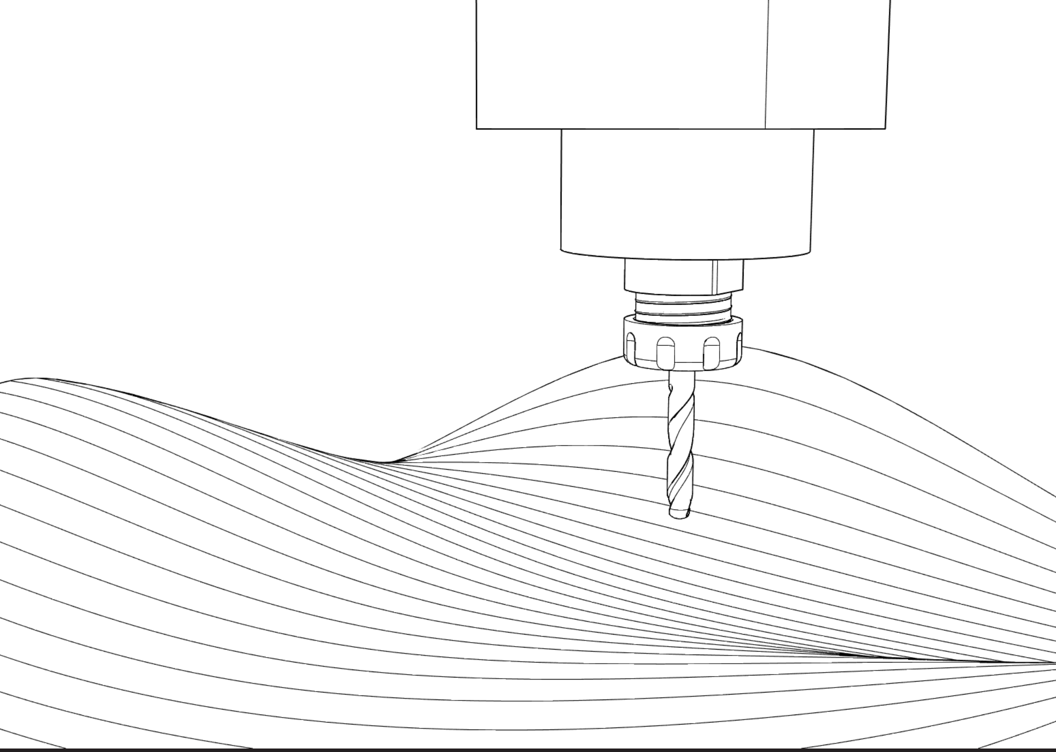
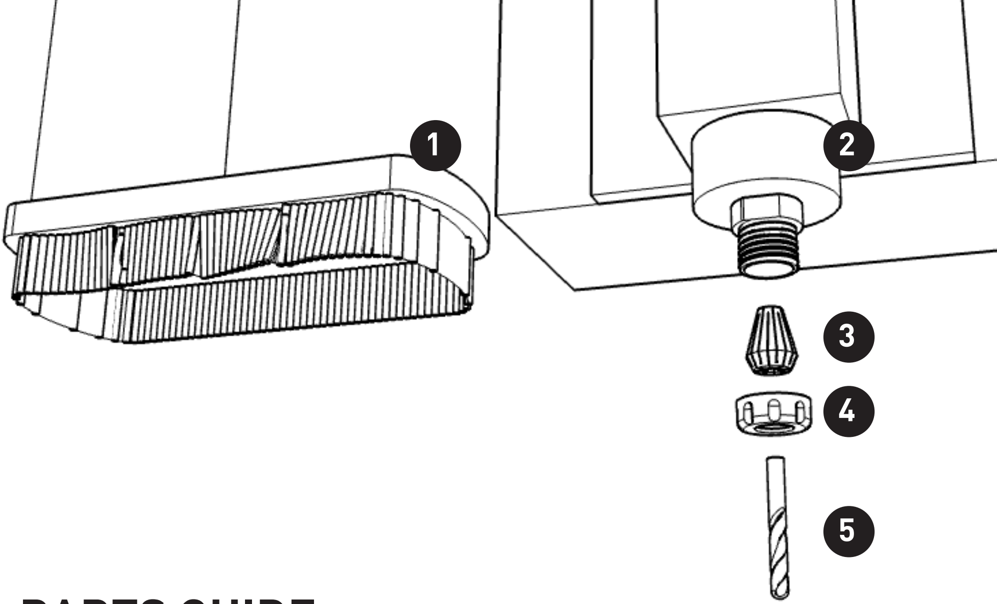


CNC PART 2 : STARTING 3D
GSAPP FABRICATION LAB 2016

this is a the second part of a student guide for skill-building and proficiency in the use of the CNC machines in the Fabrication Lab at Columbia GSAPP

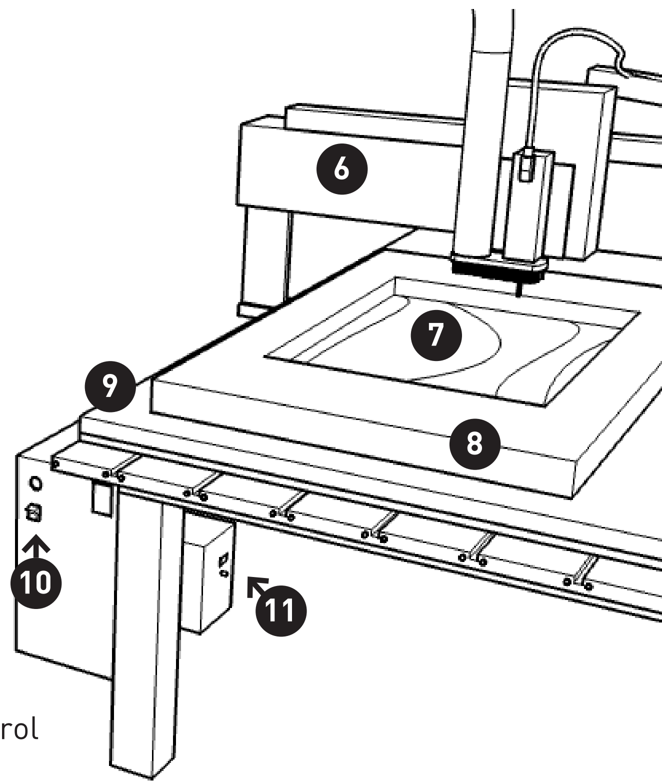
...upon completion of this walkthrough, one will be familiar with the modeling, handling, and setup of three-dimensional CNC toolpaths for volumetric digital models in Rhino 3D. unlike the intro tutorial, this guide does not prescribe a particular project; rather, it allows you to apply the walkthrough to your own work.

...for **clarification** of any content, or for extended study into the more advanced operation of the mill, talk to the GSAPP Fab Lab shop monitors, or check the Fab Lab schedule for events.



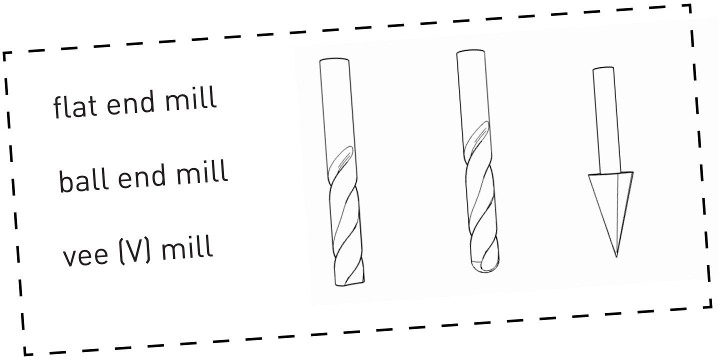
PARTS GUIDE

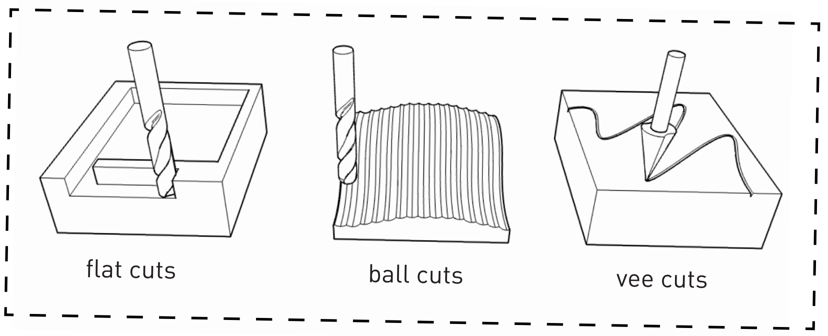
1. dust foot
2. spindle
3. collet
4. collet nut
5. end mill (bit)
6. gantry
7. part (your model)
8. stock (material)
9. spoilboard (table)
10. power
11. spindle speed control

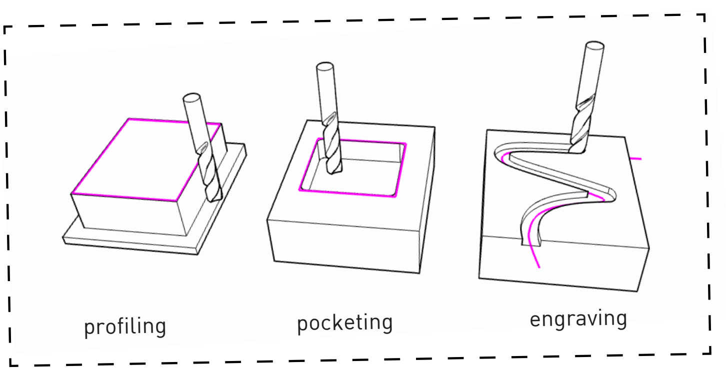


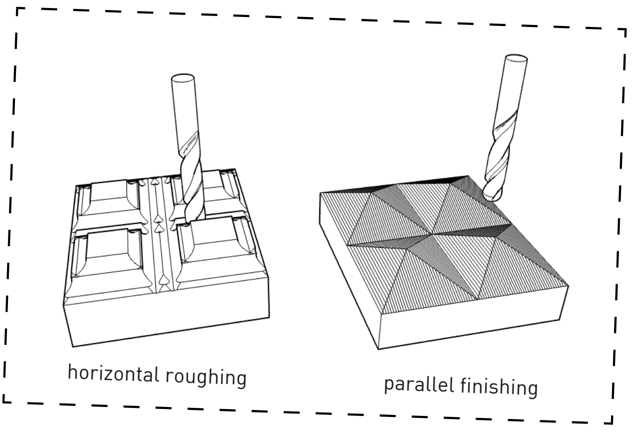
GLOSSARY

finishing	a final, slower operation intended to produce the desired shape, texture, and surface to the shape of your part **precisely**
horizontal	an operation that removes material in flat, horizontal **layers** and steps down between layers to shape the material downward
operation	a **single** cutting toolpath programmed in RhinoCAM. examples include profile, engrave, parallel finish.
parallel	an operation that removes material one small line at a time, **stepping over** gradually along parallel surface contours
post	short for post-processor. both a term for the piece of software that converts your RhinoCAM data into **coded instructions** for the CNC machine, and a term for the process of that conversion.
roughing	a preliminary operation to **excavate** lots of material quickly, with comparatively **low** precision, in preparation for a finish.
scalloping	small **ridges** left between adjacent tool passes on your material, a result of using a round bit. larger bits and smaller stepovers will result in less scalloping. scalloping can be used for **effect**.
setup	a **group** or layer of operations organized in RhinoCAM browser. provides the ability to chain several operations together.
spoilboard	the MDF **table** below your part. scuff it, but do not destroy it.
stepdown	a setting parameter that determines how much the bit will **plunge** between horizontal levels. generally, keep below 50%
stepover	a setting parameter in some operations, amount that the bit will **shift** sideways between passes in a larger operation. generally, keep at or below 50%.
stock	a term for the piece of **material** you intend to cut. also a term for a virtual stock that you will define in RhinoCAM to help establish the boundaries of your parts.
tabs	small **bridges** of material that connect your cut parts back to the larger stock. some operations have automatic options for tabs, 3D operations rely on modeling and placement of custom tabs.
toolpath	the programmed path of your cutting bit as determined in RhinoCAM, shown as a series of previewable lines in Rhino

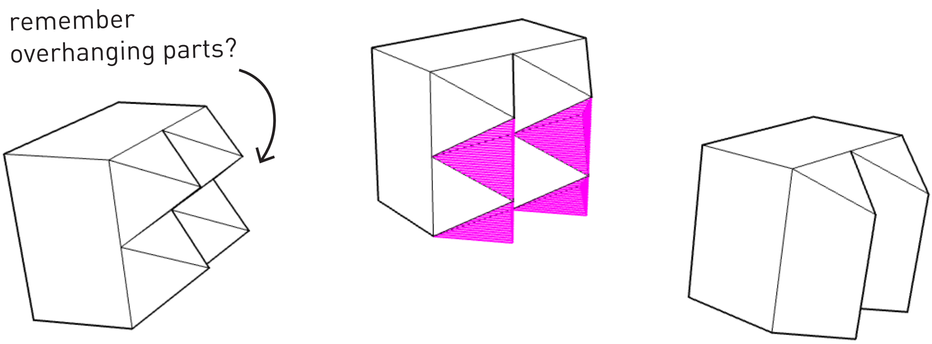

flat end mill
ball end mill
vee (V) mill


flat cuts
ball cuts
vee cuts


profiling
pocketing
engraving


horizontal roughing
parallel finishing

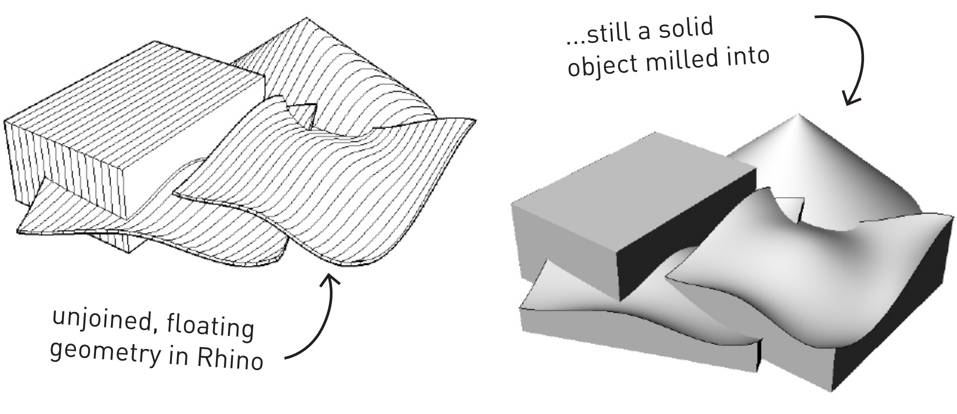
STEP ONE :
UNDERSTAND WHAT RHINOCAM SEES



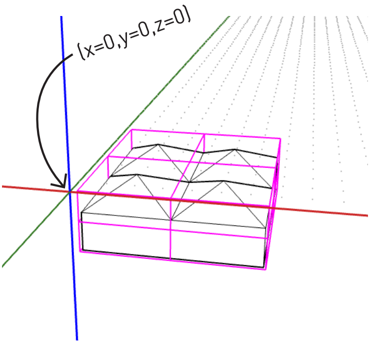
RhinoCAM takes a relatively simple view of your digital model.

...your model need **not** be **watertight**, it can be **either** volumes or surfaces, you can have **disconnected** surfaces, and nothing needs to be actually **combined**, joined, or **boolean**'ed together.

RhinoCAM simply sees the geometry you have in your model as if it is looking from directly above. As such, it **will not** see **undercut** areas (see above), and it **will** see just the upward facing portion of any and all **surfaces**, **meshes**, or **polysurfaces** in your model as they are modeled,

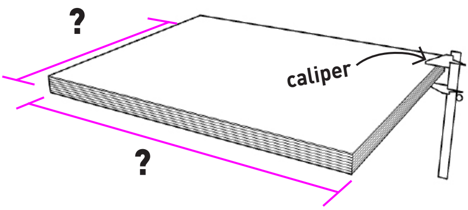


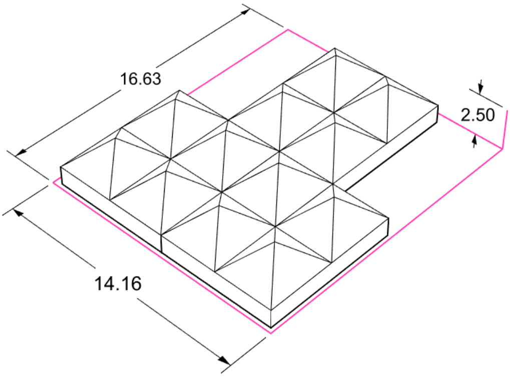
STEP TWO :
SIZE IT ALL UP



as discussed in the intro tutorial, the orientation of the CNC mill and conventions of modeling suggest that you model everything in the {+x, +y} quadrant in Rhino, and make sure that all parts of your model are beneath Z=0.

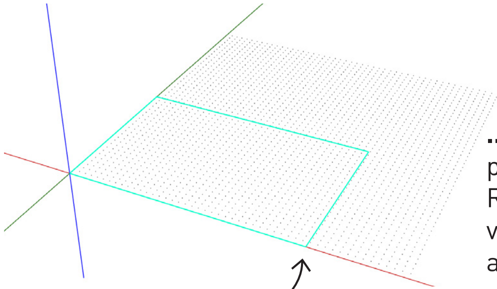
measure your material. these dimensions will serve as guides while setting up your cut file in Rhinocam. you will need a piece of material that is large enough to contain all your parts AND to allow for space to add material connections and support.



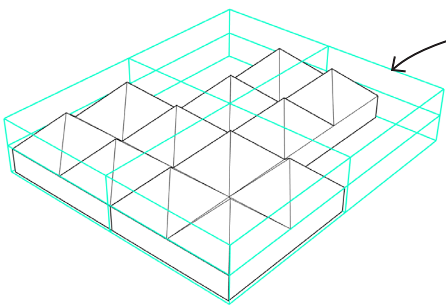


ensure that the material you have is big enough, or **scale** your digital model to fit within it.

you can glue or **laminate** several layers of material together to build up thickness... of course, this needs to be done **hours** before milling.



...draw the extents of your physical material into your Rhino model to give yourself a visual aid for the area you have available for working.



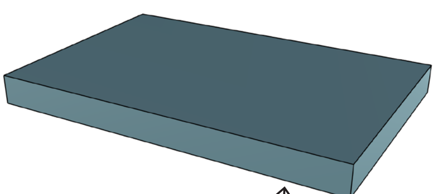
it can help to make a wireframe **guide box** around your model to assist you in locating it properly.

a quick way to do this is to select your model geometry and:

-**[BoundingBox]**

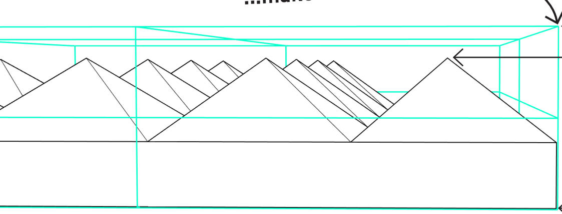
-**[ExtractWireframe]**

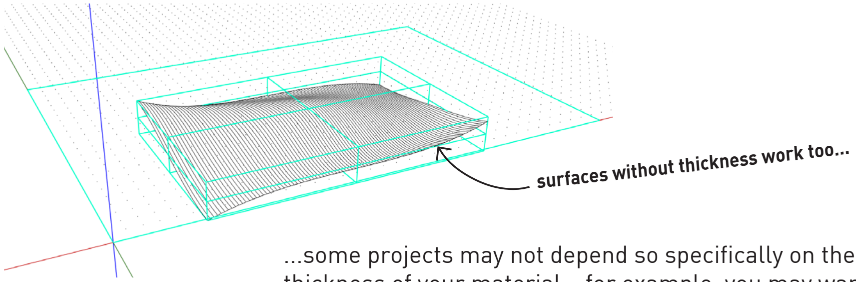
-**[Group]** (and delete the solid box)



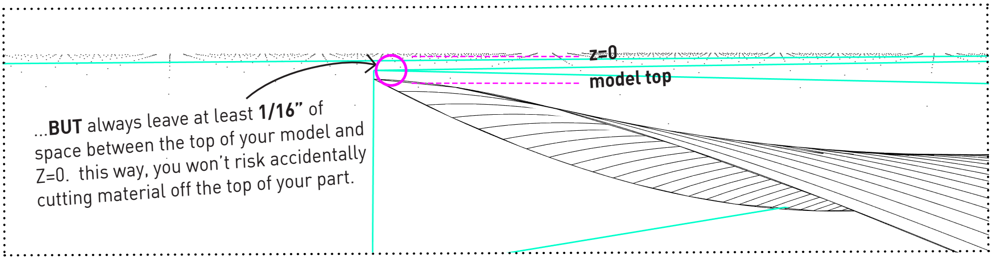
...take the accurate measurements of the thickness of your material, and **scale** (scale1D) your guide box to match that thickness.

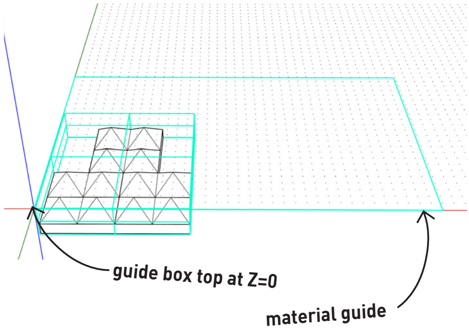
-remember that your part needs a flat side to sit on the CNC table, so your model should back up against the "**bottom**" of your guide box.

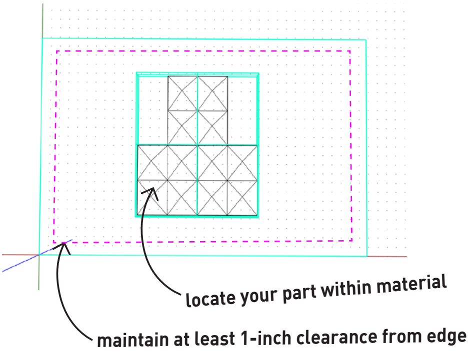




...some projects may not depend so specifically on the thickness of your material. for example, you may want to mill a 3D surface into the **top** of a material regardless of how thick it is (provided it is at least thick enough to handle the z-depth of your Rhino surface). in this case, you could locate the top of your model close to the **top of the guide box** and near Z=0.







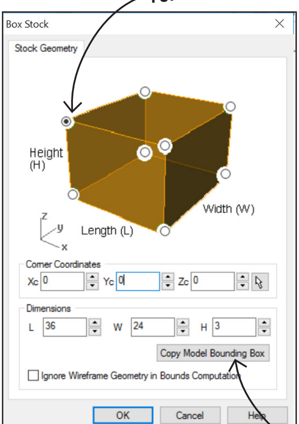
once your guide box is built around your model, and adjusted to the thickness of your material, **locate it all below Z=0**, as you did with the previous tutorial. move everything you want to cut sufficiently **inside the edges** of your material.

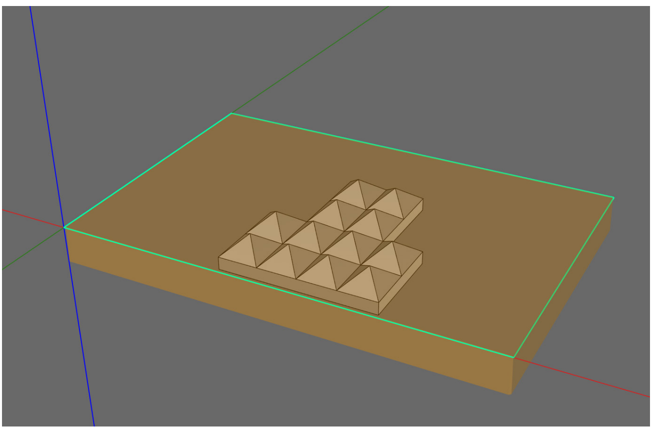
...also as seen in the previous tutorial, you should always (when you **can**) maintain at least an inch of free space around the perimeter of your material guide, to allow sufficient space to fasten the material down to the CNC table.

STEP THREE :
SET UP BOUNDARIES FOR YOUR CUTS

...in addition to your own **guides** drawn to visualize the size of your physical material and the margins to leave for fastening, you need to give RhinoCAM some information on how large your material is, so it can understand the extents to which it should be allowed to cut.

the best way to do this is to set your **stock** in RhinoCAM.   a correctly-sized stock will ensure that the CNC mill cuts no deeper than your material, and does not cut off the edge of your material.

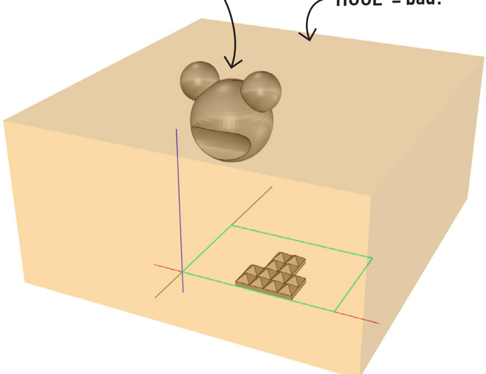




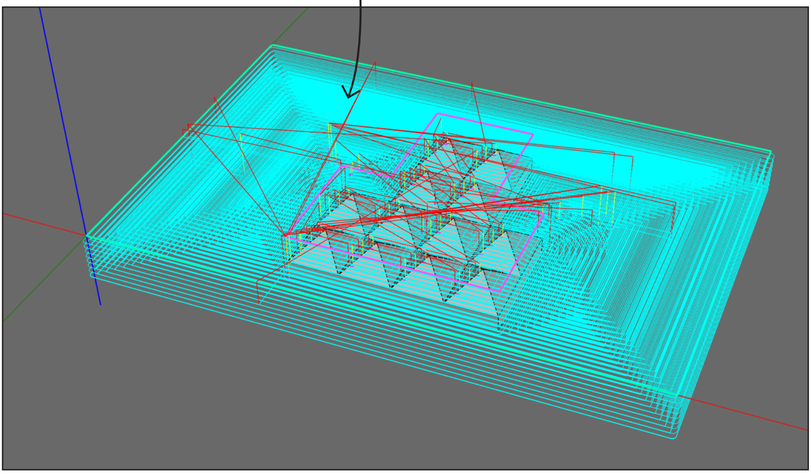
note: it's good practice that you prepare a file for the CNC in a **new, blank** Rhino file.

this will keep your workspace tidy and help avoid errors in calculating stock size.

the stock is not only important for telling RhinoCAM how **deep** your material is, it also tells the machine how high to set the **clearance plane**.   too high and your cut will not even start.



now that the stock size is set (and hopefully correctly), you can attempt to generate your tool operations. **HOWEVER**, without providing any other information to RhinoCAM, this is what you would get if you attempted to create a roughing pass:



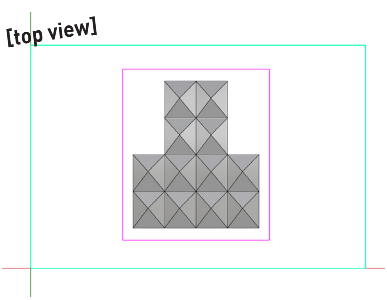
woops...

...what this shows is that RhinoCAM will attempt to remove **ALL** of the material in your stock around your model. this is clearly not only **unecessary**, but will result in **failure** once your part moves freely.

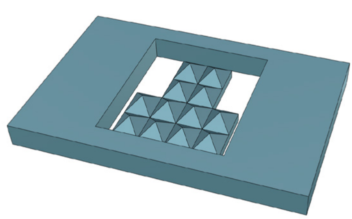
what you need to do is to define **boundaries** for the cut operations. there are a couple ways to do this.

1. a smaller bounding rectangle

[top view]

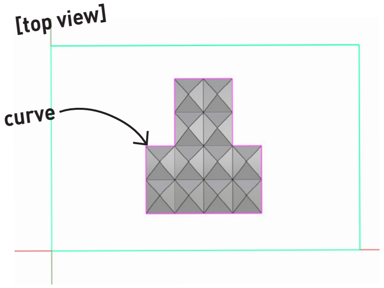


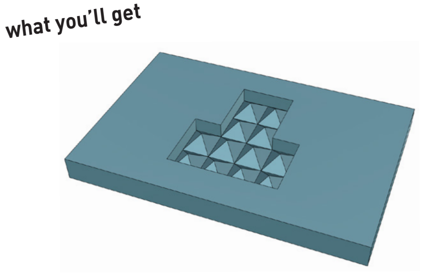
what you'll get



a smaller rectangle reduces the total amount of material you are taking away by cutting only within that region. this can work for some shapes, but for more eccentric figures you will still end up taking out way too much material.

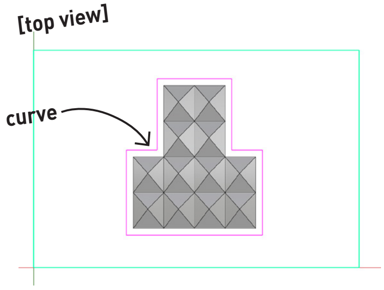
2. the perimeter of your part

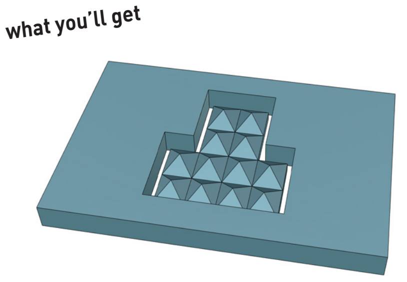




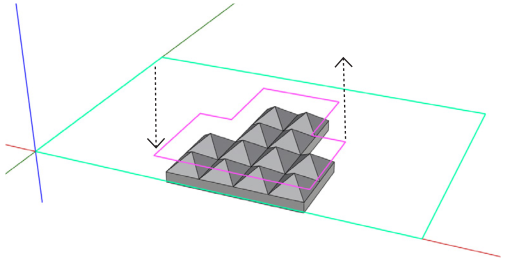
using a tight perimeter of your model as a boundary curve is the most efficient way to take out the minimal amount of material. a quick way to extract this curve is to use the [**MeshOutline**] command while in **top** view. what this method will not do however is cut out your part at all. if you need the part wholly cut from your stock, you'll need to use another cut or a different method. ...**profiling operations**, as you learned, can achieve this. also, you can try:

3. an offset perimeter



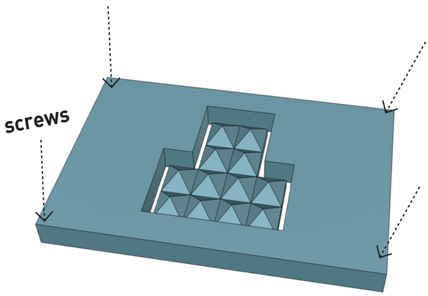


if you need to remove all of the material **around the edge** of your part, this method is a compromise that allows you to achieve this without roughing out the entire rectangle showed in the previous method. this is an efficient technique that gives you a lot of options for securing the part in place.



note: these boundary curves can be **anywhere** in Z, the only thing that matters is how RhinoCAM views them from the top. you can even use three-dimensional curves as a boundary, provided they are closed and convex. RhinoCAM uses a **projection** of the curve to determine the boundary.

STEP FOUR :
MAKE SURE YOUR PARTS STAY PUT



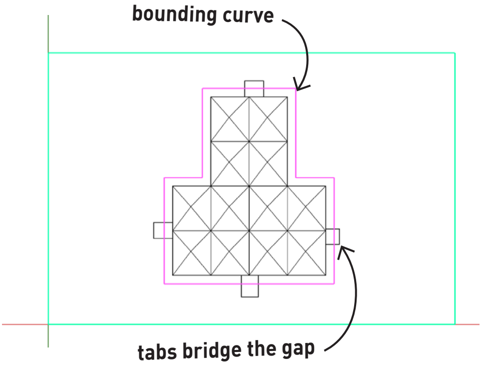
if you take this version of the cut, you'll see that your part in the center is not bound in any way to the part of the stock that is screwed to the table. it is entirely loose!

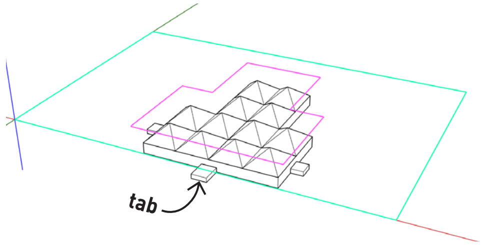
in these cases where you are cutting entirely around your part, you will need to build **tabs**, or **connections** back to the larger stock.

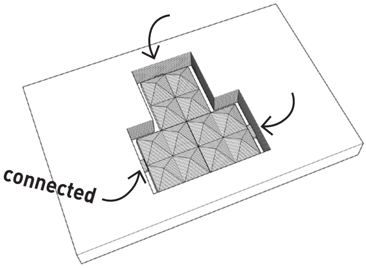
unlike the automatic tabs option with a profiling operation, tabs for 3D operations need to be **modeled manually** in Rhino.

there is no hard and fast rule for how many or how big the tabs need to be... they need only be big and frequent enough to hold everything together.

one thing they **must** do is **overlap** with both your part and the bounding region for your cutting operation.







keeping your material **stable and connected** while cutting is an art. remember you need to keep a **solid** perimeter of stock behind, keep all potentially loose parts **tied** back to the stock, and strategically **screw down** any interior parts that may become loose.

STEP FIVE :
CREATE YOUR CUTTING OPERATION

[Control Geometry] → (select your boundary curves)

[Tool] → (create and set the right tool)

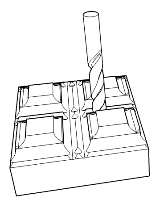
[Cut Parameters] → (set cut details and stepover)

[Cut Levels] → (set stepdown where applicable)

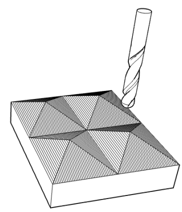
[Clearance Plane] → (make sure it's the right height)

there are many ways to perform three-dimensional cuts, shaping, and surfacing on the CNC mill. **MOST** of these techniques will use both a **roughing** and a **finising** pass.

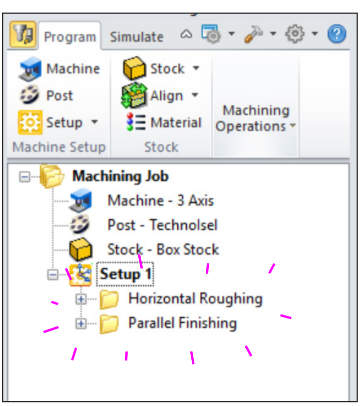
roughing will cut away material quickly but is not gentle or precise enough to provide a good finish. finishing will more carefully and accurately trim your part to shape.



horizontal roughing



parallel finishing



in this walkthrough, we will look at the most common and universal pair of 3D operations:

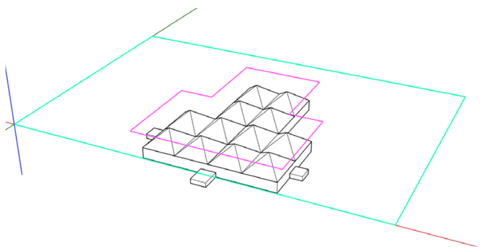
a **Horizontal Roughing**

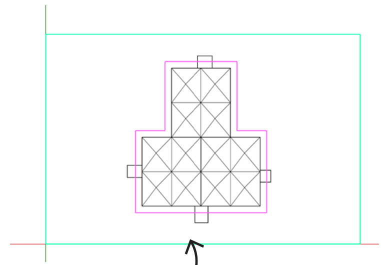
and a **Parallel Finish**

these two combined will be able to handle a wide variety of projects. but remember that there are many more types of passes that are worth exploring once you master these ones.

...when your material is chosen and measured, your digital model scaled and located correctly, and your stock set in RhinoCAM, you are **ready** to build your toolpaths →

horizontal roughing





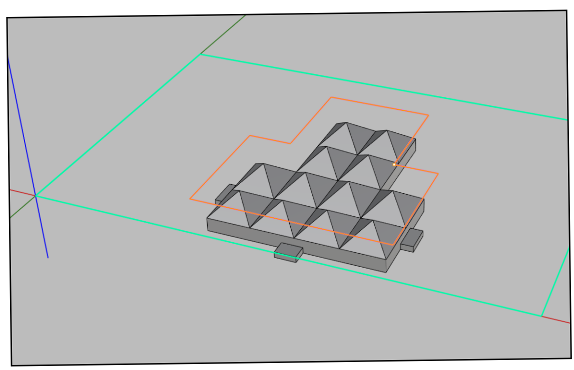
as you begin, your model in Rhino should look something like this. you need your model **located** correctly in Z, a **boundary** curve around your part to contain the area of your cut, and--if you're cutting the part out entirely--**tabs** to hold things in place.

[Control Geometry]

select the boundary curve that you made around your part.

if you are cutting around the part, make sure that this curve is **wide enough** around your part to allow the size of bit you choose.

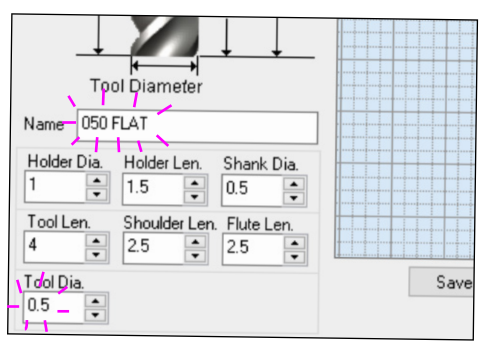
e.g. if you are using a 1/2" bit, your curve should be at least 0.6" offset from your part.



[Tool]

the larger the tool, the faster the roughing operation. but you still need to pick a tool that can fit into the spaces you want to cut.

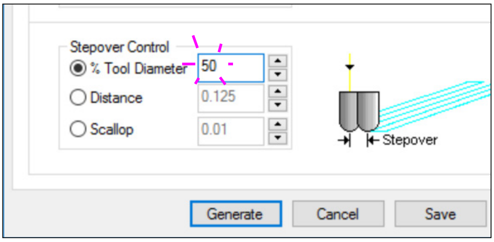
often, you will want to use a **1/2" flat bit** for roughing.



[Cut Parameters]

the primary parameter you need to set in this window is the **Stepover Control.**

the value is usually set as a percentage of your tool diameter. as mentioned, **50%** is a safe place to start for most materials.

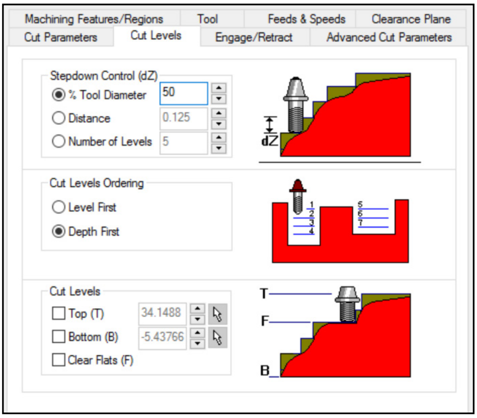


[Cut Levels]

similarly, you need to specify a value in **Stepdown Control.**

stepdown is one of the most important controls in how your material is cut. again, **50%** is a safe place to start.

horizontal passes work by clearing all material in one Z level, and stepping down once between levels.



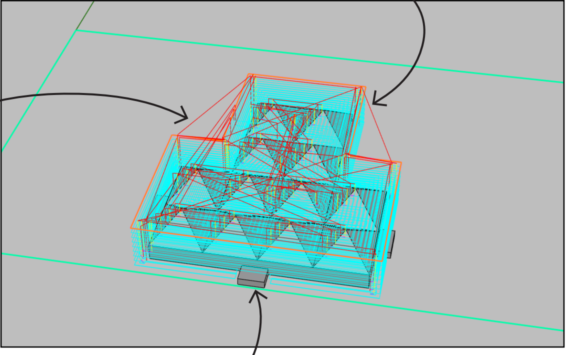
click **generate**

you should see a preview of the roughing pass, contained to the region of your boundary curve.

make sure that your **red lines** (clearance paths) are just slightly above the model.

if those paths are **too high**, you will need to check your **stock** and your **clearance plane**, and delete geometry that is **above Z=0**

steps down gradually

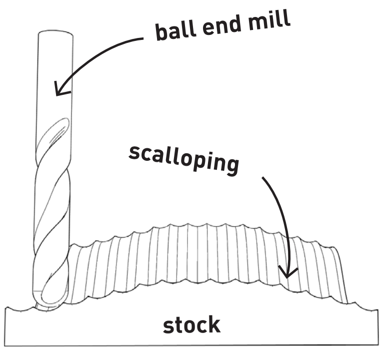


path avoids tabs

parallel finishing

a parallel finish will track your bit **back and forth** across the surface of your part, approaching a more accurate resolution of your digital model in physical form.

it's important that you performed a roughing operation **first**, since this pass will plunge your bit all the way into your material to hit that model surface. if you haven't removed the vast majority of that above material first, you will simply **break** the bit.

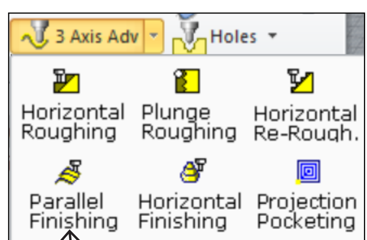


a **ball end mill** is the best choice for shaping three-dimensional surface, since it won't leave sharp contour edges between paths like a flat mill does.

it will, like any tool, leave a texture on your surfaces. this texture, **scalloping**, is more pronounced with a larger **stepover**. larger ball end bits will create a **smoother** polished surface, and smaller bits will leave a higher level of **detail**.

there a **tradeoff** between the **detail** of your final surface and the **time** it takes to cut.

a **1/4" ball end mill** will get great detail, but is impractical for finishing projects that are larger than about 3 feet square (36" by 36"). for this size, consider a **larger** bit.



set it up...

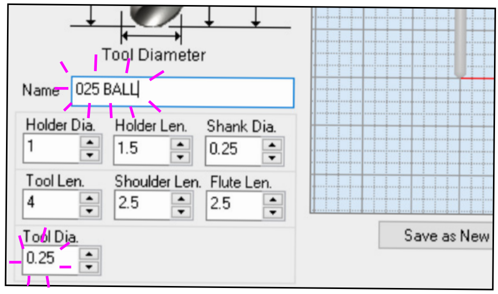
[Control Geometry]

select the same **boundary curve** that you used for your roughing pass. in certain cases, you may want to constrain the parallel finish to a smaller boundary area.

[Tool]

as discussed, the size of your bit determines everything about the quality and time of the pass.

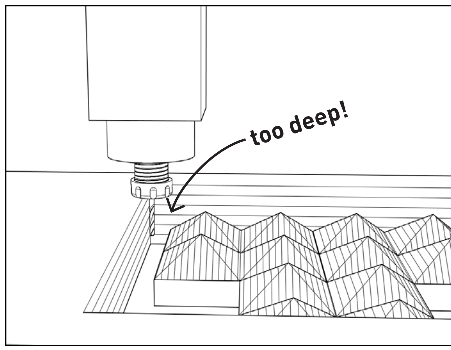
this example uses a relatively small model, and in this case it works to use a **1/4" ball end mill.**

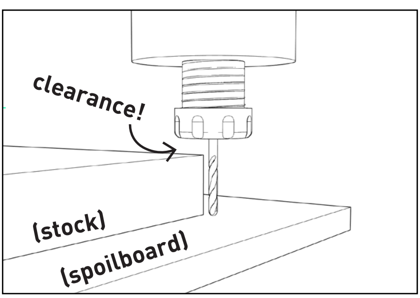


note: it's **important** (!!) to consider how deep you are trying to cut with the type of bit you have.

it's not just the bit that needs to fit into your stock, but the entire collet and spindle arm too.

you need to make sure that your cuts aren't going to be deeper than the length of the bit you are using.





1/4" bits come in **all lengths**. if your bit is not long enough to plunge into your vertical holes, you'll collide the spindle with your material. this can range from being annoying (foam) to very destructive (hard wood)

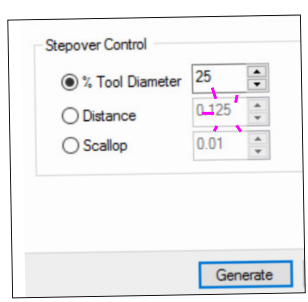
you can **double-check** this when setting up the machine, but you need to have it in mind making the toolpath.

[Cut Parameters]

a parallel pass only depends on **stepover**.

25% is a good start for an efficient finish. lower than that, to **10%**, will produce a finer surface but will take over the time.

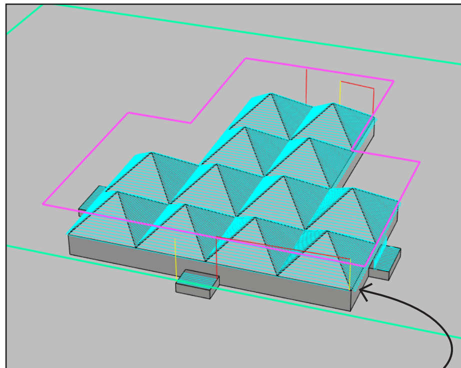
any tighter than 10% will get very slow.



click **generate**

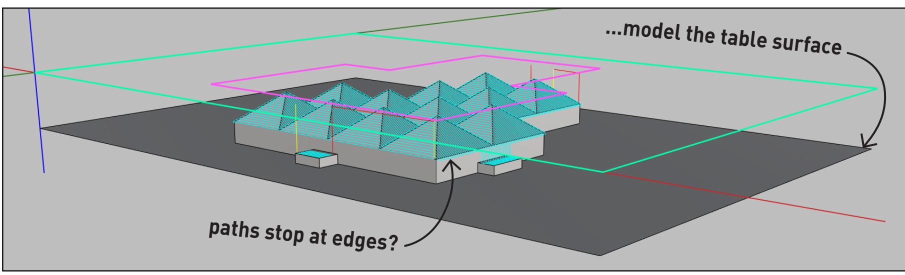
your preview may look something like this one, or may have vertical red lines at the edge of every single parallel pass. what is happening is that RhinoCAM is confused as to how deep it should be cutting.

if your goal is to cut the part out entirely, you will need to give your model something to tell RhinoCAM where the "table" is below.

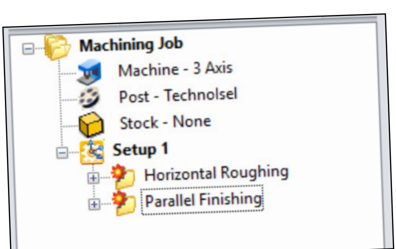


...doesn't cut through?

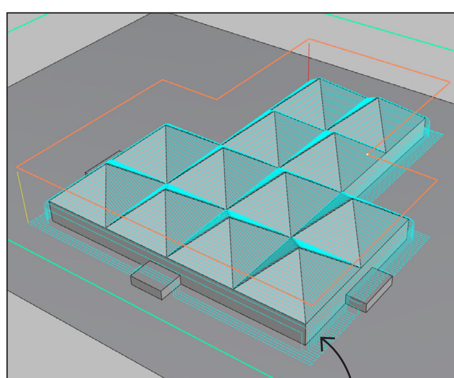
a quick way to fix this, and a handy thing in general for your model, is to model a **surface at the bottom** of your model, where the spoilboard table actually sits.



note: when you change the geometry in your Rhino model after you've built toolpath operations, a little warning flag will appear next to the title of the operation...



...so you'll need to **regenerate**



cuts entirely around part

NEXT...

1. **Go to the shop** and get a monitor to check your files

2. Show your **CNC sticker** as evidence of certification.

3. **Post** your files, instructions in intro tutorial.

4. **Book** time on the CNC with help of a monitor.

5. **Show up** on time to your appointment, and cut away!

6. **Expand** your CNC skills with subsequent tutorials.

note: the Fab Lab monitors are there to assist you with making sure your files are ready. that being said, if your files aren't ready to the point they would be with the help of the information contained in the first two tutorials, you will be directed **back** to the tutorial until your files are better prepared.

* this is a first-draft publication. for questions, corrections, or other issues, please contact Josh at **jcj2134@columbia.edu**

happy making!