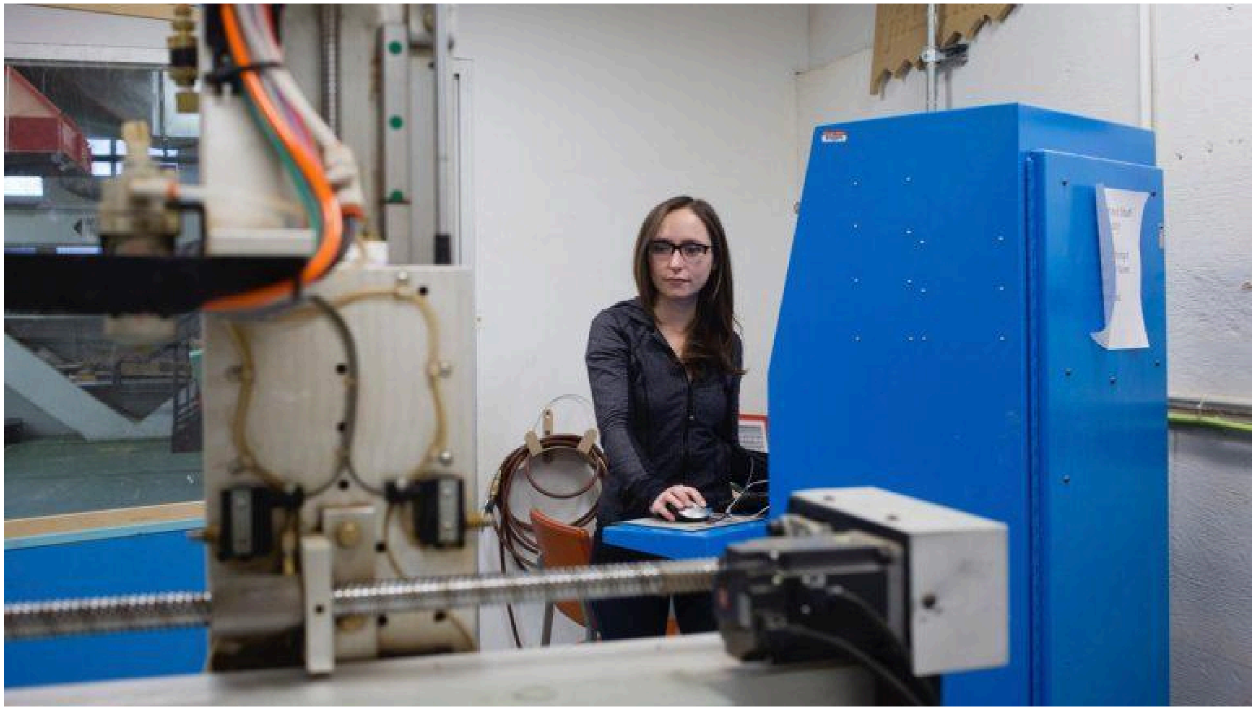
Carnegie Mellon University

School of Art



CNC Handbook

Table of Contents

1.0 Basic Operation

Fig. 1: CNC router Tool Library : T1-T11 tool cones with bits

1.1 Safety and PPE

Everyone in the CNC router room, whether they are operating equipment or simply observing, must wear the following personal protective equipment (PPE):

1. Safety glasses
2. Hearing protection
3. Dust mask especially when cutting MDF, Plywood or Foam.

In addition to this PPE, you must follow these safety guidelines during operation:

1. **Never touch the spindle** with your hand or other materials during operation.
2. The bits are **very sharp**, use caution when loading and handling tools.
3. All personnel and observers should stand behind the yellow and black caution tape during operation, the machine is incredibly powerful, can move very fast and will not stop unless you tell it to.
4. Often pieces of material can fly off the machine at high speeds. Use the **protective barrier** to position between people in the room and the machine.
5. **Never** cut material that cannot be securely affixed to the table or is larger than 48 inches in the x-dimension and taller than 2 inches in the z-dimension.

For troubleshooting see Section III of this handbook, for questions about the machine's operability, broken bits, or errors not encountered in this manual, please contact Digital + Physical Computing Technician, Jenna Boyles, jboyles@andrew.cmu.edu

1.2 Machine Inspection and Setup

1. Make sure the spoilboard is clear of any material or debris.

2. Check the oil reservoirs (one above the X axis and 2 for the Y axis) to see that there is an appropriate amount of oil in them *(Fig2).* *If the oil level is low please alert Technician.

3. Check that the main air regulator behind the machine reads between 90-120psi *(Fig 3).*

4. Check that the correct bits are in the tool holders *(Fig 1)* that correspond with the School of Art RhinoCAM tool library *(Fig 4).*

5. Ensure bits are securely tightened in collets.

6. Inspect bits to make sure the bits are not damaged and the bits for your job are good to cut. Double check that the bits do not slip from the collets.

7. Ensure tool cones are properly positioned in the tool holder.

8. Make sure all doors on the CNC computer (big blue cabinet) are closed to prevent dust from getting in.

Fig. 2: Oil reservoirs

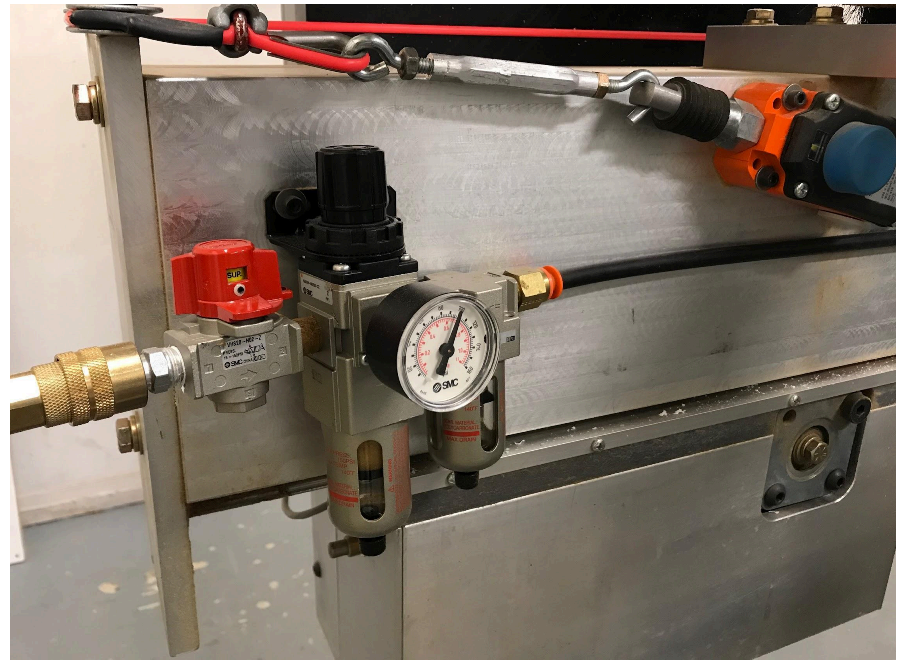


Fig. 3: Main air regulator, 90 - 120 psi

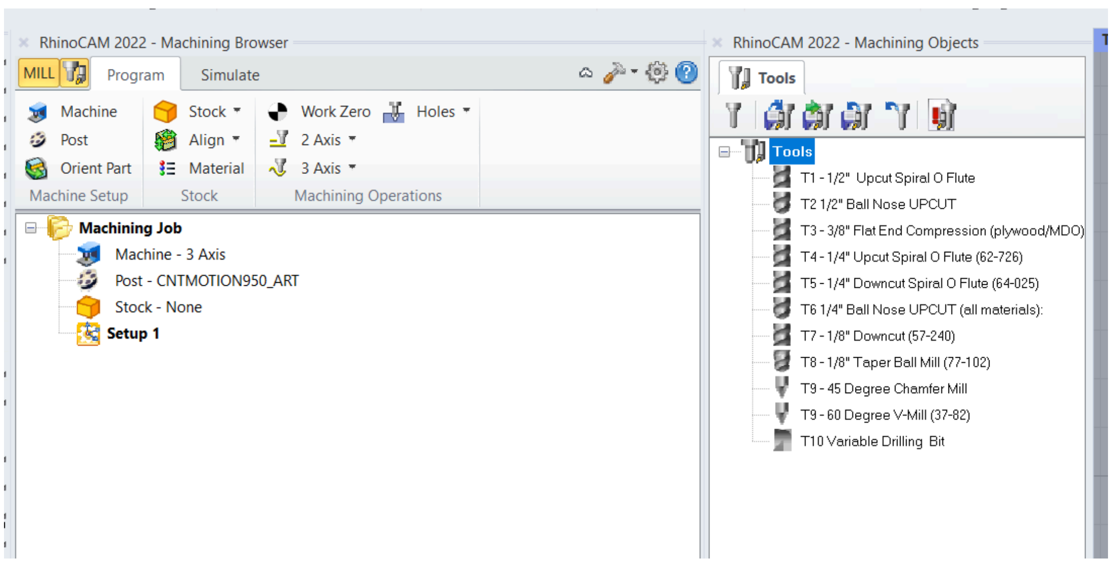


Fig. 4: RhinoCAM Tool Library (T1-T11 correspond to labels in Fig 1)

1.3 Installing Tools

The **Tool Library** *(Fig 1)* consists of a standard array of bits that are in the machine at all times. Replacing or installing bits should **only** be done by a technician. Students are not permitted to change tooling without permission or supervision.

If there is a discrepancy with the bits or you need a special bit for a specific project, contact **Digital + Physical Computing Technician, Jenna Boyles, jboyles@andrew.cmu.edu**

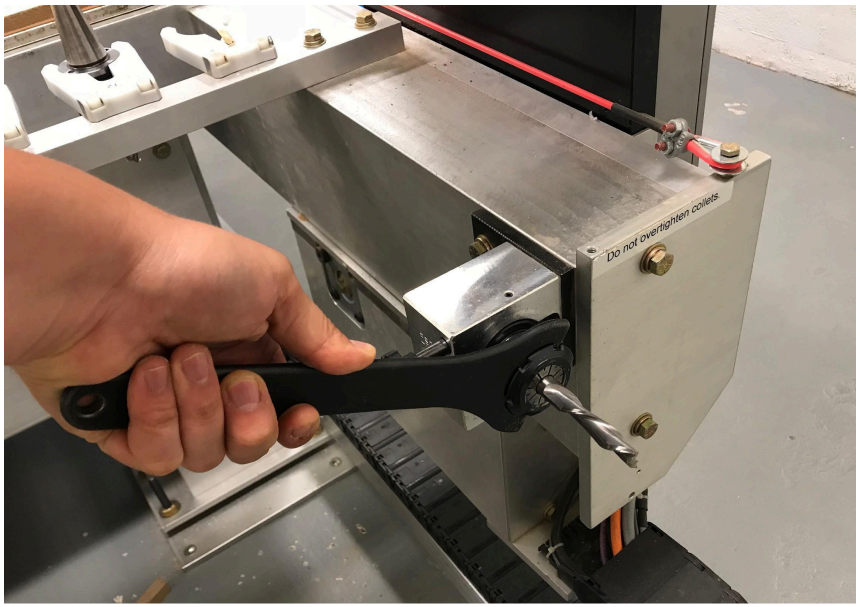


Fig. 5: Using holder for tool cone at the back of the machine and collet wrench to tighten the collet around a bit.

1.4 Turning ON the Machine

1. Turn the key to the right and the red E-Stop button should release towards you *(Fig 6)*

2. Press the Green ON button.

3. Open panel on the left side of the operator cabinet, and turn on the computer.

4. Close and latch panel, and wait for the computer to boot into Windows.

5. Turn on the Master Switch.

6. Turn on the Enable Switch.

7. Walk over near the spindle and pull the air valve down to open it.

8. Make sure the pressure valve over the spindle reads **72psi** *(Fig 7)*

9. Open the WinCNC Program, located on the desktop of the computer in the blue cabinet

Fig. 6: CNC computer cabinet

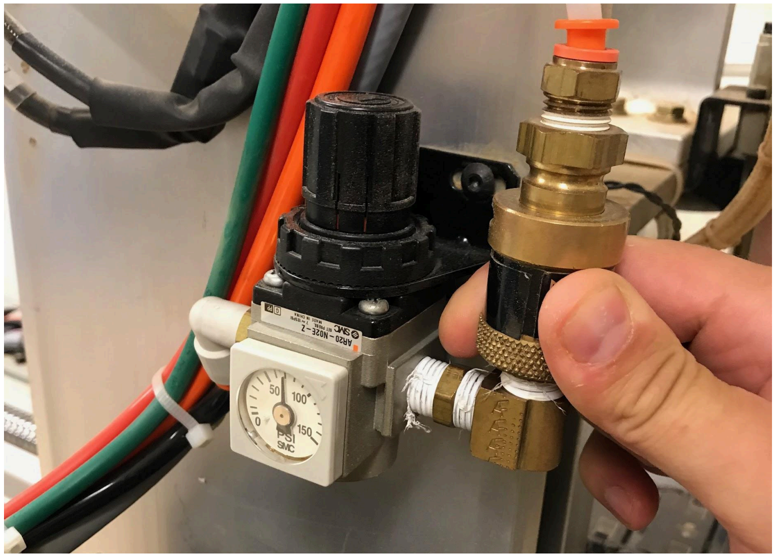


Fig. 7: Turning on the spindle air valve.

1.4.1 Warming Up The Machine

The CNC must complete the warm up sequence on start up or after remaining idle for more than 1 hour, **no exceptions**. Before running this sequence make sure that T1 has a standard half inch bit that has been properly installed.

1. Press the “**Warm Up**” button on WinCNC (*Fig 8*)

The Warm Up sequence will Home the machine, pick up T1, and warm up the spindle by running at several increasingly higher RPMs.

This should take about 5 minutes. Once the Warm Up sequence is complete, the machine is ready for use.

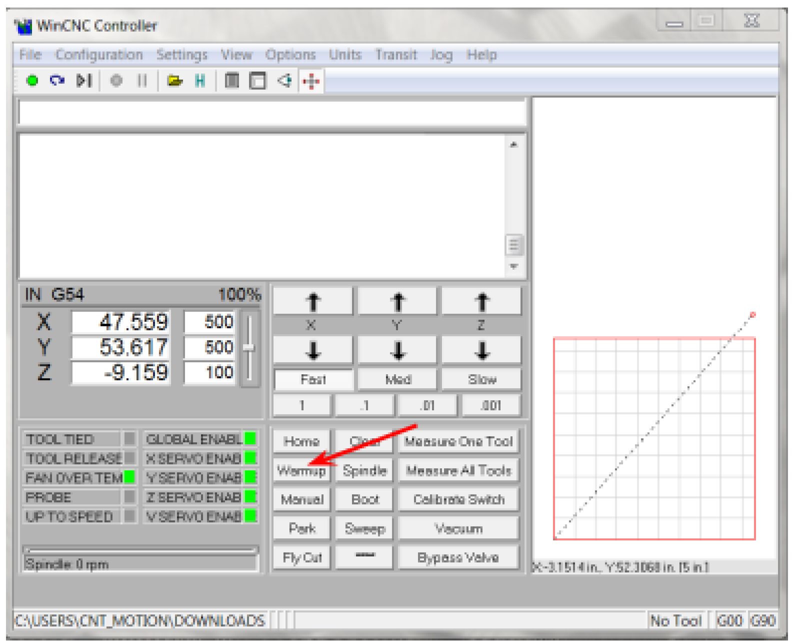


Fig. 8: The WinCNC interface

1.4.2 Turning ON Dust Collector

Before starting any cut, turn on the dust collector by pressing the green button near the door labeled Start Blower (*Fig 9). *In order for the dust collector to work properly, three blast gates must be checked.

1. **Open** the blast gate vent above the example shelves by pulling down (*Fig 10*).

2. **Close** the blast gate vent to the left of the vacuum hose (*Fig 11*).

Fig 9: Blower Controls

Fig. 10: Blast gate above shelves

Fig. 11: Blast gate by vacuum hose

1.5 Turning OFF Machine

1. Clear off all material from the spoilboard.

2. With the dust collector on, run the **Sweep** operation to clean up debris.

3. Run **Park** operation.

4. Power down the machine and computer

    i. Turn **Global Enable Switch** OFF on the blue operator cabinet console.
    ii. Turn **Master switch** OFF on the blue operator cabinet console.
    iii. **Exit WinCNC** Controller
    iv. **Shutdown** Windows computer. Wait for complete shutdown.
    v. Open panel on the left of the blue operator cabinet and **power down PC**
    vi. **Press Red E-Stop** button.

5. **Turn off air pressure flow** by popping the air spindle back up (no more hissing sound) (*Fig 7*).

1.5.1 Cleaning Up

When finishing your session, it is **your responsibility** to clean up areas both on and around the CNC. As a monitor or technician, it is your responsibility to make sure that users help you clean up after each session and at the end of every shift.

1. Use the **Sweep** button in WinCNC, with the dust collector **ON** to remove dust and debris from the spoilboard.

2. Use the **air nozzle** located to the right of the doors to spray down the machine, starting from the back of the room and moving dust and debris to the front.

3. Use the **vacuum** attachment and broom to clean up any dust and debris remaining in the room.

4. When done using the Sweep function and vacuum attachment, turn OFF the dust collector by pressing the red **Start Shaker** button (*Fig 9*).

1.6 Basic Control Operations

Jogging

Select the speed at which you want to jog the spindle head on the WinCNC interface. **Use caution** when jogging the bit to avoid collisions, especially when moving along the Z-axis, operating near tool cones, and loaded materials.

Use the **arrow keys** to move the spindle head along the **X and Y-axis**
Use "**Page Up**" and "**Page Down**" to move the spindle along the **Z-axis**

Pausing

Press **Spacebar** to pause the machine during an operation. Keep in mind that if the spindle is spinning, it will continue to spin. Press “**Enter**” to resume the operation or “**Esc**” to abort.

Aborting

If you want to completely abort an operation and stop the spindle, press “**Esc.**”

Emergency Stop (E-Stop)

The E-Stop should only be used in the event of an emergency where you need to stop the spindle and shut off the machine immediately. The E-Stop can be activated in two places: by pressing the **large red button** on the operator cabinet, or by pulling the **bright red cable** around the perimeter of the CNC bed. Once activated, the machine will completely shut off and will need to be reset (*Fig 12*) and homed to resume operation.

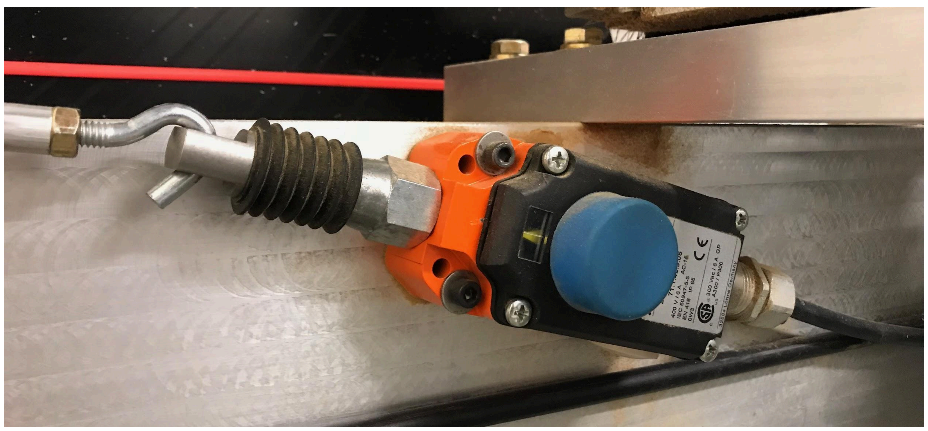


Fig. 12 : If the e-stop cable gets pulled, press this blue button to reset it. You will also need to restart the computer, the CNC, and home the machine to resume operation.

1.6.1 Basic G-Code Commands

While most of the time you will be loading files containing G-code to run a job, you can also type individual G-code commands into the WinCNC controller to run basic operations*. Below are some examples of some very basic commands used in day to day operations:

***Use caution when typing commands and do not experiment, entering certain commands by accident may lead to disastrous results.**

Picking up a Tool

T1, T2, T3.... etc

Returning a Tool

T0 will return your current tool and not pick up another tool. Press T (any other tool number) to pick up a different tool.

1.6.2 Measuring Tools

Before running any CNC job, we need to define two essential things:

1. **Where is the top of the spoilboard?**
2. **How long is each tool?**

Defining the top of the spoilboard as absolute zero for the Z-axis (**Z-Zero**) will ensure that the geometry of the CNC table is consistent with your CAD/CAM environment. Read on to learn how to conduct this calibration:

1.6.3 Calibrating Z-Zero height

If we decide to use a new spoilboard, or possibly if we notice that our bits are cutting too deep into the spoilboard or not deep enough, use the following procedure to make sure that Z-zero is properly calibrated.

1. Remove all material and debris from the spoilboard.
2. Ensure that a flat end mill is installed in T1, type "**T1"** and hit enter to pick it up.
3. Click the **Manual** button on WinCNC
4. Jog the machine, using the arrow keys, so that the bit is hovering over an untouched/uncut part of the spoil board near enough for you to have a good view of the end of the bit. (if the boot is down, raise it by pressing "**Boot**" in WinCNC).
5. Turn on the **Vacuum** in WinCNC (not the dust collector).
6. Click the "**Slow**" movement setting in WinCNC, and slowly lower the tool manually, using **Page Down** on the keyboard until the bit is just above the spoilboard.
7. Confirm that the spindle is not currently running, and give the tool cone a good spin using your hand.
8. Switch from "**Slow**" mode to "**.01**" incremental setting. Now everytime you hit Page Down, it will lower by 0.01".
9. While keeping the spindle spinning just by the force of your hand, lower the bit incrementally until the bit stops rotating by contacting the top of the spoilboard.
10. Hit the **Calibrate Switch** button on WinCNC.
11. This should set the top of the spoilboard as absolute zero, now **measure all tools**.

Once this is established, it doesn't need to be done again **unless** we flycut or change out the spoilboard.

1.6.4 Measuring All Tools

With many users, it's impossible to know for sure how someone might have left the tools prior to you using it. It is best practice to **measure all tools** right when you turn on the machine, or anytime you change multiple bits.

1. Click the **Measure All Tools** button the WinCNC or type G37 in the command line and hit enter.

1.6.5 Measuring One Tool

This is useful when you change out a single tool or need to readjust the same tool.

1. Pick up the tool you want to measure by typing T1, T2, T(n)... etc

2. Click **Measure One Tool** in the WinCNC Controller screen or type G37Z in the command line and hit enter.

3. The head will pick up and go over to the tool touch off switch. The tool will then be lowered to the tool measure switch (*Fig 13*). The tool length offset will be set to the calibrate switch offset that you set up earlier when you did calibrate switch.

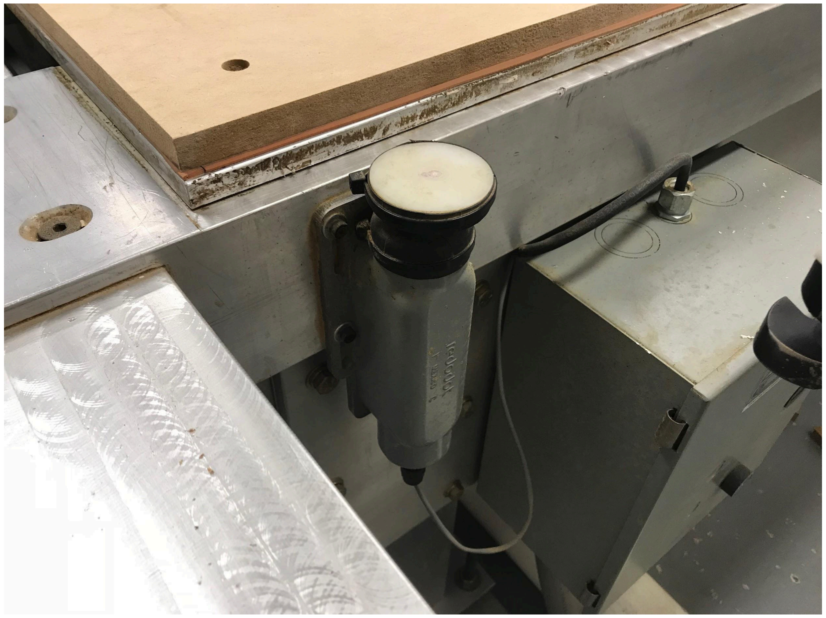


Fig. 13:
Tool Measure Switch, always keep clear.

1.7 Loading and Securing Stock Material

Properly securing your material to the spoilboard is essential to successfully complete your job. Not securing your stock can harm the machine, bits and operator.

1. Clear all dust and debris from the spoilboard.

2. Make sure that the vacuum (not the dust collector) is **OFF**. Use the bypass valve button in WinCNC to turn off the vacuum and carefully position your stock on the spoilboard.

3. Arrange your stock so that the placement matches the toolpaths and geometry on your CAM software. In absolute mode, X0Y0 should be the bottom, left corner of the table, closest to the operator. In relative mode X0Y0 may be in a user defined position. If unsure, enter **G90** to toggle to absolute mode to ensure that X0Y0 is at absolute zero.

4. Use the bypass valve button to turn on the vacuum table and test the suction of the table. If you can easily move your material around, you need to reinforce it with tape. If you are routing a small piece, always secure the piece to the table since the small part won't have enough surface area for the vacuum table to get a solid hold on it. Also, make sure to **include bridges** (Section 2.2.1) to prevent your pieces from coming loose as the router cuts.

Advanced Techniques:

- The more surface area of the spoilboard that is left exposed, the weaker the suction. To increase suction on smaller stock pieces, lay material (that is either the same thickness or thinner to avoid toolpath collisions) on exposed / unused sections of the spoilboard.

- You may use wood screws to fixture jigs and small part stock to the spoilboard. In order to protect the aluminum vacuum table, do not use screws that pierce all the way through the spoilboard. Only use screws that only go part way through the spoilboard. Additionally, screws should only be used if you are absolutely sure that your toolpath will not cause the bit to collide with a screw. This can completely ruin a bit and possibly injure an operator. Only use screws with extreme caution.

2.0 Tool Pathing with RhinoCam

2.2 Loading Post Processor

When we use RhinoCAM to post tool paths (which are text files containing G-code), we need to make sure that the code we're sending is something that our machine can both understand and is within its capabilities. A **post-processor** does just that, it's a file specific to the School of Art's CNT MOTION CNC router in Doherty Hall.

Just like loading the correct tool library, loading the correct post-processor is essential. Not having the correct post-processor will either not allow you to load your file, or worse, damage the equipment and endanger its operators.

The following tutorial will show you how to access and load our post-processor into RhinoCAM to get you up and running on our machine.

Step 1: Download the latest SOA Post-Processor file

You can find the SOA post-processor for our 4 x 4 CNT Motion CNC router in a number of places on the Facilities page of the School of Art website. The download link is available here.

Step 2: Open the Machining Browser in RhinoCAM

Open Rhino 7 (64-bit), if RhinoCAM does not open automatically, in the RhinoCAM 2024 drop-down menu at the top of the screen, select "**MILL**" to open the Machining Browser (*Fig 14*). If the browser is already open and it says "TURN" instead of "MILL", click on the TURN tab to swap over to MILL. Turn is only for CNC Lathes which is not applicable to our machine, we only want to be producing toolpaths in MILL mode.

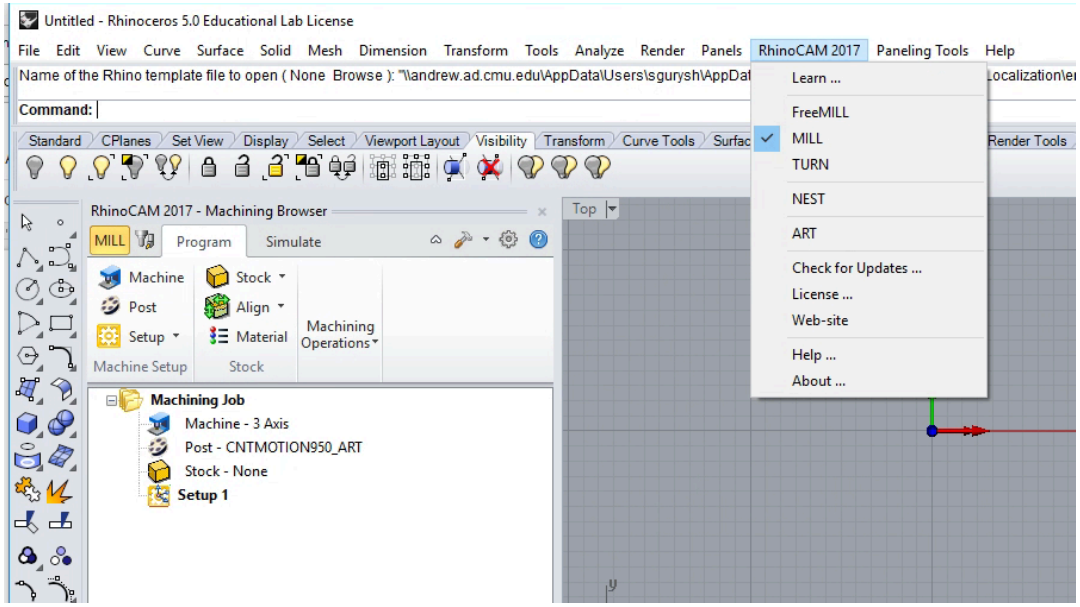


Fig 14: Opening machining browser in RhinoCAM

Step 3: Selecting a Post

In the Machining Browser, locate the **Post** icon (*Fig 15*), just below "Machine" under the MILL tab. Double Click to open up "Set Post-Processor Options" window.

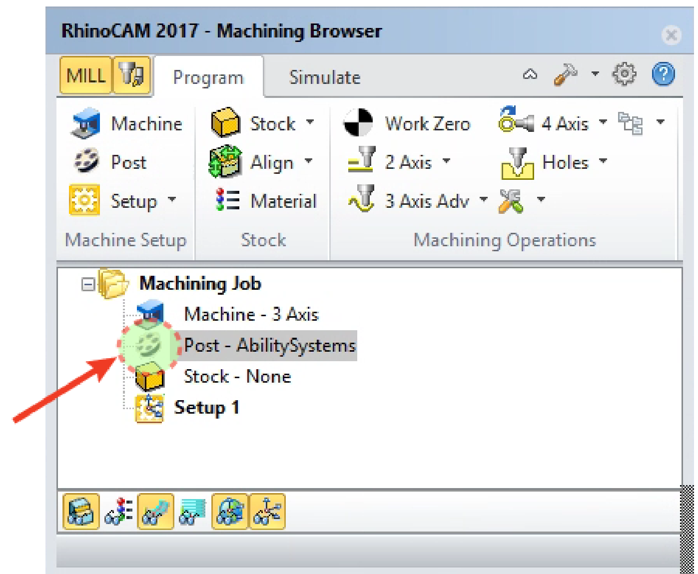


Fig 15: Post located under "Machining Job"

Step 4: Change the Folder Location

Where it says "Folder where post-processor files are located," click the ... button to the right and navigate to the folder where you saved the post-processor. Most likely it will be in **Downloads.**

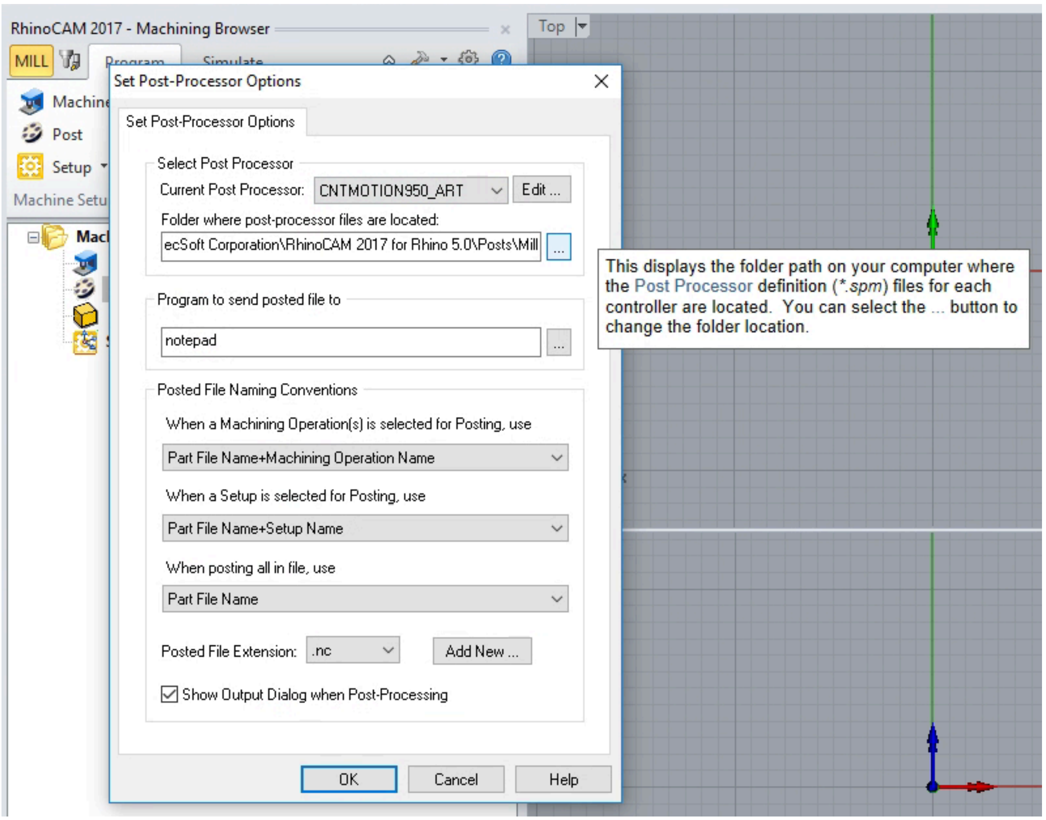


Fig 16:

Step 5: Select CNTMOTION950_ART in the "Current Post Processor" drop down menu

If you can't find it listed, return to the previous step and make sure that you have the correct folder location selected.

Hit "**OK**." Now the Machining Browser should indicate **CNTMOTION950_ART** as the current post-processor.

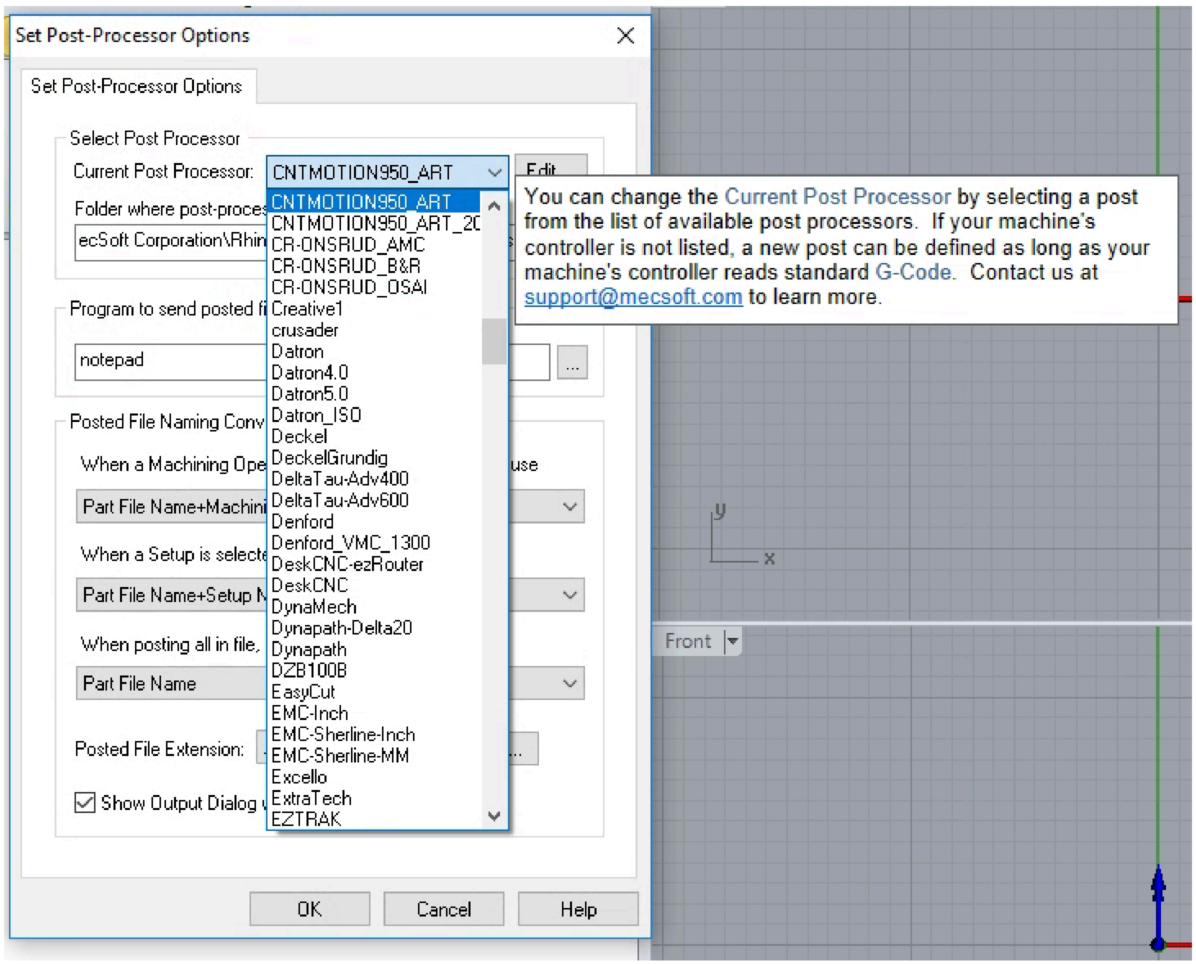


Fig 17

Step 6: Posting a Tool Path

When you are ready to post, after producing some toolpaths, right-click on your Setup and select Post (*Fig 18*). A window should pop up asking what you want to name your file and where to save it. Make sure that the **Current Post** is also indicating CNTMOTION950_ART (*Fig 19*).

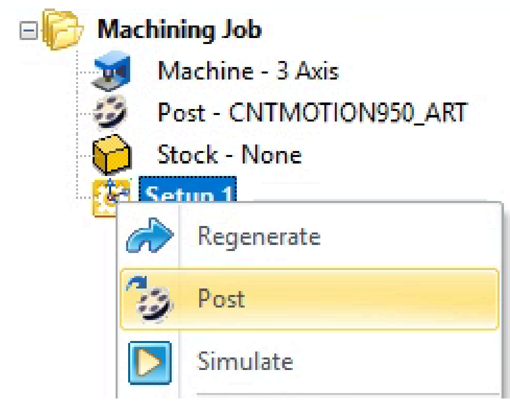


Fig 18:

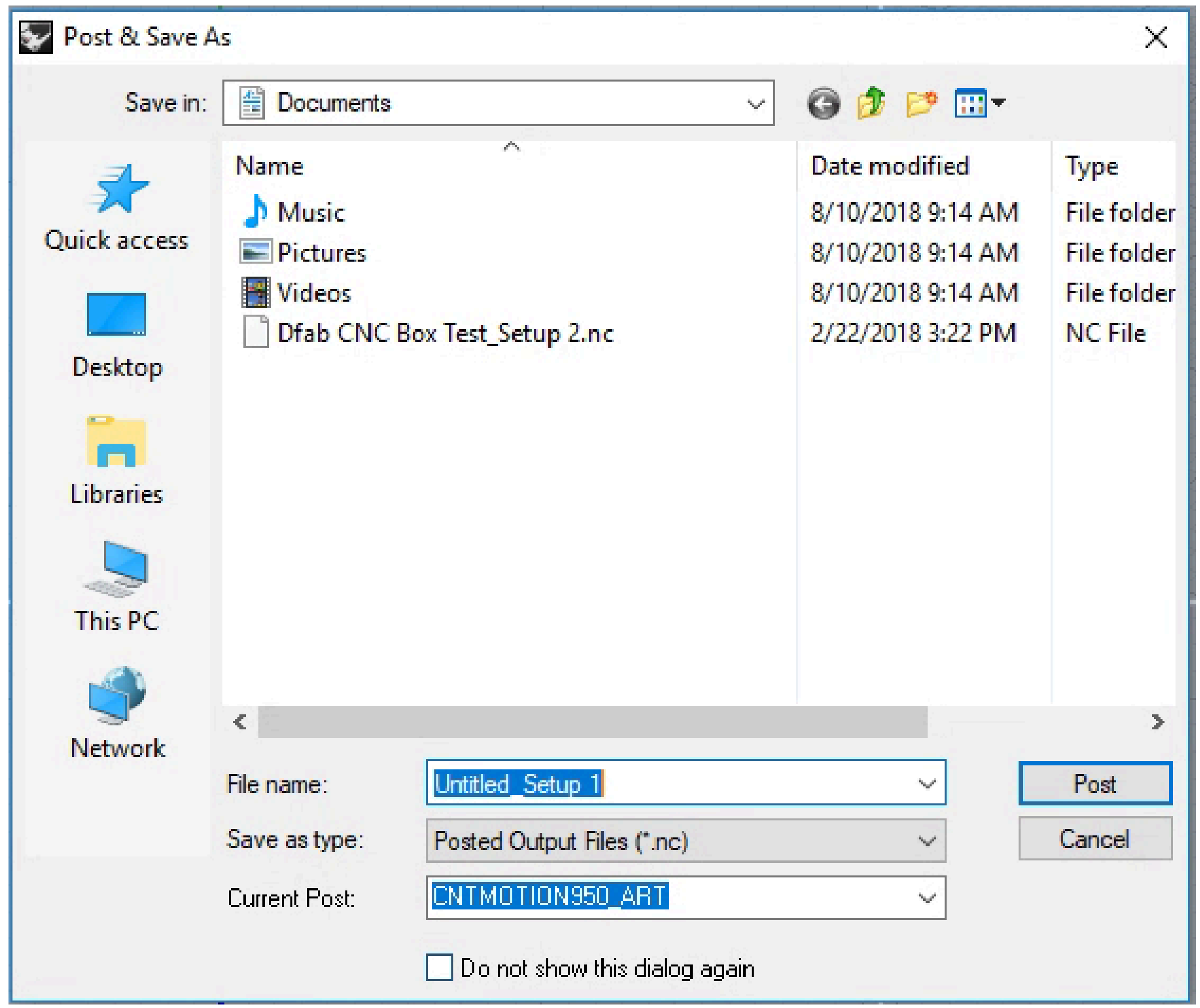
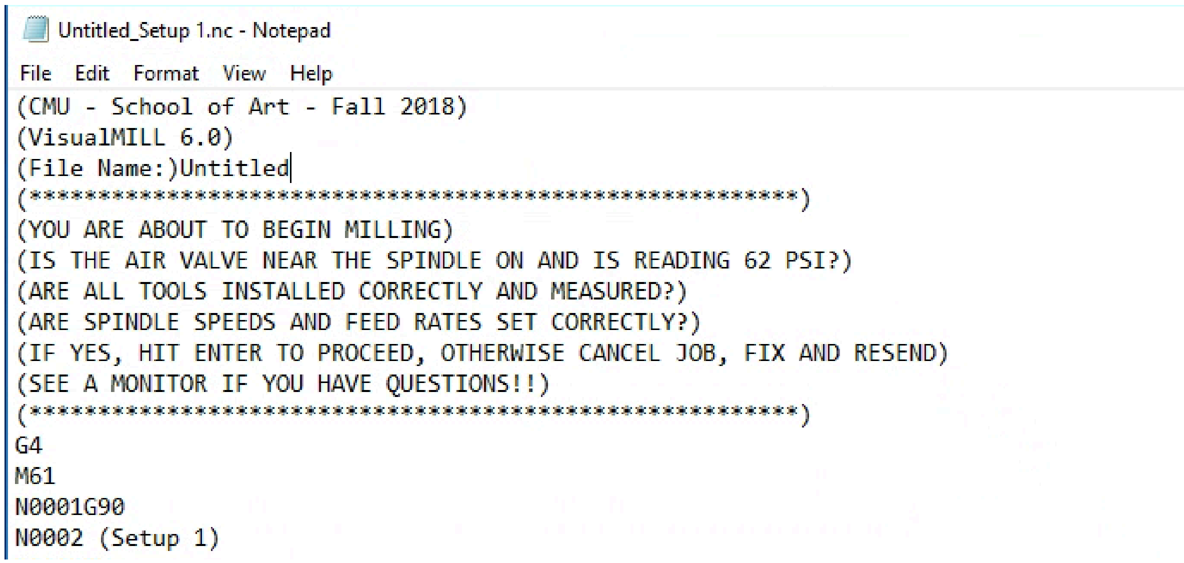


Fig 19:

Step 7: Reviewing your G-Code

After posting, the header should look like this:



Untitled_Setup 1.nc - Notepad

File Edit Format View Help

```
(CMU - School of Art - Fall 2018)
(VisualMILL 6.0)
(File Name:)Untitled
(**********************************************************)
(YOU ARE ABOUT TO BEGIN MILLING)
(IS THE AIR VALVE NEAR THE SPINDLE ON AND IS READING 62 PSI?)
(ARE ALL TOOLS INSTALLED CORRECTLY AND MEASURED?)
(ARE SPINDLE SPEEDS AND FEED RATES SET CORRECTLY?)
(IF YES, HIT ENTER TO PROCEED, OTHERWISE CANCEL JOB, FIX AND RESEND)
(SEE A MONITOR IF YOU HAVE QUESTIONS!!)
(**********************************************************)
G4
M61
N0001G90
N0002 (Setup 1)
```

Fig 20: Check that the header has these messages

2.2 Loading SOA Tool Library to RhinoCAM

In order to generate the proper tool paths for your project, RhinoCAM needs a virtual representation of the physical tools that have been installed in each tool cone. Using an incorrect tool library can cause serious damage to the tools, your material, the machine itself, and even injure the machine operator.

This is why it is essential to make sure you have the School of Art CNC tool library loaded into RhinoCAM when posting tool paths to the CNC router. Every laptop in the Physical Computing Lab, the Laser Lab, and the CNC room comes with the SOA CNC Tool Library pre-installed, however if you are using your own computer or logging onto a computer in a lab cluster, you must load that tool library into RhinoCAM the first time you use it.

If you have a special request to use a tool that is not in our tool library, please contact **Digital + Physical Computing Technician, Jenna Boyles, jboyles@andrew.cmu.edu**
Step 1: Download the SOA Tool Library

You can find the Tool library in a number of places on the Facilities page of the School of Art website. The download link is available here.

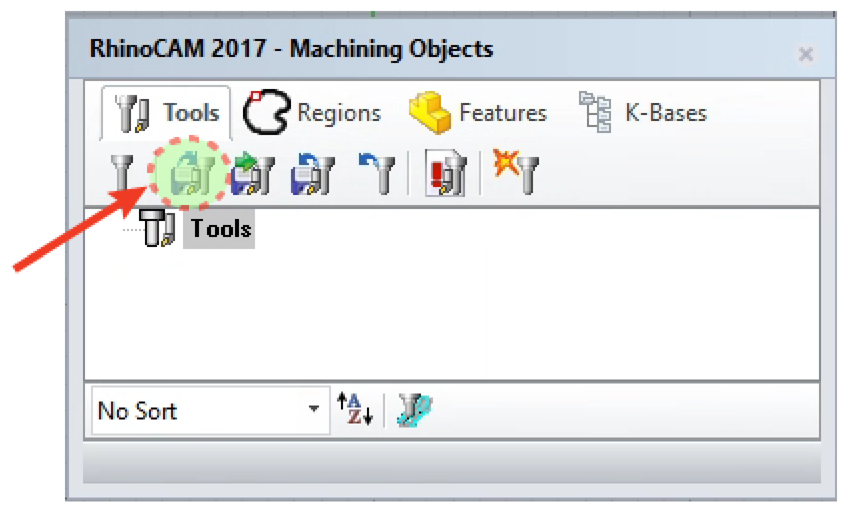
Step 2: Open the Machine Objects Browser

If RhinoCAM does not open automatically, in the RhinoCAM drop-down menu at the top of the screen, select "**Machine Objects Browser**" to open the tool library.

Step 2: Select Load Tool Library

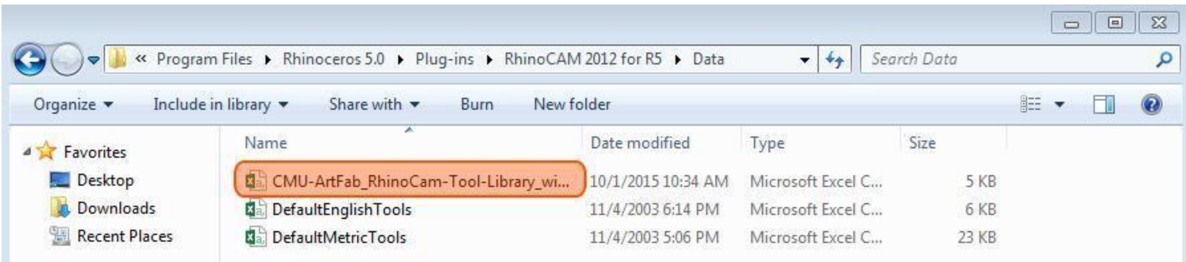
The Machining Objects Window will open. There are four main tabs: **Tools, Regions, Features and K-Bases.** Make sure you are on the **Tools** tab.

Under the **Tools** tab click the second icon from the left that says "**Load Tool Library**."

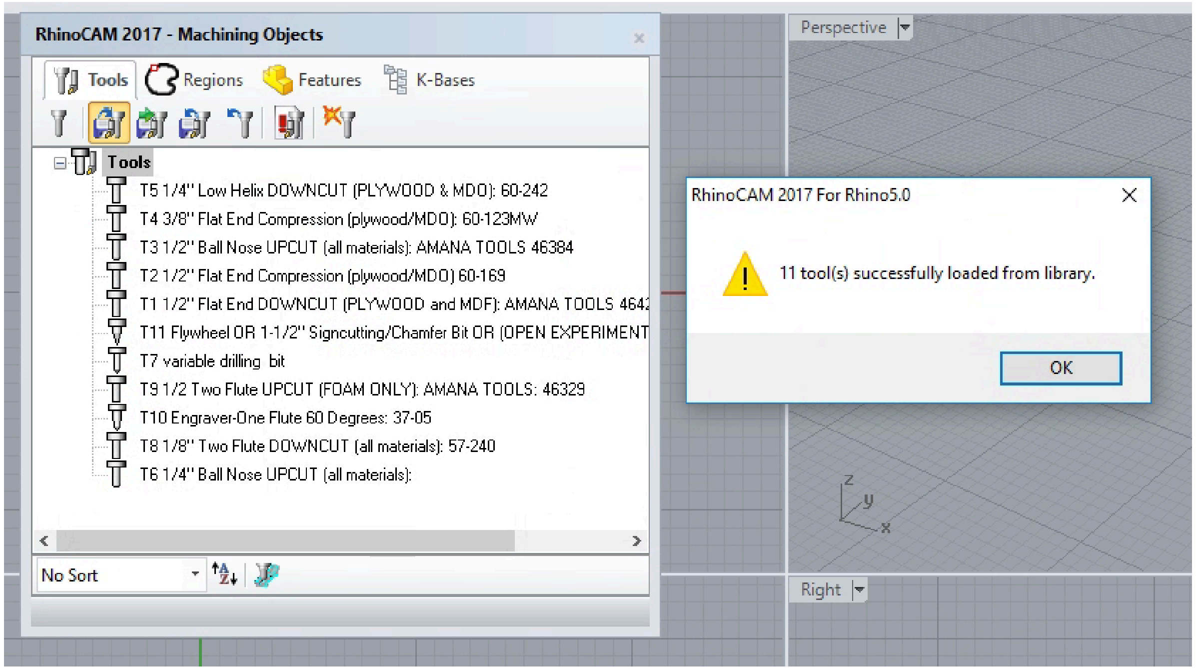


Step 3: Open tool library

A new browser window will open to let you select your file. RhinoCAM can open tool libraries as a .CSV or .VKB file. Navigate to the file you downloaded previously, and open it. Most likely it will be in **Downloads.**



The tools will appear in a list below and a dialog box should pop up saying the total number of tools that have been successfully loaded into the Machining Objects Browser. **Do not edit the tools** after they are loaded. This library is managed by School of Art Staff. If you have a special request to use a tool that is not in our current library, please contact **Digital + Physical Computing Technician, Jenna Boyles, jboyles@andrew.cmu.edu**



Step 4: Begin Toolpathing

Now that you have successfully loaded all of your tools, you can close the Machining Objects Browser to provide more visible workspace on your screen and return to the Machining Browser

When you open a new toolpath, and navigate to the Tools tab, you should now be able to view all the tools that you just previously loaded into the library. If you select one of the tools, you should see information on the right describing the geometry of the bit, such as its diameter, length, number of flutes, etc.

2.3 2 ½ Axis Operations

2 ½ Axis Machining operations create toolpaths that follow 2D curves, or flat surface geometry, to cut to a programmed depth. Cuts are determined by the X and Y coordinates at each point along the surface edge or curve. 2 ½ axis milling is used primarily for cutting sheet materials.

1. Click on the **RhinoCam 2024** tab, then click on the **Machining Operations Browser**.
2. After the Machining Operations Browser opens up, find the “**2 Axis**” category and select the appropriate operation you need.

Here are brief descriptions of the more commonly used 2 ½ operations:

PROFILING
Profiling toolpaths can follow either the inside or outside edge of a curve or surface edge. This is the most commonly used type of machining. It is easy to set up and fast to cut. By default a profile cut is offset from the selected curve geometry by ½” the diameter of the selected tool.

ENGRAVING
Engraving toolpaths follow curves on the center to apply text or engrave a design.

POCKETING
Pocketing toolpaths cut away all of the material inside of a selected region to create pockets. Islands may be created using [Avoid Regions]

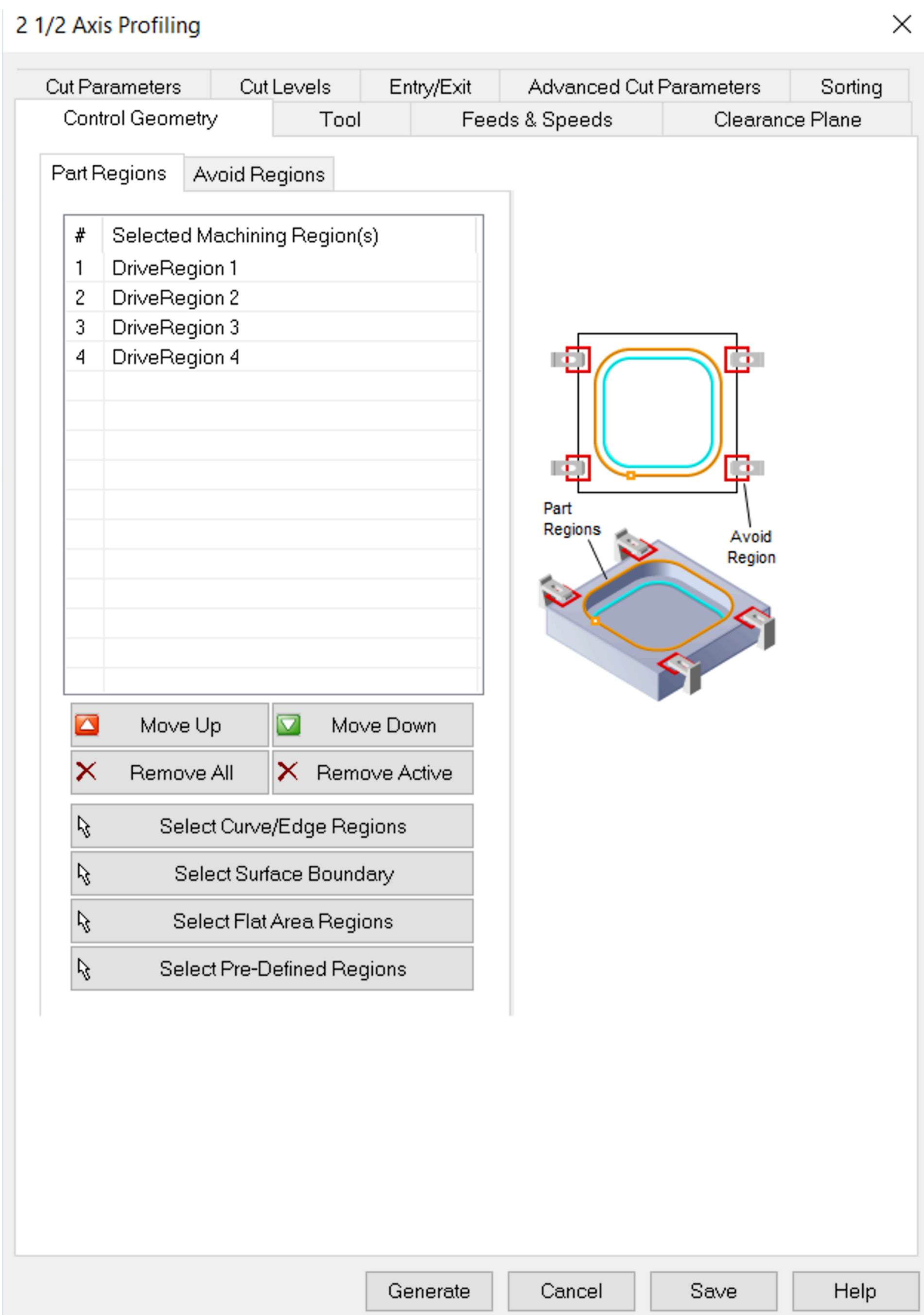
FACING
Facing toolpaths machine the entire surface to uniformly flatten an area of material.

2.4 Preparing 2 ½ Axis Toolpath Parameters

A toolpath is a series of coordinate positions that determine the movement of a tool during a machining operation. RhinoCam has incredible flexibility in defining custom toolpaths to produce a wide variety of results. However, defining a toolpath requires many parameters that must be entered and selected carefully.

Before a toolpath can be generated, parameters need to be determined in each of the tabs appearing at the top of the 2 ½ Axis Profiling dialog box:

- **Control Geometry**
- **Tool**
- **Feeds & Speeds**
- **Clearance Plane**
- **Cut Parameters**
- **Cut Levels**
- **Entry/Exit**
- **Advanced Cut Parameters**
- **Sorting**


2 1/2 Axis Profiling
Cut Parameters
Cut Levels
Entry/Exit
Advanced Cut Parameters
Sorting
Control Geometry
Tool
Feeds & Speeds
Clearance Plane
Part Regions
Avoid Regions
Selected Machining Region(s)
1 DriveRegion 1
2 DriveRegion 2
3 DriveRegion 3
4 DriveRegion 4
Move Up
Move Down
Remove All
Remove Active
Select Curve/Edge Regions
Select Surface Boundary
Select Flat Area Regions
Select Pre-Defined Regions
Part Regions
Avoid Region
Generate
Cancel
Save
Help

2.4.1 Control Geometry

All operations require that you first select the geometry with which you will use to cut a profile.

1. Click **Remove All** and then click **Select Curve/Edge Regions**. After selecting this, the window will return you to the Rhino Viewport to make your selection.

2. Click each of the curves you wish to choose. Hit **Enter** when finished. If you pick too many, use **Ctrl-Click** to deselect, or delete it from the list later.

3. Confirm that you have selected the right geometry, and move on to the next tab.

Notes:

- For certain operations, it is often useful to also specify **Avoid Regions**, for islands, fixtures, etc..

- For profiling operations, **Inside Cuts** and **Outside Cuts** must be done on separate operations

2.4.2 Tool

The tool tab allows you to select the tool for your operations. Select your tool. The laptops should be loaded with the current tool library, but always double check that the tool you select is actually in the physical slot specified on the machine.

**Do not edit the tools, the tool library is managed by School of Art staff.
**If you do not see any tools listed, you may have to import the Artfab Tool library by following the steps outlined in section 2.2

2.4.3. Feeds and Speeds

This is a very important pane, this tab determines the speeds of the spindle as is spins, and moves throughout the operation. These values depend on the material you cutting, the bit your using and the specific machining operation. Incorrect feed rates can damage the material, the bits, and even increase the likelihood of material catching on fire.

Use the spreadsheet in the binder to determine the feeds and speeds for your cuts.
Always double check the values you put in, its very easy to accidentally add an extra digit.

Here are what some of those feed rates mean:

- **Spindle Speed**: the number of Rotations per Minute (RPM) the router bit will be spinning
- **Plunge:** the speed the bit moves downward in the Z axis motions.
- **Approach:** Speed at which the tool will travel during Entry/Exit Motions.
- **Engage:** Speed at which the tool travels when 50% of the Bit diameter is engaged in the material.
- **Cut:** The speed at which the tool travels when 100% of the bit Diameter is engaged in the material
- **Retract:** Indicates how fast the tool is moving upwards along the Z axis toward the clearance plane.
- **Departure:** The speed at which the tool moves after it has reached the clearance plane.

2.4.4 Clearance Plane

The Clearance Plane is the height the machine will go to when it is not cutting, this height should be tall enough for the machine to move around without hitting anything on the spoilboard.

1) Always use "Absolute Z" and set it at least .5" more than the thickness of your material

2.4.5 Sorting

1) Select "Minimum Distance Sorting" - This will drill or mill one piece of geometry first, before moving to the next closest Drive Geometry.

2.4.6 Advanced Cut Parameters

The only option you need to change in this tab is the "Bridges" when you are doing Profiling cuts.

1) if you are doing a Profiling operation, make sure you create bridges. Bridge dimensions must be strong enough to prevent your machined part from breaking free from the material. *For instance: bridges for plywood should be a minimum of .01" tall and .02" wide

2.4.7 Entry/Exit

This tab allows us to determine how the bit engages/Enter and how it departs/Exits our stock.

1) For Profiling, engraving, and Facing operations - click none on both Entry and Exit
2) For Pocketing select Path for Entry, and Linear for Exit.

2.4.8 Cut Levels

Pay special attention to this section as it control the depth of cut for your operation The School of Architecture's dFab explains how this tab works:

These settings control how the Milling Tool steps into your material. Say for instance, we were milling 3/4″ MDF; we can't plunge the tool into the full 3/4″ of Material (unless we have a more advanced tool). Therefore, we program the tool to 'step' into the material, and repeatedly mill out the Profile Curve in multiple passes- rather than one, single pass. The tool will 'step-down' into the material; then mill out the Profile Curve; then it will 'step-down' again, and mill the same Profile Curve. It will repeat this process until it has reached a depth of our choosing.

- [Location of Cut Geometry]

1) Measure your material using a set of calipers
2) Click "Pick Top" and put the thickness of your material

- [Cut Depth Control]

1) "Total Cut Depth" - how deep the tool should go for the cut - If you want to cut through your material, set this to the thickness of your material
2) "Rough Depth" should be the same as your "Total Cut Depth" and "Finish Depth" should be 0
3) "RoughDepth/Cut" this is the height of the stepdown between passes. Base this off the feeds and speeds spreadsheet for common materials that is in the binder.

- [Cut Levels Ordering]

1) Click "Depth First" - it will save you time.

2.4.9 Cut Parameters

- [Stock] - refers to the amount of material the bit will leave between the actual cut and your model.
- If you needed to sand your piece after milling it, you may want to leave some stock
- [Cut Direction] -

1) Click "Mixed" unless you are cutting expensive/exotic hardwoods - in which case you would want to click "Conventional"

- [Stepover] - This is for pocketing, or facing operations and should be based on the step/cut value in the spreadsheet based on your material, material thickness and bit diameter
- For Profiling Operations - make sure you select "Use Inside/outside for Closed Curves" and then click either inside or outside based on what you want.

2.5 3D Operations

The router is capable of very detailed 3D operations but these cuts are more advanced and require more careful consideration when modeling and defining your CAM parameters.

One of the main things to know is that 3D operations are broken down into 2 separate operations - a “Roughing” operation and a “Finishing” operation

Roughing Operations

Roughing Operations are the first step of 3D Processes, they remove the bulk of the unnecessary stock before the finishing operations. This is to save time and prolong tool life, since one tool isn’t doing the entire 3D cut.

Finishing Operations

These operations are used for getting more detailed, and accurate parts and are typically done after Roughing operations.

2.6 3D Horizontal Roughing

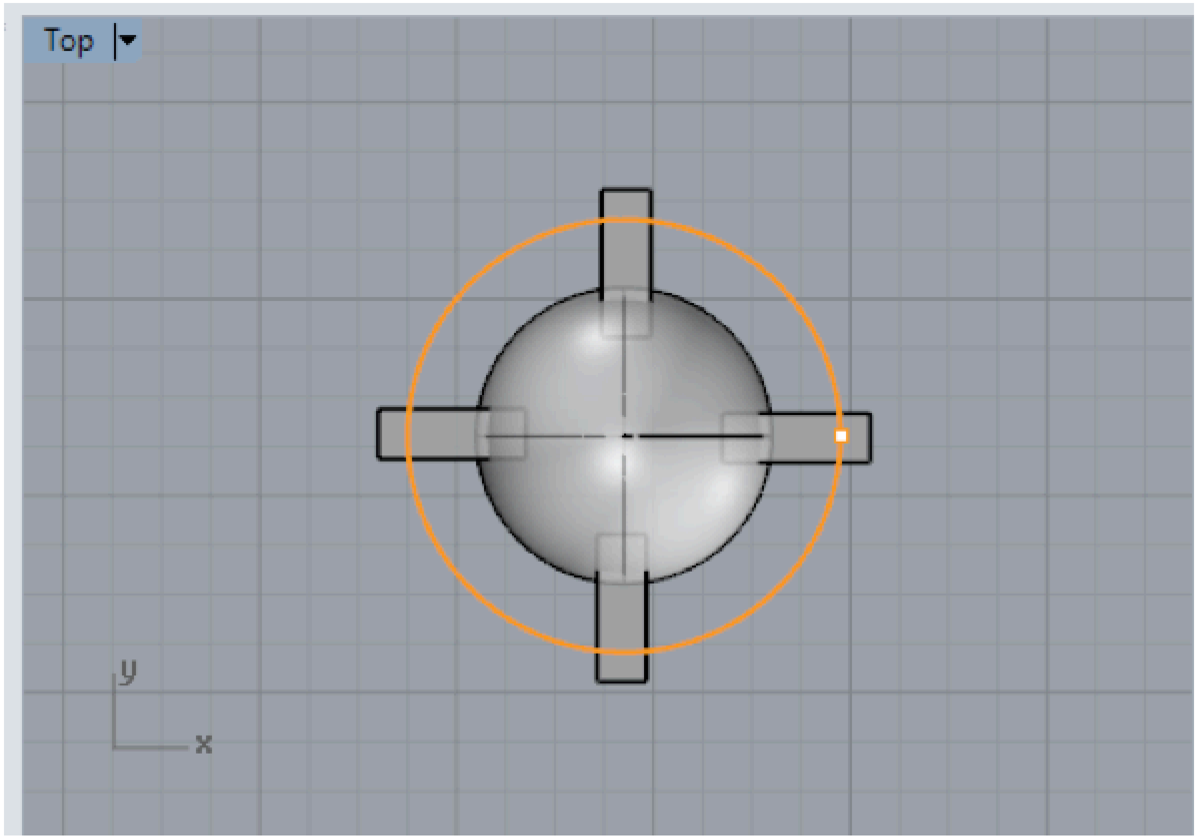
Roughing operations are the first step in a two-step process for 3D Machining. It is used to remove the bulk of unnecessary material away from the stock. We are therefore not really concerned with getting detailed cuts with this operation, just mainly removing material quickly and cleanly to prepare for the Finishing operation.

****Do Not attempt to make 3D cuts unless you are trained how to do so, this guide is meant to refresh and be used as a reference, not to substitute hands-on training.

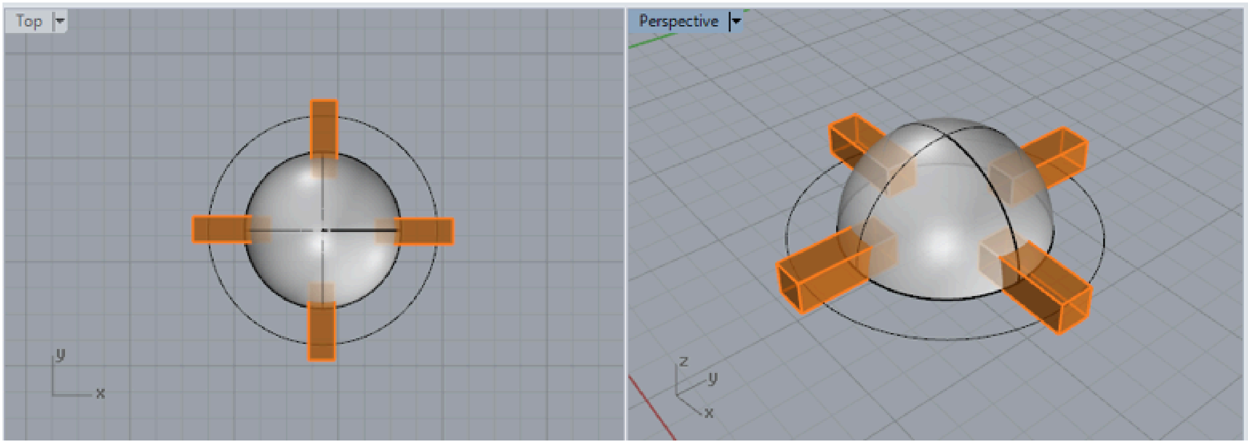
2.6.1 3D Model

There are certain things you need to consider and changes you need to make to a model if you want to make a 3D operation

- Drive Region: Horizontal Roughing requires a closed curve geometry to define the X and Y limitations. It helps the operation decide where it should look for your part. It is important to account for the bit’s diameter, cutting length and tool cone diameter. If you plan on cutting through your material, you should offset your drive containment region more than the chosen bit diameter. Example:



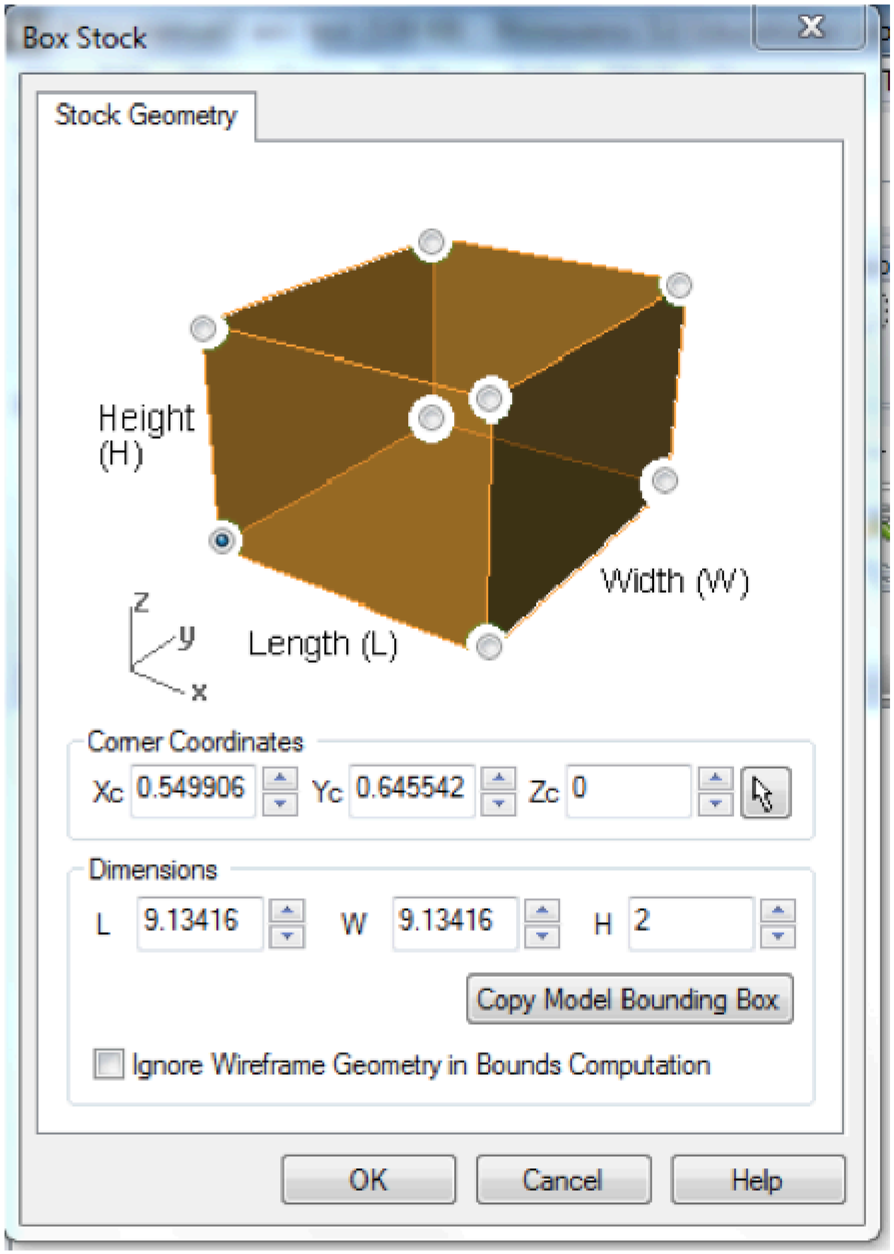
- Bridges: Unlike 2 ½ axis operations, RhinoCam cannot automatically generate bridges for 3 axis operations. if you want to have the whole object cut away from the stock completely You must model your own bridges. Simple shapes like boxes or half-cylinders work best. Here’s an example: **note that bridges fully intersect surface



- Undercuts: The machine cannot do undercuts, it will not try to cut “underneath” your top-most surface. If you model has undercuts. the machine will only cut the top surface and ignore what is below it.

2.6.2 Define Stock

The next step is to define your stock in RhinoCAM.


Box Stock
Stock Geometry
Height
(H)
Width (W)
Length (L)
z
y
x
Corner Coordinates
Xc 0.549906
Yc 0.645542
Zc 0
Dimensions
L 9.13416
W 9.13416
H 2
Copy Model Bounding Box
Ignore Wireframe Geometry in Bounds Computation
OK
Cancel
Help

1) In the Machining Operations browser, double click the “Stock” button under the Machining Job Folder
2) click on the bottom left corner of the cube model.
3) Stock geometry corner coordinates and dimensions can be determined as follows:
a) You can enter the coordinates of the bottom left corner of your material
- For most jobs where you line up the material to X0Y0 of the Machine, the Coordinates should be set to 0,0,0.
- Under Dimensions, type in the measurements of your material.

***Note: Alternatively, you can determine the corner of your box stock coordinates by drawing a rectangle that represents your material length and width in your model. Then, click the arrow button next to the Zc can be used to manually select your stocks X, Y, Z coordinates. This method is useful if your sock is not positioned at X0,Y0.

2.6.3 RhinoCAM Toolpath Parameters

In the Machining Operations Browser select the dropdown menu 3 Axis Adv and click on Horizontal Roughing and a pop-up window will appear with the Horizontal Roughing parameters:

Machining Features/Regions

Like any operation, the first tab is the Machining Features/Regions tab.
1) make sure the list is empty by clicking “Remove All” at the bottom of the window. Then click “Select Drive/Containment Regions”
2) This will return you to the Rhino viewport and allow you to select your drive/containment regions. Click the closed curve geometry you want to use as your containment regions.

*** your containment region should be a closed curve, and your surface geometry should be within your containment region.

Note - If you plan on completely cutting your object away from the material, your containment region should should account for the bit and tool cone diameter *the tool cone diameter is approx. 2”. *A pocketing operation may be necessary if the the tool cone poses a rubbing hazard.The pocketing operation should precede roughing or finishing operations

Tool

The tool tab allows you to select the tool for your operations. Select your tool. The laptops should be loaded with the current tool library, but always double check that the tool you select is actually in the slot specified.

1) Roughing Operations are meant to remove a bulk of material as quickly as possible so for this reason you would most often want to use a larger bit.
2) Make sure the bit is long enough to cut to your desired height WITHOUT the collet rubbing! The collet rubbing is a definite way of burning your material and possibly starting a fire. Do not base this just off the simulation on RhinoCAM, physically take the bit to your material and see if the collet is going to contact your material.

a) A pocketing operation around the perimeter of your drive region may be necessary if the the tool cone poses a rubbing hazard. The pocketing operation should precede roughing or finishing operations but make sure you don't cut away material where your part is during the pocketing operation. See Pocketing for Tool Cone Clearance (Section 2.8)

**Do not edit the tools, the tool library is managed by School of Art staff.

**If you do not see any tools listed, you may have to import the Artfab Tool library by following the steps outlined in section 2.2

Feeds & Speeds

This is a very important pane, this tab determines the speeds of the spindle as it spins, and moves throughout the operation. These values depend on the material you are cutting, the bit you're using and the specific machining operation. Incorrect feed rates can damage the material, the bits, and even increase the likelihood of material catching on fire.

****Use the spreadsheet in the binder to determine the feeds and speeds for your cuts.

****Always double check the values you put in, its very easy to accidentally add an extra digit.

Here are what some of those feed rates mean:

- Spindle Speed: the number of Rotations per Minute(RPM) the router bit will complete
- Plunge: the speed the bit moves downward in the Z axis motions.
- Approach: Speed at which the tool will travel during Entry/Exit Motions.
- Engage: Speed at which the tool travels when 50% of the Bit diameter is engaged in the material.
- Cut: The speed at which the tool travels when 100% of the bit Diameter is engaged in the material
- Retract: Indicates how fast the Tool is moving Upwards Z axis toward the clearance plane.
- Departure: The Speed at which the tool moves after it has reach its clearance plane.

Clearance Plane

The Clearance Plane is the height the machine will go to when it is not cutting, this height should be tall enough for the machine to move around without hitting anything on the spoilboard.

1) Always use "Absolute Z" and set it at least .5" more than the thickness of your material.

Cut Parameters

Many of the options in this will remain at their defaults, but you have to look at the stepover control.

1) Skip down to the bottom option Stepover Control. This option controls how the bit steps over in the x-y axis - how much the bit overlaps with each pass in the X-Y axis)
2) Use the Feeds and Speeds sheet to set this value.

Cut Levels

This tab is the main control for our Z axis for this operation. The Stepdown determines how deep the bit will cut into your material on the Z axis. This is similar to the “Rough Depth/Cut” parameter in 2 ½ axis operations. These parameters depend on the bit and the material.

1) Use the Feeds and Speeds sheet to set the Stepdown Control Value
2) Make sure Cut Levels Ordering “Depth First is Selected”
3) Click “Bottom (B)” and make sure that value is 0.
- Setting this to zero ensures that the lowest cut is zero and not a negative Z value.

Engage/Retract

1) Leave this tab unchanged
a) “Engage/ Retract in Material” is Path
b) Check the box that says “Always engage in previously cut area if possible”
c) “Engage/Retract in Air” is Linear Extension (D)

Advanced Cut parameters

1) Leave this tab unchanged
a) All the boxes are unchecked.

2.7 - 3D Parallel Finishing

There are many different ways to Finish your part on a CNC, but most often you will be using Parallel Finishing or Horizontal Finishing. Horizontal Finishing is very similar to the roughing operation but with much smaller “steps”and typically uses a smaller bit to complete the details.

Parallel Finishing achieves the greater detail by running a series of Parallel toolpaths over your part. Like this:

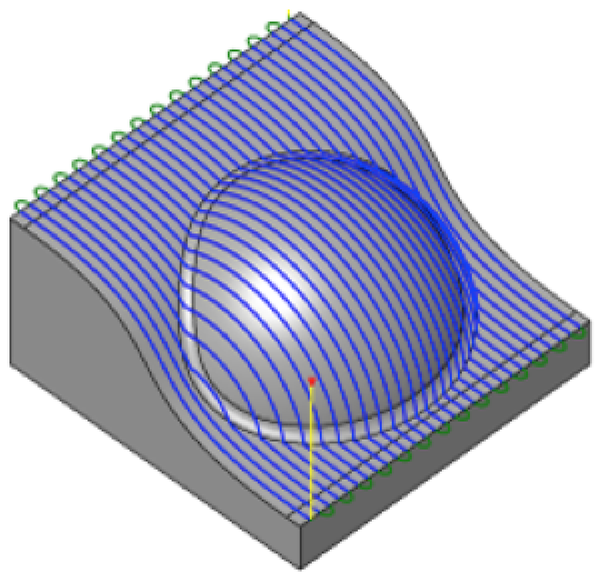


Image credit to AutoDesk

Finishing Operations are done *after* a roughing operation except for rare instances. So this section assumes you have already set up your stock, following the instruction in section 2.6.2

It is important to keep in mind that virtually anything that comes off the CNC will need some post-production work (sanding, dusting, polishing, etc.). Keep in mind that you should balance the Milling time with the post-production time. For example you choose a very small step-over then the milling time will increase, significantly, but you will achieve more detail and reduce the amount of post-processing work. Keep materials in mind, foam and mdf are very easy to sand compared to acrylic, or aluminum, these factors all help determine the proper parameters for your project.

2.7.1 Define Stock

You should have already done this in section 2.6.2, but double check to make sure you have the stock properly represented in RhinoCAM.

2.7.2 RhinoCAM Toolpath Parameters

Machining Features/ Regions

Like any operation, the first tab is the Machining Features/Regions tab.

1) make sure the list is empty by clicking "Remove All" at the bottom of the window. Then click "Select Drive/Containment Regions"
2) This will return you to the Rhino viewport and allow you to select your drive/containment regions. Click the closed curve geometry you want to use as your containment regions.

*** your containment region should be a closed curve, and your surface geometry should be within your containment region.

Note - If you plan on completely cutting your object away from the material, your containment region should should account for the bit and tool cone diameter.

Tool

The tool tab allows you to select the tool for your operations. Select your tool. The laptops should be loaded with the current tool library, but always double check that the tool you select is actually in the slot specified.

1) Finishing operations are meant to achieve greater detail, generally you want to use a bit with a smaller diameter than the one used for Roughing operation.

a) This is where it matters to think how easy it would be to sand the material by hand? Can I save time by sanding it down by hand?

2) Make sure the bit is long enough to cut to your desired height WITHOUT the collet rubbing! The collet rubbing is a definite way of burning your material and possibly starting a fire. Do not base this just off the simulation on RhinoCAM, physically take the bit to your material and see if the collet is going to contact your material.

a) A pocketing operation around the perimeter of your drive region may be necessary if the the tool cone poses a rubbing hazard. The pocketing operation should precede roughing or finishing operations but make sure you don't cut away material where your part is during the pocketing operation. See Pocketing for Tool Cone Clearance (Section 2.8)

**Do not edit the tools, the tool library is managed by School of Art staff.
**If you do not see any tools listed, you may have to import the Artfab Tool library by following the steps outlined in section 2.2

Feeds and Speeds

This is a very important pane, this tab determines the speeds of the spindle as it spins, and moves throughout the operation. These values depend on the material you are cutting, the bit you're using and the specific machining operation. Incorrect feed rates can damage the material, the bits, and even increase the likelihood of material catching on fire.

****Use the spreadsheet in the binder to determine the feeds and speeds for your cuts.
****Always double check the values you put in, it's very easy to accidentally add an extra digit.

Clearance Plane

The Clearance Plane is the height the machine will go to when it is not cutting, this height should be tall enough for the machine to move around without hitting anything on the spoilboard.

1) Always use “Absolute Z” and set it at least .5” more than the thickness of your material.

Cut Parameters

1) Go down to the “Stepover Control” section of the tab. This controls how the bit “steps over” in the X and Y axis.
2) Use the Feeds and Speeds sheet that contains the Parallel Finishing operation to input this value.

Z-Containment

Parallel finishing operations there is no “Step-Down” value. This is because RhinoCAM determines the Z-axis movements from the surface of your model.
However, this tab gives you some options and controls over the z axis movement.

1) Make sure that the “Specify Lowest Z” is not a negative value. Even if you are cutting an object completely out of the material, this value should never be a negative Z value.

Entry/Exit

1) Leave this tab unchanged.
a) The Entry and Exit tabs are set to Linear with a vertical distance of 0.025.
b) The Cut Connections is set to Straight.

2.8 Pocketing for Tool Cone Clearance

This is a helpful tip when your bit is not long enough to finish your cut. This section will explain how to create a “valley” around the perimeter of your part that will allow for the tool cone, and your bit to cut deeper into your material without the tool cone rubbing on the material.

1. Offset your drive region by at least 1.25” in Rhino.

2. Select the Pocketing operation on the RhinoCam Machining Operations Browser.

3. Select your original drive region curve, and the new offset curve as your Drive Containment Regions in the Pocketing Toolpath Parameters

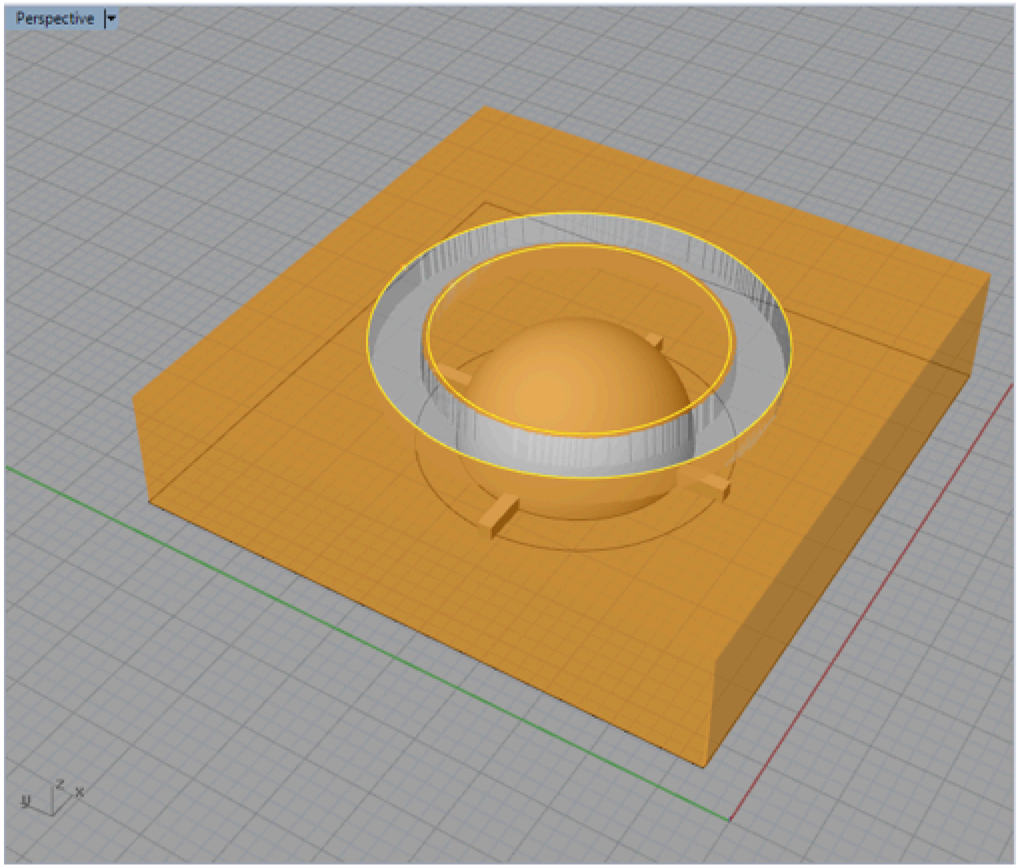
4. Set this up as you would any other 2.5 pocketing operation - outlined in section 2.4 of this handbook - except for the Cut Levels tab.

5. On the Cut Levels tab, the “Total Cut Depth” should be set to the depth you need for your 3D operation to be completed.

A. I.E. if my parallel finishing operation used a bit with 1 inch of cutting depth but my material was 1.5” I would set up my Total Cut Depth to at least .75”

6. Make sure this Pocketing Operation is performed before the Roughing or Finishing operations. Make sure this is first in your Setup Folder in the RhinoCAM machining Operations tab.

Visualization of what this does:



2.9 Planing the Spoilboard

Monitors might be asked to resurface the spoilboard or to plane a new spoilboard after it has been installed. There is a pre-programmed command on WinCNC that has the appropriate G-Code to complete the operation but there is a little set up required.

1. Make sure the Fly-cutting bit is in **T11**.
2. Make sure the spoilboard is **calibrated** using the steps outlined in section 1.3 of this handbook.
3. **Measure** the fly cutting tool.
4. Turn on the **dust collector**.
4. Click the "**Fly Cut**" button on WinCNC and follow the on screen instructions.
5. After the operation is done, check to see if the board is flat, it should be noticeable if the board is flat by the evidence of even cutting marks.
6. If the board is still not flat, you need to recalibrate the machine to the new top of the spoilboard and run the operation again until the board is flat.

3.0 Troubleshooting

3.1 Safely aborting a job

Even when all the safety precautions have been taken and proper steps followed, things can still go wrong and accidents can happen. Knowing what to do when something goes wrong and being prepared to handle the situation will help minimize the damage or danger.

- Keep a close eye on the machine before it begins to cut your material. Did it pick up the right tool? Is it making any strange sounds? Is the stock vibrating or moving on the spoilboard?

- Catching any mistakes before the machine cuts into your material allows you to make changes without damaging your stock.

If something goes wrong during your cut and the bit is already engaging your material these are the steps you should take:

1) Pause the job by pressing Spacebar
2) Abort the job by pressing the Escape Button
3) Type in G28Z - this will home the Z axis.
4) Re-Home the machine

Note: Pausing the machine doesn't stop the bit! The pause button just halts the machine on the current line of your .NC file.

Note: Do not park the machine if material is on the spoilboard.

If your material moves while the machine was cutting out your object, safely abort the operation. Do not try to reposition you material and rerun the same NC file. It not safe and it won't be a successful cut.

It may be the case where you might need to start over again with entirely new stock.

3.2 Soft Limit Errors

Software limits (Soft Limits) are programmed limits on the machine that prevent the machine from going beyond a certain point in the XYZ axis. There are also "hard" limits which are the physical limitations on how far the machine can travel.

Soft Limit errors will only show up on WinCNC and **not** on RhinoCAM.

If you are receiving a Soft Limit Error during your operation it means that your .NC file is attempting to make the machine go beyond its software limits.

The Soft Limits on our machine are:

X axis low limit = -1.45
X axis high limit = 49.5

Y axis low limit = -1.7
Y axis high limit = 56.45 (This is longer than the bed so that it can pick up the tools)

Z axis low limit = -0.02
z axis high limit depends on the length of the bit

1. If you are getting a Soft Limit Error, double check your toolpaths and rhinocam parameters to make sure you are not going beyond any of the limits.
2. It helps to do a search (control + F) on your .NC file to double check before you post your files to the blue cabinet computer.

Warning: If you receive a Soft Limit error while the bit is plunged into material, the spindle will not shut off automatically and will keep running unless you do one of the following:

1. Press Page Up on the keyboard to raise the bit out of the material.
2. Type T0, to return the tool.
3. Press the Spindle button on WinCNC to stop the spindle.

3.3 Incorrect machine coordinates

If the machine and the WinCNC coordinates do not line up, i.e. the machine is at x10y10 over the spoilboard but WinCNC is saying something very off, then the machine coordinates need to be reset.

1) Re-Home the Machine by clicking the HOME button or typing G28 on the WinCNC command line.

3.4 Manual Release of Tool Cone

If something goes wrong and you cannot put the tool back with the regular g-code commands there is a way of manually releasing a tool from the machine.

Do not bypass any of the safety interlocks or perform a manual tool release with the spindle on. Serious injury can occur, make sure the spindle is off.

1) On Command line type M1 and press enter. Or hit the Manual button.
2) Hold onto the tool cone and press the green button on the left side of the spindle.
3) Once the button is pressed the draw bar will release the tool cone with 60 psi of air pressure.
4) On the command line type L110T0 and press enter. This will reset the current tool register to NO TOOL.

3.5 E-Stop after Shutdown

1) Reset E-Stop pull switch, if E-stop cable was used.
a) to reset, press blue button on the back right side of router.
b) If red mushroom button was used, turn the key.
2) Turn all switches to the off position
3) Press green Start Button
4) Turn on Computer, Boot up windows
5) Start CNC Controller
a) turn on Master Switch

b) Turn on Enable switch
6) Once all switches are on, Home machine.

NOTE-

The e-stop cable will stretch over time. On the switch there is a glass window, with a line scribed in the middle. Behind the window is a yellow triangle. The point of the triangle needs to be pointed at the scribe mark. Adjust the turnbuckle until the point meets the scribe mark.

The switch will be actuated when pulled, or when the tension has been released, i.e. cut.

3.6 Setting of Tool Measure Deeper

Manual Setting of M37

1) Home Z axis - G28Z
2) pick up the tool you want to adjust deeper
3) calibrate the tool of the switch like normal
4) jog machine until the bit is not over the table, so off to either side of the spoilboard
5) type G0Z0 to have the Z head go to the top of the spoilboard but off the table
6) Jog the bit in the down direction to the location of how much deeper you wish for it to go to into the spoil board. Depending on the depth, you may need to turn of soft limits.
7) Type M37T?Z0 (T is for the tool cone number)

a) there will be a blue box to the right of the Z axis coordinate value on Z. There will also be a G43 above the axis coordinate window. The Z coordinate will change to zero. This value will be recorded internally and will not change until the next time the tool bit has been changed or the tool has been re-measured.

4.0 Emergency Response

4.1 Fire

5.0 Maintenance

Following the maintenance schedule described in this section will increase the life of your machine and reduce down-time due to breakdown or malfunction. These are guidelines only. These guidelines assume that the machine is being operated approximately 40 hours per week.

Main Power and Circuit Breakers to the machine **MUST BE TURNED OFF** during all maintenance.

As required:

1. Empty Dust Collection

2. The gear rack drive system on the 1000 & 950 Series Machines has an oil/air mixture blow off tube at the pinion gear. Make sure that a proper flow is maintained to insure the gear rack is kept clear of material. Check rack for particles that may become lodged in teeth and remove with a stiff brush.

3. Blow out and clean the intake filters on the pressurized electronics box and computer control cabinet. Dirty air filters will reduce the flow of air to the components in the cabinet and may shorten their useful life.

4. Most high Speed Routers come with sealed bearing and require no lubrication for maintenance. For routers that require lubrication refer to the manufacturer's maintenance guide for details. Routers with brushes would be checked for wear and rotated 180 degrees to insure proper operation and longer life. Cooling fans should be kept clean and clear of obstruction. Periodic checks to confirm proper calibration and orientation of your router to the work surface should be performed if the router runs into fixturing or any other obstructions by mistake.

Weekly:

1. Clean and wipe down the machine while making a visual inspection. This will allow the operator to become familiar with the mechanics while maintaining the machine.

2. Check the oil level in both bearing oiler lubricators. The oil is a light 10 weight air tool oil.

3. Backup all your programs and files.

Semi - Annually:

1. Remove the endplates on the two Y axis transmission boxes. Using compressed air, blow out any dust and debris. (You must wear eye protection during this)
2. Check all timing belts for adequate tension and wear. The belts should be tight but not so tight that it takes pressure to achieve deflection.

3. Check Belt tension on all axis.

Manufacturer Contact:

CNT Motion Systems, INC
(412) 244-5770
(412) 244-5760 Fax
website: www.cntmotion.com
email: cnttech@cntmotion.com

6.0 Glossary / CNC Terminology

CLIMB CUT (AKA DOWNCUT): In a climb cut, the cutter motion is downriver, opposite its rotation—the cutter rotates clockwise but is fed counter-clockwise and away from the rotation. The resulting cut can produce a finer surface edge but climb cutting is extremely hard on bits and can be easily overdone. Climb cuts should only be made with extreme caution and in special situations.

CNC: Computer Numerically Controlled

COLLET: The holder that locks cutting tools into the spindle of the CNC router.

CONVENTIONAL CUT (AKA UPCUT): In a conventional cut, the cutter rotation and tool motion are in parody, the cutter is fed upstream into the material, producing a robust cut but with a tendency to tearout. Conventional cutting is the standard cutting direction and should be used in most situations.

CUTTING EDGE: The sharpened edge of the tool used for cutting.

DOWNCUT BIT: The flutes and cutting edge of a downcutting bit are spiraled downward pushing chips and grain down into the material.

FEED RATE: The rate at which the cutter moves from point A to point B. Expressed in inches/ minute.

FLUTE: The grooves cut into the tool to allow chip removal during cutting.

FIXTURE: A template or clamping device which helps to either locate or affix your material to the machine. If, for example, a student is milling many parts from identical sized stock, it would make the process go faster to affix a location template on the spoil board so each new piece of stock would not need to be re-zeroed and could instead use the same origin point as the last milled piece.

GANTRY: The horizontal support member of the CNC, on which the Spindle is mounted. The gantry moves along the X-axis. The router moves laterally along the gantry to describe the Y-axis.

G-CODE: A system of numeric code and machine instructions. When you Post a file, Rhinocam will produce G-Code. Once the file is posted a plain text file will open in a new notepad window. This is the G-Code. Do not alter this code! You will notice the file type is .nc, this stands for numeric code.

GENERATE: The command in Rhinocam which tells Rhinocam to interpret the information you have provided in the MOp and develop toolpaths.

HOME: An absolute coordinate position programmed into the CNC machine. When you send the machine Home you are sending it to this absolute coordinate position. If you are running many files with a consistent origin, it is useful to know where that origin is in relation to the Home coordinates. In the event of a CNC shutdown or some other loss of your origin coordinates, the coordinates can always be reproduced from this Home position.

KNOWLEDGE BASE: A group of basic tool-paths with speeds and other parameters already defined. Most often, when a student begins developing a toolpath they should start with one of the toolpaths found in the Knowledge Base file so that some options are already established.

MACHINING OPERATIONS (MOPS): The term used to describe the tool path and settings created for machining.

ORIGIN: In Rhino, this is the world 0,0,0 coordinate. On the CNC machine you establish this point when you zero out the bit on the material. When you do this you must zero out all three coordinates. Once you have established this origin, the CNC will interpret that origin to be the position of the 0,0,0 origin in your Rino file.

POST (OR POSTPROCESS): The process by which Rhinocam converts tool paths into numeric code which are interpretable by the CNC machine.**28**

POST PROCESSOR: In the Rhinocam setup tab, there is an option to define the Post processor. When you define this you are telling Rhinocam what kind of CNC machine you are using. Every CNC speaks a slightly different form of numeric code (.nc) so it is necessary to tell Rhinocam what kind of machine it is going to be communicating with so that it posts the .nc file in the correct format. In SFS, we only have two machines, TechoIsel and Onsrud TM3. If your post processor is setup for any other machine, your file will not run correctly.

REGION: The area in Rhinocam you use to designate a milling area. Generally, regions are closed curves on or around the geometry you wish to mill. Think of a region as a way to tell Rhinocam where to look for milling geometry.

SPINDLE: The rotating motor that spins the cutting tool.

SPOIL BOARD: Sheet of underlying material, beneath your working stock, which serves as both a place to affix your stock as well as a sacrificial cutting surface. While it is allowable to set your toolpaths such that they cut into the spoil board, be sure they do not cut through it!

STOCK: The material that will be machined on the CNC Router.

STRAIGHT FLUTE: Straight flute bits are used in applications where the material may flex or lift during cutting, when for instance the material is thin and light.

SUPPORT MATERIAL: The material outside of the part you are milling. When you are milling, there must generally be some sort of support material in addition to the part being milled so as to secure the parts to the machine as they are being milled.

TOOL: The rotary cutter used in the CNC machine to cut the material. Tools are made from a variety of materials and in a wide range of sizes and geometry.

TOOLPATH: A definition of the motion of the tool as it makes a given cut. In Rhinocam, the toolpath itself is expressed by a light green line. Each green line represents one linear motion of the tool and defines the center location of the bit at any one time.

UPCUT BIT: The flutes and cutting edge are spiraled upward pulling chips and grain up out of the material.