# CNC INTRO WALKTHROUGH
GSAPP FABRICATION LAB 2016

this is a student guide to the procedure of gaining access to the CNC router digital fabrication equipment in the Fabrication Lab at GSAPP.   The guide contains instructions on understanding the basic setup of a file and its relation to the operation of a CNC router [AKA: *the mill*].

...upon completion of this walkthrough, one will have gone through essential concepts and features of the CNC equipment, drawn a digital model in Rhino, and prepared the file for CNC tooling via the RhinoCAM plugin.

...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.
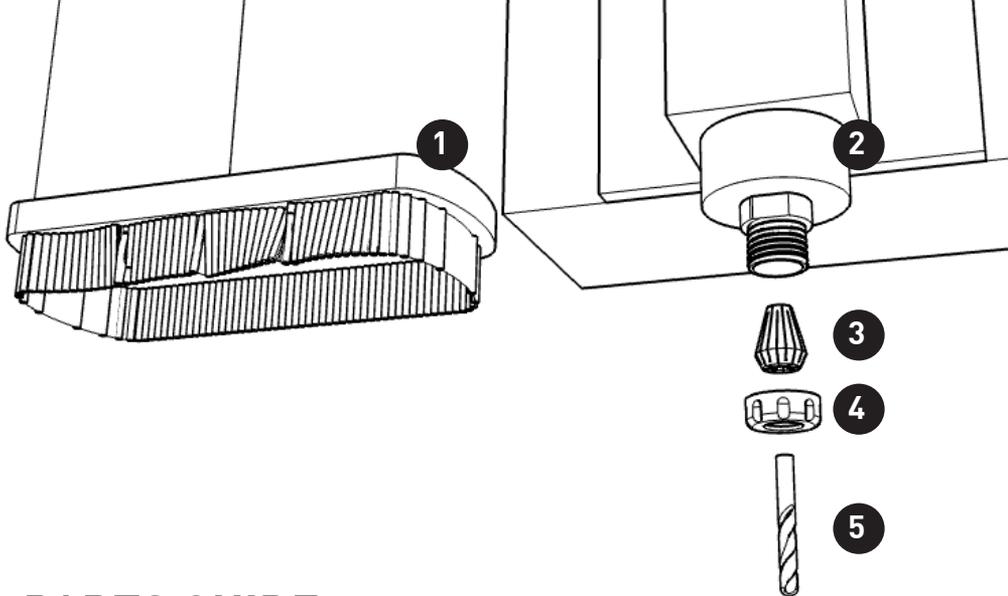
PARTS GUIDE

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
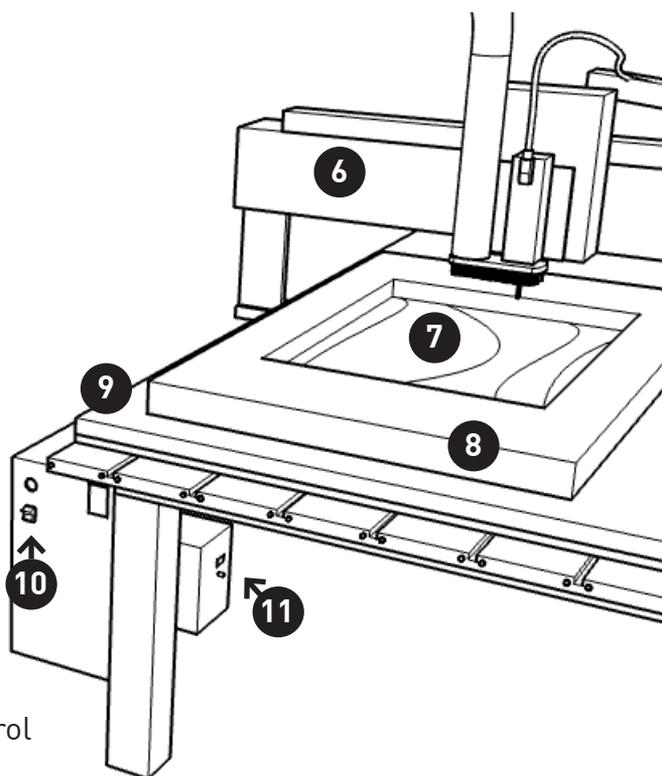
STEP ONE :
UNDERSTAND WHAT YOU WANT TO CUT

like the laser cutter, the mill allows you to cut incredibly complex models with a high precision. **unlike** the laser cutter, the mill asks you much more about what you want to cut and how you want to cut it.

the **two** things you need to have in mind before you prepare a model to be fabricated on the mill are the **type of tool** you are going to use, and the **type of operation** you're going to ask the tool to perform. the two factors give you a broad palette of techniques on the mill.

**most of the time**, you'll use one of these:

flat end mill

ball end mill

vee (V) mill
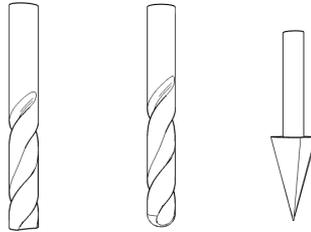
the different bits will produce different cuts, which you will use for different situations:
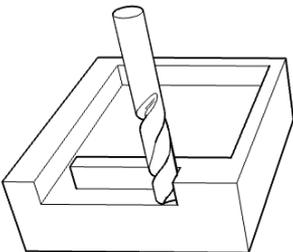
flat cuts
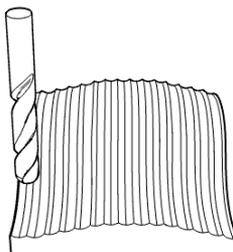
ball cuts
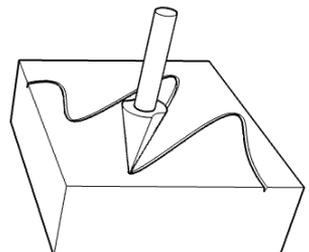
vee cuts

(COMMON OPERATIONS)
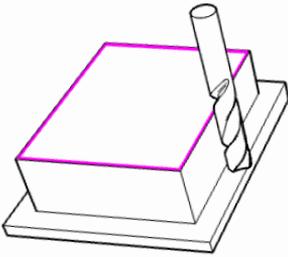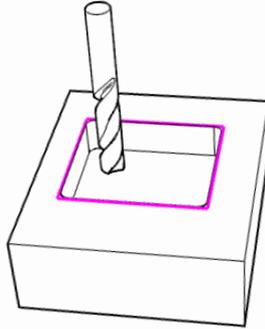
profiling   pocketing   engraving
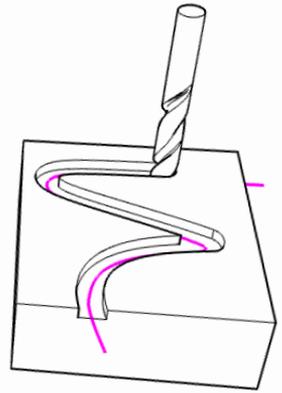
these operations are considered **2D**. this is how to "cut out" pieces or how to carve simpler grooves, notches, and pockets.
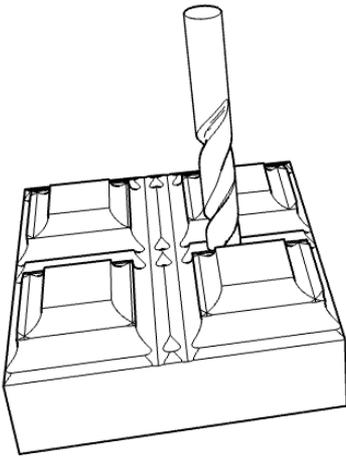
horizontal roughing   parallel finishing
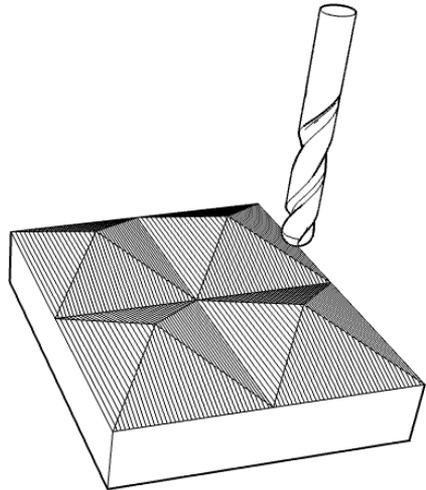
3D operations almost always come in pairs: a **roughing** pass to cut away large amounts of material in a faster, more imprecise way... and a **finishing** pass, to closely approximate your digital model shape.   a parallel finish is the most common and universal, but you can determine almost any kind of finish pattern.

(LIMITATIONS!!)

1: max depth

all CNC routers have a **working envelope**, or a maximum size object you can cut. the biggest limitation is usually the z-axis. on the Techno routers, that maximum is about **3 inches**.

don't make your model taller than that. you can slice your model into multiple parts to achieve greater depth.

> 3"
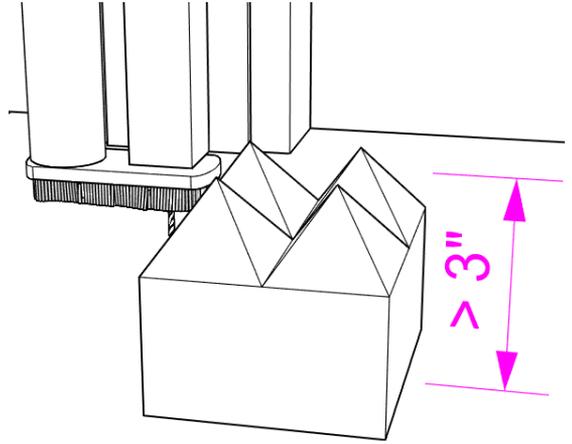
2: end mill shape
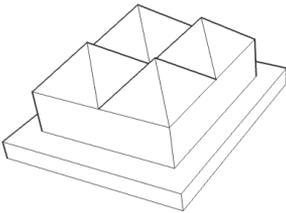
inside corners?

digital...
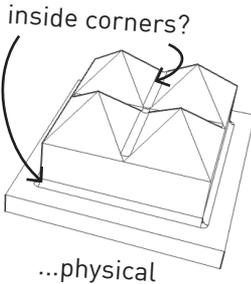
...physical

...remember that you can only cut into spaces **wider** than the bit you decide to use.

as a result, **inside corners** on your model may be left with a small radius of material

3: no undercuts!

if your model has overhanging parts...
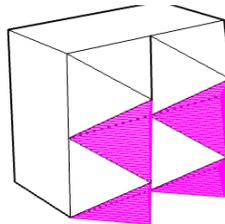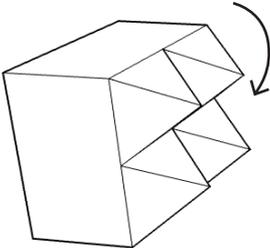
the machine won't reach those undercuts...

...you can still send the file, but you'll get this
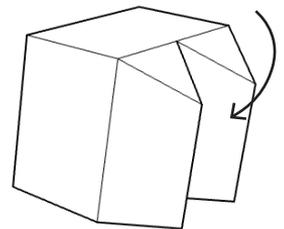
STEP TWO :
UNDERSTAND WHERE YOU ARE
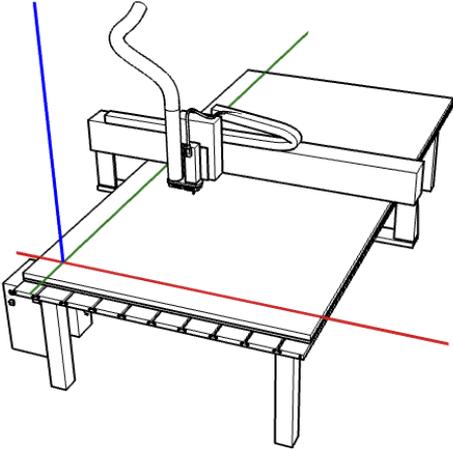
the CNC mill is a physical representation of the virtual space of your model in Rhino. some conventions always hold:

- the x-axis in Rhino runs along the short edge of the mill

- the y-axis in Rhino runs along the long edge of the mill

- the z-axis in Rhino is "vertical" in the real world.

- x increases to the "**right**" of the mill, y increases toward the "**back**", and z increases "**up**"

accordingly, it makes sense to locate your model in Rhino in the quadrant where X and Y are both positive. in 'top view,' this is the upper-right quadrant.

(top view)

(model here!)
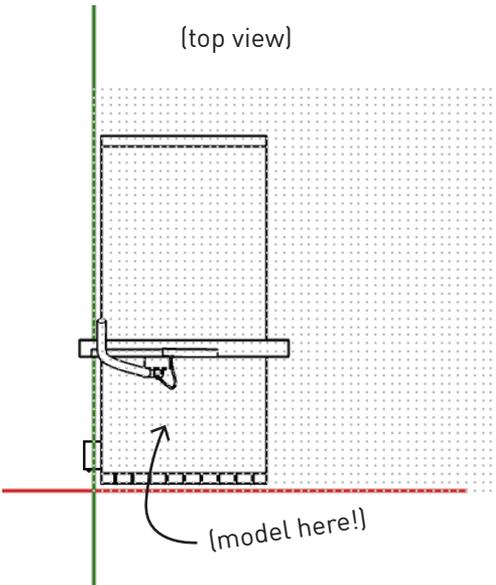
the CNC depends on you to manually indicate where your model "starts," or where to set as a reference point the origin in your Rhino model.

YOU SET THE ZEROES MANUALLY ON THE MACHINE, LATER!

where you set the zeroes depends on how you want to cut something. often, it works to set the X and Y zeroes at the physical **corner** of the CNC table. but you can just as well set those zeroes anywhere you like.
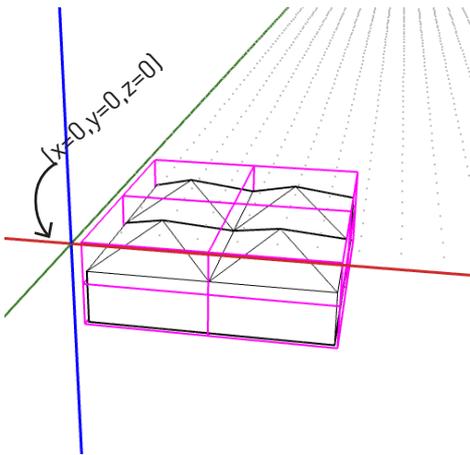
in **most** cases, you'll be starting the mill at the top of your material and cutting **down** into it.

this means you'll **zero the z-axis** of the CNC mill to the TOP of your material.

in this case, z=ZERO in Rhino needs to be at the **top** of your digital model.

in Rhino, this means moving your model **below the origin** x-y plane.

as mentioned, the location of your model in Rhino relative to the origin (0,0,0) determines where the part will be cut on the mill, after you have **manually** set where x=0 and where y=0 on the machine.

if you locate it **here** in Rhino...

(origin)
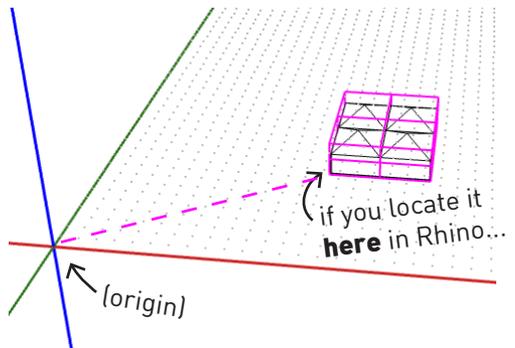
...and if you zero X and zero Y **here**

...remember this relationship! it's important because it will determine where you need to fasten your material to the bed of the table.

...then your part will be cut **here**
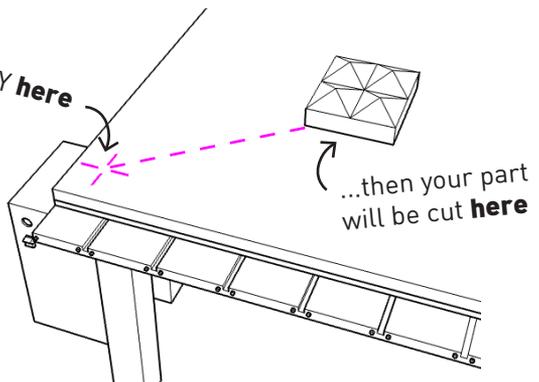
To prepare your digital model for cutting on the mill, you'll need to do the following:

1. Pick your piece of blank material (**stock**).

2. Measure your material!   Length, width, and THICKNESS all matter.
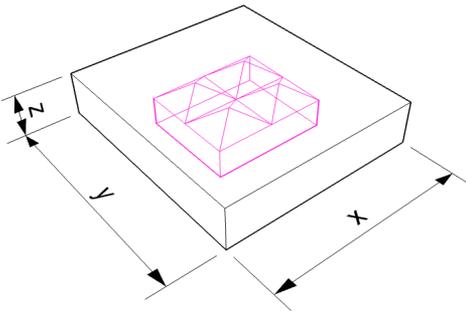
pick a piece of material **larger** than your model! remember that you need extra space around the edge to fasten your material to the table.

for this exercise, you'll be using plywood that's 0.75" thick.

3. Locate your model in Rhino within where you know it will fit in your **stock**.
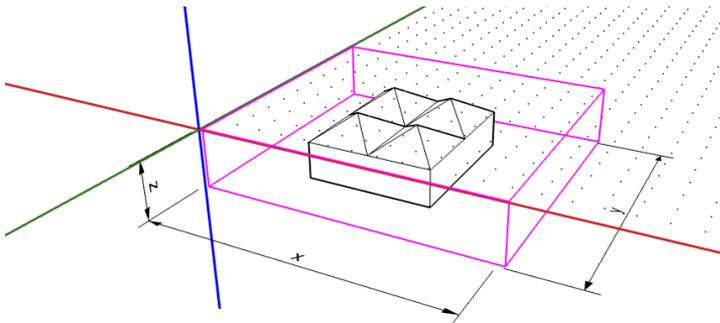
4. Decide what kind of cuts you need to make.   For this exercise, you're going to do a **drill holes operation**, and a **profiling operation**.

5. Decide which kind of tool you need to use. For this exercise, you will be using a **1/8" drill bit** and a **1/4" flat end mill**

...ready?

TUTORIAL EXERCISE

1. Ask for a piece of plywood from the shop (ask a monitor) that's **15" x 12"**. **Draw this rectangle** in Rhino

2. **Draw a figure** in Rhino smaller than the size of your plywood rectangle: Make an eccentric shape that's a closed curve, with one or more closed curves entirely within the shape.

Also, draw (on another layer if you wish) some **guides** (below, pink) that will help you make sure to leave enough space to fasten your plywood to the table.

your plywood size

1" offset from edge

the shape to cut

holes within your shape

0.5" offsets within holes
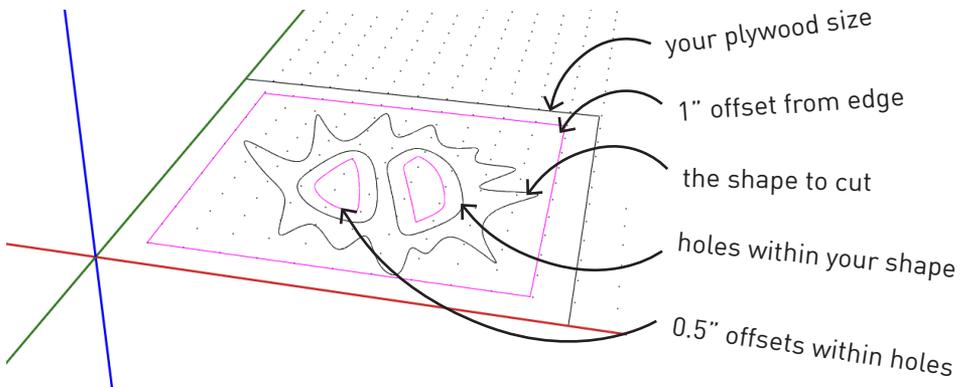
...the idea is to produce something like **this**

you leave the 1" offset from the edge to make sure you have space to screw your material down without risking collision of the bit with screws.

the 0.5" offsets in the holes ensure your interior cuts won't interfere with the screws you put in those pieces.
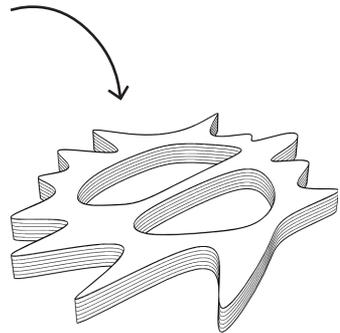
3. RhinoCAM is the machining plugin that runs as a series of windows in Rhino. You can have access to RhinoCAM on the computers in the **Fab Lab** and on the computers in **202 Fayerweather.**

Here both RhinoCAM 2014 and 2016 versions are shown. You'll use **2016**. Note there may be small differences.

make sure your RhinoCAM window is switched to **MILL** with the upper-left button.
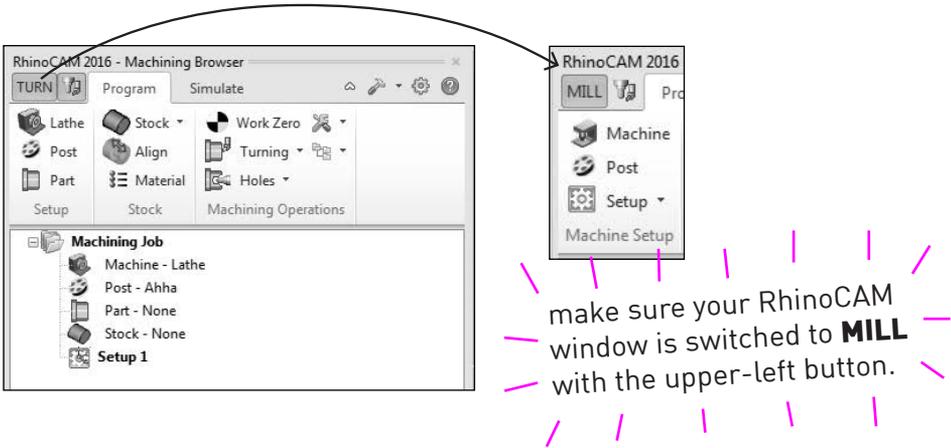
4. Set up your first operation! **DRILL HOLES**

make circles or points for drilling
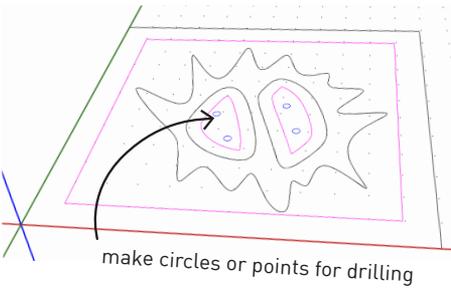
fastening down your material can be tricky when you are going to have a lot of complex parts.

having the CNC mill drill holes gives you the exact location of good points to fasten screws.

...**screwing interior parts down** like this ensures that pieces won't hop out and interfere with the cut mid-way. this technique can be extended to larger, complex jobs. the takeaway is: you use the CNC to set up and help your work, as well as cut your model.

5. Select that type of operation, and a new window will pop up...
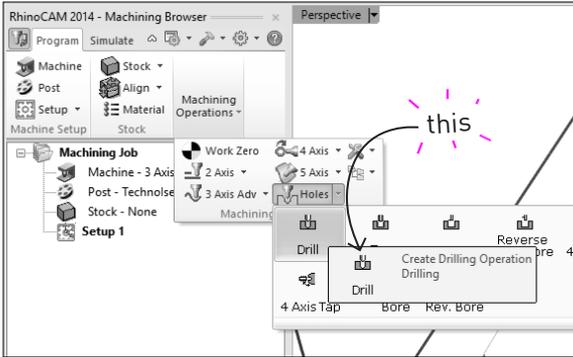
...where you will configure all the settings for the cut.
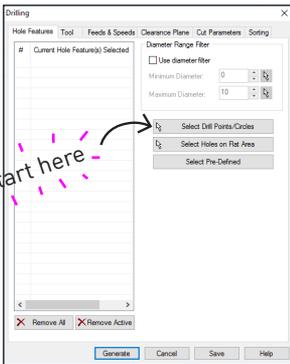
every operation in RhinoCAM has its own window of setting, but some things are common to all of them.

1. selecting your **geometry** to cut

2. telling RhinoCAM which **tool** you are using

3. specifying details for the cut like **total depth**, **step-down** distances, and **step-over** distances.

...for the drill operation, the only one of these that matters is **total depth**.

start here

...the hole drilling operation gives you the option to select points or circles for your drill locations.   select the points/circles you made inside your cutouts.

once you pick your points, they will become highlighted in your Rhino window.

note: RhinoCAM has its own layers of previews that can be switched on and off.
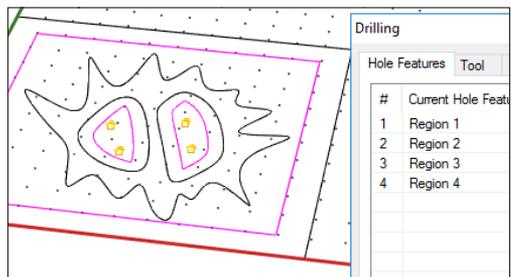
click the next tab, [**Tool]**, then the button **Edit/Create/Select Tool...**

shape...

name it...
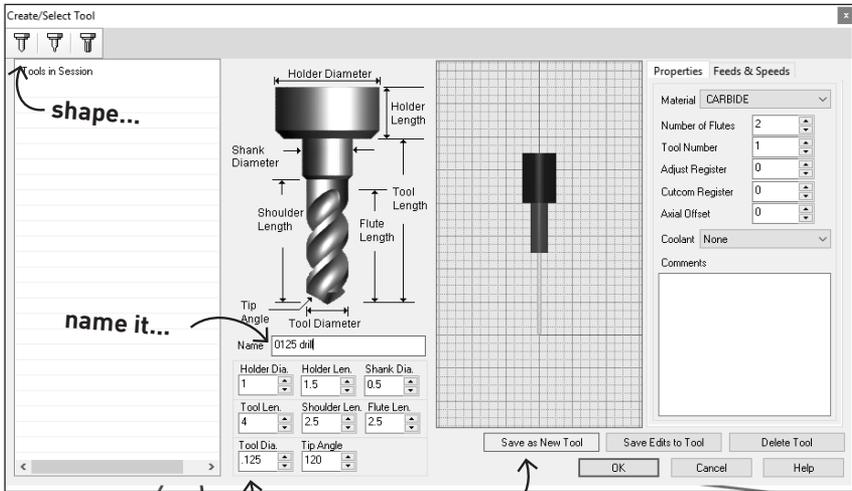
enter diameter...

...and save it.

...the **only** setting here that matters is the tool diameter. there are reasons to model the whole tool, but none that this lesson will address.

once you create a tool, you'll be able to use it in other operations by just picking it from your list instead of creating it all over again.

...next, set your drilling depth in the [**Cut Parameters**] tab.

here!
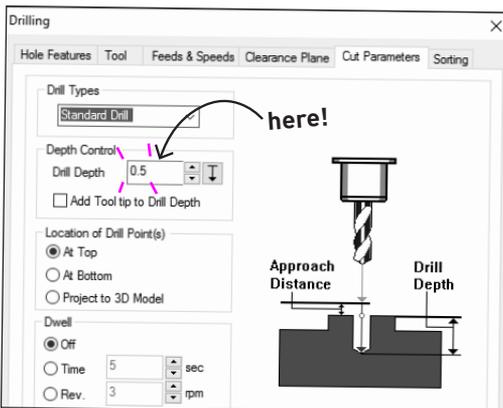
you want to drill to the depth of your material.

remember measuring the thickness? (you can use a caliper in the shop)...

the drill depth is a simply a number. the tip of the drill will plunge to this depth.

6. generate your toolpath by clicking "**generate**"
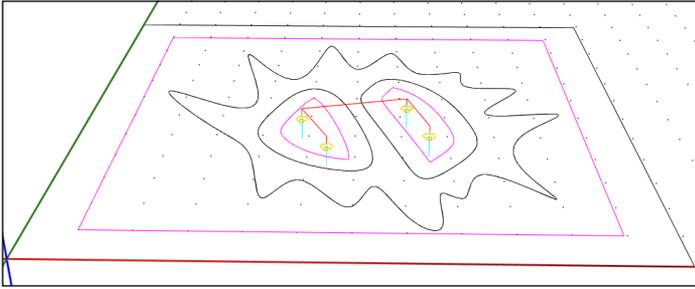
...what you generated is previewed as a set of **toolpath** preview lines. the **red** lines are the tool jogging between cuts in the material, and the **light blue** (sometimes dark blue) lines are the actual cuts within your material.

make sure that your **red lines are 1/4" above** the surface of the drawing, and orbit around your model to check that your holes are drilling at the depth that you actually want. **do this every time**!

note: if you don't see a preview of your toolpaths at this point, you can turn them on **here**
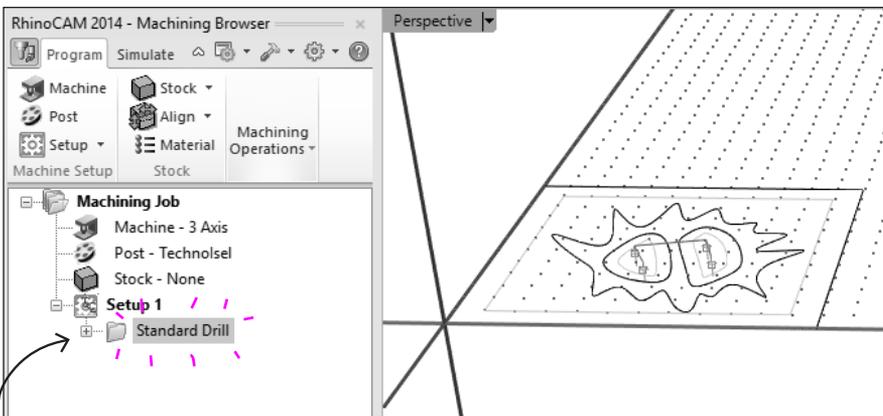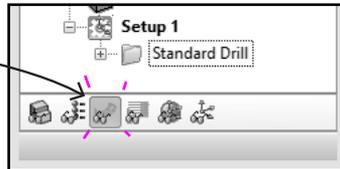
your operation now shows up as an item in the RhinoCAM window. you can go back and edit its settings if you need to make changes.

next... you need to create a **profile** path to cut out your parts.

7. Set up your profile path.   A **profile** is a path that takes a line, curve, or surface edge and cuts along it with your chosen bit.

It's great for cutting holes in things, or cutting parts out entirely.
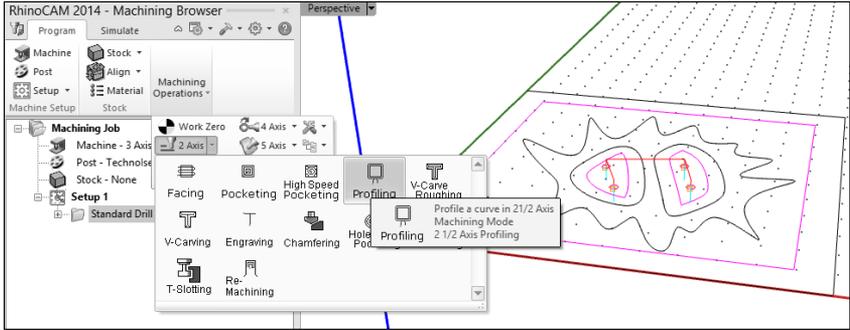
Since this operation is a bit more complex than drilling, there are more settings you need to ensure are dialed in:

[Control Geometry]

click to pick

inner   curves
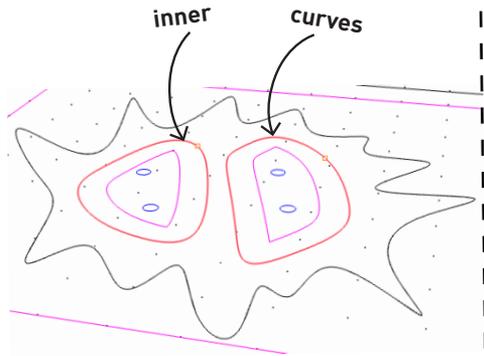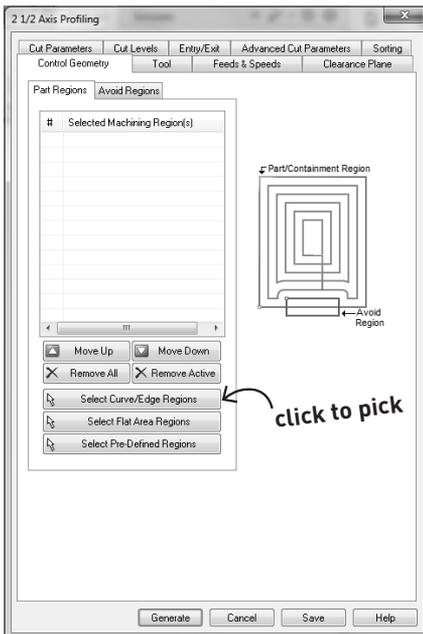
First, pick the curves that are the interior parts you want to cut out.

You might have more than two.

Generally, RhinoCAM calls these lines and curves that guide your cut **control geometry.**

[Tool] Click [Edit/Create/Select Tool...] again and you'll see a slightly more complicated tool creation window.

pick tool shape...

...name it

...diameter?

save it
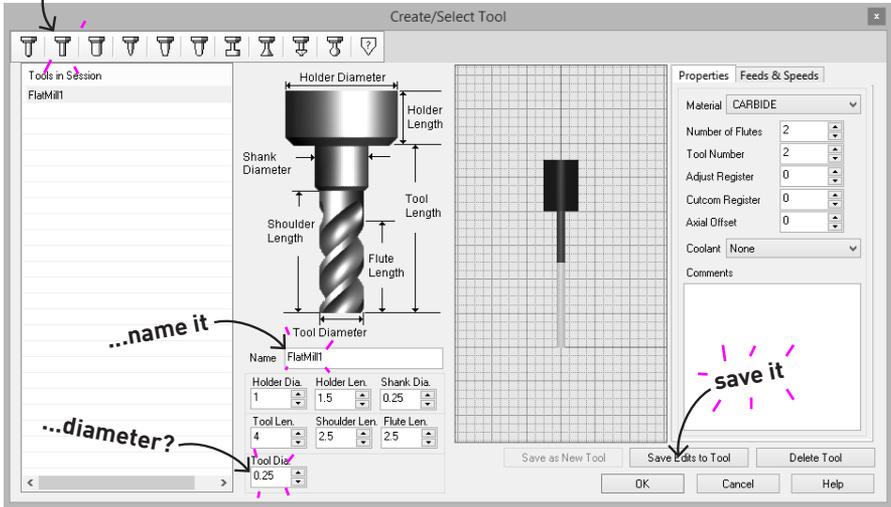
The only settings that urgently matter are that you pick the correct tool shape, and the tool diameter. For this exercise, you're using a **FLAT**, **1/4"** bit.

Name the tool something to indicate what it is, like "**025 FLAT**"

[Cut Parameters]

pick a side
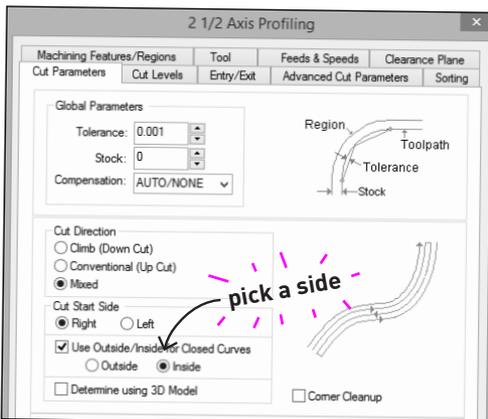
since the profile operation makes cuts that run the **side** of your bit along your curve for a clean cut, you need to **specify** which side of the curve you want to cut.

...in the case of closed curves, like your part, you can pick to cut on the **outside or inside** of the curve.

...and since these are parts to cut from the interior of your final piece, you want to pick "**inside**" here.

[Cut Levels]

this is the setting that requires a little bit of work, but is key to understanding how the mill works, and what to tell it to do

caliper
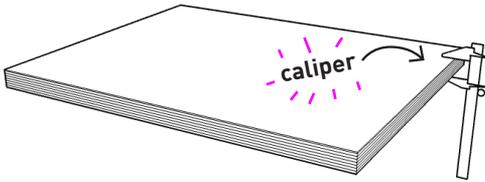
It's also where it's important to understand **exactly** how thick your material is.

Your plywood will be **0.72"**

...when you cut all the way through something, you will want to tell the cut to go *slightly* deeper than the thickness of your material. adding **0.01"** is effective. so, **0.73"** deep.

the *bit cannot cut* this deep at once. in most cases, you will need to use multiple descending passes to cut a profile operation. the [**Cut Levels**] tab also allows you to determine that "**stepdown**."

big rule of thumb   ← helpful!

this **stepdown**, or "**plunge**" is a setting that you determine based on your material, and the diameter and type of bit you want to use.

a good place to start to be safe at **50% of the bit's diameter**

...so, your **1/4"** bit should only be dropping **1/8"** at a time.

... a 3/8" bit should be set to plunge 3/16" at a time

... a 1/2" bit should be set to step down 1/4" at a time... **and so on**

fun fact: in many operations, you will be asked for a "**stepover**" as well as a stepdown. same idea: a safe starting point is **50%** of the bit.

**2 1/2 Axis Profiling**

Machining Features/Regions | Tool | Feeds & Speeds | Clearance Plane
Cut Parameters | Cut Levels | Entry/Exit | Advanced Cut Parameters | Sorting

Location of Cut Geometry
- At Top
- At Bottom
- Pick Top 0

total depth →

Cut Depth Control
Total Cut Depth: **0.73**
Rough Depth: **0.73**   Finish Depth: 0

stepdown →

Rough Depth/Cut: **0.125**   Finish Depth/Cut: 0

Use 3D Model to Detect Depth
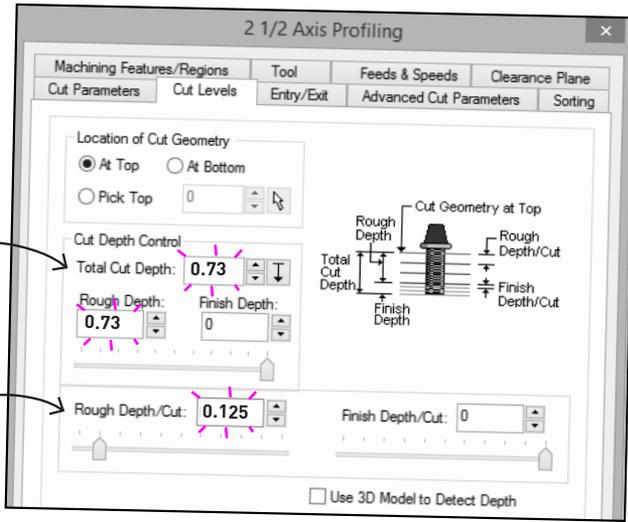
set your **total depth** and amount to **step down** in this window.

...we measured the plywood at about 0.72, so the depth is **0.73**
...you know the bit is 1/4″, so that stepdown is entered as **0.125**

[Entry/Exit]

entry and exit paths are little curls that your bit can perform on the way in and out of a cut. it's used to preserve finer finishes and to be delicate cutting into harder materials.

but in many of your cases such paths will be unnecessary and even potentially damaging.

turn both of these to **None**

2 1/2 Axis Profiling

Machining Features/Regions | Tool | Feeds & Speeds | Clearance Plane
Cut Parameters | Cut Levels | Entry/Exit | Advanced Cut Parameters | Sorting

Entry Motions
- None
- 2D Entry
- Along Path 3D Entry

2D Entry

Approach Motion
- Normal Tangent Specify
Length (L) 0.25
Angle (A) 0

Engage Motion
- Linear
- Radial
Length (L) 0.125
Angle (A) 20
Radius (R) 0.25

Entry point on Path

Along Path 3D Entry
Along Path Angle 10   Along Path Height 0.05

Exit Motions
- None
- 2D Exit

2D Exit

Retract Motion
- Linear
- Radial
Length (L) 0.25
Angle (A) 20
Radius (R) 0.25

Departure Motion
- Normal Tangent Specify
Length (L) 0.25
Angle (A) 0

Exit point on Path

Apply entry/exit at each cut level

Generate | Cancel | Save | Help

8. Generate and Check

Click "**generate**" and orbit around to check your toolpath previews.
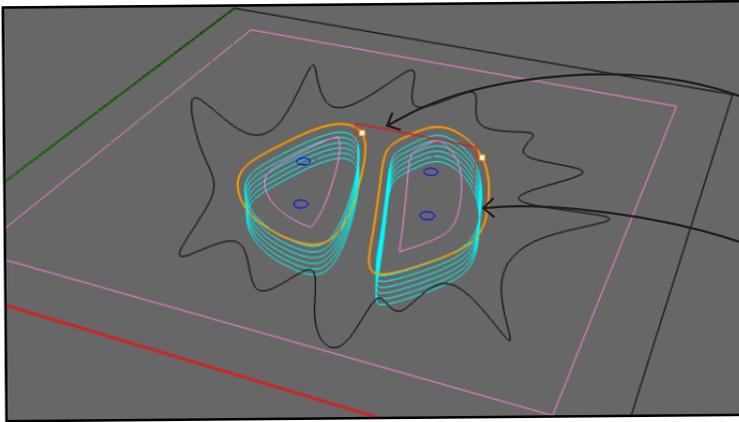
looks ok...

red lines at **clearance plane** are .25" above drawing

cuts are **inside** your interior curves.

from side...

depth looks ok
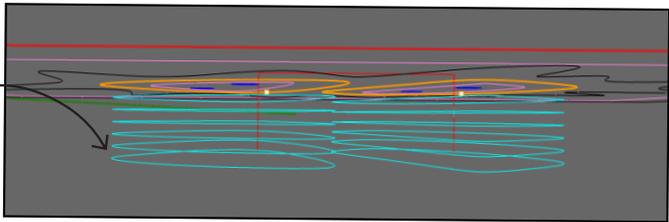
9. Make another profile operation for the outside cuts of your part.

[Control Geometry] → (select your exterior curve)

[Tool] → (use the same 1/4" you created)

**[Cut Parameters]** → (this time, select **outside**)

[Cut Levels] → (same)

[Entry/Exit] → (same, None)

hint: save time by **copy/pasting** your previous **Profile** operation

...most of the settings (like total depth/stepdown) will carry over.

(new) [Advanced Cut Parameters]
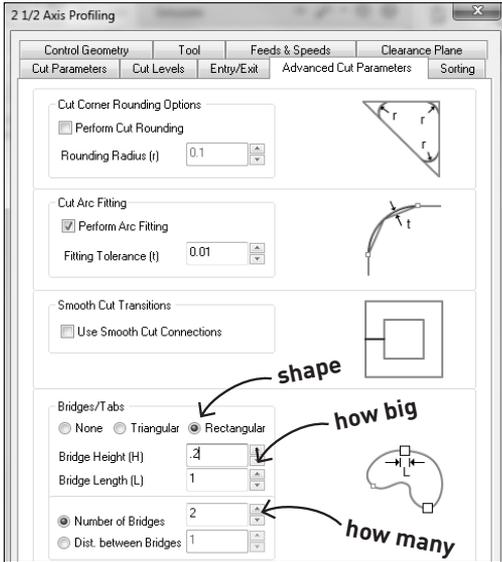
the final thing you need to be concerned about when cutting parts out with a profile operation is the **issue of how your pieces are going to be held in place on the table.**

you gave yourself an inch border to screw down the edges of your plywood. the screws in the center parts will hold those pieces down

but there's still the problem of your actual part being held down. there are many ways to do this, but the easiest is to ask RhinoCAM to generate **tabs** that connect your part back to the larger stock.

preview: when the cuts are finished,
your project should look something like this:

- interior screws
- first profile pass
- second profile pass
- tab
- tab
- exterior screws
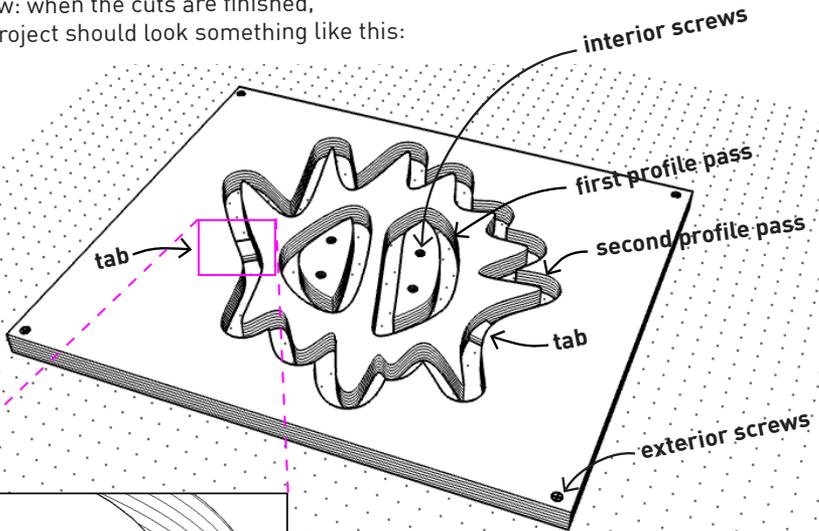
tab

* make your tabs at least **0.5"** long and **0.125"** high

finding ways to hold your material down while performing intricate cuts is a challenge.

combinations of CNC-drilled screws, tabs, and clever organization of stock and geometry will allow you to achieve very complex cuts

10. **Generate** again and check…
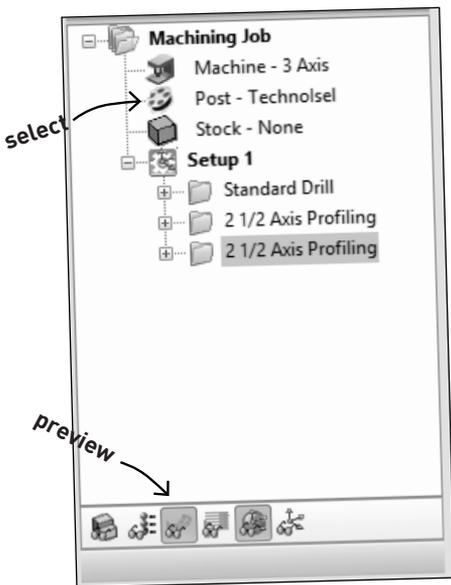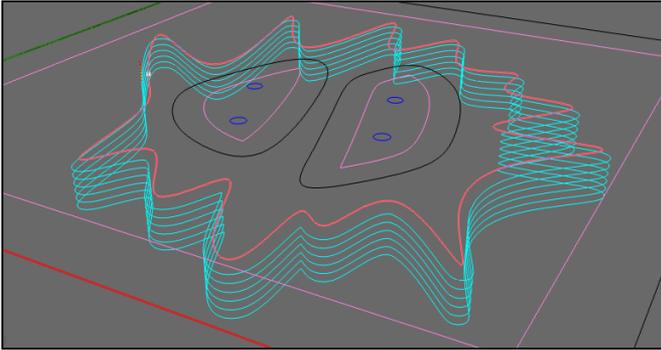
you should now have **three** operations saved in the RhinoCAM job window.

at this point you are **ready** to take this file to the Fabrication Lab to complete the in-person segment of the tutorial.

check the Fabrication Lab web page for the best hours you can walk in to the shop and get this hands-on step done.

before you start, you will "**post**" your file. posting, or post-processing, is a process that takes your work in RhinoCAM and interprets it into code files which the machine can read and perform. every CNC machine has a different post-processor. For the large Techno CNC, use the **Technolsel** post. For the smaller mills, use **Techno HD**
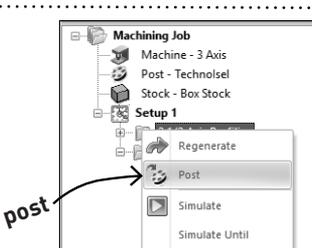
…posting generates an ***.nc** file that pops up in a notepad window as a text file.

when you save these post files, it's important to give yourself a **naming convention** that you'll understand later.

01-0125_Drill.nc   ← **OK**
02-025_Profile_Inner.nc
03-025_Profile_Outer.nc

asdfghjkl;.nc   ← **FAIL**
thing.nc
final_final_really_final.nc

NEXT...

1. **Go to the shop** and get monitor assistance to get on the mill.

   ...the monitors will help you:

   + post your files

   + load them into the CNC control software

   + set your feed rate and spindle speed (very **critical**)

   + equip the machine with your chosen bits

   + fasten your stock to the table

   + move and zero the bit to your project origin point

2. Get a **CNC sticker** upon completing your plywood part.

3. **Learn** 3D passes and machine setup in **next** PDF tutorials

4. **Bring** your prepared project files to the Fab Lab

5. After file approval, **book** your appointment on the CNC mills.

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

happy making!