
# CERTIFICATION TUTORIAL

## part 2: creating and cutting a file

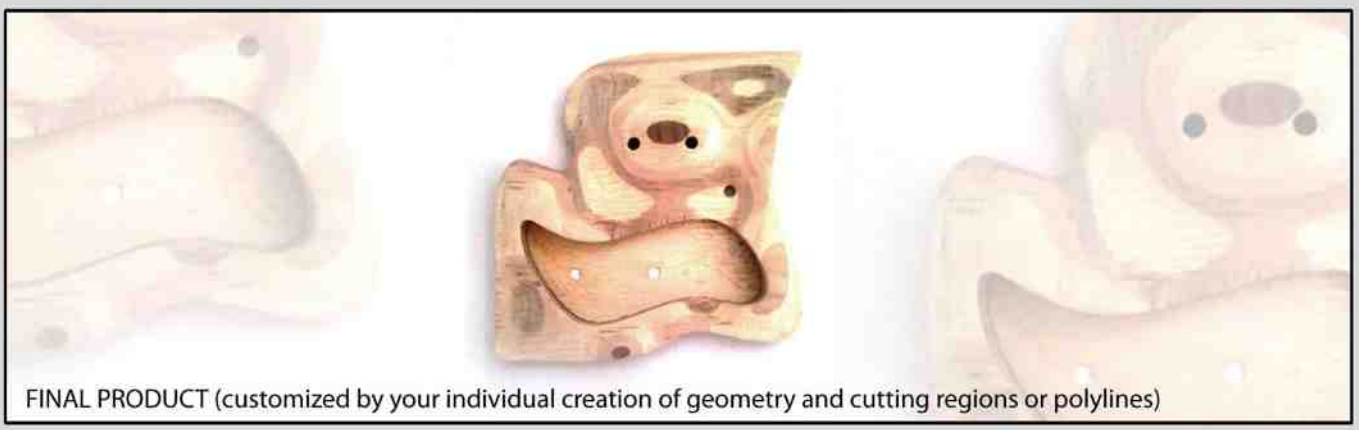
# STEP 1 SKILLS / LEARNING OBJECTIVES

This is part 2 of the certification process to use CNC machines in the architectural production facilities. You must have attended part 1 of certification, the group tutorial session. To schedule cutting times for Part 2 and for general contact please email **cnc315info@gmail.com** (if possible please send your Rhino file, NC files, shop documentation file as well as your cutting checklist in a zipped folder). Part 2 will familiarize you with file setup, basic operation of the routers and five basic tool path techniques:

1. HORIZONTAL ROUGHING
2. PARALLEL FINISHING
3. DRILLING OPERATIONS
4. POCKETING OPERATIONS
5. PROFILING OPERATIONS

**TUTORIAL STEPS / LEARNING OBJECTIVES:**

- RHINO MODELING
- RHINOCAM SETUP
- CREATE SURFACE ROUGH PATH
- CREATE SURFACE FINISH PATH
- CREATE DRILL PATH
- CREATE POCKET PATH
- CREATE CONTOUR PATH
- EXPORT / POST FILES



FINAL PRODUCT (customized by your individual creation of geometry and cutting regions or polylines)

**REQUIRED MATERIALS AND BITS:**

- **8 x 8 x 3/4" MDF square** (NOT PROVIDED)
- **1/2" ball nose endmill** (surface rough) (PROVIDED only for tutorial. You must purchase your own bits in the future)
- **1/4" flat endmill** (finish, drill, pocket, contour) (PROVIDED only for tutorial. You must purchase your own bits in the future)

MATERIAL: **NOT PROVIDED**

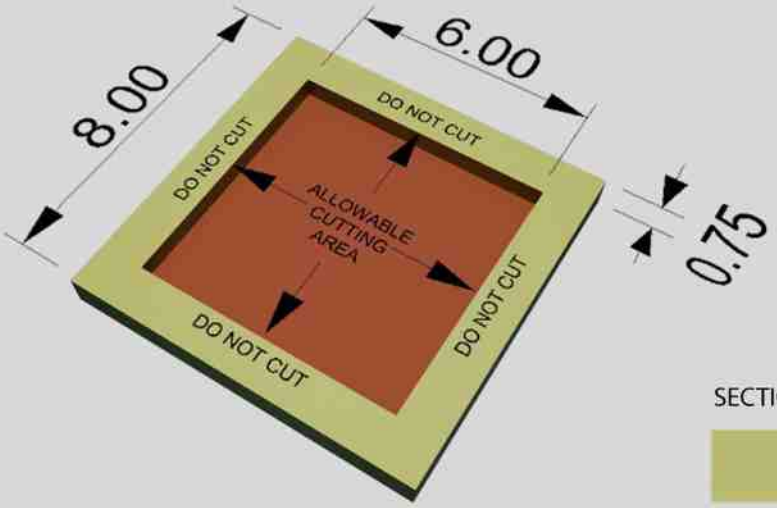
BITS: PROVIDED **(for the tutorial only)**

# STEP 2 ALLOWABLE CUTTING AREA

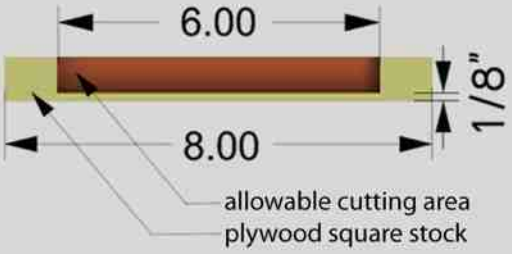
Use the template file attached to this tutorial. First make sure that your Rhino units are set to inches. Model only with in the allowable cutting area. Make sure the bottom left corner of your geometry is located a an X and Y coordinate of 1". Make sure none of your geometry rises above 3/4" in the Z Axis and does not drop below 1/8" in the Z Axis. All lines to be cut and holes to be drilled should be placed at an elevation of 3/4" in the Z axis.

**ALLOWABLE CUTTING AREA=6"x 6"x 5/8"**

PERSPECTIVE VIEW




SECTION VIEW

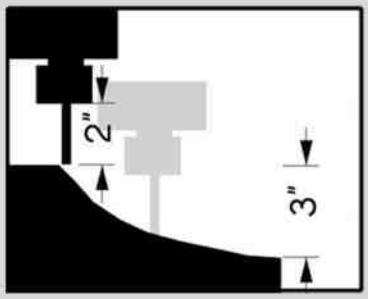



SECTION WITH GEOMETRY

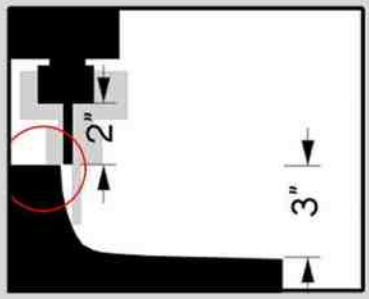
**Things to note:**

1. The actual allowable cutting area for the Techno4396 CNC machine is 48"x 96" x 6". These are maximum dimensions but the machine is only capable of cutting certain geometry. Too steep of a cut and too short of a bit can result in conflicts. The example shows how a cut depth of three inches can be both good and bad depending on the specific geometry

GOOD CUT

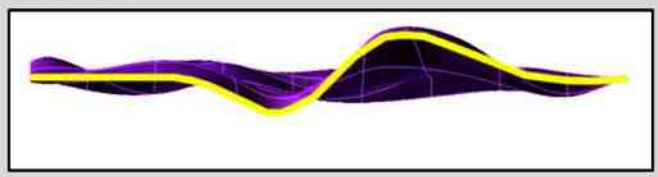



BAD CUT

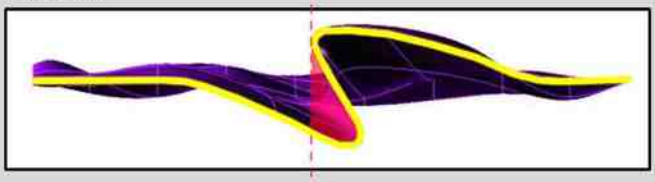



2. The machine cannot do undercuts. See example below.

GOOD CUT



BAD CUT



# STEP 3 MODELING AND ALIGNING GEOMETRY

Create any continuous 6" x 6" piece of geometry, making sure it is no more than 5/8" in depth. Place it at an X,Y value of 1". Place it at a Z value that insures your geometry does not raise above 3/4" and does not drop below 1/8" in the Z dimension. (see figure 1)

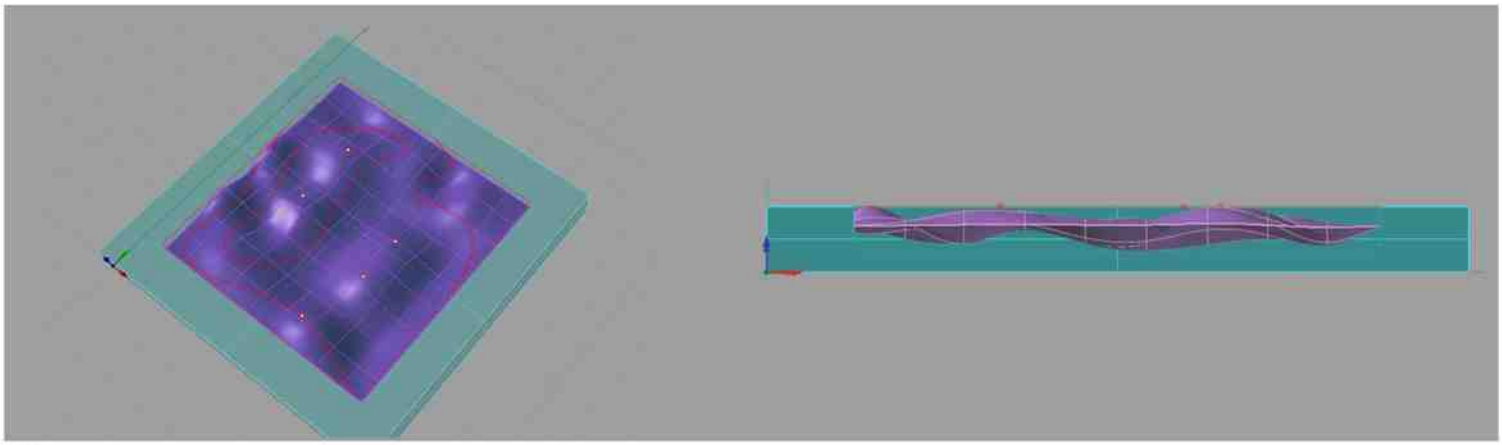


figure 1

Create a flat polyline that that will define the outer geometry to be cut. (continuous poly-line). Create an interior smaller flat polyline that will define the shape of the pocketing operation. Make sure these lines are placed at a Z value that equals the height of your material or in this case 3/4". See figure 2.

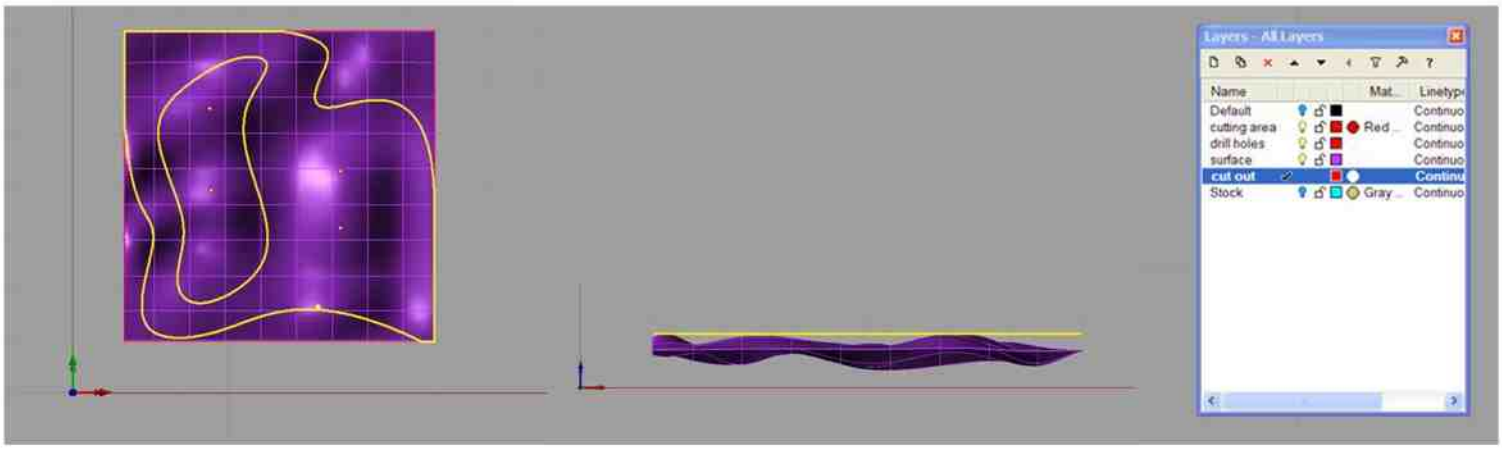



figure 2

Create 5 points that will become drill holes. Make sure all of this geometry is places at a Z value that equals the height of your material or in this case 3/4". See figure 3.

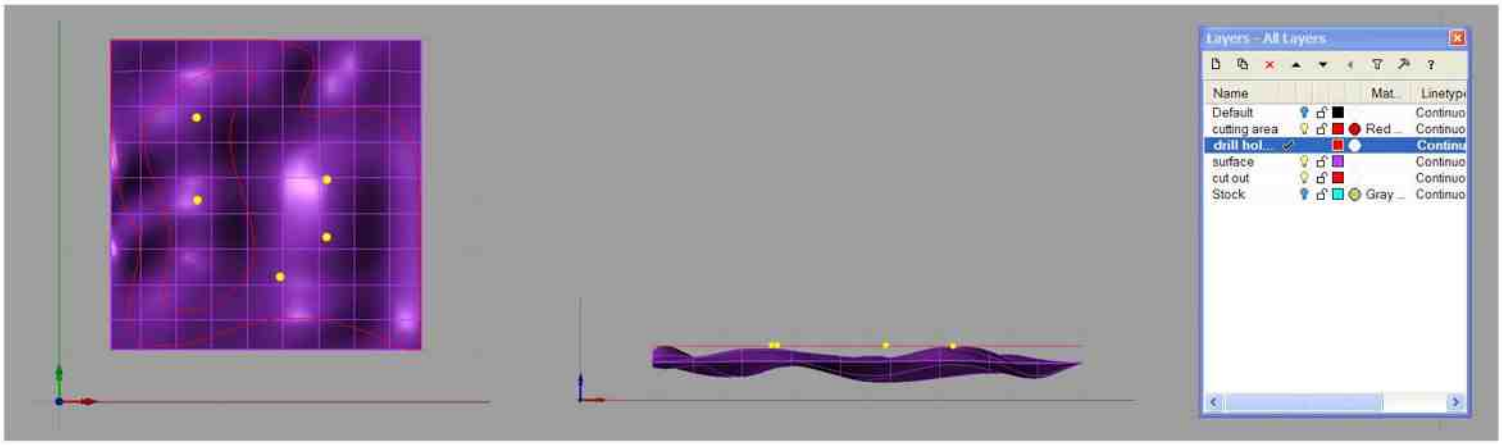
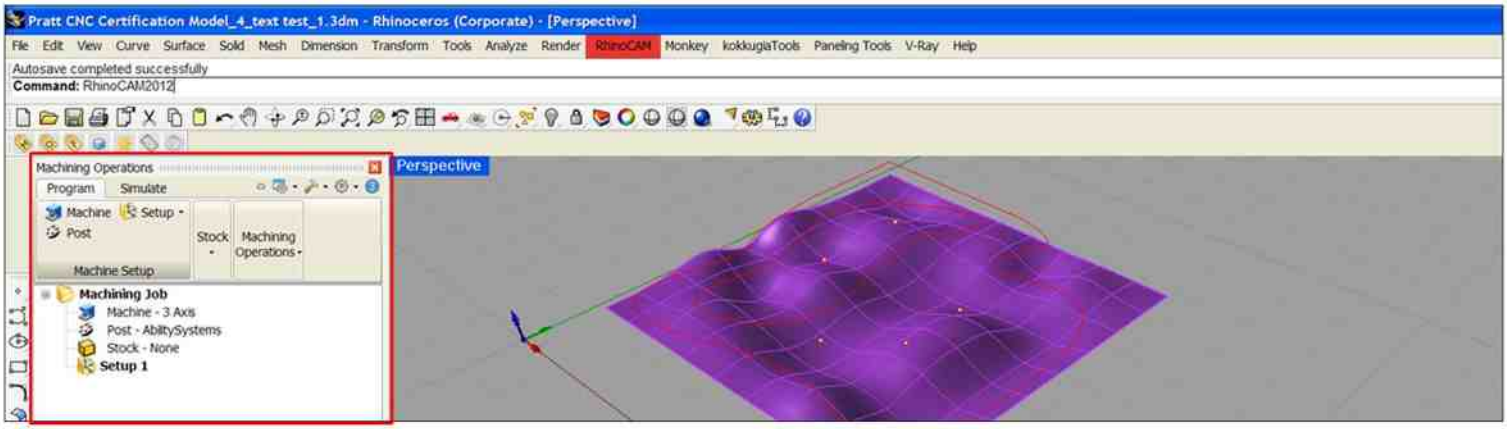


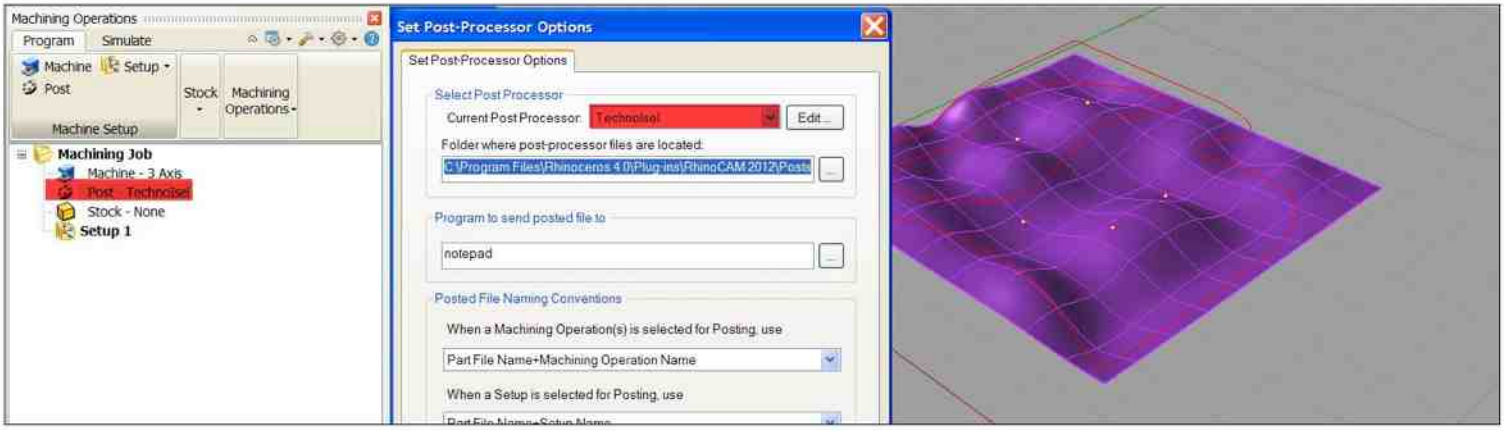

figure 3

# STEP 4 RHINOCAM SETUP AND OPERATIONS

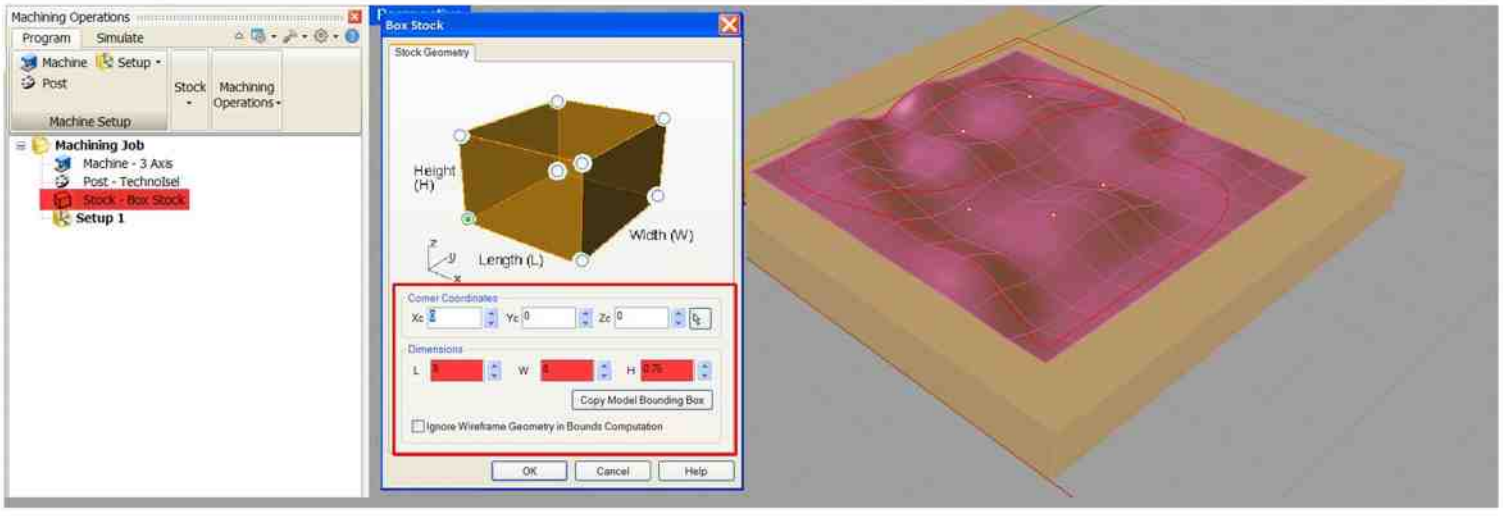
A) In the command line, type "RhinoCAM2012" to open the RhinoCAM interface. Note that all numbers will be in inches.
B) The "Machining Operations Browser" window should appear at the left. If not, click on the RhinoCAM menu bar option at the top of the screen, then select "Machining Operations Browser".




C) Verify that "Machine" says "3-Axis". If needed, double-click on the option to change machine settings.
D) Verify that "Post" says "TechnoIsel". If needed, double-click and selection "TechnoIsel" from the "Current Post Processor" drop-down. This allows proper communication between the geometry, the computer and the router.

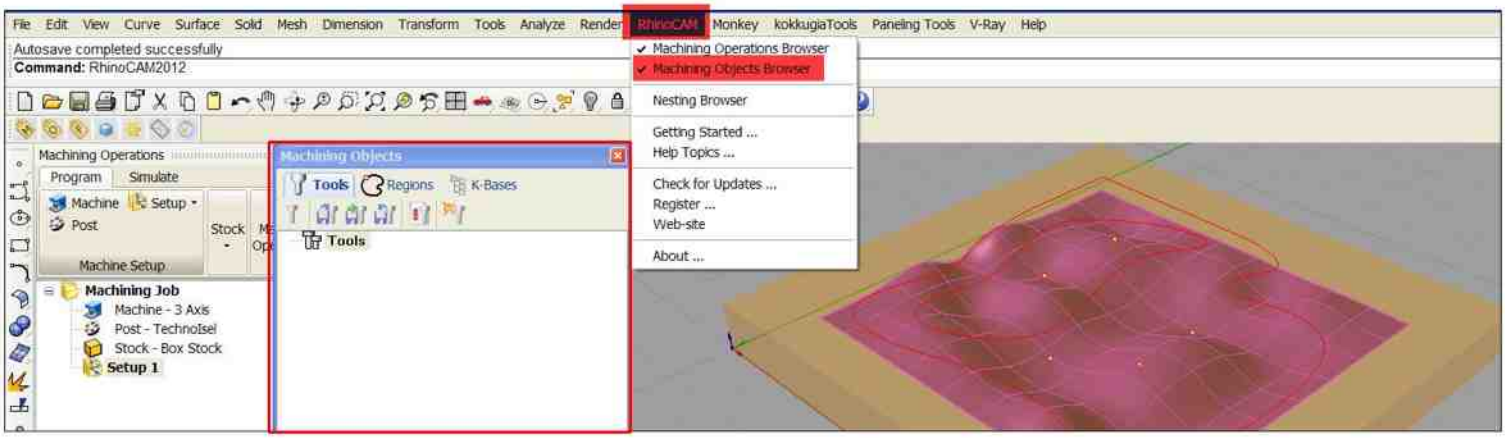



E) Double-click on "Stock" to set stock geometry. This is a simulation of your material, appearing in Rhino as a translucent yellow box. Make sure the upper left corner is selected in the diagram. Set "Corner Coordinates" to the origin (0, 0, 0). Set the dimensions to match your stock material, in this case L = 8, W = 8, H = 0.75. You should see an 8 x 8 x 0.75" box geometry in Rhino, aligned to the X/Y origin and above the Z plane. Note that you can change transparency/color under the "Preferences --> Simulation Preferences" button. **(Stock geometry, cut paths and points, as well as cut depths should NEVER BE BELOW THE Z=O POSITION)**

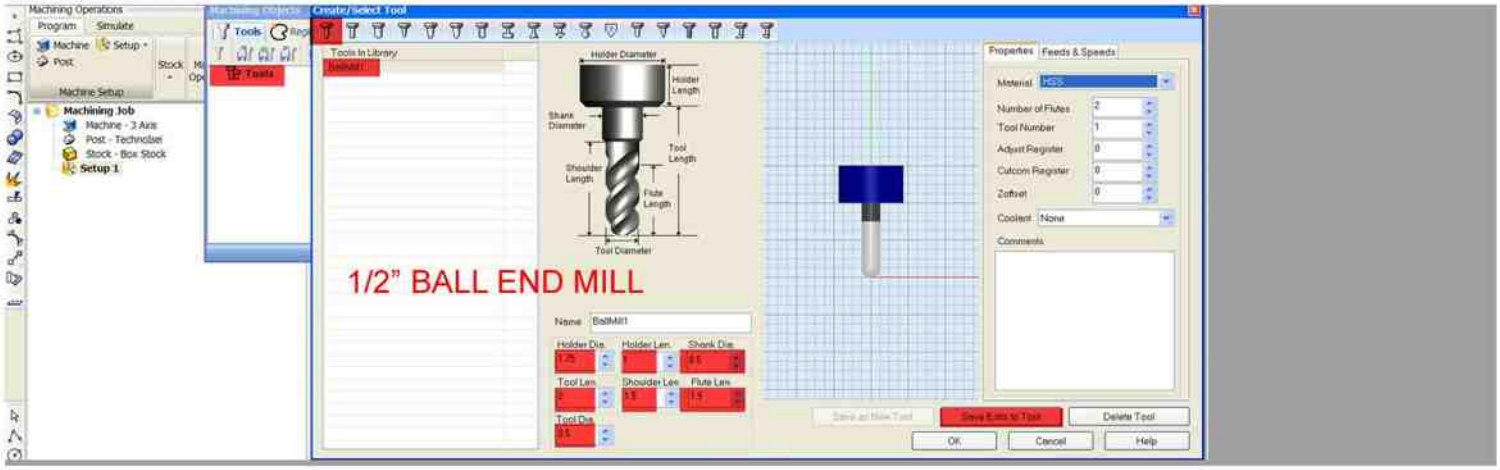


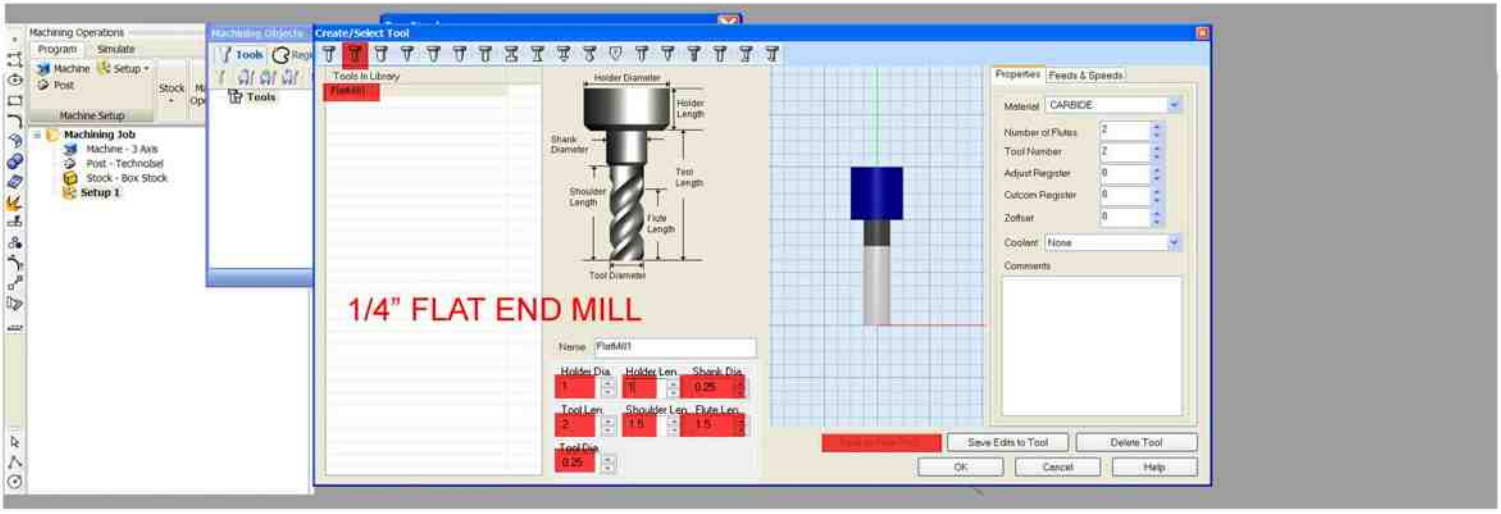
# STEP 5 RHINOCAM: CREATING TOOLS

A) Click on the RhinoCAM menu bar option at the top of the screen, then select "Machining Objects Browser". The new window should appear next to the Machining Operations Browser.



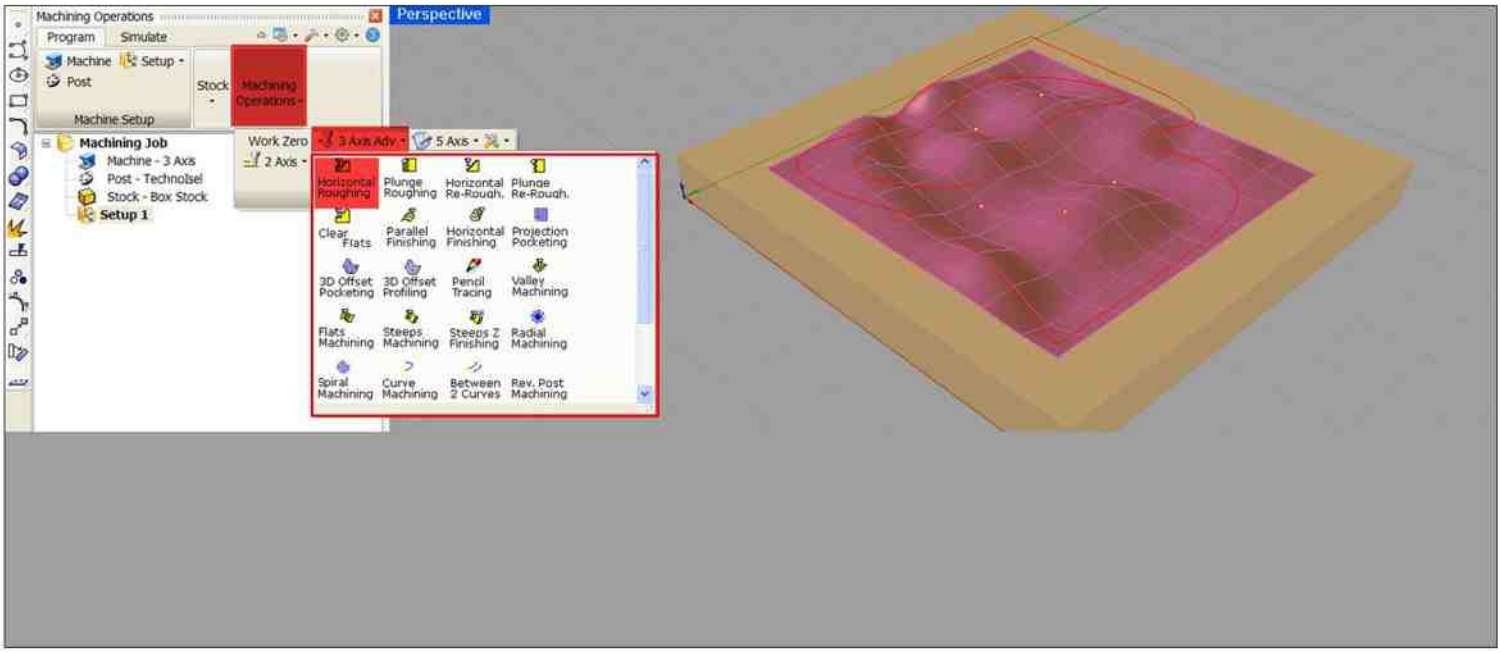

B) Click on tool icon in the upper left of the Cutting Tools Browser. At the top of the new tool options window, select the type of endmill (in this case, the "Ball Mill" icon). Set ONLY the options for "Tool Length" and "Diameter". Set "Diameter" to 0.5". Note that the other options (especially "Tool Length" and "Flute Length") are useful for a more accurate simulation of the mill, but are not necessary to change for this tutorial. O **(Note that the tool holder diameter is always 1.75 " and the other dimensions are based on the bit you are using. These dimensions will be CRITICAL to future cuts. Fill them in appropriately for future cuts)** Name the tool "0.5 ball" to indicate a 1/2" diameter round/ball end mill. Finish by clicking "Save as New Tool". Repeat these steps for the 1/4" flat end mill, being sure to set Diameter = 0.25 and rename it "0.25 flat."



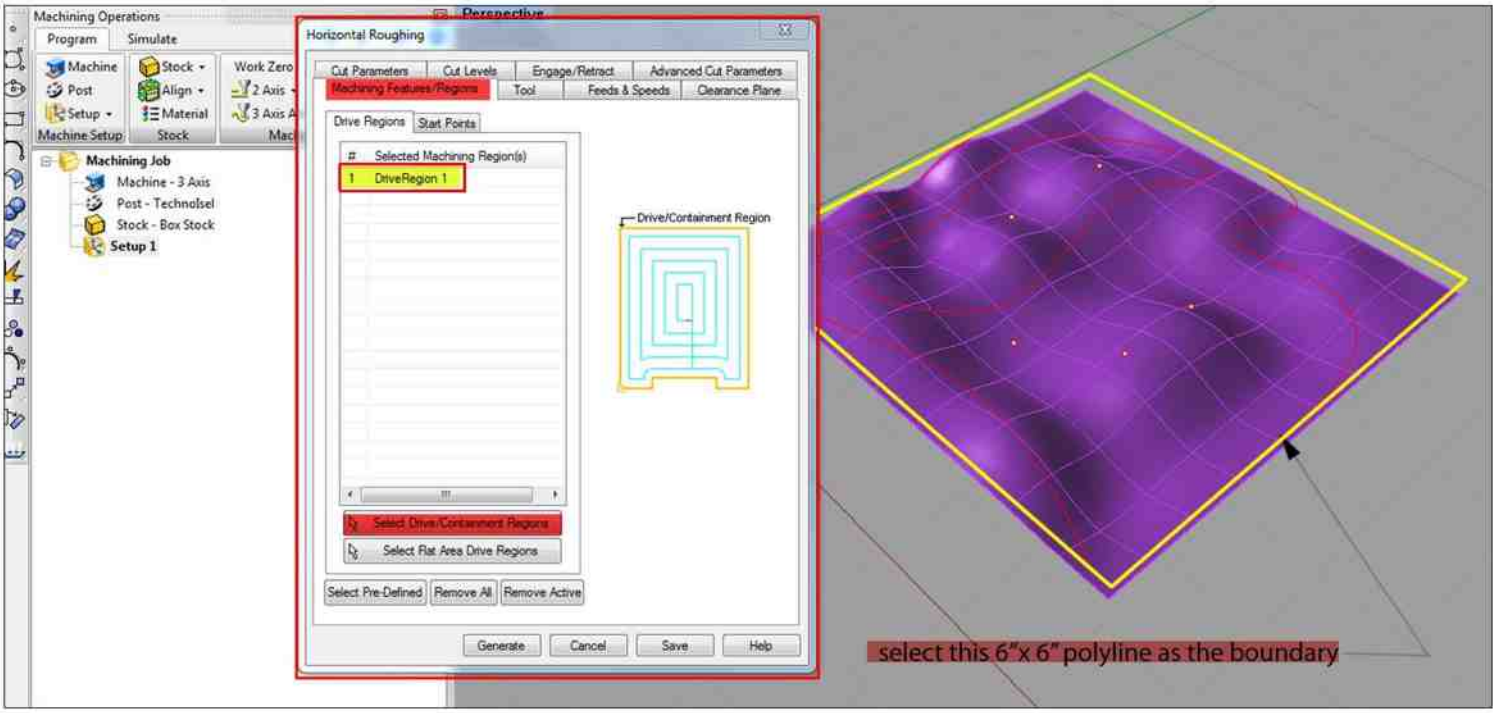



# STEP 6 RHINOCAM: CREATING HORIZONTAL ROUGHING PATH

A) Click on the "Create" tab in the Machining Operations Browser (Mops). Click on the button for 3-axis operations and select "Horizontal Roughing" from the drop down menu. A window will pop up that has all of the cut parameters and settings.

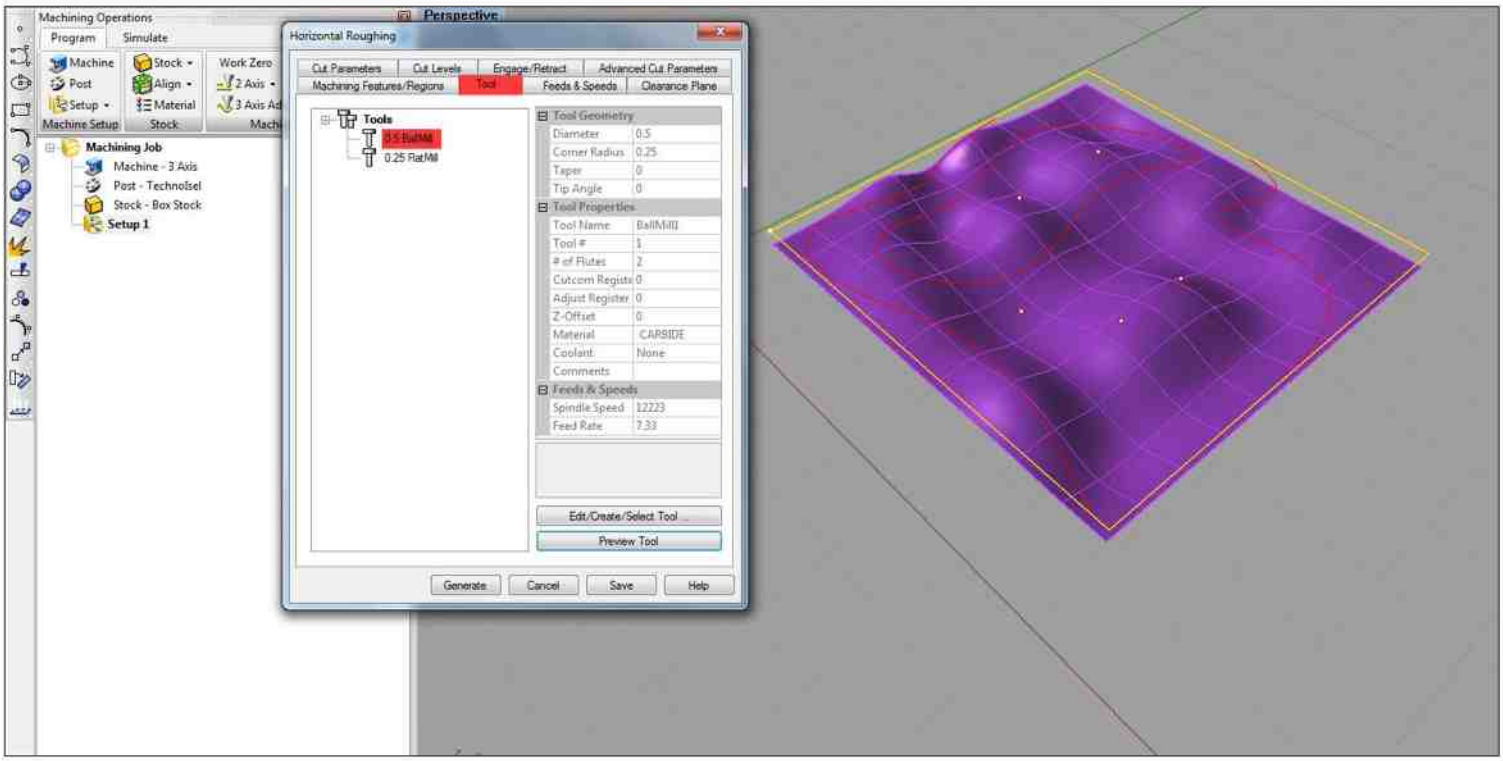



B) This pop up window has several tabs that control the parameters of the cut. We will start with the Machining Feature/Regions tab which allows you to select a bounding region for milling operations. In this example, we only want the 6x6" square to be milled out. Click on "Select Curves as Regions". The window will minimize and you'll be asked to select closed polylines from the Rhino model. In this case, select the 6x6" square on the layer "cutting area" to limit milling to the 6x6" square region. Press enter when done selecting. You should notice the selected polyline that defines the machining area is now yellow and that there is now one region (Drive Region 1) on the list of selected Machining Regions.

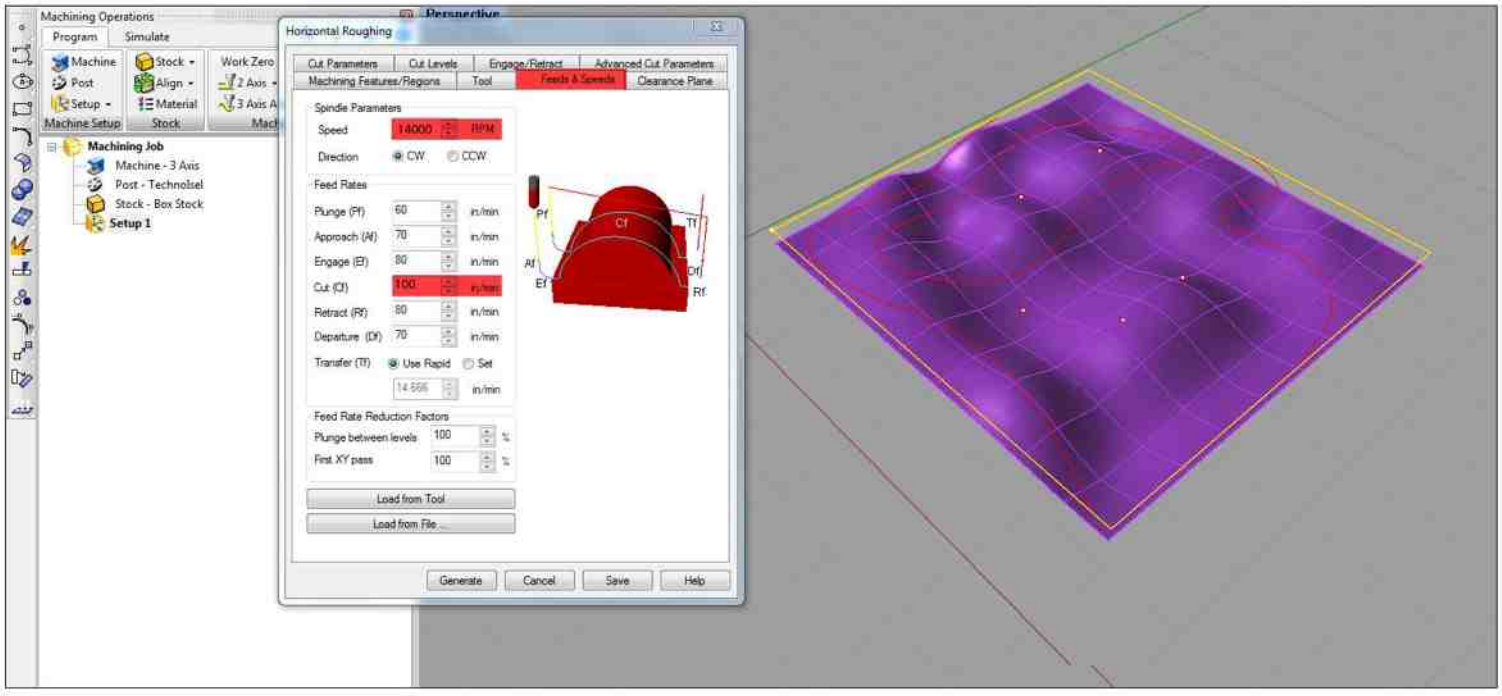


# STEP 6 RHINOCAM: CREATING HORIZONTAL ROUGHING PATH

C) Click on the "Tool" tab. Select the endmill "0.5 BallMill" that was created earlier

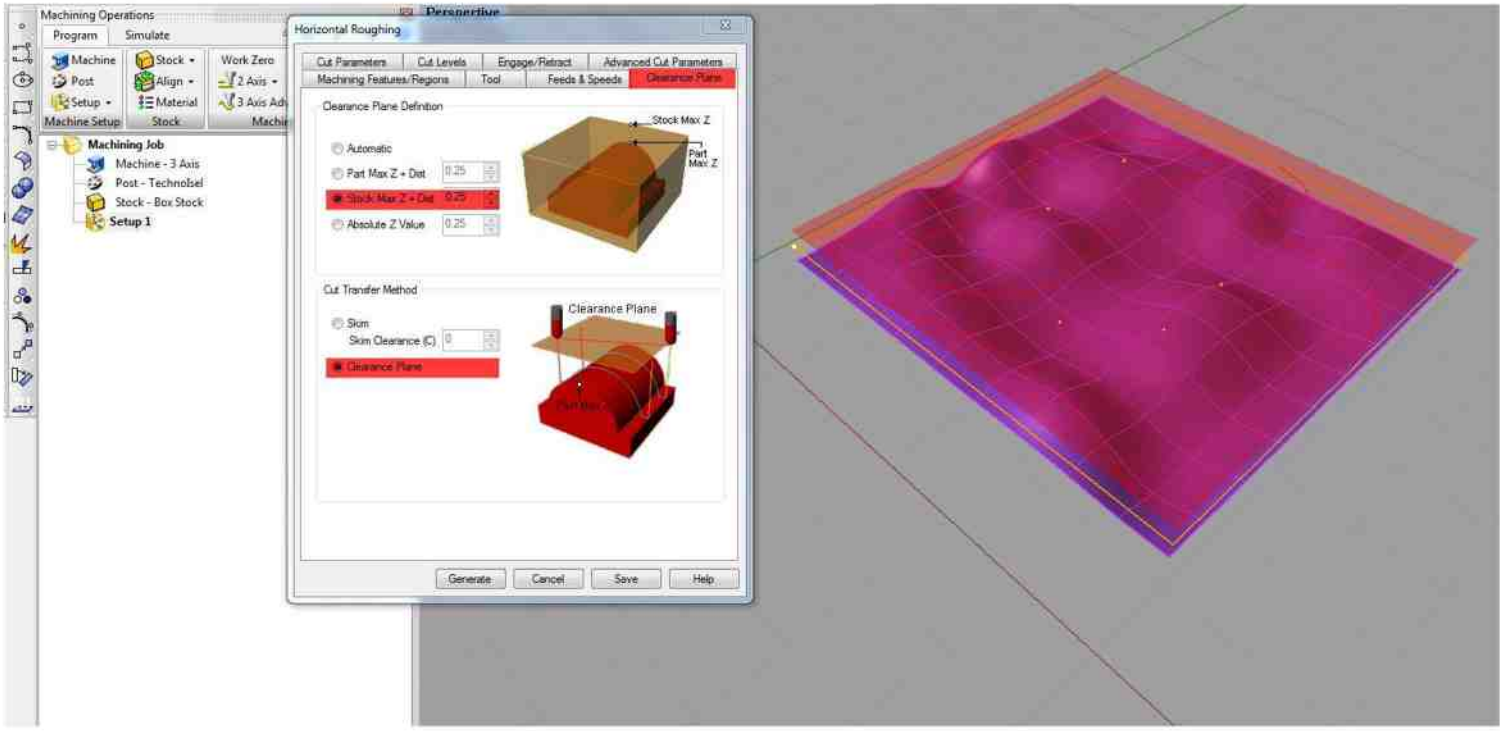



D) Select the "Feeds & Speeds" tab. These **speeds** (the rate at which the router moves or cuts, measured in IMPs or inches per minute) and **feeds** (the rate at which the spindle rotates, measured in RPMs) will change depending on the material being cut, the bit being used and the desired smoothness of the cut. **(THIS IS ONE OF THE MOST CRITICAL CALCULATIONS WHEN MILLING)** For further information on these parameters please consult your instructor or CNC shop literature. For this tutorial the CNC staff has calculated values based on the MDF material selection and the end mill bits supplied. For this tutorial the spindle speed should be set at 14,000 RPM in a clockwise direction and the cutting speed should be set at 100 IPM. See below for descriptions and values of Plunge, Approach, Engage, Retract and Departure Speeds. The "Transfer" field should be set to "Use Rapid" and both of the "Feed Rate Reduction Factors" should be set to 100%.

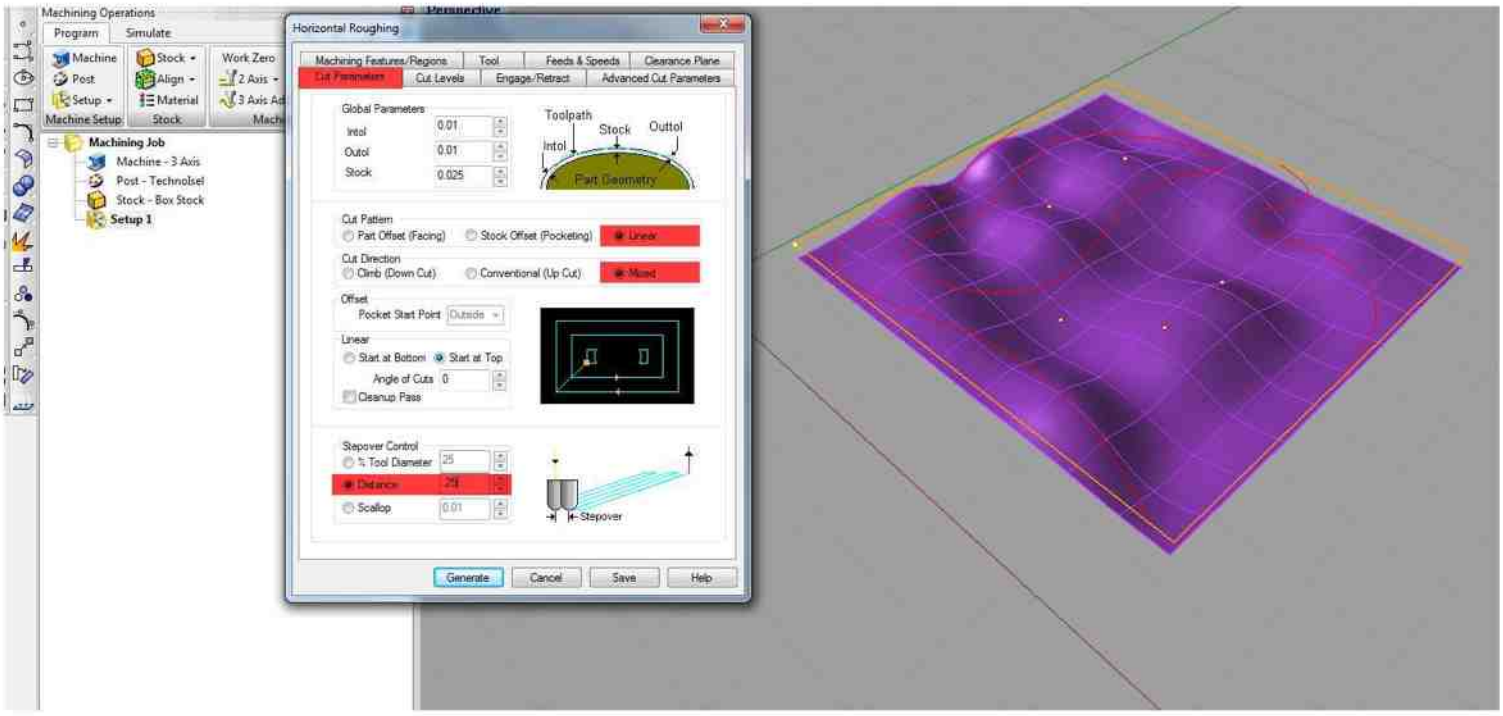


# STEP 6 RHINOCAM: CREATING HORIZONTAL ROUGHING PATH

E) Click on the "Clearance Plane" tab. Set the "Clearance Plane Definition" to Stock Max Z + Dist and select a distance of .25". This means when the machine jogs across the piece it will always travel at a height of .25" above your stock material (which was previously set to .75"). Note: This distance should be adjusted to account for variation in stock height. For example if you have areas of your stock that protrude 0.25" above the Z axis, your clearance plane needs to be increased. Set the "Cut Transfer Method" to Clearance Plane.

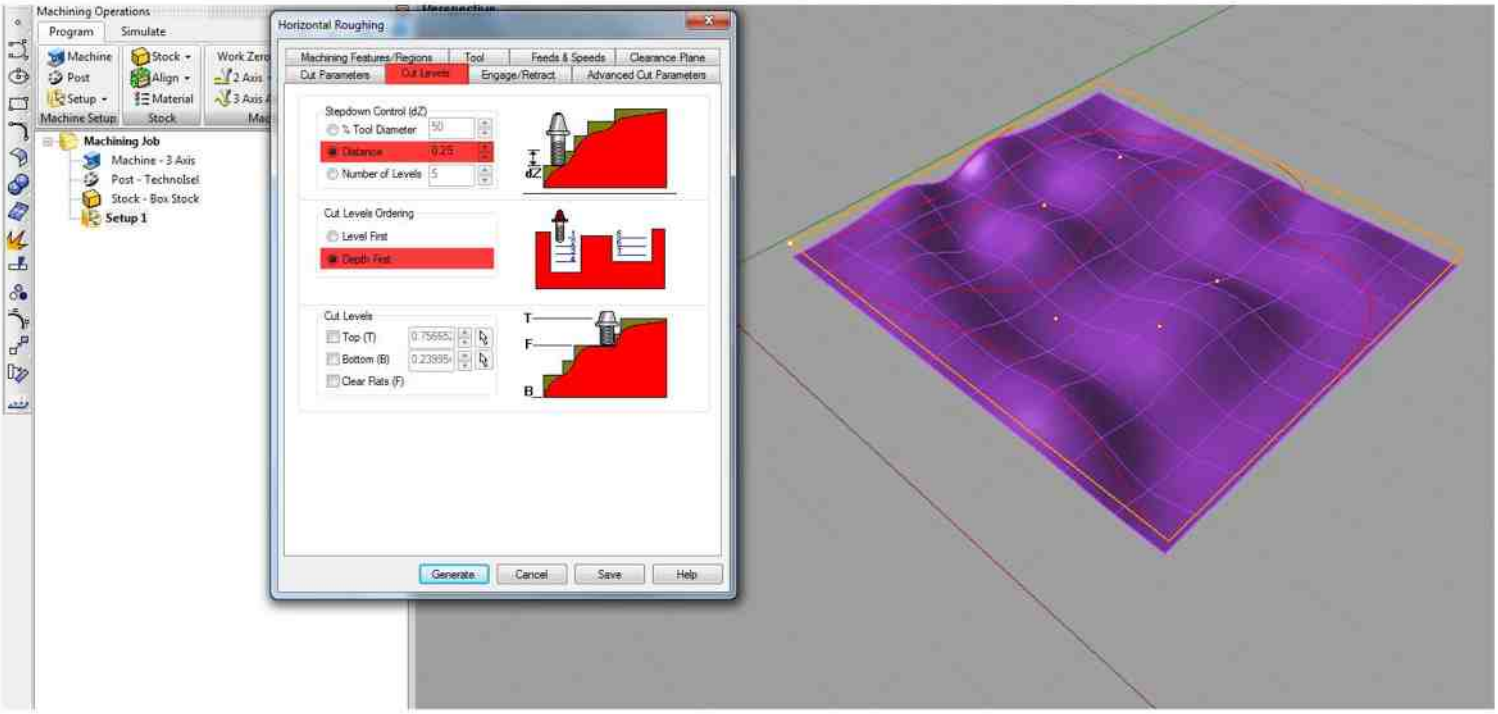



F) Select the "Cut Parameters" tab. Set the "Cut Pattern" to "Linear". Set the "Cut Direction" to "Mixed". Set the "Offset Pocket Start Point" to "Inside". The "Stepover Control" should be set to a distance equal to half the diameter of the bit being used. In this case that value is .25".

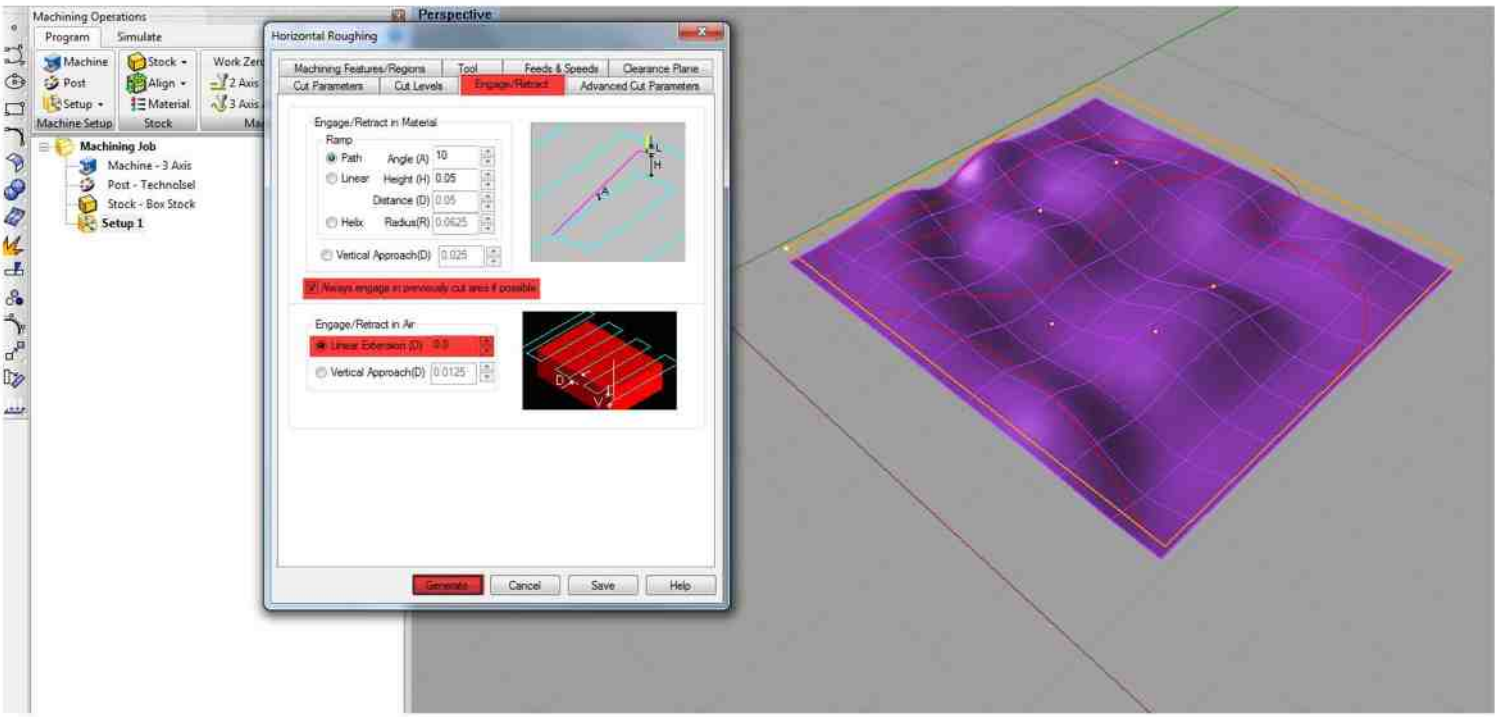


# STEP 6 RHINOCAM: CREATING HORIZONTAL ROUGHING PATH

G) Select the "Cut Levels" tab. Set the "Stepdown Control" to a distance equal to half the diameter of the bit being used. In this case that dimension is .25". Note that this dimension is the distance between vertical cutting passes and is usually equal the diameter of the bit being used but this tutorial has shallow geometry. Finally set the "Cut Levels Ordering" to "Depth First". PLEASE NOTE THAT- CUT AND DRILL DEPTHS ARE A CRITICAL SETTING AND SHOULD ALWAYS BE DOUBLE CHECKED BEFORE YOU GENERATE YOUR CUT PATH. NEVER CUT TO DEEP OR THROUGH THE SPOIL BOARD!

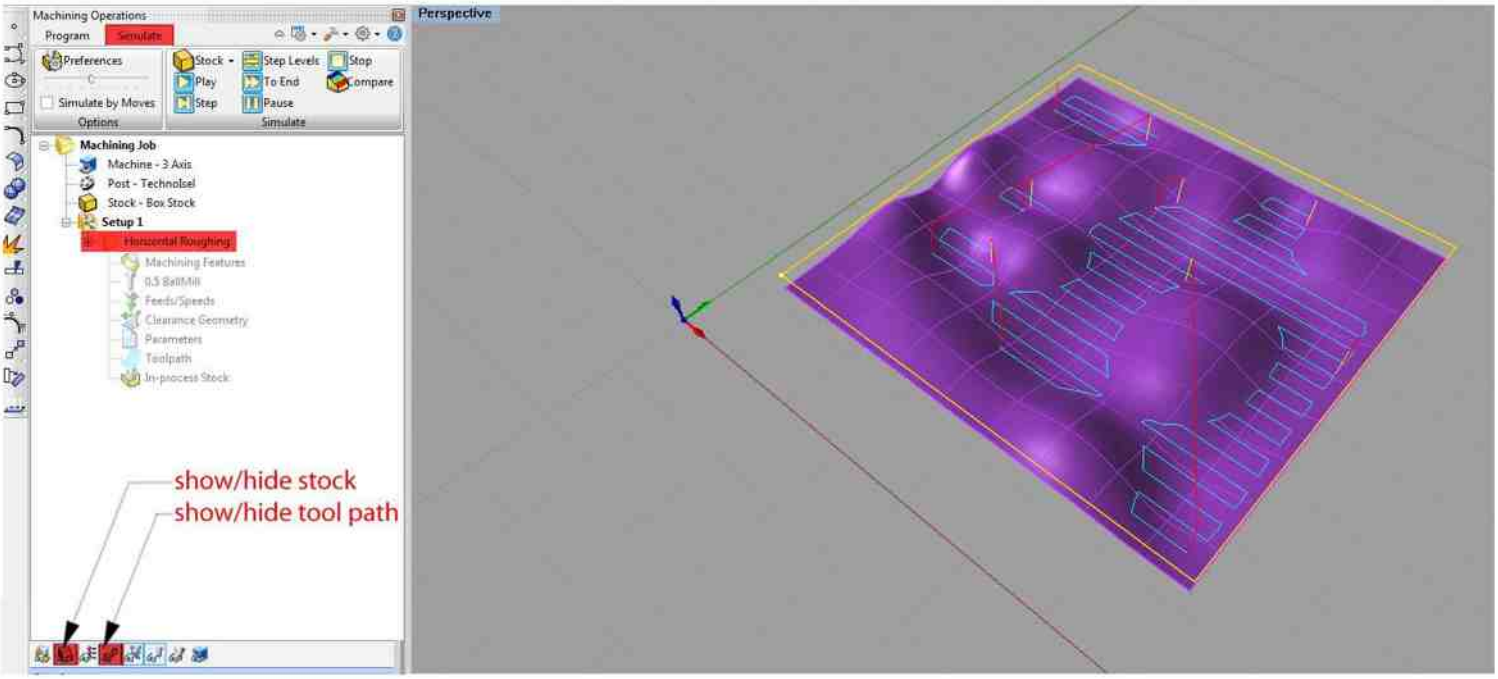



H) Select the "Engage/Retract" tab. Make sure the "Always engage in previously cut area if possible" tab is selected. Make sure the "Engage/Retract in Air" option is set to "Linear Extension" with a value of 0". This means that the machine will cut only with in the selected region. Once all of these options have been set, click on the "Generate" tab at the bottom of the pop up window. This will calculate the tool paths based on the inputs provided and create a new machining operation in the "Setup 1" folder.

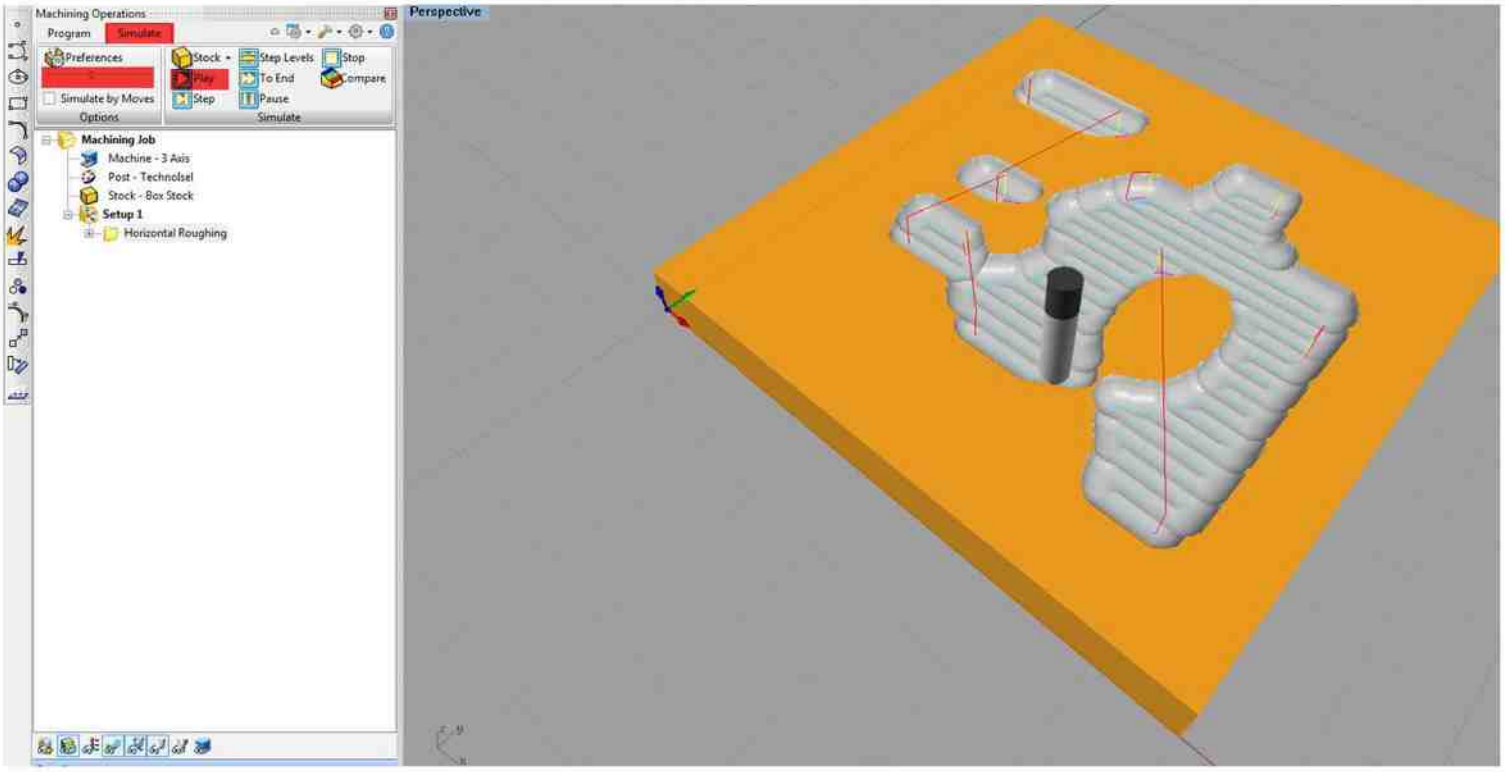


# STEP 6 RHINOCAM: CREATING HORIZONTAL ROUGHING PATH

I) You should now see an overlay of tool paths over your geometry in Rhino. This is a preview of the way the router will carve out your geometry. Blue lines indicate paths of travel through the material. Red lines indicate paths of travel across the top of the material. Yellow lines indicate movement in the Z axis. You will also see the tool path you created under the Mops Browser to the left. If you need to make a change to your tool path, double click the folder to reveal the different options you set and double click them to edit settings. At the bottom left, you'll see options to show/hide tool paths.

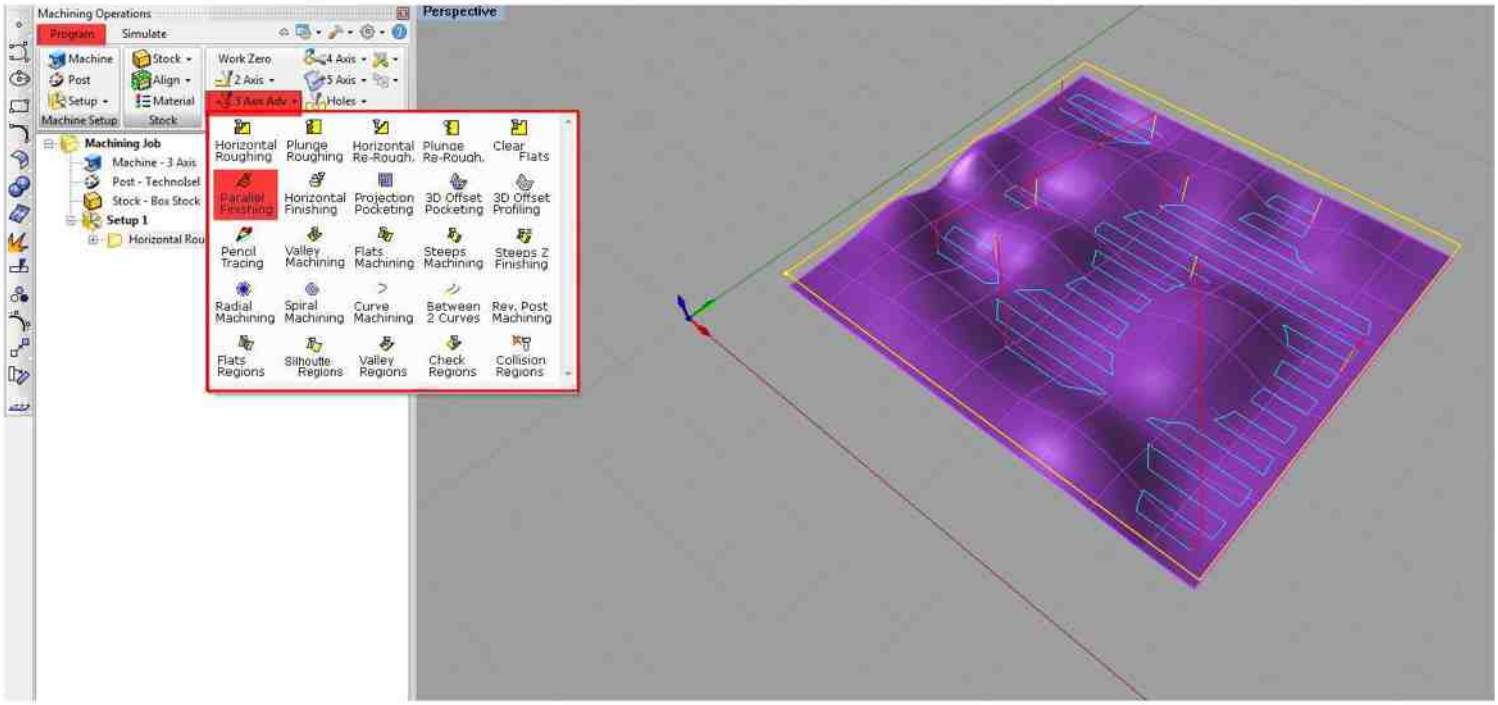



J) Highlight the tool path folder ("Horizontal Roughing") by single clicking on the folder. Click on the "Simulate" tab to preview your tool path. On the left, drag the slider to the far left to slow down the speed of the simulation. Click on the "Play" button to start the simulation. You should see the simulated bit running along the blue tool paths and at the end you should see the stock with material removed. Note: Horizontal Roughing only mills large regions. Your surface will NOT look like your Rhino geometry until you run a finishing pass.

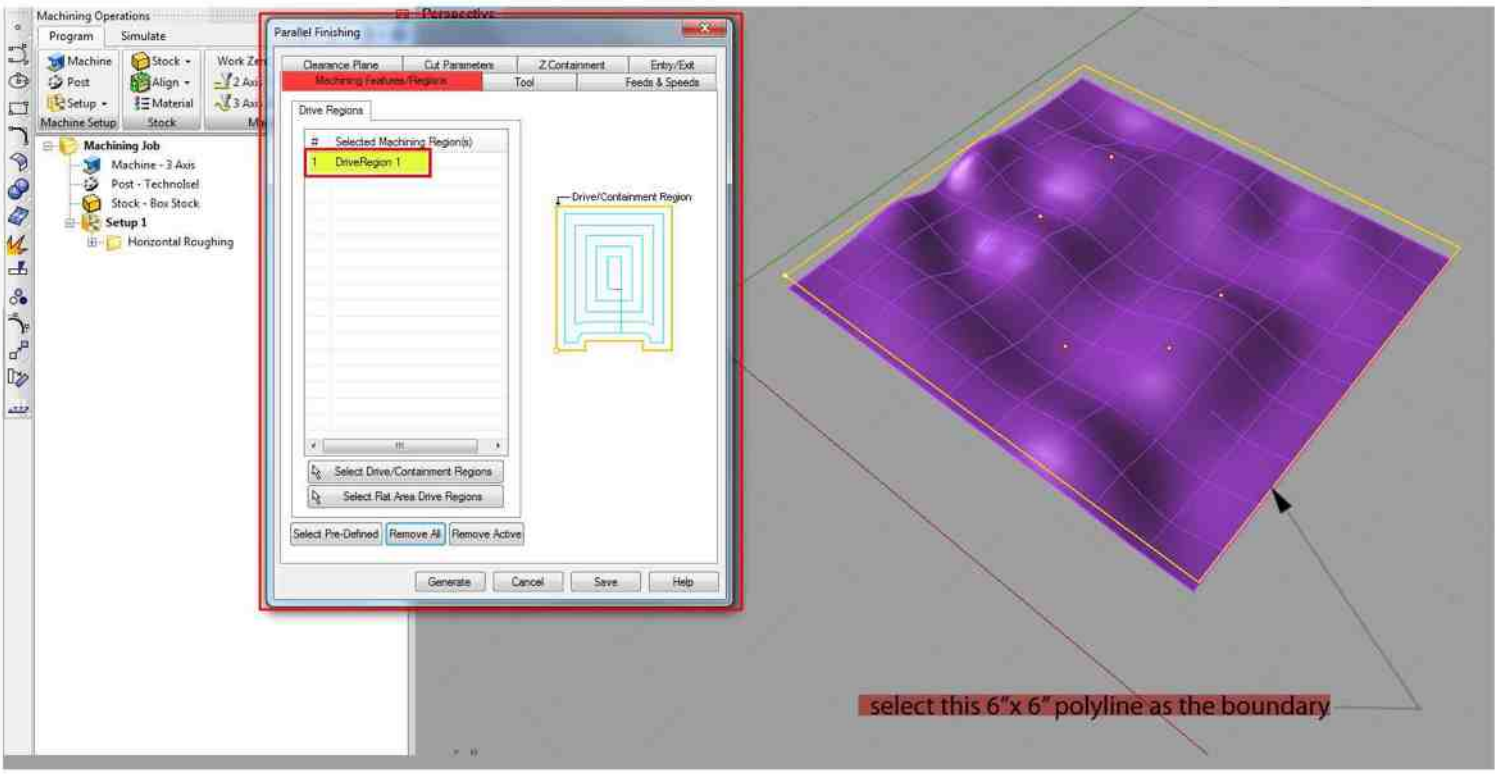


# STEP 7 RHINOCAM: CREATING PARALLEL FINISHING

A) These procedures will seem familiar because they are similar to those used to create the "Horizontal Roughing" operation. First click on the button for 3-axis operations and select "Parallel Finishing" from the drop down menu. A window will pop up that has all of the cut parameters and settings. Note: Parallel Finishing cleans the geometry created by your Horizontal Roughing pass.

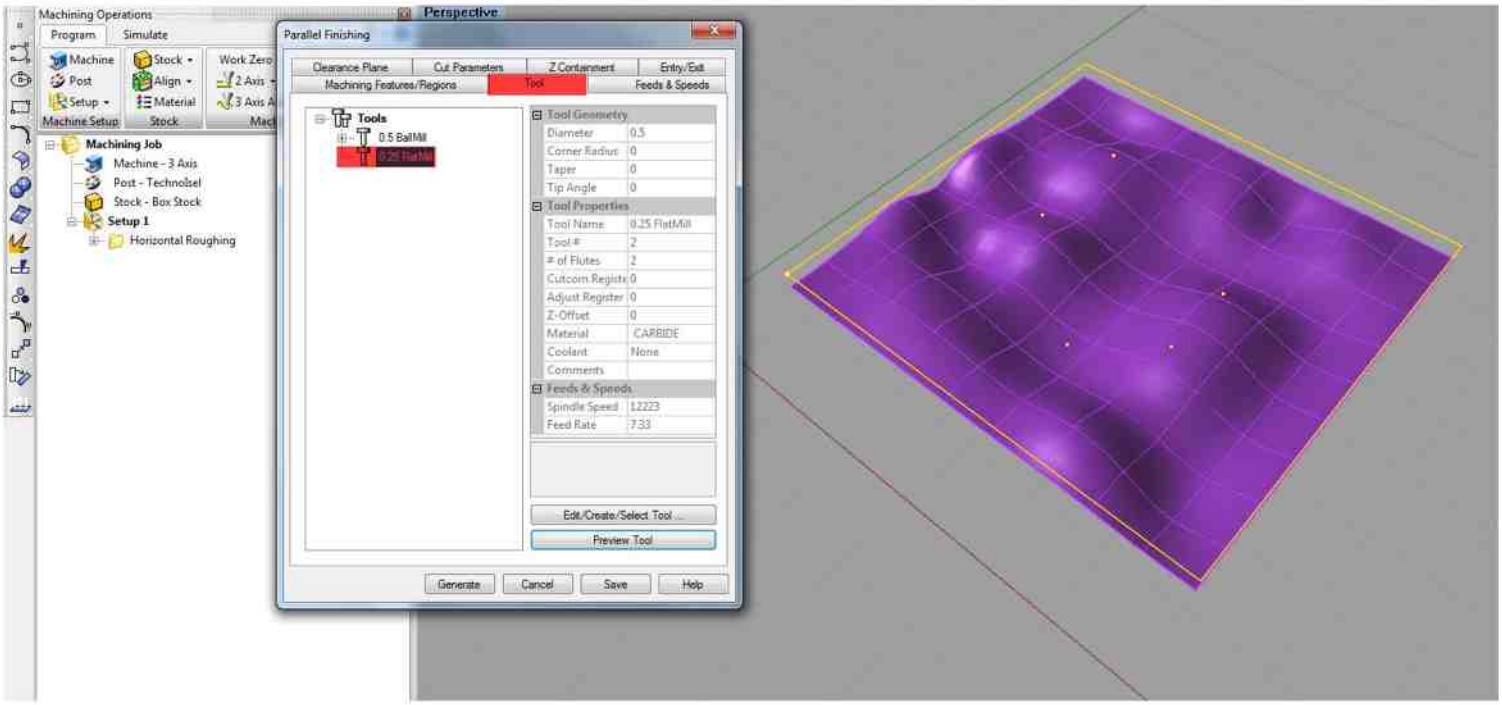



B) The same dialog box will appear with a series of similar option tabs. Under the "Maching Features/Regions" tab make sure you use the same polyline that was used for your horizontal roughing pass. Selecting the same region means you are roughing and finishing in the same boundary.

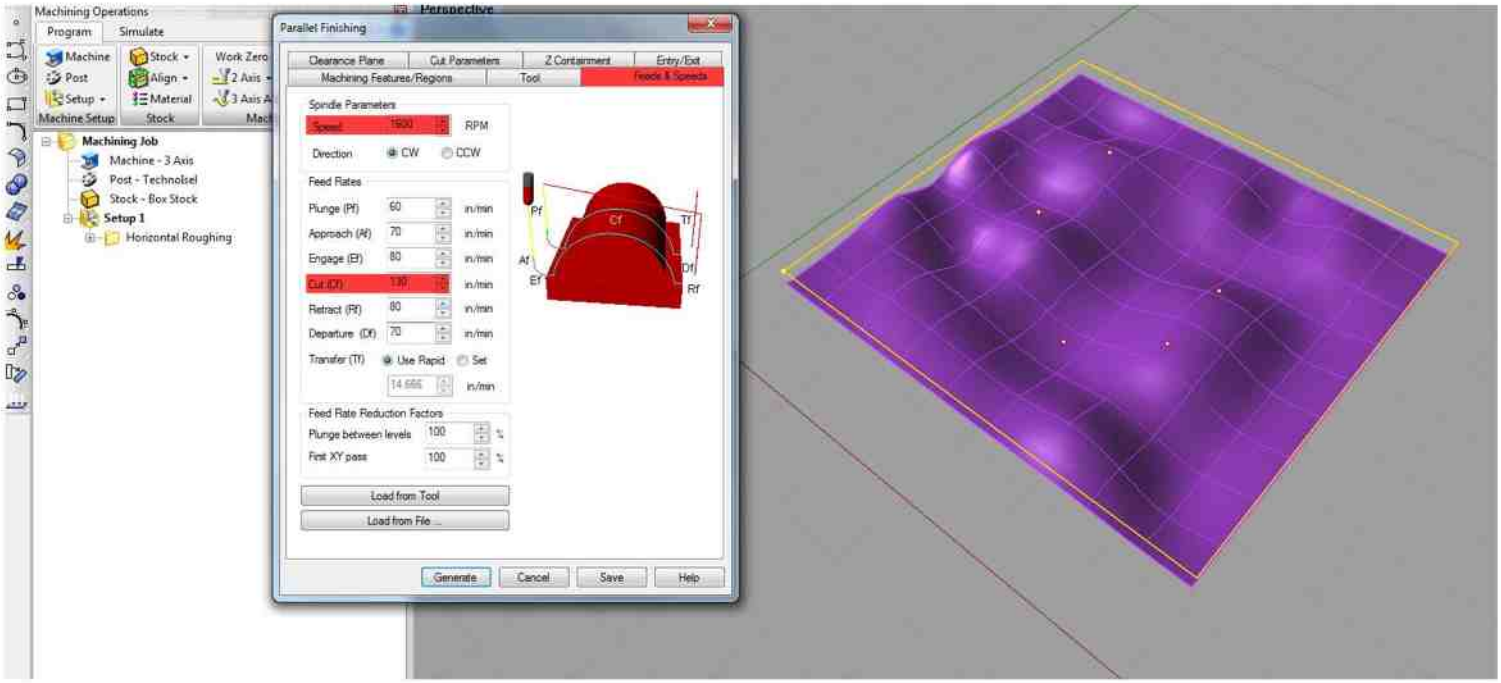


# STEP 7 RHINOCAM: CREATING PARALLEL FINISHING

C) Click on the "Tool" tab. Select the endmill "0.25 flat." When running a surface finish/parallel finishing path, you want to use an endmill that can produce a finer resolution than your surface rough, generally 1/2 the diameter of the surface rough endmill.

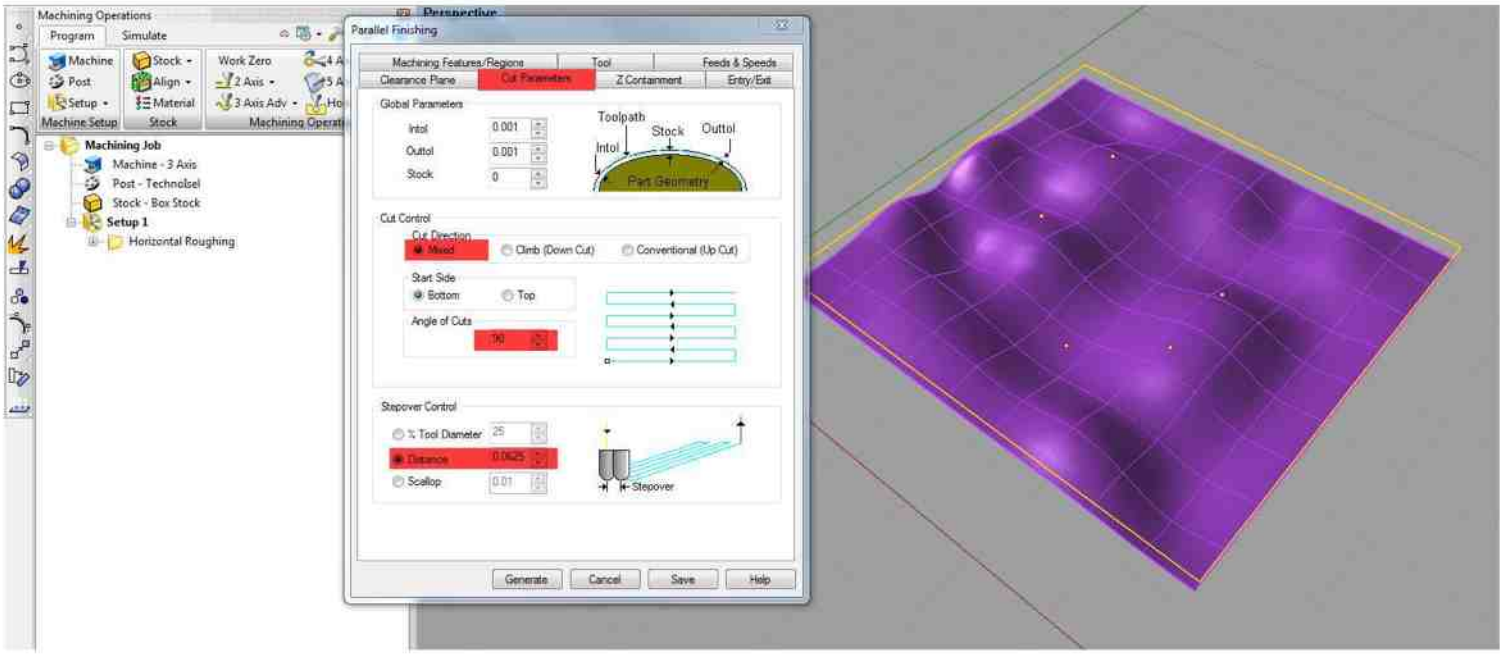



D) Select the "Feeds & Speeds" tab. These **speeds** (the rate at which the router moves or cuts, measured in IMPs or inches per minute) and **feeds** (the rate at which the spindle rotates, measured in RPMs) will change depending on the material being cut, the bit being used and the desired smoothness of the cut. **(AS NOTED, THIS IS ONE OF THE MOST CRITICAL CALCULATIONS WHEN MILLING)** For further information on these parameters please consult your instructor or CNC shop literature. For this tutorial the CNC staff has calculated values based on the MDF material selection and the end mill bits supplied. For this tutorial the spindle speed should be set at 14,000 RPM in a clockwise direction and the cutting speed should be set at 100 IPM. See below for descriptions and values of Plunge, Approach, Engage, Retract and Departure Speeds. The "Transfer" field should be set to "Use Rapid" and both of the "Feed Rate Reduction Factors" should be set to 100%.

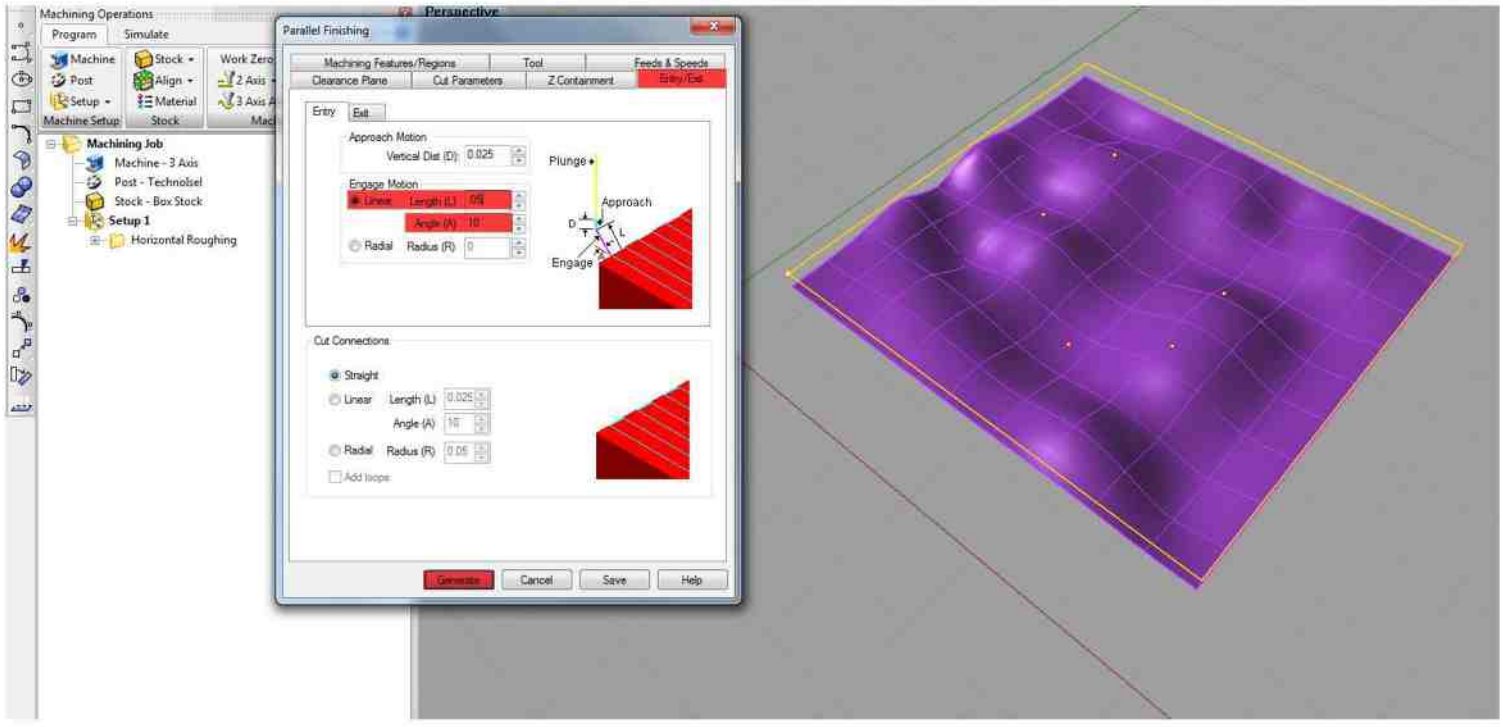


# STEP 7 RHINOCAM: CREATING PARALLEL FINISHING

E) Click on the "Cut Parameters" tab. Under "Cut Control," select "Mixed." Under "Angle of Cuts," set desired degree between 0 - 90. This will run the finishing tool path a chosen degree angle from the rouging tool path, giving a smoother finish. Under "Stepover Control," select "Distance" and set this to a dimension no less than 1/4 and no more than 1/2 of the diameter of the bit being used. The smaller this dimension the finer the finish. In this case set this dimension to 1/4 of diameter of the bit being used, or .0625" Note that with Parallel Finishing, you set the stepover but not the stepdown. This is because the tool path goes right down to the surface you modeled. Be sure to run a roughing pass before the finish pass, otherwise you may break the endmill.

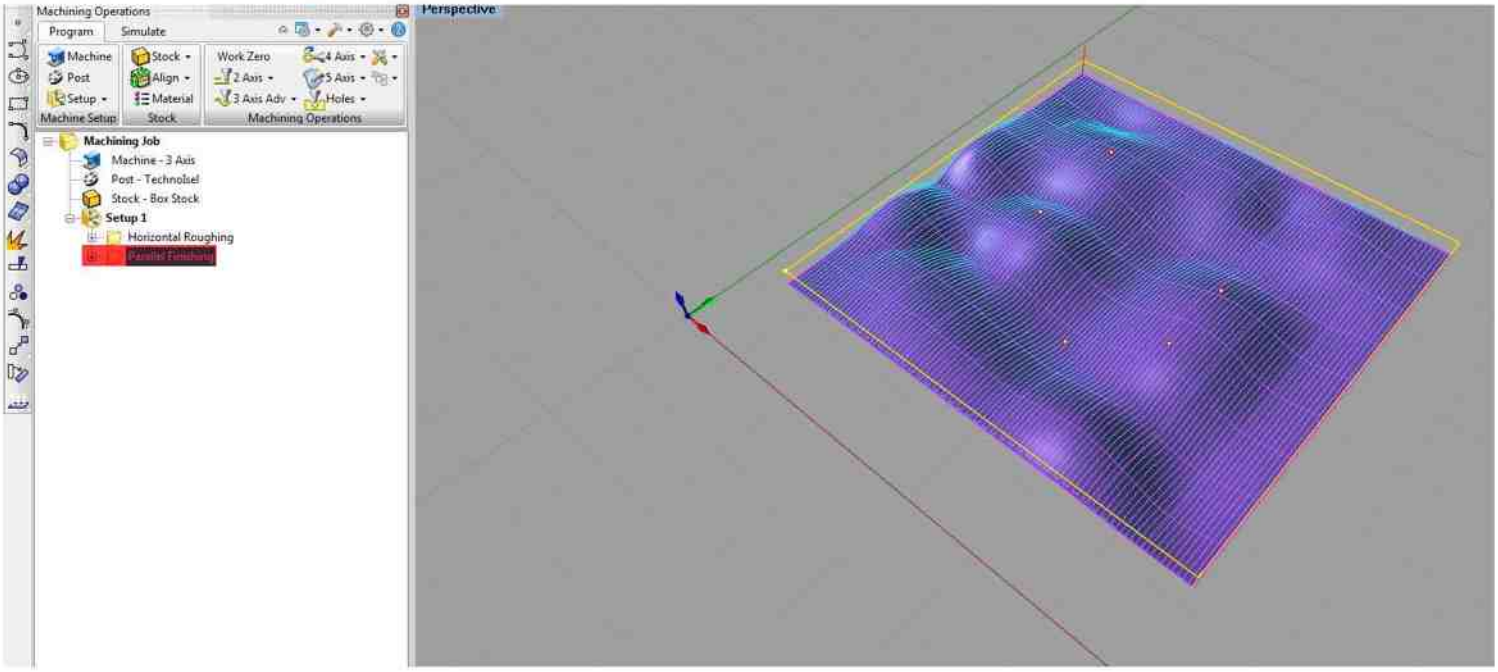



F) Click on the "Entry/Exit" tab. Make sure the "Engage Motion" is set to "Linear" with a length of .05" and and angle of 10 degrees. Make sure the "Cut Connections" are set to "Straight" This means that the machine will cut only with in the selected region. Once all of these options have been set, click on the "Generate" tab at the bottom of the pop up window. This will calculate the tool paths based on the inputs provided and create a new machining operation called "Parallel Finishing" in the "Setup 1" folder.

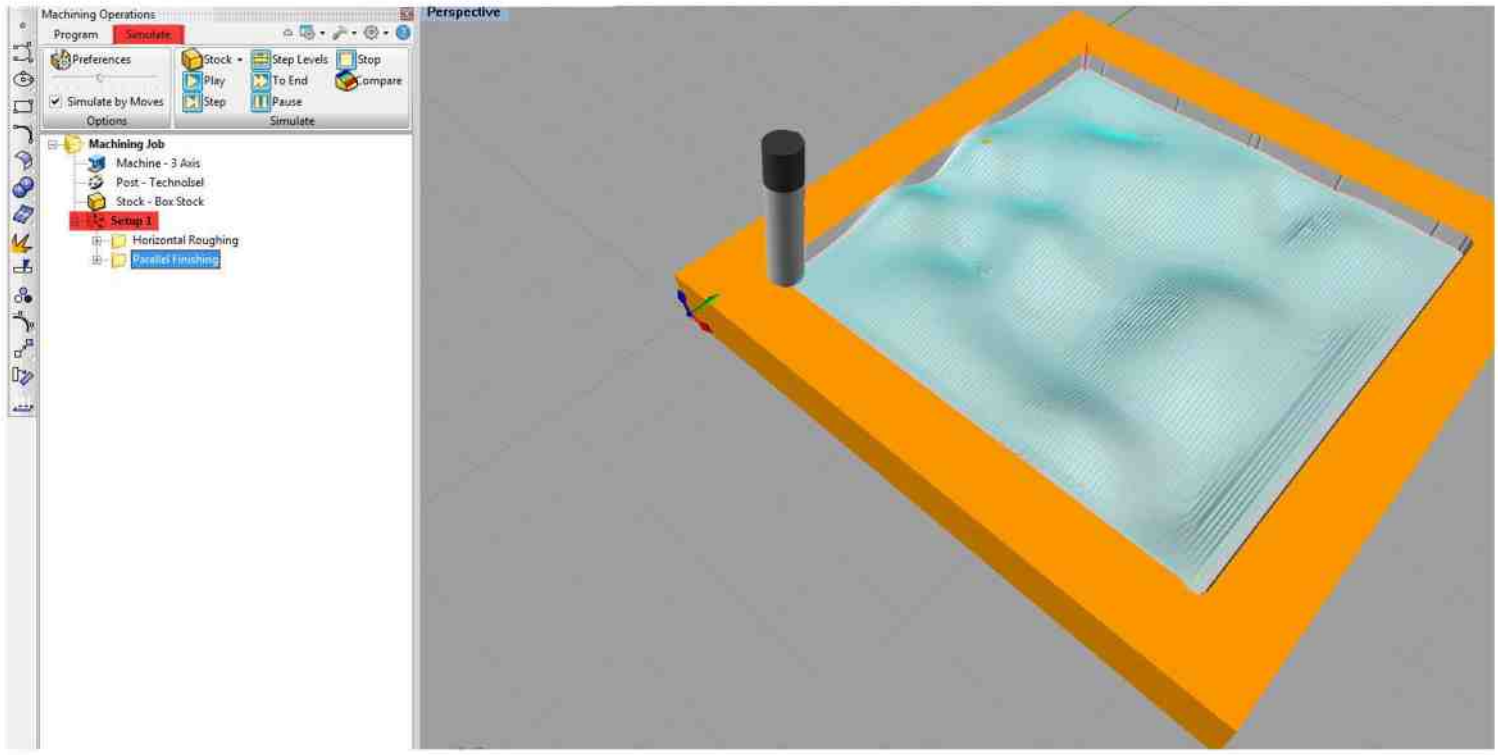


# STEP 7 RHINOCAM: CREATING PARALLEL FINISHING

G) You should now see an overlay of tool paths over your geometry in Rhino. This is a preview of the way the router will carve out your geometry. Blue lines indicate paths of travel through the material. You will also see the tool path you created under the Mops Browser to the left. If you need to make a change to your tool path, double click the folder to reveal the different options you set and double click them to edit settings.

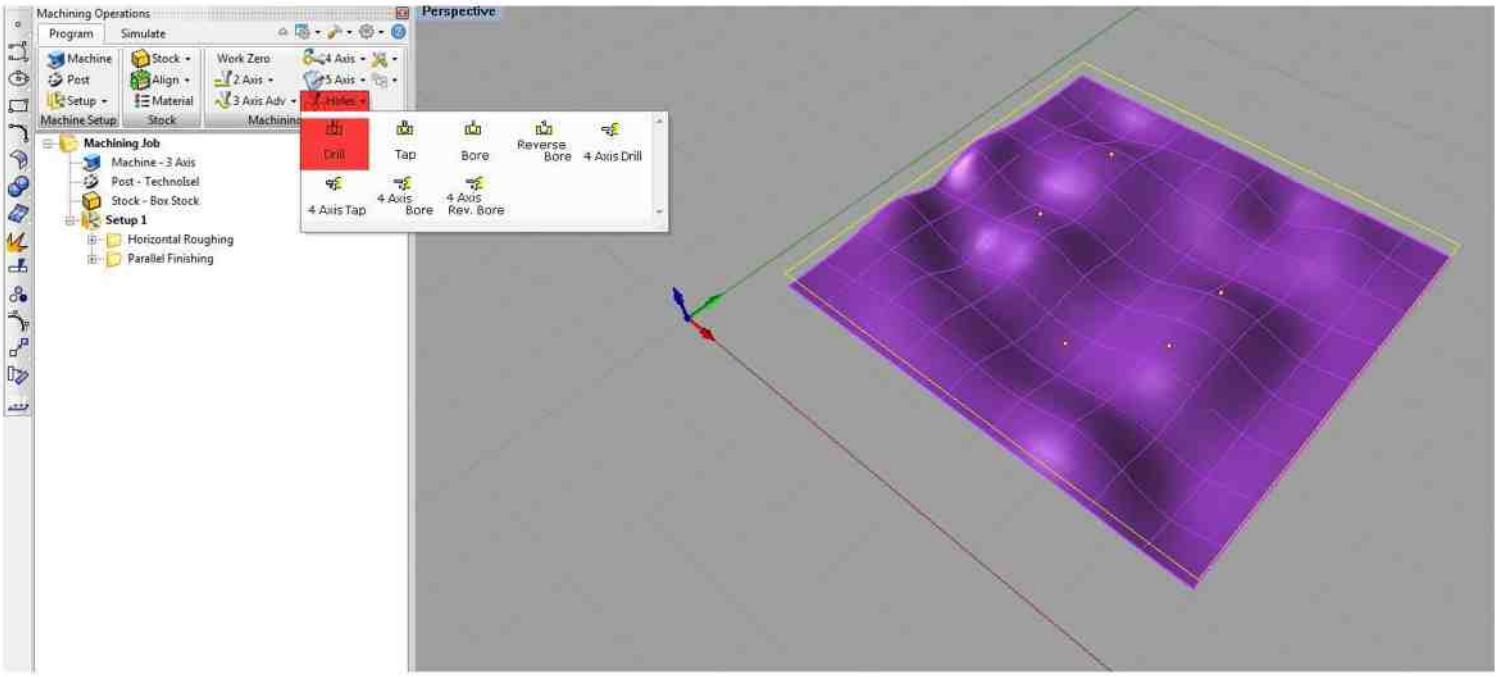



H) Highlight the tool path folder "Setup 1" by single clicking on the folder. Click on the "Simulate" tab to preview your tool path. On the left, drag the slider to the far left to slow down the speed of the simulation. Click on the "Play" button to start the simulation. You should see the simulated bit first running your roughing pass then running along the blue tool paths of your parallel finish pass and at the end you should see the stock with material removed after the two operations.

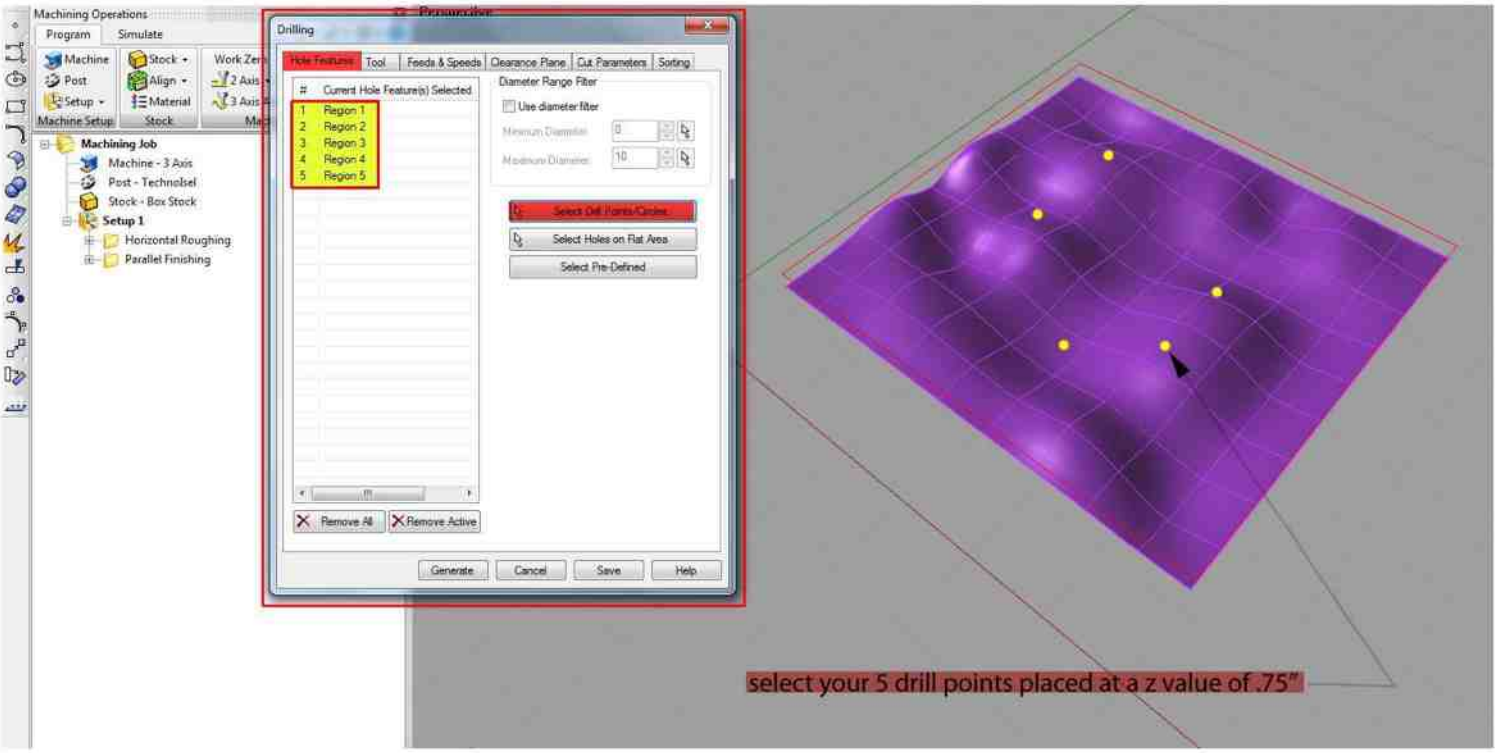


# STEP 8 RHINOCAM: CREATING DRILLING OPERATIONS

A) Next will be the creation of the drilling operations. First click on the button for "Holes" and select "Drill" from the drop down menu. The window will pop up that has all of the parameters and settings for your drilling operations. Again, these options will look familiar.

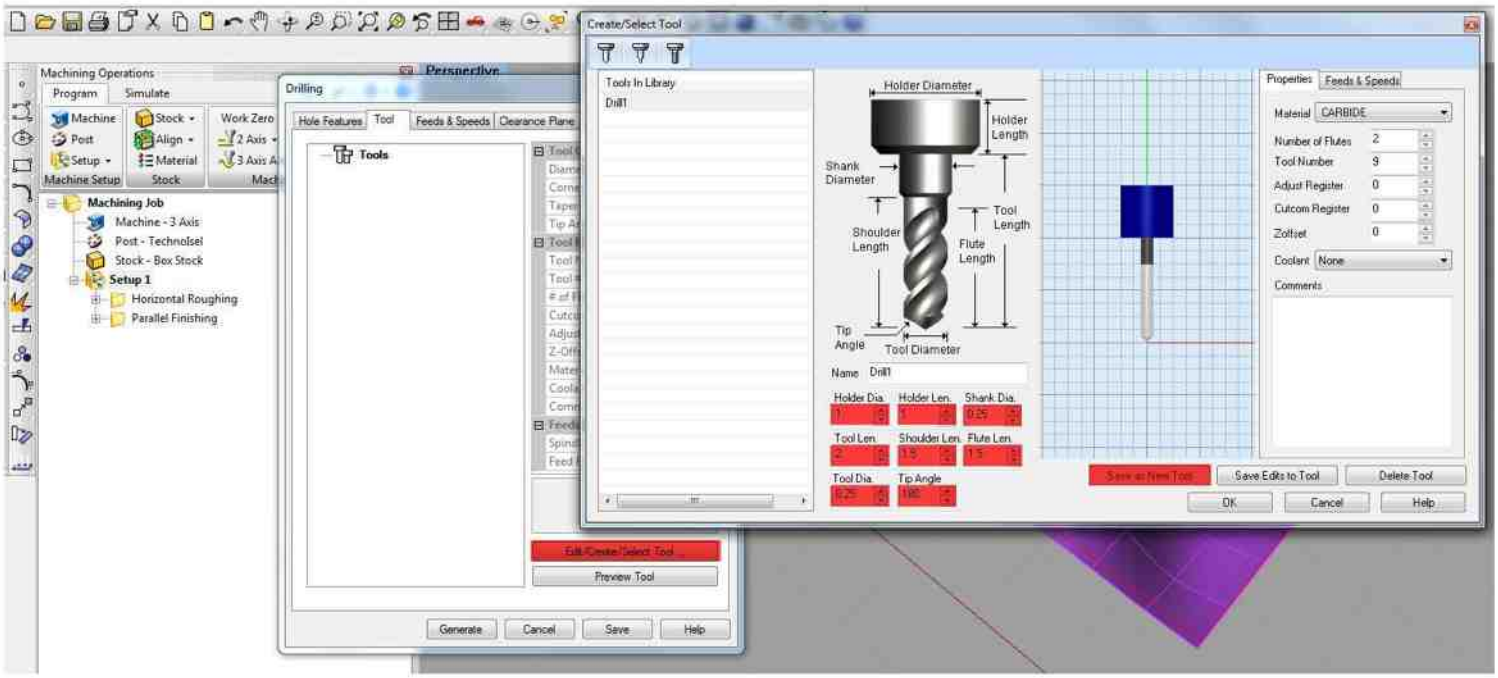



B) Under the first tab, "Hole Features," you will choose the drill points. Click on "Select Drill Points/Circles". The window will minimize and you'll be asked to select your drill points from the Rhino model. In this case, select your 5 points. Press enter when done selecting.

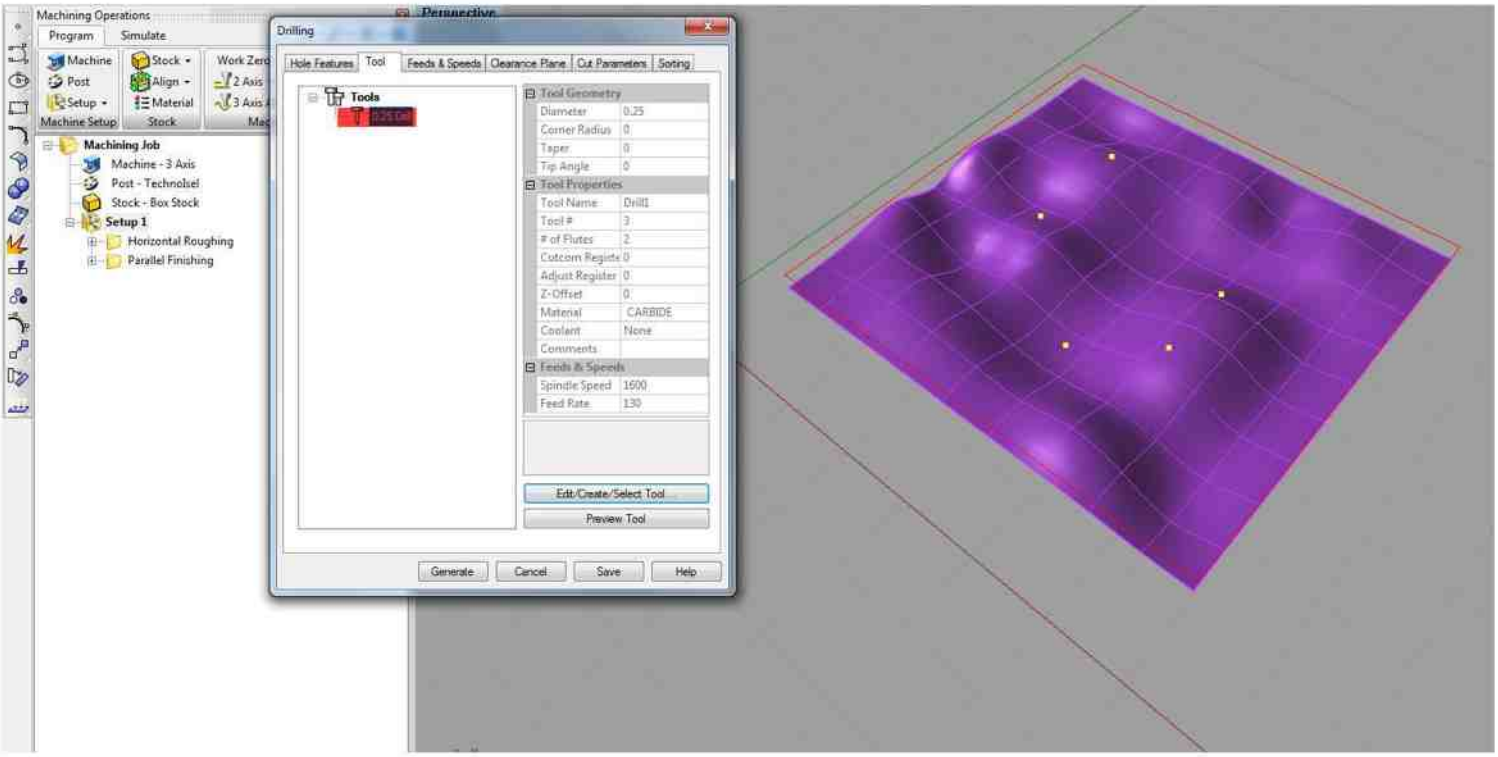


# STEP 8 RHINOCAM: CREATING DRILLING OPERATIONS

C) Click on the "Tool" tab. Even though you'll be using the 0.25 flat endmill, you will have to set up a new drill tool. Click on "Edit/ Create/Select Tool." A window will pop up. In the new tool popup window, name the tool "0.25 drill." Set "Tip Angle" to 180 degrees (a flat-ended drill). Set "Tool diameter" to 0.25. Click "Save as New Tool." Click OK to close the window.

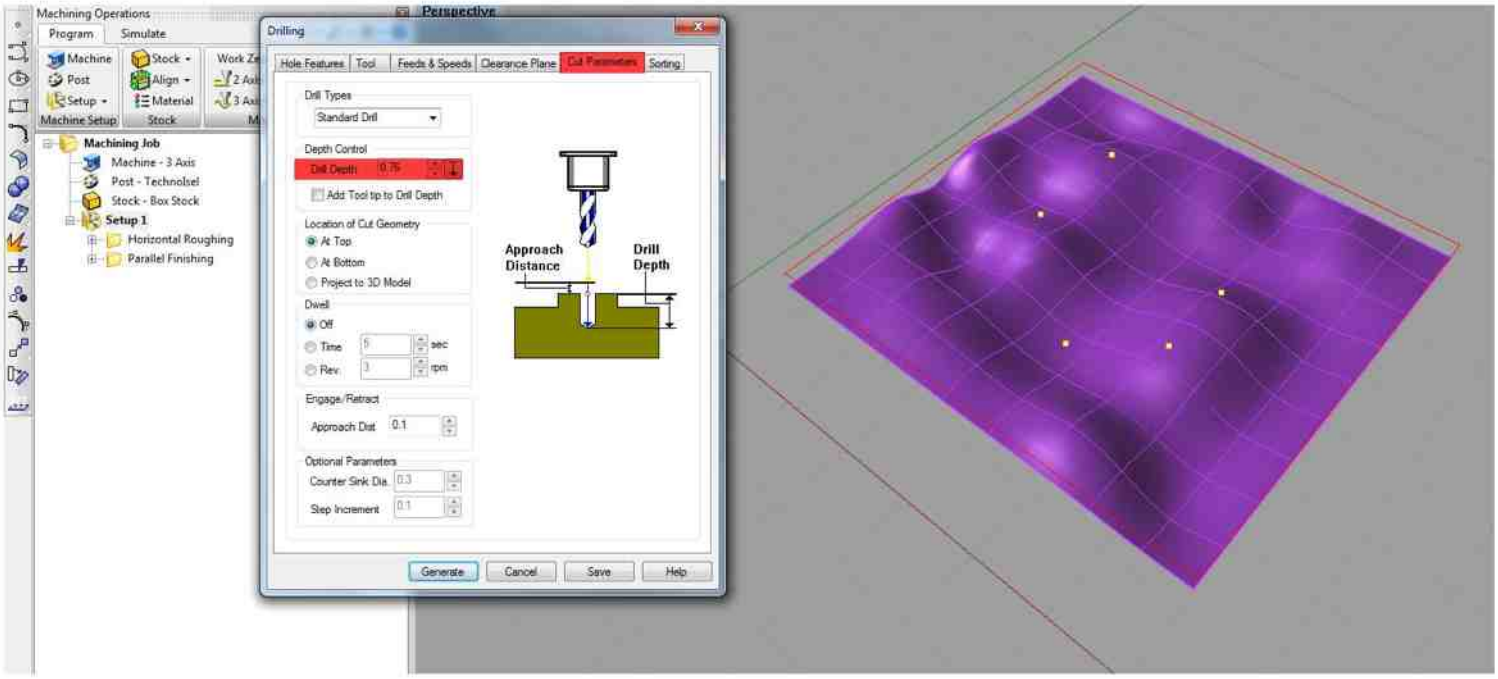



D) Under the "Tool" tab, the new 0.25 drill tool should appear. Click on it to make sure it is selected.

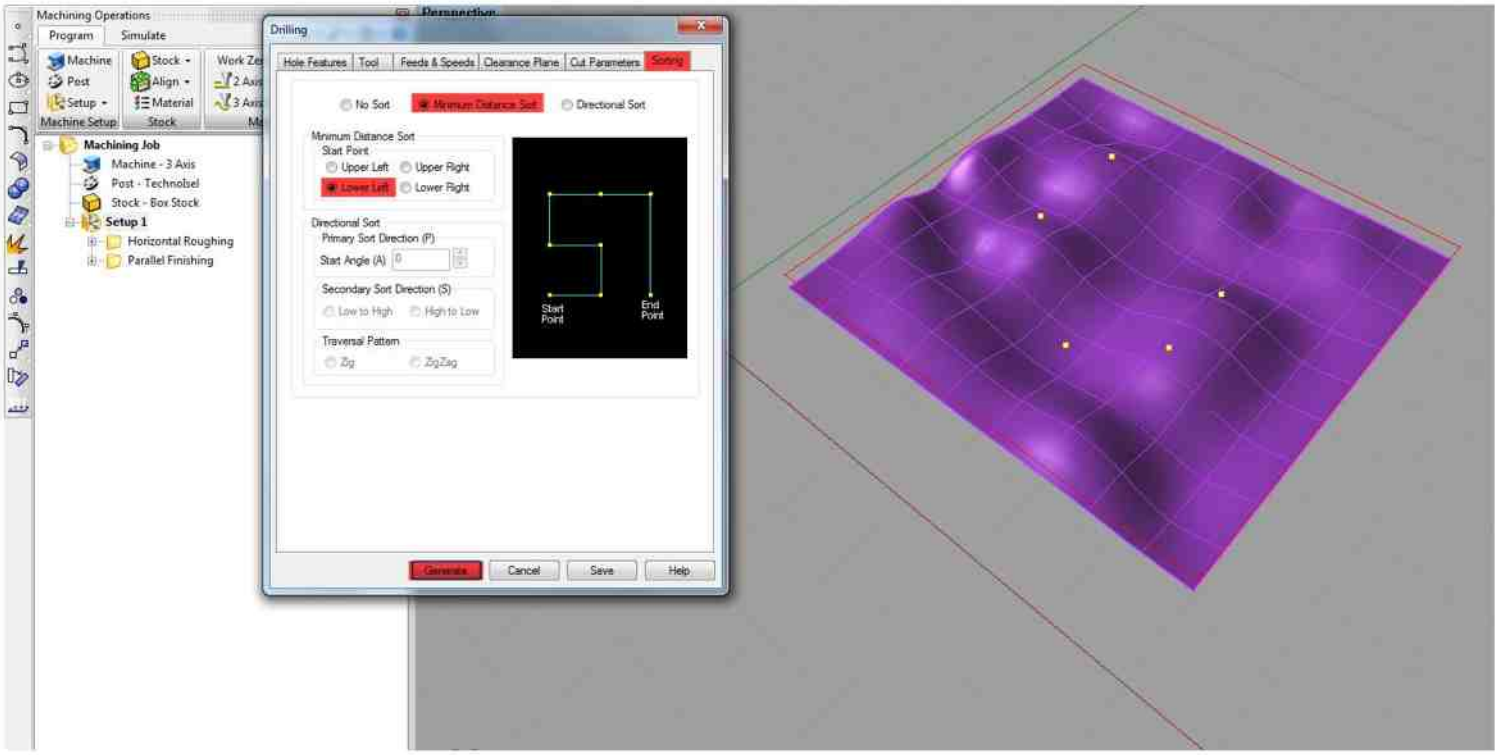


# STEP 8 RHINOCAM: CREATING DRILLING OPERATIONS

E) The "Feeds & Speeds" and "Clearance Plane" options should be the same as before so we can skip them. Next click on the "Cut Parameters" tab. Under "Depth Control," set "Drill Depth" to be 0.76 inches. The tutorial material is 0.75" thick, so setting the drill depth slightly deeper ensures a clean drill through the material. However do not go too deep! You don't want to be drilling into spoil board to deeply and need to be sure you NEVER CUT THROUGH THE SPOIL BOARD! CUT AND DRILL DEPTHS ARE A CRITICAL SETTING AND SHOULD ALWAYS BE DOUBLE CHECKED BEFORE YOU GENERATE YOUR CUT PATH.

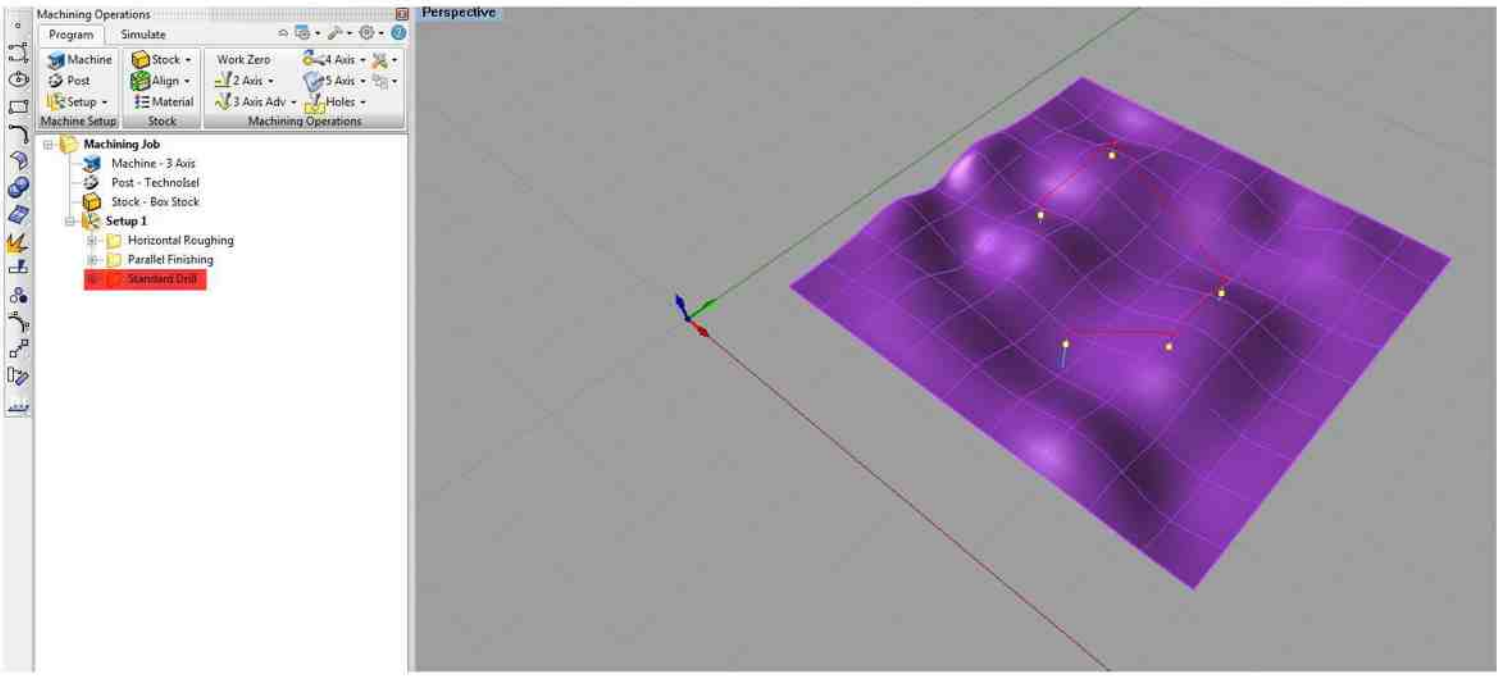



F) Click on the "Sorting" tab. Select "Minimum distance sort." Under "Start Point," select "Lower left." This will start your drill operation at the lower left hand hole, and continue on a minimum distance path from hole to hole. Once all of these options have been set click "Generate."

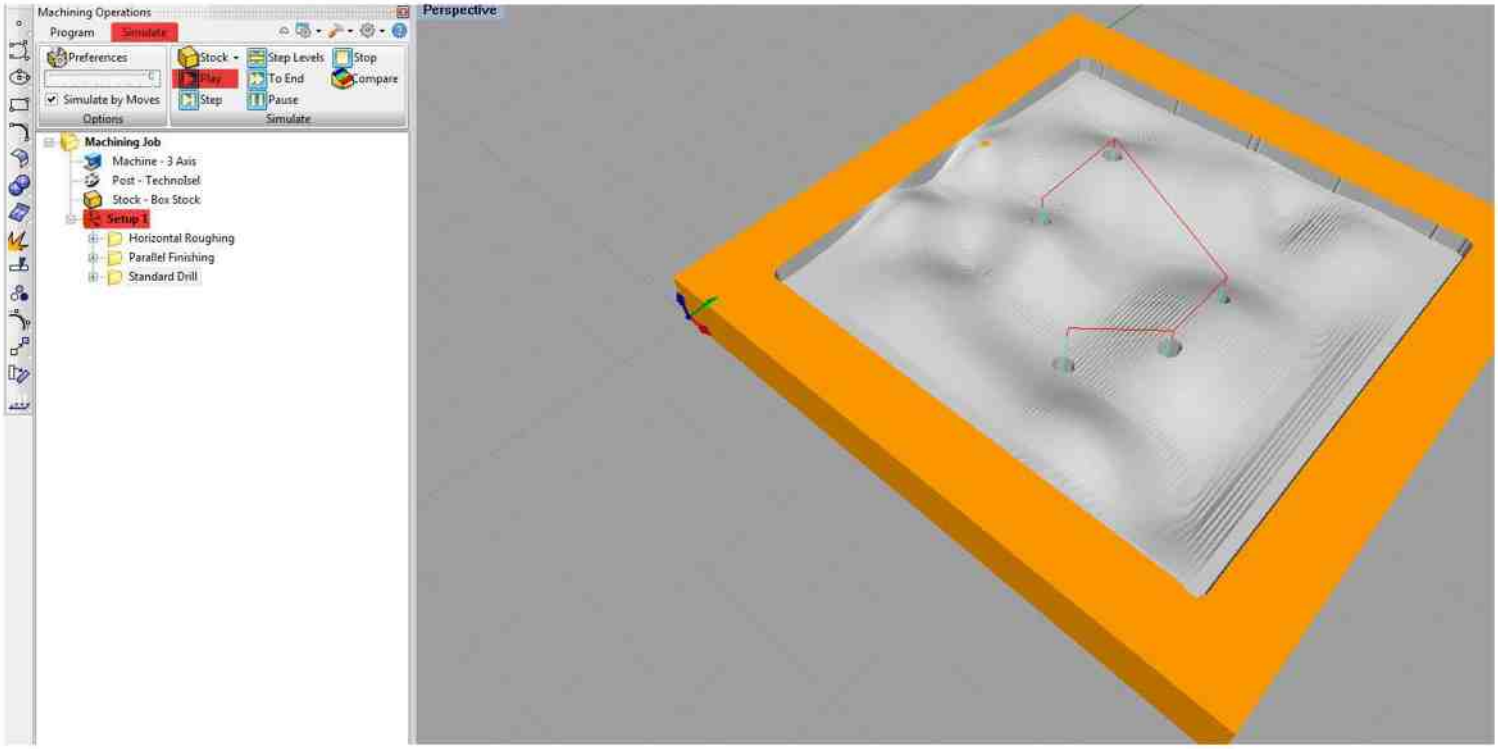


# STEP 8 RHINOCAM: CREATING DRILLING OPERATIONS

G) Once again you should now see an overlay of the new drilling tool paths over your geometry in Rhino. In this case the red lines are jogs and the blue lines are drilling operations. You will also see the "Standard Drill" tool path you created under the Mops Browser to the left. If you need to make a change to your tool path, double click the folder to reveal the different options you set and double click them to edit settings.

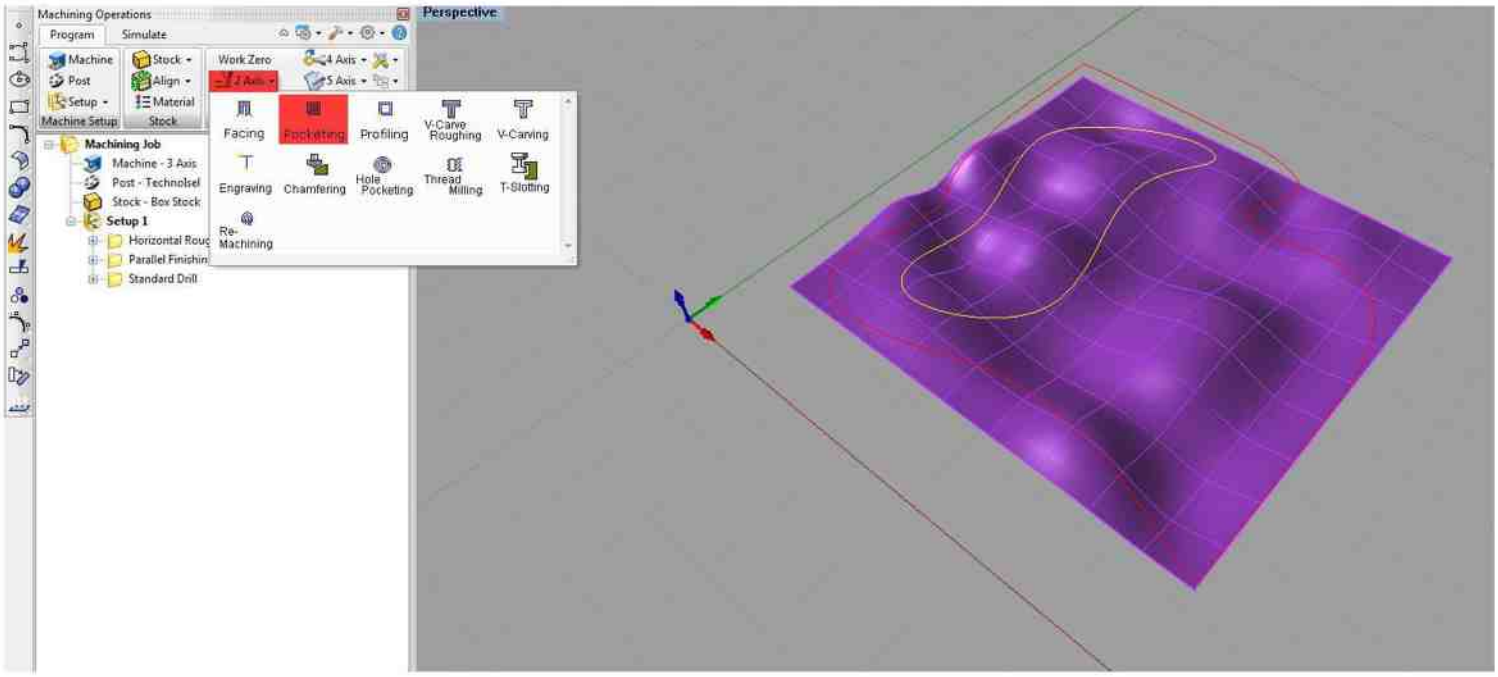



H) Highlight the tool path folder "Setup 1" by single clicking on the folder. Click on the "Simulate" tab to preview your tool path. On the left, drag the slider to the far left to slow down the speed of the simulation. Click on the "Play" button to start the simulation. You should see the simulated bit first running your roughing pass then running along the blue tool paths of your parallel finish pass and finally, drilling your 5 holes. You should now see a simulated geometry that looks much like what you created in Rhino. Next we will create the pocketing operation that cuts the aperture in your geometry.

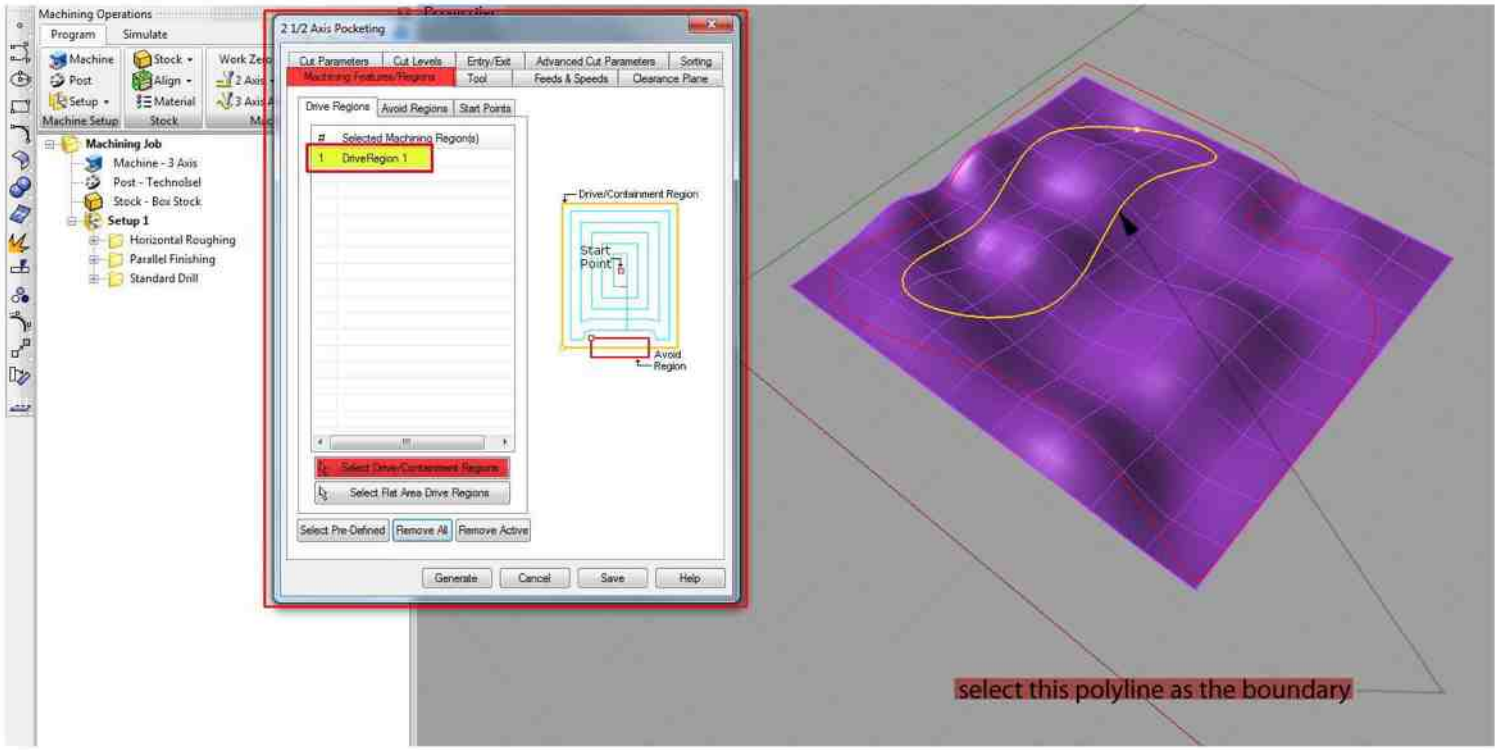


# STEP 9 RHINOCAM: CREATING POCKETING OPERATIONS

A) Click on the button for 2-axis operations and select "Pocketing" from the drop down. A window will pop up that has all of the cut parameters and settings.

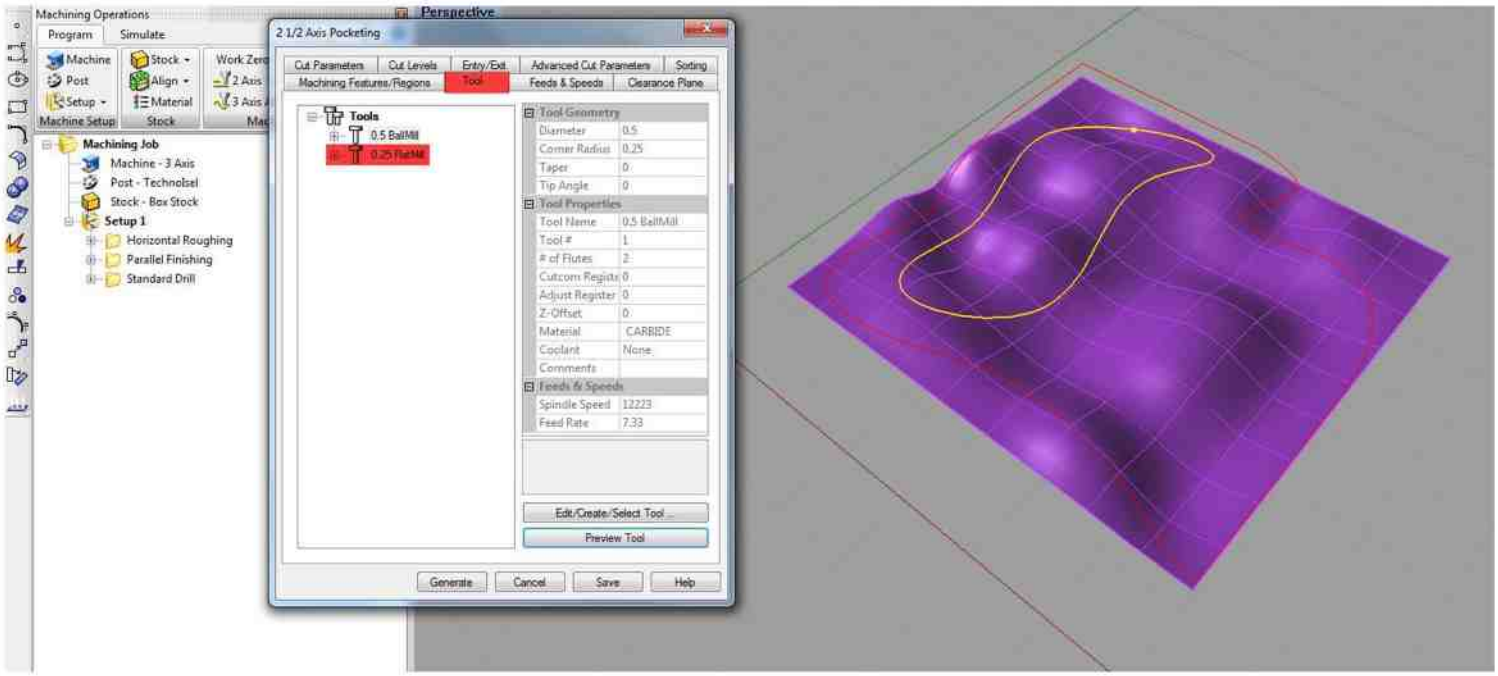



B) The same dialog box will appear with a series of similar option tabs. Under the "Maching Features/Regions" tab as before click the tab to "Select Drive/Containment Regions". Select the polyline you created on the interior of your geometry. Again this line should be located at the top surface of your stock with a Z value of .75

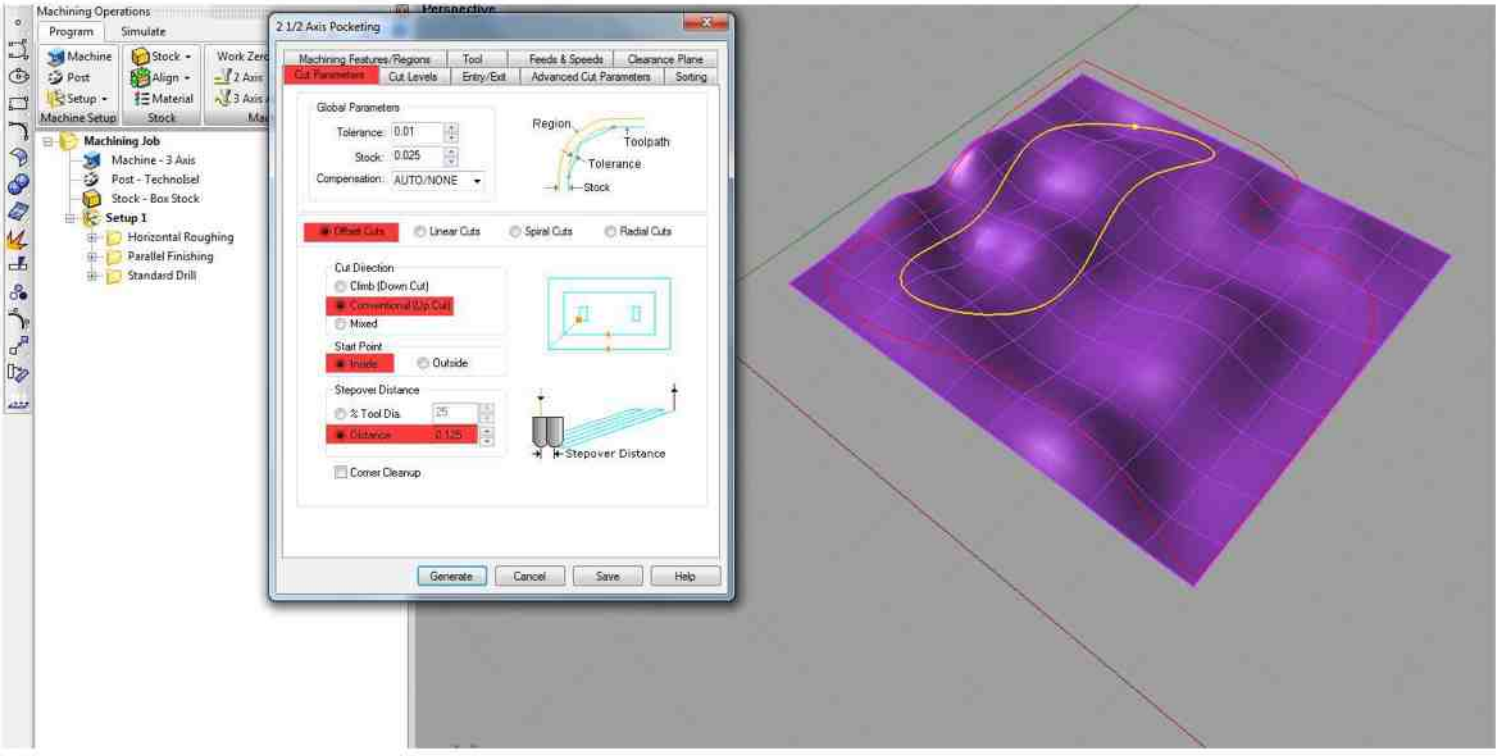


# STEP 9 RHINOCAM: CREATING POCKETING OPERATIONS

C) Click on the "Tool" tab. Select the endmill "0.25 FlatMill" that was created earlier

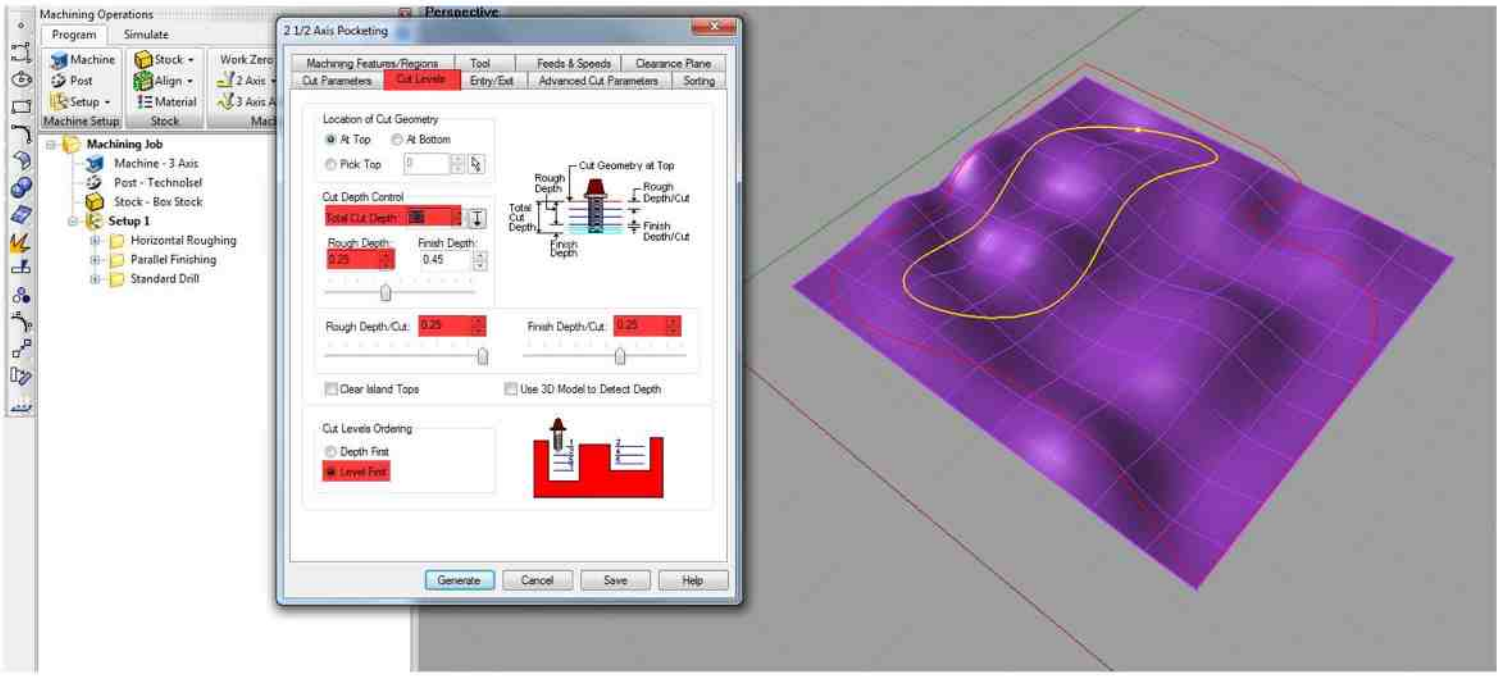



D) The "Feeds & Speeds" and "Clearance Plane" options should be the same as before so we can skip them. Next click on the "Cut Parameters" tab. Make sure "Offset Cuts" is selected. Under "Cut Direction," select "Conventional (Up-Cut)." Under "Start Point," select "Inside." Under "Stepover Distance" select a distance equal to half the diameter of the bit you are using. In this case that value is 0.125".

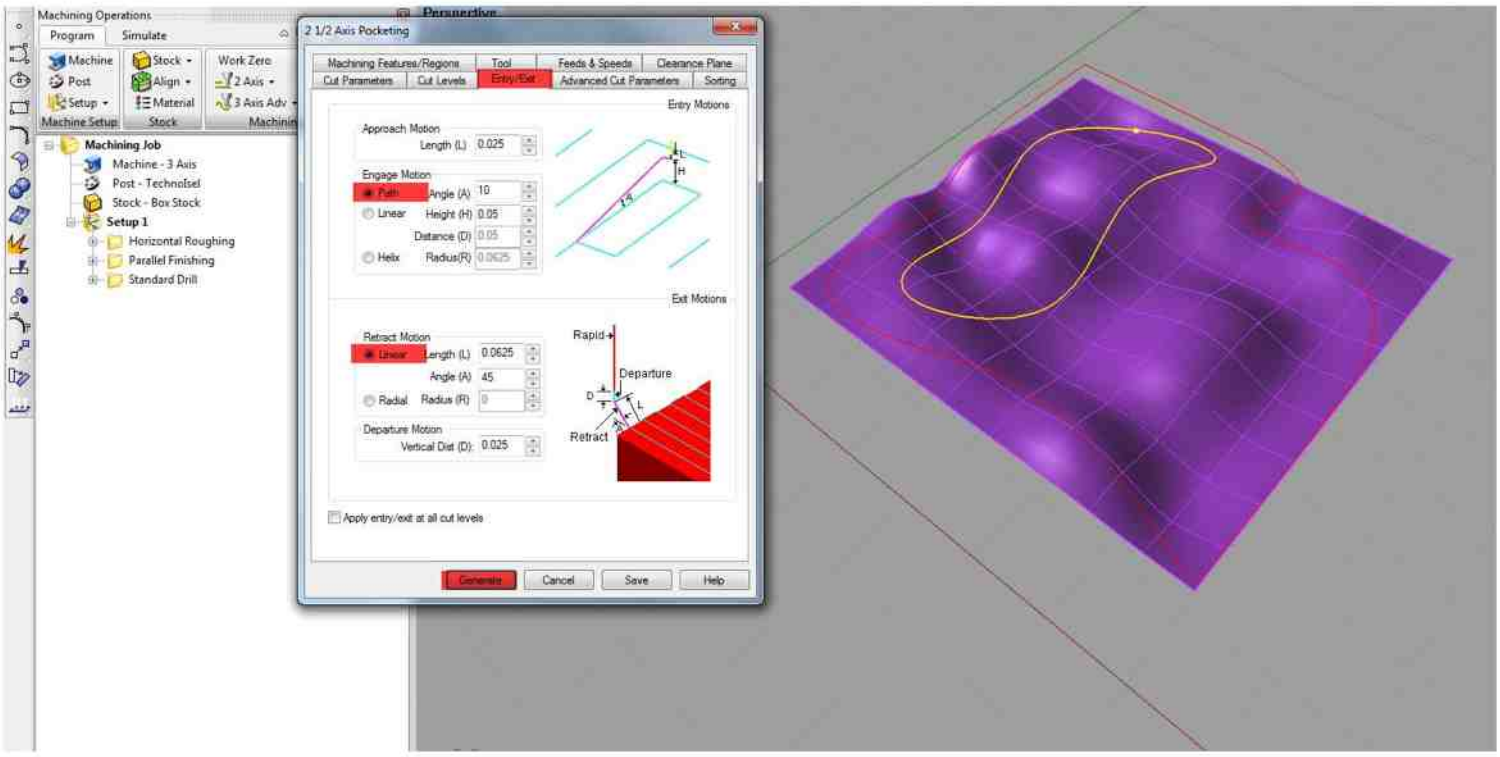


# STEP 9 RHINOCAM: CREATING POCKETING OPERATIONS

E) Click on the "Cut Levels" tab. Make sure "At Top" is selected. Under "Cut Depth Control," set to the depth you would like your pocket. The example uses 0.7", but if you wanted a shallow pocket, you could input 0.25", etc. Under "Rough Depth/Cut," set equal to the diameter of the endmill (0.25"). This is the same as "max. stepdown." Set the "Cut Levels Ordering" to "Level First". PLEASE NOTE THAT CUT AND DRILL DEPTHS ARE A CRITICAL SETTING AND SHOULD ALWAYS BE DOUBLE CHECKED BEFORE YOU GENERATE YOUR CUT PATH. NEVER CUT TO DEEP OR THROUGH THE SPOIL BOARD!

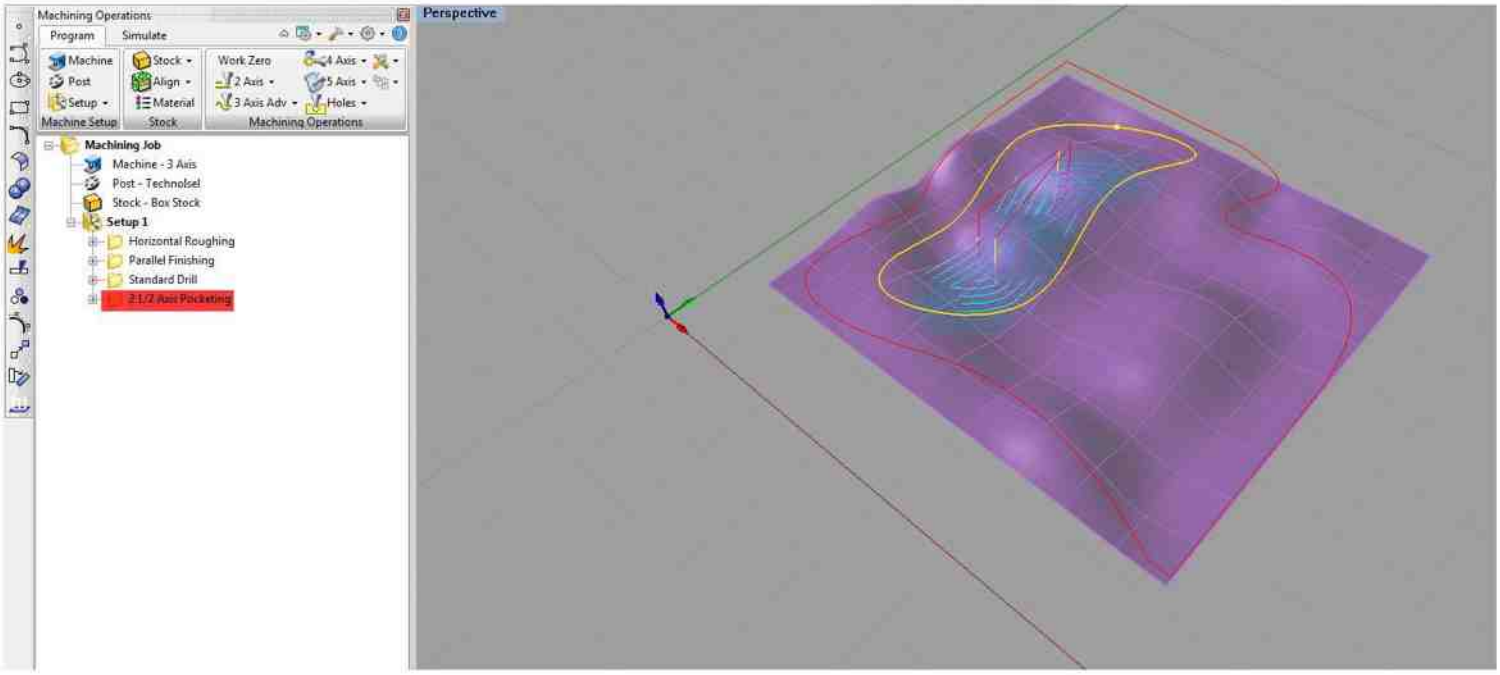



F) Click the "Entry/Exit" tab. These values should be a default based on the parameters of the Techno machine but its best to double check to be sure. Make sure the "Engage Motion" is set to "Path" with an angle of 10 degrees and a height of .05". Make sure the "Retract Motion" is set to "Linear" with an angle of 45 degrees and a length of .0625". Once all these options are set click on the "Generate" tab at the bottom.

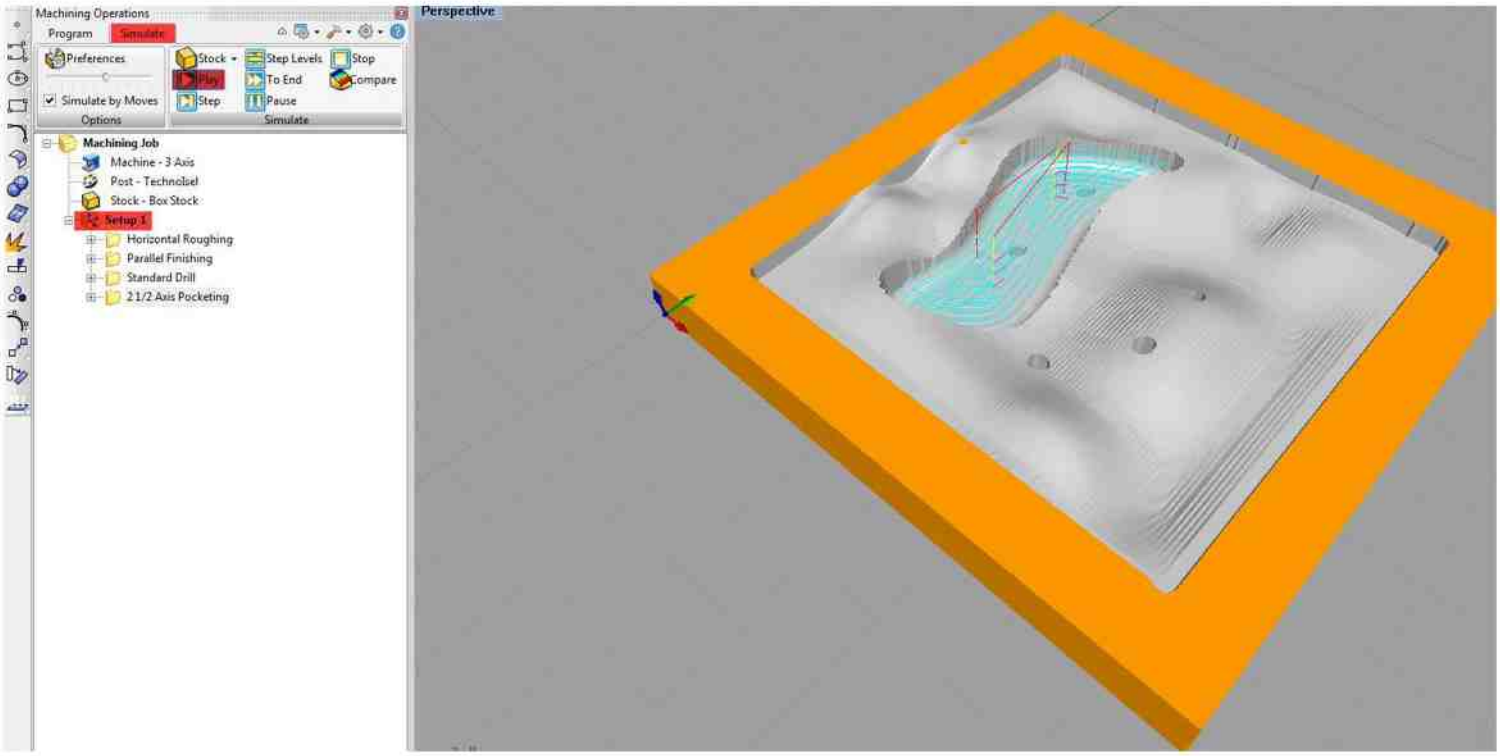


# STEP 9 RHINOCAM: CREATING POCKETING OPERATIONS

G) You should now see an overlay of the new pocketing tool paths over your geometry in Rhino. This is a preview of the way the router will carve out your Pocket. Blue lines indicate paths of travel through the material. You will also see the tool path you created under the Mops Browser to the left. If you need to make a change to your tool path, double click the folder to reveal the different options you set and double click them to edit settings.

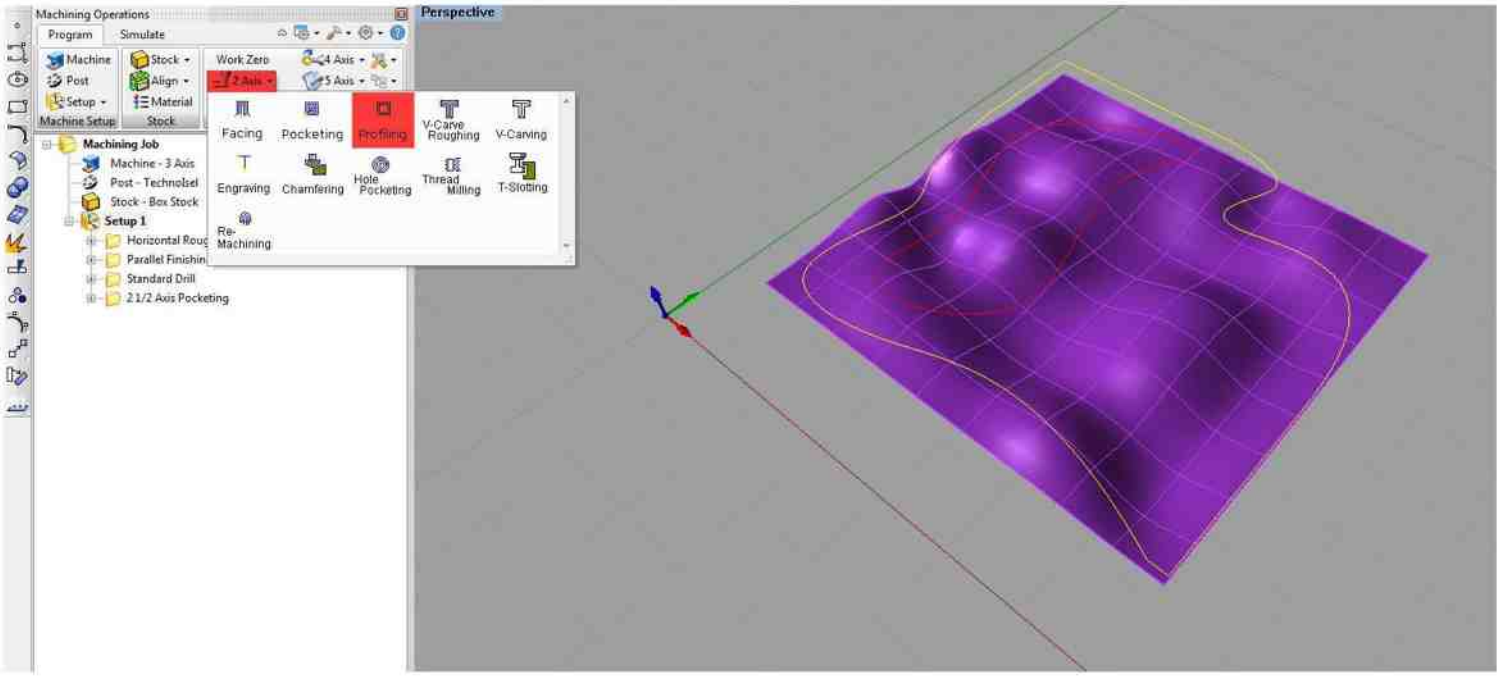



H) Highlight the tool path folder "Setup 1" by single clicking on the folder. Click on the "Simulate" tab to preview your tool path. On the left, drag the slider to the far left to slow down the speed of the simulation. Click on the "Play" button to start the simulation. You should see the simulated bit first running your roughing pass then running along the blue tool paths of your parallel finish pass and then drilling the holes and finally cutting the pocket operation.

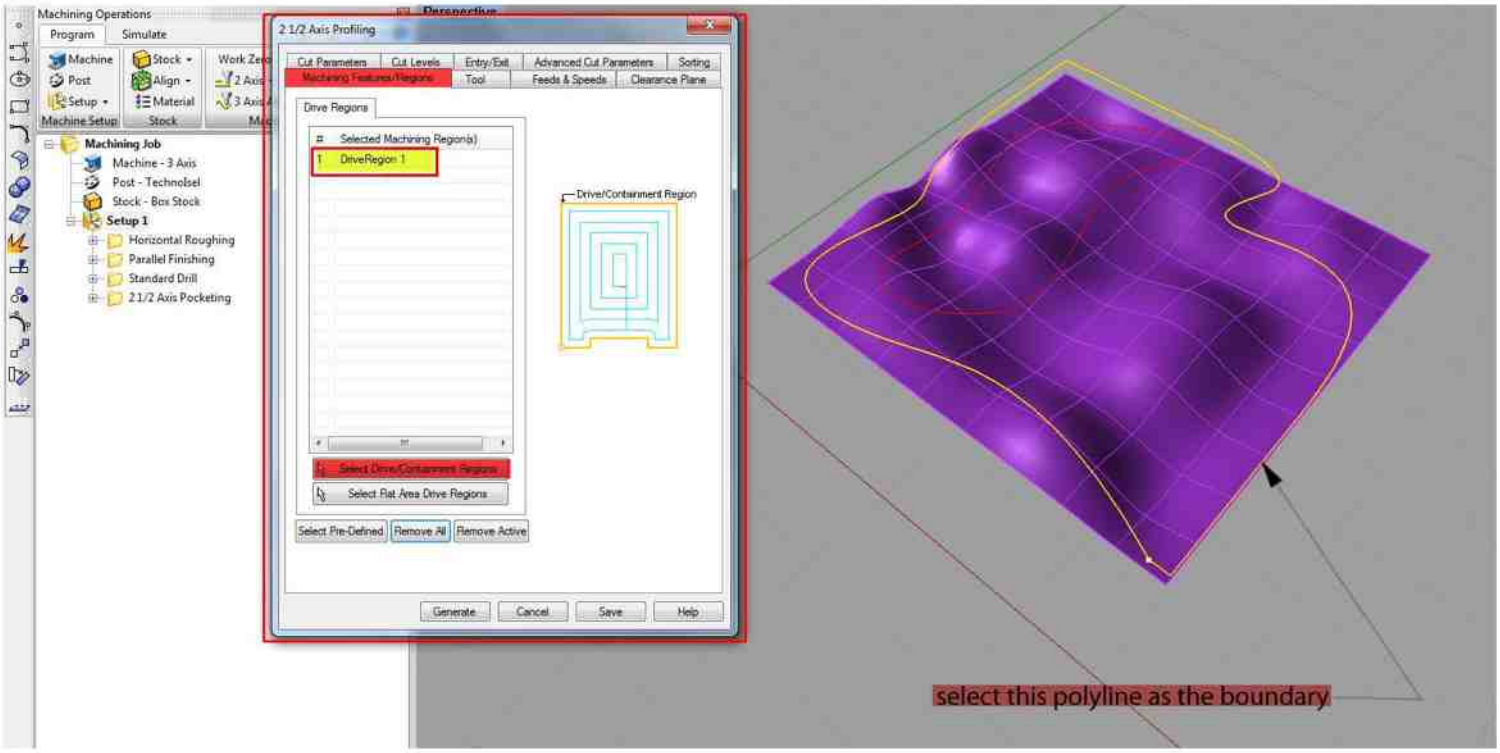


# STEP 10 RHINOCAM: CREATING PROFILING OPERATIONS

A) Click on the button for 2-axis operations and select "Profiling" from the drop down. A window will pop up that has all of the cut parameters and settings.

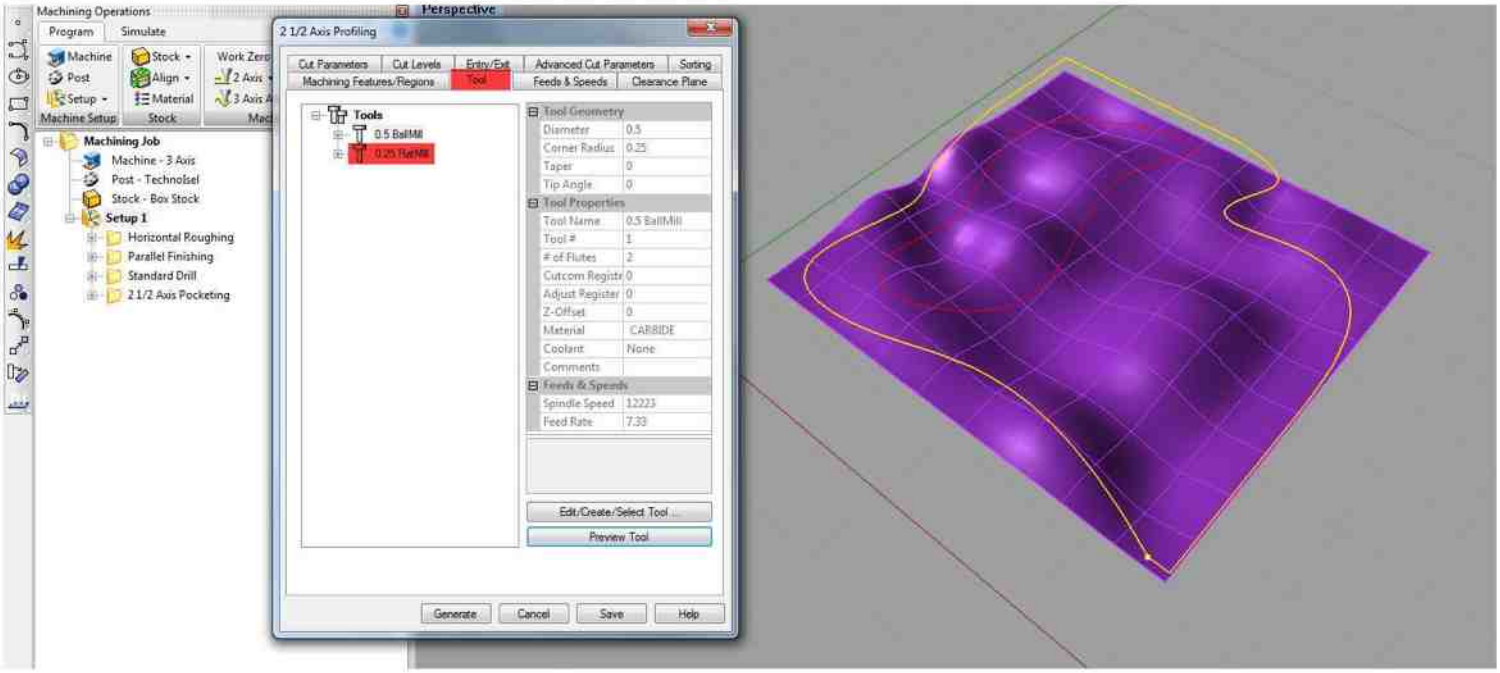



B) The same dialog box will appear with a series of similar option tabs. Under the "Maching Features/Regions" tab as before click the tab to "Select Drive/Containment Regions". Select the polyline you created to cut out your geometry. Again this line should be located at the top surface of your stock with a Z value of .75"

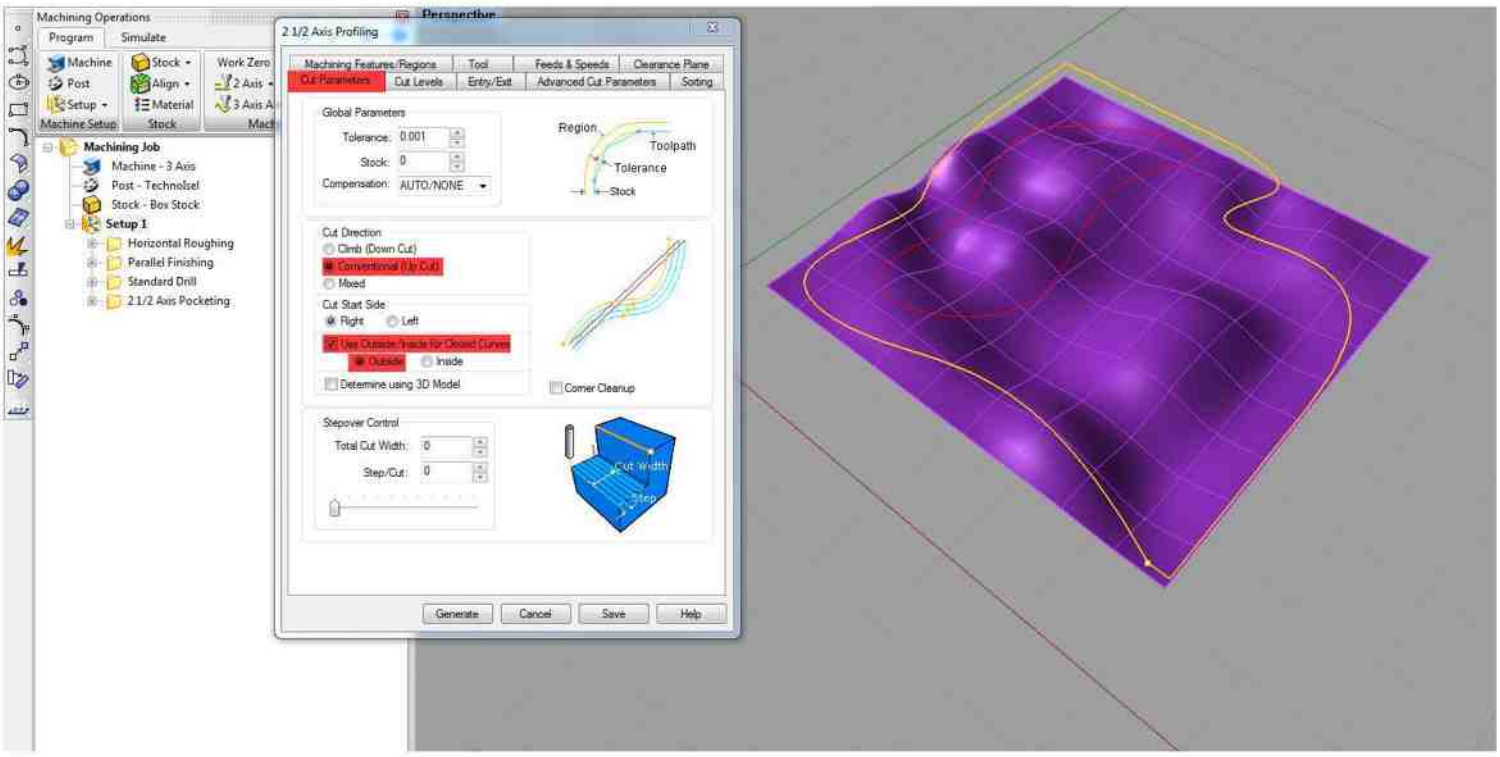


# STEP 10 RHINOCAM: CREATING PROFILING OPERATIONS

C) Click on the "Tool" tab. Select the endmill "0.25 FlatMill" that was created earlier

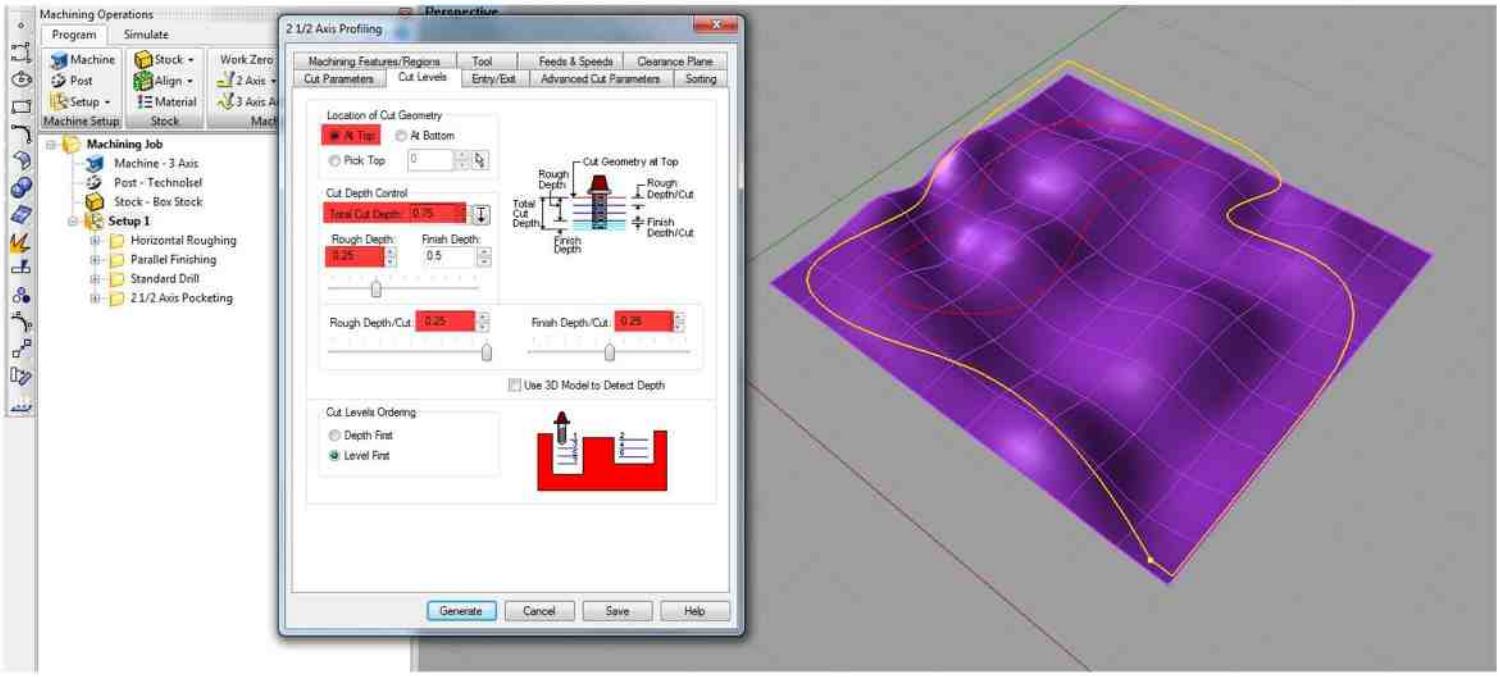



D) The "Feeds & Speeds" and "Clearance Plane" options should be the same as before so we can skip them. Next click on the "Cut Parameters" tab. Under "Cut Direction," select "Conventional (Up-Cut)." Select "Use Outside/Inside for Closed Curves" and choose "Outside"

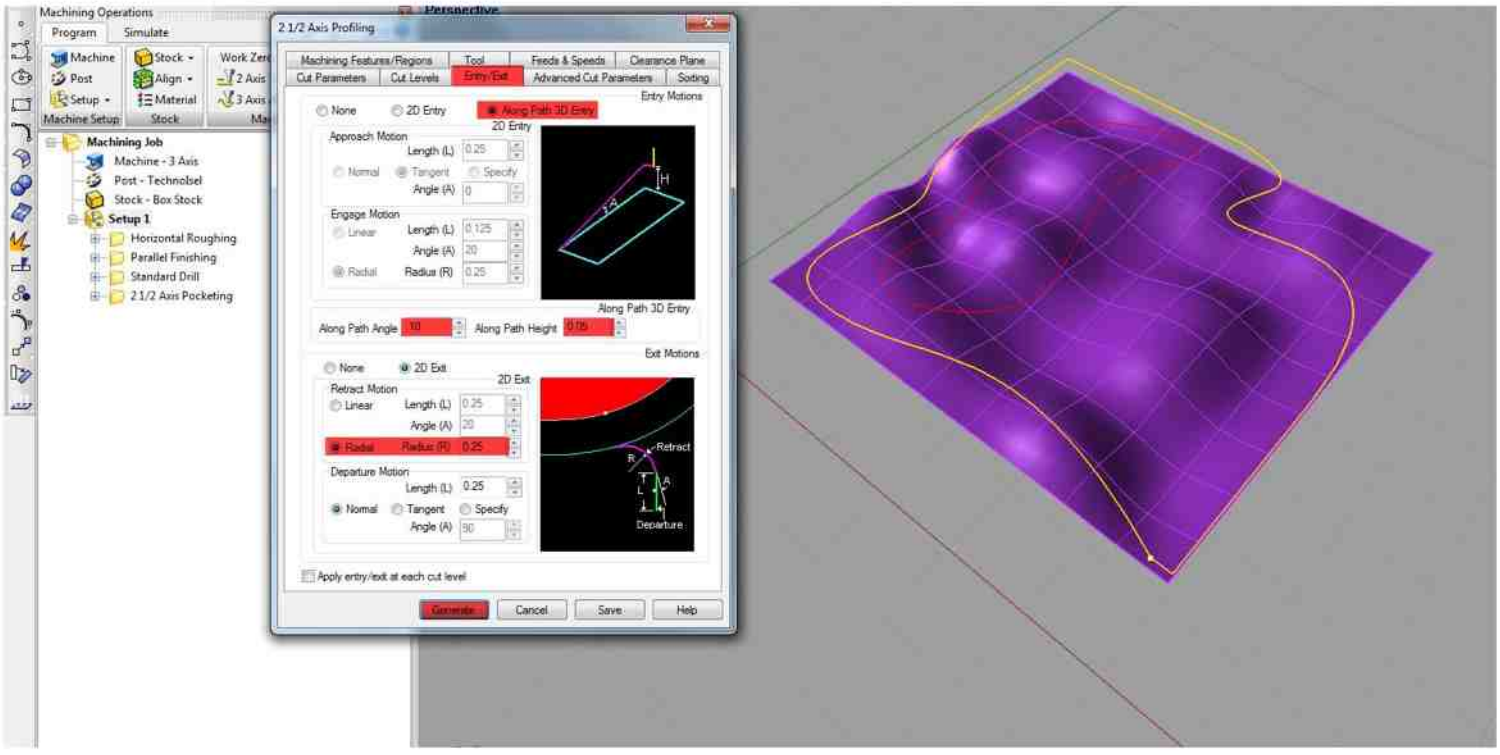


# STEP 10 RHINOCAM: CREATING PROFILING OPERATIONS

E) Click on the "Cut Levels" tab. Make sure "At Top" is selected. Under "Cut Depth Control," set to the depth of your stock material or .75". Under "Rough Depth/Cut," set equal to the diameter of the endmill (0.25"). This is the same as "max. stepdown." Set the "Cut Levels Ordering" to "Level First". PLEASE NOTE THAT CUT AND DRILL DEPTHS ARE A CRITICAL SETTING AND SHOULD ALWAYS BE DOUBLE CHECKED BEFORE YOU GENERATE YOUR CUT PATH. NEVER CUT TO DEEP OR THROUGH THE SPOIL BOARD!

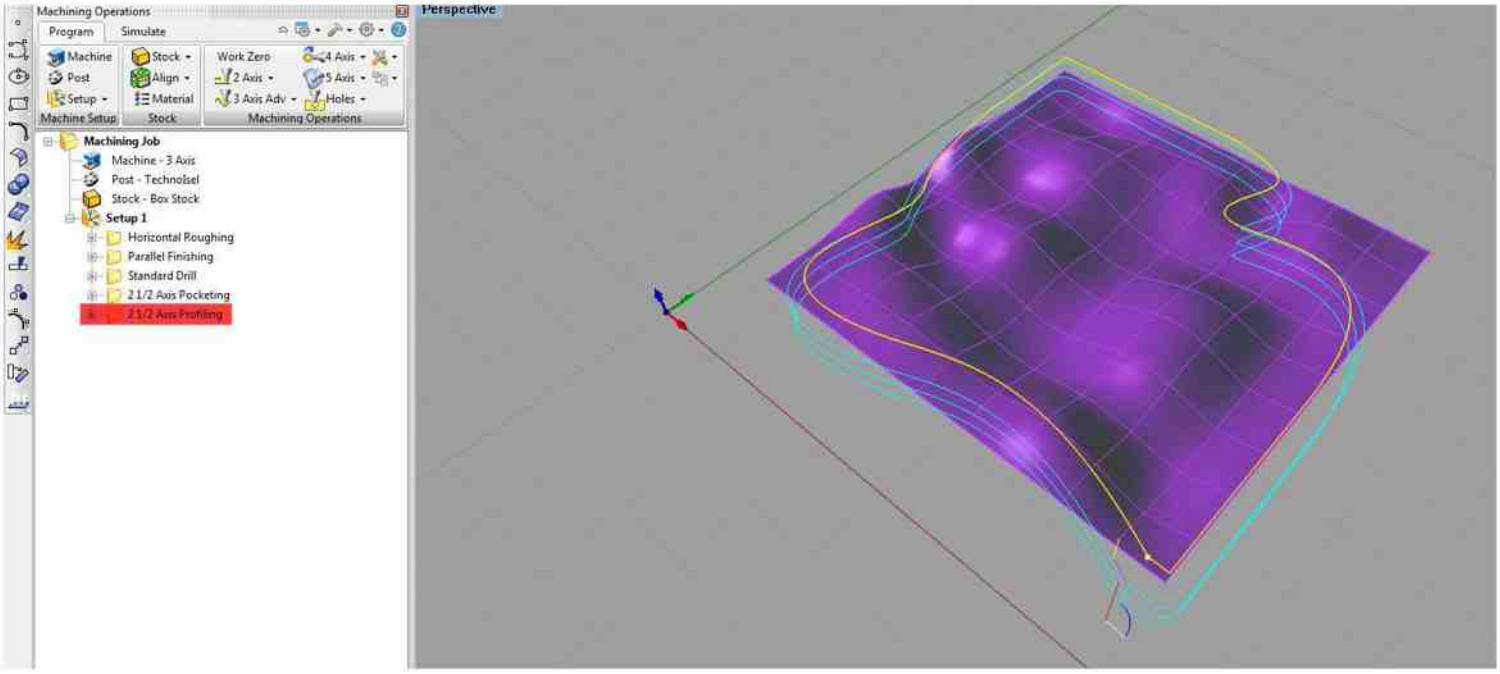



F) Click the "Entry/Exit" tab. These values should be a default based on the parameters of the Techno machine but it's best to double check to be sure. Make sure the "Entry Motion" is set to "Along Path 3D Entry" with an angle of 10 degrees and a height of .05". Make sure the "Exit Motion" is set to "Radial" with a radius of .25". Once all these options are set click on the "Generate" tab at the bottom.

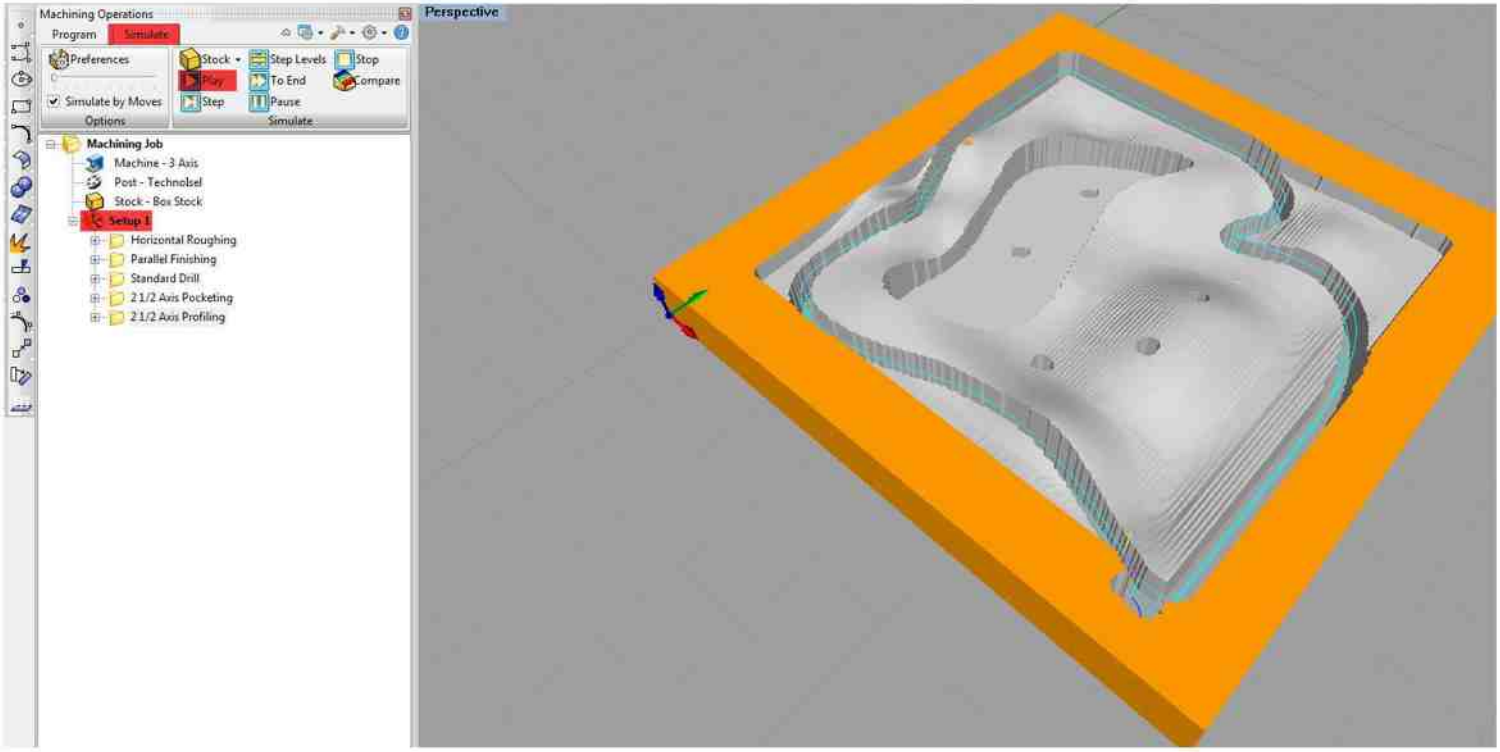


# STEP 10 RHINOCAM: CREATING PROFILING OPERATIONS

G) You should now see an overlay of the new profiling tool paths over your geometry in Rhino. This is a preview of the way the router will cut your profile. Blue lines indicate paths of travel through the material. You will also see the tool path you created under the Mops Browser to the left. If you need to make a change to your tool path, double click the folder to reveal the different options you set and double click them to edit settings.

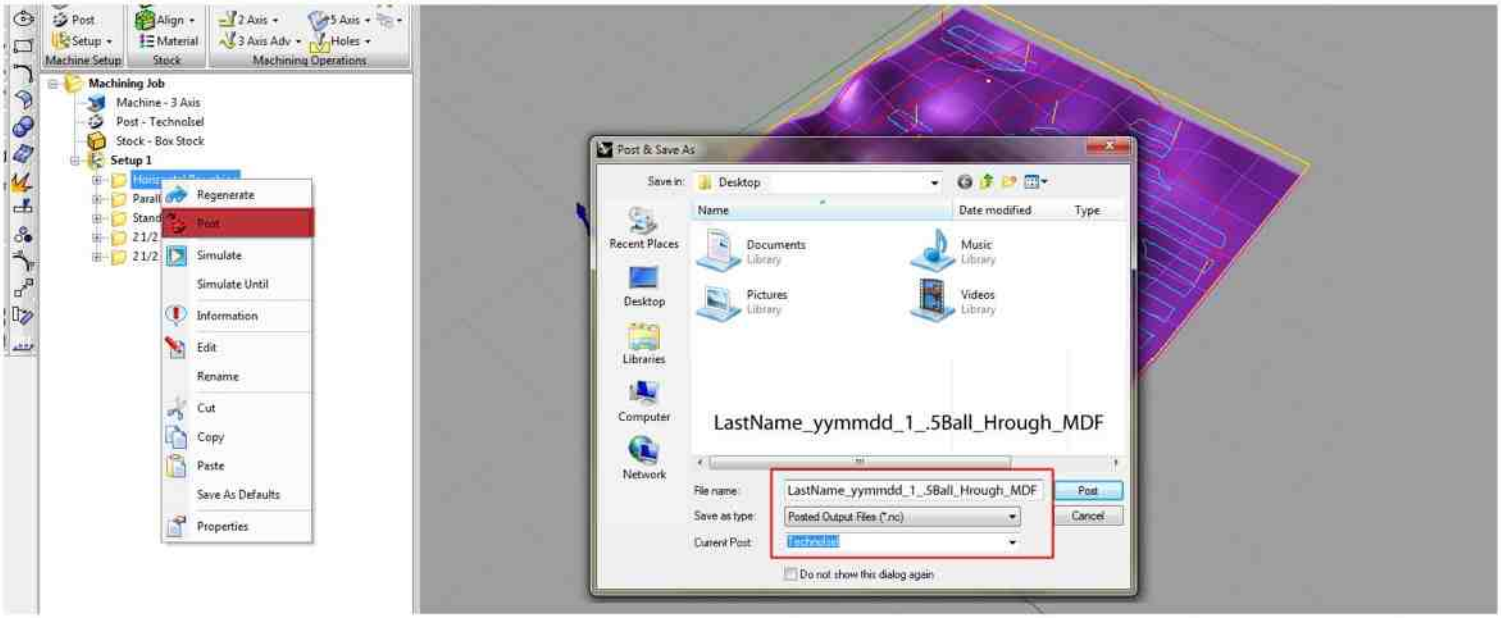



H) Highlight the tool path folder "Setup 1" by single clicking on the folder. Click on the "Simulate" tab to preview your tool path. On the left, drag the slider to the far left to slow down the speed of the simulation. Click on the "Play" button to start the simulation. You should see the simulated bit first running your roughing pass then running along the blue tool paths of your parallel finish pass and then drilling the holes and cutting the pocket operation and finally cutting the profile. You now have a completed set of operations that that will produce a finished piece.

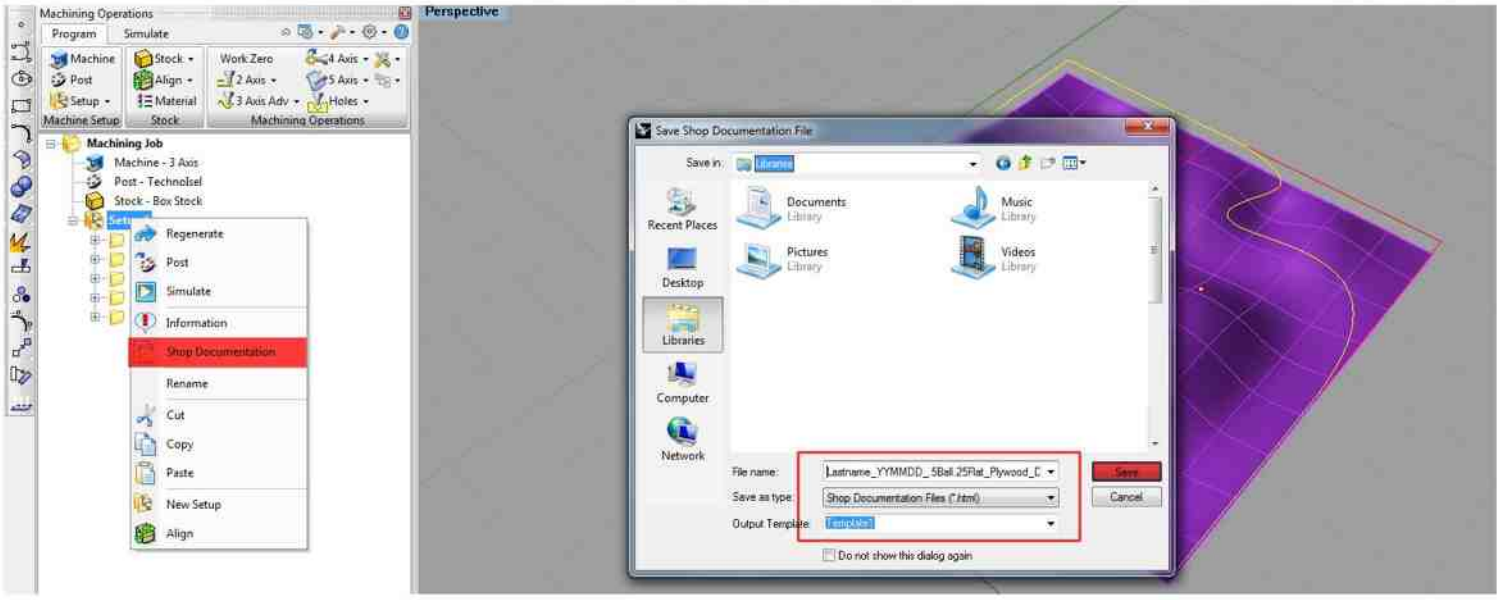


# STEP 11 RHINOCAM: POSTING(EXPORTING) FILES AND SHOP DOCUMENTATION

A) Make sure you **DOUBLE CHECK** all of your settings for each operation especially the **FEED RATES, SPINDLE SPEEDS, CUT DEPTHS and STEP OVER.** Next it is important to post each individual milling operation one by one rather than the whole MOPset. Right Click the first milling operation, in this case it is the "Horizontal Roughing" and choose "Post" from the drop down menu. This will convert your operations in RhinoCAM to a file with a series of text based instructions for the Techno CNC machine. This is called an NC file. In the "Posting" dialog box, make sure you have the file type set as NC and that the Current Post is set to Technosel. Finally make sure you name the file as indicated with your *Last Name* followed by the *6 character date (YYMMDD)* followed by the order of the cut *(first=1, second=2, etc..)* bit you are using *(.5Ball)* followed by the type of milling operation *(HRough)* followed finally by the material you are using *(MDF)*. Once you have set these options and correctly named the file, click on the "Post" tab. This will create your NC file. Repeat this process for each milling operation and bring all of your NC files in a single folder to the CNC shop.. The files should look something like this: LastName_yymmdd_1_.5Ball_Hrough_MDF




B) Right Click the "Setup 1" folder and choose "Shop Documentation" from the drop down menu. This creates a description of your milling operations including information on the tools, the speeds, the cut time, etc. In the "Save Shop Documentation File" dialog box, make sure you have the file type set as .html and as with your NC file. This will help to keep files organized. Once you have set these options click on the "Save" tab. This will create your Shop Documentation file. You must print this file and bring it to the CNC shop. When you come to schedule your cut, bring your **Student ID** as well as **NC file** and your **Rhino file** on a flash drive. Be sure to print your **"Shop Documentation"** and fill out a "**Cutting Checklist"** that can be obtained from the CNC Shop. Failure to bring these five items will result in lost cut time.




**When you come to schedule your cut, bring your Student ID as well as your NC file/s and your Rhino file on a flash drive. Be sure to print your "Shop Documentation" and fill out a "Cutting Checklist" that can be obtained from the CNC Shop. Failure to bring these five items will result in lost cut time.**