

OVERVIEW

RhinoCAM, is a computer-aided manufacturing (CAM) software plugin for Rhinoceros 3D, a popular 3D modeling software. Rhinocam integrates with Rhino and adds CAM functionality, allowing users to generate toolpaths and control CNC machines for various manufacturing processes.

With Rhinocam, you can perform tasks such as 2D and 3D machining, engraving, mold making, and more. It provides a range of features and options for toolpath generation, including support for different cutting strategies, tool libraries, simulation, and post-processing capabilities. Rhinocam is widely used in industries such as aerospace, automotive, jewelry, and prototyping for converting 3D designs into physical parts.

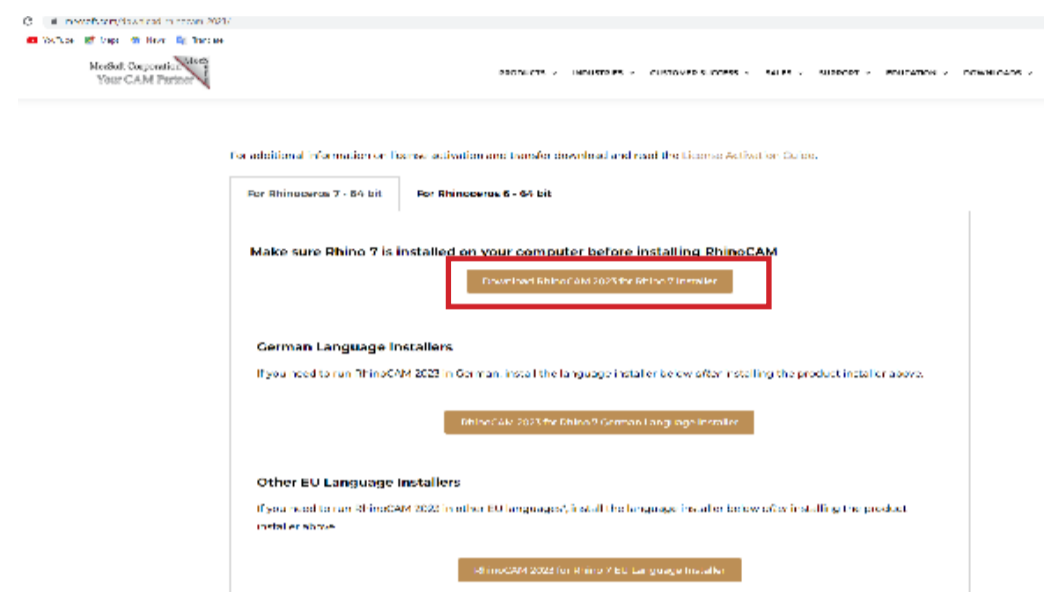
Please note that my knowledge cutoff is in September 2021, so there might have been updates or newer versions of Rhinocam since then.

1. INTRODUCTION

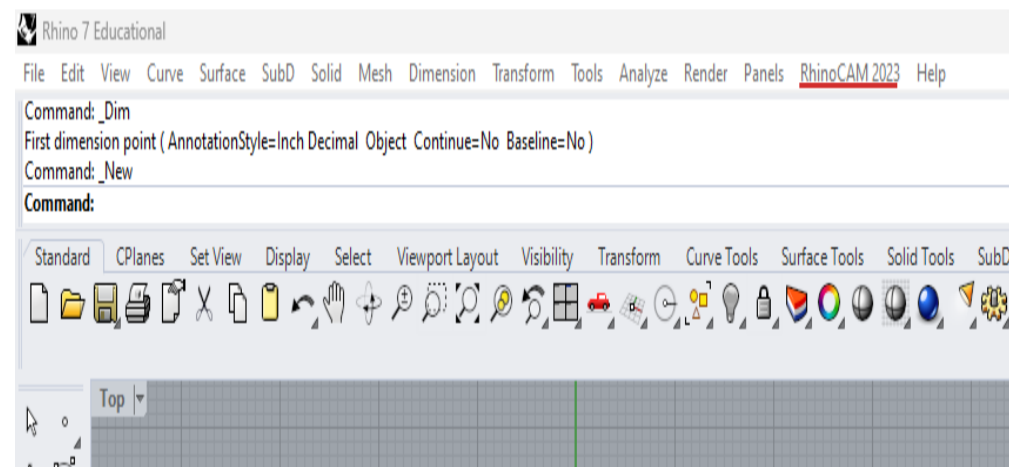
A. Software Setup

a. RHINOCAM DOWNLOAD

- Ensure you have Rhino installed on your computer.
- Download RhinoCam from the following link: <https://mecsoft.com/download-rhinocam-2023/>



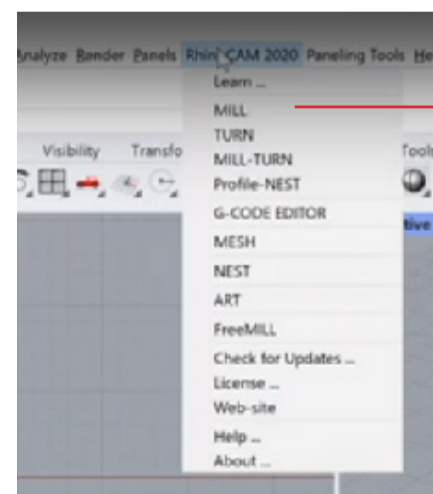
- Once it is installed, open your Rhino software.
- On the menu bar of Rhino, you should see the RhinoCam option. If you don't see it, it may not have been installed correctly. Alternatively, check and set your units to inches.



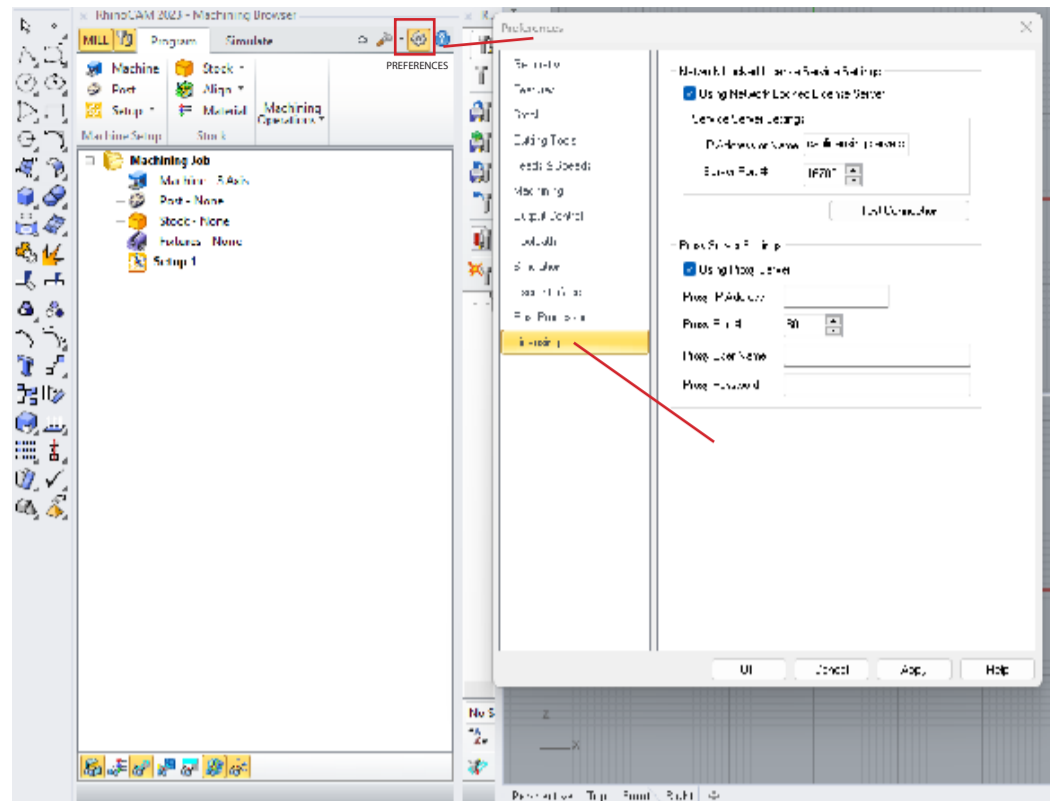
B. LICENSE SETUP

- If you are a Ball State student and not connected to the Ball State network, you need to connect to the network to access the license server and obtain a license.

- To input your license, go to the top menu and click on RhinoCam 2023. You will see multiple options; choose the Mill option (the tool you want to work with).



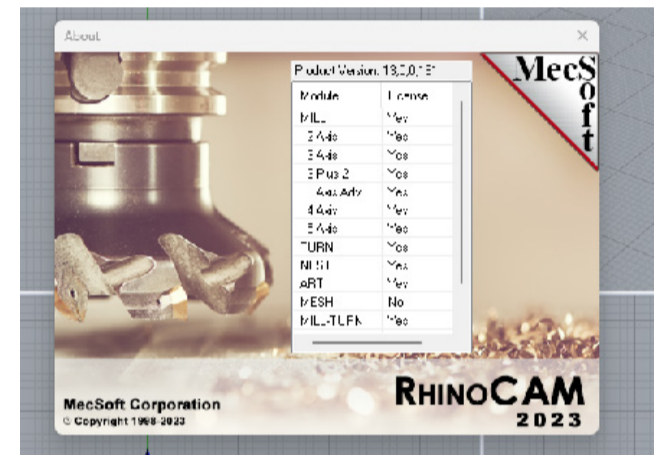
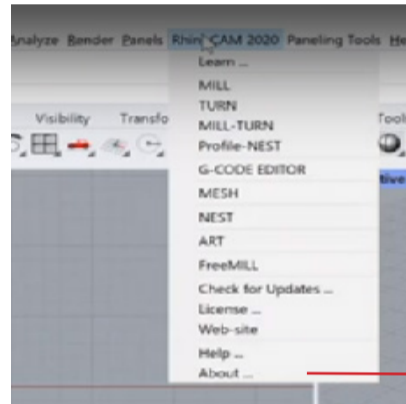
- After clicking on Mill, a new tab will open. Click on the gear icon located at the top of the tab, which is named "Preferences." This will bring up the preferences dialog. Go to the Licensing section and fill in the license information. Click OK when you are done.



- Alternatively, you can go to the RhinoCam 2023 menu at the top of Rhino and select the license option. A license dialog will open, and you need to fill in the text boxes. Then click Register, and it will connect to the license server and fetch one of the licenses. A dialog box will appear, indicating that it is being activated.

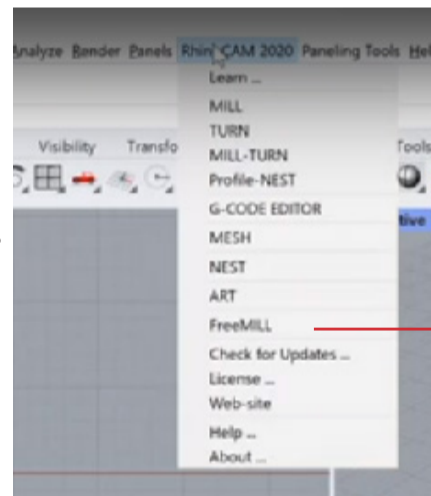
- To check if you have licenses for the tool you want to use, go to the "RhinoCam 2023" menu on the side and click on "About." It will display "Yes" for the tools that have a license.

- Please note that the license information does not persist once Rhino is closed. You will need to enter the license information again before using RhinoCam.



C. RHINOCAM DEMO

- To start with RhinoCam DEMO, run free mill under RhinoCam 2023

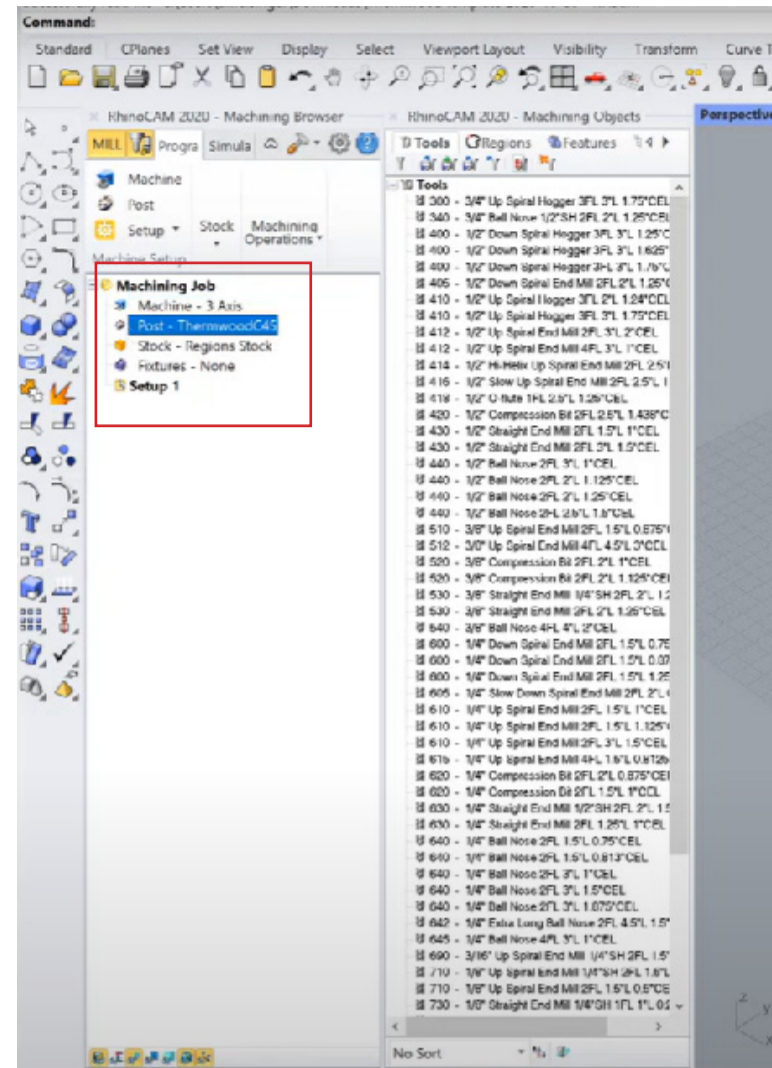


E. MACHINE SETUP

This is where the details of your specific milling job will be set up.

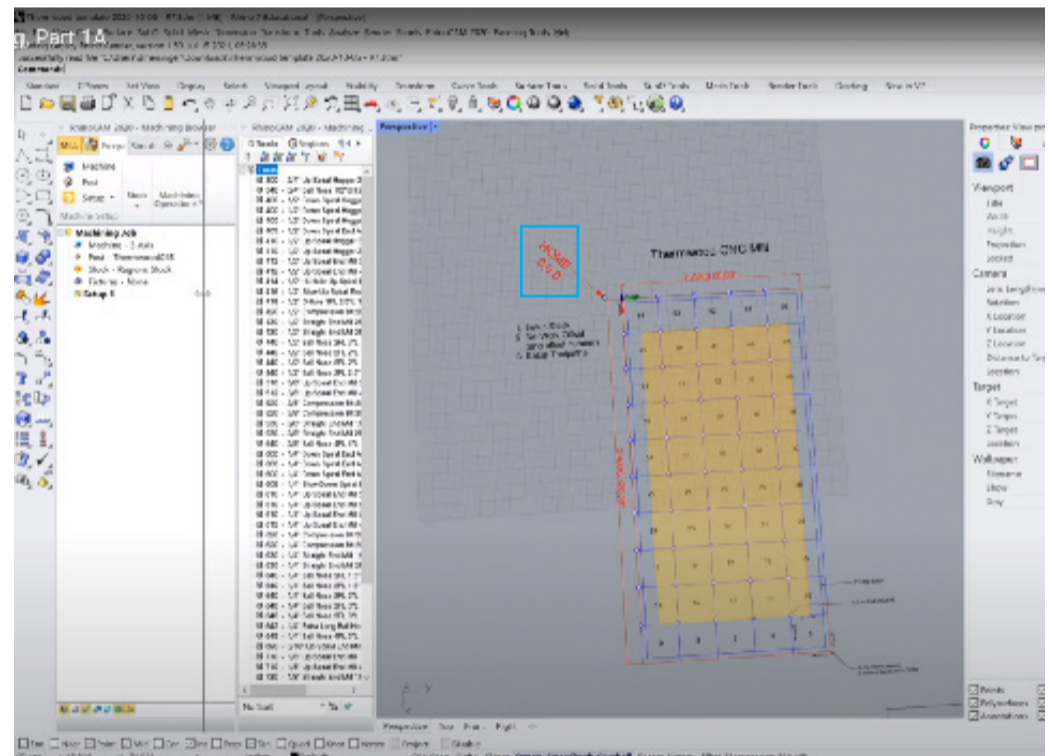
Most times it is located at the corner left of your Rhino panel.

- You will always need to use the 3-axis unless you are using a robot.
- Stock refers to the raw material you are using for the job.
- Fixtures – to hold down material for milling, it could be clamp or anything to hold down the stock for milling.
- Setup - everything that you Set up for your job is going to be added under this setup section.
- The things that get added all come from the machining operations drop down at the top which has different types of tool paths. Also, they are divided up by the number of axes involved.
- The two axes toolpath – are for jobs that only use 2D geometry for setup while the three axes' toolpaths require 3D geometry for setup.



F. GRID

- Notice that there are points at the upper left corner of each grids.
- Be careful when rotating your CNC mill template on your Rhino, the numbers actually flip direction such that the rhino appears to be facing upright direction.
- To avoid confusion, make sure you rotate the bed such that the Home text close to bed is at the upper left corner, and the controller box is at the end of the mill.
- Each of the boxes on the grid represents a fixture offset or a work offset; So, after the stock material has been set up, the next is to set up the work offset, and then set up the tool path.



2. BASIC EXAMPLE

STEP 1: SETUP STOCK

- A simple way is to draw a rectangle in Rhino and place it at your desired grid, let's say grid 23. The grid number refers to the point at the edges of the grid boxes that is closest to "home."

- For grid 23, the 0 0 location for the grid cell is located at the upper left of the 23 grid box.

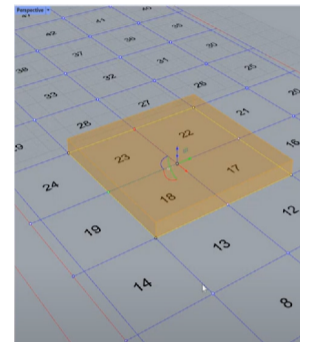
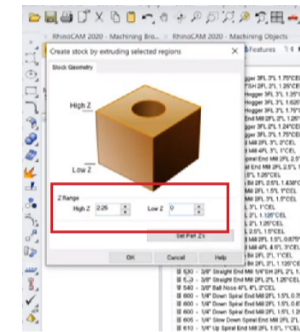
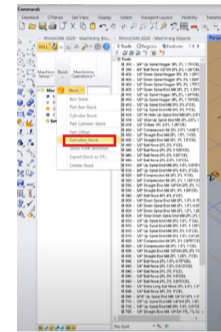
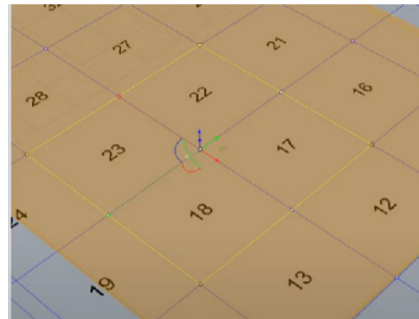
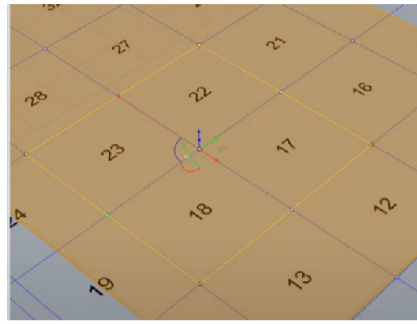
- Next, draw a box starting from the 0 0 grid cell at any distance, let's say a two-foot square. This two-foot square is located at grid cell 23.

- There are different ways to do this:

a. The simplest way is to go to the Rhino tool panel, draw a rectangle starting from grid 23 at a distance of a two-foot square,

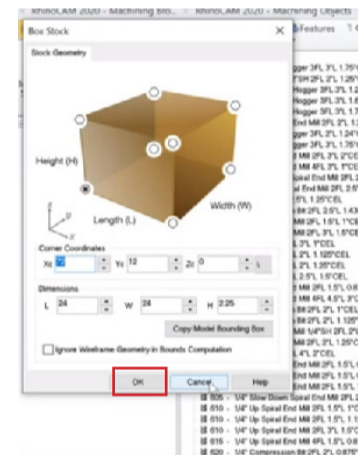
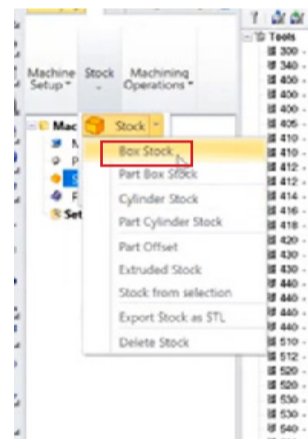
- then go to the RhinoCam panel and navigate to the stock menu at the top. Click the drop-down menu of the stock option and select "extruded stock."

- In the tab that appears, enter the desired thickness (e.g., two and a quarter inches) for the extrusion. Ensure the bottom of the material (low z) is set to zero before clicking OK. After clicking OK, you will see an extruded stock geometry that resembles a box drawn in Rhino.



b. Another method is to go to the RhinoCam panel, select the stock menu, and choose the "box stock" option from the drop-down menu.

- This method allows you to define the box by specifying its dimensions (length, width, and height).

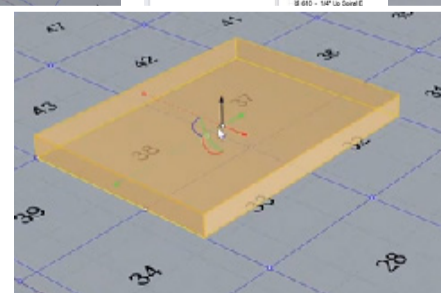
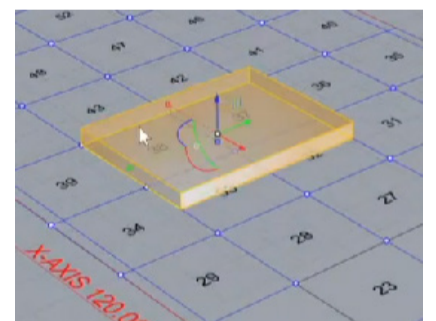
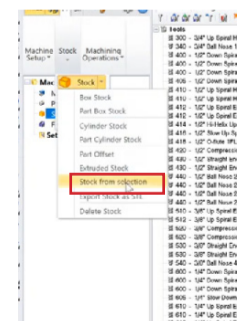
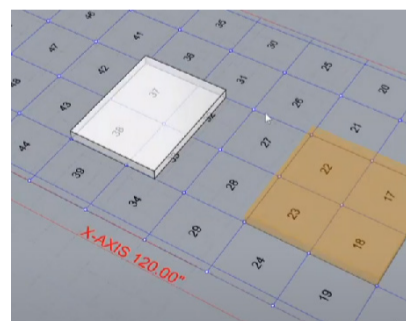
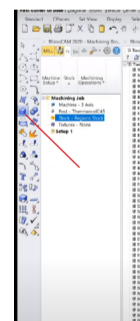


c. An alternative method related to option "a" is to use the box tool in Rhino, which provides 3D geometry.

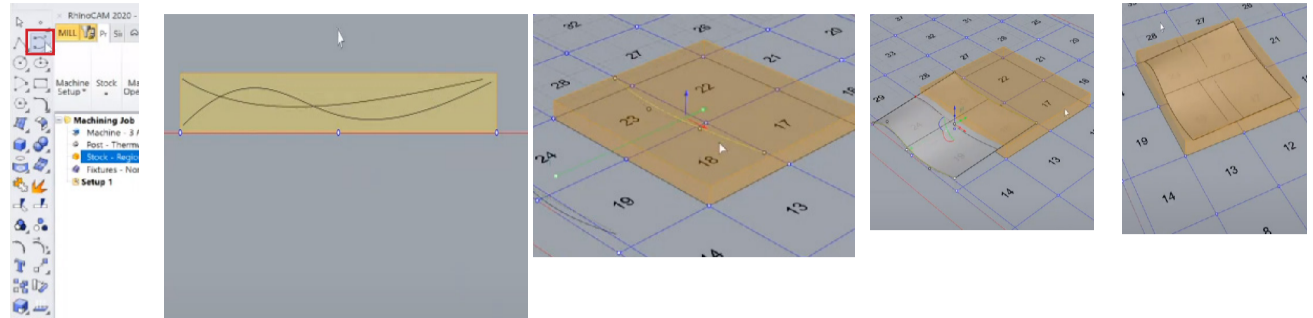
- Place the box at the desired grid position with the desired dimensions.

- After creating the Rhino geometry, select the box, go to the stock menu at the top of RhinoCam panel, click the drop-down menu of the stock option, and choose "stock from selection."

- This will convert the Rhino object into RhinoCam stock material. To avoid confusion, hide the Rhino geometry and leave only the RhinoCam geometry on the grid. This method (c) is useful for stock geometries with non-standard shapes.



- After extruding the stock material, you can create a loft surface that is not flat within the extruded box; To do this, create two or multiple curves at different distances from each other, select the curves, and use the "loft" command. This will generate a non-flat surface.



STEP 2: SET WORK OFFSET

- Once the stock is set up and the part geometry (the non-flat surface) is within the stock, the next step is to establish the work offset.

- The work offset is used to specify the reference points that Rhinocam will use to locate this geometry on the table.

- One of the grid boxes or locations must be selected as the reference point.

- You can choose any grid box on the table as the reference point or select the grid box where your stock is located.

- To set up the reference point, go to the Machining Operations and select "work zero" (work zero is another term for work offset).

- There are two things you need to do. First, select the point on the grid that you will use as the reference point.

- To do that, click the small arrow icon in front of the z input, as illustrated in the image. This will allow you to choose your reference point on the Rhinocam grid.

- If your Osnap is not enabled, make sure to turn it on and snap to the grid point that defines grid 23 (remember, that is where we placed our stock).

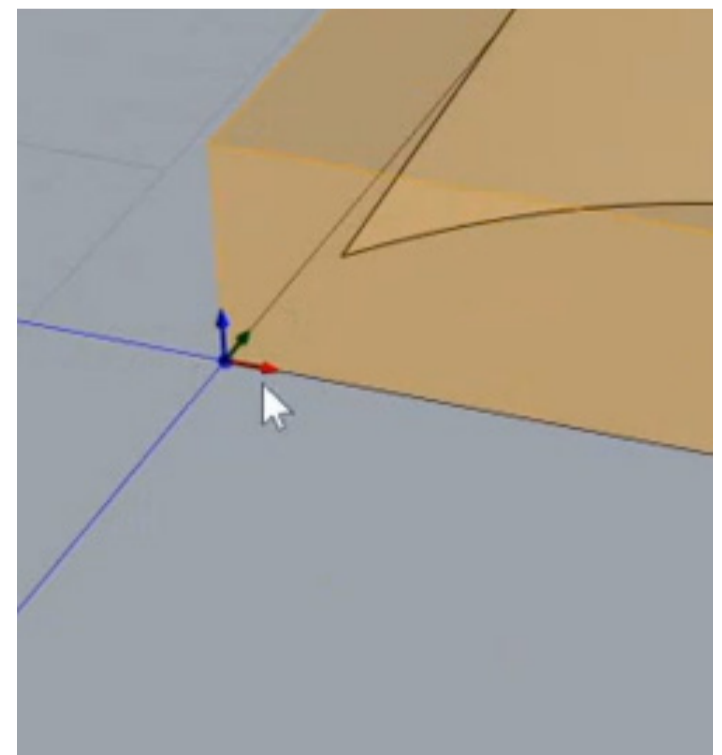
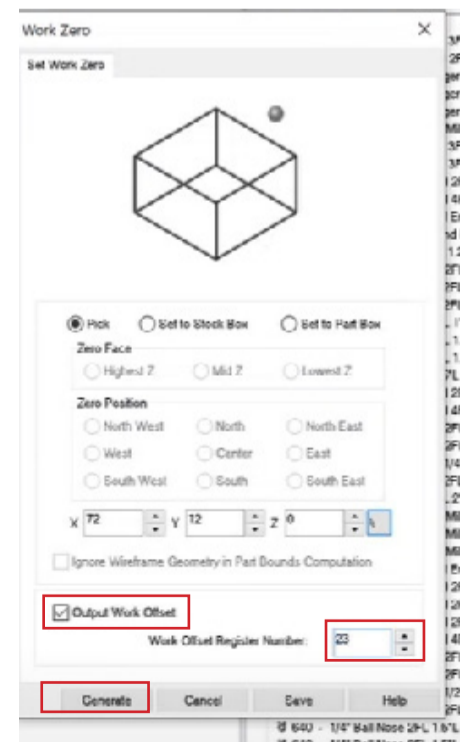
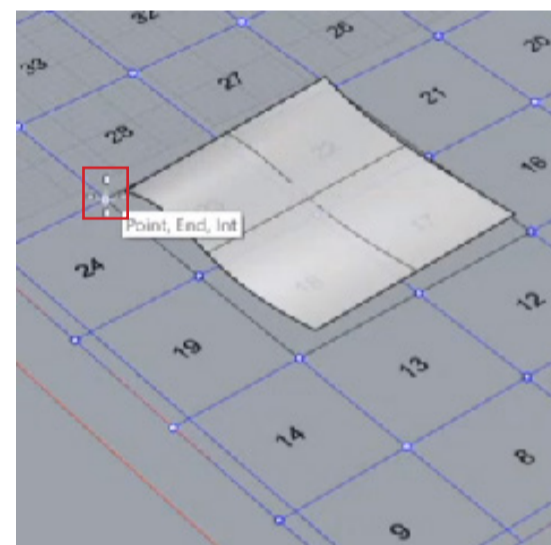
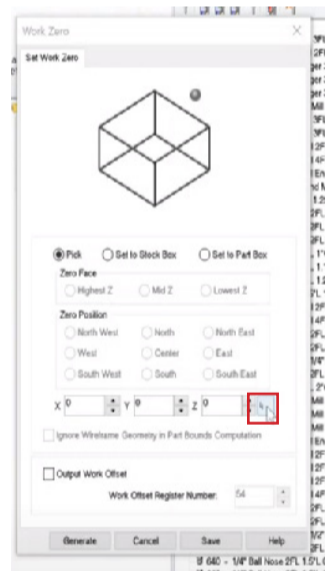
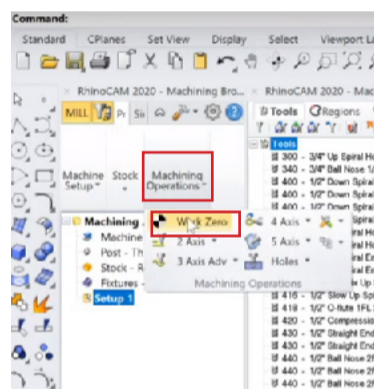
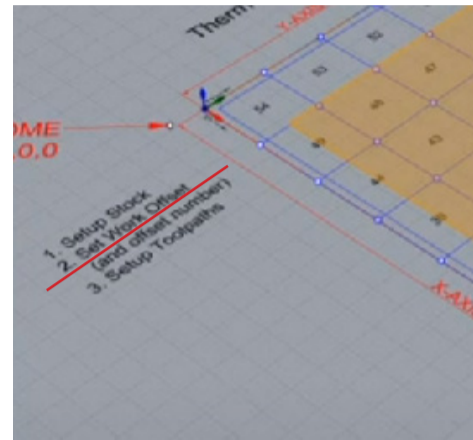
- Also, remember that for grid 23, the 0 0 location (closest to home) for the grid cell is located at the upper left of the 23 grid box.

- As you select the point, you will see the reference coordinates loaded into the box for that point.

- The next step is to output this work offset to 23 by checking the box. This means you need to enter 23 to tell the controller which number to use. The controller knows what grid 23 is in terms of the actual physical dimensions in relation to its home, which will be different from the XY coordinates we have in Rhino.

- By instructing RhinoCam to use grid 23 with the zero position, we are telling the controller to use that same point as 0 0.

- Once you have finished outputting the work offset, click "generate." You will also notice a small gumball at your reference point 23, indicating that this is now the new 0 0 position.



STEP 3: SETUP TOOLPATHS

- Once you have set up your work zero (work offset), it is time to set up your tool paths to instruct Rhinocam in the machine on what we want to do.

- Go to the machining operations. Since we are working with 3D geometry, click on the "3 Axis Toolpaths" dropdown. You will see multiple roughing options, finishing options, and other specific types of machining.

- For this example, our first toolpath is the "Horizontal roughing", then the parallel finishing and other toolpath we might consider, which will create a simple kind of topography mill.

A. HORIZONTAL ROUGHING

- Let's start with the horizontal roughing. Under the 3-Axis, click on the "Horizontal roughing", and a new window will open with various options and tabs, including tabs within tabs.

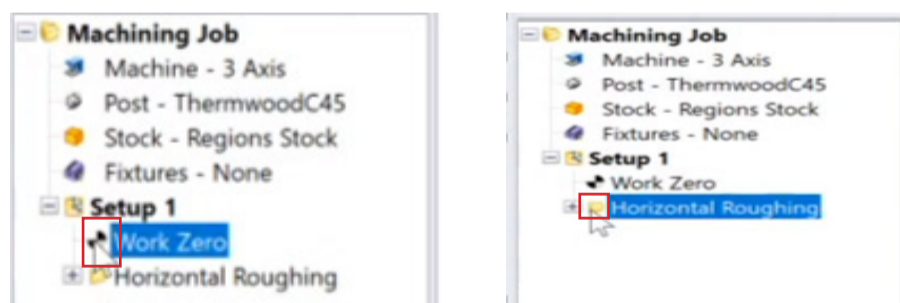
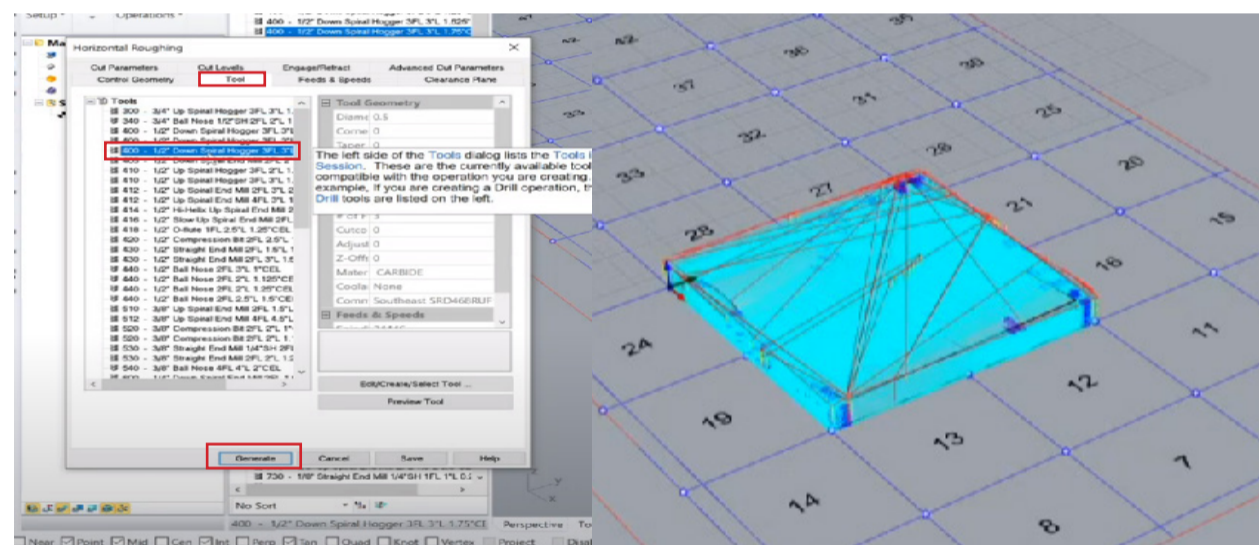
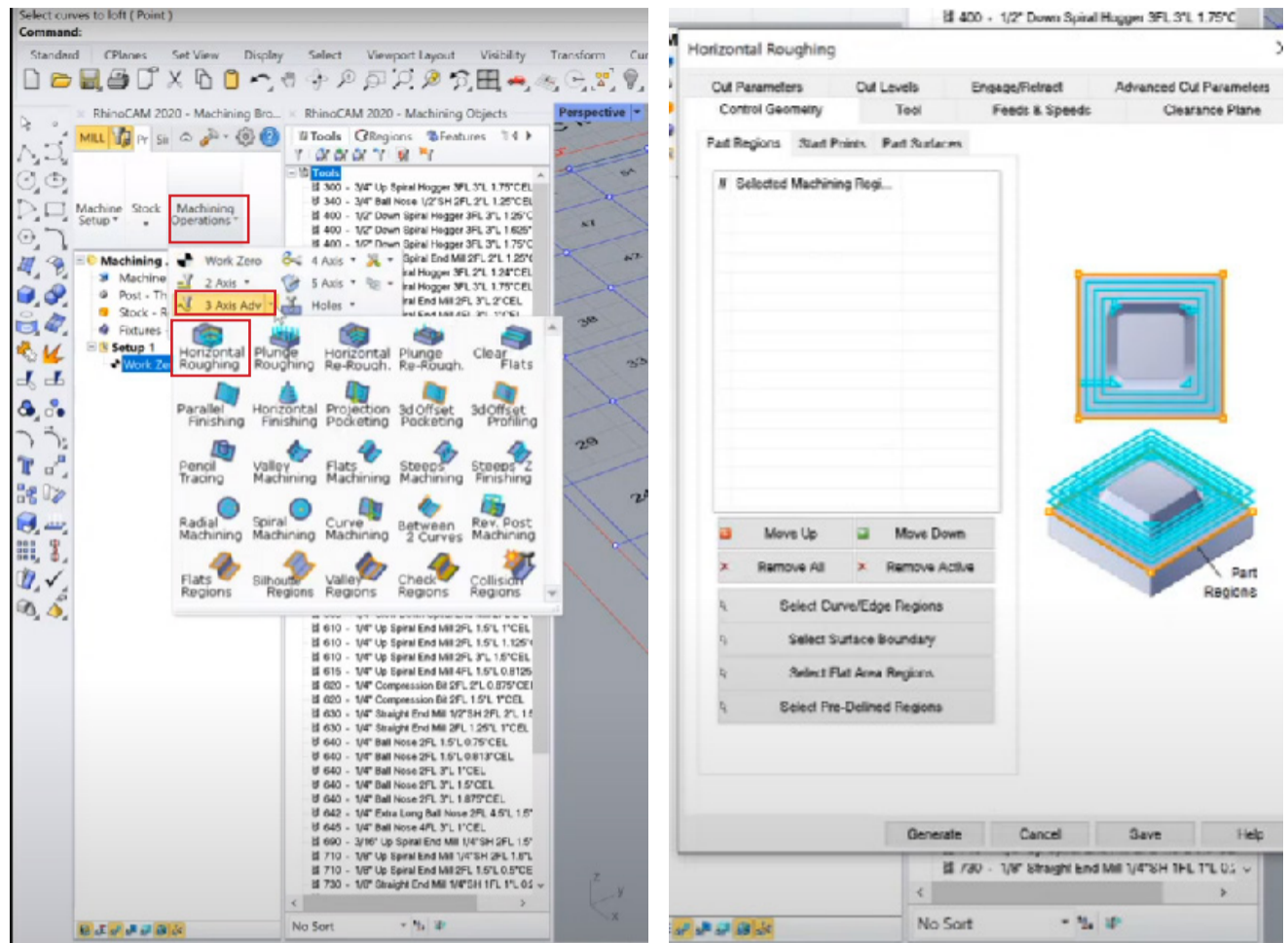
- For every tool path, there will always be control geometry that defines what we are milling.

- For horizontal roughing and all other 3D roughing toolpaths, you do not need to specify control geometry.

- The next step is to select a tool. Click on the tool path, and a tool tab will appear. Then, select a half-inch down spiral hogger (you can choose your desired tool, but for this example, we are using the half-inch down spiral hogger).

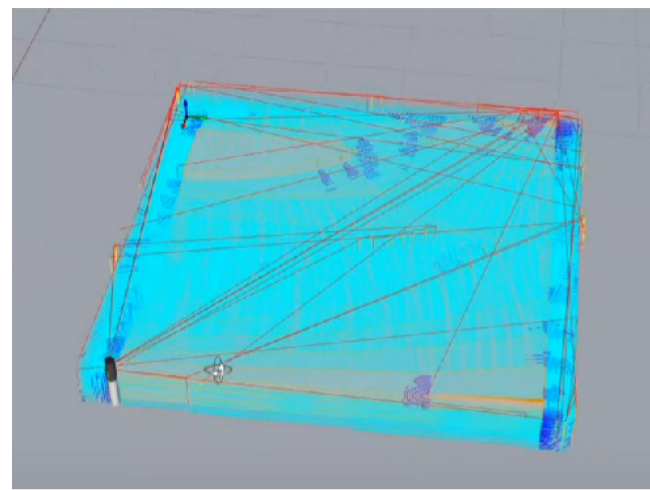
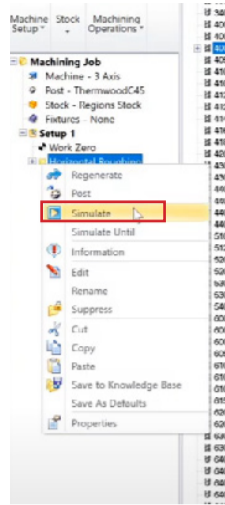
- After selecting the tool, click on the "Generate" tab. This will load and generate paths for the bit to follow and remove all the material that is not part of the surface.

- To hide the generated tool path, click on the small icon on the left side of the work zero. If you want to bring back the roughing tool path, click on the horizontal roughing tab.

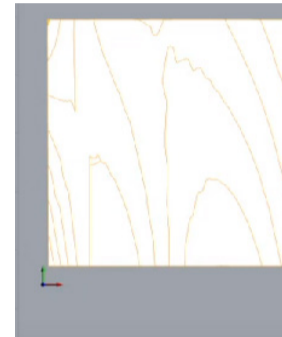
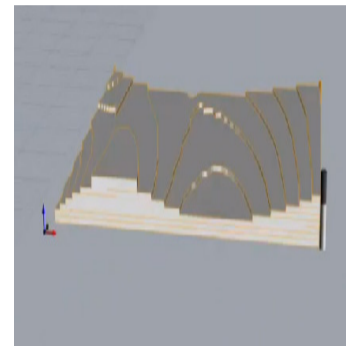
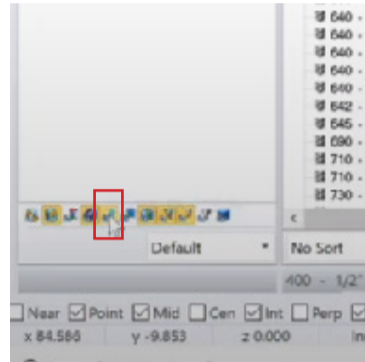


Simulation

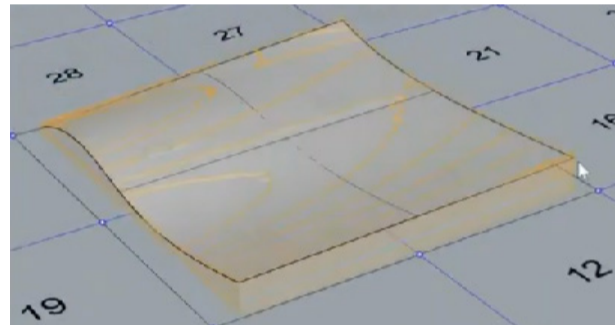
- To simulate this, right-click on the horizontal roughing tab and select "Simulate." You will notice that it starts removing the material.
- The result may appear obscure from the tool path, so you will need to turn the tool path off.



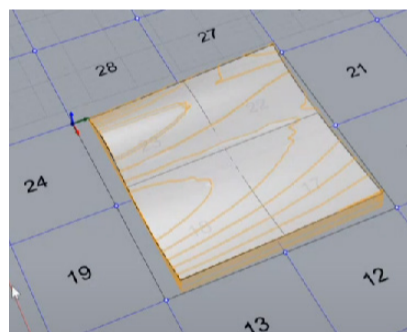
- The result may appear obscure from the tool path.
- To turn off the tool path, deactivate the tool path visibility at the lower left side of the Rhino interface.
- It seems that the result shows a kind of step-down effect.



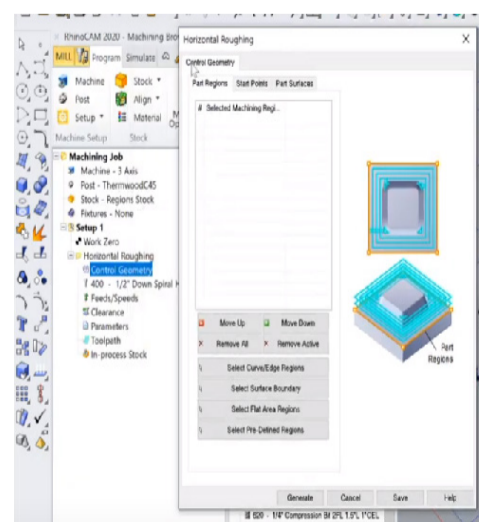
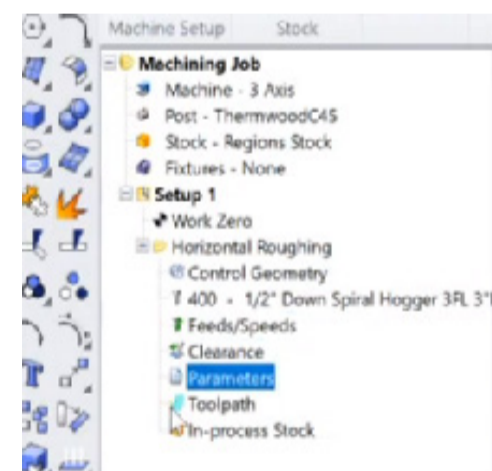
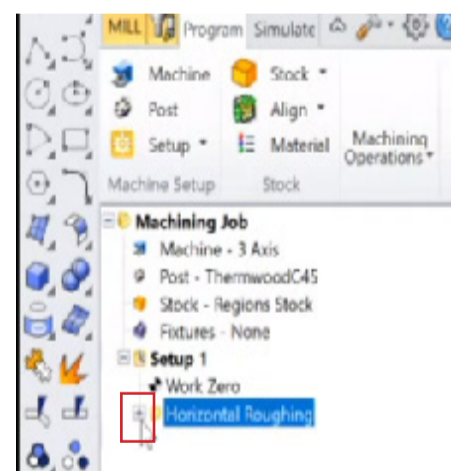
- Also, if you click on the program, it will show the result with the actual milling geometry, allowing you to compare the actual model result with the simulated geometry.
- Horizontal roughing is used to cut out the final geometry or cut out the object from the stock material.



- If you had taped or clamped the stock material to the bed, running the tool path will mill away all the tape around the edges of the stock. You need to rely on the seal between the final geometry or the final milled object and the bed, without the tape.



- There are various ways to limit what is milled out as part of this tool path.
- When you click on horizontal roughing, a new window with different features will open or you click the plus icon (check the image on your right) next to the horizontal roughing, it will show details about the job, primarily the tools that have been selected. This allows you to select any kind of tool path you need. For example, If you click on any of them, let's say the control geometry, it opens a dialog box specifically for the control geometry. It is much faster than opening the entire thing. This is a quick way to access the features.



Control geometry

- Let's focus on the control geometry, which directly impacts the output. There are several ways to select what will control what we are milling.

i. Click the plus icon (check the image on your right) next to the horizontal roughing, then click the control geometry.

- Since we are working with the surface here, we can use the "Select Surface Boundary" option in the control geometry dialog box (check the image on the right) and then select the surface of the model.

- Right-click or hit enter to complete the selection.

- You will notice that the surface has been extracted, and its boundary curves are listed in the control geometry tab. Clicking on them will highlight each one in a different color.

- Then click "Generate" and turn the tool paths back on by activating the tool's visibility at the lower corner of the Rhino interface.

- Now, instead of generating tool paths for the entire area where there is stock material, the tool paths are limited to where the control geometry, or the final surface, is located.

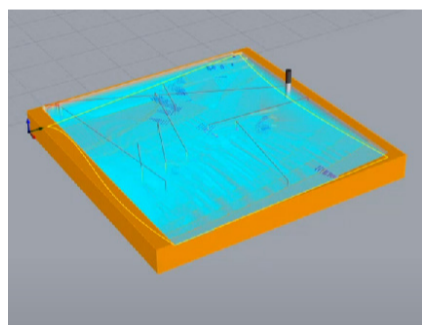
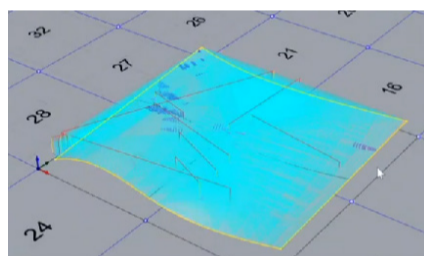
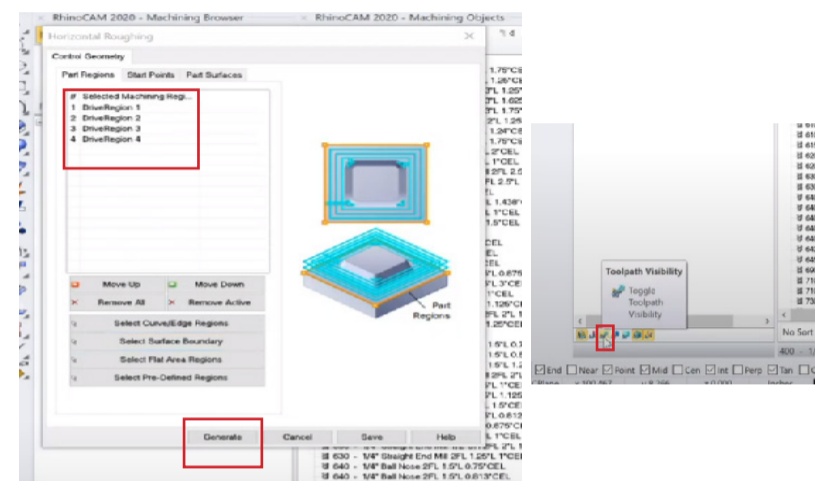
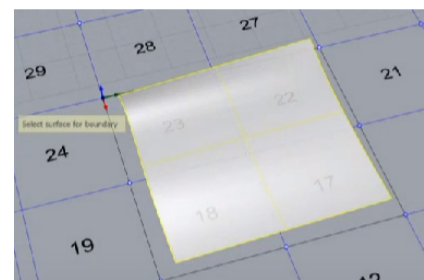
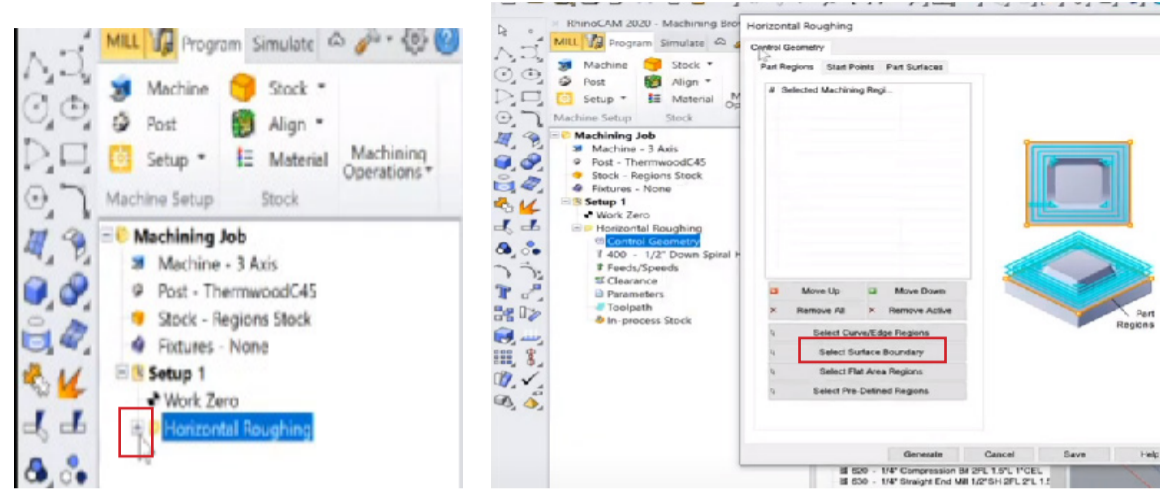
- Then simulate this, and you will notice that it leaves behind all the parts of the stock that are outside of that control geometry. If we take this approach and think about the stock material being taped down to the bed, the edges will not be affected by running the tool path in this way.

- We must consider a subsequent step to cut out the final surface from the remaining stock material.

- By restricting the cut geometry, we limit the amount of material that is removed.

- This can be useful because it takes a lot of time to remove all the stock geometry. If we have more stock geometry than model geometry, we can save a lot of time by taking this approach and cutting out the final geometry rather than removing all of the stock material.

- So, if our stock material were much larger, this would be a way to reduce the milling time required.



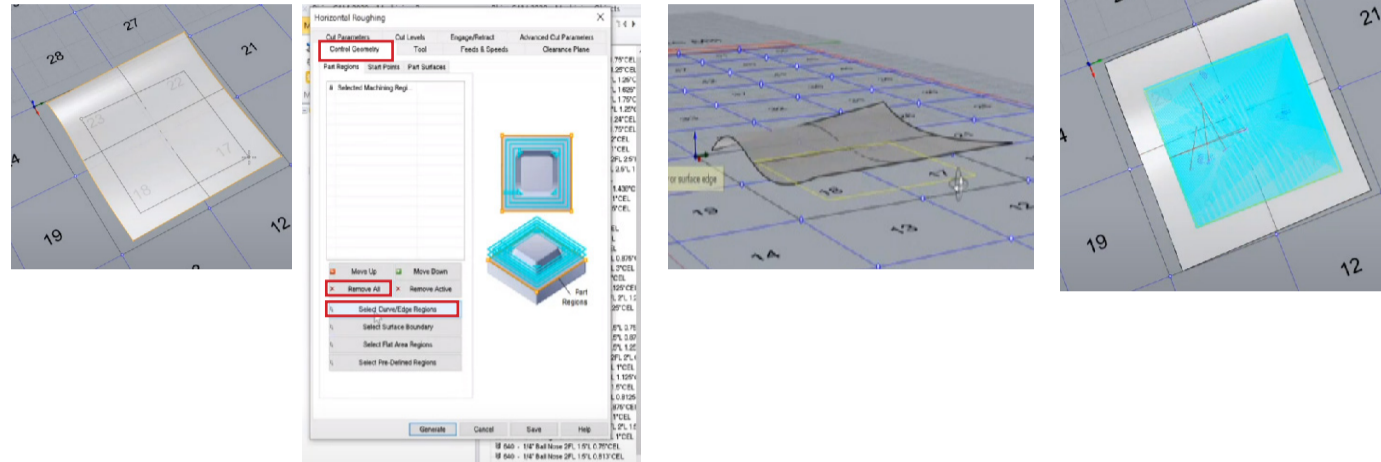
ii. Another way is to go to the horizontal roughing and click "select curve" under the control geometry section and select all the curved edge regions of your model.

- This would give you the same result as the first approach.

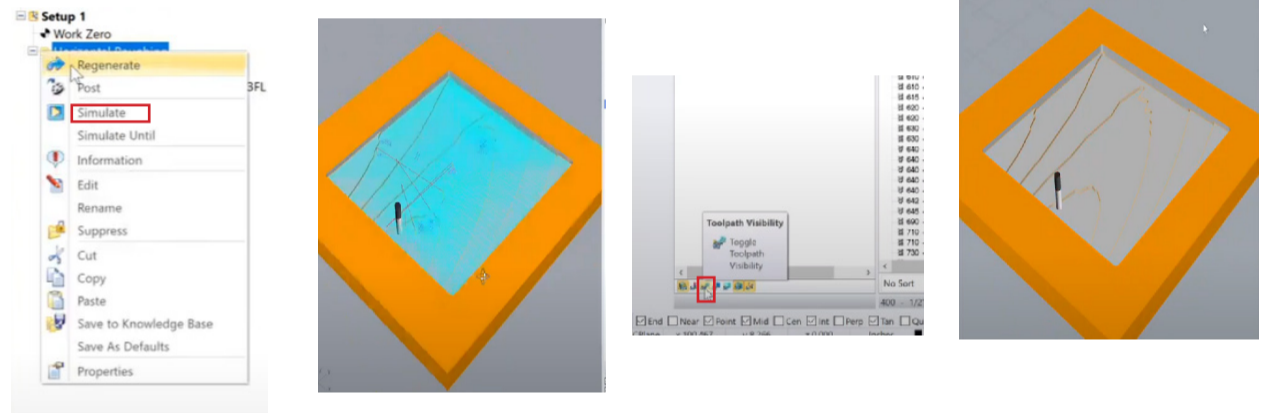
iii. Another option is to draw your own control geometry by drawing a rectangle using the rhino rectangle tool.

- Also remove all the current cut geometry, by clicking "remove all";
 - Click "Select curve" under the control geometry,
 - Then select the drawn rectangle and then right-click and click "Generate."

- You will need to turn your tool paths visibility (located at the lower left corner of the rhino interface) back on to see that you have limited the cutting area to just that rectangle.



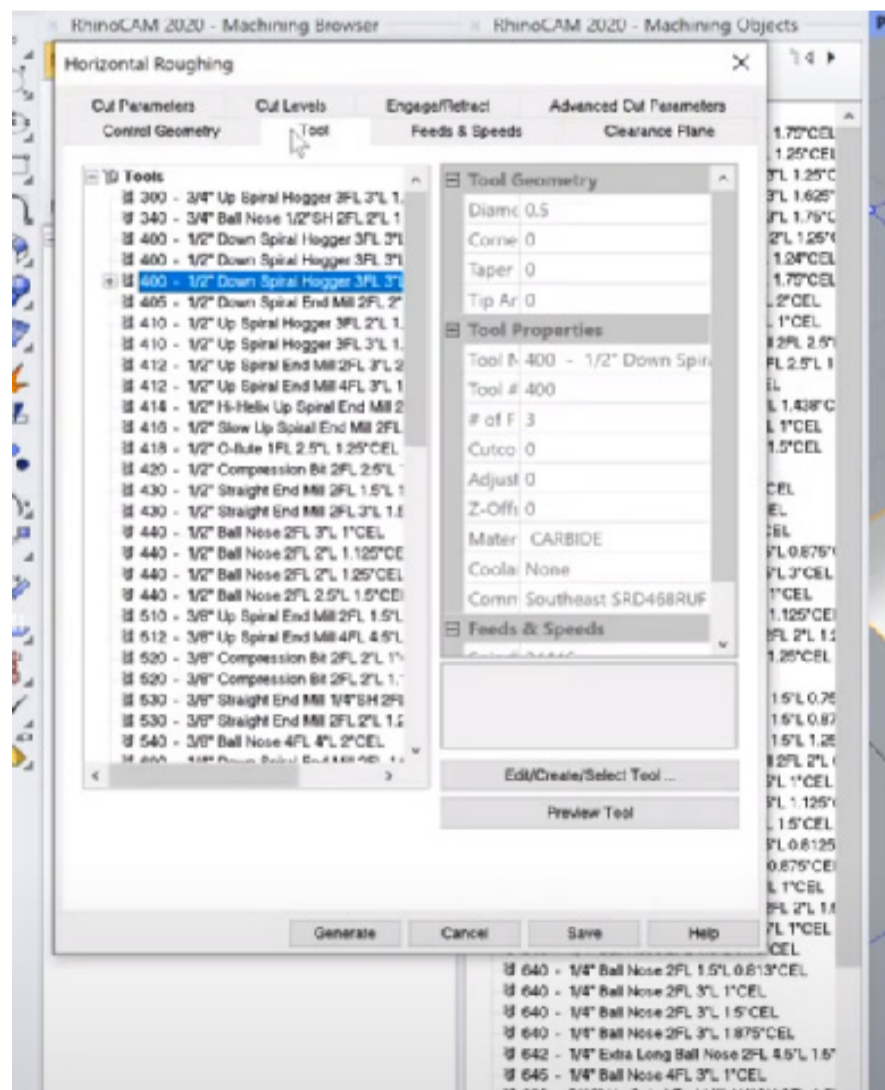
- Simulate and see your result. Ensure you turn off the toolpath visibility.



Let's use the option "i" of control geometry (found in step 3A of this module), which is the first approach _don't forget to remove all the cut geometry.

- This involves using the surface boundary. Under horizontal roughing (control geometry), click "select surface", and click the surface of your geometry, and right click or hit enter.

- Then go to the tool section and select your desired tool; for this example let's use "half-inch down spiral hogger."



Cut parameter and Cut level

- This is a continuation of Step 3A.
- These two (cut parameter and cut level) are found in the horizontal roughing.
- After selecting your desired tool, by default, the cut parameters and cut levels of the geometry should be set.

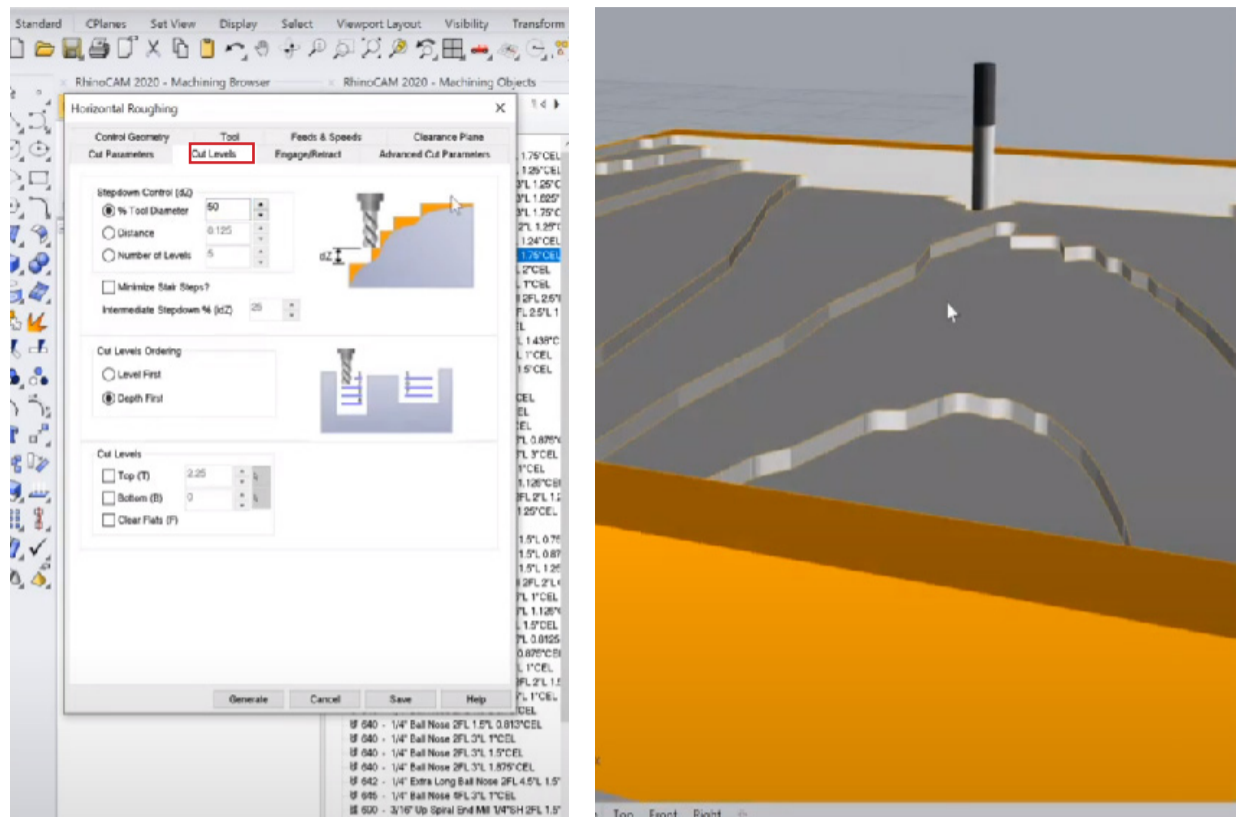
i. Cut level

- Click on the "Cut Levels" tab first. This tab allows you to control the size of the stair-stepping effect caused by the tool path. By default, the step-down (how much it goes down with each step in the roughing pass) is set to 50% of the tool diameter.

- Since we have a 1/2-inch spiral down tool selected, the step-down for the cut level will be 1/4 of an inch (check the dialog box).

- Click "generate" to generate this tool path.

- Simulate it, and a 1/4-inch step-down will be generated. In other words, we will be removing up to 1/4 of an inch of remaining stock above our control geometry (the surface).

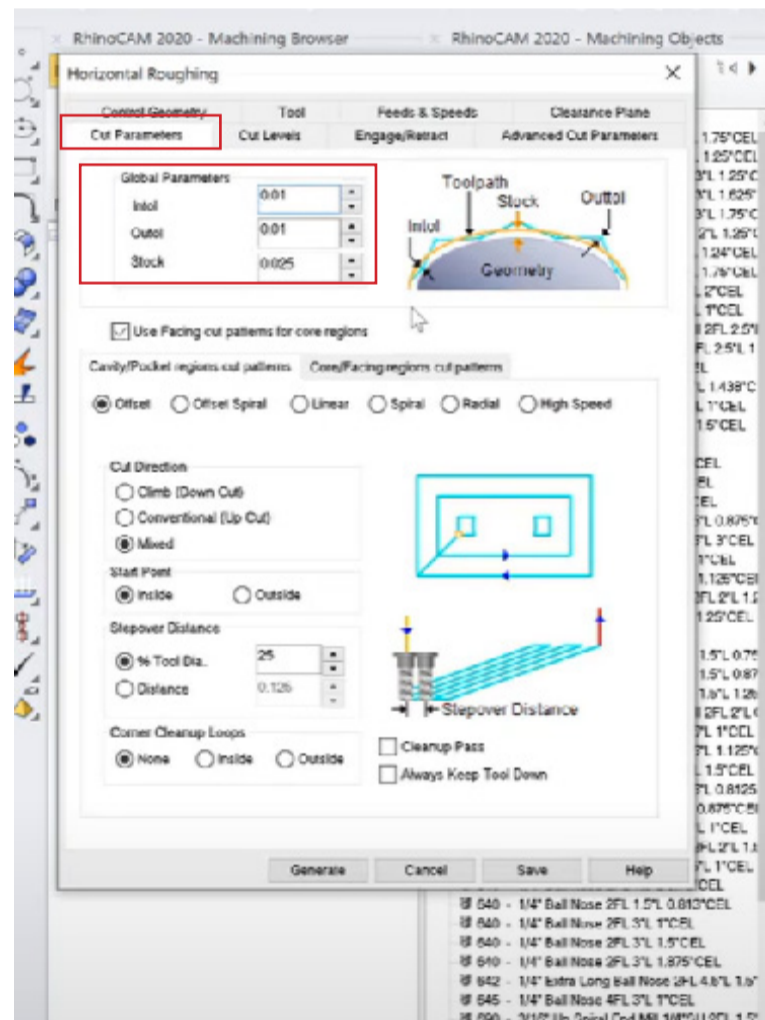


ii. Cut parameter

- Click on the "Cut Parameters" tab, which is controlled by the Cut Levels tab. You will see a legend in the upper right corner (that legend represents the cut geometry); the orange line represents the stock geometry, and the other blue parts represent the tool paths.

- The settings on the left, control how close the tool paths come to your final geometry.

- The stock settings determine the minimum amount of stock that will remain after this roughing tool path is completed. The stock setting here is 0.025.

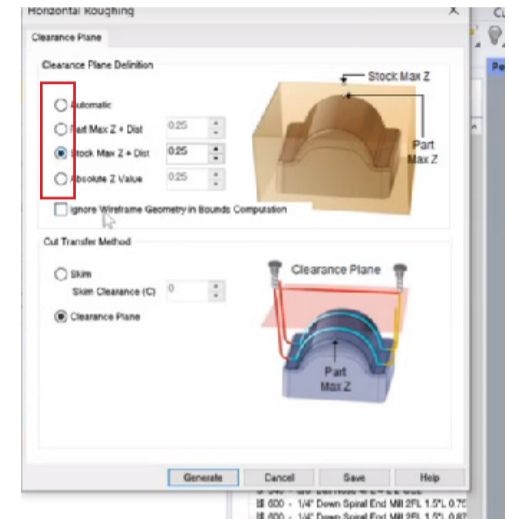
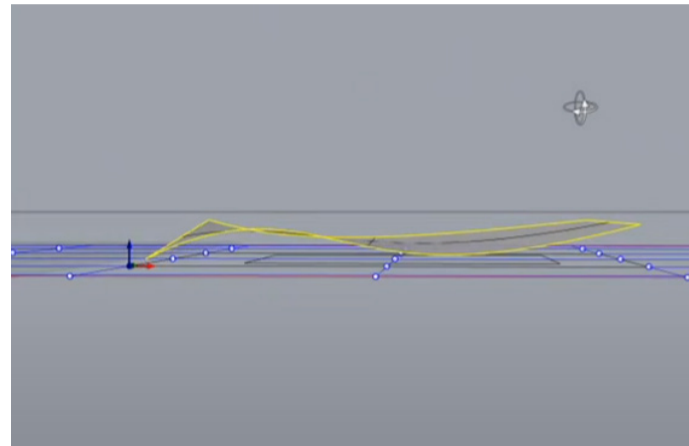
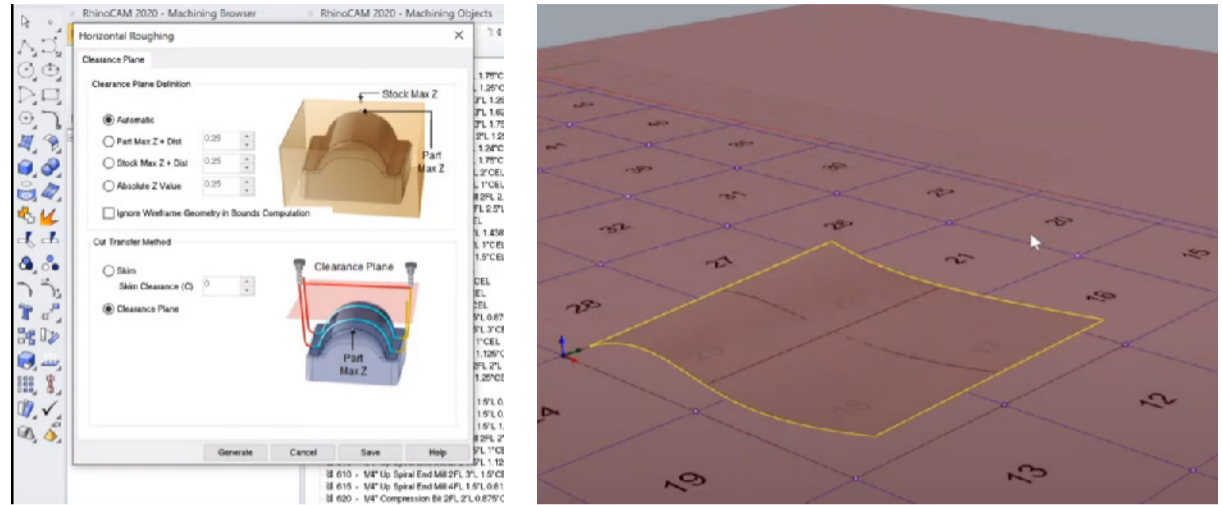


Clearance plane:

- Click on the clearance plane under your Horizontal toolpath
- The clearance plane is defined as a safe height above the material where the machine can move to avoid collisions.
- The colored plane by default is above the geometry.

- By default, the automatic setting calculates the clearance plane based on the maximum z-height of the part and stock, setting it 1/4 of an inch above those values.

- The settings can also be manually adjusted by clicking on them, but automatic is preferable.

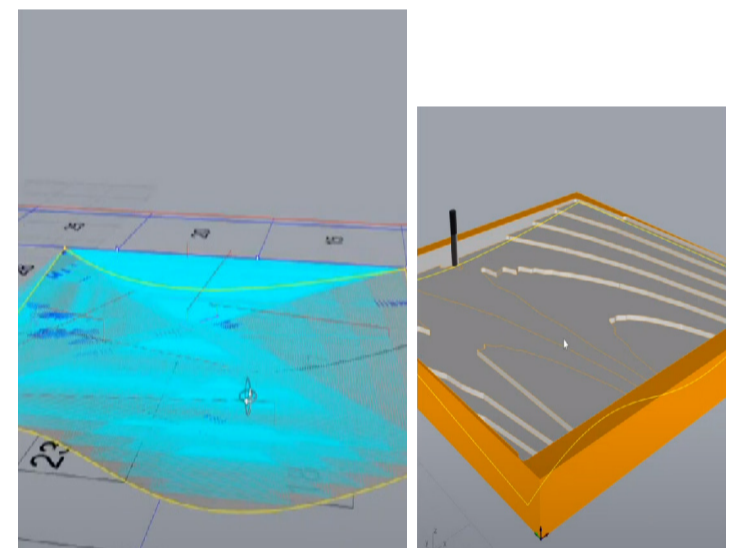
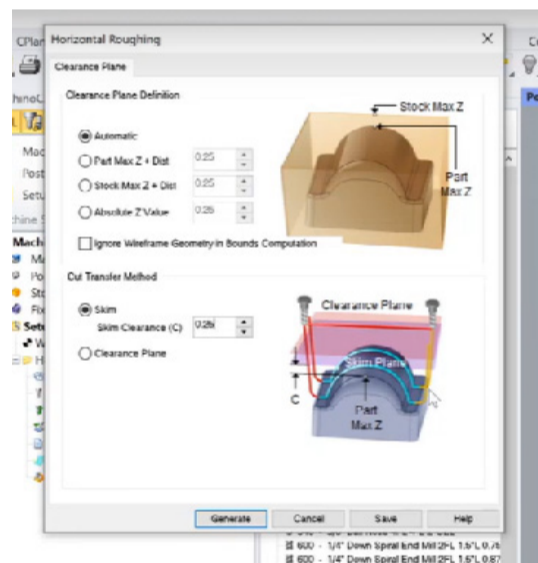


- The skim clearance option is used for jobs with significant topography or complex pieces. Instead of always returning to the clearance plane, it calculates a clearance height above the current cut-out geometry.

- To use the skim clearance, ensure that it is set to a higher value, at least 1/4 of an inch. You can visualize the effect by clicking on the "generate" button.

- However, when toolpaths visibility is turned on, you will notice that the red lines are not all at the same height.

- You can also check the result by simulating.



Cut level tabs:

- To access the cut levels tab, navigate to the horizontal roughing section.

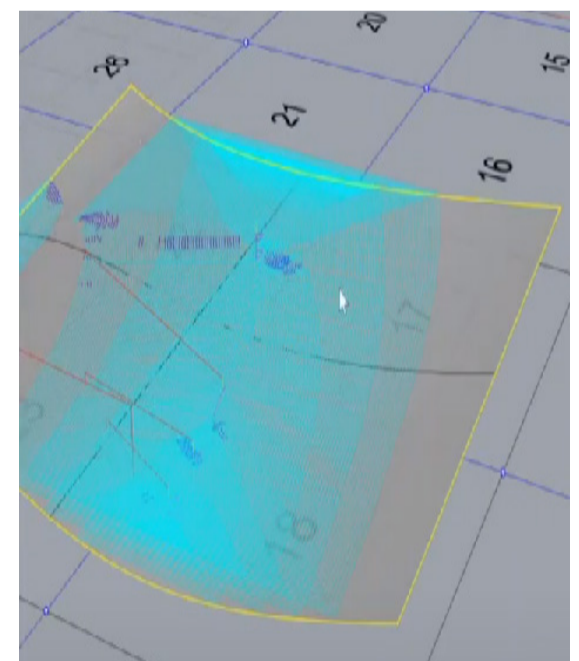
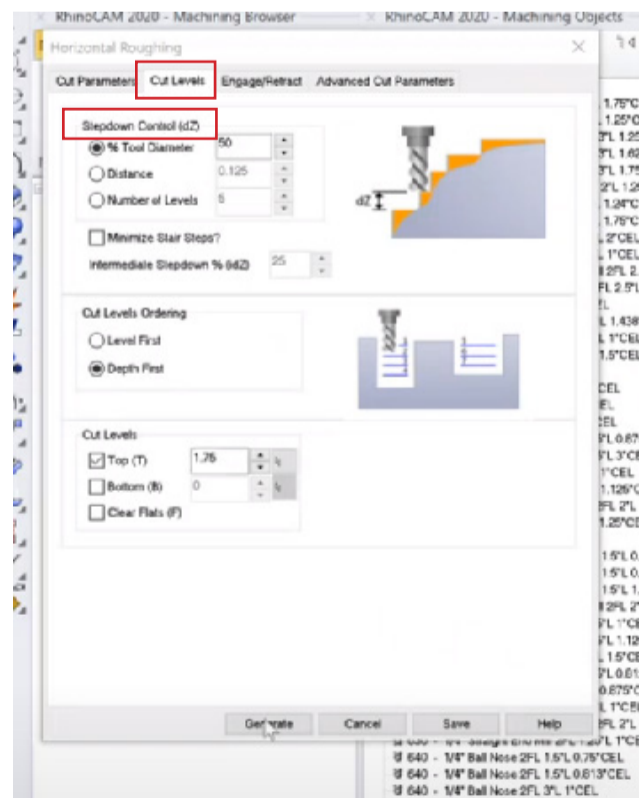
- There are various ways to adjust the results. - In the dialog box, the step-down size can be changed by adjusting the percentage. The specific height (distance) and the number of step-downs can also be adjusted.

- The cut level ordering determines the sequence when there is more than one area to clear during roughing.

- It is possible to specify where the toolpath starts and stops by setting the top level. - This feature is useful if a job needs to be interrupted and resumed later.

- For example, setting the top level to 1.75 inches places the plane at the top of the cut.

- Clicking "generate" shows the resulting toolpath.

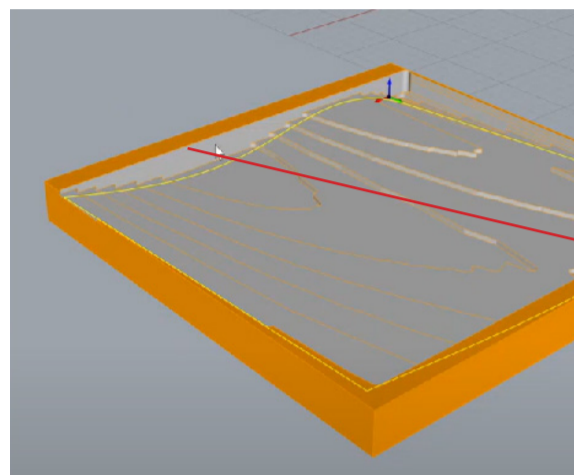
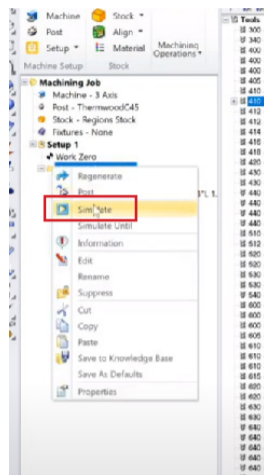
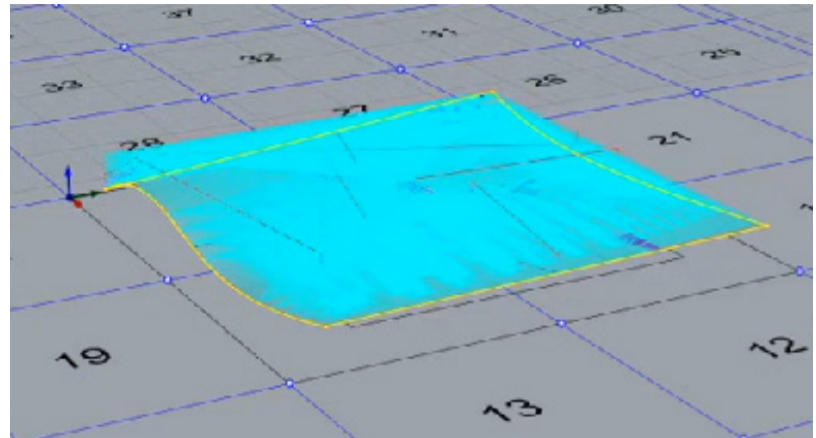
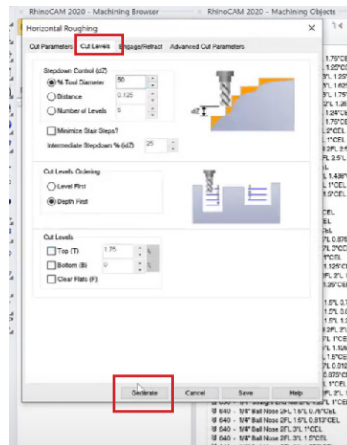


- Another way to divide a toolpath is by setting a first roughing path (top part) and a second roughing path (bottom part).

- Setting the bottom to 1.5 and the top to 1.5 continues the roughing from that point.

- Under cut level, remove the top-level, generate the toolpath again, and simulating it.

- In the simulation, you may notice some leftover walls around the model that need to be removed to avoid potential collisions.

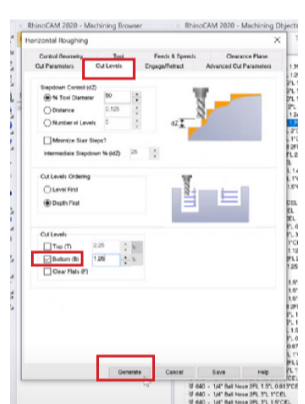
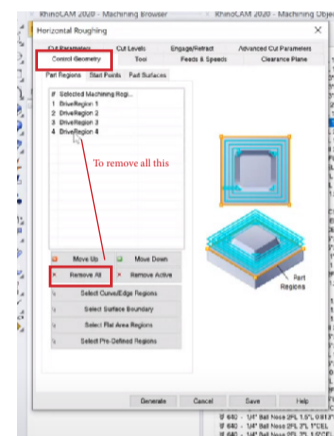
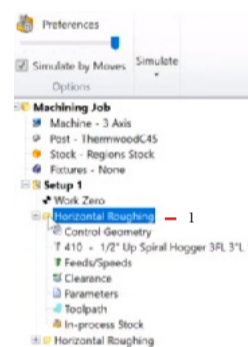
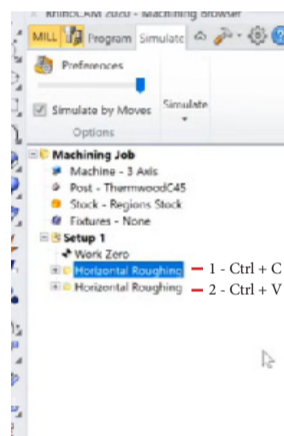


Problem identified with this wall - which could lead to collisions.

B. SECOND TOOLPATH - HORIZONTAL ROUGHING

- To avoid collisions, create another horizontal roughing by copying (ctrl + c) and pasting (ctrl + v) the existing roughing pass.

- In the first horizontal roughing, go to the control geometry section and click "remove all".



Cut level

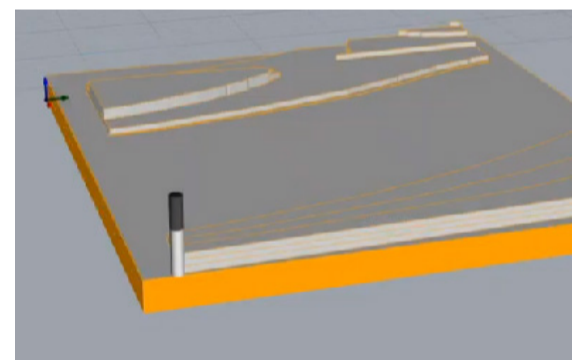
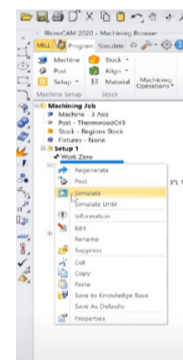
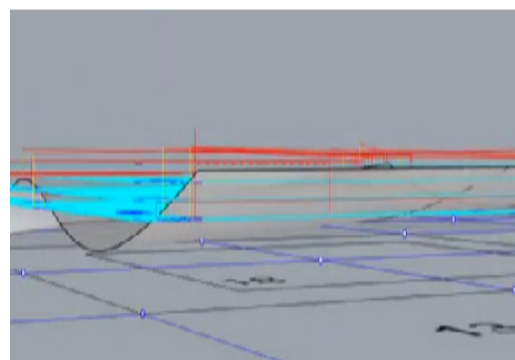
- Then go to the cut level section, and set the bottom level to 1.25.

- Click "generate".

- Clicking "generate" will stop the toolpaths at 1.25 inches above the bed.

- Click simulate

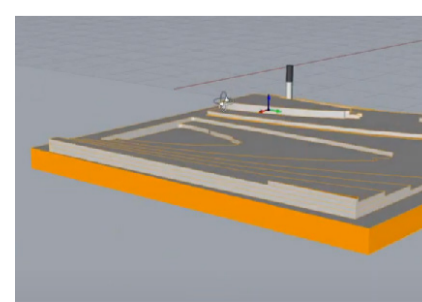
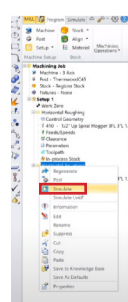
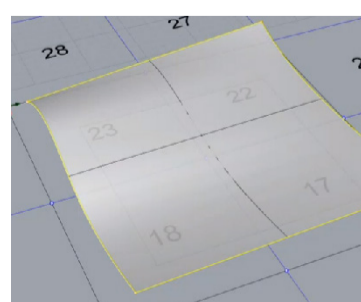
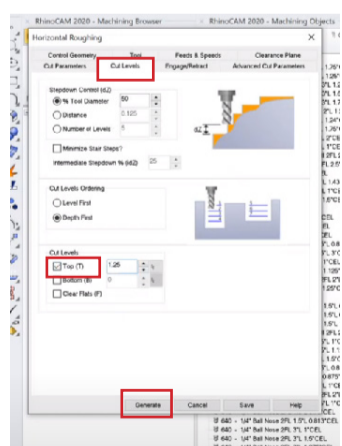
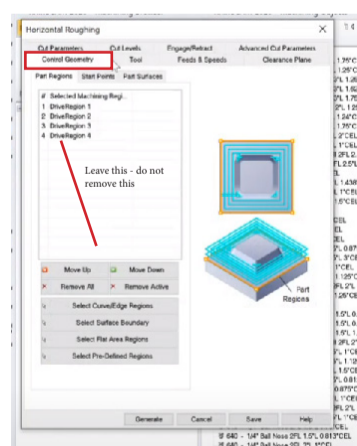
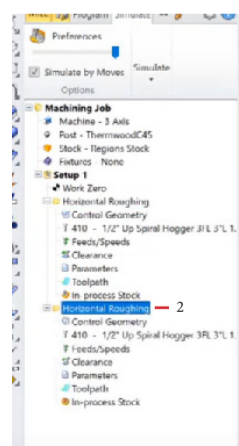
- Clicking "simulate" will clear those edges



- Next, select this second horizontal roughing and leave the control geometry in place.

- Go to the cut level section, and set the top level to 1.25 and limit the rest of the roughing to this geometry.

- Click "generate."



- Simulating the toolpath will show the continuation from the previous path, resulting in only a small remaining wall.

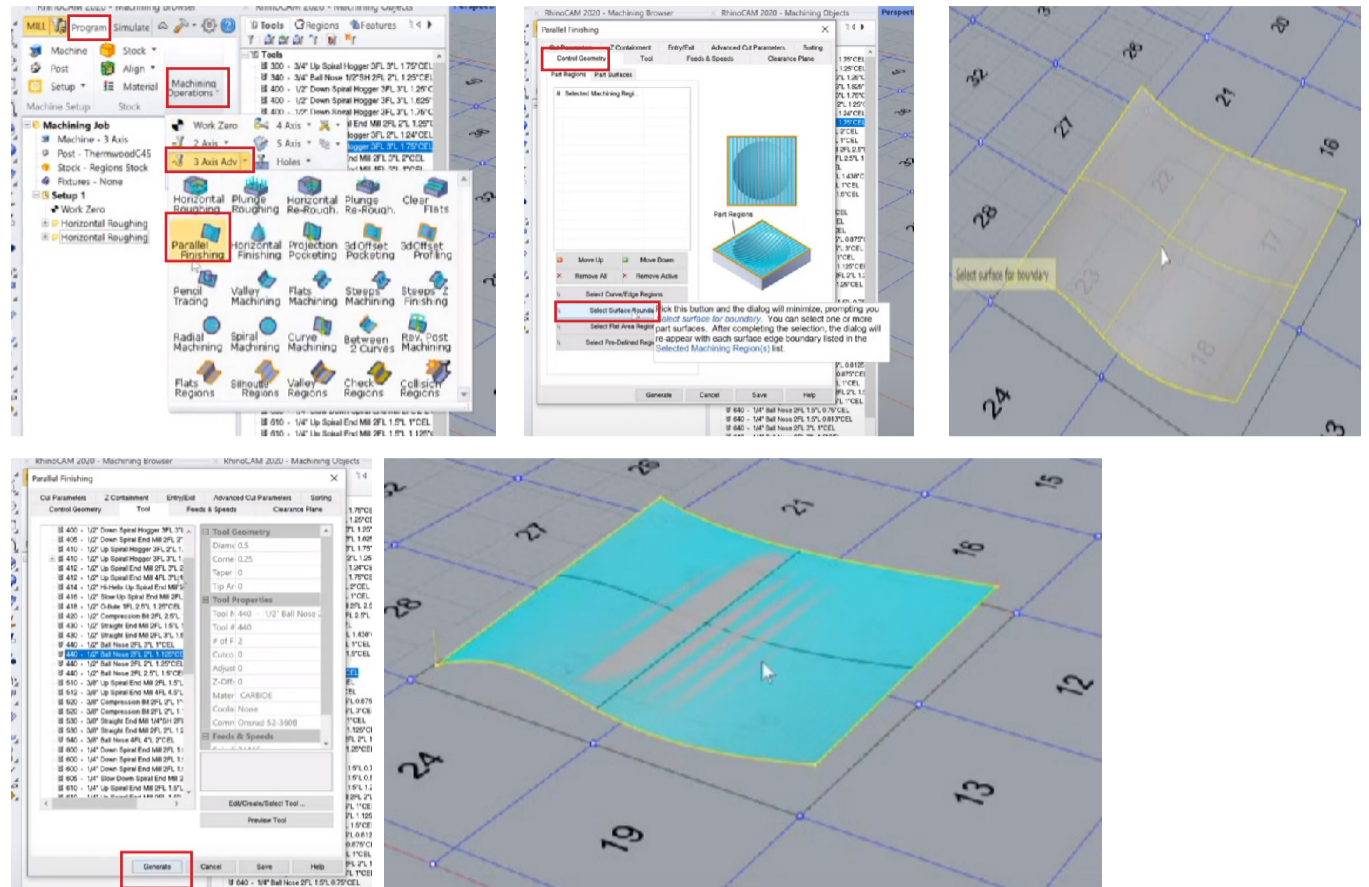
1n MODULE 8: RHINOCAM

C. PARALLEL FINISHING PATH:

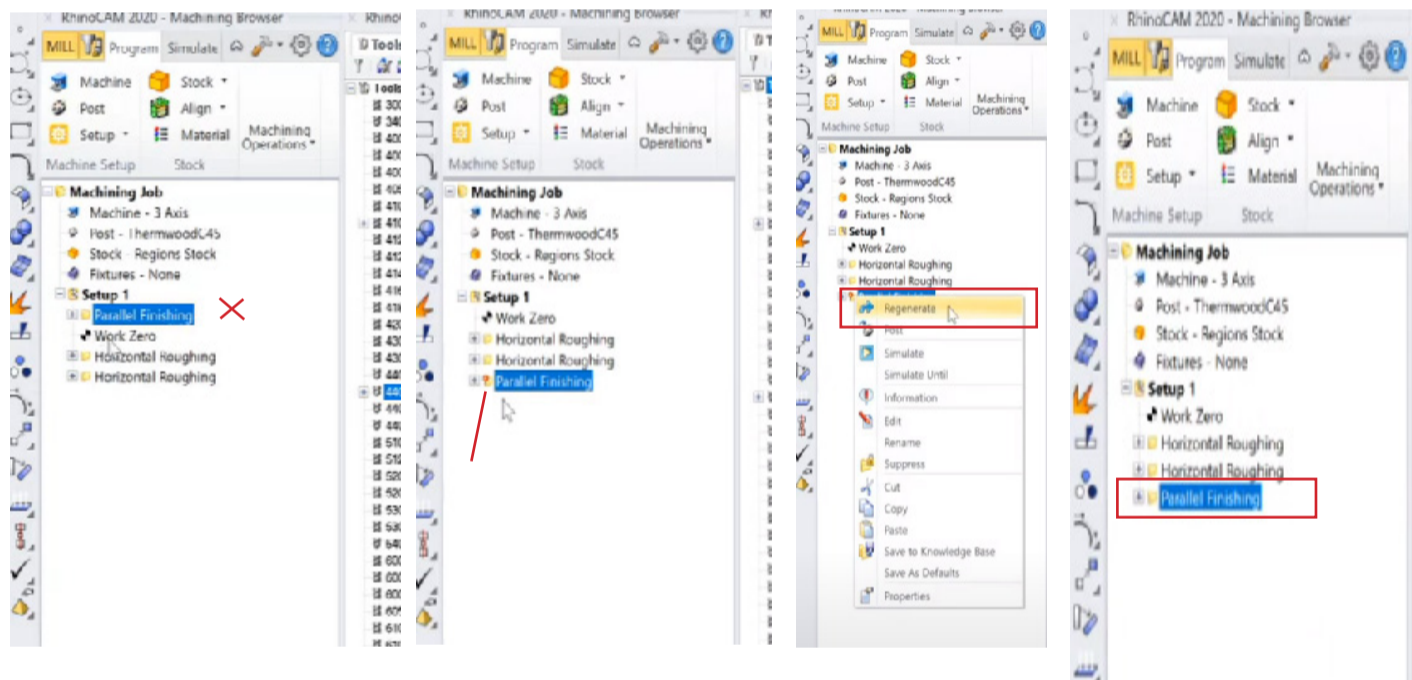
- The next step is the finishing path.
- Click on the program tab and set up a new machining operation.
- Select the 3-axis tab and choose parallel finishing. Parallel finishing produces a step-down effect, while parallel finishing follows the curved geometry.

Control geometry

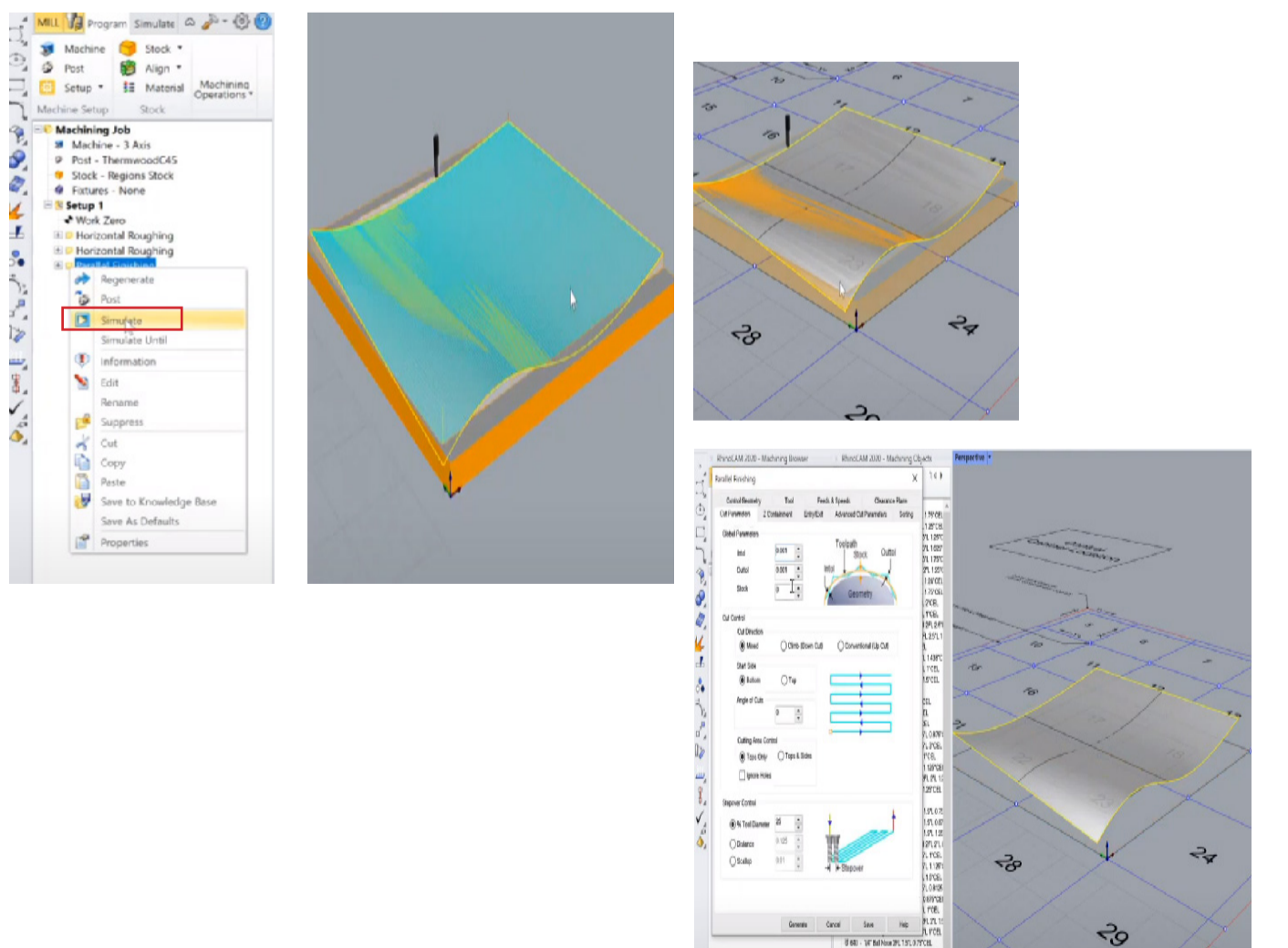
- Access the control geometry under parallel finishing and select the surface boundary as before (in step 3A).
- In the tool section, choose the desired tool, such as a 1/2 inch ball nose.
- Clicking "generate" will show the resulting toolpath.
- By turning on the toolpaths visibility, you will see projected lines on the surface that the tool follows back and forth.



- In a situation where the toolpath (parallel finishing) appear at the top of the list. If necessary, drag it to the bottom to determine its replacement.
- Please, note this: Remember that the order of the toolpaths matters as they are executed from top to bottom.
- When the toolpath (parallel finishing) is dragged, a red icon appears beside it, and this indicates that the toolpath is not valid.
- To generate a valid toolpath, right-click on the parallel finishing operation and select "regenerate."



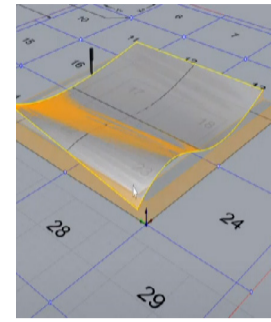
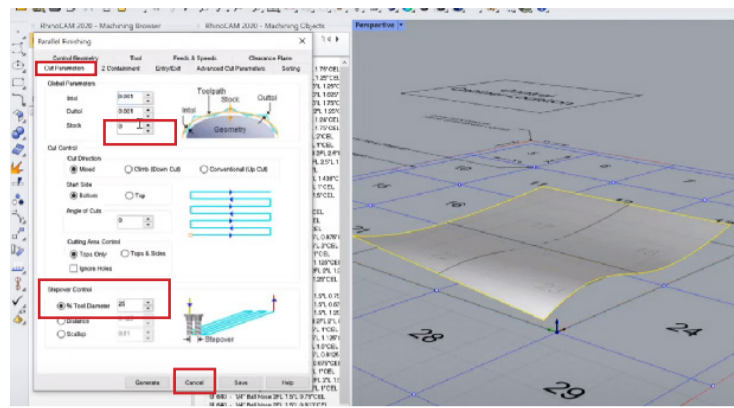
- The result will be the same, but the work zero is used as a reference.
- Simulate the toolpath by right-clicking on the parallel finishing operation. The simulation will show the tool moving back and forth along the surface, removing the remaining stock geometry to achieve the final surface.
- As a finishing toolpath, it leaves no remnants of stock.
- After the simulation, turn off the toolpath visibility to view the final surface.



Cut parameters

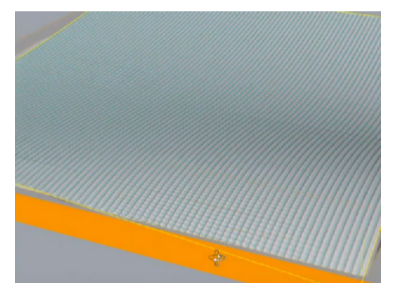
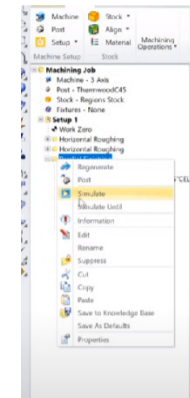
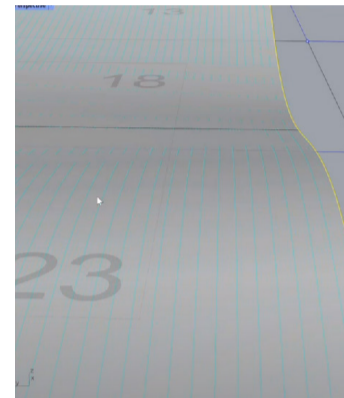
- Check the cut parameters to ensure that the stock option is set to zero, indicating no remaining stock.

- In the stepover control section, the default stepover is 25% of the tool diameter. Since we're using 1/2 inch tool bit or tool here, that means that our step over is an eighth of an inch right.



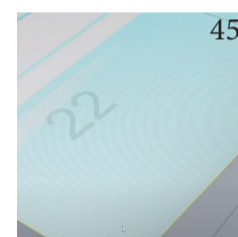
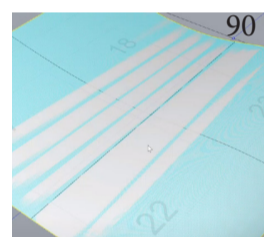
- Adjusting this value allows control over the stepover distance. For example, setting it to 50% would result in a larger stepover.

- Simulating the toolpath with a 50% tool diameter will show higher ridges, and the simulation may be faster, but the surface may be less precise and not as smooth compared to a 25% tool diameter.



- For smaller areas, consider using a smaller bit and reduce the tool diameter under the tools and cut parameter section.

- The angle of cuts can be specified (try 90 and 45) under the cut parameter section, affecting the toolpath direction based on the curvature and geometry nature.

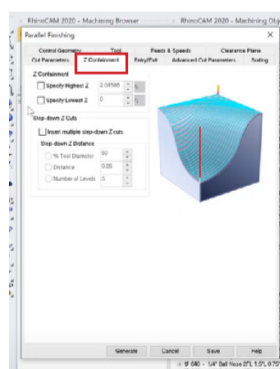


- Another way to control the result is by specifying the scallop height, which determines the stepover distance to minimize the scallop.



z containment

- Similar to specifying the top and bottom for the roughing pass, the z containment allows specifying the top and bottom levels for the finishing path.



D. FEED AND SPEED SETUP

- For this session, you will need to download two files: "GWizard" and "RhinoCam to Gwizard"
- These downloads will be provided by the director of Digital Fabrication.
- First of all, we need to download and install GWizard. This is an old version, but once installed, it will prompt you to upgrade to the newest version.

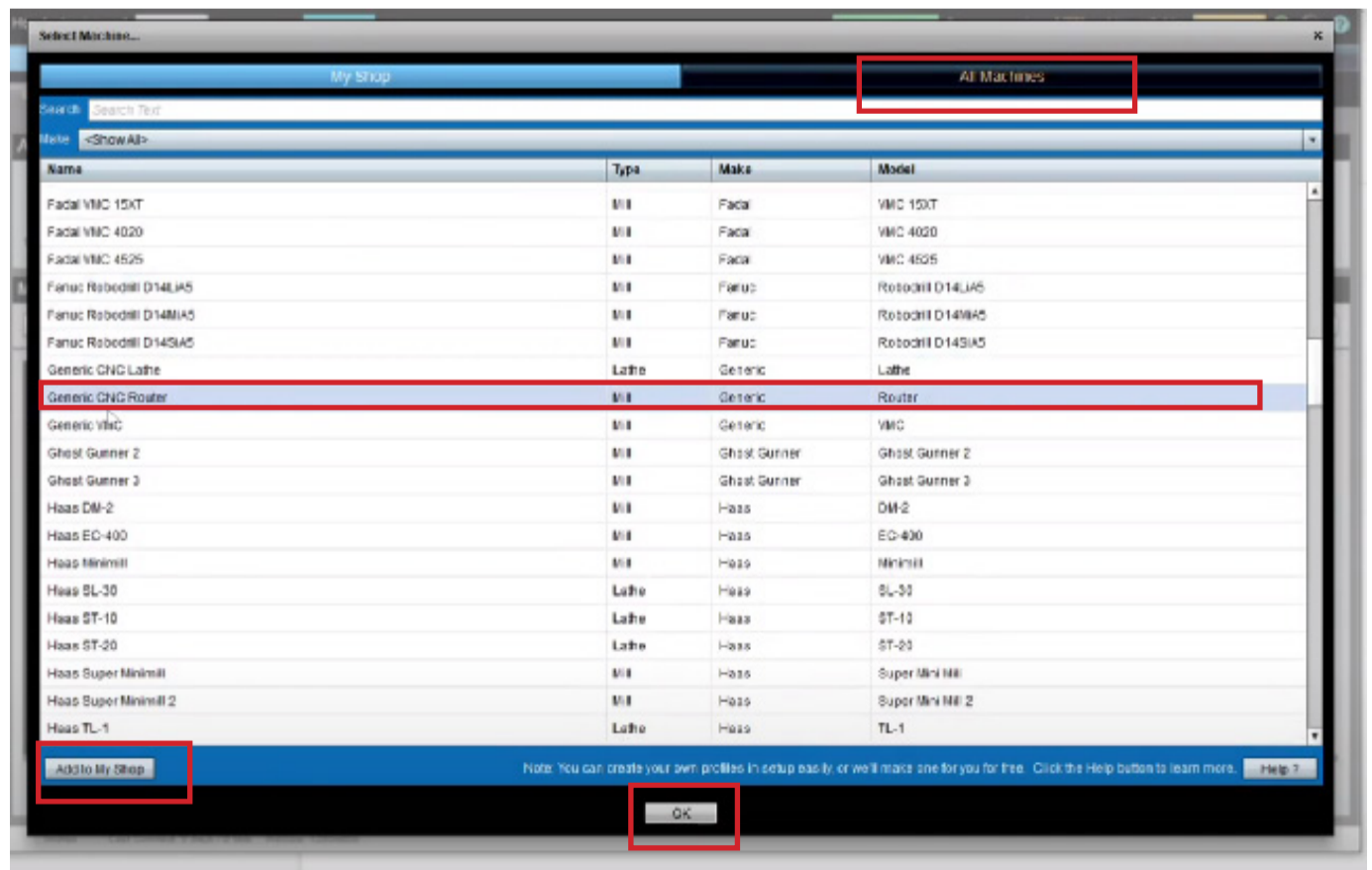
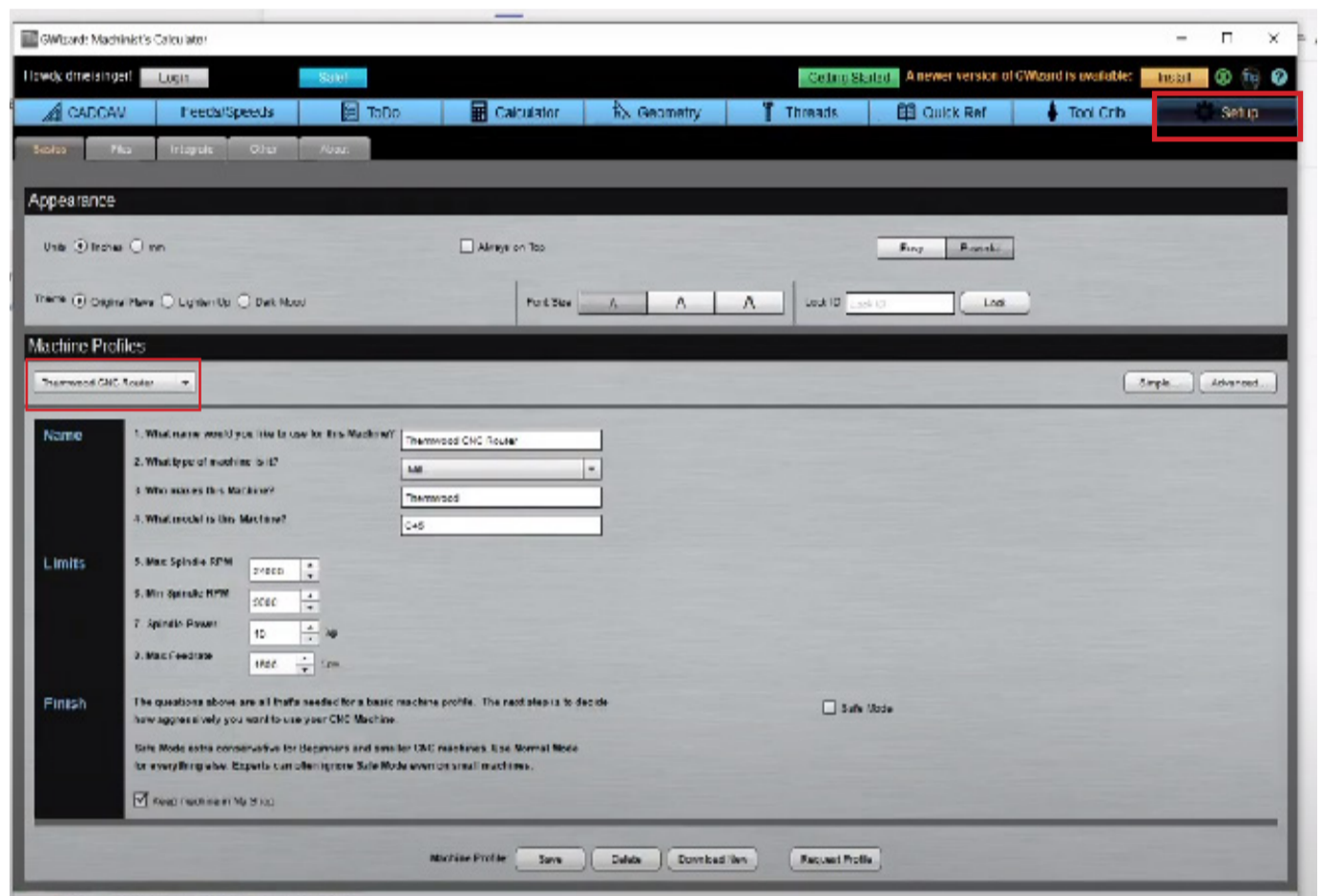
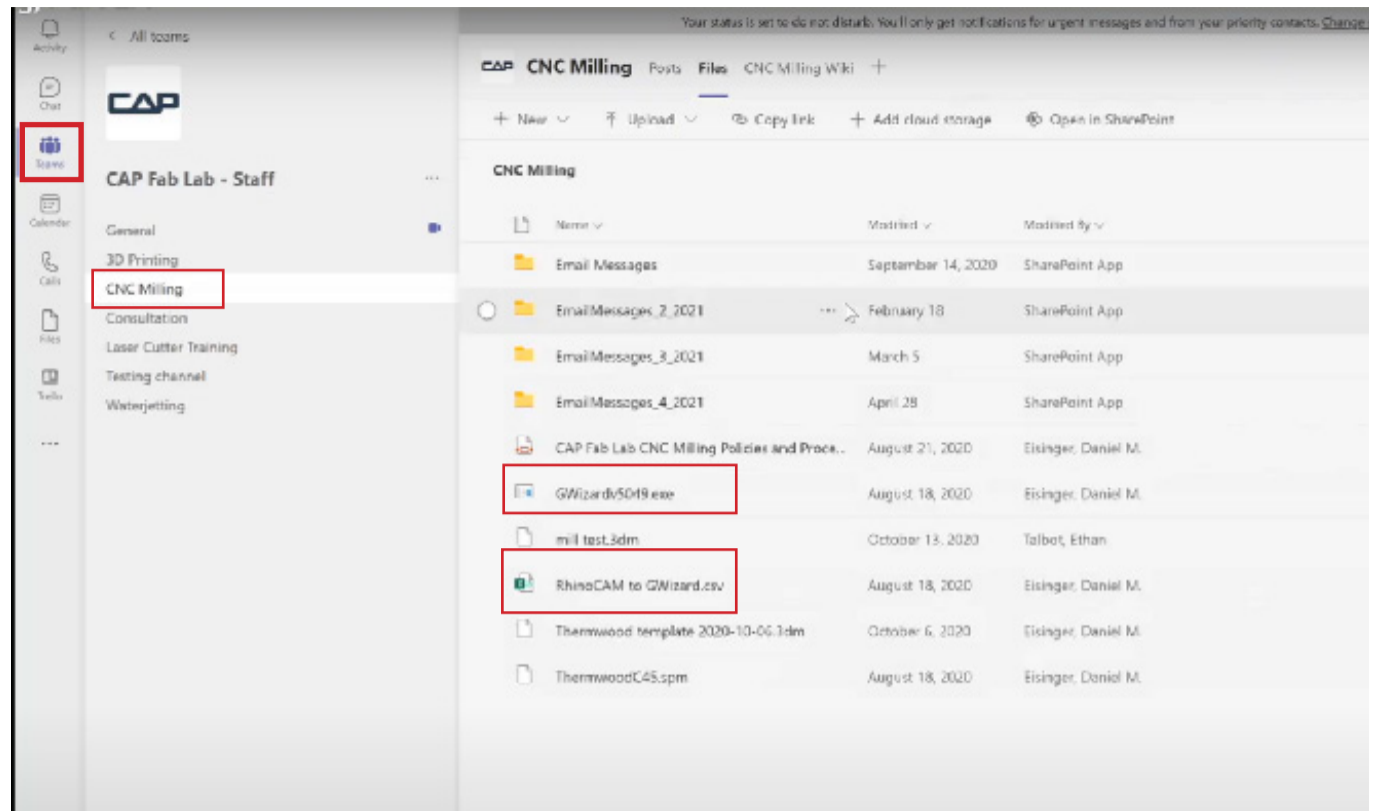
- GWizard is a tool that helps with the calculation of feeds and speeds. When installed, some setup needs to be done.

- After downloading and installing G Wizard from the file, open the software.

- The first configuration step is to ensure that the right machine profiles are set up.

i. Setup on GWizard

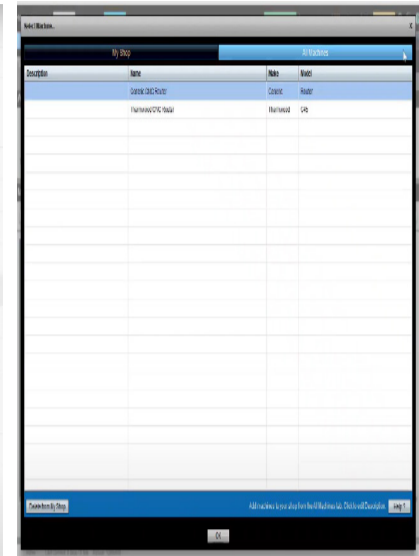
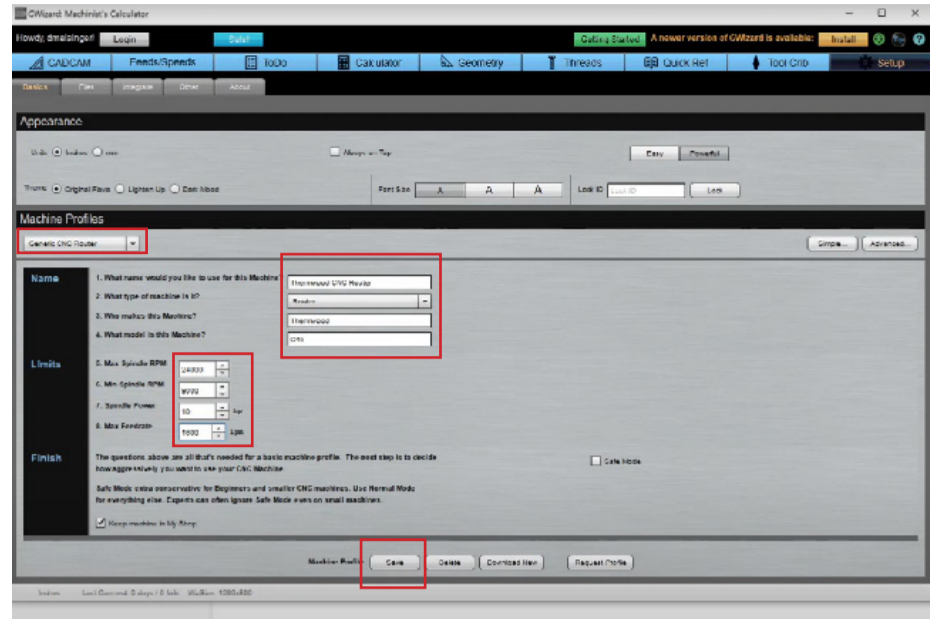
- Click on the setup tab to begin.
- Click the Thermwood CNC router tab on the left (sometimes it may appear as another name like Generic router or anything).
- Once opened, you will see two tabs: "My Shop" and "All Machines."
- Click "All Machines" and scroll down. Then, click the Generic CNC router.
- After selecting "Generic CNC Router," click the "Add to my Shop" tab. Then click OK, and it will appear at the Setup home.



- Click the Generic CNC Router and click "My Shop." You will see that it has been added to your shop. Close this new window to go back to the setup home.

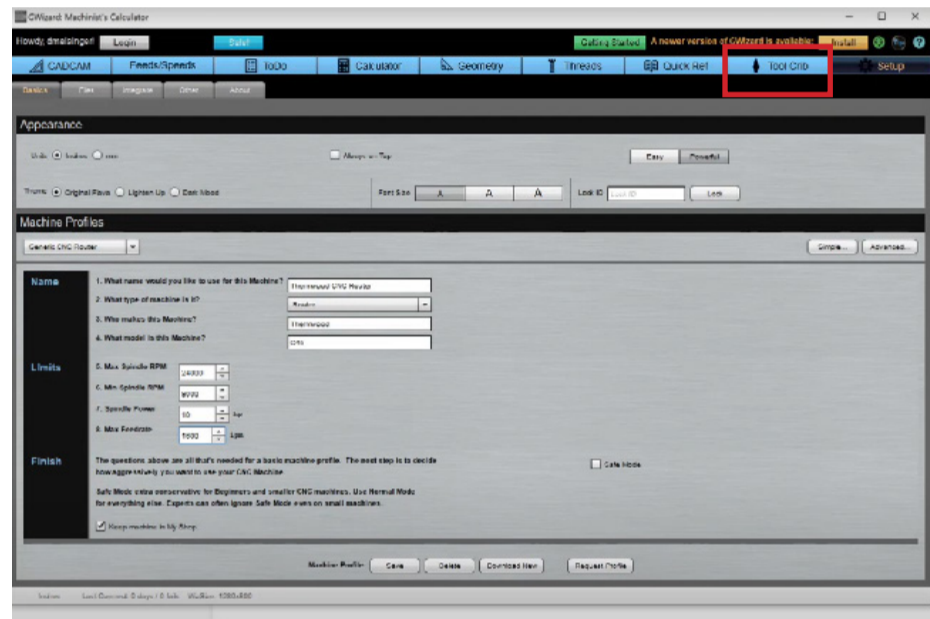
- The next step is to specify details to match what we have. Check the image on the right for specific input. Use the Router option for the type of machine.

- The important settings to configure are the maximum spindle speed (24,000), the minimum spindle speed (9,000), the spindle power (10), and the maximum feed rate (1,600). Once you have set those values, hit save.



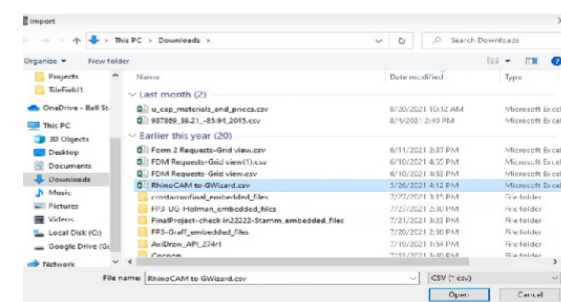
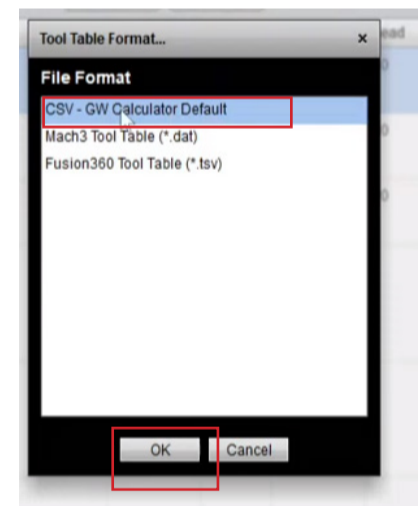
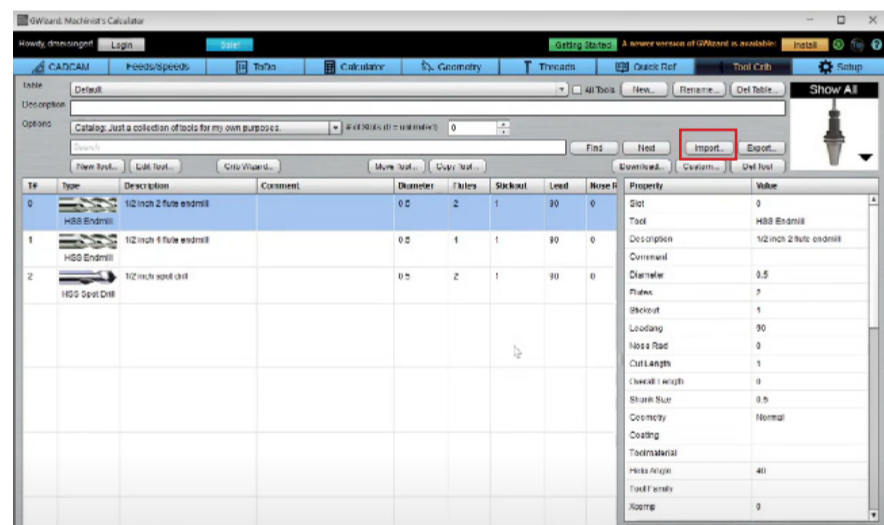
ii. Toolcrib in G Wizard

- After completing the setup, go to the basics section and click the "tool crib" tab. This is where the list of tools is located, and it needs to match what we have in RhinoCam in terms of tool numbers and geometries.

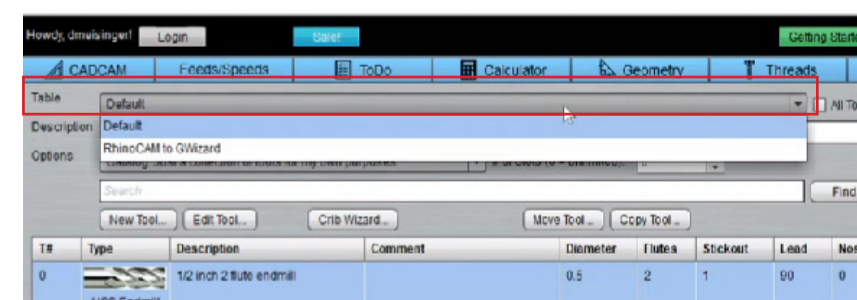


- Go back to Teams where you downloaded G Wizard and download the file named "RhinoCam to G Wizard" (refer to step 3G, on where to download). This file is a database of tools in the format that G Wizard needs.

- Once downloaded, click the import tab under "Tool Crib," and choose the top option for a CSV file. Then click OK and select the downloaded file. This will load the tool database into G Wizard.

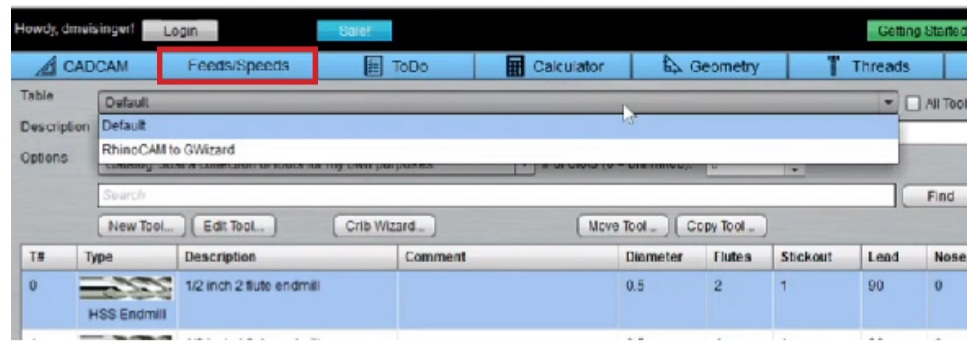


- After loading the tool database, click the top table dropdown and select the "RhinoCam to G Wizard" option. It will provide a list of tools. Ensure that the tool numbers, descriptions, and details match RhinoCam.



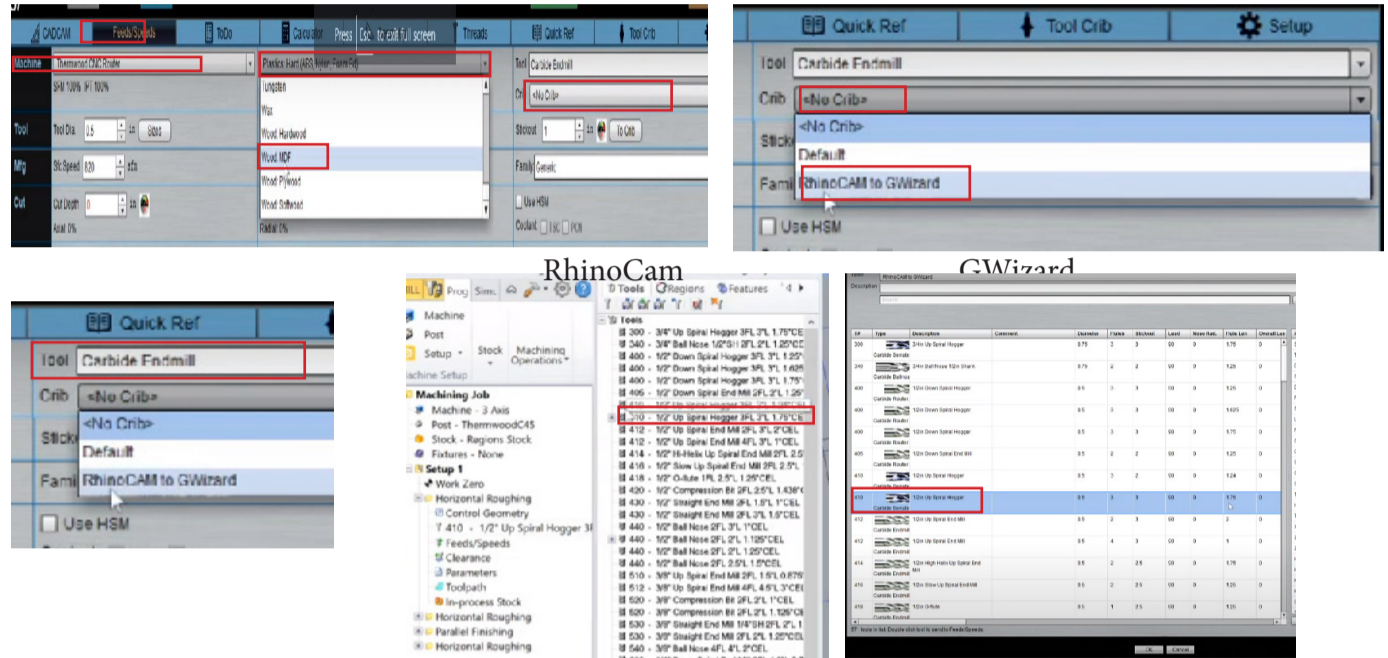
iii. Feedrate in G Wizard

- Once you have set up those two things, the main tab where you'll spend time is the "Feeds and Speeds" tab.
- Click the "Feed/Speeds" tab and fill out all the details.



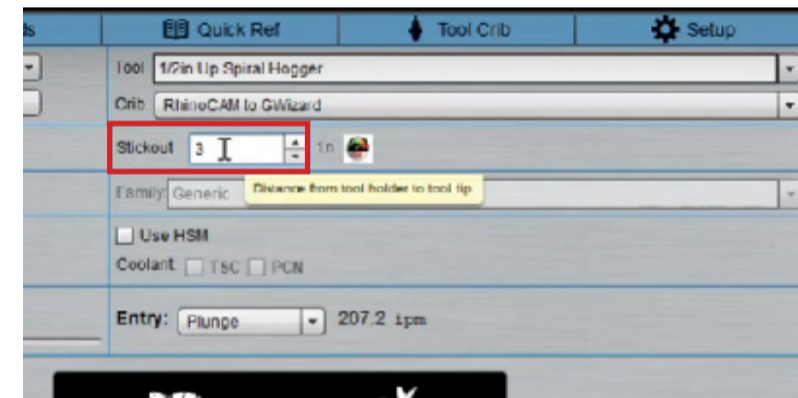
Machine

- Select the Thermwood CNC Router as illustrated in the image. To do that, click the dropdown button, it will lead you to "my shop", and select "Thermwood CNC Router."
- Select the material you are working with, in this case, Wood:MDF.
- Choose the "RhinoCam to G Wizard" tool crib.
- Click the tool dropdown on the right to load the tool selection window. Select a tool that corresponds to what you have in RhinoCam.



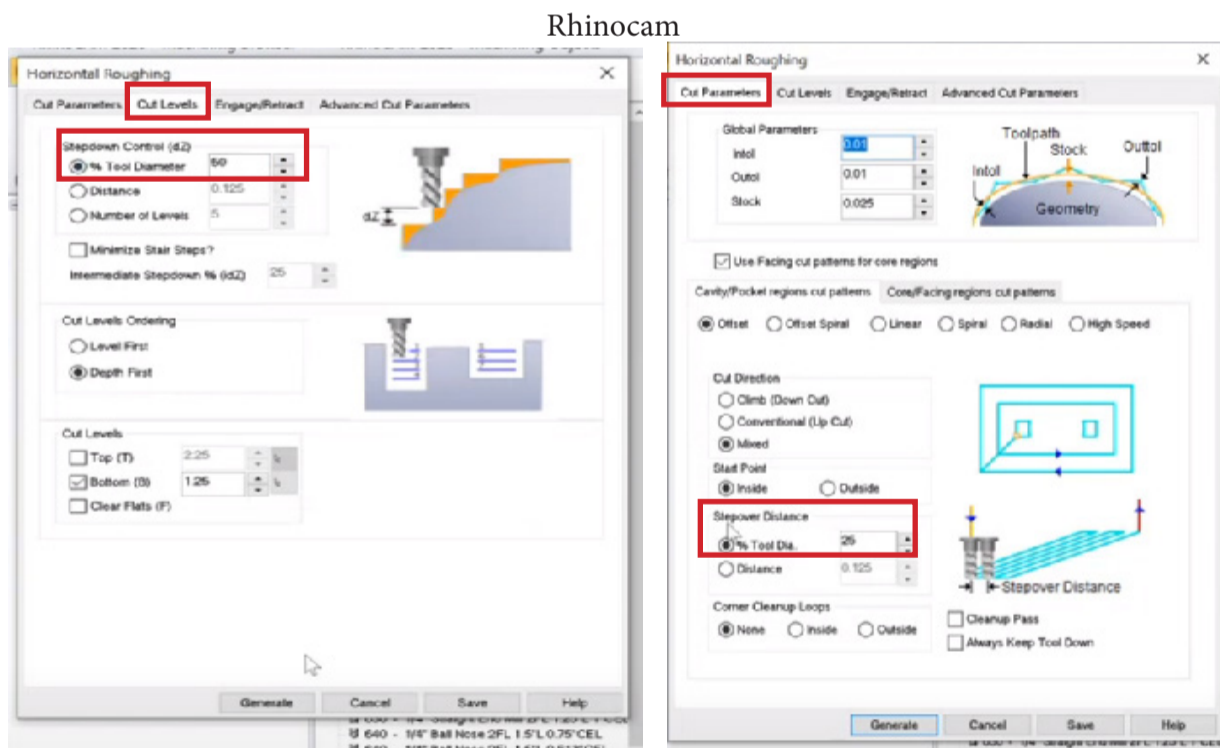
Tool

- You will notice that some information is automatically filled out.
- The thing to fill out here is the stick out, which refers to how you load the tool in the tool holder. For example, if you've loaded a four-inch tool into the tool holder, you probably have a three-inch stick out. Reducing the stick out reduces deflection.

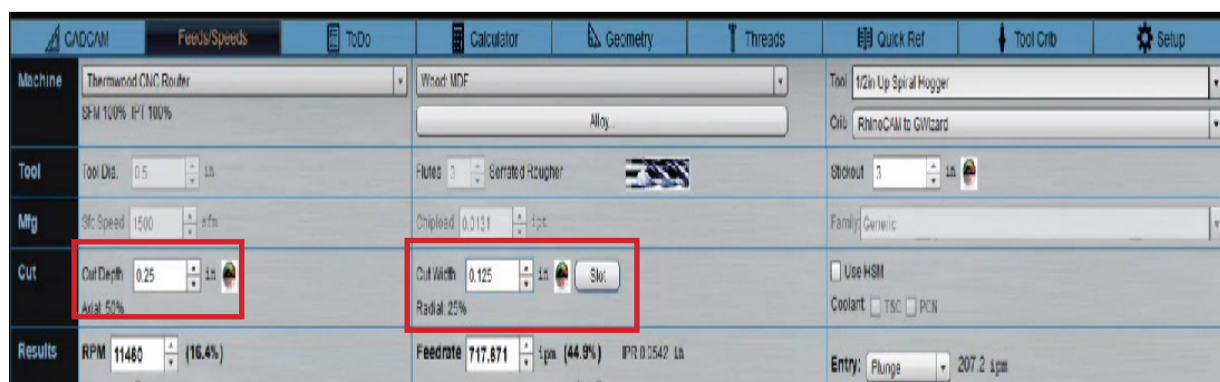


Cut

- To fill in the cut depth, we need to go to the RhinoCam toolpath to check the step-down input.
- Go to the first Horizontal roughing path, and click cut levels.
- For this toolpath, we had set the cut level to 50% of the tool diameter, which would be 1/4 of an inch cut depth since we have a 1/2 inch tool.
- In G Wizard, fill in 0.25, which is equivalent to 1/4 inch in RhinoCam. You'll see that it gives you the 50% value.
- Next, to set the cut width in G Wizard, you also need to go to the RhinoCam toolpath and check the cut parameter section. In this toolpath, we had set the step-over distance to 25%. Fill in an eighth of an inch, which is equivalent to 25% of the half-inch tool.



GWizard

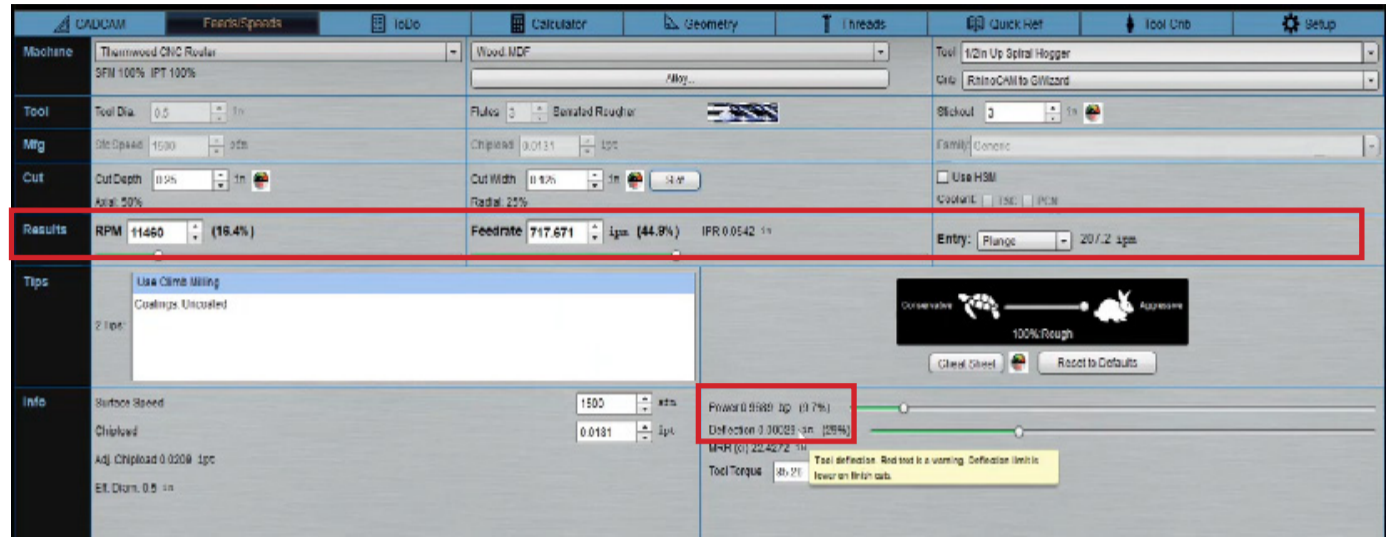


Result

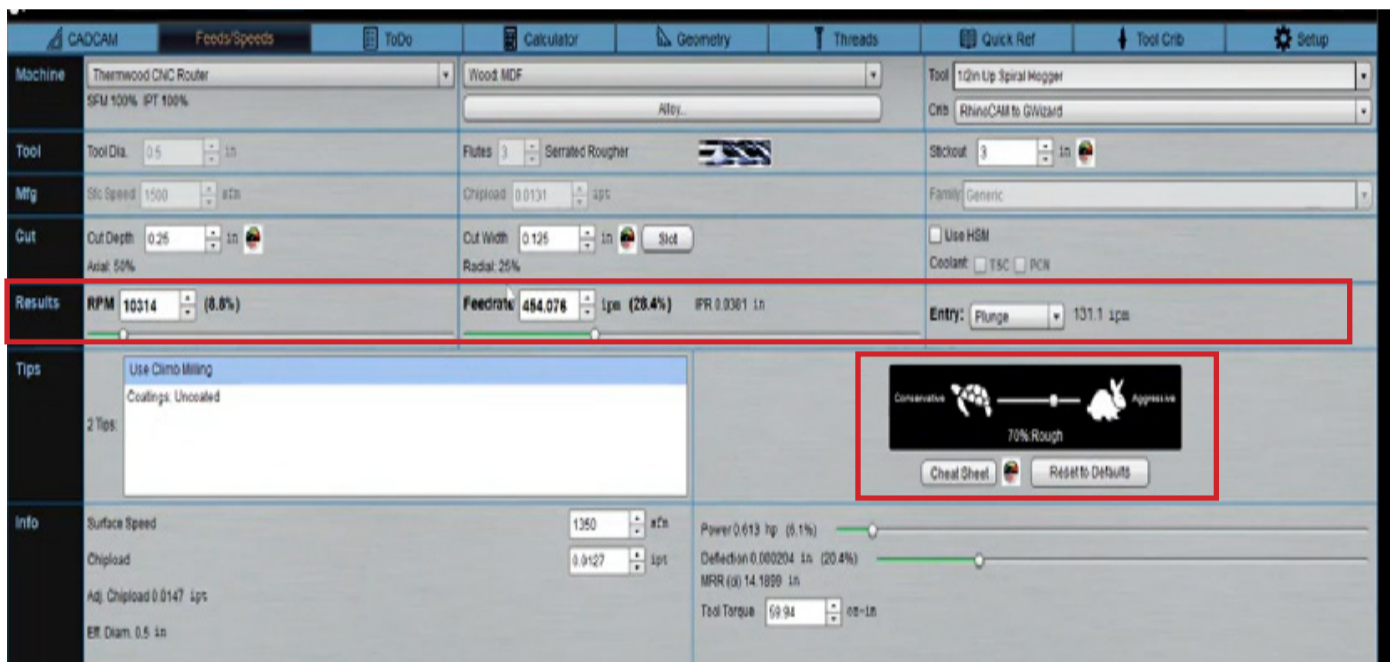
- Under this section, we have two important outputs: RPM (spindle speed) and feed rate (lateral speed). G Wizard calculates these values based on the input provided (machine type, tool, cut information).

- The software calculates a feed rate of approximately 700, which is very fast. It takes into account the tool diameter, number of flutes, and directly affects the result.

- It is using only 10% of the horsepower, and the tool is deflecting only a tiny amount. The 29% which appears refers to the 29% of the maximum allowed deflection that we're okay with.



- There is a slider between conservative and aggressive on the right. You can adjust it based on your requirements. For example, if you are using a smaller amount of material or unsure if it will stay in place, you can reduce the feed rate (set it to 70%), which will also reduce the spindle speed (RPM) and feedrate.



Way to calculate the feedrate and RPM: Chipload chart

- There is a direct relationship between spindle speed and feed rate, which is determined by the chip load value along with other tool and material factors. The chip load is obtained from the chosen material and can be found on a website that provides a chip load chart.

- Search for "Vortex chipload chart" on a browser.

- For MDF material, you need to look up the chip load chart for the tool diameter you are using and insert it into the formula for calculating feed rate and RPM. The formula is found in the chart, check the image at the right.

The feed rate is defined in terms of RPM and the number of flutes in the chip load, while RPM is defined in terms of feed rate and the number of flutes in the chip load. Generally, higher RPM results in better surface finish, but it also generates more friction, causing mechanical wear on the tool cutting edge. The goal is to select the lowest RPM possible for each application, unless surface finish is the priority.

Chip Load Chart

TOOL DIAMETER	HARD WOOD	SOFTWOOD/ PLYWOOD	MDF/PARTICLE BOARD	HIGH PRESSURE LAMINATE	PHENOLIC/ PAPERSTONE
1/8"	.003"-.005"	.004"-.006"	.004"-.007"	.003"-.005"	
1/4"	.009"-.011"	.009"-.013"	.013"-.016"	.009"-.012"	.004"-.006"
3/8"	.015"-.018"	.017"-.020"	.020"-.023"	.015"-.018"	.006"-.008"
1/2" & up	.019"-.021"	.021"-.023"	.025"-.027"	.023"-.025"	.010"-.012"

TOOL DIAMETER	HARD PLASTIC	SOFT PLASTIC	SOLID SURFACE	ACRYLIC	ALUMINUM
1/8"	.002"-.004"	.003"-.006"	.002"-.004"	.003"-.005"	.003"-.004"
1/4"	.006"-.009"	.007"-.010"	.006"-.009"	.008"-.010"	.005"-.007"
3/8"	.008"-.010"	.010"-.012"	.008"-.010"	.010"-.012"	.006"-.008"
1/2" & up	.010"-.012"	.012"-.016"	.010"-.012"	.012"-.015"	.008"-.010"

Other Valuable Formulas:

Feed Rate = RPM x number of flutes x chip load

RPM = Feed Rate / (number of flutes x chipload)

Metric Conversion: Divide inches per minute by 39,374

(x, 300 inches per minute divided by 39,374 = 7,62 meters per minute)

RPM Selection

The general operating RPM for tooling contained in this catalog is between 10,000 and 20,000 revolutions per minute. Usually the higher the RPM, the better surface finish becomes. However, the higher the RPM, the more friction is generated between the tool and the work piece. This friction is what creates the mechanical wear on the cutting edge. Your goal is to select the lowest RPM possible for each application.

iv. FEED AND SPEED SETUP IN RHINOCAM

a. First toolpath: Horizontal roughing path

- Go back to RhinoCAM, and go to the first horizontal roughing toolpath, and navigate to the "Feed and Speeds" tab, which will be the same for every toolpath you set up.

- Copy the RPM value (10,300) from G Wizard (you can round it up) and paste it in RhinoCAM as the spindle speed, setting the direction to clockwise (CW).

- For the feed rate, copy the feed rate value from G Wizard and enter it in the "Cut" section under the "Feederate" tab. In this case, it is approximated to 450.

- In the "Approach and Engage" section, set the "Approach" to 120 (assump.) and the "Engage" to the value provided in G Wizard (which is 131 (Entry - plunge)). You can change these values if needed.

- In this module, we assumed 120 for the "Retract" which means the tool is leaving or retracting from the material.

- When you are done filling the feed and speed information, hit the generate button.

- The plunge and departure values can be set automatically, but pay attention to the engage and cut feed rates.

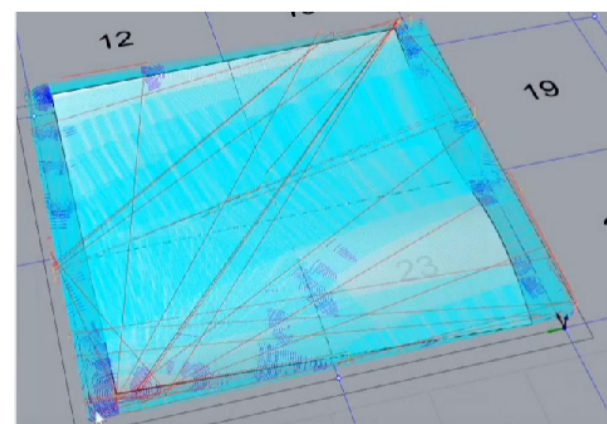
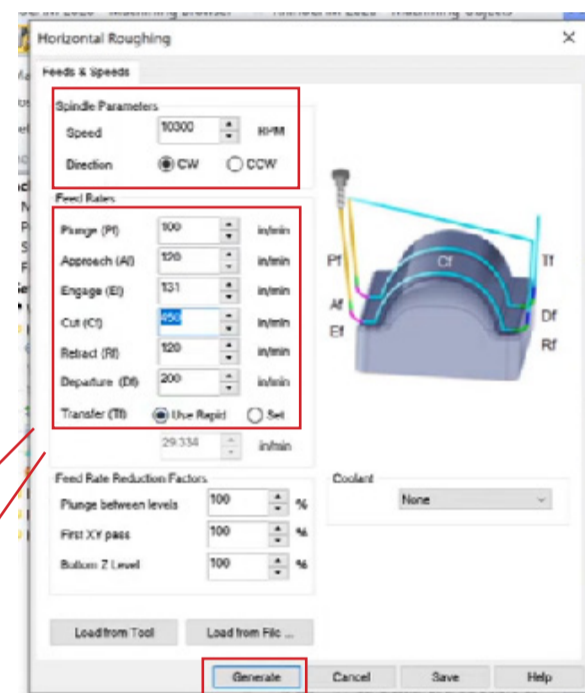
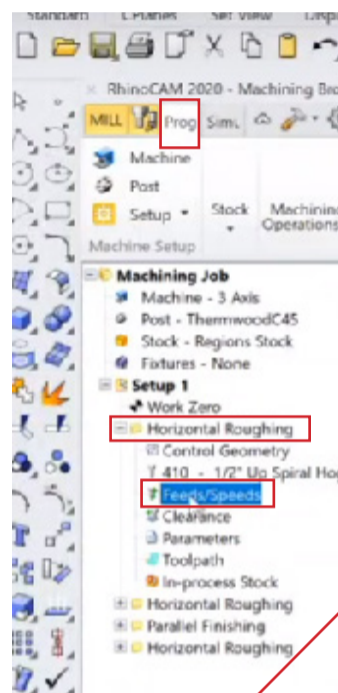
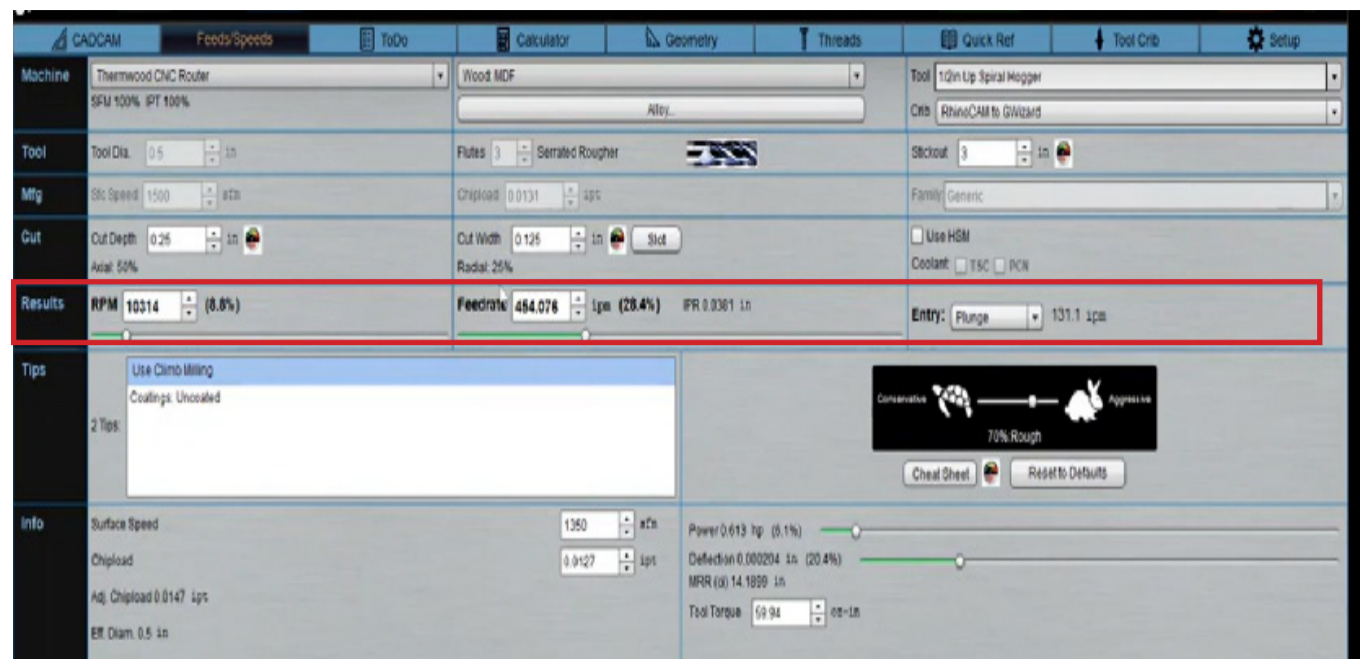
- Set the plunge to at least 100 and the departure to at least 200.

- The "Transfer" refers to the red lines you see on the geometry when the toolpath is turned on, which represent the transitions between different cuts in the toolpath.

- In the "Feederate" section, there are two options for the transfer: "Rapid" (which uses a built-in speed for moving from point to point without cutting) or "Feed" (which follows the feed rate specified for cutting). You can set the transfer to "Rapid" if desired.

- Once you have finished setting the feed and speed sections in RhinoCAM, click "Generate."

- After clicking "Generate," the toolpath will be recalculated, updating all the associated codes to reflect the entered speeds without changing the geometry being followed for the toolpath.



- You can change how the machine contacts or cuts the material by double-clicking on the first roughing path and accessing the “Engage Retract” tab.

- The “Plunge” in G Wizard corresponds to the vertical approach in RhinoCAM, so enable the vertical approach in the “Engage Retract” section.

- There is also a “Ramp” option under the “Engage Retract” section, offering different types of ramps that move along the cut path at an angle until reaching the cut depth.

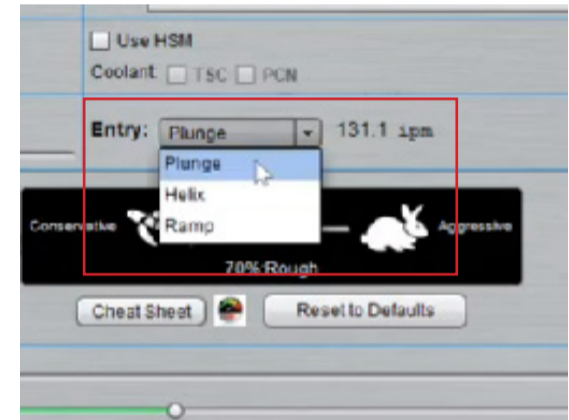
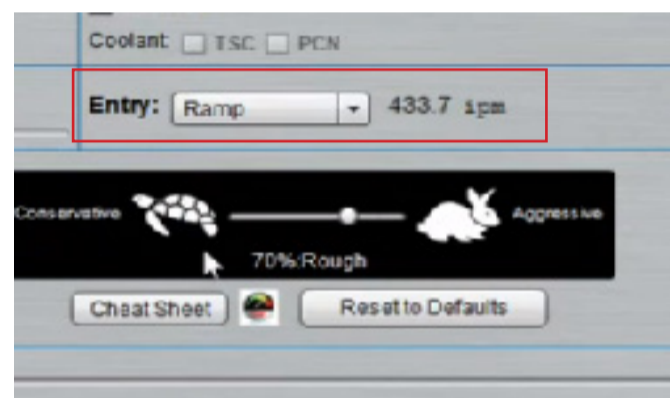
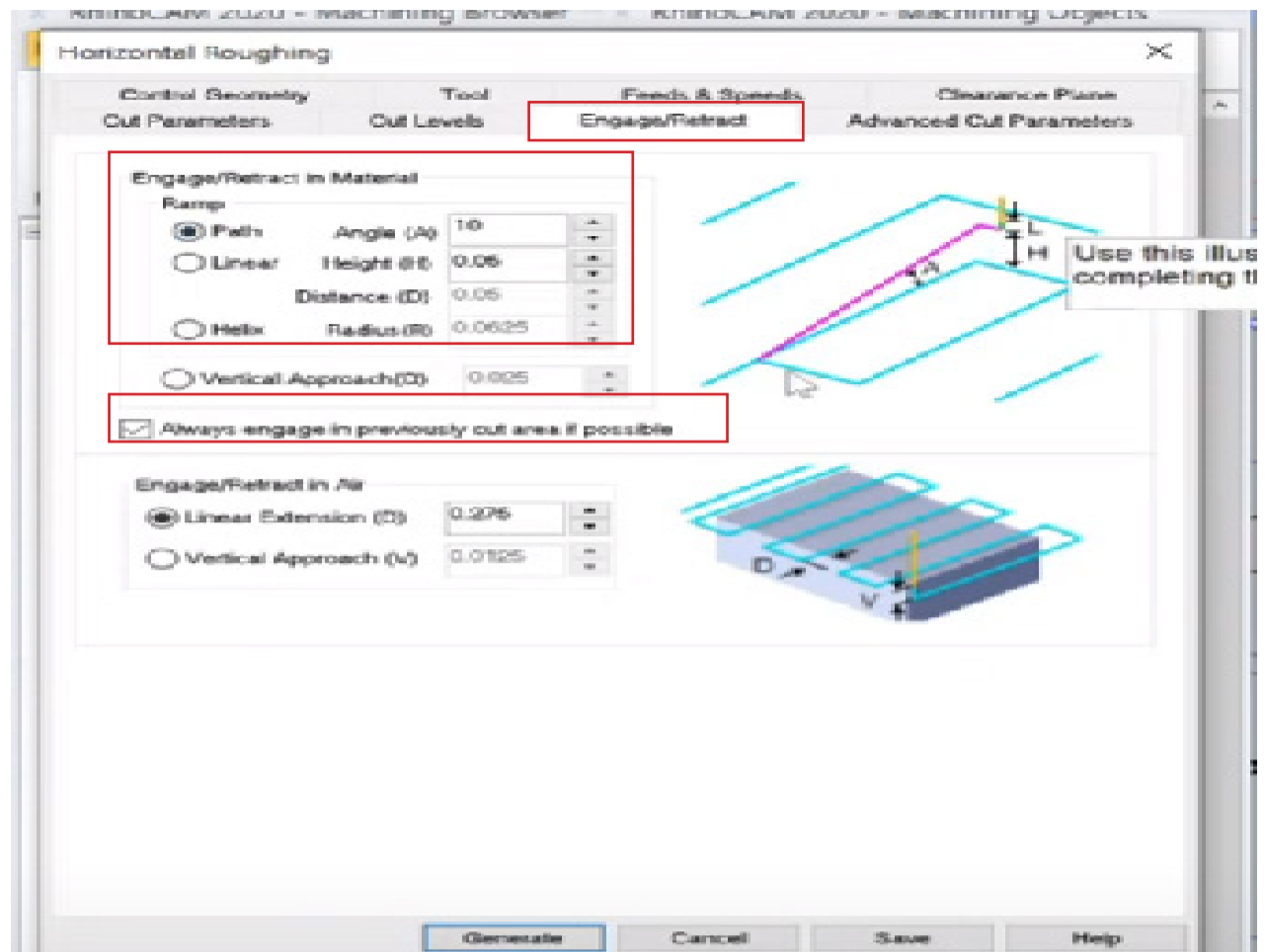
- The ramp options in G Wizard correspond to the ramp options in RhinoCAM.

- Ramping allows faster entry into the material compared to straight plunging.

- It’s important to note that router bits, which have flat ends, move laterally and require slower speeds, while drill bits, which have sharp ends, move vertically and require slower speeds as well.

- The ramp option is particularly useful when using ball nose bits, which are not designed for plunging.

- In RhinoCam, enable the “Always engage in previously cut area if possible” option to avoid entering the material in new places.



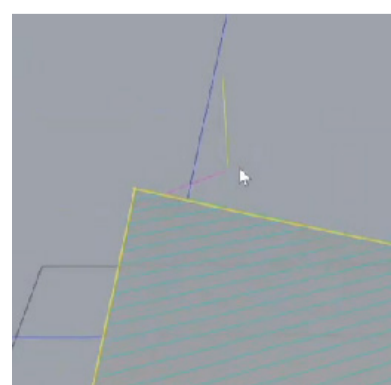
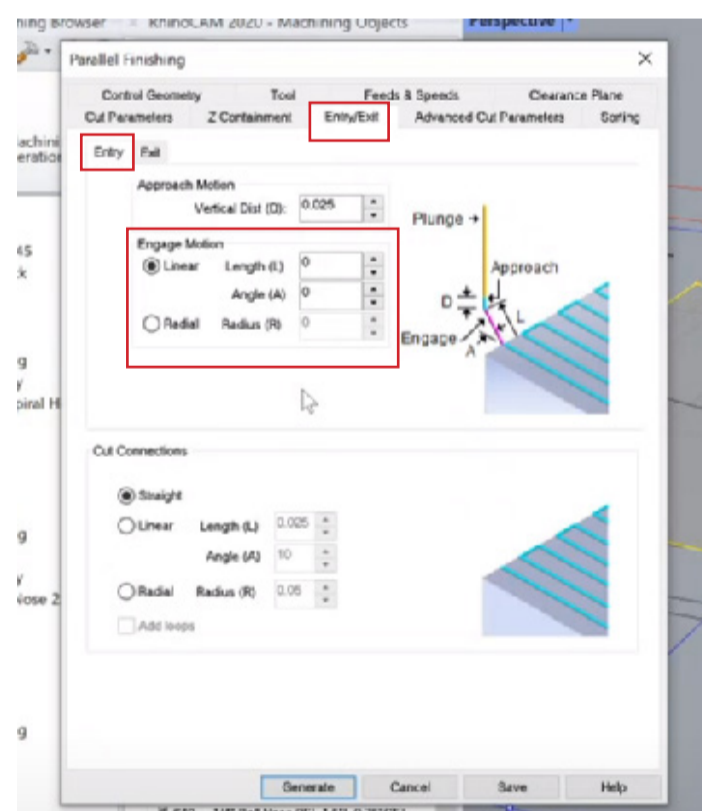
b. Third toolpath: Parallel Finishing

- Click on the “Parallel Finishing” toolpath and compare the “engage retract” in the first tool path (horizontal roughing path) with the second toolpath (parallel finishing).

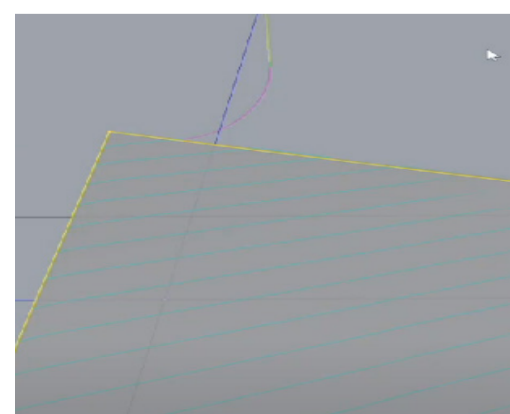
- You will notice that it is called “Engage Retract” in the roughing toolpath and “Entry/Exit” in the parallel finishing path.

- Click on the “Entry” tab within the entry/exit section, and you will see an image showing the plunge before engaging any material. The blue path represents the approach, while the engage area is where the tool actually cuts into the material.

- The engage motion can be changed to “Linear” (a straight line that angles into the material) or “Radial” (a curved line that angles into the material). You can also set it to “Radial with 0.025 radial” and generate to compare different options.



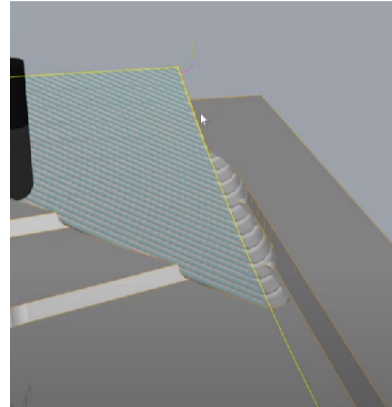
Linear



Radial

- To check the result, simulate each toolpath starting from the first roughing path, then the second roughing path, followed by the parallel finishing path, and finally the third roughing path.

- During the parallel finishing simulation, you may encounter scenarios where the bit cuts through the outside of the toolpath. The intention is to preserve the outside wall and not affect the material outside of that surface.



- To change this, go back to the programming tab, double-click on the parallel finishing toolpath, and go to the "Control Geometry" section. There are two options: "Part Region" and "Part Surface."

- First of all, click the "part regions" at the top and click "remove all."

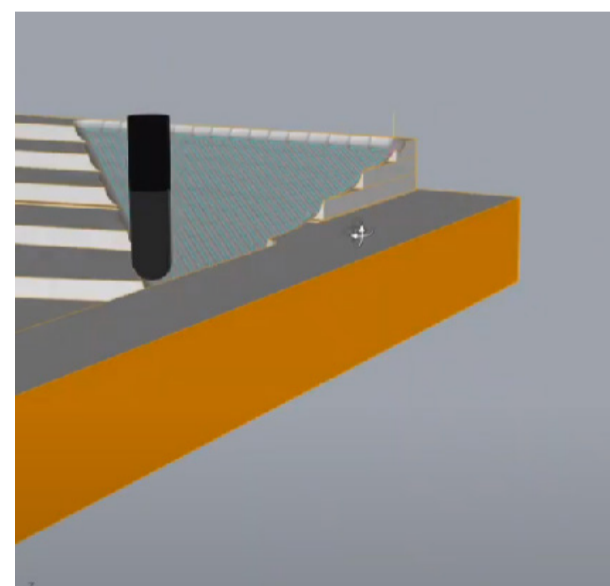
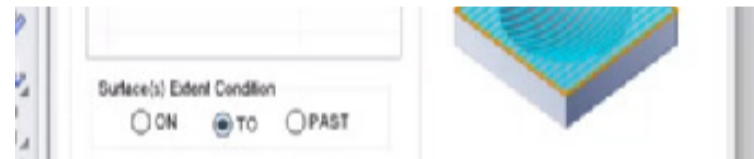
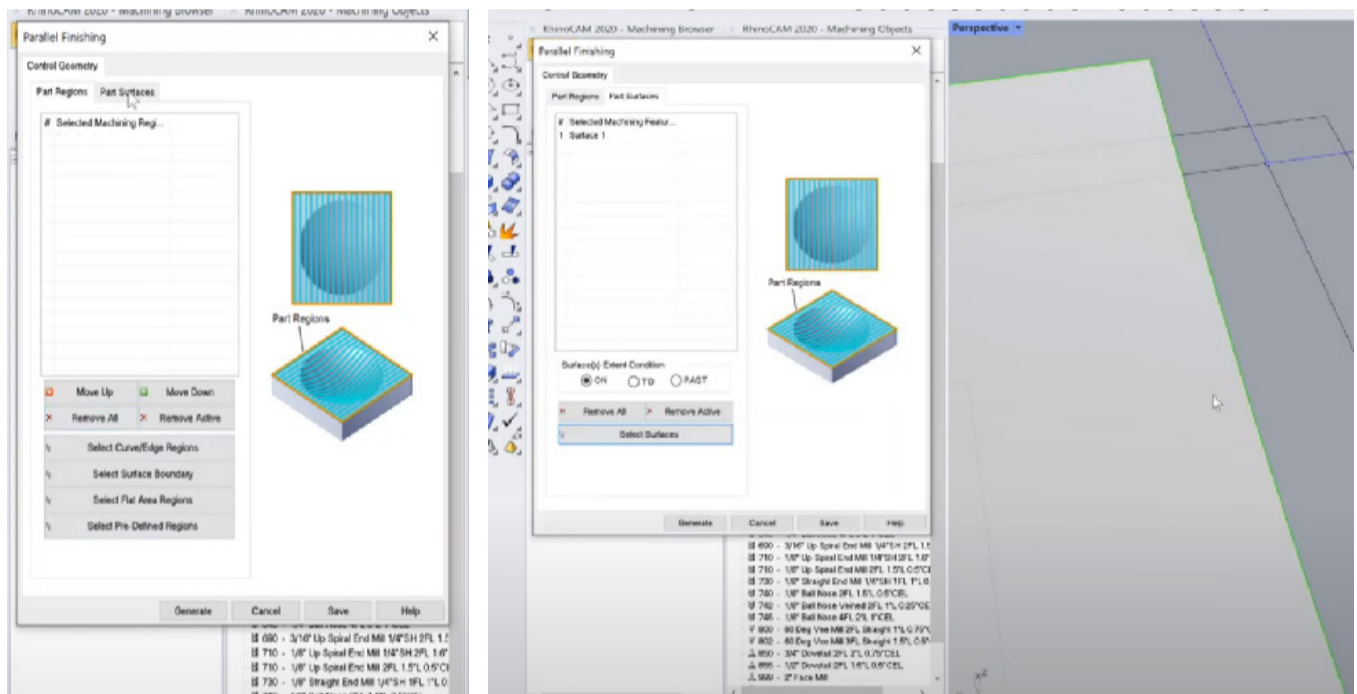
- Then select the "Part Surface" option at the top, and click "select surfaces."

- Select the surface of your geometry.

- Then explore the three different options: "On," "To," and "Past." Each option controls the surface extent condition when cutting. Click generate when done selecting.

- The "To" option is likely the one you need to use, as it will ensure that the bit never exits the outside of the surface you want to cut.

- Right click on the parallel finishing toolpath, and simulate again to check the result, and you will see that it does not exceed the surface boundary this time.



c. Feed and speed for parallel finishing path:

- Go back to G Wizard to set up the feed and speed for the parallel finishing path.

- In G Wizard, go to the "Tool crib" and select the exact tool used in RhinoCam (440 half-inch ball nose for this module) for the parallel finishing. Then click "OK."

- The stickout has changed to 2 inches. The length listed in RhinoCam refers to the length of the bit sticking out. It should be called "stickout" instead of "length" to avoid confusion. Stickout is important to ensure there are no collisions during simulation.

- The cut depth for parallel finishing varies because of the stair-stepping from the horizontal finishing path. To be conservative, you can set the cut depth to 0.25 inches or use 0.125 inches, especially with MDF.

- In Rhinocam the stepover for parallel finishing (under cut parameter) was set to 25% of the tool diameter (which is 25% of half-inch bit diameter), equivalent to 1/8 inch. In G Wizard, set the cut width to 0.125 inches.

- Increase the "conservative to aggressive" value to 100%.

- Set the values (RPM, Feedrate) in G Wizard for the parallel finishing. Go back to RhinoCam and open the feed and speed section for parallel finishing.

- Copy and paste the RPM generated in G Wizard to the Spindle speed (RPM) input box. You can round up the value. The value we got here is 13,200

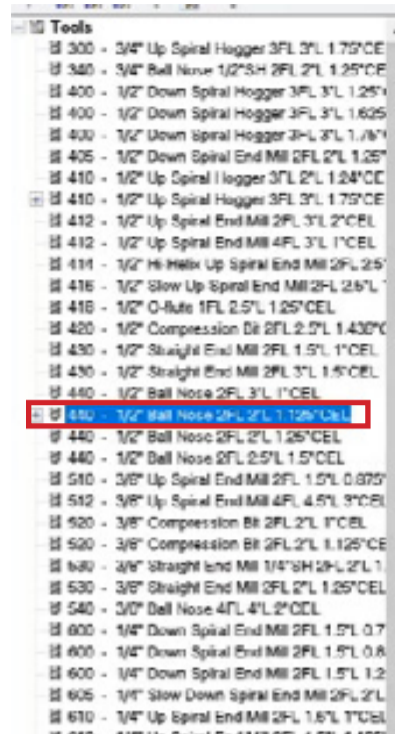
- Assume: Set the plunge to 100, and the approach to 120.

- For the engage, check the ramping value in G Wizard and copy-paste it into the Engage input box in RhinoCam. Here, it is approximately 370.

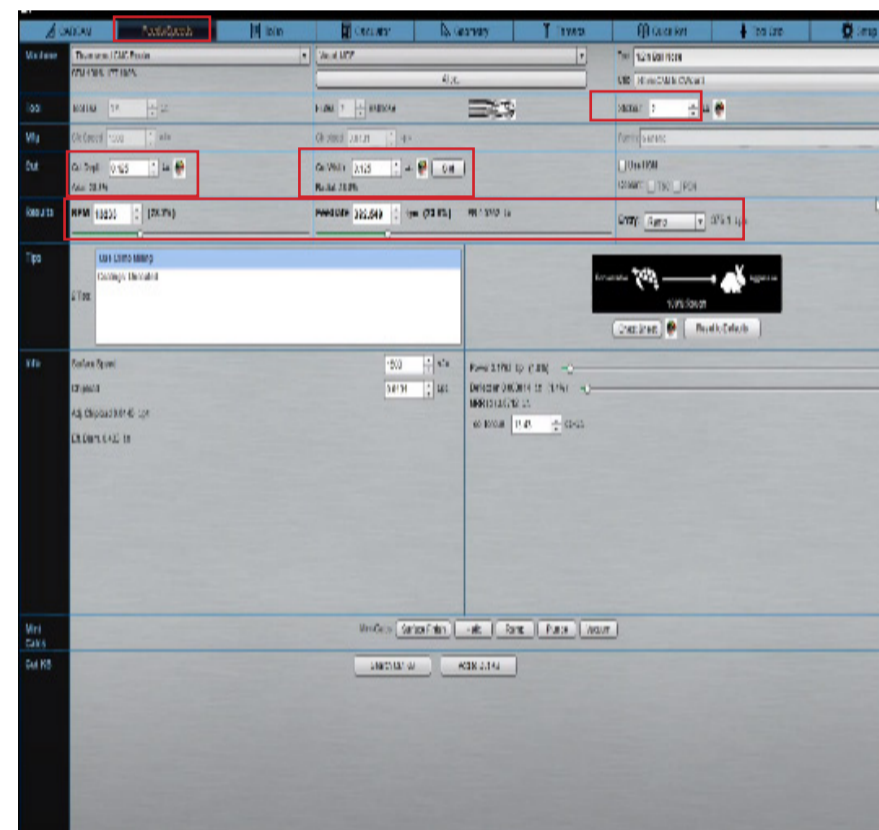
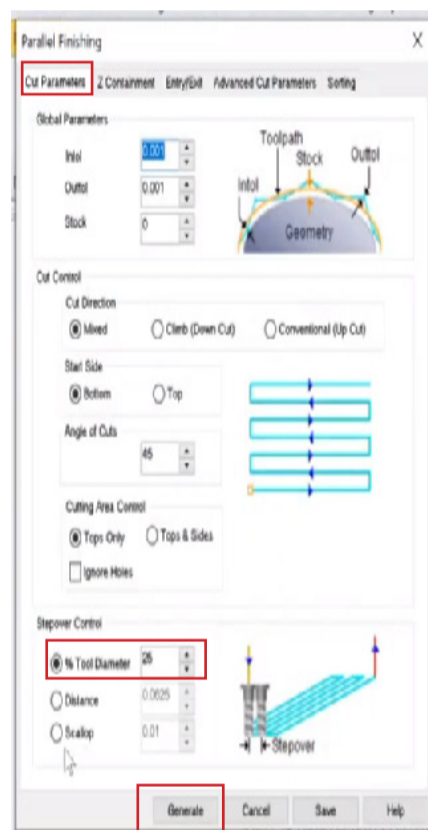
- For the "cut" input in RhinoCam, use the feed rate value generated in G Wizard. The value we got here is 380.

- Assume: Set the retract value to 100 and the departure to 200.

- Click "Generate."



T#	Type	Description	Comment	Diameter	Flutes	Stickout	Lead	New Rad.	Flute Len	Overall Len
410	Carbide End Mill	1/2in O-Ride		0.5	1	2.5	90	0	1.25	0
420	Carbide Router	1/2in Compression Bit		0.5	2	2.5	90	0	1.438	0
430	Carbide Router	1/2in Straight End Mill		0.5	2	1.5	90	0	1	0
430	Carbide Router	1/2in Straight End Mill		0.5	2	3	90	0	1.5	0
440	Carbide Router	1/2in Ball Nose		0.5	2	2.5	90	0	1.5	0
440	Carbide Router	1/2in Ball Nose		0.5	2	2	90	0	1.125	0
440	Carbide Router	1/2in Ball Nose		0.5	2	2	90	0	1.25	0
440	Carbide Router	1/2in Ball Nose		0.5	2	3	90	0	1	0
510	Carbide End Mill	3/8in Up Spiral End Mill		0.375	2	1.5	90	0	0.875	0
512	Carbide End Mill	3/8in Up Spiral End Mill		0.375	2	4.5	90	0	3	0
520	Carbide Router	3/8in Compression Bit		0.375	2	2	90	0	1	0
520	Carbide Router	3/8in Compression Bit		0.375	2	2	90	0	1.125	0
530	Carbide Router	3/8in Straight End Mill 1/4in Shank		0.375	2	2	90	0	1	0

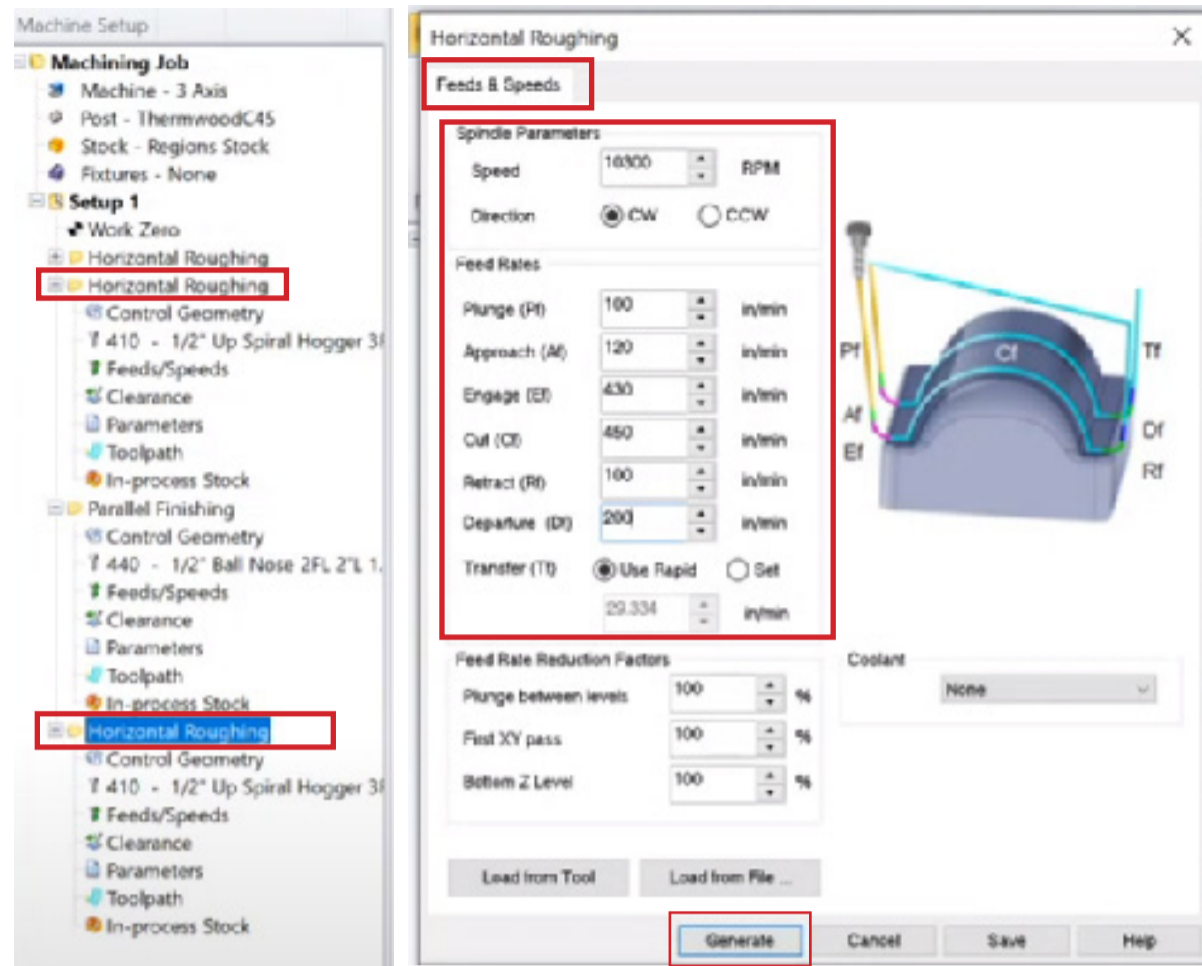


d. Second and Fourth (last) toolpath: Horizontal Roughing

- The feed and speed information previously entered in the first horizontal roughing toolpath will be used for the second and last horizontal roughing path. Make sure to write it down or have it recorded for reference.

- Repeat the process for the third horizontal roughing path which is the fourth toolpath.

- At this point, the feed and speed for every toolpath have been set.

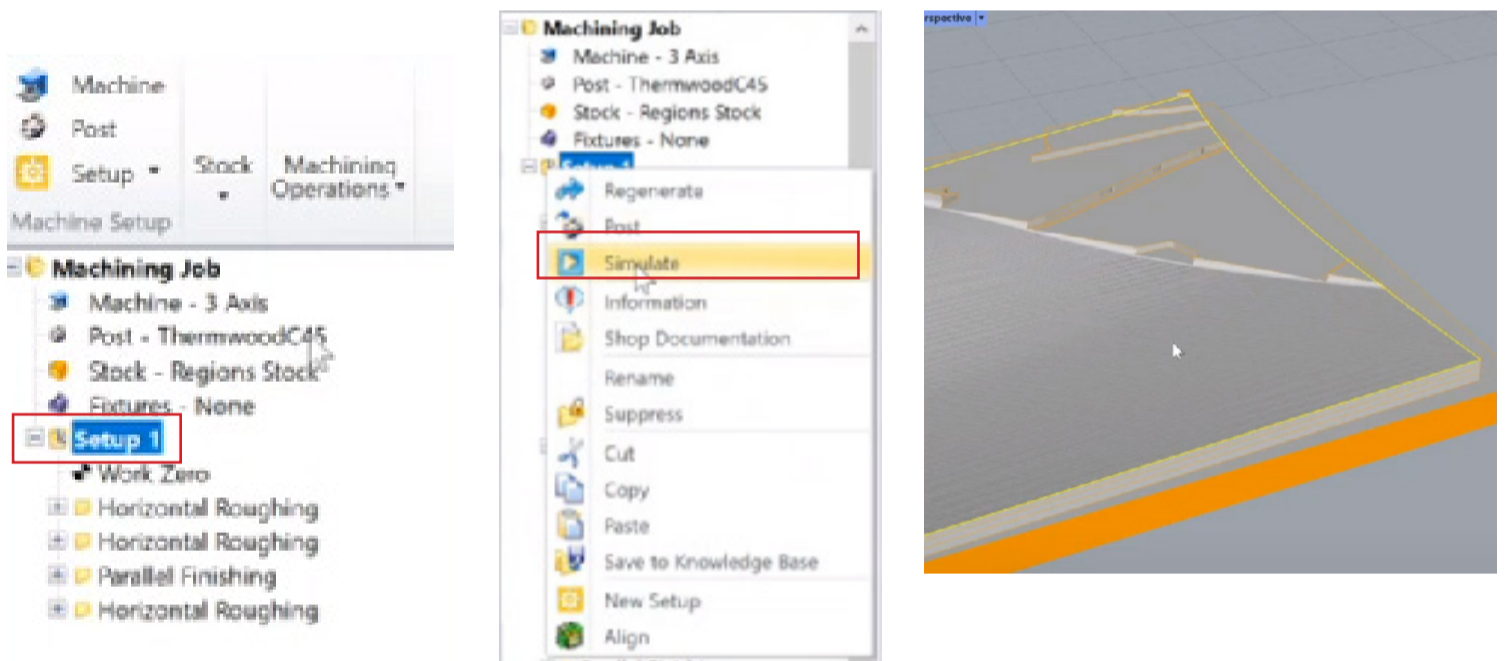


Simulation

- Simulate the entire toolpath by clicking on setup, then right-click and select simulate to go through all the toolpaths.

- To see a summary of all your toolpaths, including the tool number, right-click on the setup and select information. It will also show you the estimated total time for the entire job.

- Once you have set up all the feeds and speeds, confirmed the tool bit, and ensured there are no collisions during simulation, you can send the code to the machine.



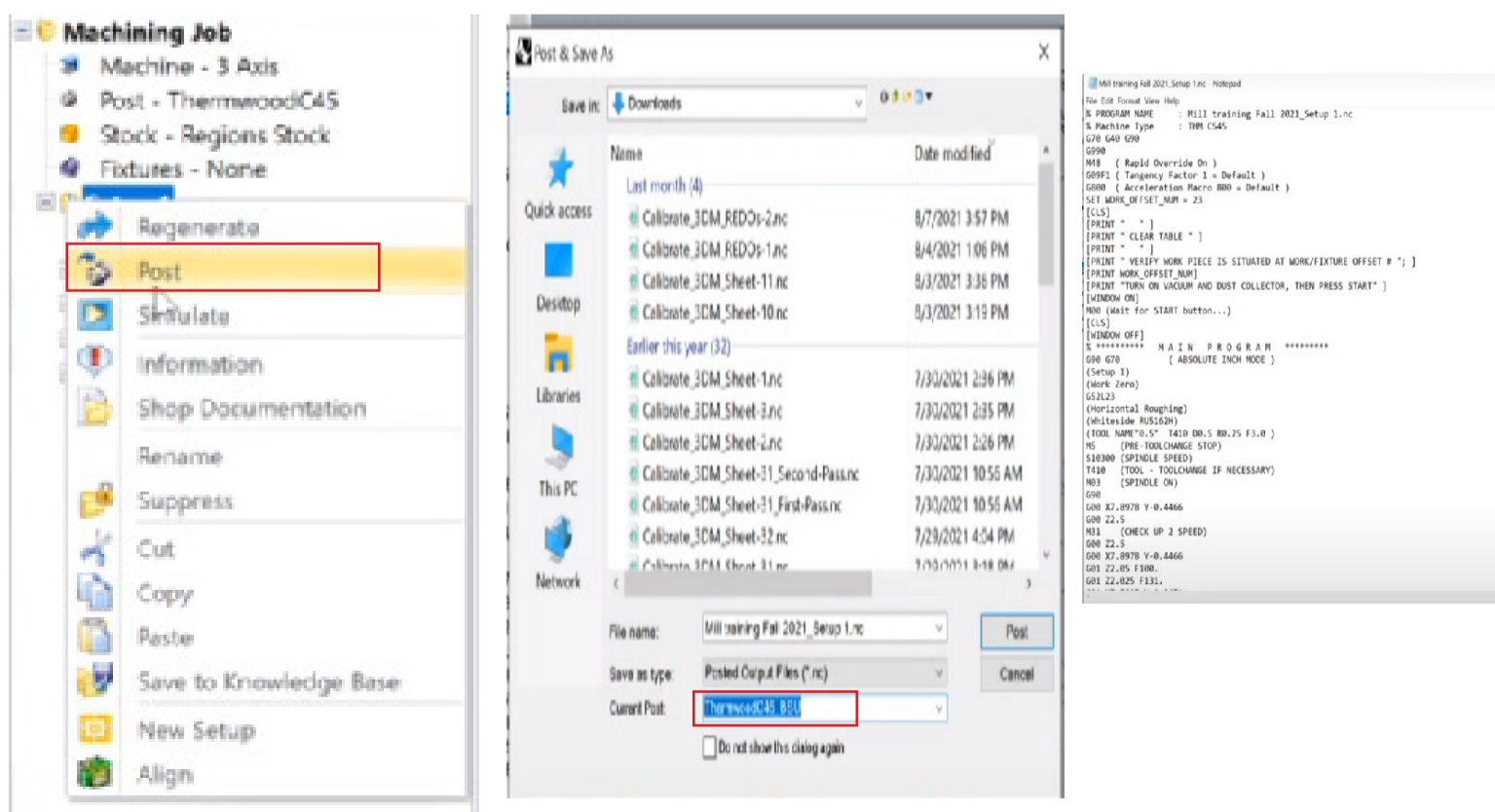
E. POST G-code

- The final step is to post the code. Click on the setup (make sure it is selected), right-click, and select "Post."

- A new window will appear, which generates the G code based on the machine selection made at the beginning of the module (e.g., Thermwood C45). Click the "Post" button.

- The code will be processed and converted to G code. It will show you the actual code that will be sent to the machine.

- The code will open automatically after posting, else open the location where you posted and saved the G code. All the setup and input information will be written in the code.

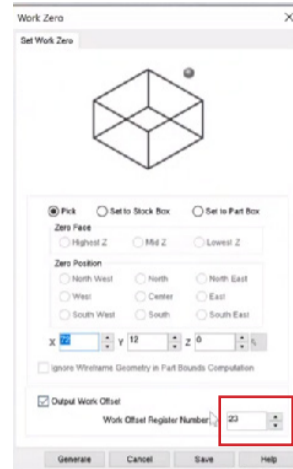
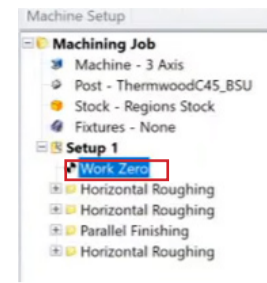


F. CHECK:

- One important thing to check in the G code is the Work offset number, which must be correct.
- Also, double click the work zone to see if you input the right work offset.

```

Mill training Fall 2021_Setup 1.nc - Notepad
File Edit Format View Help
% PROGRAM NAME      : Mill training Fall 2021_Setup 1.nc
% Machine Type      : THM CS45
G70 G40 G90
G990
M18 ( Rapid Override On )
G09F1 ( Tangency Factor 1 = Default )
G800 ( Acceleration Macro 800 = Default )
SET WORK OFFSET NUM = 23
[CLS]
[PRINT " "]
[PRINT " CLEAR TABLE " ]
[PRINT " "]
[PRINT " VERIFY WORK PIECE IS SITUATED AT WORK/FIXTURE OFFSET # "; ]
[PRINT WORK_OFFSET_NUM]
[PRINT "TURN ON VACUUM AND DUST COLLECTOR, THEN PRESS START" ]
[WINDOW ON]
M00 (Wait for START button...)
[CLS]
[WINDOW OFF]
% *****  M A I N  P R O G R A M  *****
G90 G70      ( ABSOLUTE INCH MODE )
(Setup 1)
(Work Zero)
G52L23
(Horizontal Roughing)
(Whiteside RUS162H)
(TOOL NAME"0.5"  T410 D0.5 R0.25 F3.0 )
M5 (PRE-TOOLCHANGE STOP)
S10300 (SPINDLE SPEED)
T410 (TOOL - TOOLCHANGE IF NECESSARY)
M03 (SPINDLE ON)
G90
G00 X7.8978 Y-0.4466
G00 Z2.5
M31 (CHECK UP 2 SPEED)
G00 Z2.5
G00 X7.8978 Y-0.4466
    
```



- Another thing to check is to go to the "Edit" menu on your G-code window and click "Find." Type "Z-" in the input box. If there is a Z- in the code, it means it is cutting into the table. Check this to ensure you are not accidentally cutting too deep.

- Z zero refers to the table height, and anything below Z zero or Z- will cut into the table to some degree.

- Once you have checked all these aspects, load the G code onto your USB or external device and take it to the machine. Load the code onto the machine to run it.

```

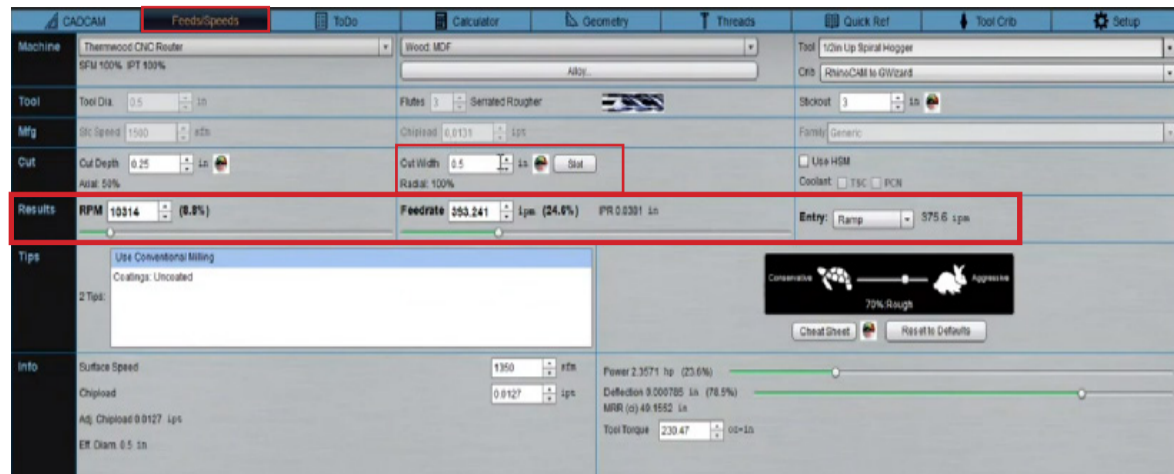
Mill training Fall 2021_Setup 1.nc - Notepad
File Edit Format View Help
% F      Undo          Ctrl+Z
% F      Cut           Ctrl+X
G70     Copy          Ctrl+C
G90     Paste         Ctrl+V
M18     Delete        Del
G800    Search with Bing... Ctrl+E
SET      Find...      Ctrl+F
[CLS]   Find Next     F3
[PRINT] Find Previous  Shift+F3
[PRINT] Replace...     Ctrl+H
[PRINT] Go To...       Ctrl+G
[PRINT] Select All     Ctrl+A
[PRINT] Time/Date      F5
[WINDOW ON]
M00 (Wait for START button...)
[CLS]
[WINDOW OFF]
    
```


A. ESSENTIAL TOOLPATH FOR PROPER MILLING

i. Axial profiling toolpath

For the 2 axis profiling toolpath, follow these steps:

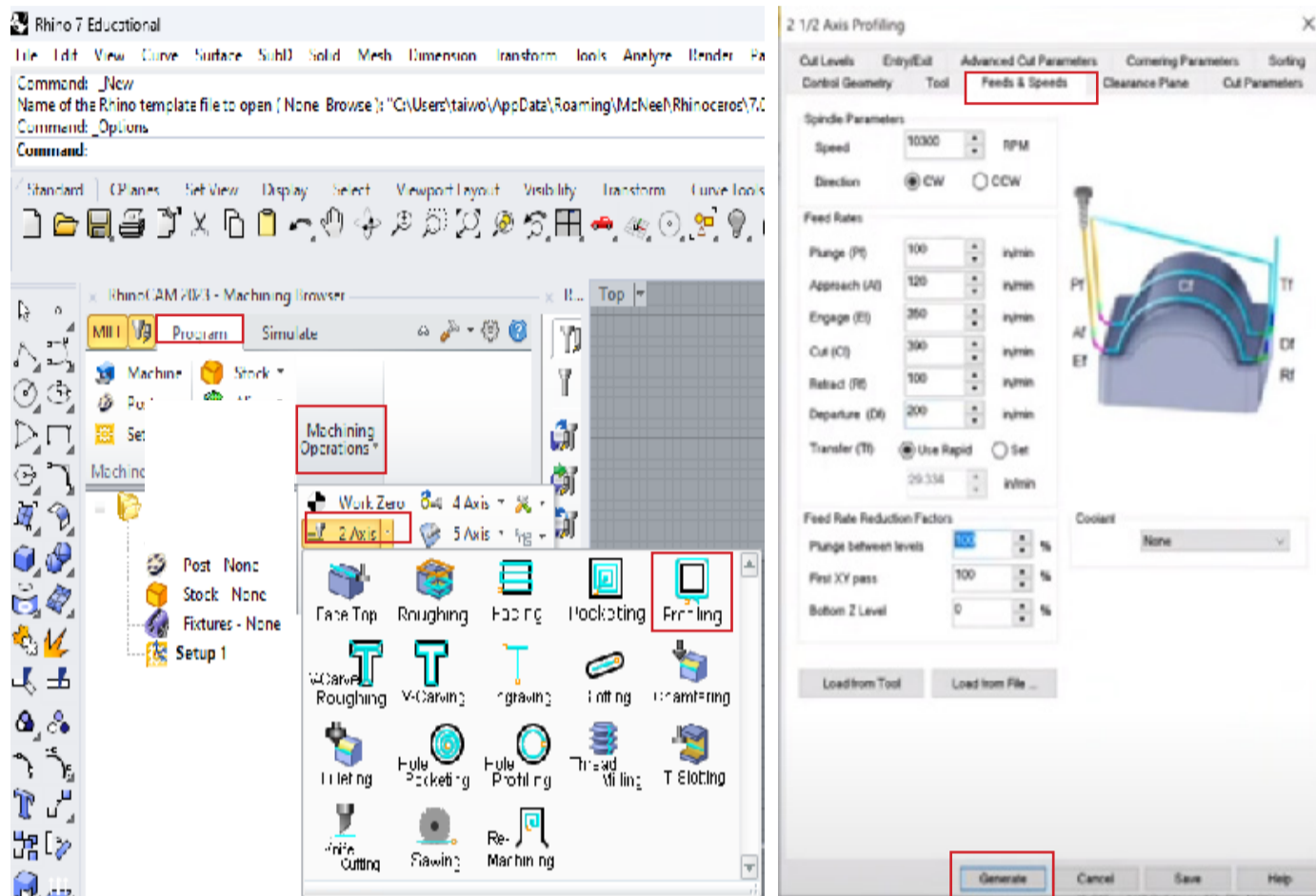
- Go to GWizard and under “Feeds and Speeds,” set the slot to 100% for the cut width. This will automatically adjust the feed rate and RPM.



- Return to RhinoCam and go to “Program,” then “Machine Operations,” and click on “Profiling.”

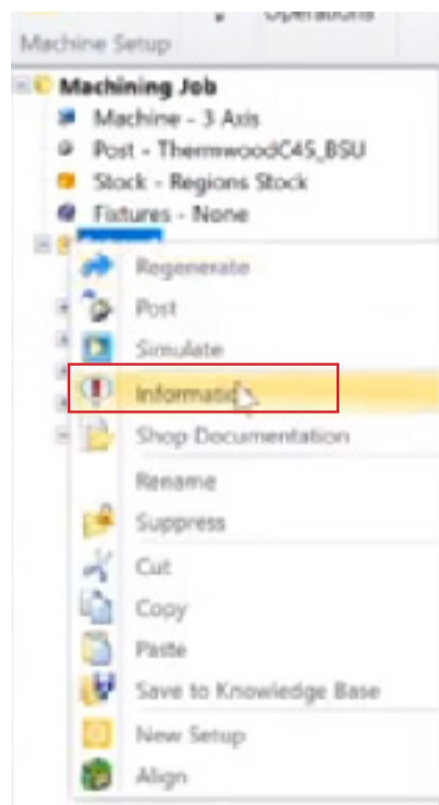
- An Axial Plotting Profile window will open.

- Go to the “feed and speed” section.
 - Fill in the information generated from GWizard in the cut level section.
 - For the cut, use the feed rate generated in GWizard (approximately 390) and the spindle speed (approximately 10,300 RPM).
 - For the engage value, use 350, which is the entry section value in GWizard.
 - Assume the input values for the rest of the parameters: Plunge: 100, Approach: 120, Retract: 100, Departure: 200.



- Once you have filled in the information, click “Generate.”

- Before proceeding, check the information about your job by right-clicking on the setup and selecting “Information.” This will display the machine time, which can be significantly reduced compared to cutting out all the material at the end.



Name	Stat	Tool	T...	Cut Fe...	# of GOTOs	Machine Time
Setup 1						
Work Zero	Clean	No ...	-	0.0		
Horizontal...	Clean	410 ...	410	450.0...	17704	1 hr 3 min
Horizontal...	Clean	410 ...	410	450.0...	1287	3.19 min
Parallel Fl...	Clean	440 ...	440	380.0...	38151	10.47 min
Horizontal...	Sup	410 ...	410	450.0...	1993	
2 1/2 Axis	Clean	410 ...	410	390.0...	91	1.08 min
Setup-Total						1 hr 18 min

- The most complex part of the 2D axis toolpaths is the “Cut Level” section. It can be confusing, especially when using the top two options, as it determines where the cut is made.

- Make sure you understand the desired outcome, which is to cut a slot following the edge of the final geometry rather than removing all the material around the edge.

a. Cut Level

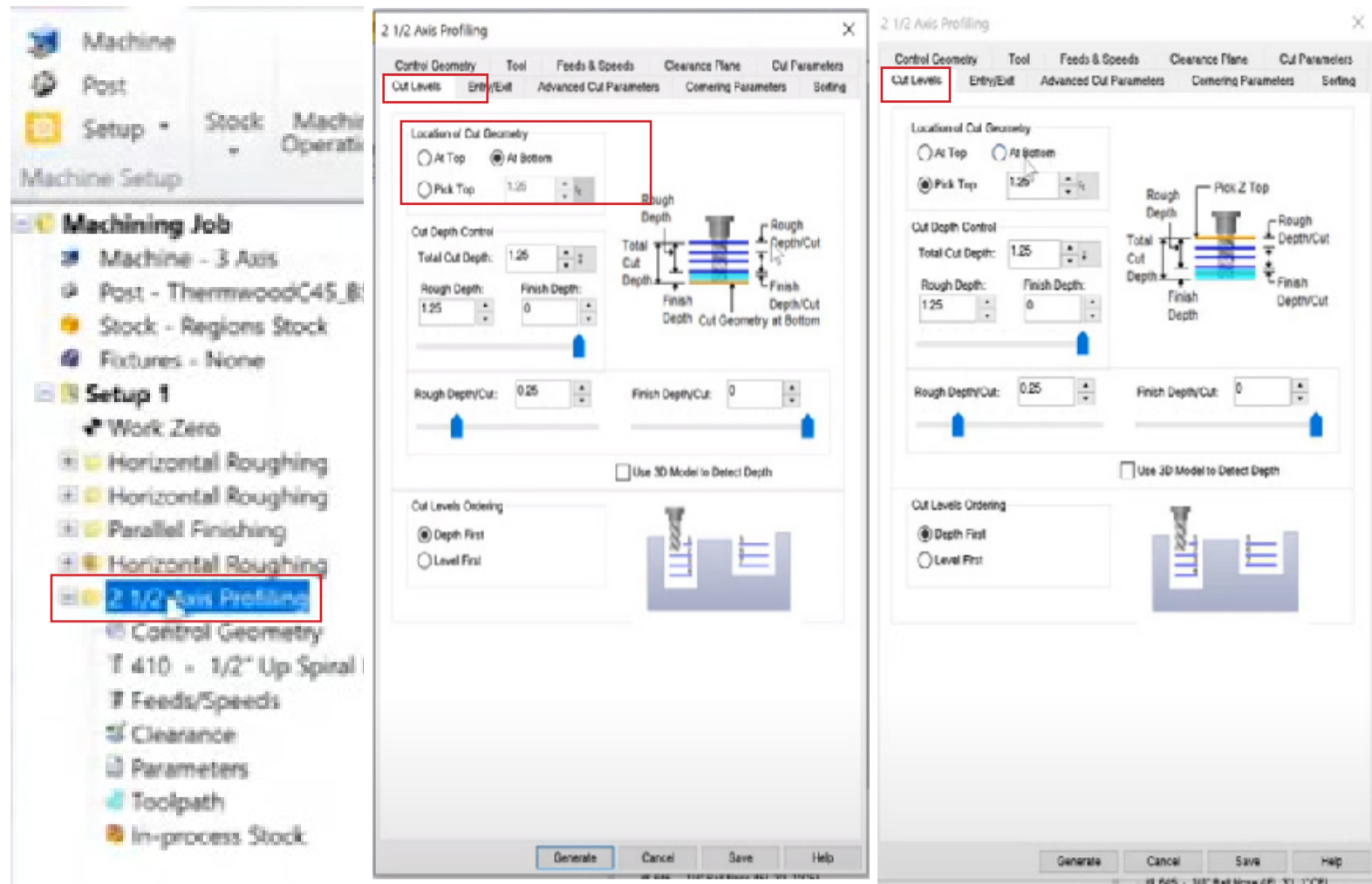
- Double-click on your axial plotting toolpath and click on the "Cut Level" section.

- Here you will find three options for the location of the geometry.

- If you navigate to the "At Bottom" option, check the image beside it, which shows that the cut geometry is at the bottom, and the cut depth is going to be above your cut geometry.

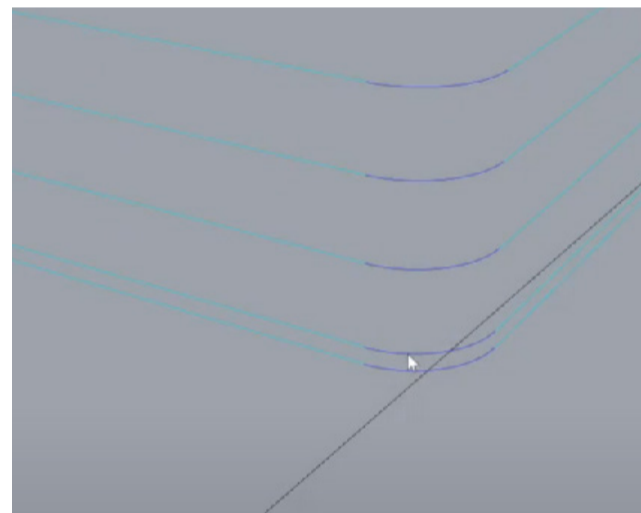
- The "Pick Top" option is the safest because you have more control, and it represents the top of your cut.

- Under the "Finished Depth Control" option, you can separate the finished depth or split up the total cut depth into a rough depth and a finished depth. Let's say the finished depth is 0.05 inches.



- This could theoretically divide it into multiple step-downs.

- Try to zoom in, and you will notice that it adds some extra levels here.

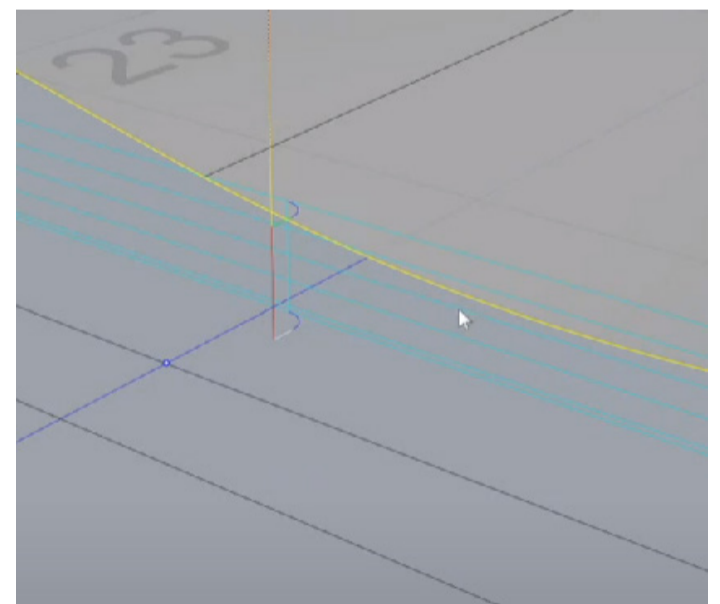
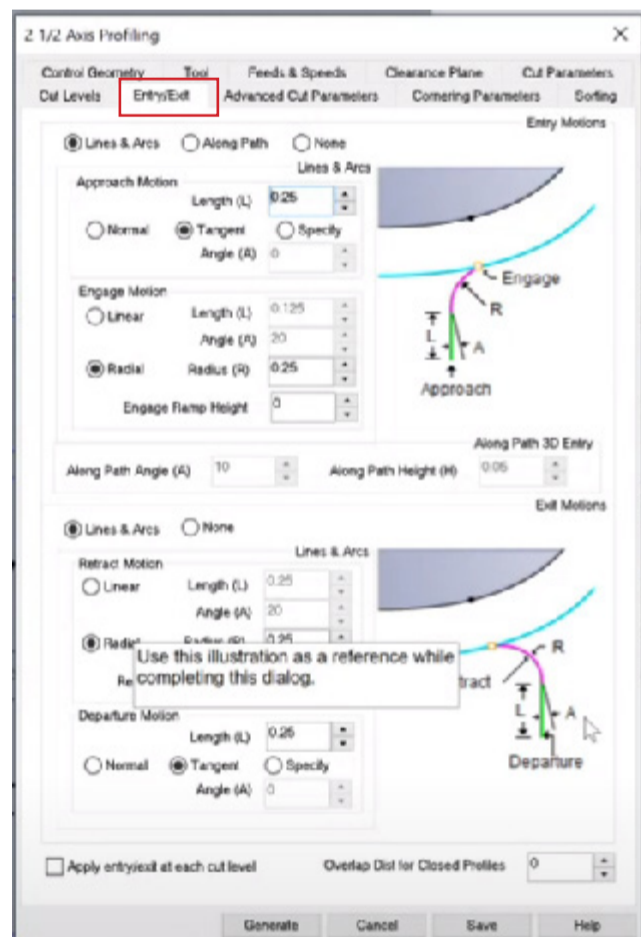


- Another thing to look at in this axial profile toolpath is the start and end of the cut.

- Double-click on the axial profiling toolpath and click on the "Entry/Exit" section.

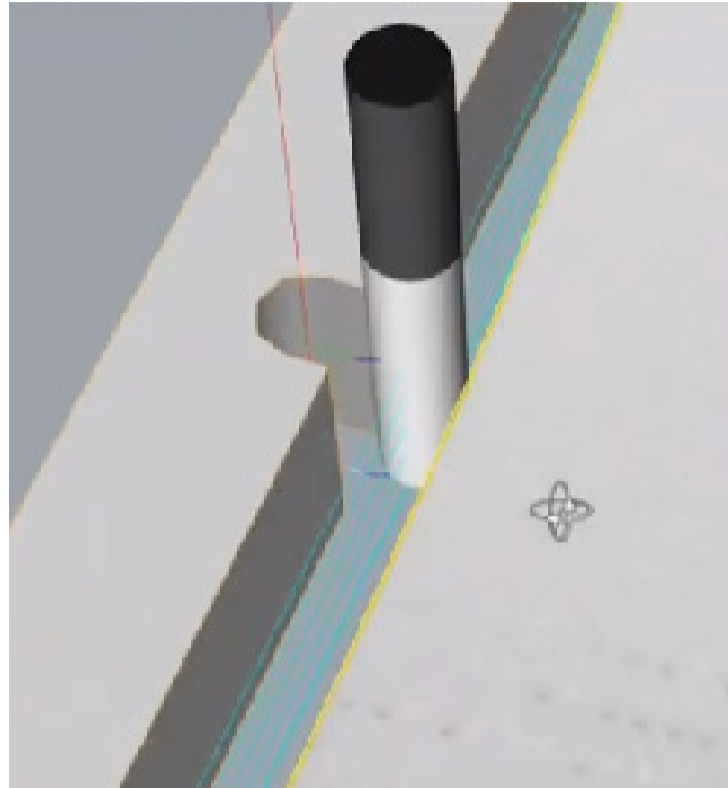
- You will see two images that explain how it curves into the cut as it goes around the material and how it curves out of the cut as it leaves the material.

- Check the image on the right side of this page for illustration, and look closely, you can see the start and end where the top is curving into the cut and at the bottom, it's curving out.



- Try to simulate this toolpath by right-clicking on the axial toolpath.

- You will notice that at the end, it is curving out into material that has not been cut; at the end there, you've got this big chunk that's cut out where it curved in and curved out.



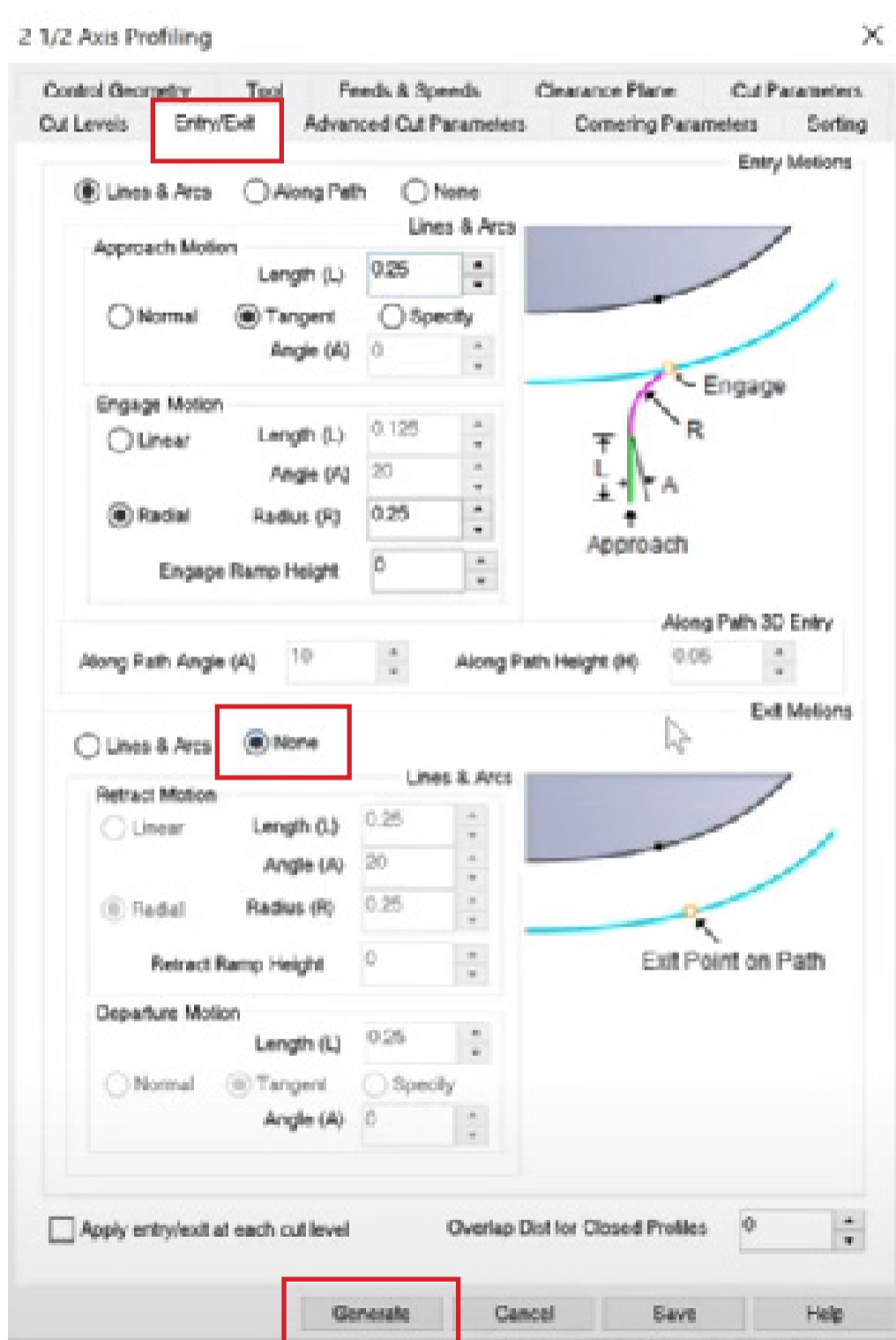
- This can be a problem and cause damage. For instance, if you were working with hardwood and you're moving really fast and you're exiting the material and cutting a full inch at once, that could cause tool breakage or dislodge the material.

- This default setting, which could cause problems, can be avoided.

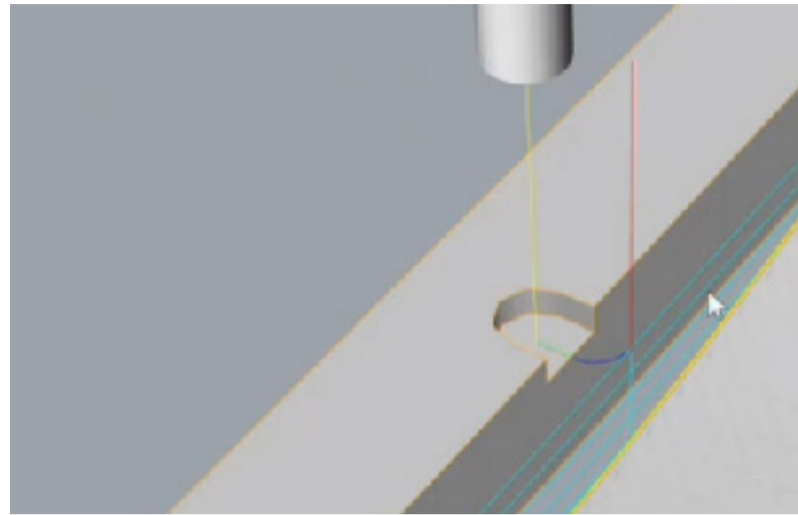
- The simplest method to fix it is to go back to the axial plotting toolpath, double-click it, and select the "Entry/Exit" option.

- Then disable the "none" icon here (check the image for illustration): This will ensure that the exit does not curve out of the material at the end.

- Try to generate it now. You will realize that the curve at the bottom level is removed.



- Now, simulate by right-clicking the axial toolpath. You'll see that it only did that first quarter of an inch, but then at the end of the cut, it's just moving straight up out of the cut.



- What we finally use for this module: Go to the axial plotting toolpath, then click on the "Entry/Exit" section.

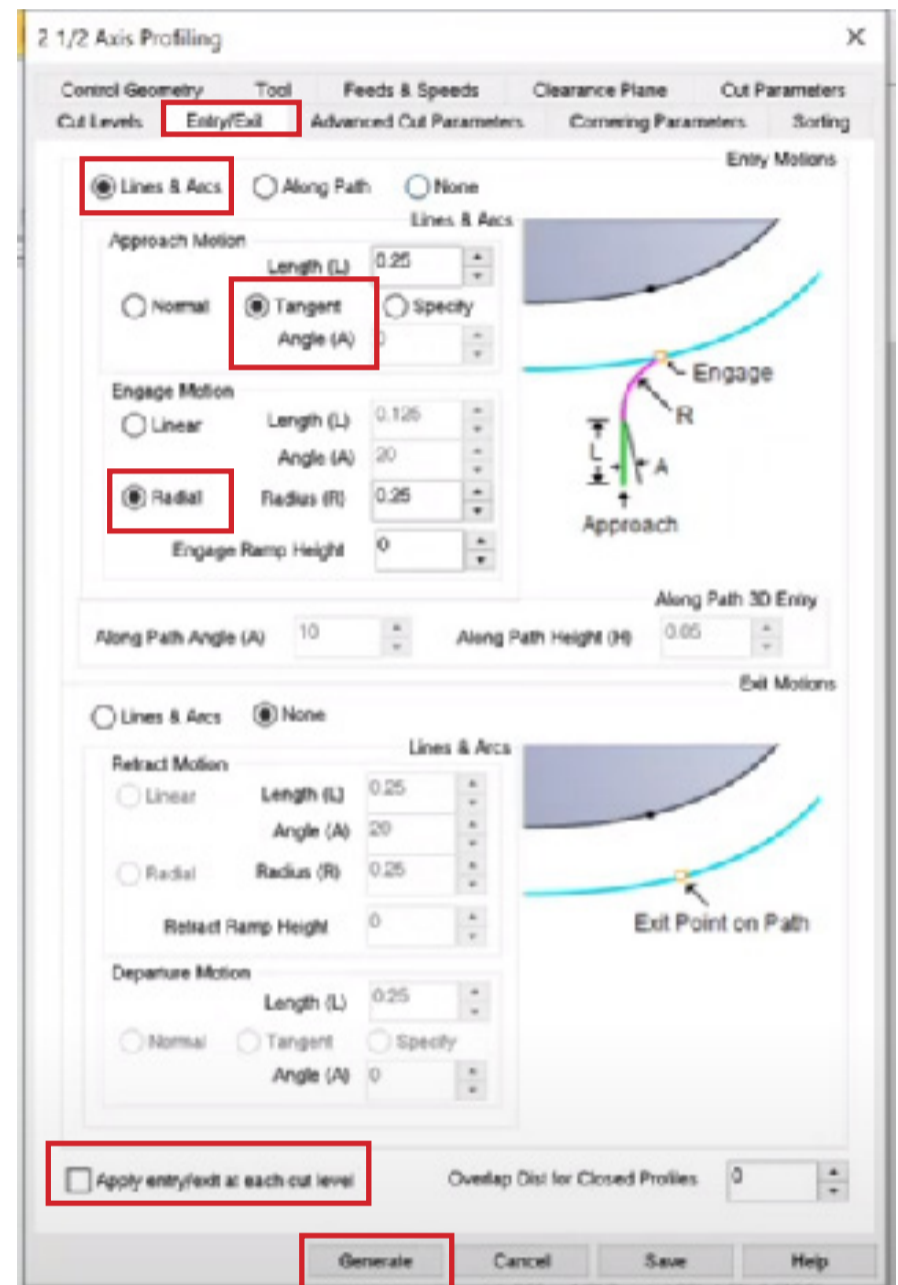
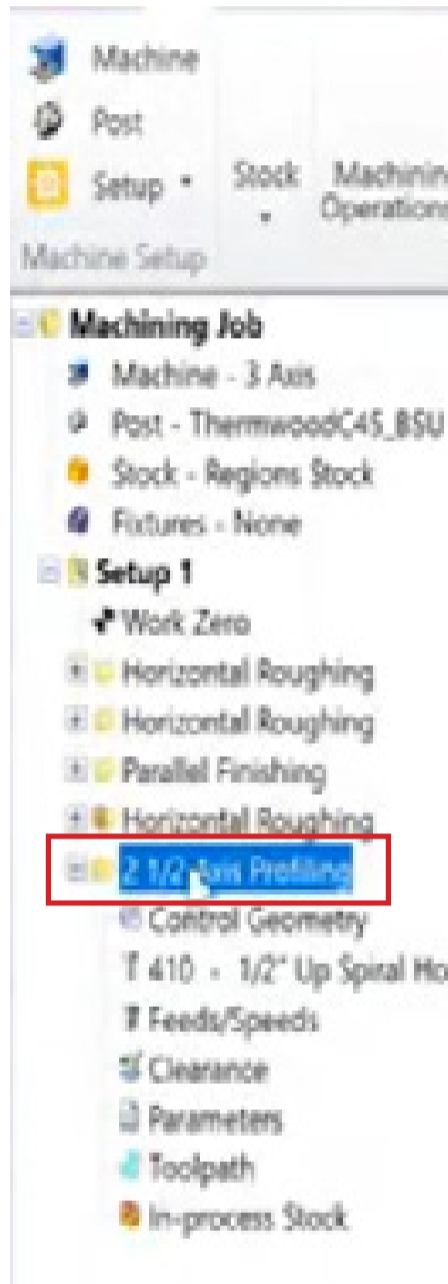
- Disable the tab below that says "Apply Entry/Exit at each cut level."

- Then click the option above "Line & Arcs," under the Engage motion section, click "Radial."

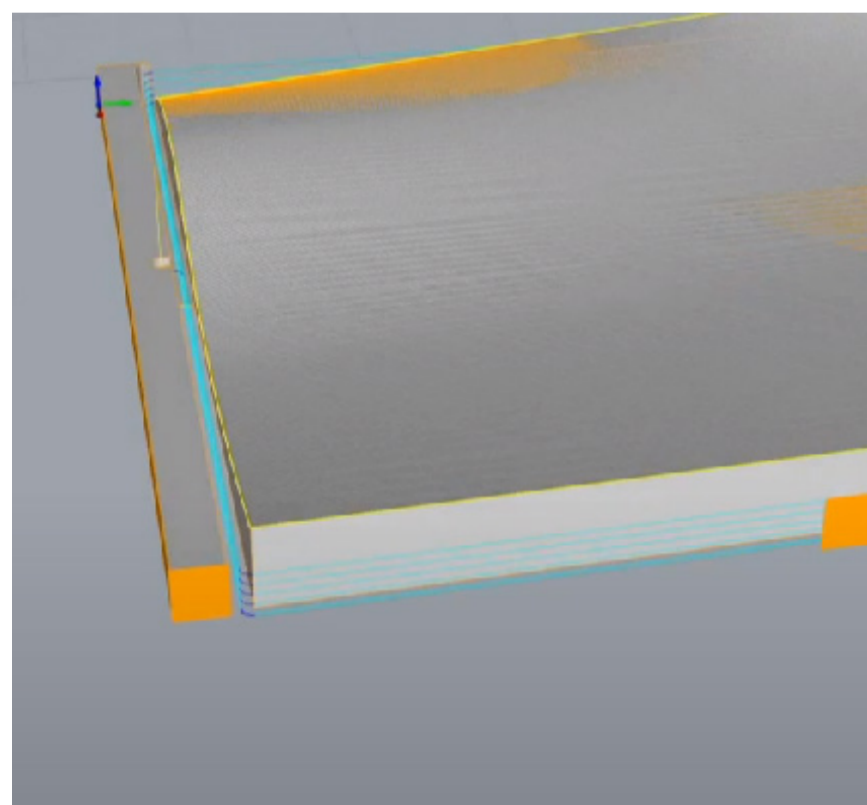
- Under the Approach motion section, click "Tangent."

- Then click "Generate."

- In situations where you're cutting on the outside and there's no remaining material, you might notice the outcome more. It ultimately depends on the cut direction you choose. So, I believe it's best to stick with whatever cut direction G Wizard recommends.



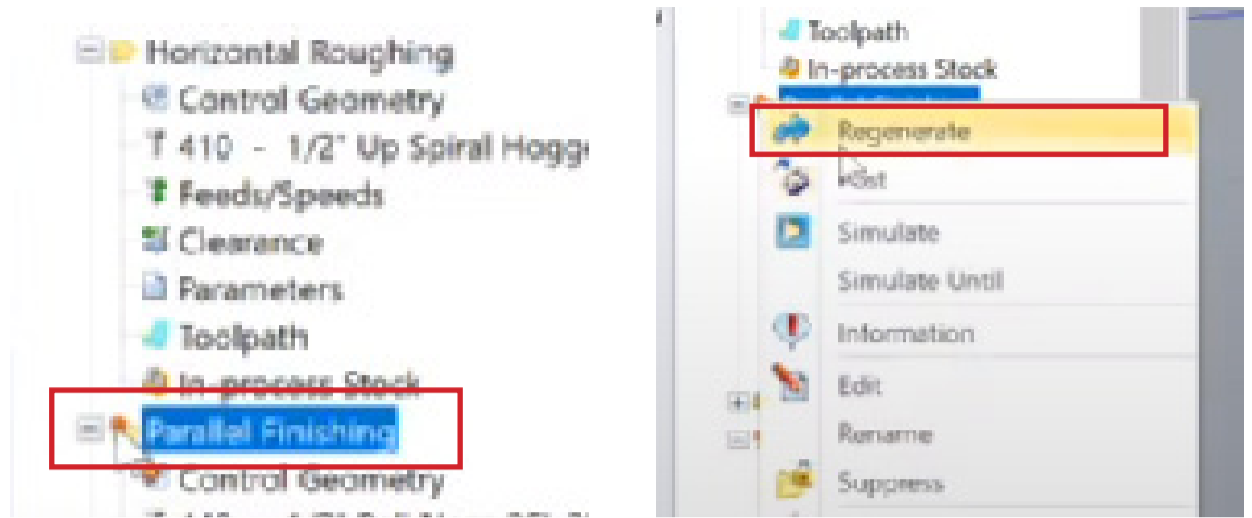
- Try to simulate the Axial toolpath to check the result.



COLLISION CHECKING

- We need to check by simulation if there is a collision in our geometry.

- The first step is to right-click the parallel finishing and click "Regenerate."



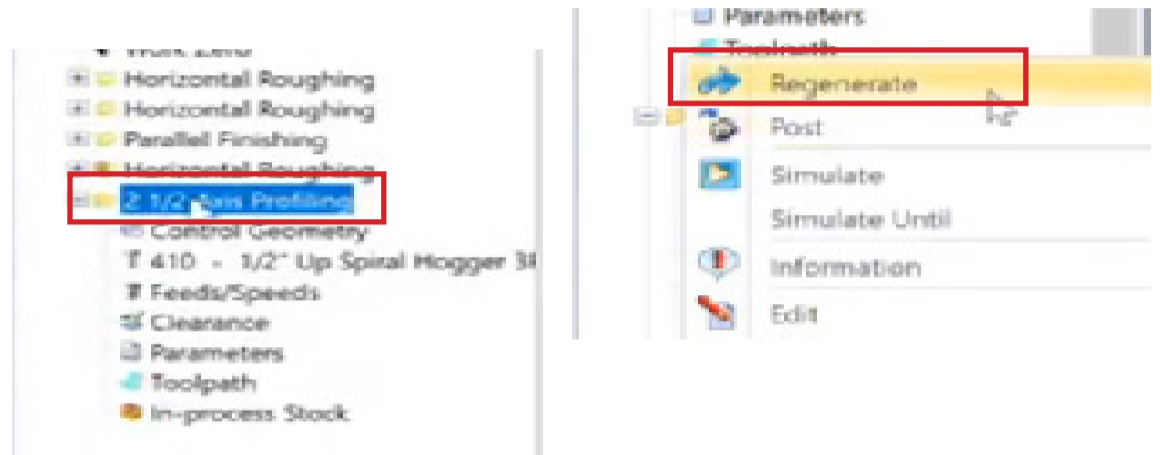
- Also, click the Axial profiling toolpath and right-click, then click "Regenerate."

- You will realize that the cut level is no longer valid for the material thickness.

- Double click on the Axial profiling toolpath and go to the cut level section.

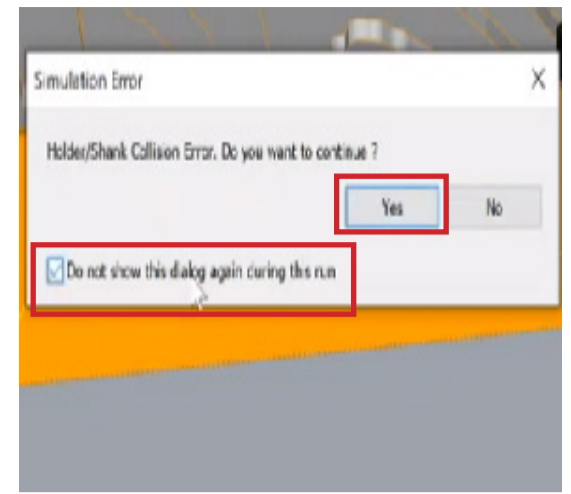
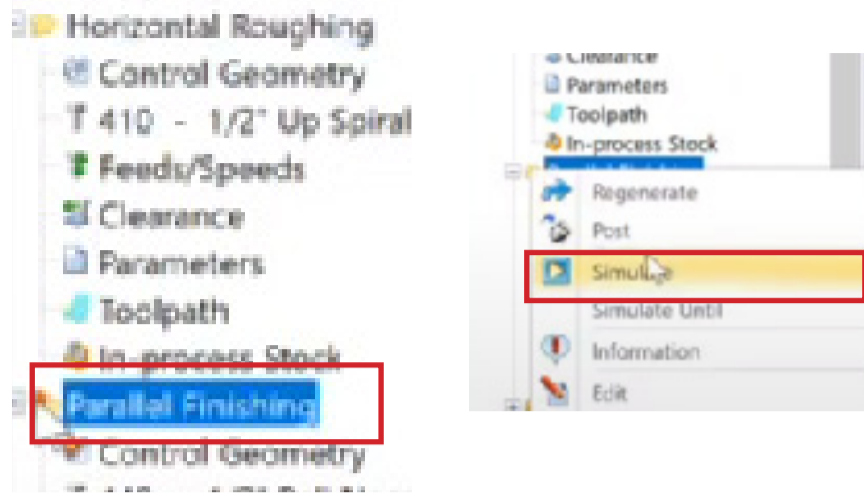
- Change the location of the cut geometry to 2.5 and the total cut depth also to 2.5.

- Click generate.

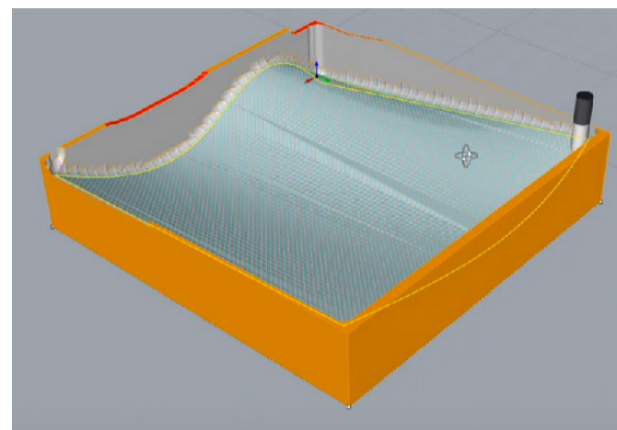


- Right click on the parallel finishing toolpath, then click simulate.

- Now, a small window pops up to notify us that there is a collision error. Click "Yes" to stop at the next collision.

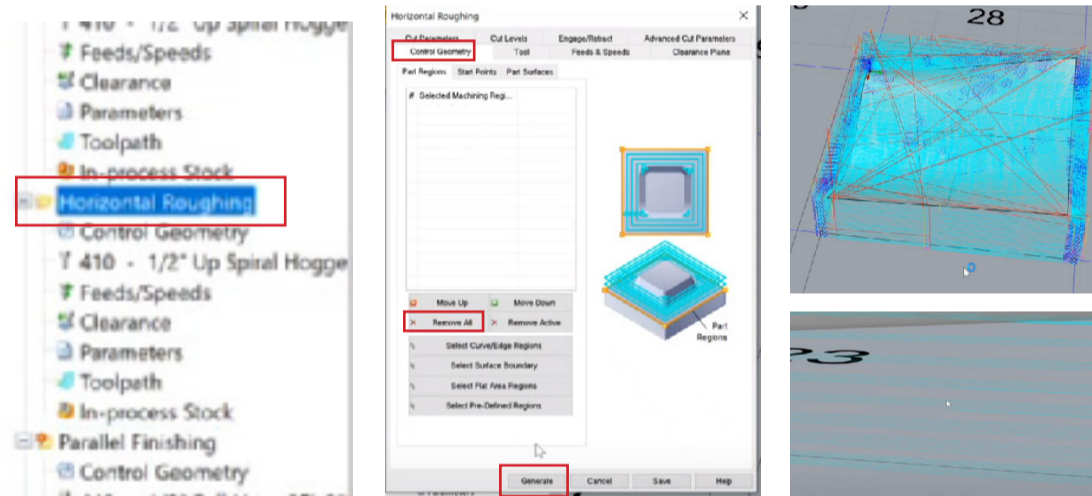


- There are several areas in this short finishing bit where we encounter problems. This highlights the issue when dealing with a thick piece of material that has significant variations in height. To address this, we have two options: either choose a longer ball nose bit or find a way to eliminate the lip around the edge to avoid any collisions.



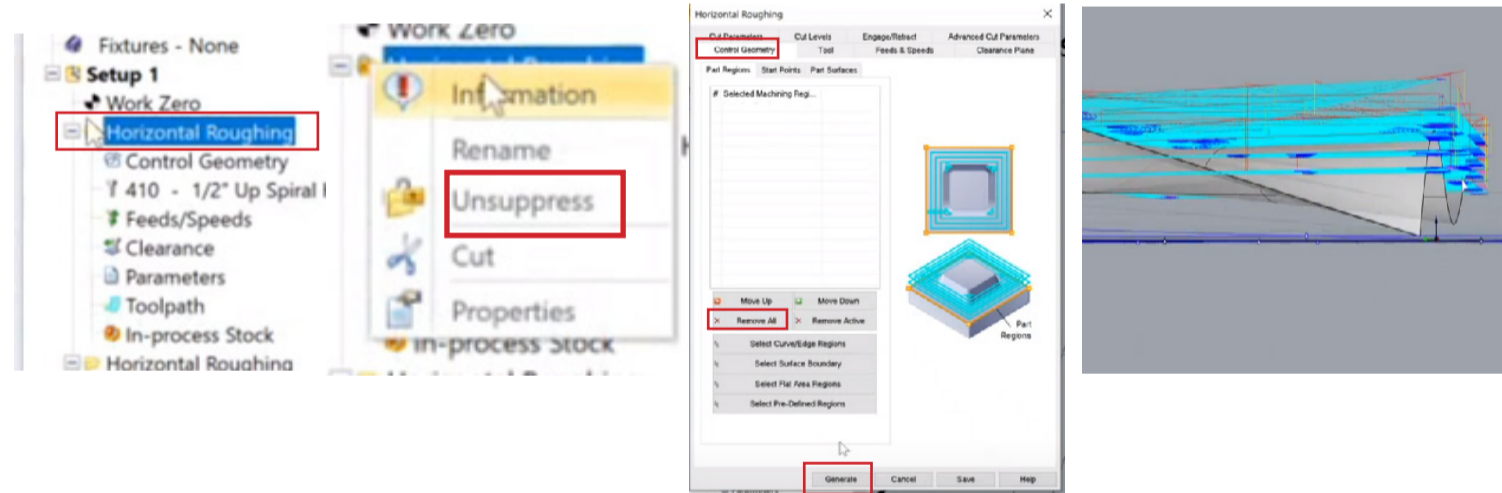
- To achieve this, double click on the second horizontal toolpath and go to the control geometry section. Then click "Remove all" because that will generate toolpaths all the way down to the ground.

- Looking at this image here, you can see that at each level, it's going around this object three or four separate times. That could cause an issue.

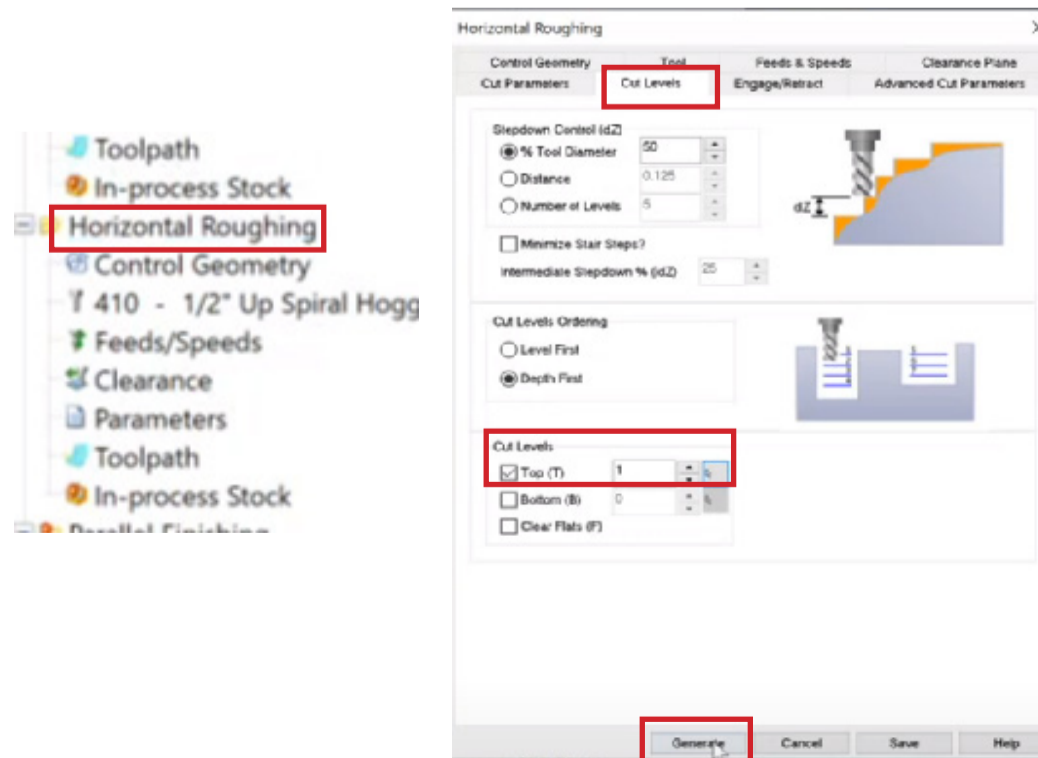


- First of all, right-click on the first horizontal toolpath and click "unsuppress."

- Then double click on the first horizontal toolpath and go to the control geometry and click "remove all."



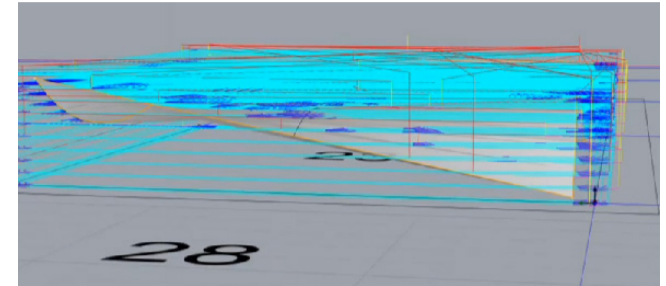
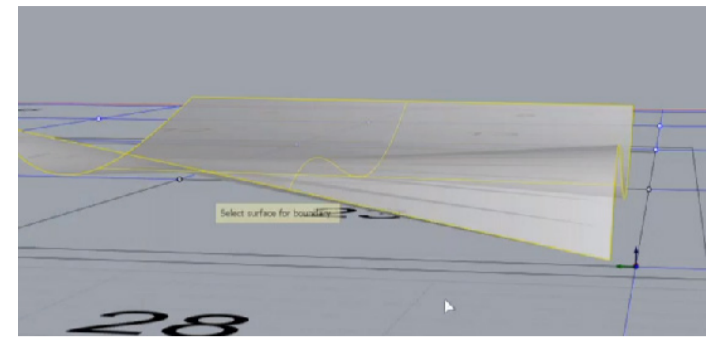
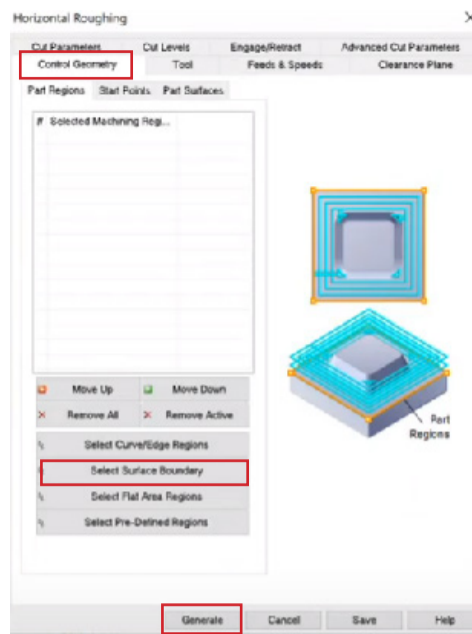
- Then go to the cut level section and set the cut level bottom to 1 inch, then click "Generate."



- Double click on the second horizontal roughing toolpath, then go to the cut level section and set the "Top cut level" to 1 inch.

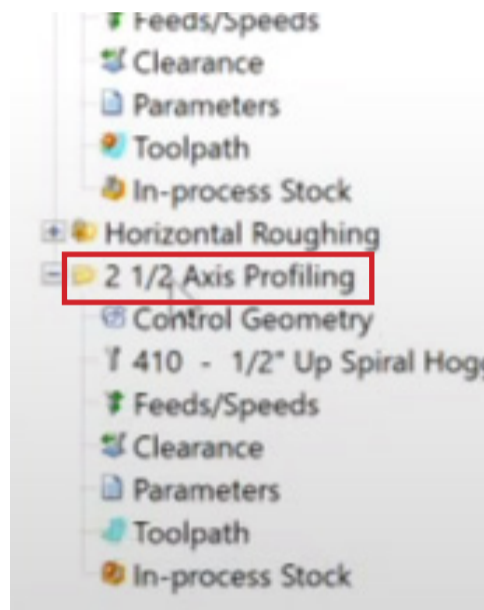
- Then go to the control geometry section and click "select surface boundary," then click the surface of your geometry and right-click, then click "Generate".

- This selects the surface boundary and limits this path here to that bottom and allows this Axial toolpath to remove that last inch.

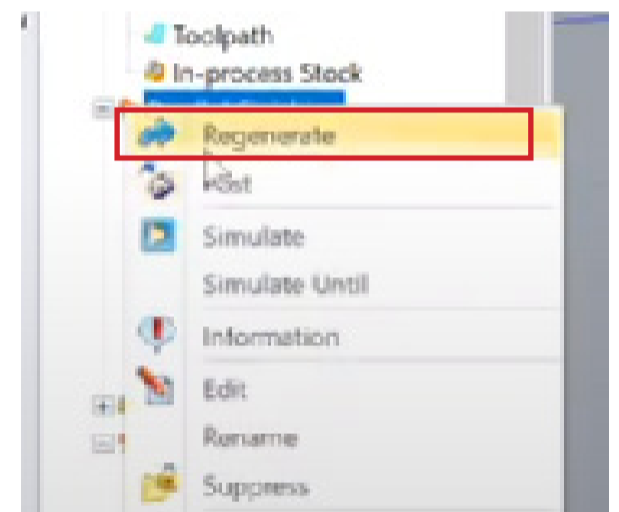
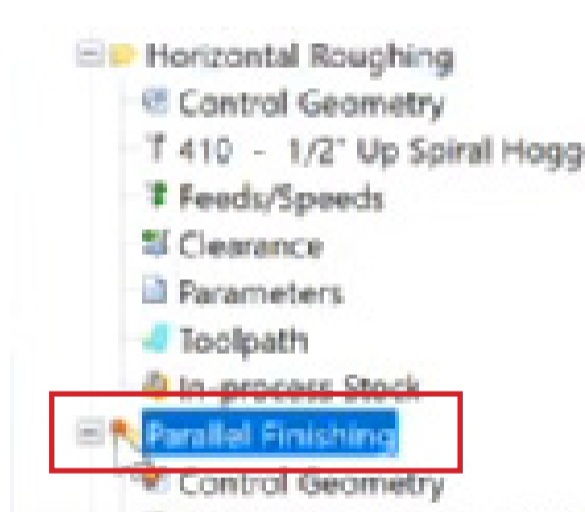


- Next, double click on the Axial profiling toolpath and go to the cut level section. Enable the "Pick Top" and set the location of the control geometry to 1 inch and the "Total cut depth" control to 1 inch.

- Then click "Generate."



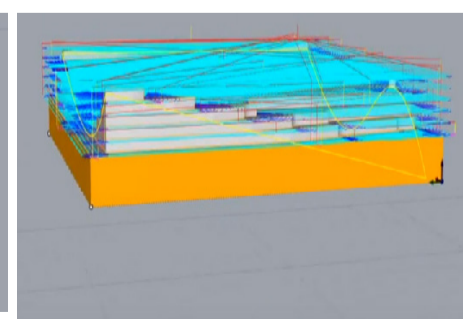
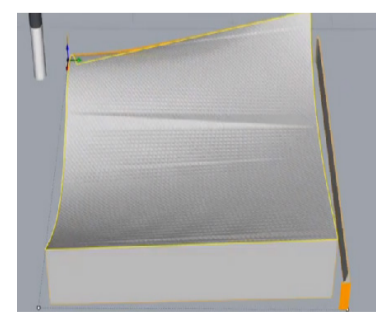
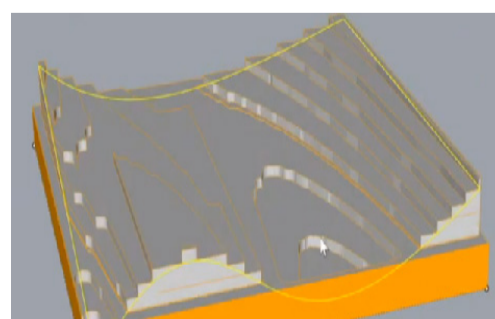
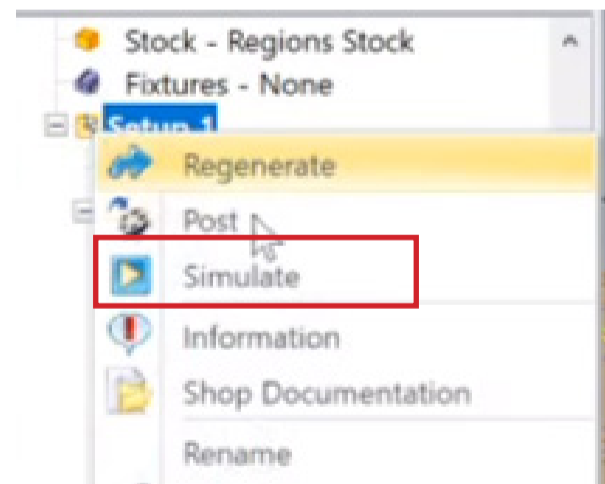
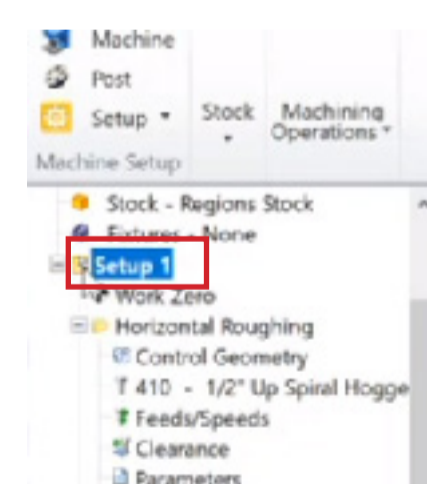
- Right-click on the parallel finishing and click "Regenerate."



- So, let us run through the whole simulation of the job. Right-click on "Set up" and click "simulate."

- Checking the result from the simulation, it is removing all of the material around the edge down to an inch and now that wall that was there before is gone, so there's no collision with the tool holder anymore.

- Well, with the second roughing path, the control geometry was reactivated to confine it to the main part of the surface we are milling. This prevents it from cutting outside that area and reaching all the way to the ground, which could disrupt the seal on the mill table. Without this adjustment, the toolpaths would have extended all the way to the bottom.



- Since running through this process with MDF and using a half-inch tool can be time-consuming, it would be practical to increase the stepover beyond the default value of 25% (equivalent to 1/4 inch).

- To make this adjustment, follow these steps:

- Double-click on the first horizontal roughing toolpath.

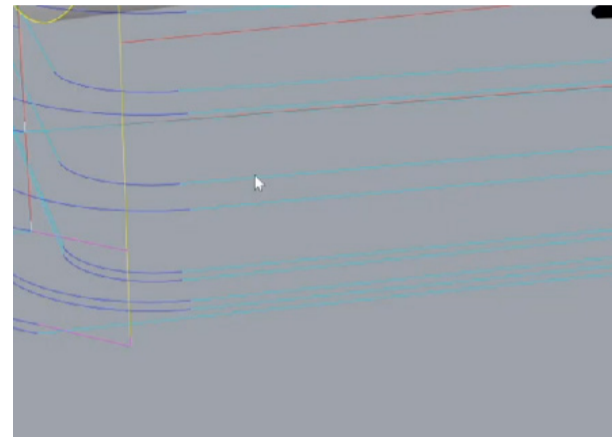
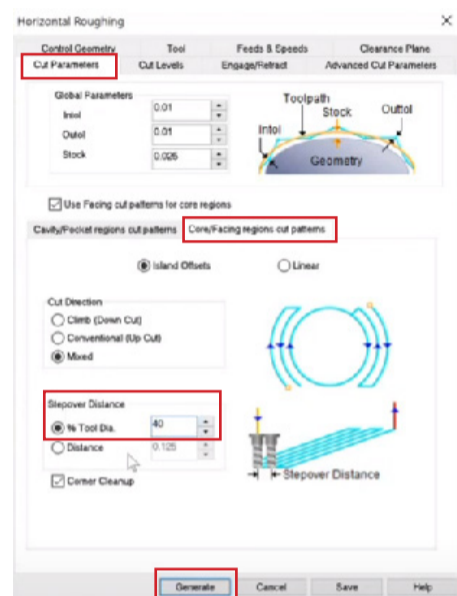
- Access the cut parameters.

- Choose the "core/facing region cut pattern".

- Enable the "Island offset" option. Set the stepover distance (% of tool diameter) to 40.

- Finally, click "Generate."

- You will notice that it is a whole lot less dense, which is better, and this will run a lot faster than with the 25% stepover. However,



NOTE: If you are working with MDF or softwood like plywood, increasing the stepover to 40% to save time is an efficient approach. However, if you are not using MDF or if you are working with hardwood, it is advisable not to exceed a stepover of 25%.

B. SCENARIO WHERE WE CREATE A CAVITY IN OUR GEOMETRY

Sometimes students or professionals may want their object or building model to sit flat on a curvy topography. We can model a couple of different things.

Rhino Basics

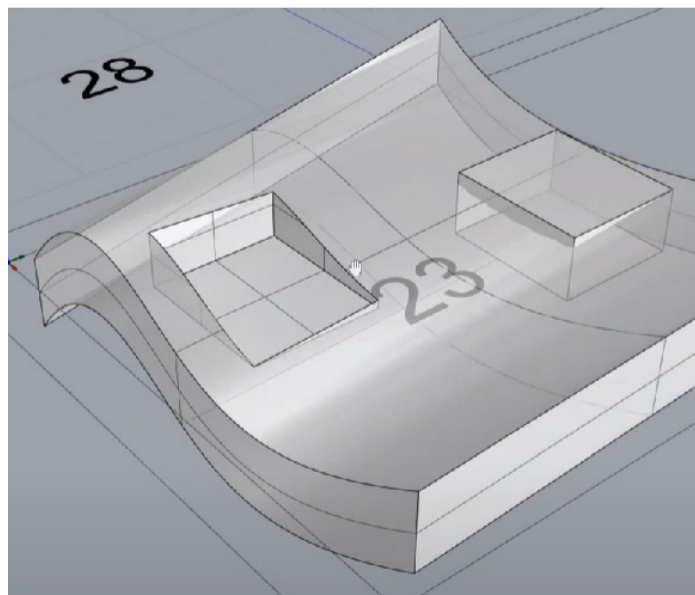
- You can use Boolean operations in Rhino to Boolean out the area, let's say the foundation where the building will need to sit on.

- In Rhino basics, pick the box tool and draw a box of a size that your object/building will fit on the surface geometry. Draw two boxes.

- Also, ensure the box does not go deep into the solid geometry.

Ensure your geometry surface is extruded such that it is solid and probably delete any standalone surfaces.

- Use the Boolean difference to Boolean the first box out. For the second box, leave it as a platform (as seen in the image).



RhinoCam

- Right-click on the first horizontal roughing toolpath and click "Regenerate."

- Turn on the toolpath visibility below the screen on the left side.

- Then right-click on the first horizontal roughing toolpath and click "Simulate."

- Turn on the "Tool path visibility" you will realize that the trench you created from the Boolean operation tool has been cut out. But there are some issues with the cut geometry of this cavity.

- See, we need to redefine the cut geometry because I deleted that surface. you can create a rectangular box around the geometry. Try to make the rectangle box slightly larger than the entire geometry.

- Go to the control geometry section under the second roughing toolpath.

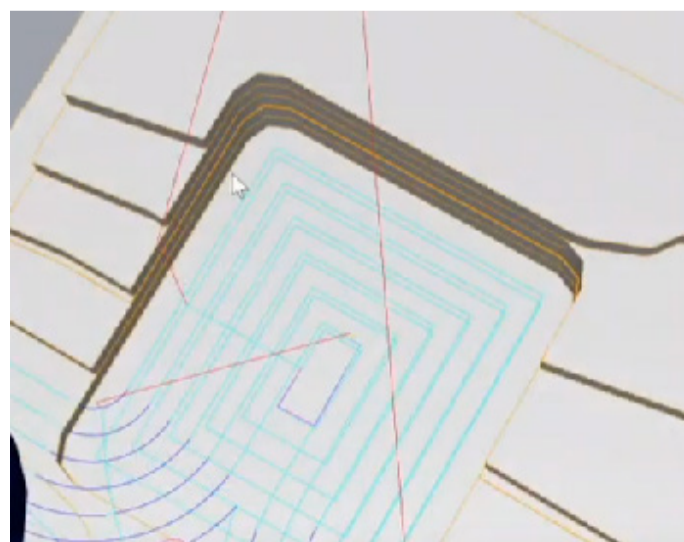
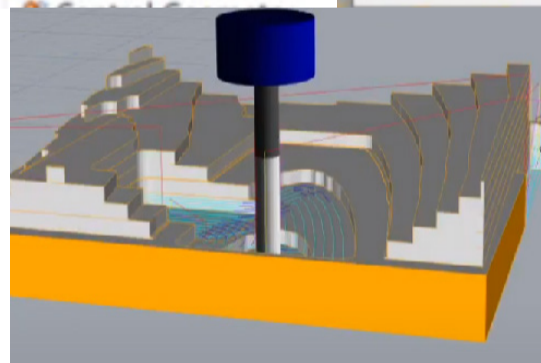
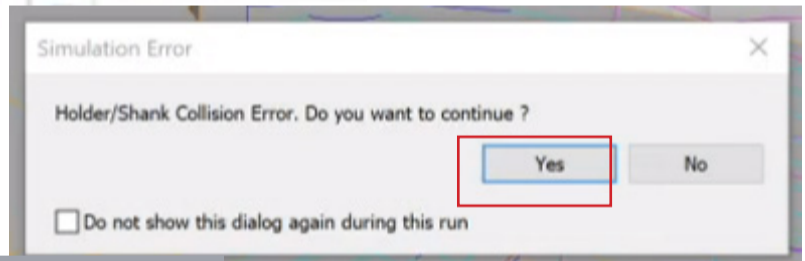
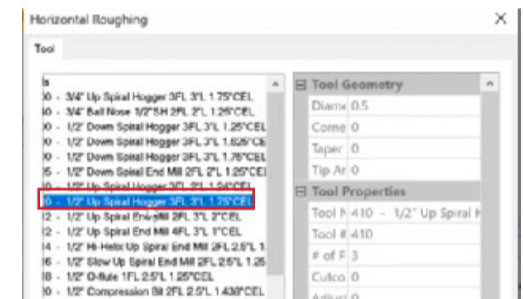
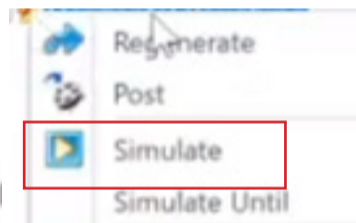
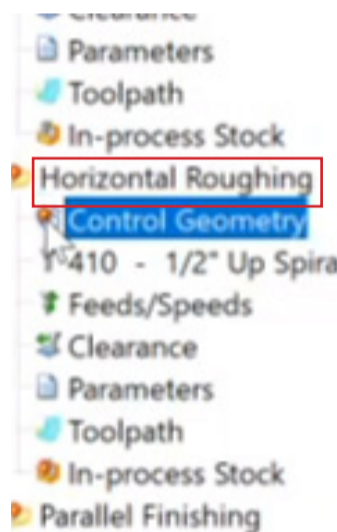
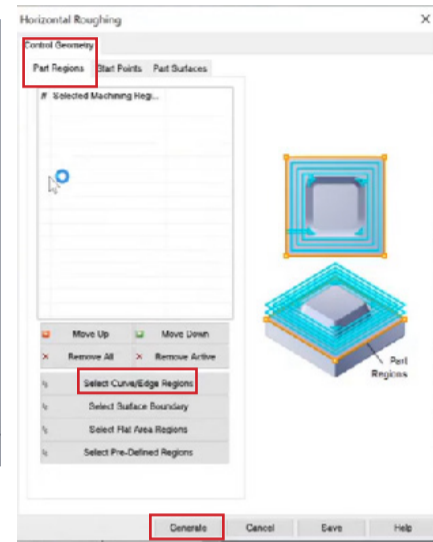
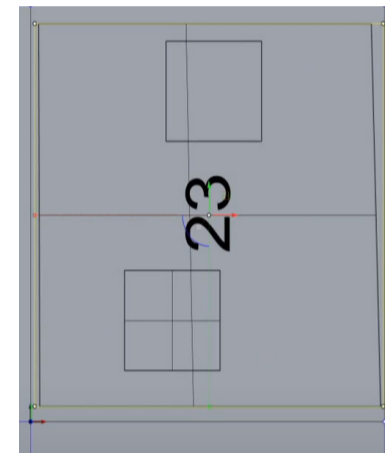
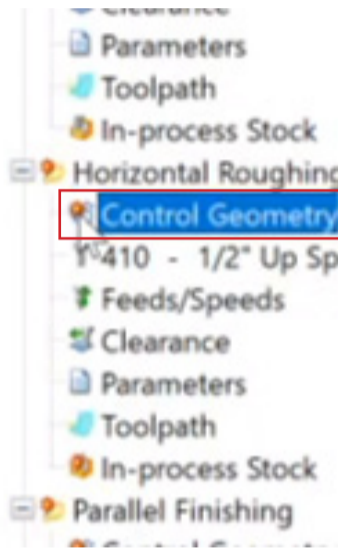
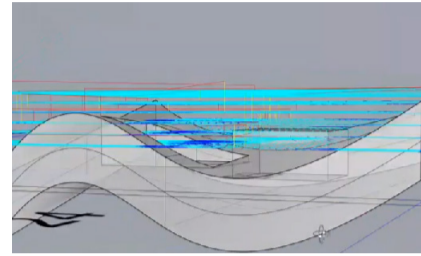
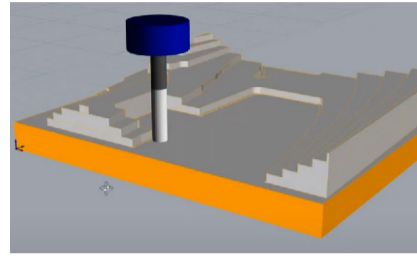
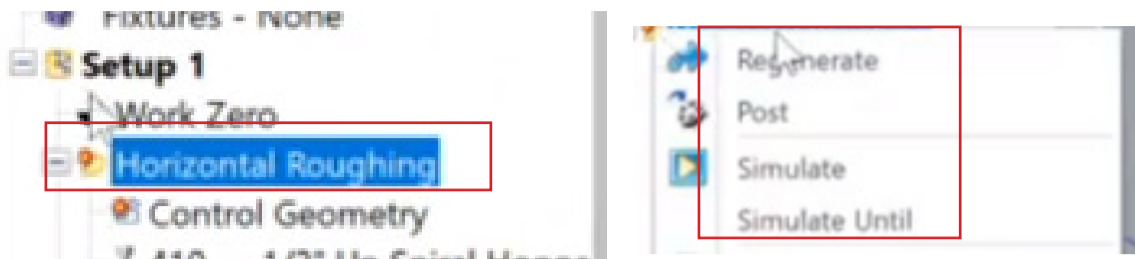
- Click on "select curve/edge region." Then select the curves of the rectangle box you just created and right-click. Next, click on "Generate."

- It seems that it was able to recognize that it needs to clear out that area with this, which is what we wanted.

- Right-click on the second roughing toolpath and click "Simulate." Here, there was an issue, it brought out a collision error. In case you also faced this, click "yes" to continue. To solve this issue, you might probably need to increase the length of your tool bit. If you remember, in this module example we had used a 1/2-inch diameter 2L bit. So, let's select a 1/2-inch diameter 3L bit. Go to the tool section under the second roughing toolpath and select the tool with 3L.

Once the tool bit has been altered, right-click on the second roughing toolpath and click "Simulate." So, no more "collision error" notification.

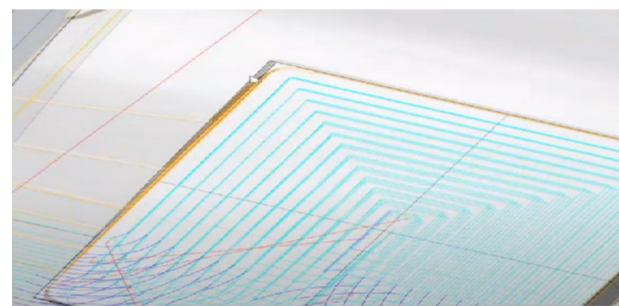
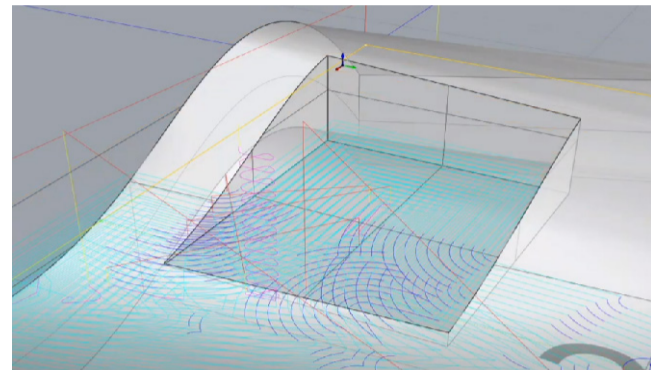
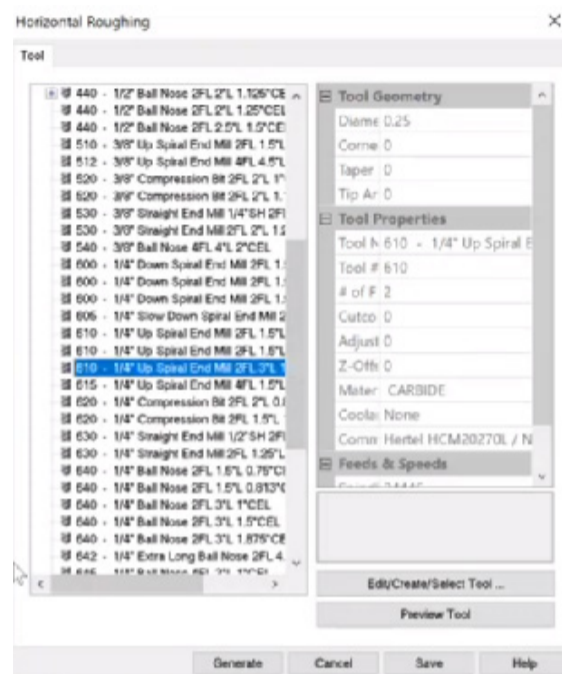
- The real challenge here, which applies whenever you're cutting anything with an inside corner, is that you won't be able to achieve a sharp inside corner. As you can see, the closest the bit can get to that corner will still leave a 1/4-inch radius curve. This limitation arises because CNC mills that use round bits cannot reach all the way into the corner. When setting up the mill, it's important to convey to the student that if they desire a similar outcome, they need to ensure the cut-out area (trench) is large enough (add a little allowance) to accommodate the model.



- Alternatively, they may have to manually clean out those corners using a chisel or a similar tool, though this method may not yield perfectly clean results. It's crucial to note that inside corners won't be sharp, and planning accordingly by either increasing the size of the trench or preparing to clean it up by hand is necessary.

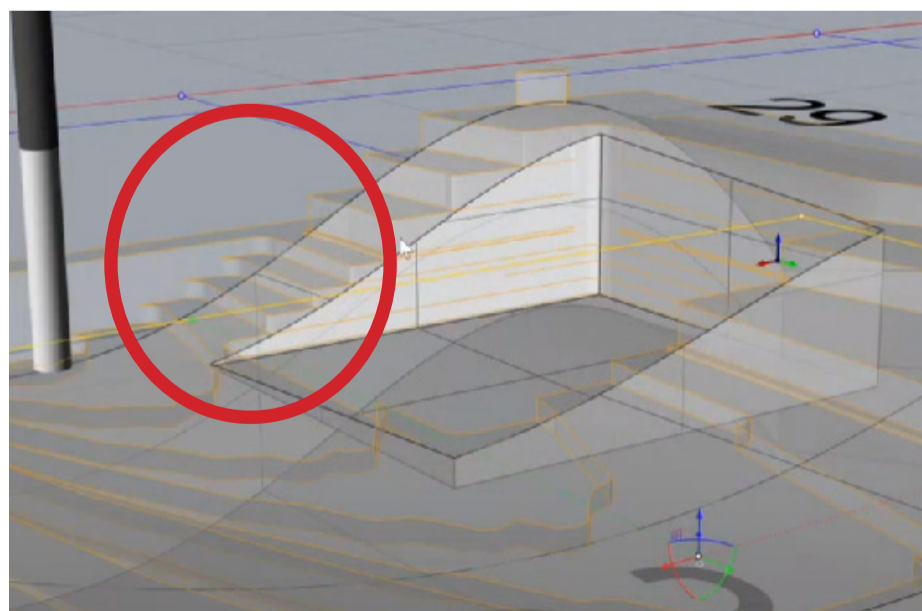
Now, let's continue with the instructions:

- Now we can achieve a tighter corner by using a smaller bit diameter.
- Go to the tool section under the second roughing toolpath and change the tool bit diameter from 1/2 inch to a 1/4 inch bit (610-1/4" up spiral end mill, 2FL, 3"L).
- By doing this, we will naturally see some trade-offs. The reduced bit size means that other aspects will take longer as well, such as the step down and step over, which are now halved. Let's simulate this.
- As a result, we now have a 1/8 inch radius instead of the previous 1/4 inch radius.

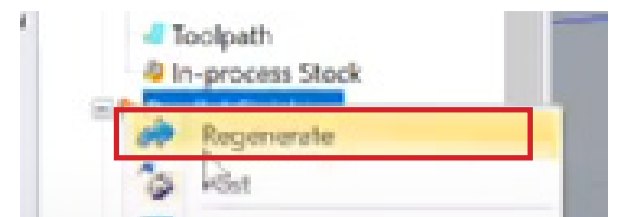
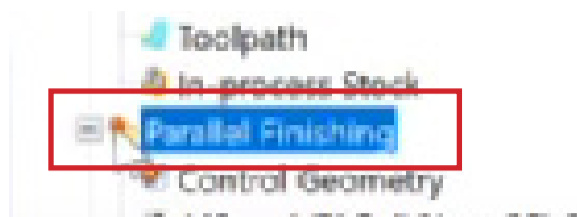


- However, it's important to note that working with a smaller bit requires smaller step downs and slower movements. This will impact speeds, deflection, and other factors associated with using a smaller bit diameter.
- Whenever you change the bit diameter, it's crucial to adjust the feeds and speeds accordingly.
- Additionally, it may introduce additional challenges that could make it not worthwhile.

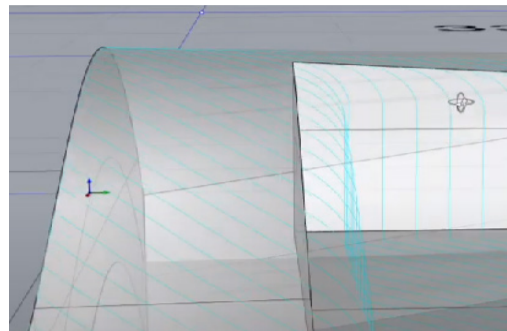
- If we turn off these toolpaths, you can see that when we switched to the 1/4 inch bit, the step down was reduced by half, resulting in smaller steps. The step over, being a percentage of the bit diameter, is also significantly smaller. Consequently, the overall machining time will be considerably longer.



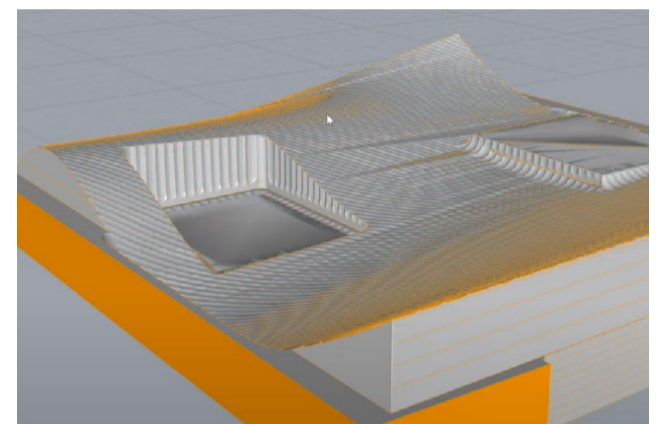
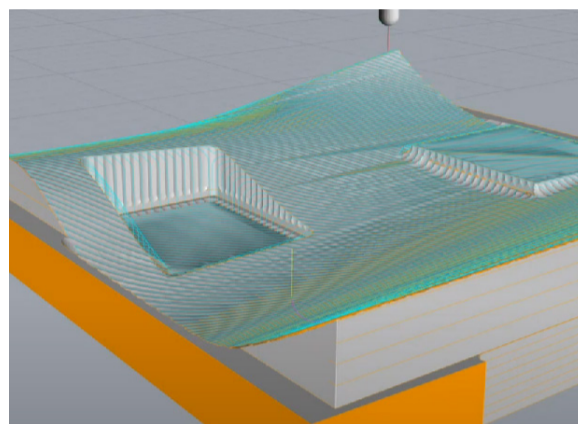
- Let's move on to the parallel finishing toolpath.
- Right-click on the finishing toolpath and select "Regenerate."



- Now you'll notice that while it still follows the surface, there is a challenge when it comes to the pocket (cavity).
- The tool must navigate over the edge and then descend, ensuring it avoids touching the edge.
- The result here shows that it is moving out into space before plunging into the cavity. Similarly, the opposite occurs when approaching the edge to avoid affecting the geometry.



- If we simulate this parallel finishing toolpath, you'll observe some interesting effects caused by using the ball nose bit.
- Disabling the toolpaths reveals that the scalloping seen on the top surface is now potentially visible on the edges of the pocket and the platform. Furthermore, it appears that the ball nose bit is too large to fully clean up all the remaining rough edges left by the roughing path. Clearly, this outcome is not what we desire.

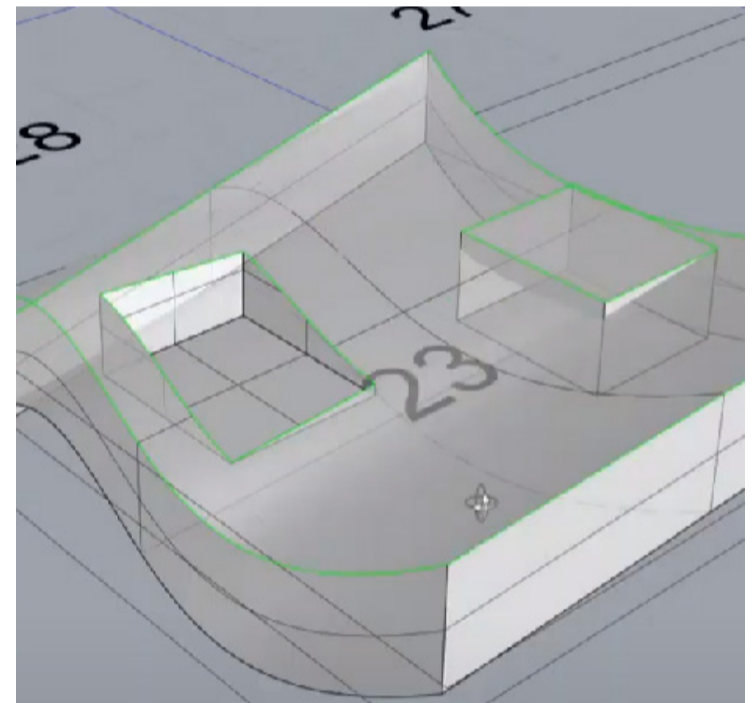
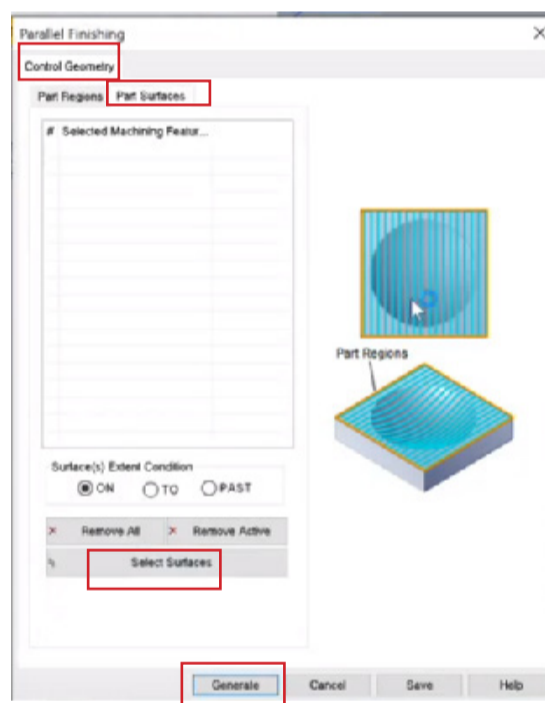


- The challenge lies in finding a solution to improve the results of the edges.
- One possible approach is to reconsider the use of the parallel finishing toolpath for this specific pocket since it has already been cleared by the roughing paths.

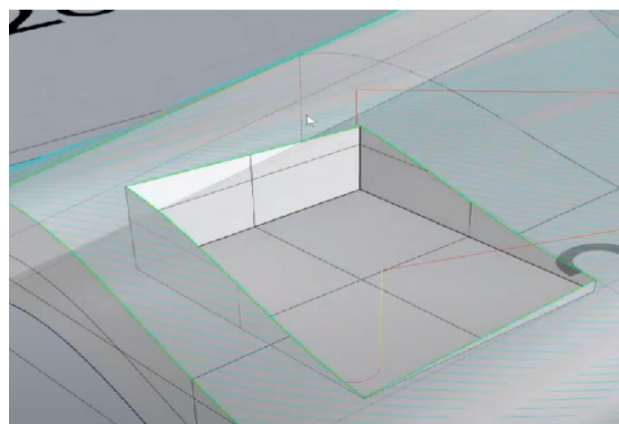
- Instead, we can select only the necessary surfaces for the parallel finishing operation.

- Go to the parallel finishing toolpath and click control geometry. Then, click the "part surface" at the top. Then click "select surface." Then click only on the surface of the solid geometry and the platform we created too.

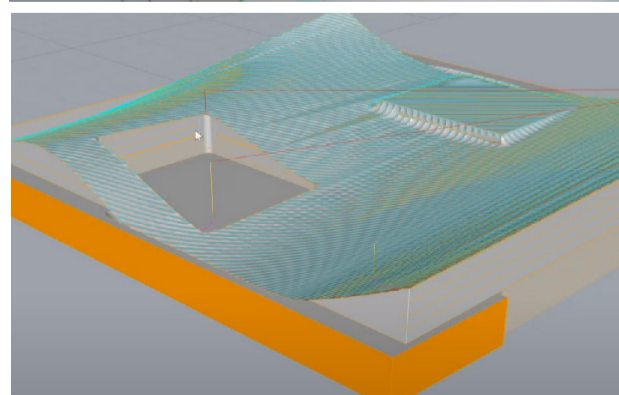
- Right-click and then click "Generate."



- Turn on the "Tool path visibility" and you'll observe that the toolpath now zigzags along the edge without entering the cavity. This preserves the cleanliness of the cavity, preventing any potential damage or scarring caused by the tool reaching inside. Although the ball nose bit may not leave much residue, it's crucial to address the underlying issue that likely caused the previous scalloping or evidence of the bit entering the pocket.



To check the result of the finishing path, right-click on the finishing path and click "simulate."



FINISHING

- To clean up the edges, go to the program, then click the machining operation, then the 3D axis, and click "horizontal finishing."

- The difference between a horizontal finishing and a horizontal roughing path is that a roughing path is just focusing on removing material, while the horizontal finishing path is just moving along the surface to clean up that edge. Therefore, it is not going to fill the whole area of your solid geometry with toolpath; it is just going to clean up the edges.

- Once you select the horizontal finishing, a new window will appear.

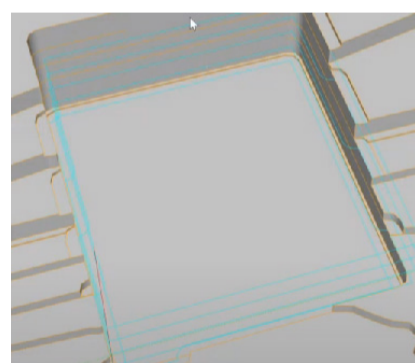
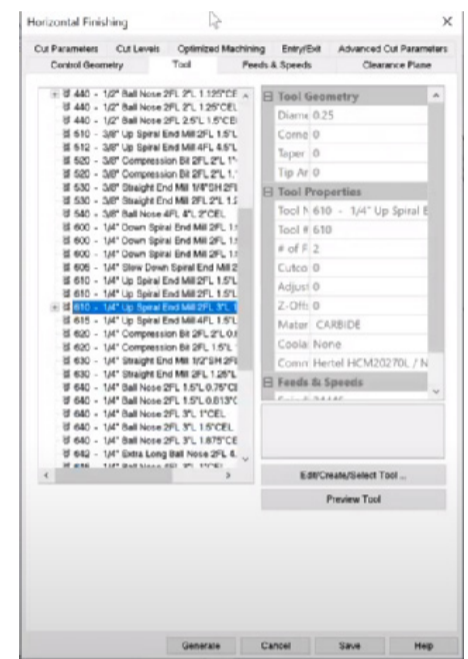
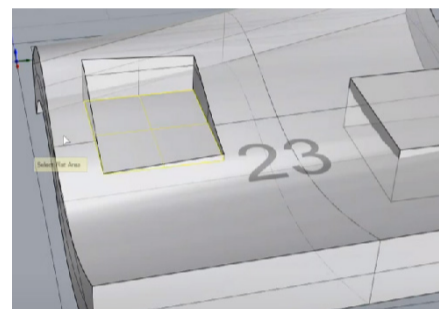
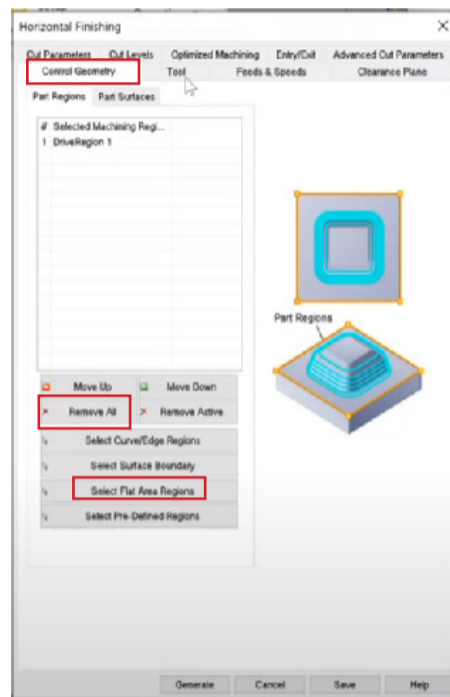
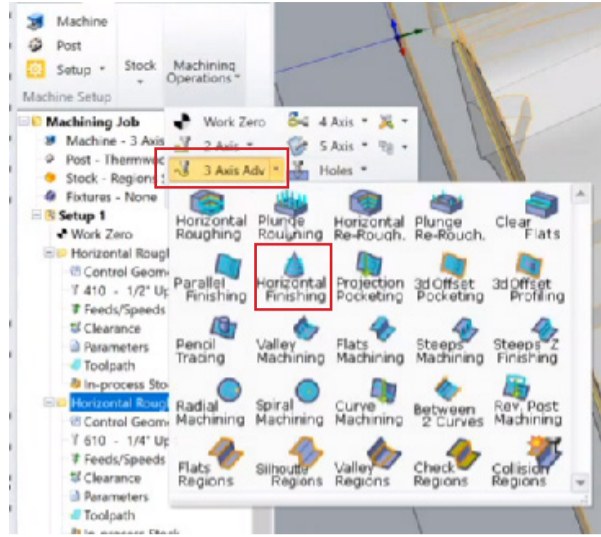
- Then, go to the control geometry section and click "Remove all."

- Then click "Flat region." Then click the bottom surface of your cavity and right-click to go back to your control geometry tab.

- Then go to the tool section and click the 1/4" bit (610-1/4" up spiral end mill, 2FL, 3"L).

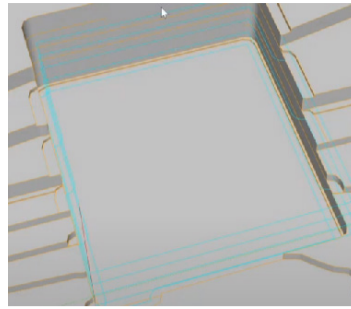
- Go to the cut parameter section and set the stock to "0." Go to the cut level section and set the stepdown tool diameter from the default (25% to 50%) since we are removing a tiny amount of material.

- Then click "generate." Right-click on the horizontal finishing toolpath and click "simulate."



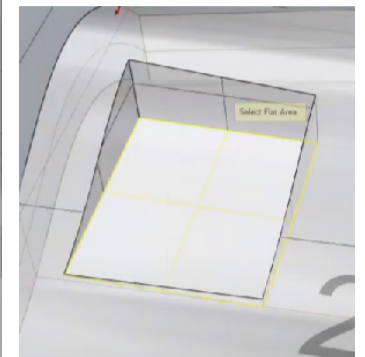
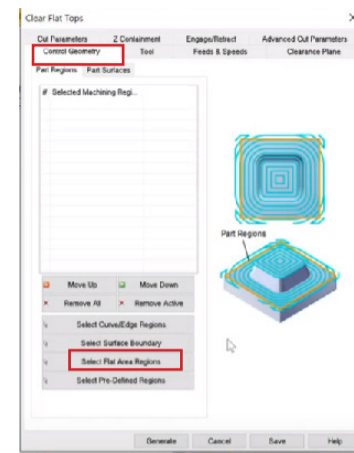
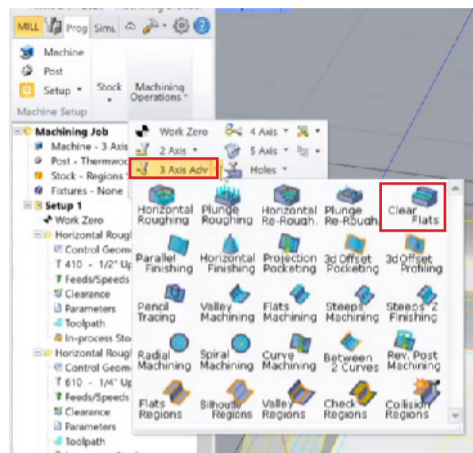
ii. CLEAR FLAT

- We are going to create another toolpath because when we simulated the horizontal finishing, some edges were not flattened (see image).

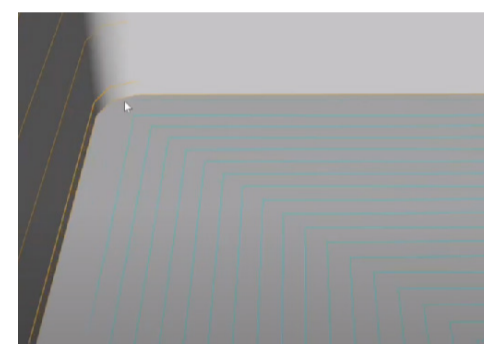
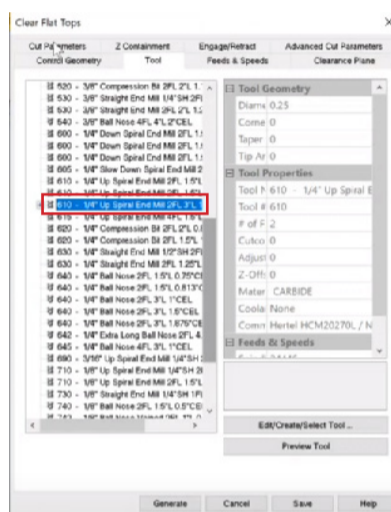


- To clear out this bump, go to the program, then to the machining operation, and to 3D axis. Click on the "Clear flats." A new window will appear.

- Then go to the control geometry section there and click on "select flat area region" and click the bottom surface of your cavity and the top surface of the box platform, and right-click.



- Go to the Tool section and select the 1/4" bit (610-1/4" up spiral end mill, 2FL, 3"L). Go to the cut parameter section and input "0" for the stock.



SIMULATION

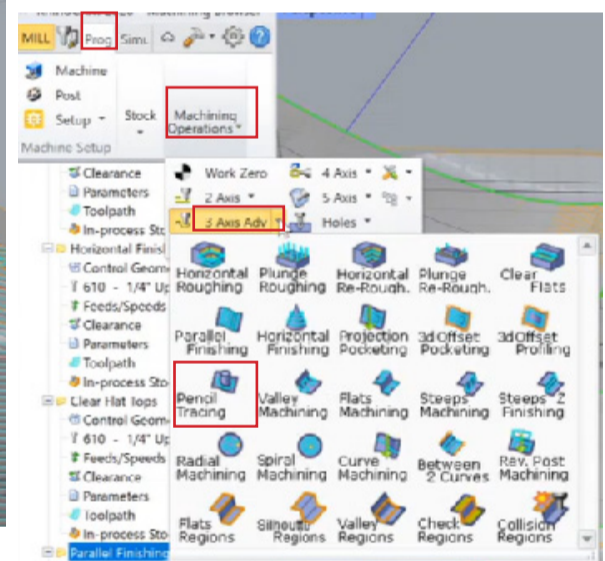
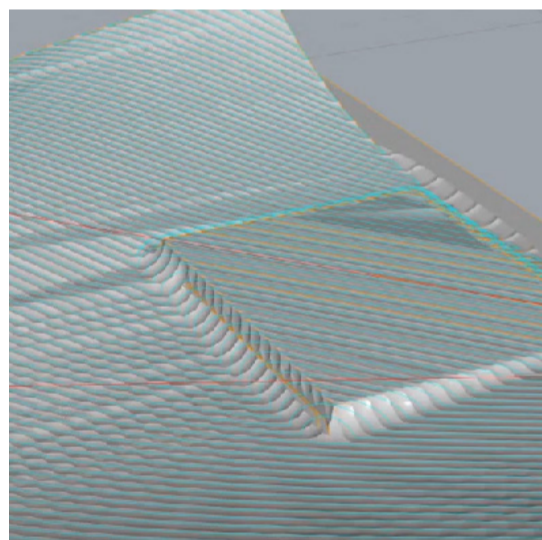
- Then right-click on the "Clear flat" toolpath and click "simulate." So, when simulating the whole job from start to finish, it goes through the clear flat to create more finished and cleared edges.

iii. PENCIL TRACING TOOLPATH

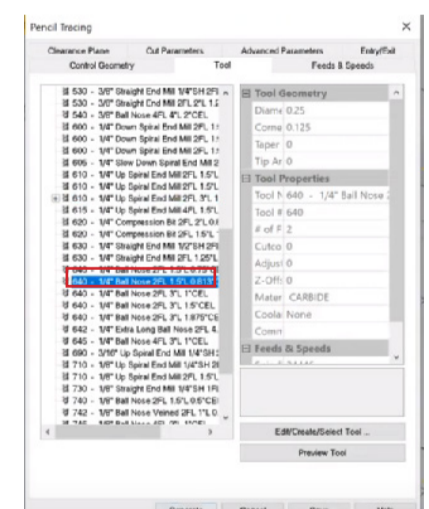
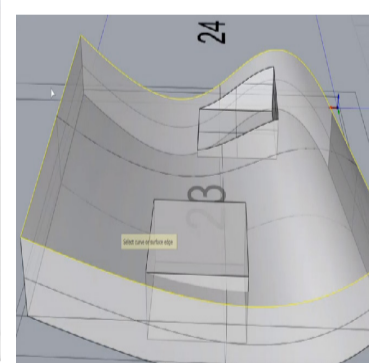
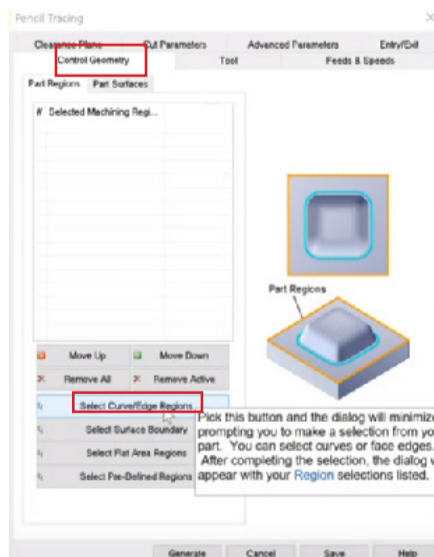
- After simulating the clear flat toolpath, regenerate the parallel finishing.

- Then, you may probably realize the sharp edges of the box platform are not looking too good; they have some rough finishing.

- To achieve a sharp edge, go to the program, then to the machining operation, and to 3D axis, and click on "Pencil Tracing." A new window will appear.

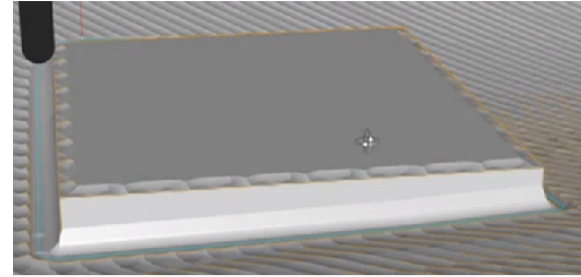
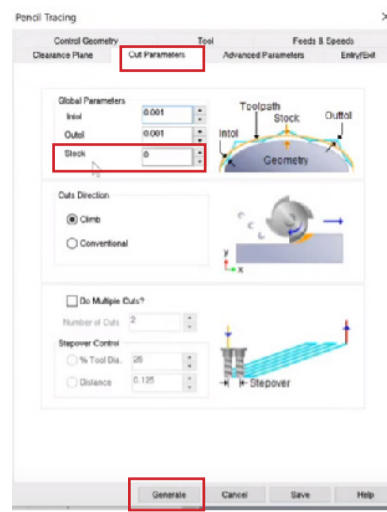


- Go to the control geometry section and click "Select curve region," then select the entire four edges of your top geometry, then right-click to find the location where it needs to be cut out cleanly.



- Then go to the Tool section and choose a tool, let's say a 1/4" ball nose (640-1/4" Ball nose 2FL, 1.5"L).

- Go to the cut level section and make sure that the stock is set to zero.
 - Then click "Generate."
 - Turn on the "Toolpath visibility."
- I am still not satisfied with the clean-up; my objective is to make the toolpath go around the edges of this box platform to make the edges come out cleaner and not to go over it. To do this, select the solid extruded geometry and the box platform, then click Boolean union.



C. ANOTHER TIP

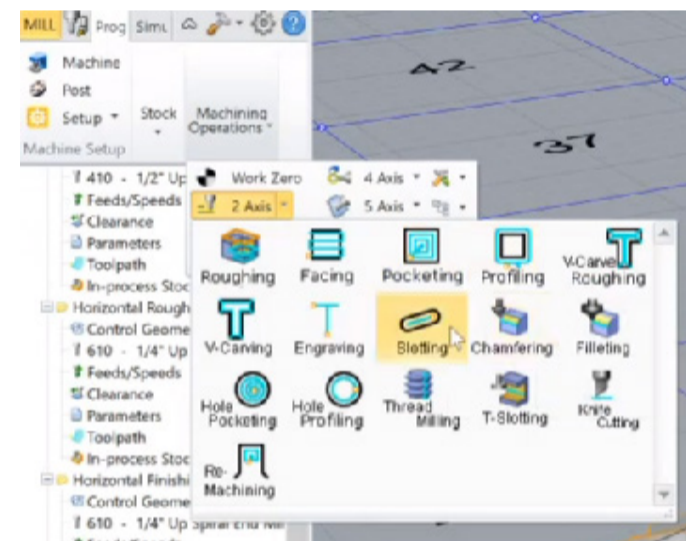
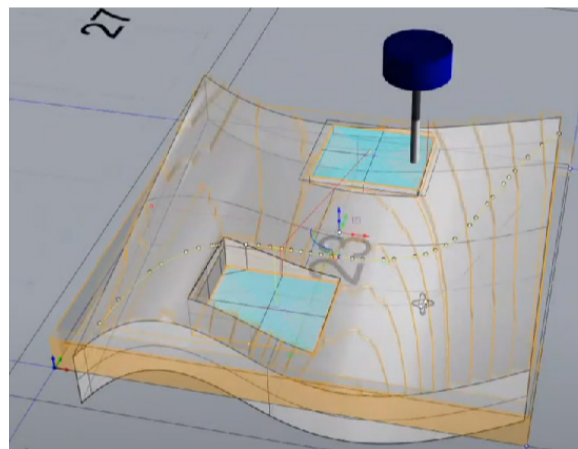
i Engraving toolpath

- The engraving toolpath is used to run the bit down a given line so the center of the bit will follow that line. The interesting thing about this is to engrave something to follow a 3D line. Let's look at an example.

- Let's draw a curve on top of our geometry.

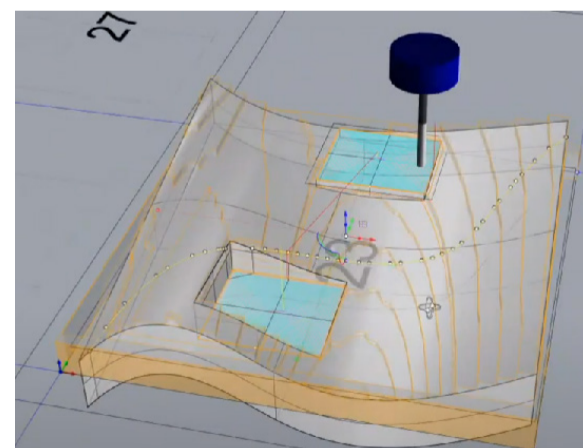
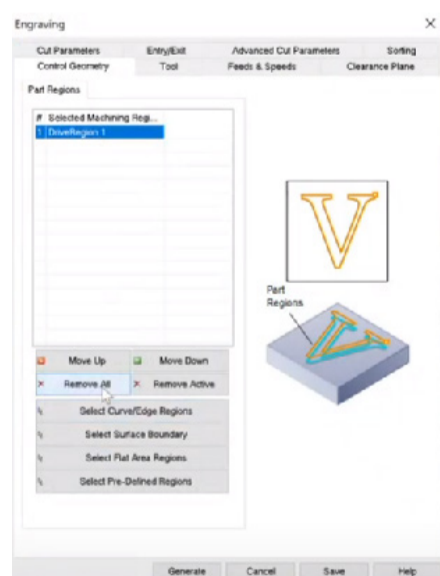
- Let's create this engraving toolpath after the pencil tracing toolpath. Click once on the Pencil tracing toolpath to make the engraving toolpath follow. Then go to the program tab, then to the machining operation, then to 2-axis, then click "Engraving."

- The engraving window will appear.



- Next, go to the control geometry section and click "Remove all."

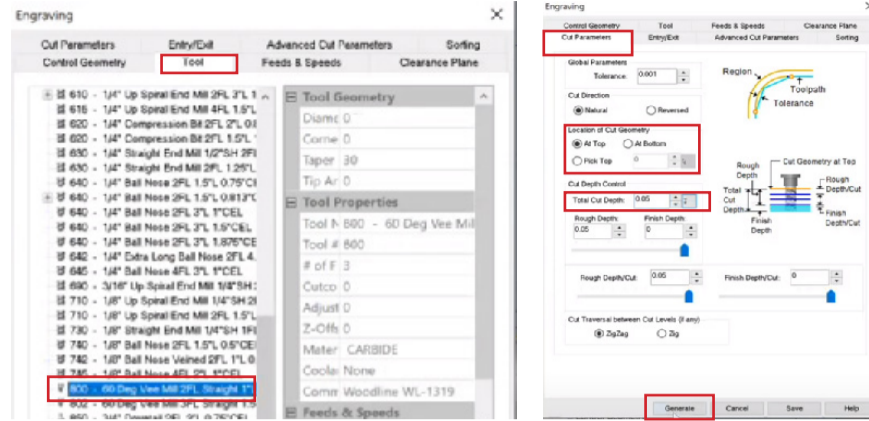
- Then click "Select curve/edge region." Then select the curves you just drew on top of the geometry and right-click.



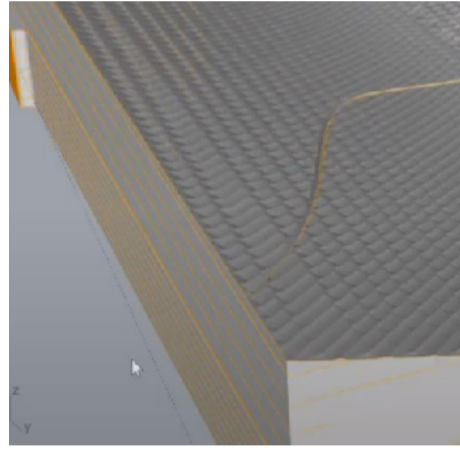
- Then go to the tool section and select a tool bit, let's say (800 60-degree vee mill 2FL straight, 1"L).

- Ensure to specify a depth. Go to the cut parameter section, under the "Location of the cut Geometry," choose a desired location, let's say "At Top." Then specify "Cut depth control" and make the "Total Cut Depth" a value, let's say 0.05.

- Then click "Generate."



- Then right-click on the "Engraving" toolpath and click "Simulate." The bit follows the curve on your geometry.



- This will give you a nice groove. You can also etch in letters or text by using a single-stroke font under text in Rhino (Font style recommended is the "SLF RHN Architect"). The only thing you might want to enable here under output is the "Curve" and "Engraving font." This type of font can be used as your control geometry and follow the Engraving toolpath process explained earlier.

