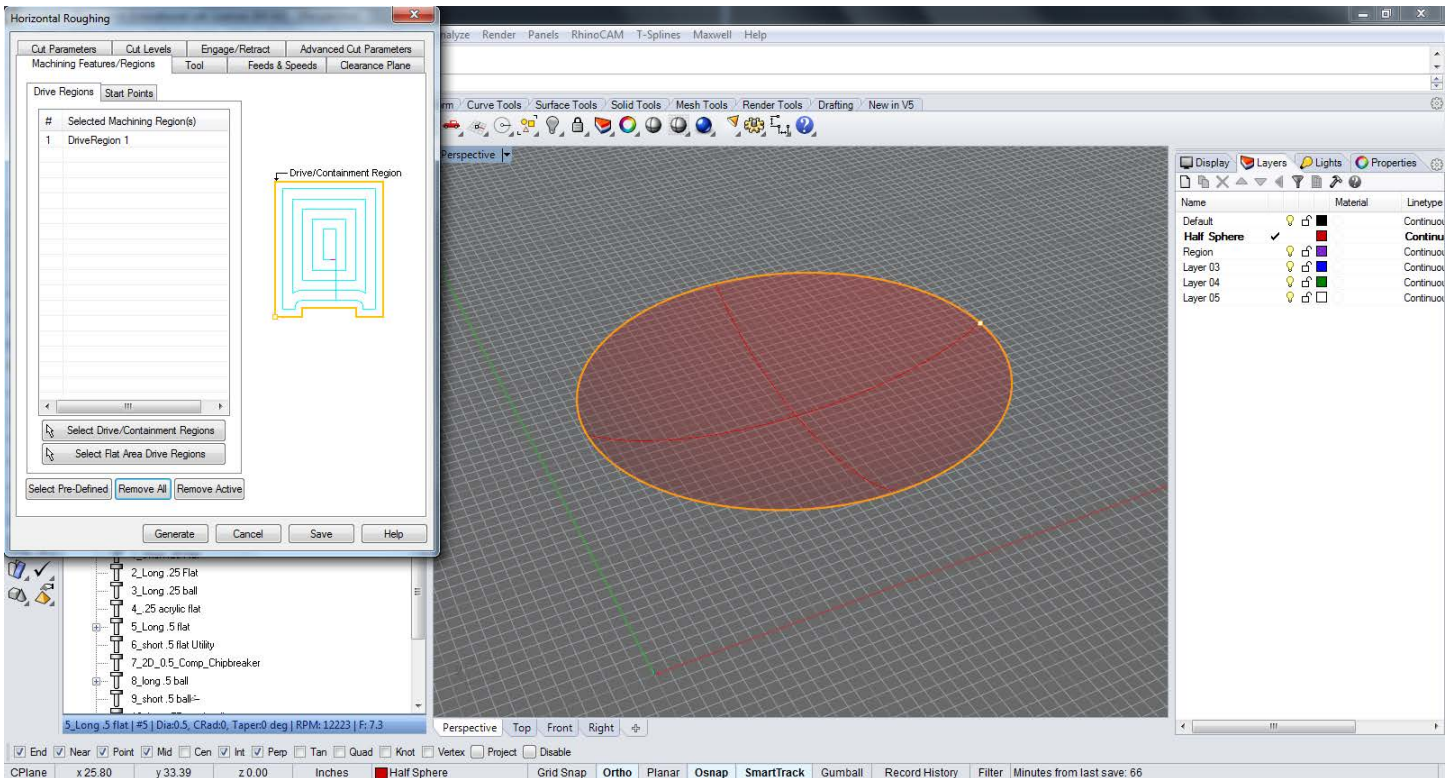
Before the Pocket Projection pass we will start with a Horizontal Roughing pass to clear out most of the material from inside the dome before the finishing pass.



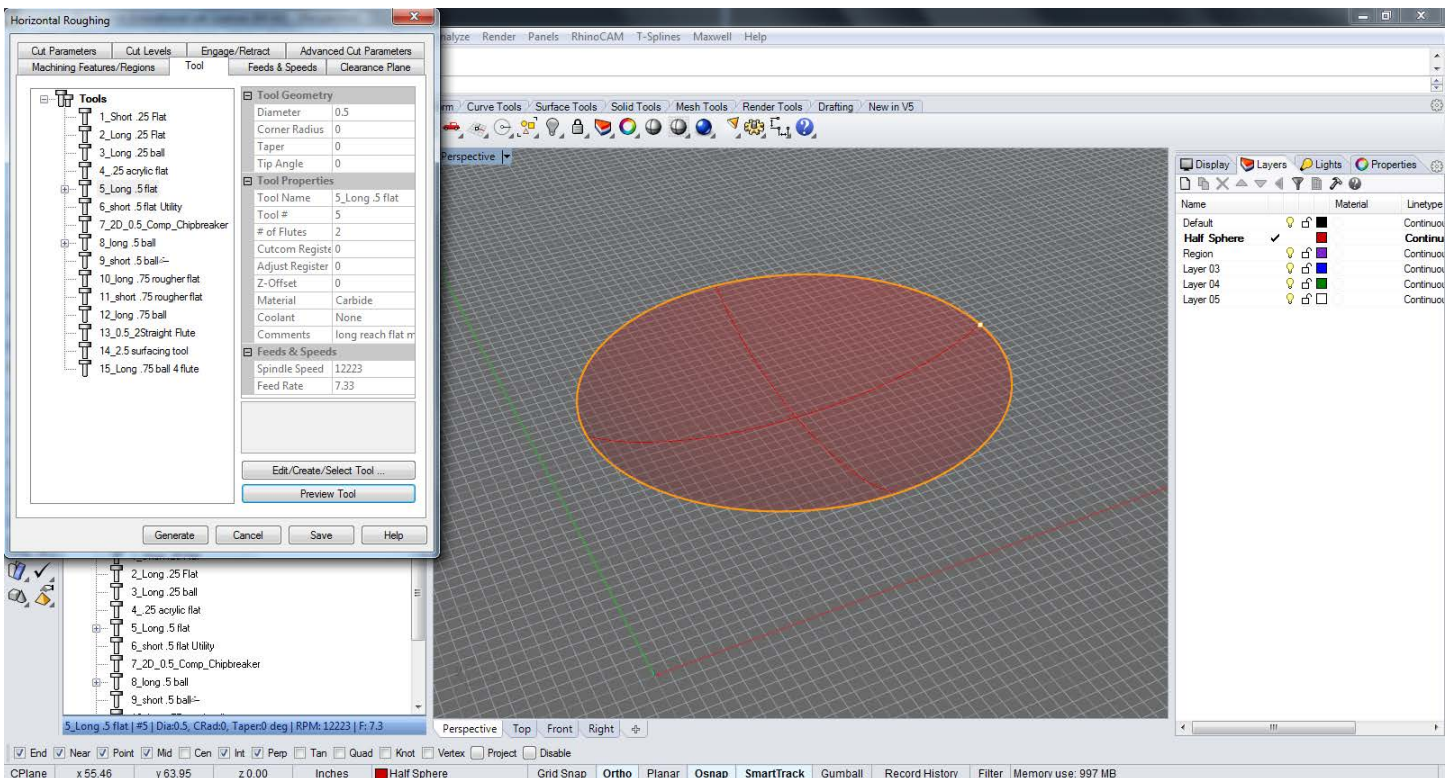
Create your stock using the 'Bounding Box' option, then create a bounding region that encompasses the area you wish to mill during the roughing pass.



# Rhino\_CAM 3 Axis Machining Operations\_Projection\_Pocketing

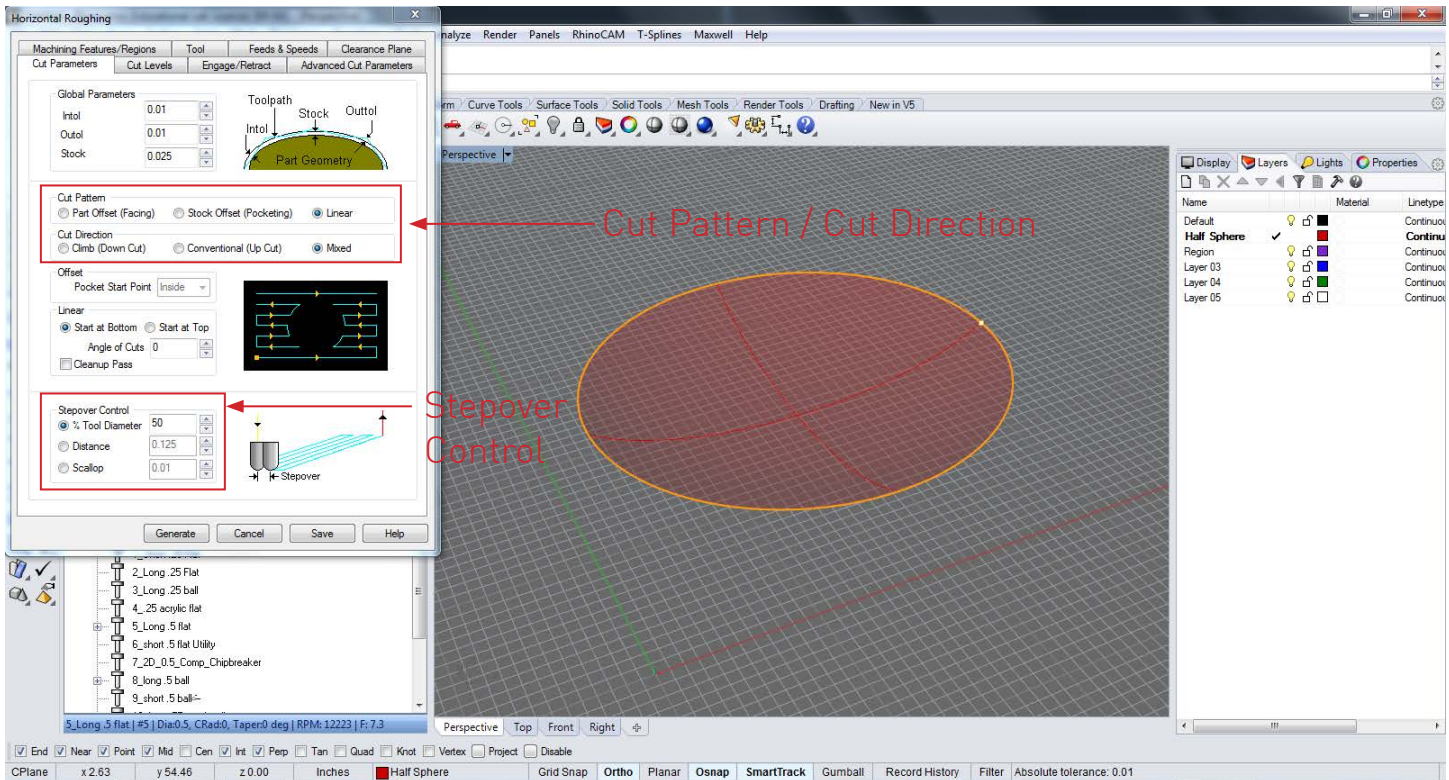


Select Horizontal roughing from the 3 axis adv drop down menu, select the drive region, this is the **bounding** region just created for the roughing pass.

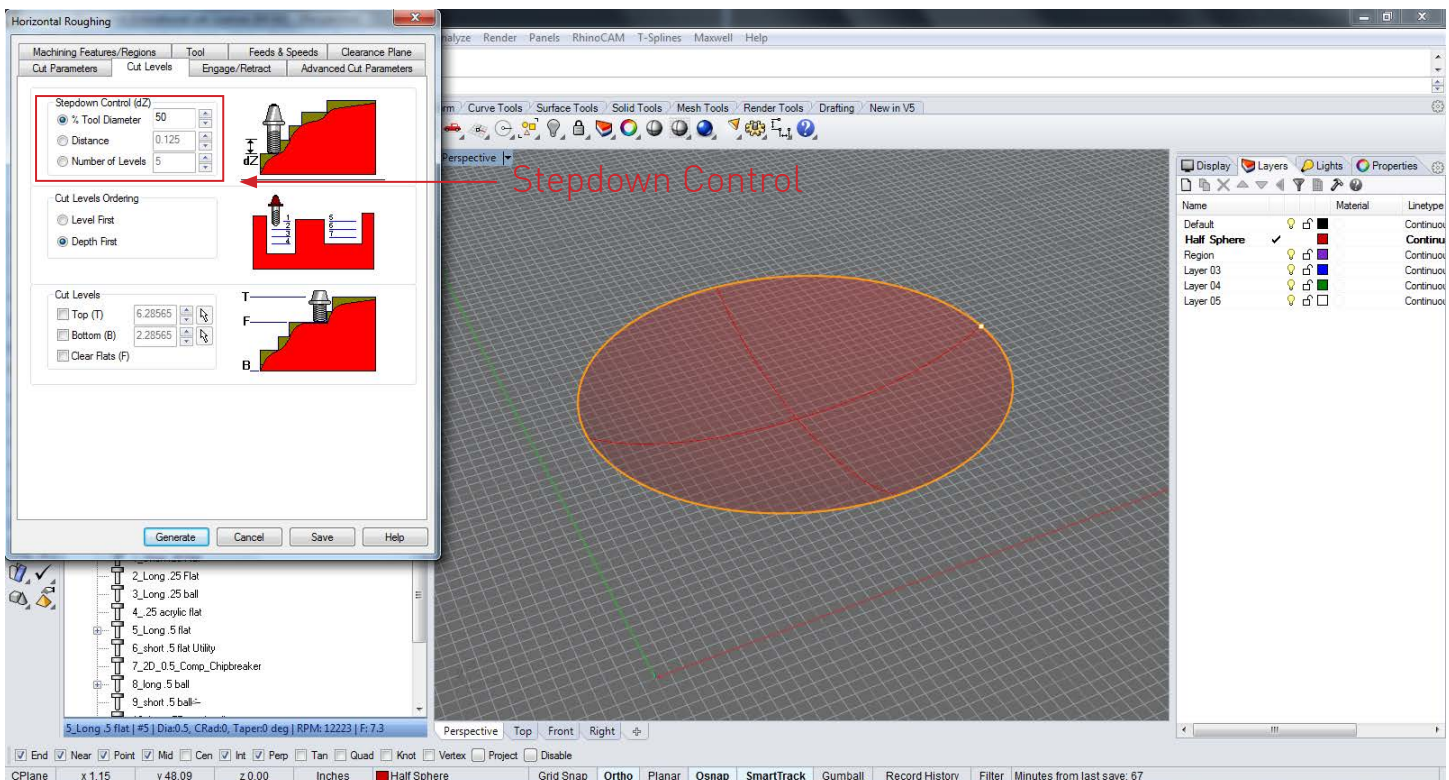


Select the proper endmill, depending on the material being milled and the cutting depth required. In this example a 1/2" flat end mill is chosen from the UMass A+D Tool Library.

# Rhino\_CAM 3 Axis Machining Operations\_ Projection\_Pocketing

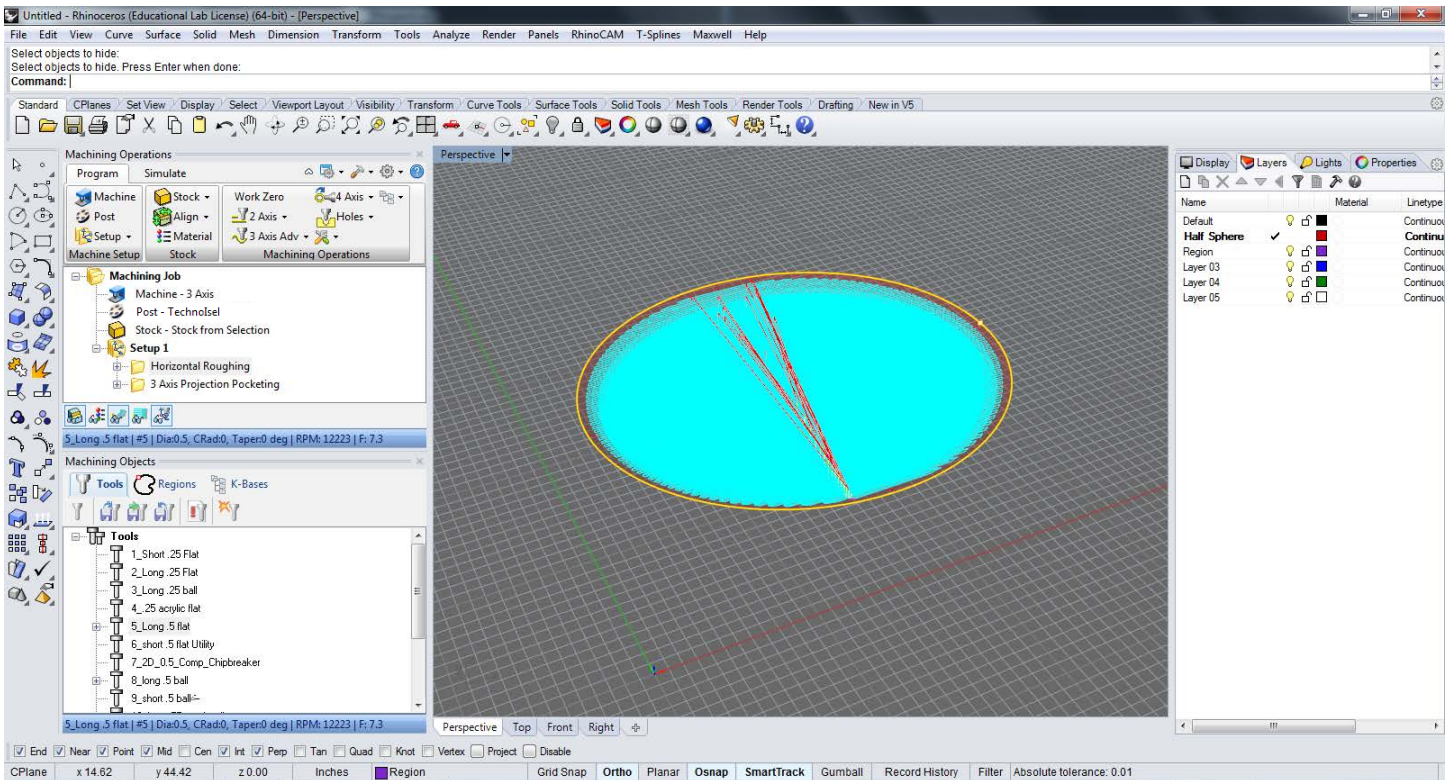


Under the Cut Parameters tab set the cut pattern to Linear and the cut direction to mixed, the stepover should be set accordingly to the material being milled.

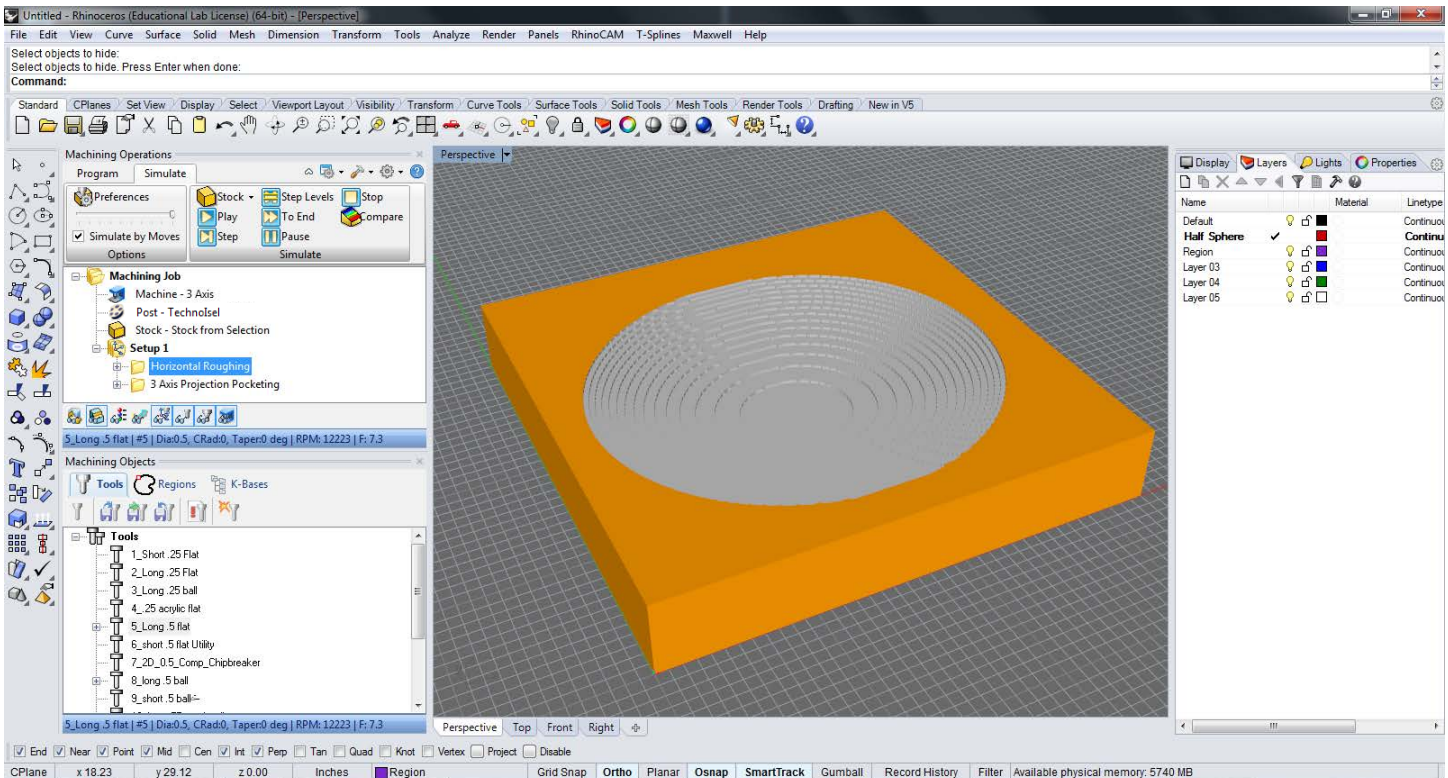


Under cut levels set the stepdown control to 50 percent of the tool diameter, this percentage can be higher for less dense material (ie foam).

# Rhino\_CAM 3 Axis Machining Operations\_Projection\_Pocketing

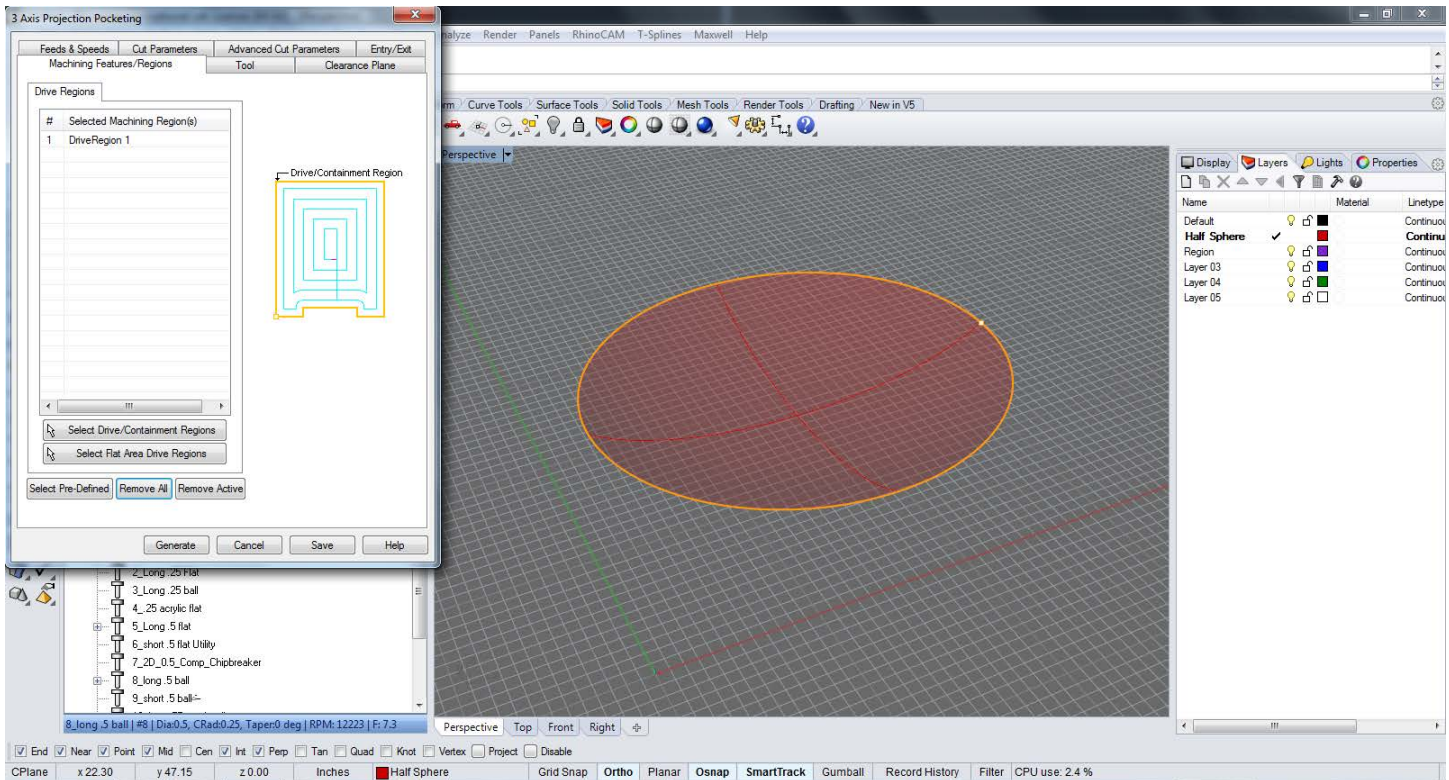


Once all the settings have been completed press generate to view the tooling paths created by the input parameters.

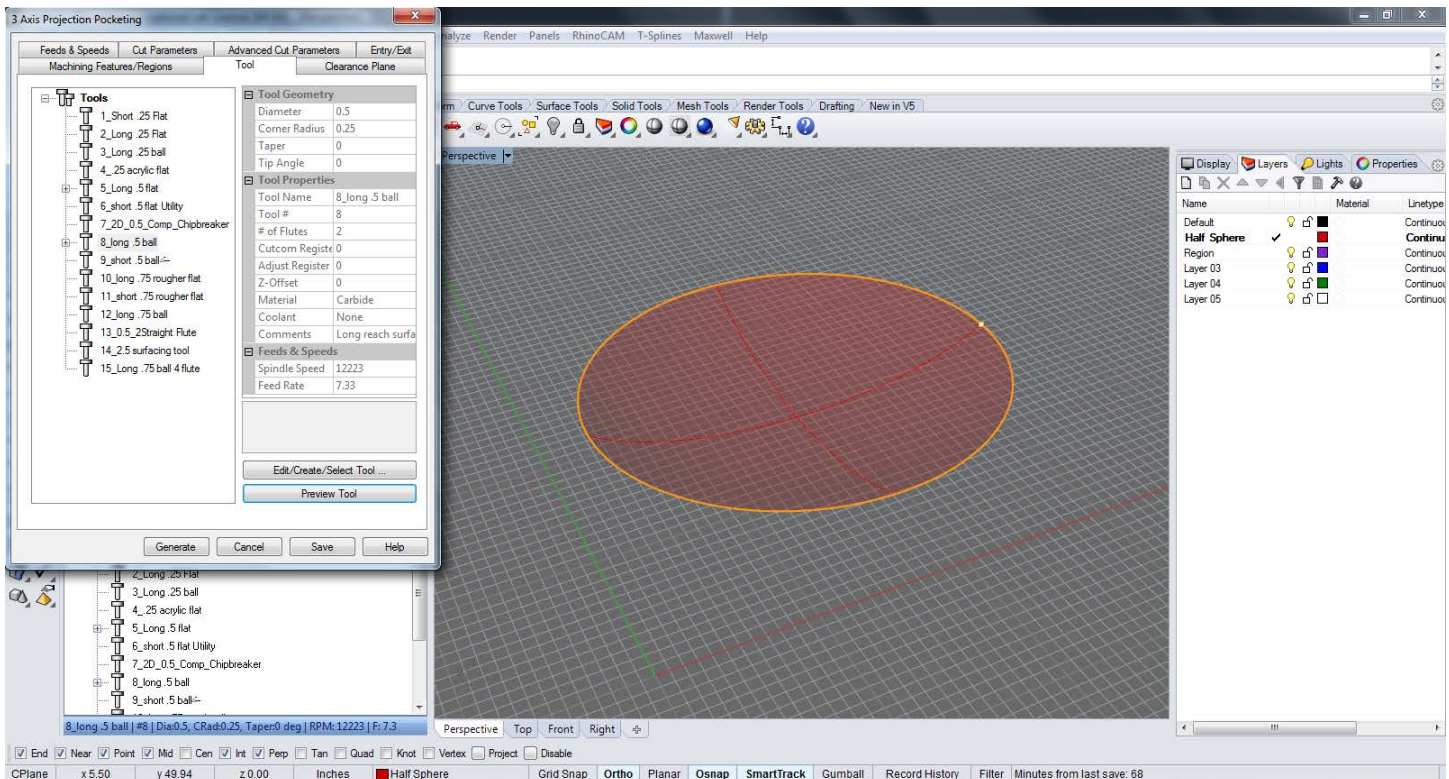


Go to the Simulate Tab and run the simulation to make sure there are no collisions or errors in the tooling path setup.

# Rhino\_CAM 3 Axis Machining Operations\_Projection\_Pocketing

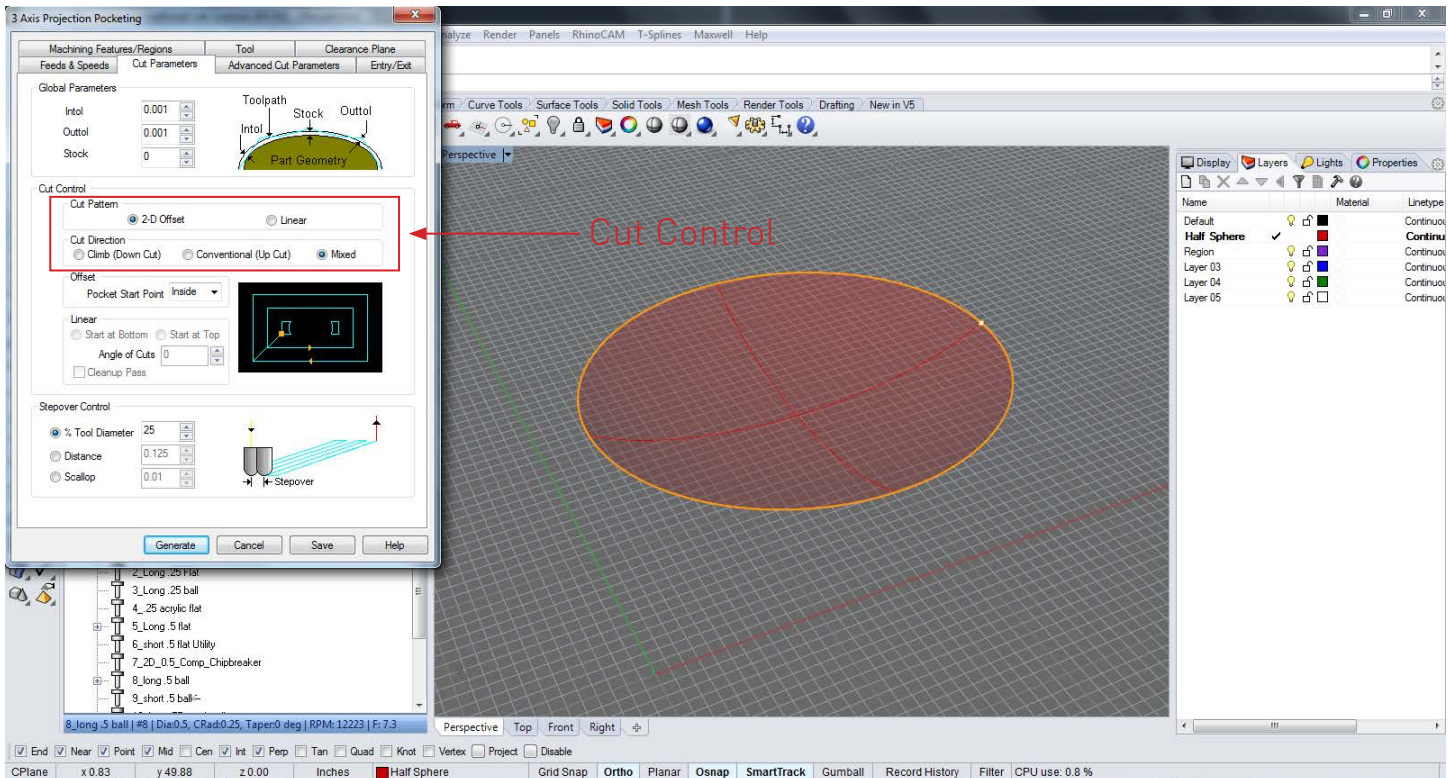


Select Projection Pocketing from the 3 axis adv menu, select the same drive region that was just used the in the previous horizontal roughing pass.

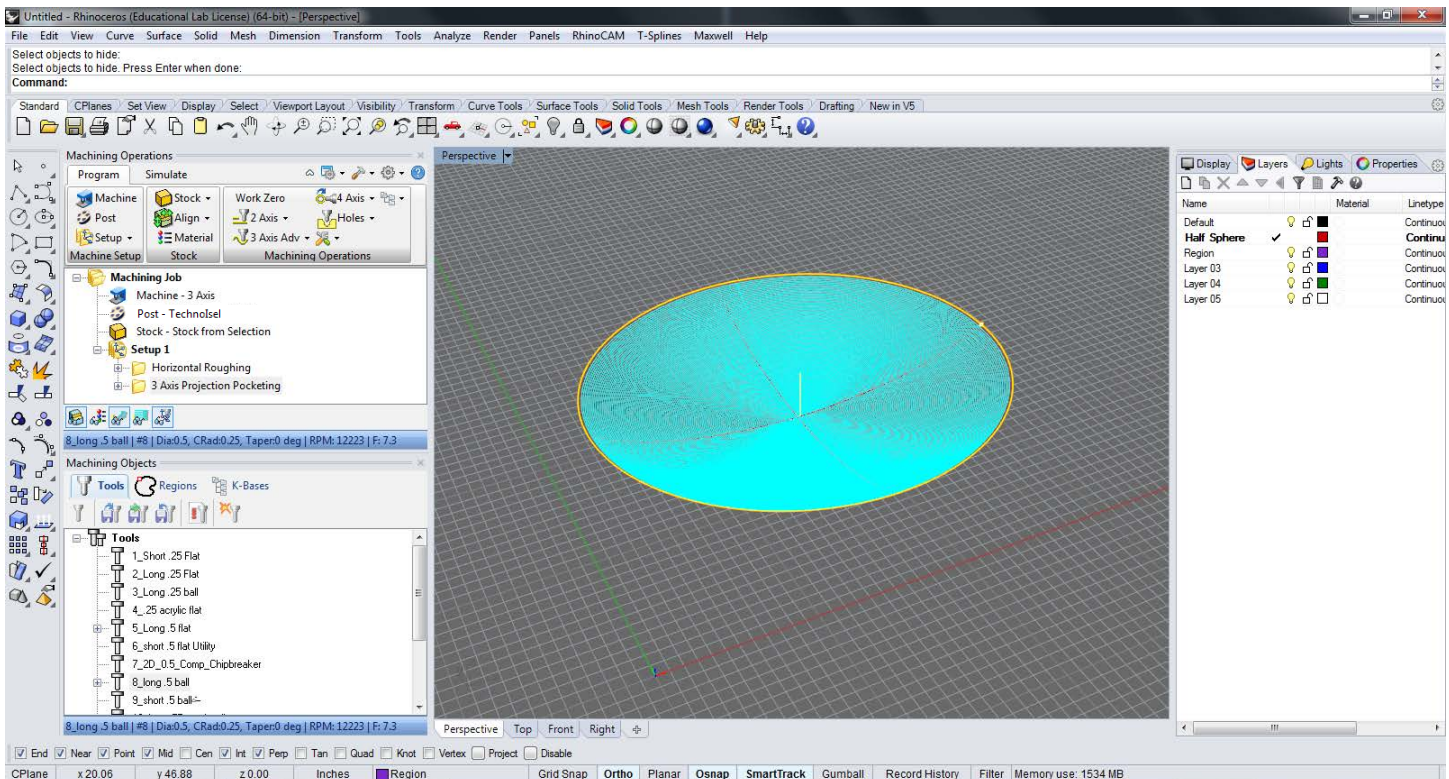


Under 'Tool' select a ball end mill to create a smoother finishing pass. In the example the 1/2" ball endmill is selected.

# Rhino\_CAM 3 Axis Machining Operations\_Projection\_Pocketing

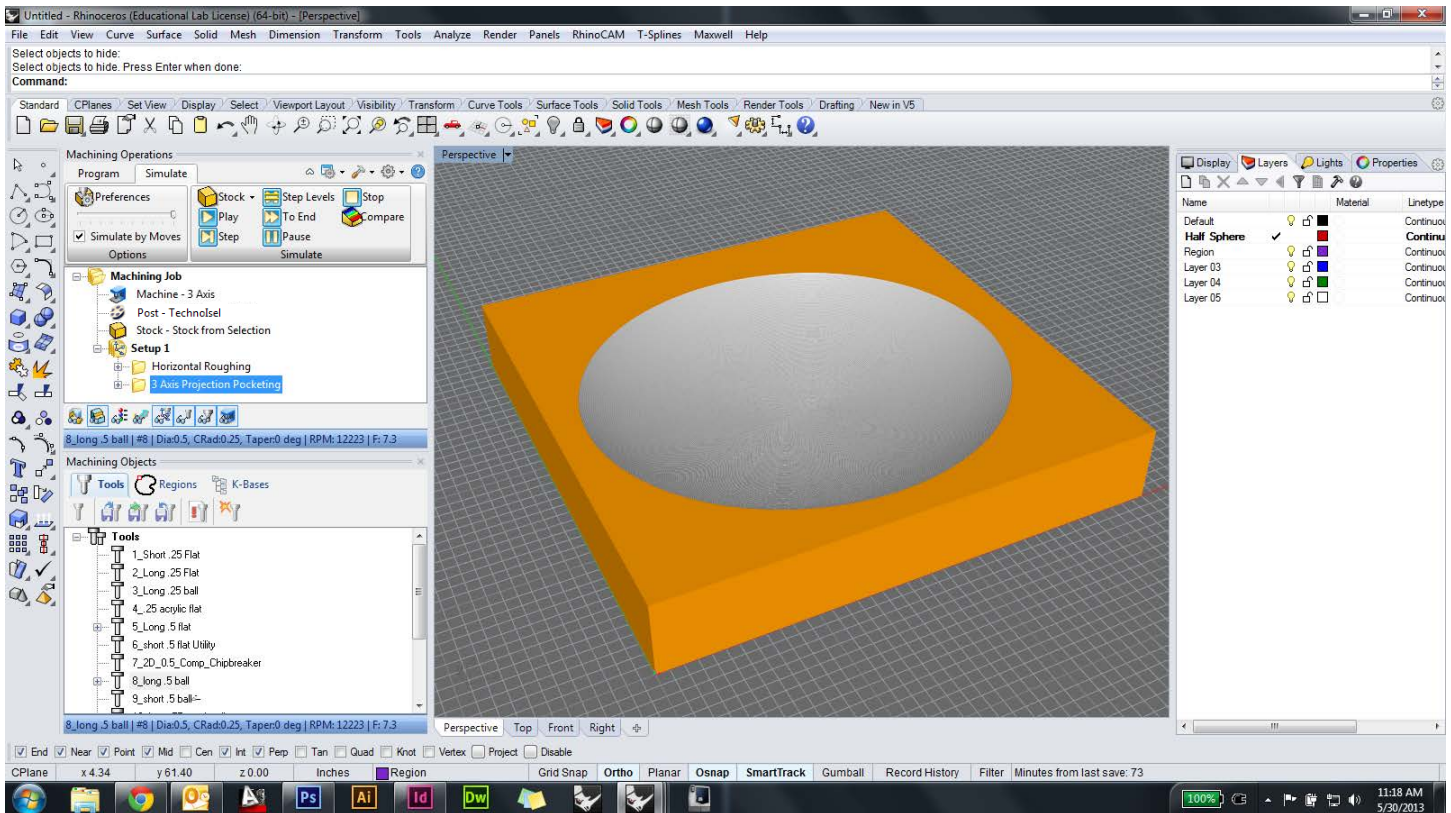


Under Cutting Parameters set the cut direction to mixed, the cut control is set to 2D Offset, linear can be selected depending on the resultant tooling paths desired. The stepover is set to 25% for a smoother final pass.



Once all the parameters have been set, press 'Generate' to create the toolpaths. Parameters can be readjusted if tighter/looser milling paths are desired.

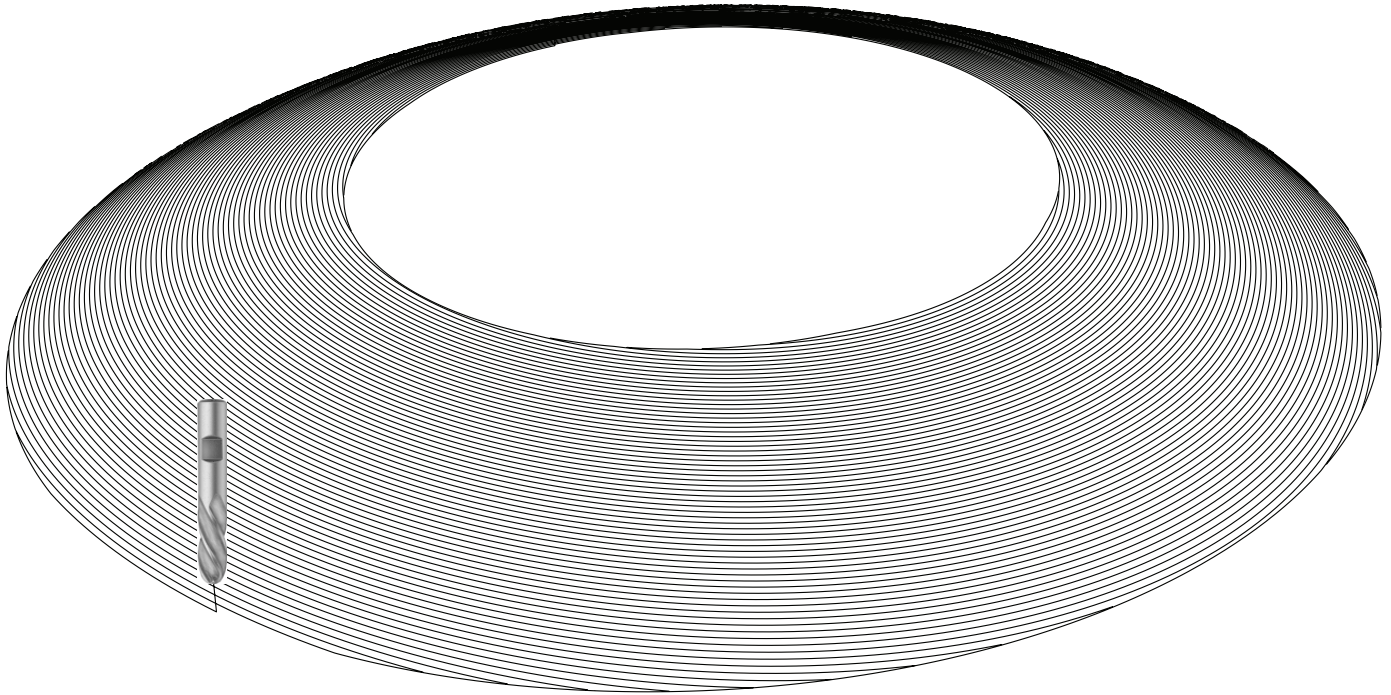
# Rhino\_CAM 3 Axis Machining Operations\_Projection\_Pocketing



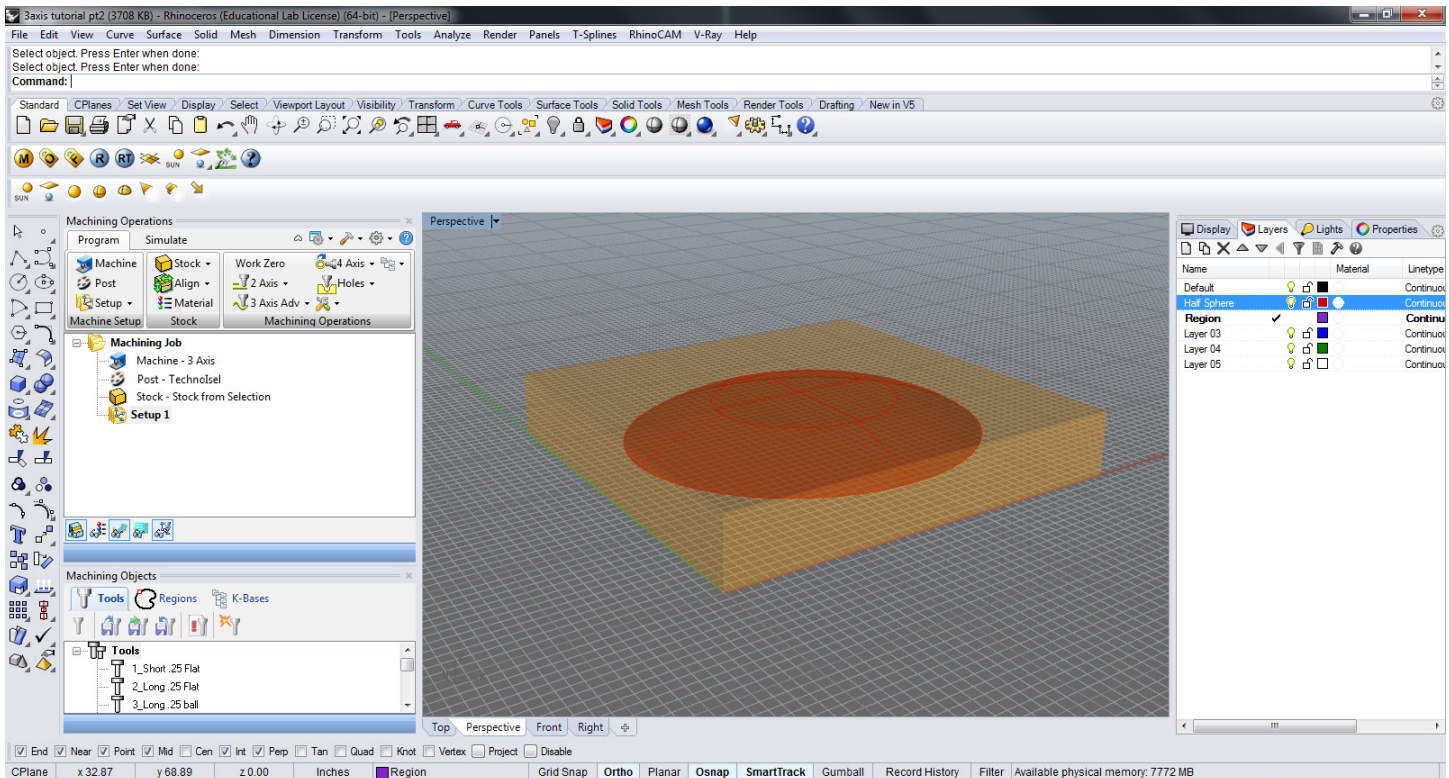
Under the Simulate tab run the **simulation** to see the final surface with the Projection Pocketing. Check to make sure there are no collisions or errors in the milling paths.

## Rhino\_CAM 3 Axis Machining Operations\_ Spiral\_Machining

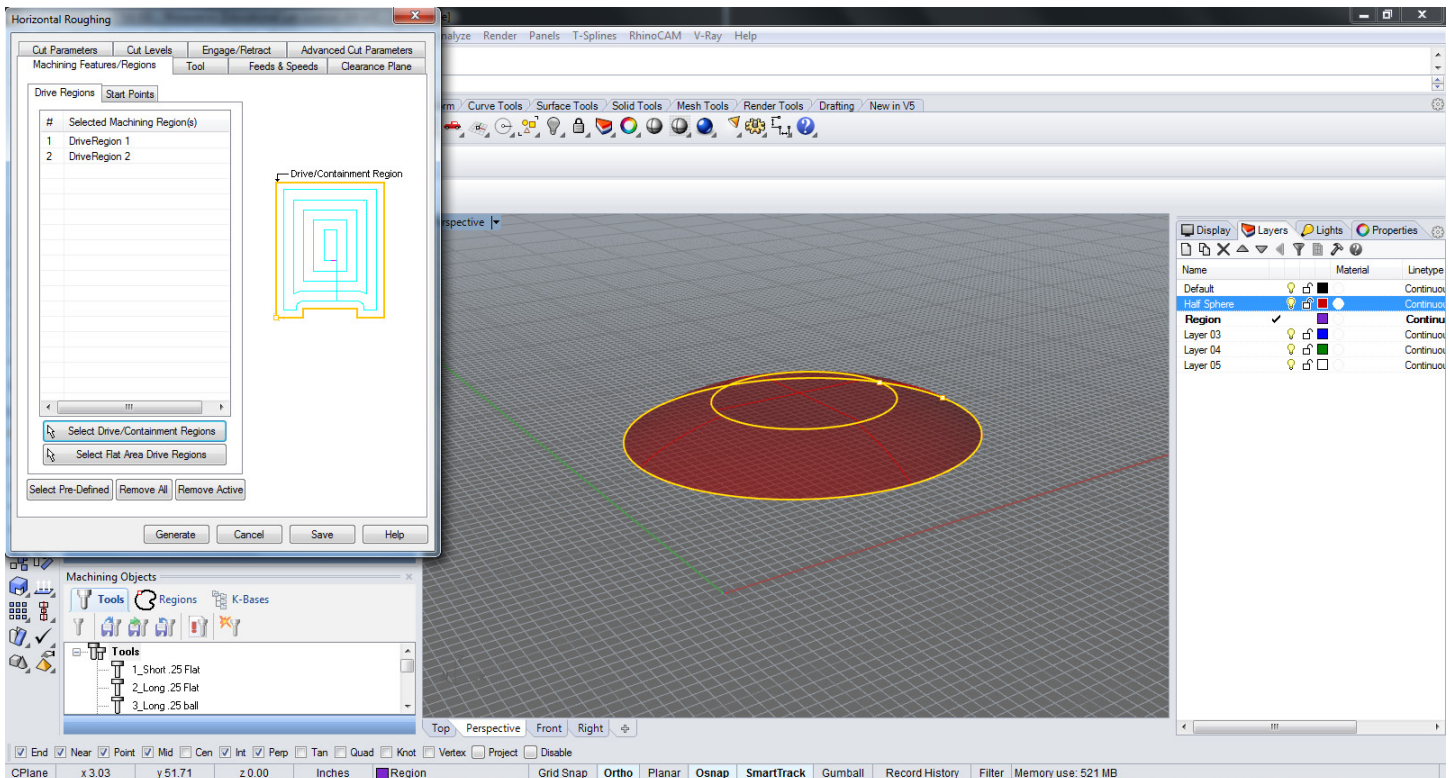
Spiral machining is a method of generating a spiral toolpath. It can be used efficiently for circular regions. Single/multiple regions must be selected and activated to generate the spiral toolpath. The toolpath will be generated only within the activated regions.



# Rhino\_CAM 3 Axis Machining Operations\_ Spiral\_Machining



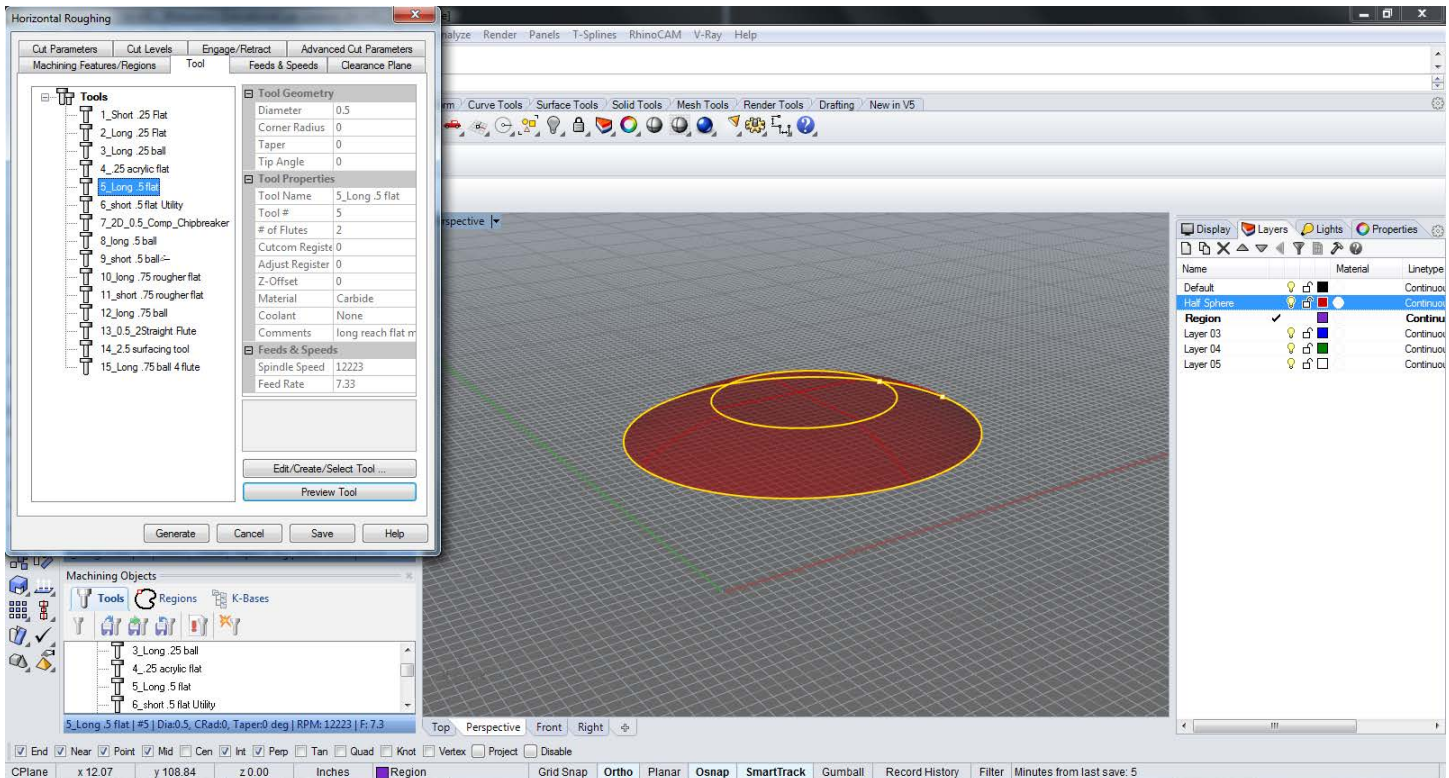
For Spiral Machining the same process will be repeated from the projection pocketing example. Begin with placing the polyobject in the stock and then use Horizontal Roughing to remove the material before the final milling pass with Spiral Machining.



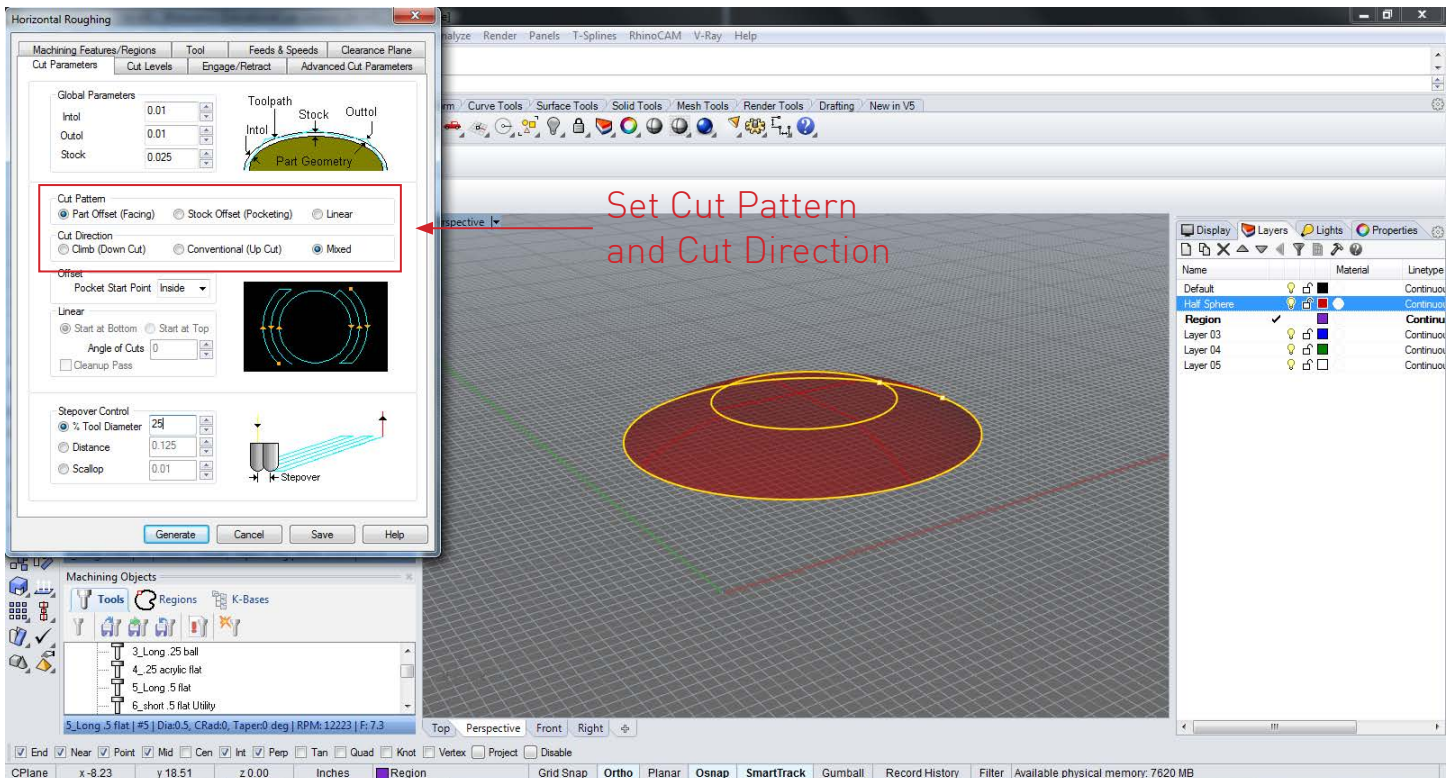
For the Horizontal Roughing create to drive regions to contain the Horizontal Roughing pass, the pass will run between this two defined regions.



# Rhino\_CAM 3 Axis Machining Operations\_ Spiral\_Machining

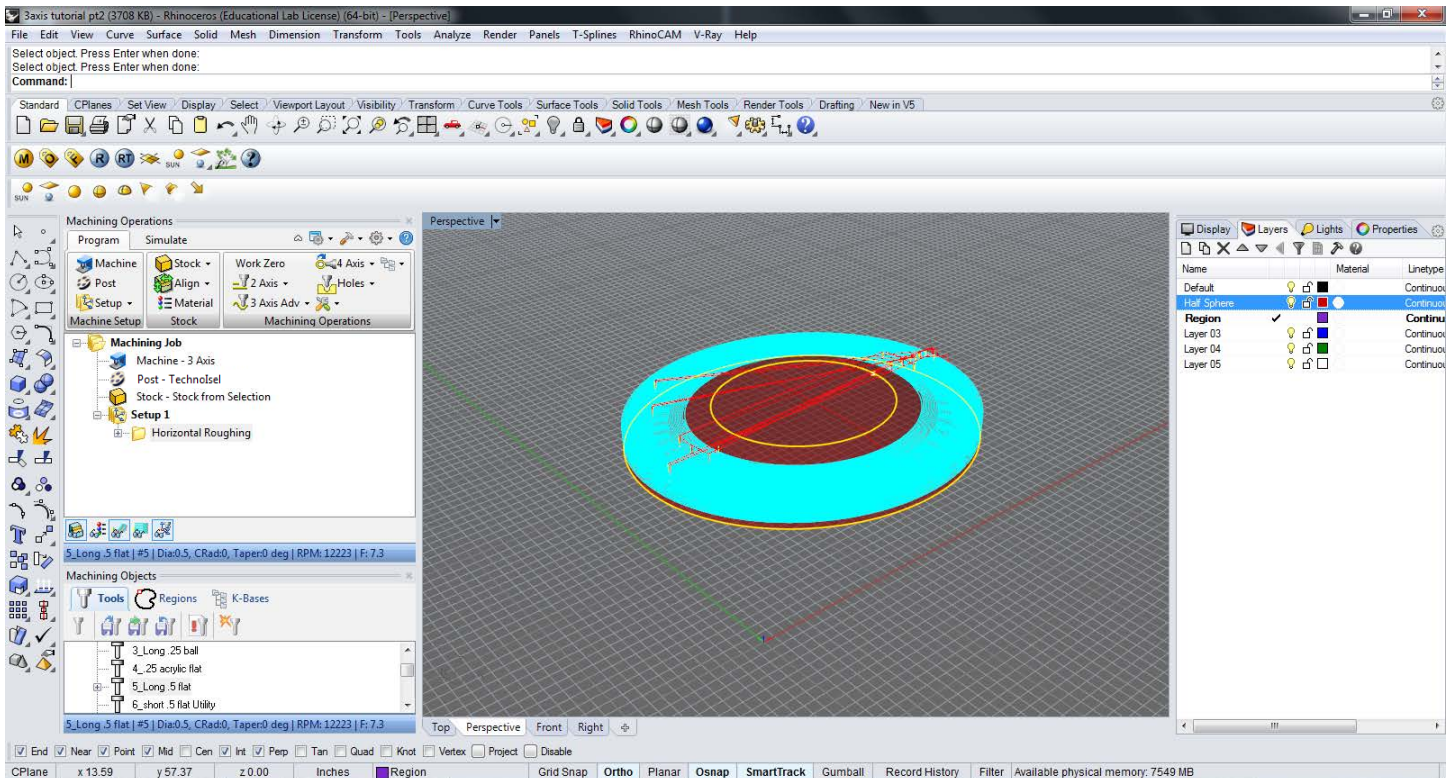


Under Tool for the Roughing pass select the the .5" long Flat endmill.

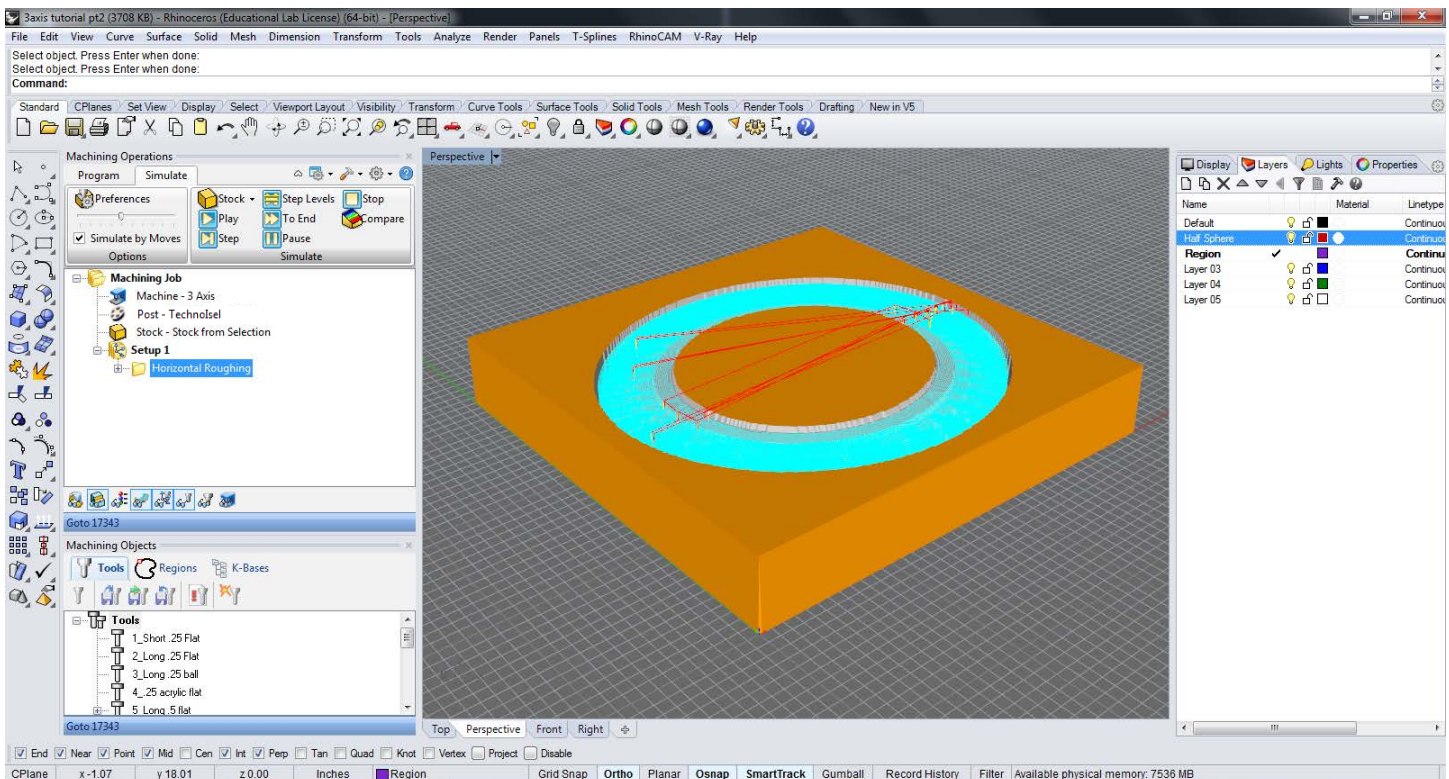


Under the Cut Parameters set the Cut Pattern to Part Offset and the Cut Direction to mixed. The tool diameter is set to 25% in this example, for less dense material it can be set higher to speed up the Roughing Pass.

# Rhino\_CAM 3 Axis Machining Operations\_ Spiral\_Machining

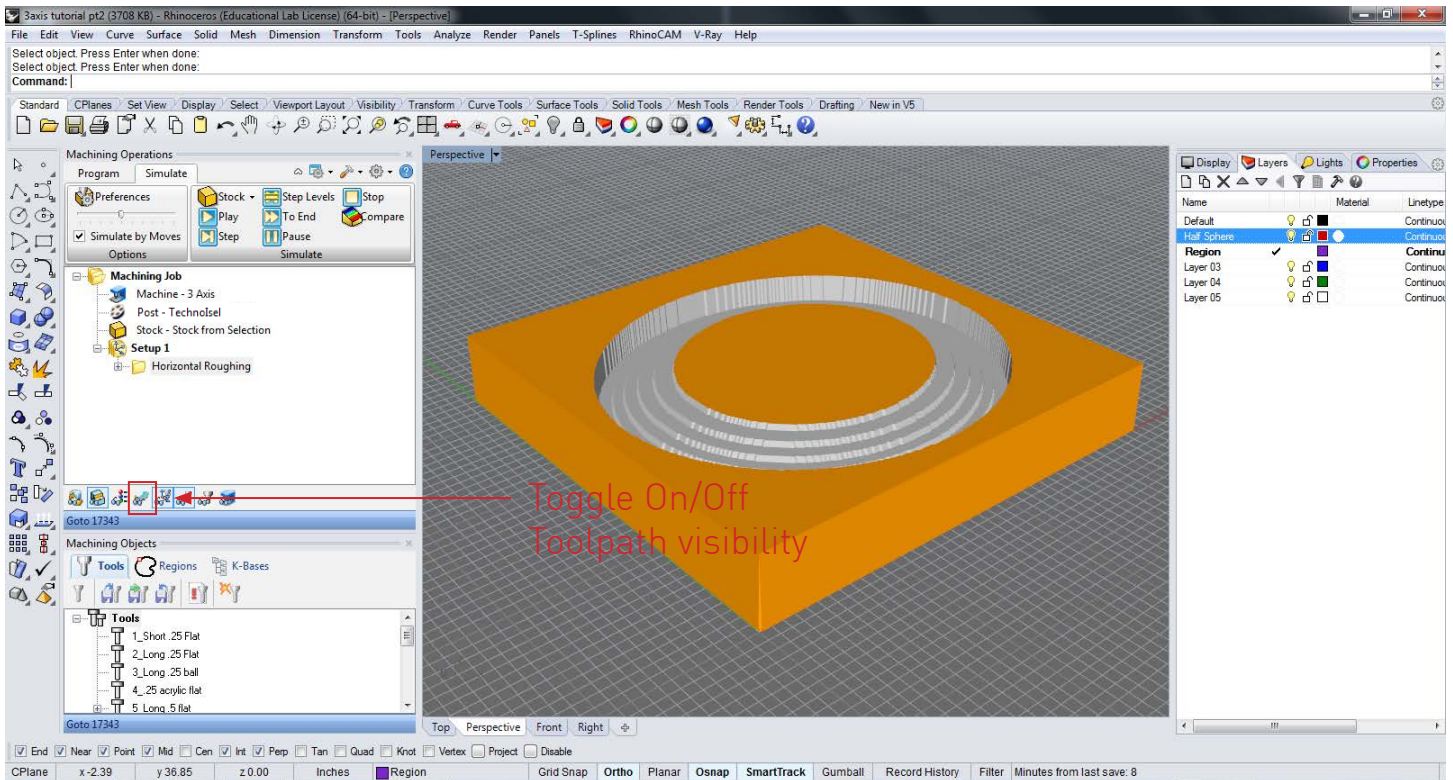


Once all the parameters have been set up press generate to create the milling paths.



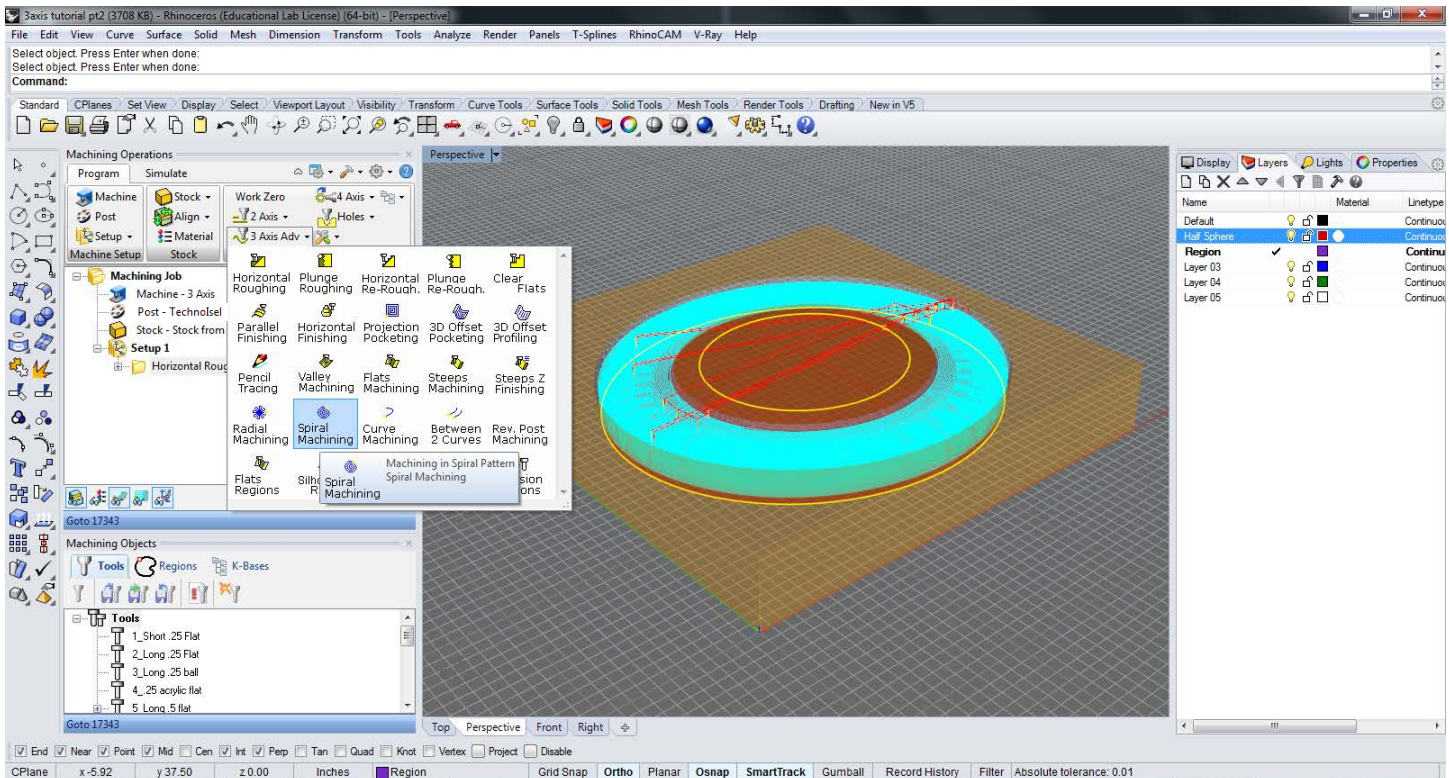
Under the Simulate tab run the simulation to check for any collisions with the head and make sure the milling paths are the desired ones.

# Rhino\_CAM 3 Axis Machining Operations\_ Spiral\_Machining

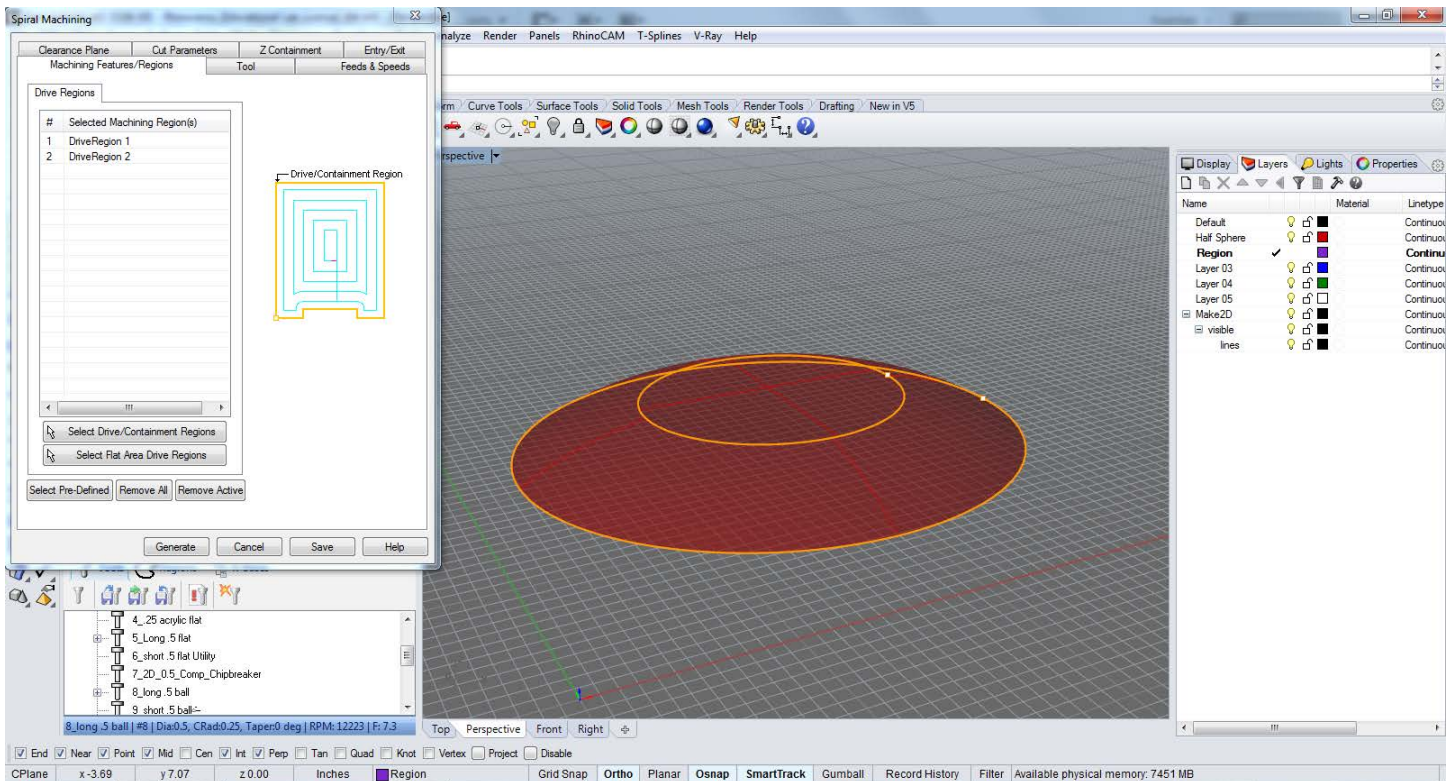


Toggle off the tooling paths to view the Horizontal Roughing path surface.

# Rhino\_CAM 3 Axis Machining Operations\_ Spiral\_Machining

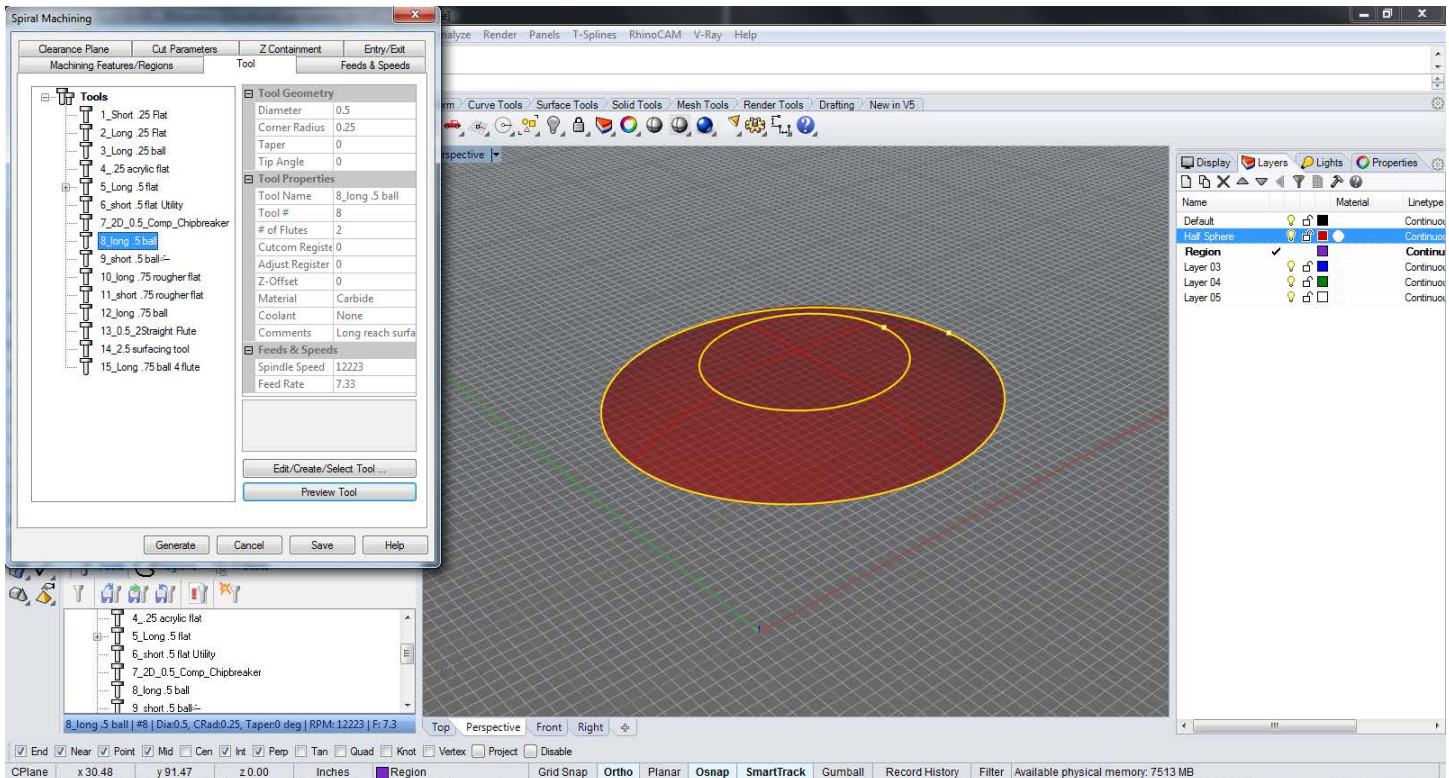


Under the 3 Axis Adv select the Spiral Machining tab.

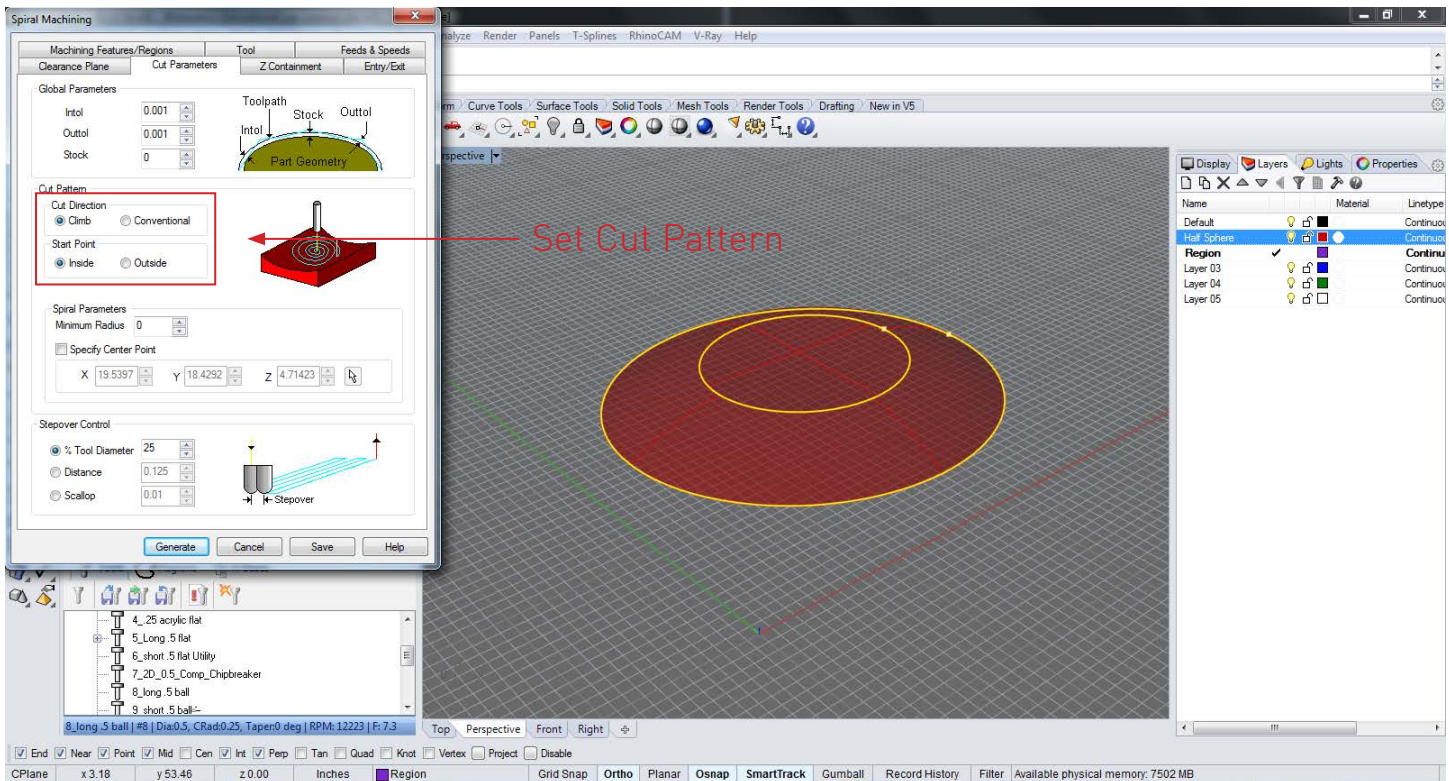


Under Machining Features/Regions select the same two drive regions from the Horizontal Roughing Pass.

# Rhino\_CAM 3 Axis Machining Operations\_ Spiral\_Machining



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%.