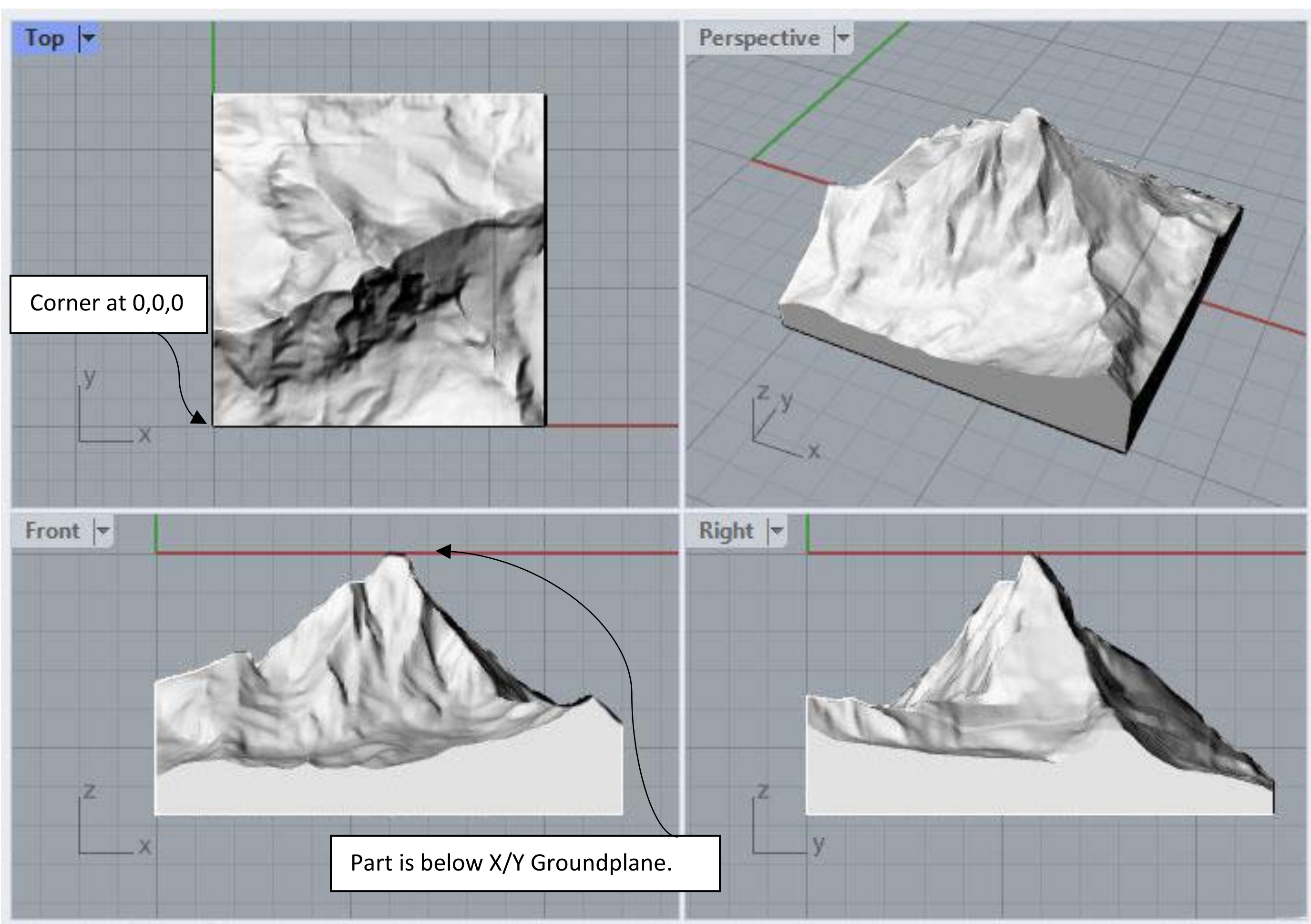
MadCAM Toolpath Generation for CNC Milling

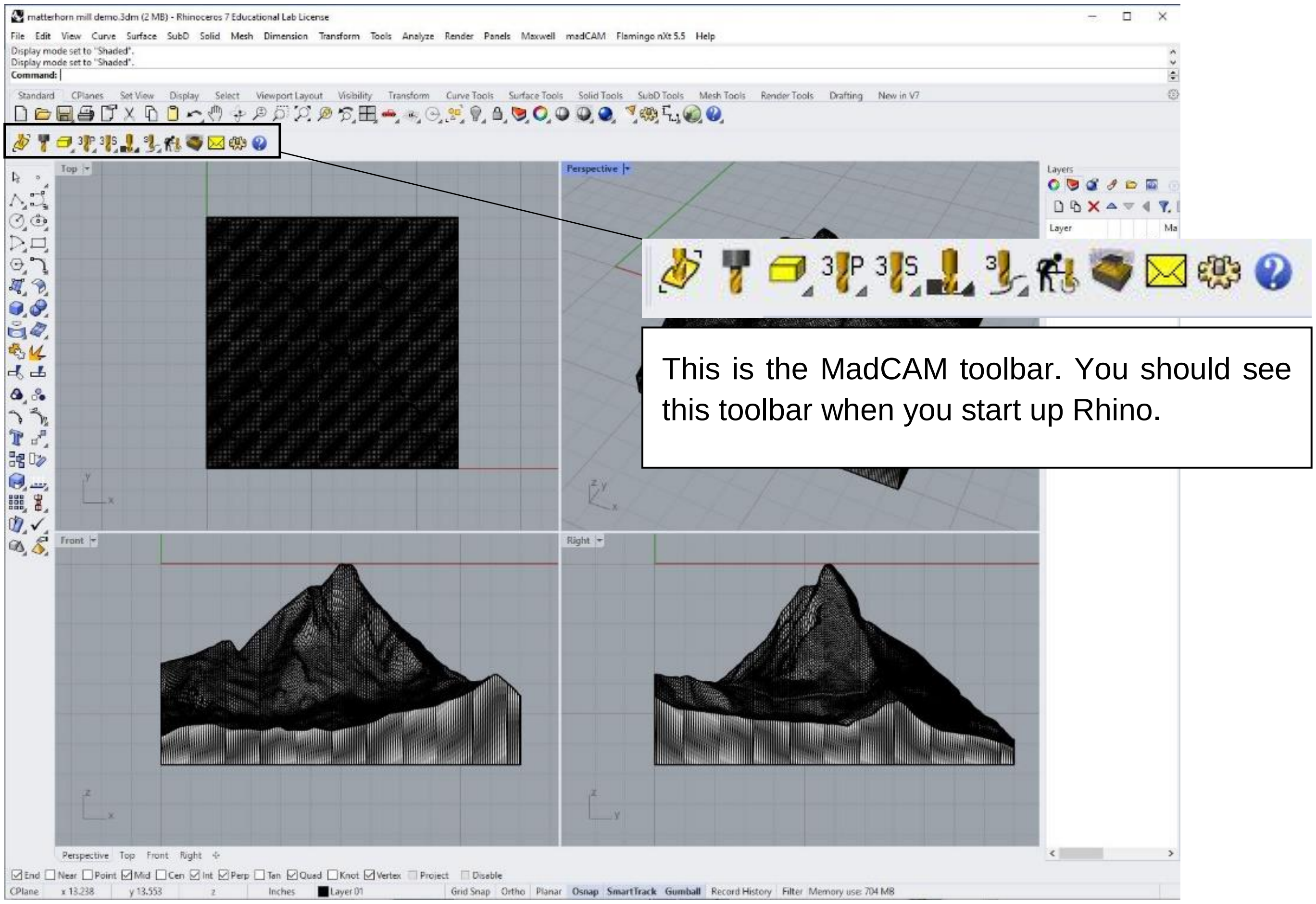
Fabrication Lab Tutorial
Written By Nathan Burnell

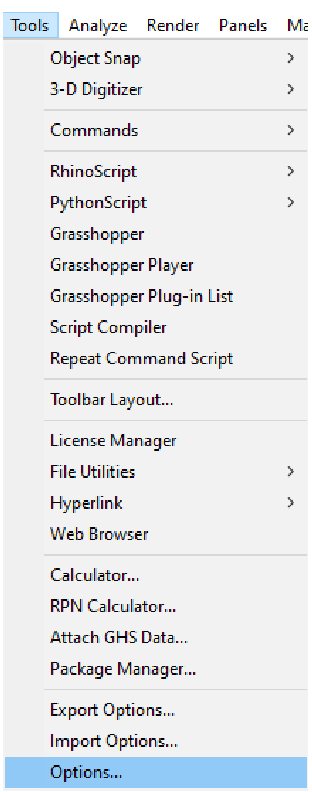
MadCAM is a plug-in for Rhino that uses Rhino's interface to generate and export toolpaths for a CNC mill. The CAM file that is created is simply a very long line of code that is mostly made up of both XYZ coordinates, and commands that inform the mill how fast to move. The mill will read this file and move the tip of a cutter to the specified coordinates at the correct rate to remove any unwanted material.

Step 1: Start Rhino and open your CAD model. Double check that the units of your model are in ***inches*** and that the size of your model is the exact same size as the part you are milling. MadCAM will work on any type of model whether you are using surfaces, solids, or meshes. It will only be able to create the objects that you can see from the top view down.

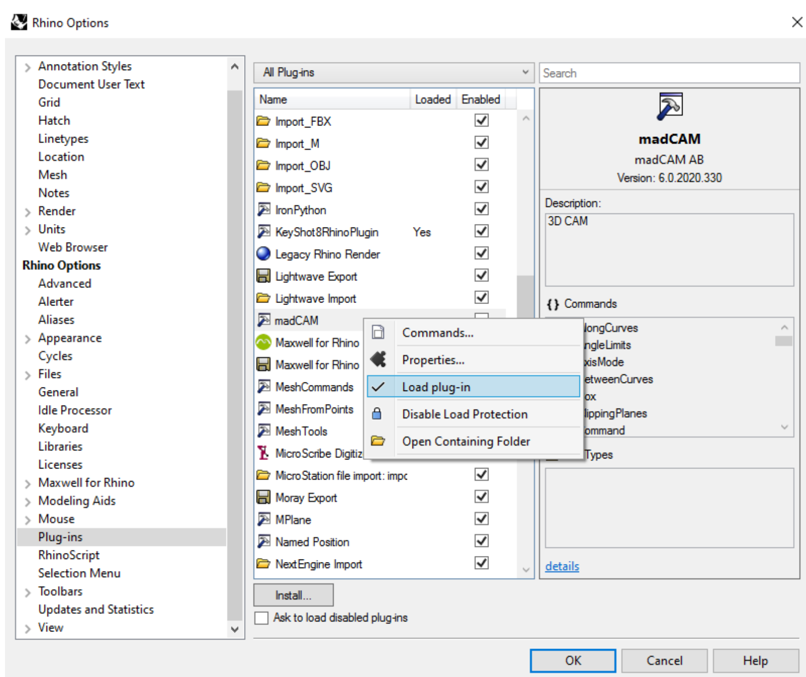
Step 2: Position your model so that it is ***located at zero*.** It should be ***positive in the X and Y axes*** and the ***tallest point at zero***. This step is important for aligning your part later on when you are setting up the CNC mill. The BoundingBox command can be useful for locating the extents of your model.



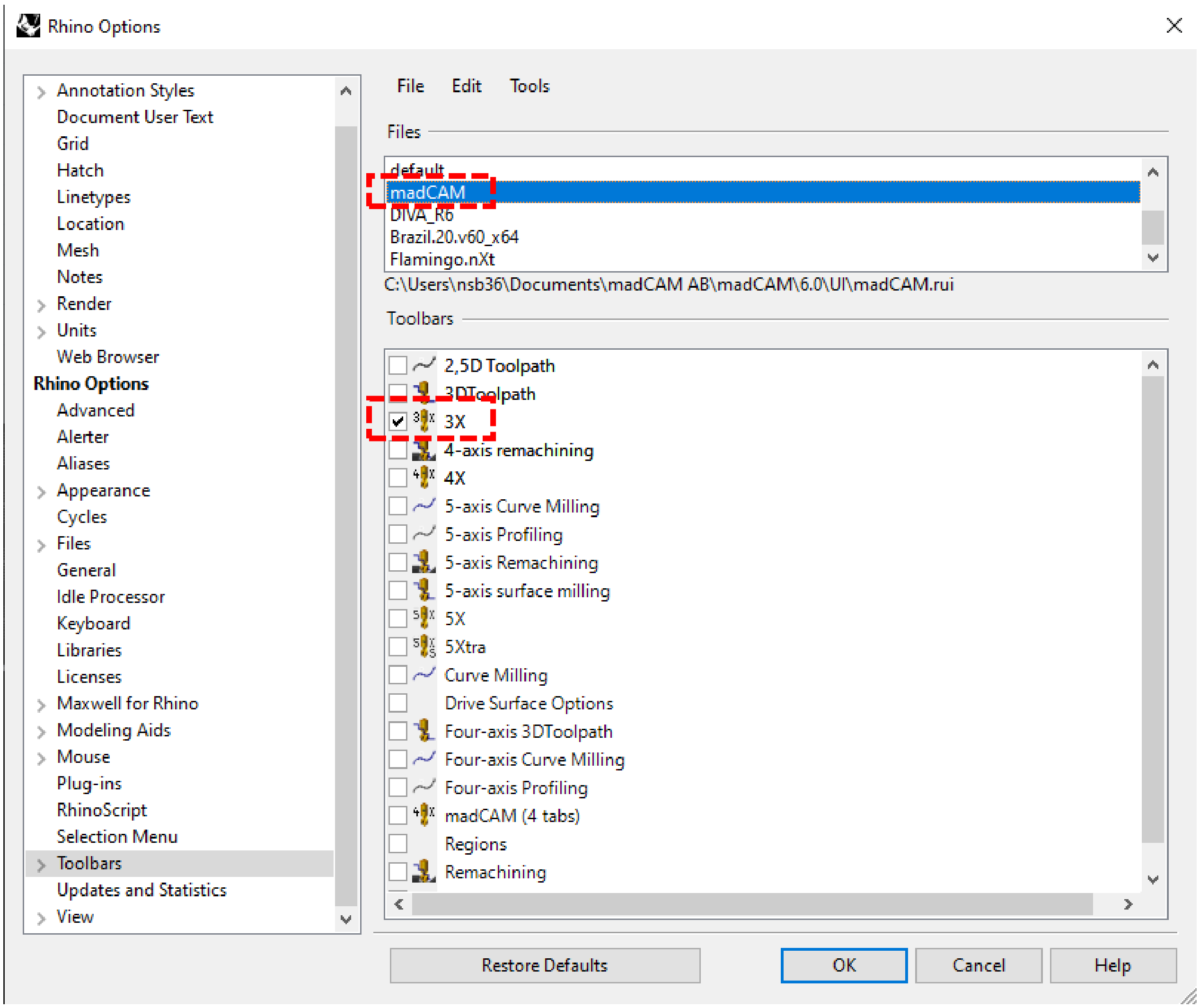




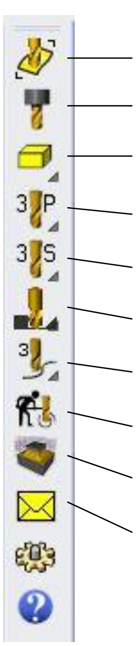
If you do not see the above toolbar, the MadCAM Plug-In may need to be loaded. To load this Plug-In, click ***Tools > Options*** and the following window will appear. Click on Plug-Ins in the options menu, then right click on MadCAM and select load Plug-In.



If MadCAM is already loaded in the Plug-Ins menu but you still do not see the MadCAM toolbar, click on the toolbars tab in the Rhino Options menu. Select MadCAM in the list of files, then check the box next to ***3X*** to turn on the 3 axis milling toolbar and hit **OK**.



The MadCAM Toolbar:

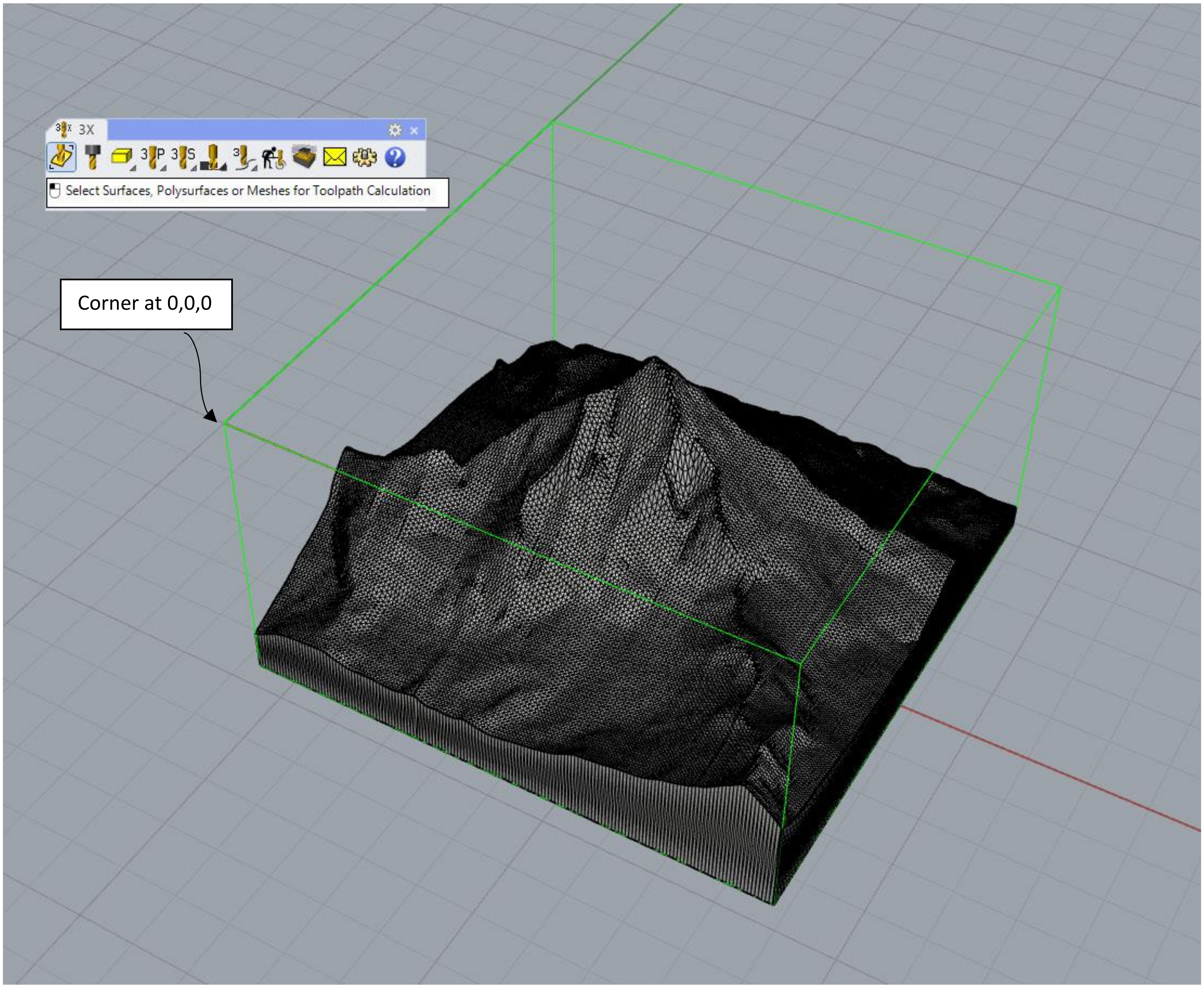


Select Surfaces to be milled
Create and define cutting tool parameters
Create or modify work piece, define regions or clipping planes
Create 3 axis tool paths for profile cutting
Create 3 axis tools paths that follow your selected surfaces
Surface remachining
Create 3 axis tool paths that follow curves
Recalculate a tool path
Graphics simulation of created toolpaths
Convert toolpaths to a CAM file and export for the Large or Small Mills

Creating Tool Paths:

For MadCAM to correctly generate a tool path, the process must be followed precisely. If a mistake is made: you need to change your units, update your geometry, or move your part to the correct location; you will need to start over by reselecting your surfaces after you make these changes.

Step 1: Click the first button in the MadCAM toolbar titled ***Select Surfaces***. Follow the prompt box and select the surfaces or meshes that you would like to mill, then hit enter.

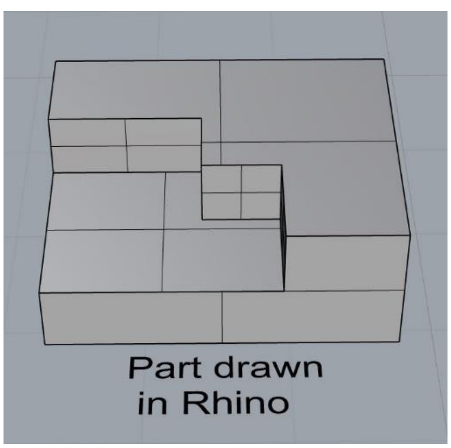


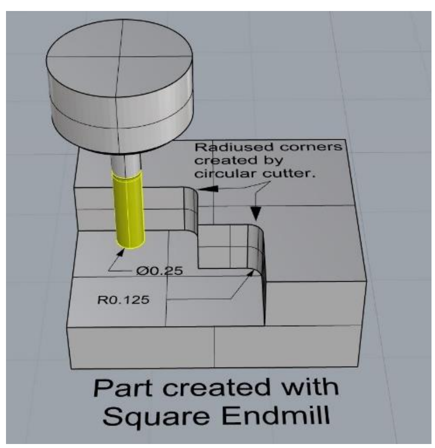
Once you have loaded your surfaces, polysurfaces or meshes, the object will have a bounding box created around it. Make sure that the bounding box lines up with the origin in Rhino and that your object is positive on the X and Y axes and negative on the Z axis. This bounding box should be the exact same size as the material that you want to mill your object out of.

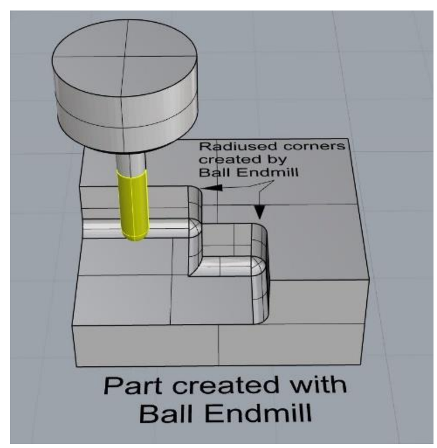
Step 2: Choose an endmill that is appropriate for creating the object that you would like to mill. When choosing an endmill there are a few major factors that will help you decide which endmill will give you the best result. These factors include: overall ***size*** of the object, the general ***shape*** of the object and the level of ***detail*** you wish to achieve. These factors will help you determine the Diameter, Length and Shape of your endmill.

Size: The overall size of your model will be the number one deciding factor on the size of your endmill. An endmill with a larger diameter will allow you to remove material more quickly and will be more appropriate for a larger sized model. The length of your endmill must also be long enough to reach into the deepest parts of your model without colliding the tool holder into your work piece.

Shape: The most common shapes for an endmill are ***square*** or ***ball*** endmills. This refers to the shape of the tip of the endmill. Each has its advantages and disadvantages when creating certain geometries. A square endmill is better at creating faces that meet at right angles and a ball endmill is better at creating curved sloping surfaces. Both will give you a radius equal to the radius of the endmill on vertical inside corners.



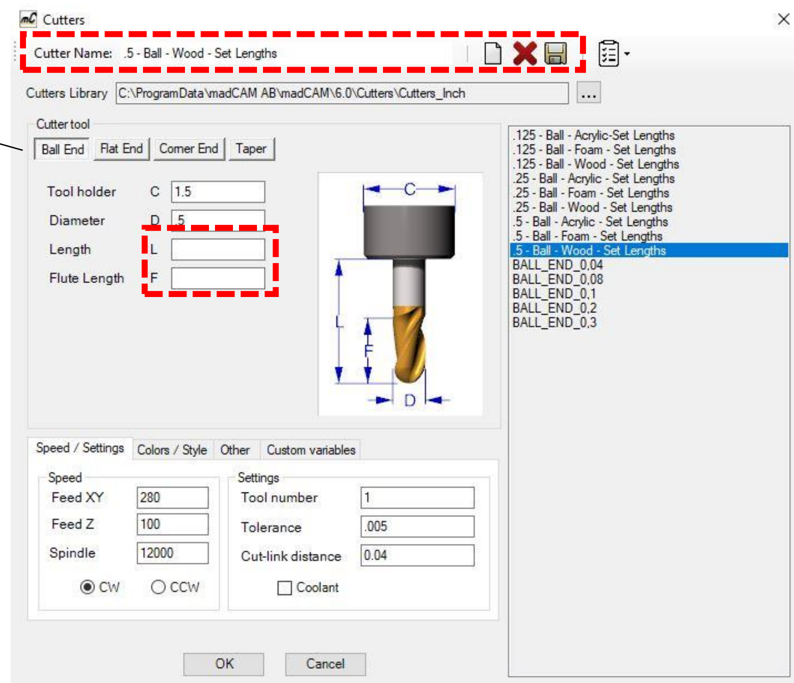




Detail: The last thing to consider is the level of detail you need the mill to create. You should use an endmill that is smaller in diameter than the features you are trying to mill into the surface of an object. For example a ½" diameter endmill would not be able to create a groove that is only ¼" wide. You may also choose to remove the bulk of the material with a larger endmill and then swap it out for a smaller endmill to create the detail you are looking for.

Step 3: Define your endmill. Once you have decided which endmill you want to use, you will need define it in MadCAM. Click the ***Create/ Load Cutter*** button in the MadCAM toolbar and carefully define the endmill you have chosen in the cutter parameter window.

Step 4: Load a predefined endmill based on the material you want to cut by selecting it from the list of endmills in the column on the right. This will load the predefined Feed and Spindle Speed rates for the endmill you chose. The density of the material you are cutting, the number of flutes on your endmill, and the diameter of your endmill are all taken into account when calculating your feed and speed rates. These calculations have been done for you and the correct values are stored in this bank for you to select from.



1. Start by selecting the shape of your endmill here.

2. Then choose the endmill from the list on the right that matches your diameter and material.

3. Set Length L to the length of your endmill after it has been attached to the tool holder. (Usually about 0.75" shorter than the overall length.)

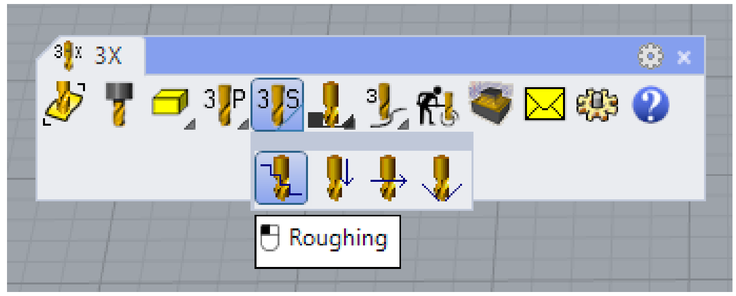
4. Set Length F to the length of the cutting edge of your endmill. (Typically ~1")

5. At the top of the dialog box where it says "Cutter Name", input a unique name for your endmill and click the save button on the right to save these settings for later. Once you have saved your settings, your endmill will now show up in the list of predefined cutters.

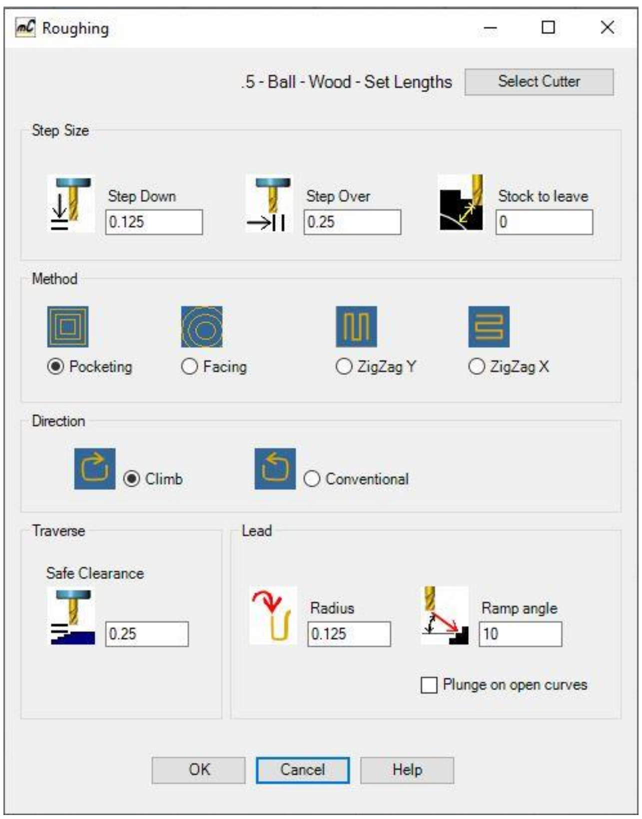
6. Hit **OK** to make this tool active. Any toolpaths that you create will now be pathed for this endmill until you define a different one.

If your endmill is not in the predefined list, you can start with an endmill that is most similar and adjust the settings as necessary. Your Feed XY should not exceed 280 in/min, Feed Z should not exceed 100 in/min and, Spindle should not exceed 12,000 rpm. These settings can also be adjusted up or down on the CNC mill if necessary.

Step 5: Generate a roughing toolpath. The purpose of a roughing pass is to remove the bulk of the unnecessary material as quickly as possible by milling away layers from the top of the block down to the surface of your object. To begin creating a rough pass, click the ***3-Axis surface milling*** or "***3S***" button on the MadCAM toolbar and then select the ***Roughing*** option to open the roughing pass dialog box.



Step 6: Adjust your roughing settings accordingly. *(**Note:** The settings shown here are for a 1/2" ball endmill cutting wood. Your settings may change depending on your project.)*



Step Down: The distance between each layer that is removed as you move down the part. A larger step down means less mill time, but increases the load on your endmill. This value will default to the optimal setting of ***1/4 the diameter of the endmill***. Your step down should not exceed 1/3 the diameter of your endmill when cutting wood or other dense materials. A larger step down is acceptable for less dense material such as foam, however your step down should never exceed your flute length.

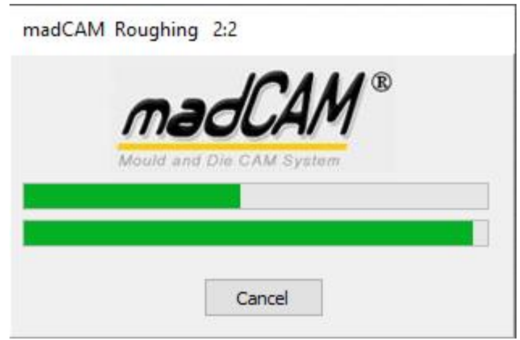
Step Over: How far the endmill will shift over after each pass. Your step over will default to ***1/2 the diameter of the endmill.*** In general this is the most optimal roughing step over. If you are milling a softer material like foam you could increase the step over up to a maximum of 2/3 the endmill diameter to save time.

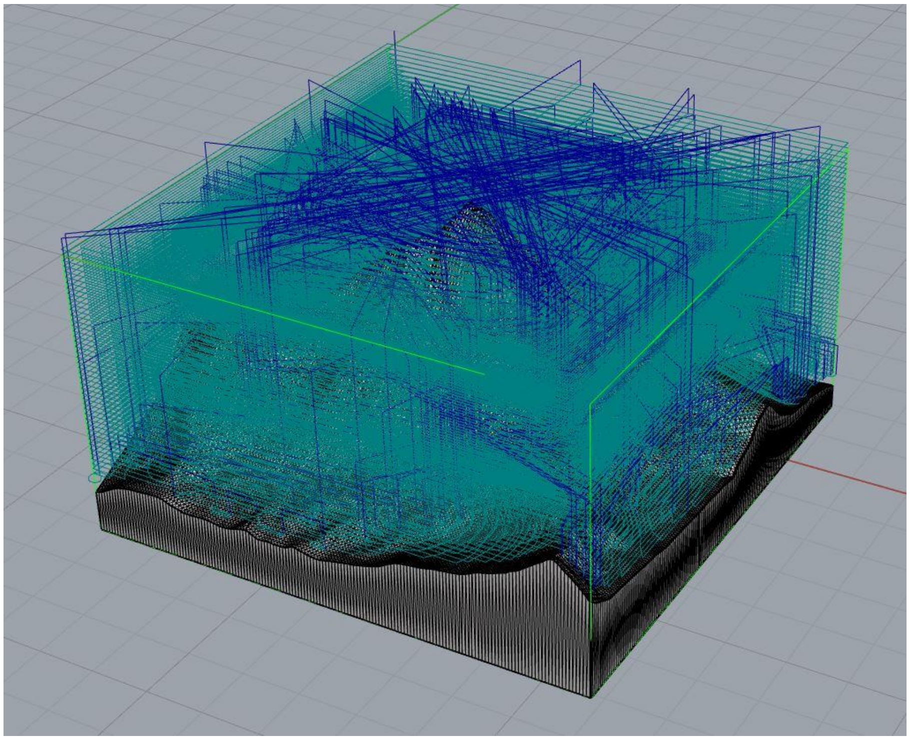
Stock to Leave: Offset distance from your surface to the roughing pass. ***Default is 0"***.

Direction: The order and general organization of the passes as they move through your geometry. ***Pocketing*** will be the default setting and most likely the fastest option as well. You may consider using a ZigZag Y or ZigZag X path if you are milling a larger object.

Safe Clearance: When the endmill needs to move across your part without cutting, it will move up to this distance to ensure it will not collide with your part before moving sideways. This setting will default to 1/2 the diameter or your endmill, however you should ***always set this to a minimum of 0.25".*** If you are using a modified step down, always ensure that your safe clearance is greater than your step down.

Once your settings are entered and you click **OK**, MadCAM will display this loading bar and begin to calculate your roughing toolpath. Once your path is calculated you will see the toolpath lines covering your part.

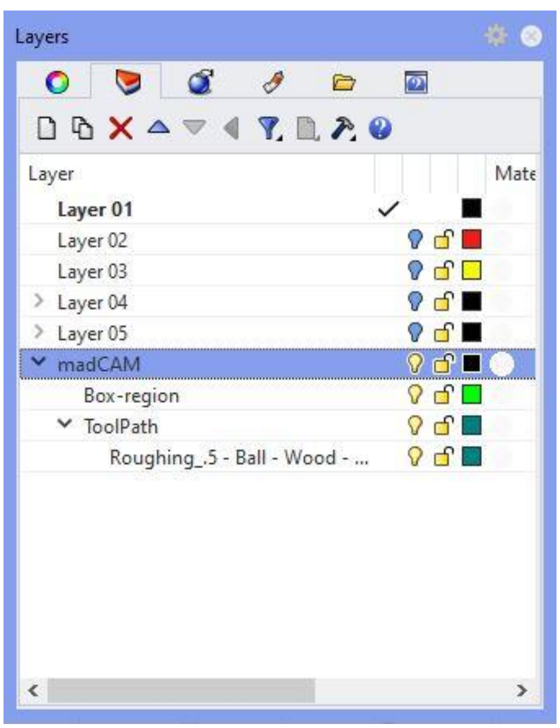




Blue lines indicate safe clearance movements where the mill is not touching your part.

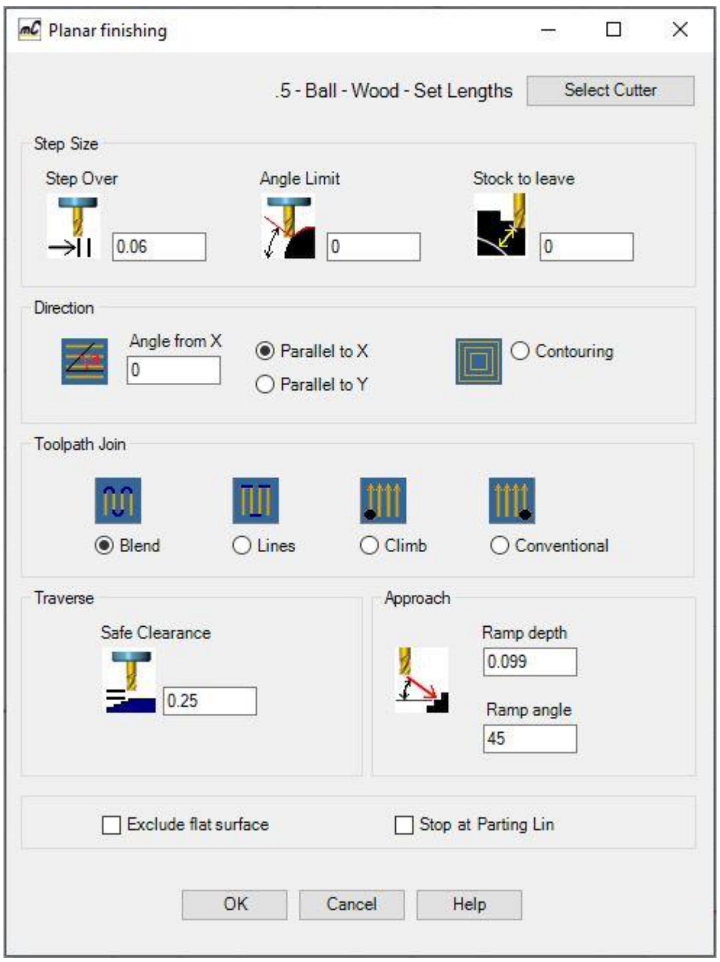
Green lines indicate cutting paths where your endmill is cutting through your material.

All of these lines are indicating where the tip of the endmill will be traveling. You will notice that these lines will be offset from your part by the radius of your endmill.



MadCAM will store your toolpath and box-region data in Rhino's layers window. Each toolpath will automatically be named after the type of path it is, followed by the name of the endmill that was used to create it. The layer order of your toolpaths is important as they will export in this order when you generate your CAM file. Always make sure that your rough paths are on top and any other paths are below. You can also prevent a tool path from exporting by clicking off the light bulb next to that layer. You may export more than one toolpath at once, just be sure that they are all pathed for the same endmill. When you are working on a larger object, it is wiser to export each of your paths as separate files so that you can be sure you have enough time to run each one while the shop power is on. When you export your file later on, MadCAM will provide a time estimate for the paths that are turned on when you export.

Step 7: Create planar finishing paths. Finishing paths will follow tight to your surfaces and are used to remove any material left behind by the roughing path. A planar finishing path will move back and forth along parallel lines across your part. In most cases you will want to create two separate planar finish paths that run perpendicular to each other to create the smoothest possible surface. Start by clicking the ***3-Axis surface milling*** or “***3S***” button on the MadCAM toolbar and then select the ***Planar Finishing*** option to open the Planar Finish dialog box.



Step Over: The distance the endmill will shift between each parallel pass. Your finishing step over should be much smaller than your roughing step over. In general the smaller your step over, the more detail you will uncover in exchange for a longer mill time. A good base setting would be ***between 1/5 to 1/3 the diameter of your endmill***. A smaller setting will always result in a better finish, especially with a ball endmill.

Angle Limit: This setting will prevent your planar tool path from touching any surfaces below the desired angle measured from 0. If you do not wish to limit your tool path, ***set Angle Limit to 0***.

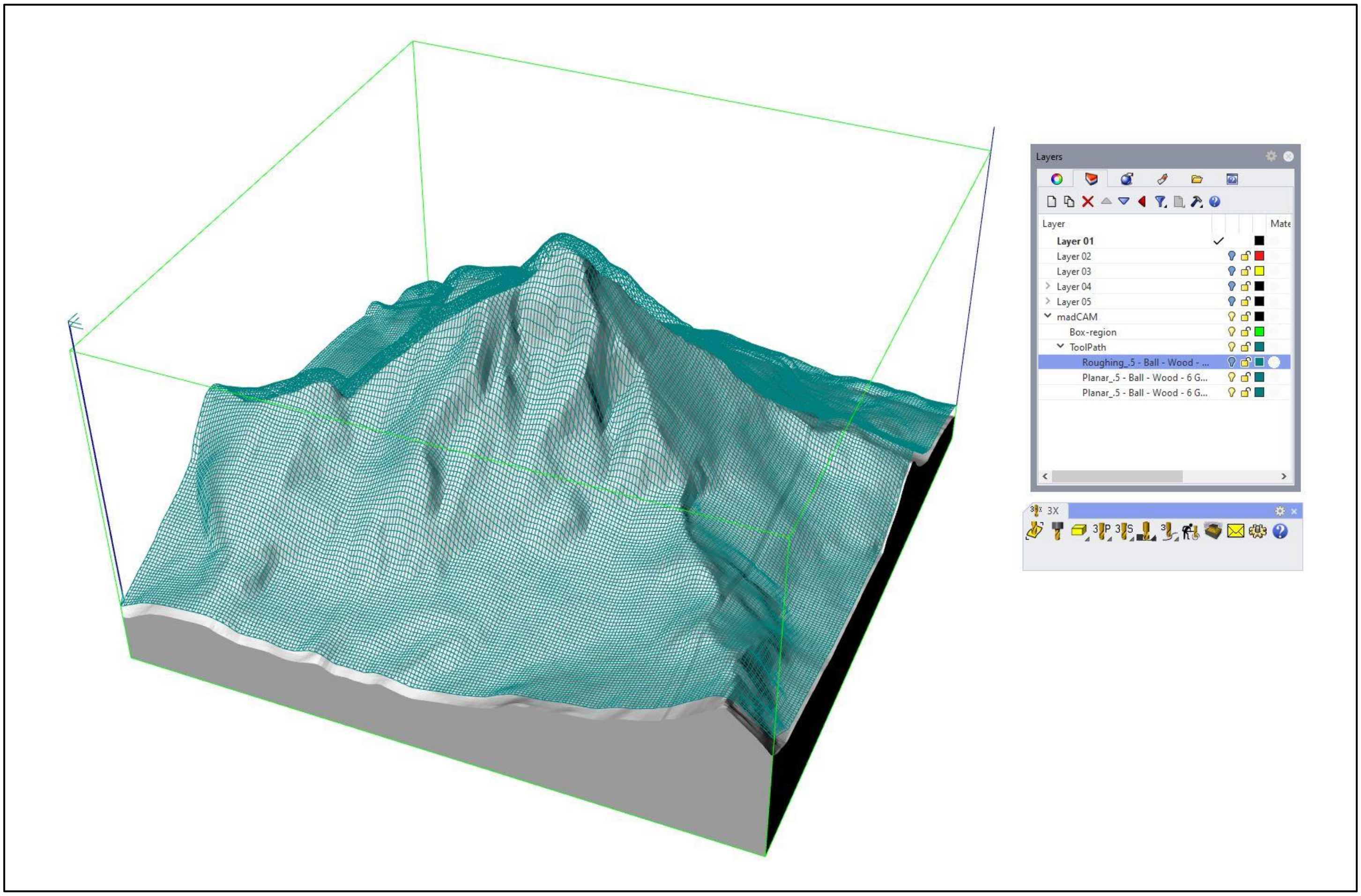
Stock To Leave: Offset the finish path above or below the surface. ***Default is 0”.***

Direction: You can set your paths to be parallel to the X or Y axis, or you can set the paths at an angle defined from X. For the best finish, create two Planar Finish paths that are perpendicular to each other. Your first finish path should be cross grain if you are milling wood.

Toolpath Join: Defines how your parallel passes are connected to each other at the end of each pass. ***Blend*** is the most efficient connection.

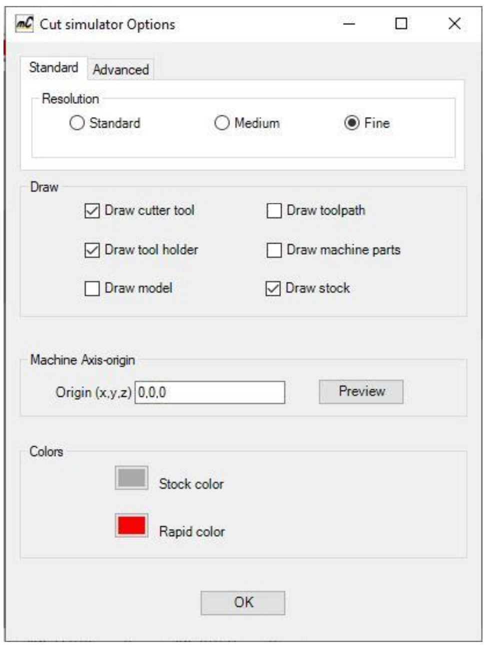
Safe Clearance: When the endmill needs to move across your part without cutting, it will move up to this height above your surfaces to ensure it will not collide with your part before moving sideways. This setting will default to 1/2 the diameter or your endmill, however you should ***always set this to a minimum of 0.25”.***

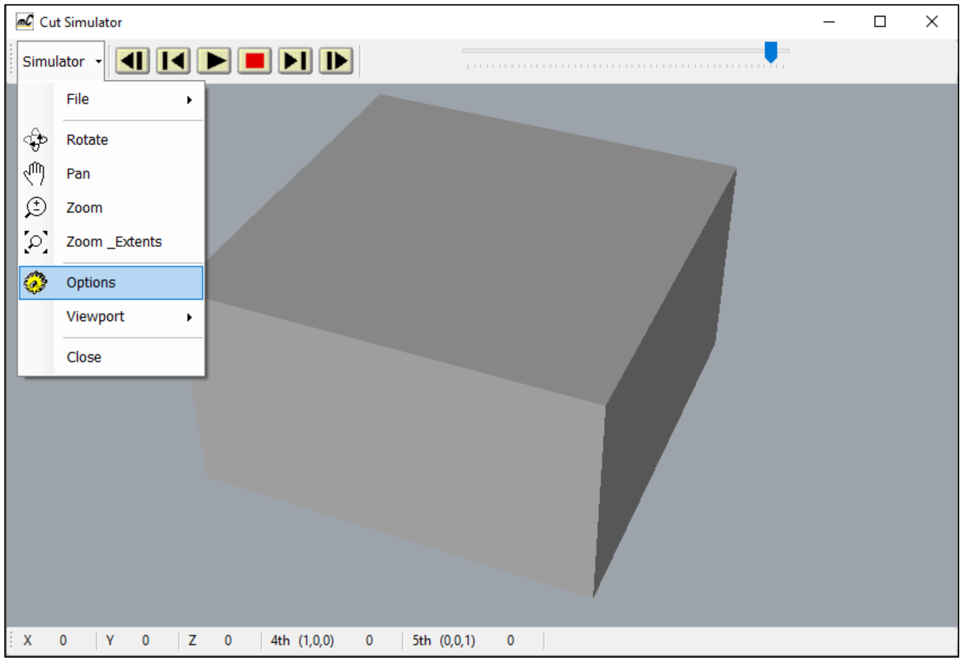
Step 8: Examine your toolpaths. Once your finishing toolpaths are created you can temporarily turn off your roughing path in the layers menu to get a better look at just your finish paths. Take a look at how your paths are interacting with your surfaces and make sure they will capture the detail you are looking for. Keep in mind that these lines are showing you the location of the tip of the endmill and will be offset from your actual surface by the radius of your endmill. In this example you can see the two perpendicular finish paths have created a grid pattern over the part. The quality of the finish path on the X axis starts to degrade on the steeper parts of the peak, but the finish path on the Y axis will recapture any detail that is missed in the same area.



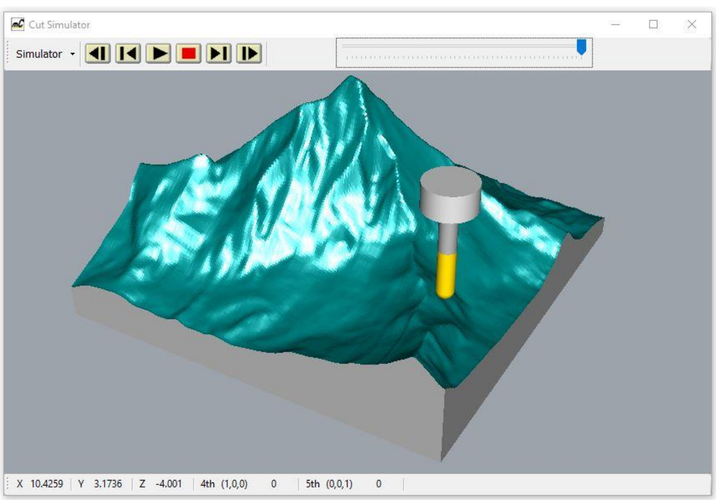
Step 9: Simulate your tool paths. You may also wish to simulate your toolpaths to ensure there are no collisions and to get a preview of your finished part. First, make sure that all of the toolpaths you wish to simulate are turned on and are in the correct order in the layers menu. The simulator will only be accurate if you saved your endmill settings to the tool library before generating your toolpaths. Once your toolpaths are setup correctly and you are ready to simulate, click the ***Simulate*** button in the MadCAM toolbar.

The buttons at the top of the simulator window are used for controlling the simulation and the slider on the right side will adjust the simulation speed. To get the best simulation you may need to adjust some settings. Inside the Simulator window, click the simulator drop down menu and then select the ***Options*** menu**.**

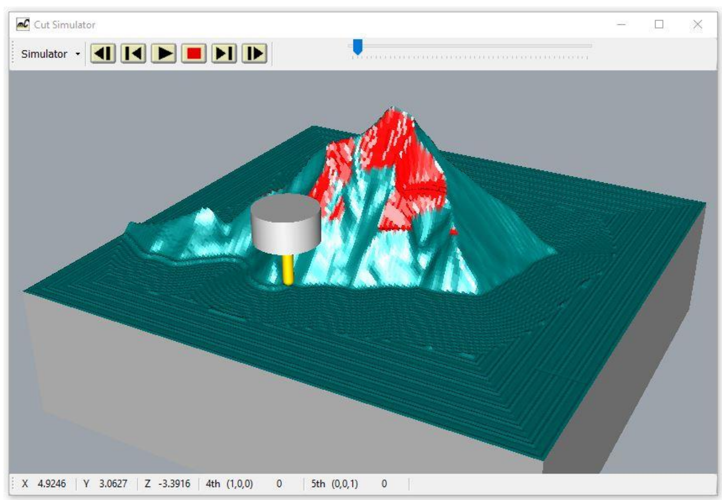




In the options menu, ***set the resolution to Fine***. You can also adjust the settings of what is displayed in the simulation. Typically you will want to ***turn on Draw Cutter Tool, Draw Tool Holder, and Draw Stock, then turn off Draw Model***. Once you have changed these settings you may need to ***close and then reopen the simulator*** to load any changes that you have made. Always make sure to save your progress before clicking play on the simulator as there is a possibility the simulator will crash Rhino, especially with larger files. Press the ***Play Button*** to begin the simulation.



What you see at the end of the simulation is a rough approximation of what you will get. Typically the simulation will be a bit rougher than your actual part even after changing the settings to fine.

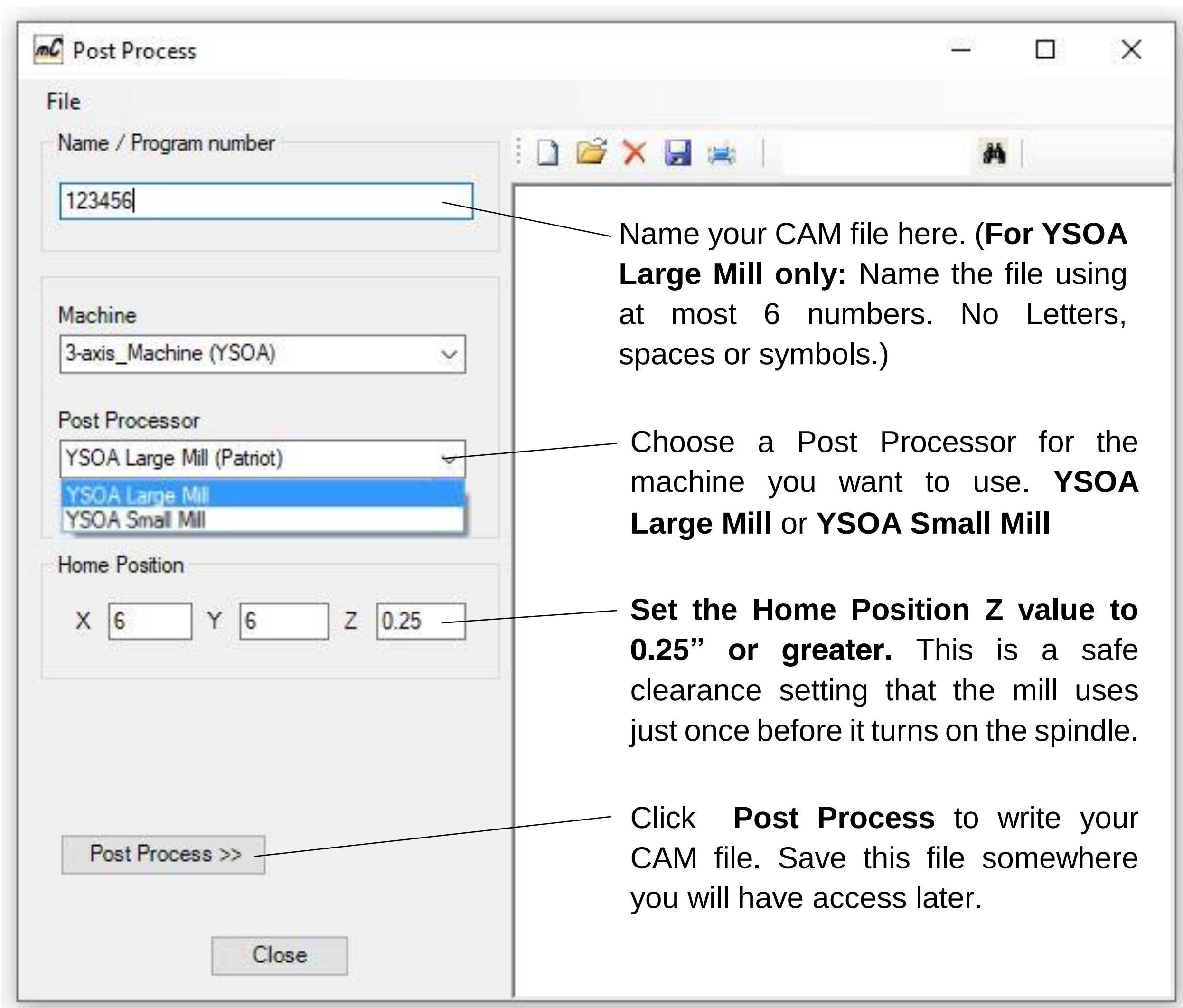


The simulator will display red areas if you have created a tool path that will cause collisions. In this example the endmill used is too short causing the tool holder to collide with the part. This toolpath would be unsafe to run and a longer endmill would be required.

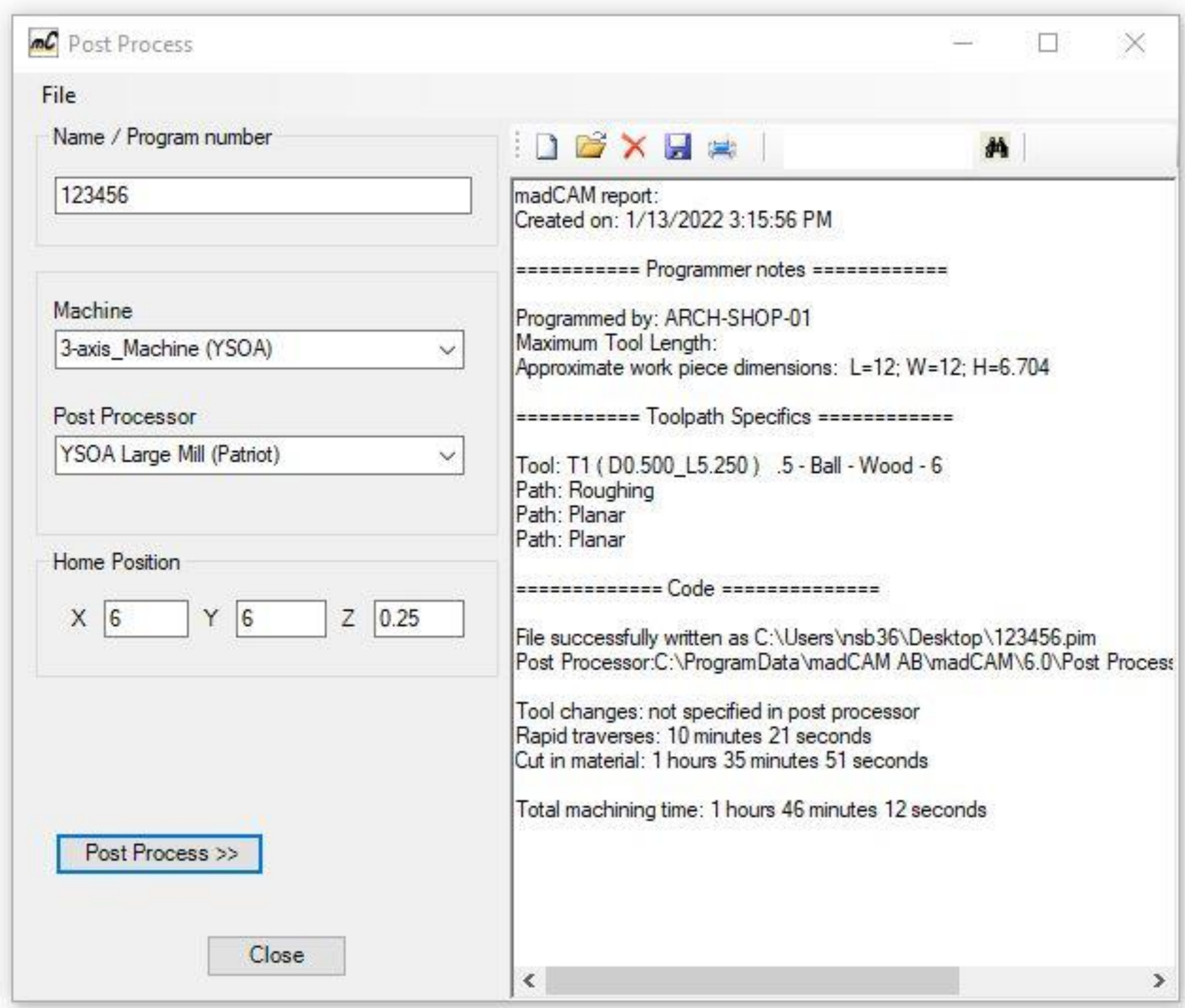
Exporting CAM Files:

Step 1: Organize your toolpath layers. Make sure your toolpaths are turned on and organized properly in your layers menu. Always run your rough paths first and finish paths last. Any toolpaths that you would like to run with the same endmill can be exported to the same CAM file. If you are planning on using multiple endmills in the same job, make sure to manage your toolpath layers so that they are grouped by the corresponding endmill. Then you will export separate CAM files for each endmill.

Step 2: Click the ***Postprocess*** button in the MadCAM toolbar and verify the following settings.



Once your file is saved you will see the following information in the Post Process window. There is a lot of useful information in this window such as where the file was saved, the size of your block, the endmill you pathed for, the order of the toolpaths, and a time estimation for your entire file. Typically rapid traverse times are under estimated while cut in material times are pretty accurate. The best way to reduce your mill time is to plan your paths strategically to reduce the number and length of your rapid traverse paths.



Step 3: Move your CAM file to the mill. The small mills are connected to a computer that is on the network so you can just save your file to a shared folder and then move it to the desktop of the computer connected to the mill. Never run your file from the network or a flash drive. The large mills are not connected to the network in any way so you will need to save your CAM file to a flash drive and move it to the large mill's hard drive from your flash drive. (*Flash drives must be formatted with a "fat 32" file system to be recognized by the large mill. See an AT Staff member for assistance with this if needed.*)

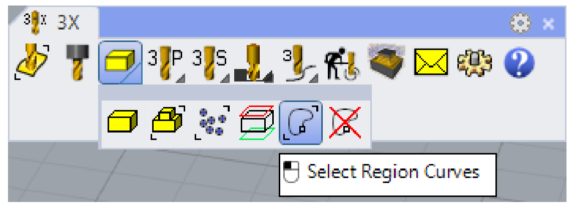
Additional Options

Setting Region Curves:

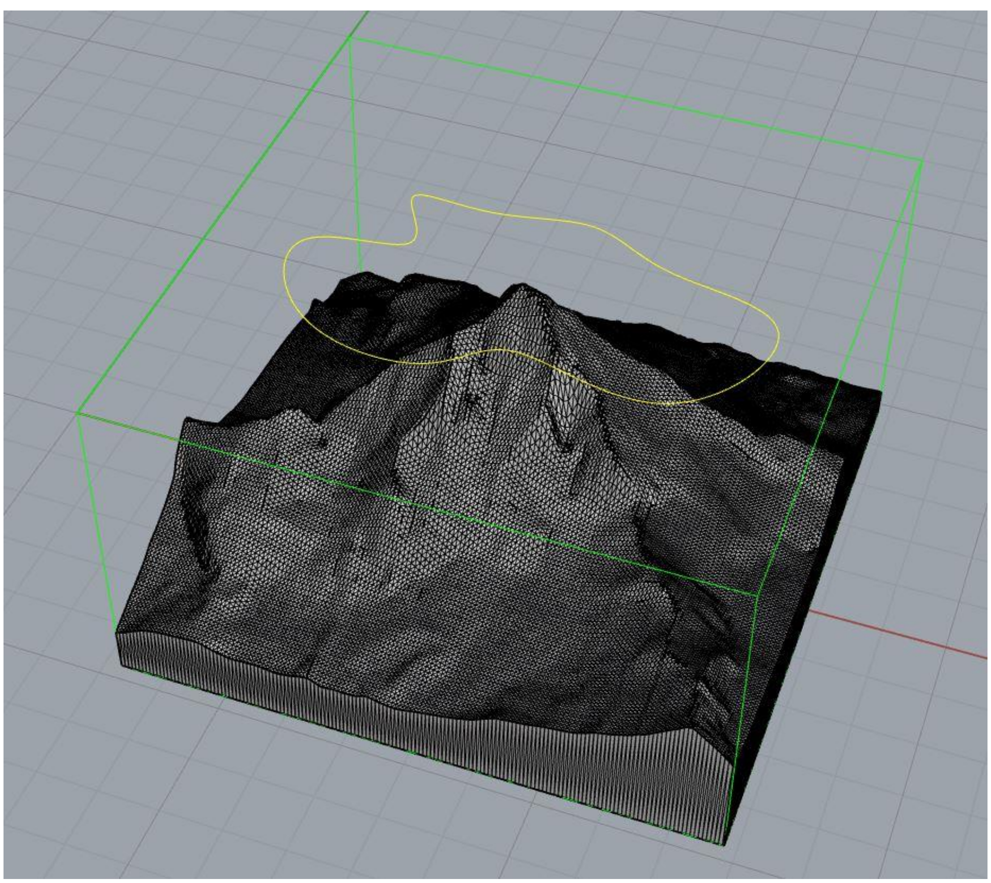
Region curves will allow you to refine the area that you would like to mill by restricting your toolpaths to the inside of a closed curve. This is a useful tool that can help you achieve a higher level of finish on certain areas without having to generate a toolpath that covers the entire model, greatly reducing the overall mill time. Region curves can also be useful for breaking up a larger roughing toolpath down into multiple smaller paths thereby shortening the length of your rapid traverses and your mill time.

Step 1: Create your region curve. Your region curve does not need to be 2D**,** but it ***must be a closed curve*** that encompasses some part of your model when you look at it from the top view.

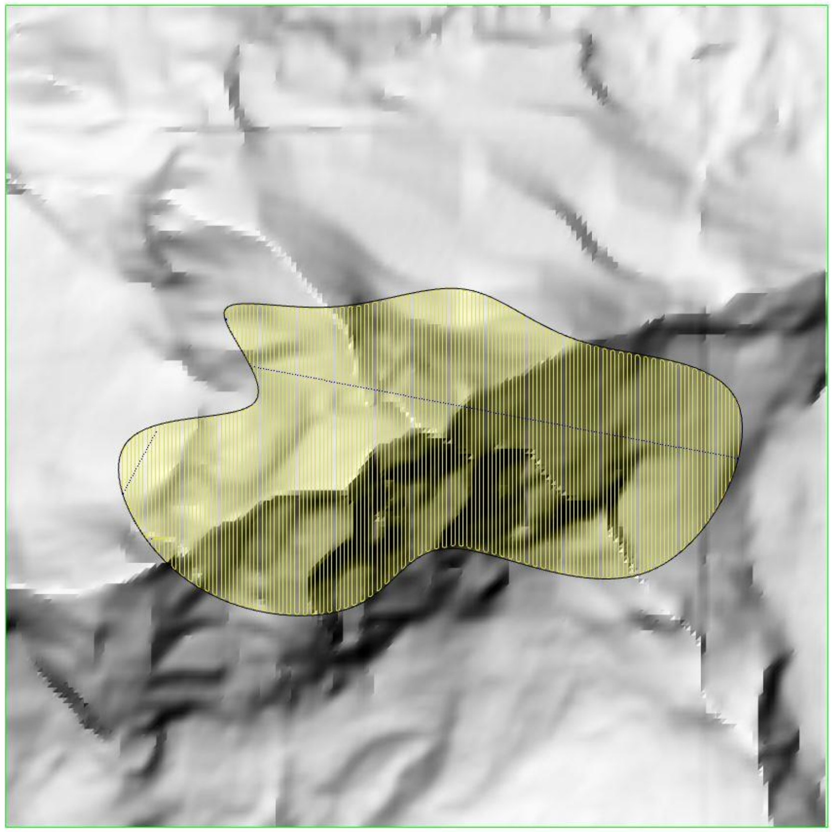
Step 2: Click the bottom right corner of the ***Create Box*** button to cascade the ***Regions*** menu, then click the ***Select Region Curves*** button.

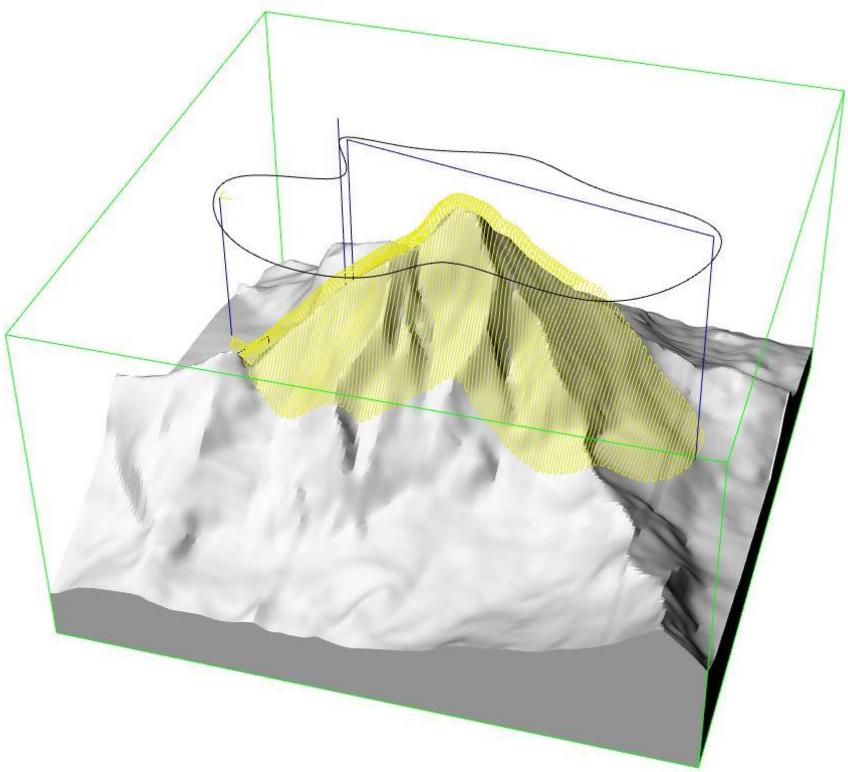


Step 3: Select your desired region curve and hit enter. Any toolpath that you create from this point forward will be limited to this region.



Examples of a planar path limited to a region:

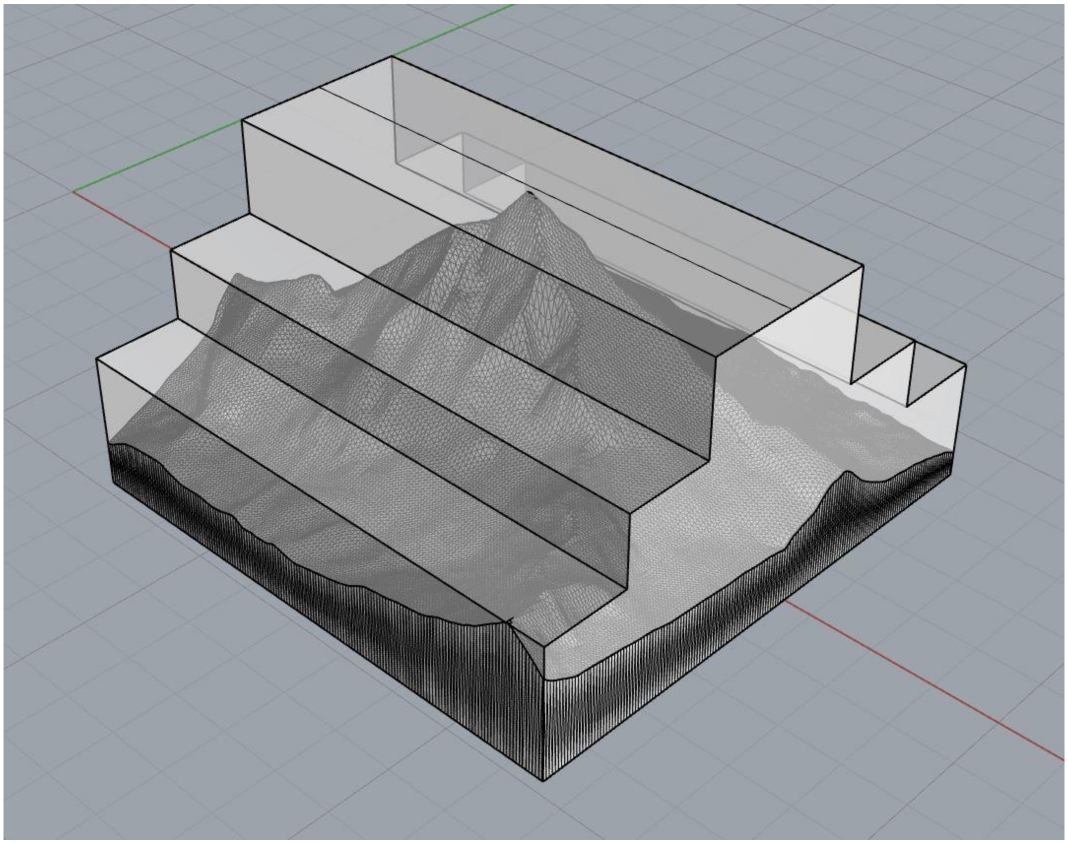




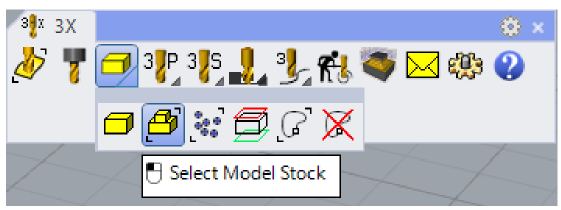
Defining Model Stock:

In most cases you will be milling a rectangular block of material which is exactly what MadCAM will assume by default. If your material is not a rectangular block, you should draw the shape of the material in 3D as accurately as possible and then locate your part within the block. Be sure to line up the top plane of the material at the zero point in Rhino to make setting up the mill easy later on. In some situations you may want to reduce the volume of your material before milling by gluing up several smaller blocks. This will allow you to save time and material by reducing the amount of work your rough pass will have to do.

Step 1: Draw your material as accurately as possible in your Rhino file and fit your part inside of it. If you are gluing up blocks to create your model stock, you may want to glue it all up before drawing it so that you can account for any small deviations.



Step 2: Define your model stock in MadCAM. Click the bottom right corner of the ***Create Box*** button to cascade the ***Regions*** menu, then click the ***Select Model Stock*** button.



Step 3: Select the model that defines your material and hit enter. Once your model stock is defined, you can hide it so that you can see your part.

Step 4: Generate your roughing toolpaths. When your rough pass is calculated it will follow to your specific model stock.

Example of Rough passes generated with a special model stock: