SAS CNC MILL

SCIENTIFIC ART STUDIO'S INTRODUCTORY GUIDE TO SUBTRACTIVE MANUFACTURING

SAM LOMBARDI NOTES - SUMMER 2017

1 RHINO AND THE PART

1.1 Set up the File

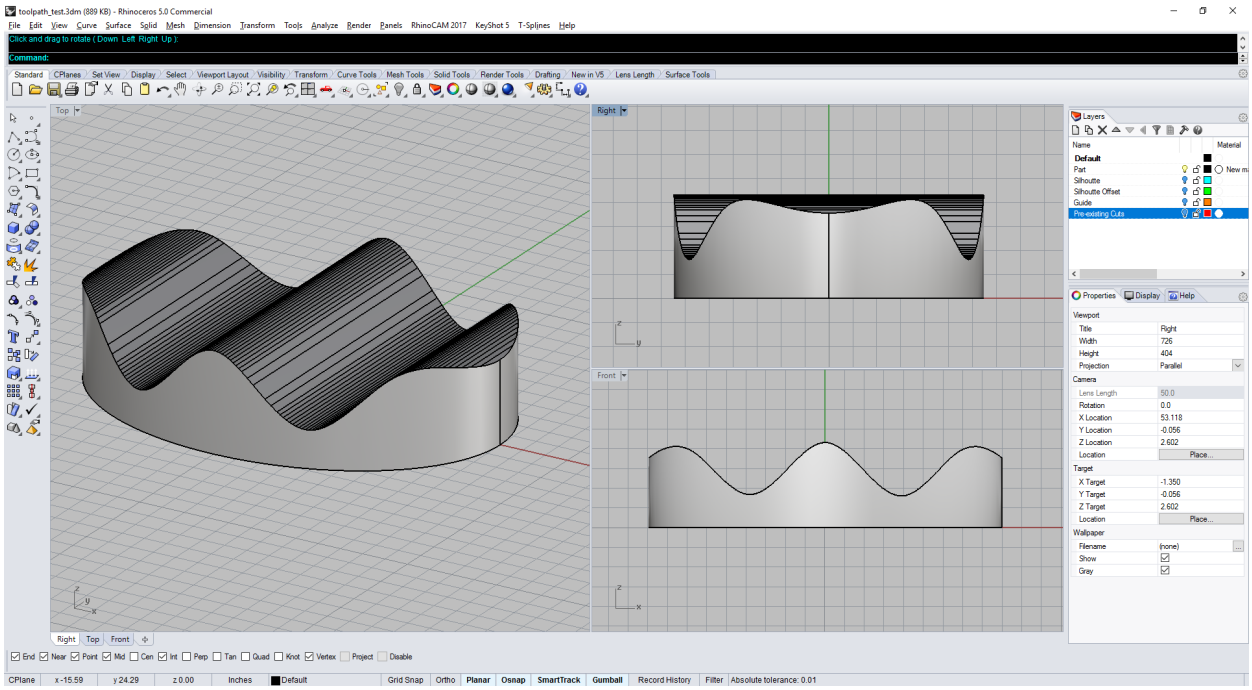
Note: Record the size and useable space of the stock, the size of the part you wish to model, and the detail you wish to model it with. All this information will be crucial throughout all steps. Multi-stock parts that cannot be digitally split, must be milled in two files, and therefore will have some different milling criteria, read all notes.

If the part is coming from Vectorworks, it should be exported as a .3ds or .3dm

Note: The part may need to be split into multiple layers to avoid overhang or size restrictions. This should be done in Vectorworks if possible, as an imported file may not be editable in Rhino.

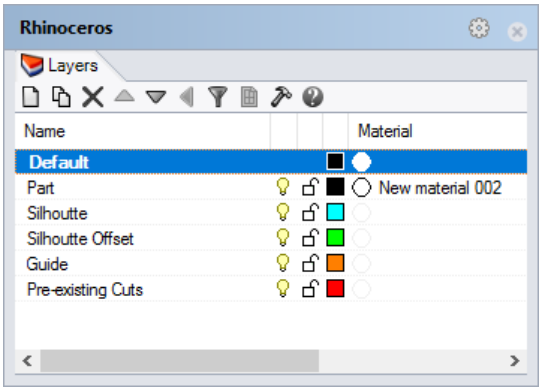
Open Rhino and change the Units, both model and layout, to that of the Vectorworks file.
Use Insert to scale and place the part into Rhino.

Change both Rhino Units to inches.
Move the part so that it is sitting on the Z, 0 plane.



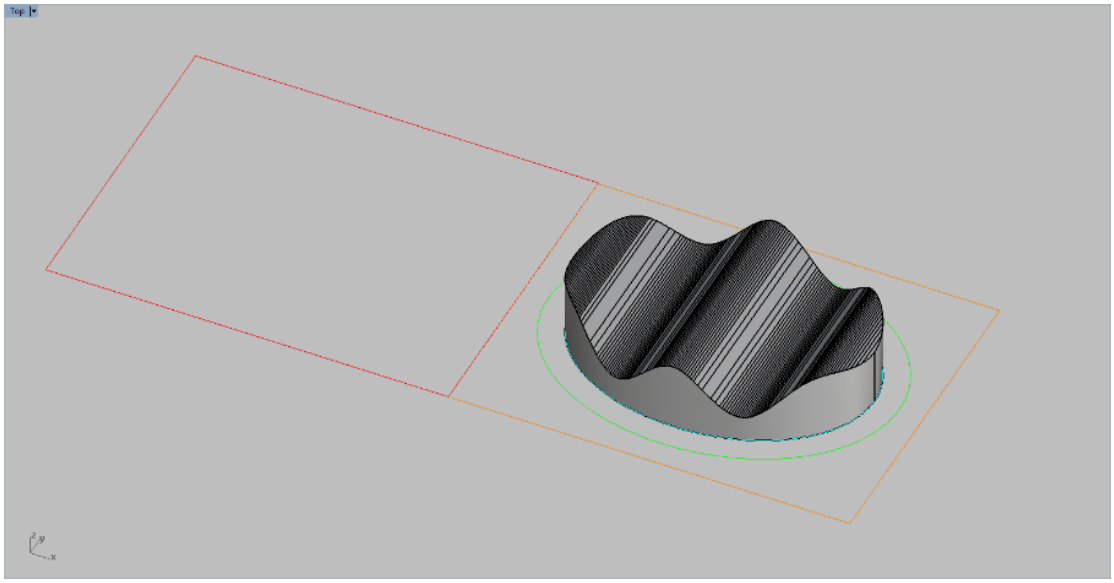
1.2 Create Drive Regions and Mill Guide

Using the layers in rhino, create each of the following shapes in this section on different layers.

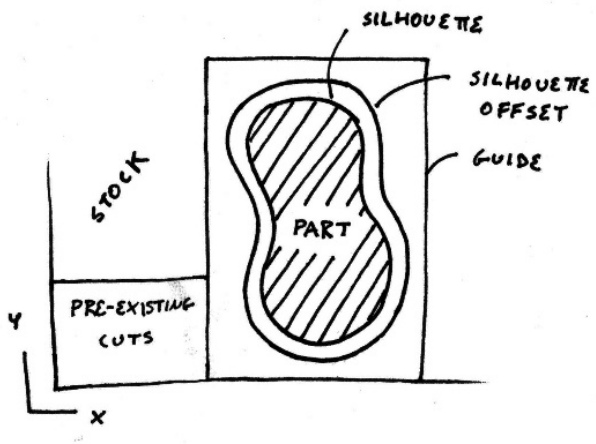


Use the Silhouette to project part outline onto the Z,0 plane.
Offset that curve by 1½" and keep the original.
Note: The drive region will be area the CNC will mill. This offset outline can also be made with any geometry, as it may be easier for a part with a simple footprint.

Draw a Rectangle around the part on the Z, 0 plane, leaving 1 ½" around the part's drive region.
Note: This mill guide will be used to align the part appropriately within the stock. The 1 ½" is the distance from the edge of the stock, take any screws or edge damage into account.



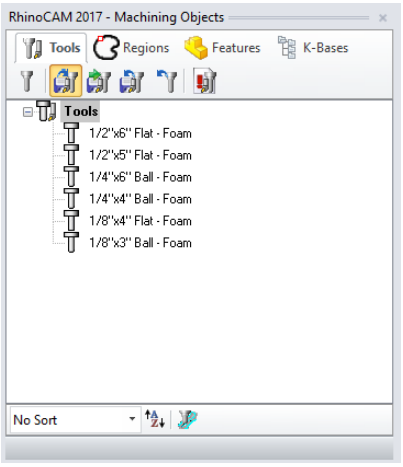
Note: Also add any geometry, like additional rectangles, to account for areas that have already been cut. This is only needed if the part will not be located at the origin of the stock, rather offset.



2 RHINOCAM AND THE TOOL PATHS

2.1 Define the Machine and Post-Processor

Open the RhinoCAM 2017 Plugin.



In the Machining Objects window, click Load Saved Tool Library.
Open *SAS CNC Tools.vkb*.

Post > Current Post Processor: Mach 3-Inch
File extension: .nc

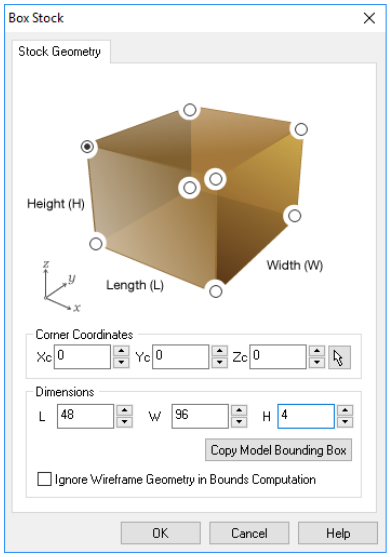
Machine > Number of Axis: 3 Axis

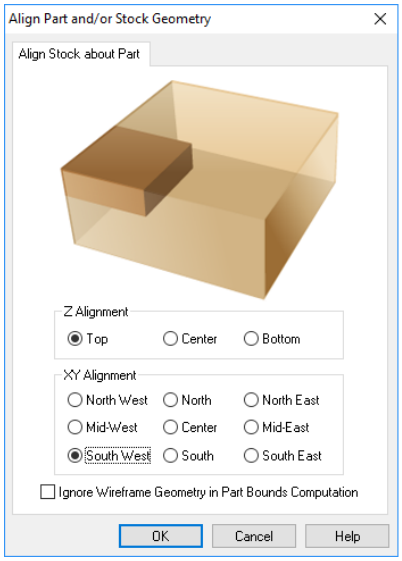
2.2 Define the Machining Setup

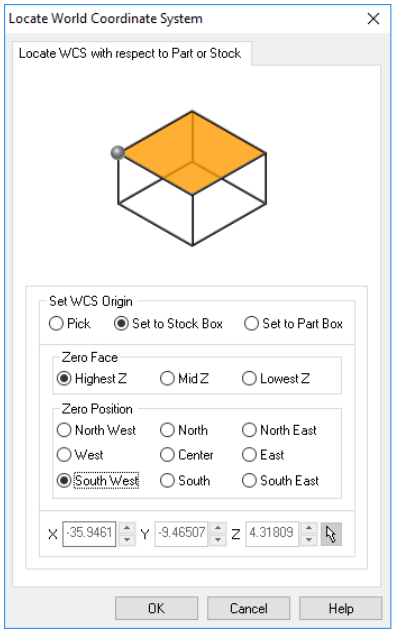
Stock > Box Stock
Click the top left corner of the stock.
Set Corner Coordinates: Xc: 0, Yc: 0, Zc: 0
Set Dimensions: L: 48, W: 96, H: 2 or 4
Note: H is dependent of foam thickness.

Align > Align Stock
Set Z Alignment accordingly.
XY Alignment: South West
Note: Ensure that the rectangular border from earlier has created 1 ½" margin from stock edge.
Note: If only one part is to be cut in the stock, just Center the XY Alignment.

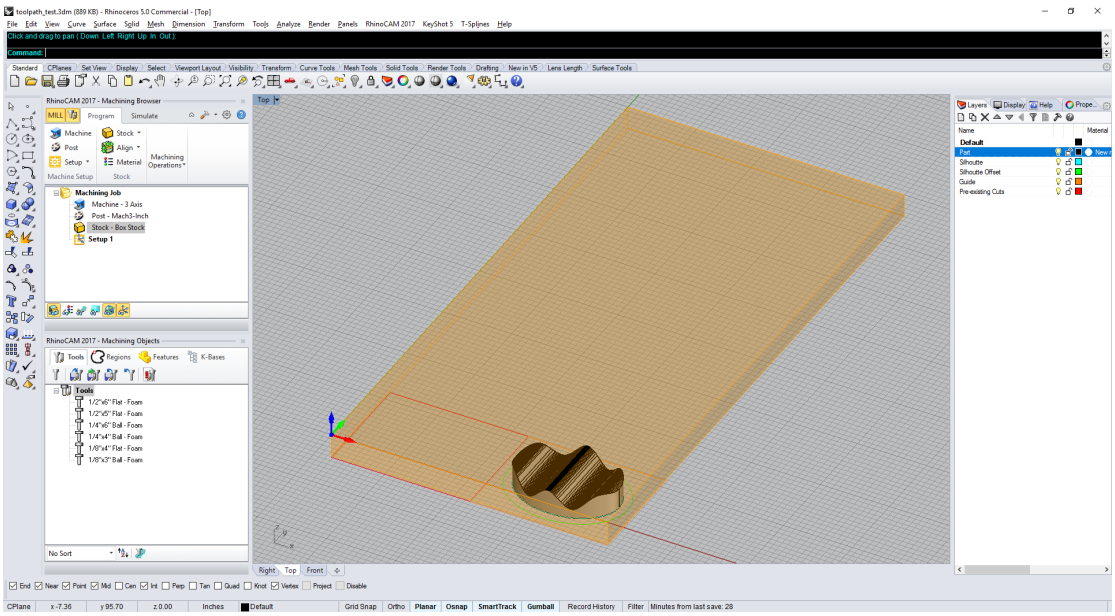
Align > Set World C.S
Click Set to Stock Box.
Zero Face: Highest Z
Zero Position: Southwest Corner
Note: This is the zero point for the machine.







Note: For a muli-stock part, that has not been split: Change the Zc for the Box Stock. Don't Align Stock, Move the part manually. Keep the World C.S. as shown above.



2.3 Set Tools Paths

Note: Keep in mind the stock setup and height, the longer bits may not fit over the stock, and the shorter bits may not reach the bottom. For example, the 6” bit will not fit over 8” of foam.
Note: The following are a set of example tool paths, change and add paths as appropriate. For multi-stock parts, Cut Levels and the Clearance Plane will be non-default.

3 Axis > Horizontal Roughing

Machining Features/Regions > Click Select Drive/Containment Regions > Select the *Offset Silhouette* curve from earlier.
Tool > Select one of the ½” flat bits.
Feeds and Speeds > Click Load from Tool.
Clearance Plane > Click Automatic and Clearance Plane.
Cut Parameters > % Tool Dia.: 40
Cut Levels > % Tool Dia.: 50
Click Generate.

Note: This will rough out the part and a trench around it. Make sure the bit does not cut all the way through the stock on this pass.

3 Axis > Parallel Finishing

Machining Features/Regions > Click Select Drive/Containment Regions > Select the *Silhouette* curve from earlier.
Tool > Select one of the ¼” ball bits
Cut Parameters > Angle of Cuts: 45, % Tool Dia.: 15
Click Generate.

Note: Change the Tool Dia. for a more precise finish. You can also add similar passes to get a better finish.

2 1/2 Axis > Parallel Finishing

Machining Features/Regions > Click Select Drive/Containment Regions > Select the *Silhouette* curve.
Tool > Select one of the ½” flat bits.
Cut Parameters > Cut Start Side: Use Outside/Inside for Closed Curves > Outside
Cut Levels > Location of Cut Geometry > At Bottom, Set Total Cut Depth appropriately
Advanced Cut Parameters > Bridges/Tabs > Rectangular
Click Generate.

Note: This pass creates the tabs and cuts out the part. It is a risky pass, and is prone to failure. Go over every option carefully and consider if this operation is better than freeing your part by hand with a knife. Or modeling tabs manually.

Note: Simulate and Regenerate all the paths after every change. Go through every tab and check the settings. Most things can be left as the defaults, but double check.

2.4 Generate Shop Documentation and Send to Post

Right Click each of the machining jobs > Post.
Save the Files and close Notepad.
Note: These are the documents that will be loaded into Mach3 in the CNC room.

Right Click Setup 1 > Shop Documents.
Output Template: SAS Template
Save/Print the Report.

Note: Save these files for use in shop and future reference. Take notes as the part is milled, record problems and solutions and write them in the notes section. Save these documents in the binder located in the CNC room.

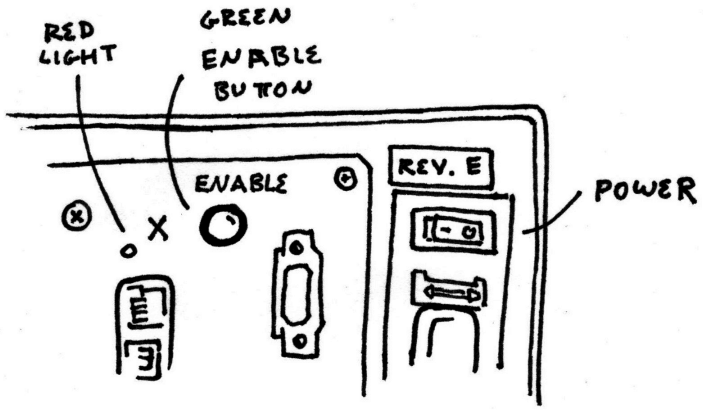
3 MACH3 AND THE K2 CNC MILL

3.2 Turn on and start the CNC

Note: Clear all dust, and screw the stock to the lower foam, orienting it as programed.

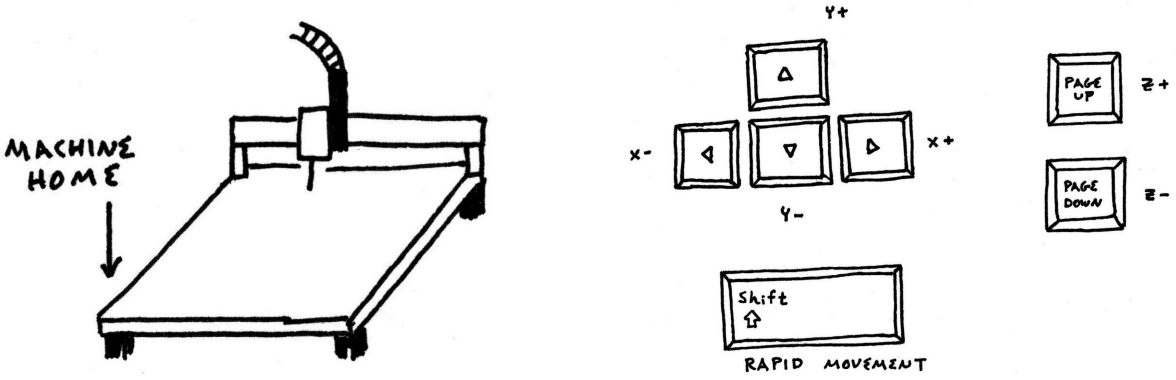
Turn on the Computer and open Mach3.
Open the profile KG12062_10.
Click Reset.

Turn on the Control Box, and press the green Enable button.



Note: The red light on the back on the Control Box should turn off. The CNC is now on, and any further movements should be cautious. Wear safety glasses and eye protection, and avoid loose clothing.

Move the Router Head to the Machine Home (The front left corner).



Note: Use the arrow keys and page up and down to move the Head. Holding Shift will increase the speed of the movements. Try not to move the head to the edge of the track, the arms will violently smack into the edge, and the Enable button on the back of the Control Box will have to be hit again.

Attach the first bit.
Note: Refer to the tool references in section 4. Confirm the Shoulder Length matches the listed tool.
Note: The first time home is set for a part, consider using the smallest tool, for a more accurate home.

Lower the head below the maximum height.
Click Reference all Home.
Note: X,Y,Z,4 buttons should all turn green.
Move the bit tip to the work zero position. This should be the front, top left corner of the stock.
Note: Press tab to bring up the MPG menu to adjust the movement distance per keypress.

Click Zero for X,Y,Z,4. All should be set to zero at this point.

3.3 Load the Code and Start to Mill
Place the .nc file on the desktop.
Click Load File, and load the first tool path.

Turn on the router.
Click Cycle Start.

Note: As soon as Cycle Start is clicked, get ready to emergency stop the CNC. A problem could occur immediately.

3.5 Change the Tool
Turn off the router.
Remove the first bit and attach the second bit.
Note: Refer to the tool references in section 4. Confirm the Shoulder Length matches the listed tool.

Re-zero the mill with the new bit.
Click Zero for Z. Only the Z axis should be changed.
Note: Pressing Goto Zero does NOT know the clearance plane of your part, and can damage the part and stock, use accordingly.
Note: Ref All Home, as you may have nudged the Head while replacing the tool.

Load the new code.
Turn on the router.
Click Cycle Start.

3.6 Using the Camera to Remotely Watch
Open IP Camera Tool.
Double click: SAS01 Http://192.168.1.130:88 M

ID: SASadmin
PW: topsecret

Click Server Push Mode.

3.7 Resume the Tool Path After a Stop

Re-Ref All Home and Re-Zero the mill if zero was not saved.
Find and select a line just before the mill stopped.
Note: A visual representation of current code line can be seen on the tool path simulator to the right, highlighted by a extremely small white line.

Move the Head to that general area, and click Run From Here.
When Mach3 has located the position and pseudo-ran the code to that line, it will stop.
Click Cycle Start.

3.8 Removing the Part and Shutting Down

Move the Head to left, back of the table.
Turn off the Control Box.
Close Mach3.
Note: Click Yes to Fixture Save? to save the zero position from this session. Before using the zero again, make sure to Ref All Home, regardless if you are Zeroing at that time.

Remove the part, return bits, and clean the dust.

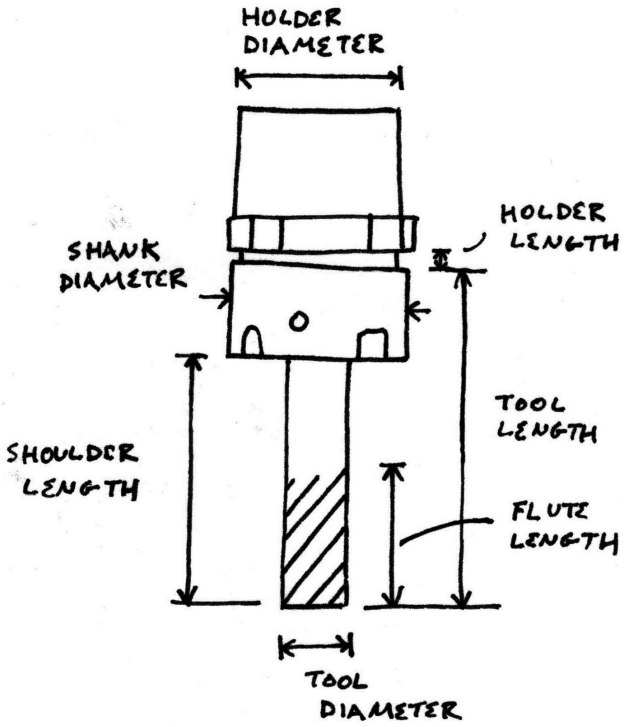
4 REFERENCE & NOTES

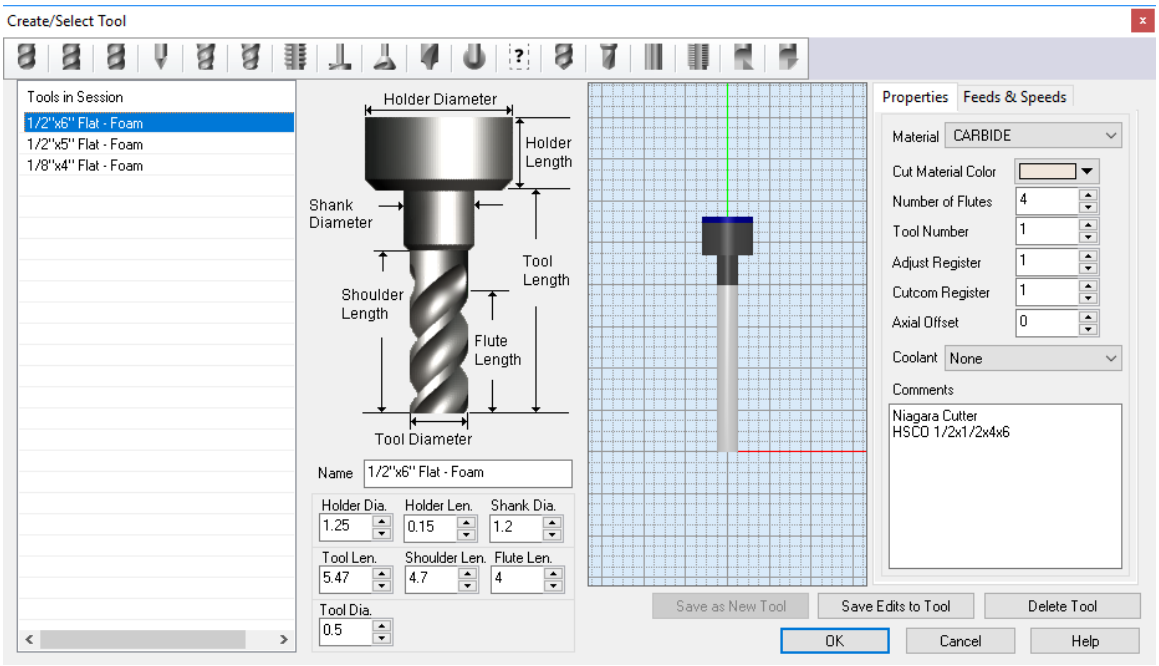
This guide should serve as a step by step introduction for going from 3D model to milled part. It will explain how to use Rhino to orient the part, RhinoCAM 2017 to code the tool paths for the part, and Mach3 with the CNC mill to fabricate the part. It is written with the Scientific Art Studio, and its specific workflow, software, and mill in mind. This is not a complete user manual, and should not be used as such.

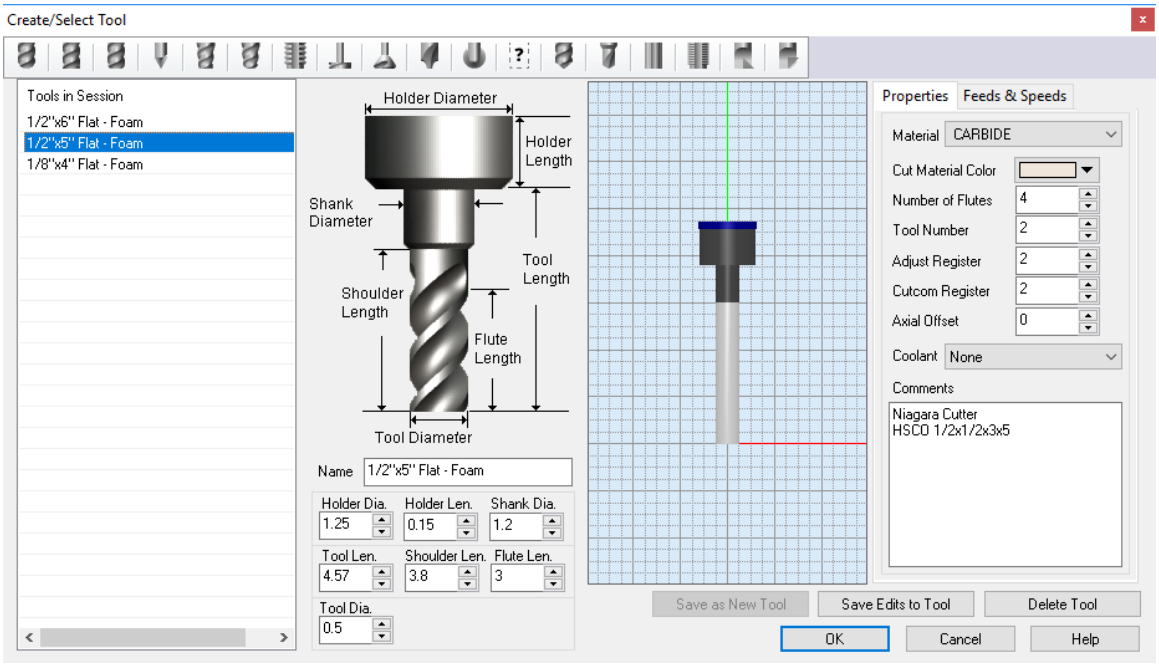
Software:
- Rhino
- RhinoCAM
- Mach3
- IP Camera Tool

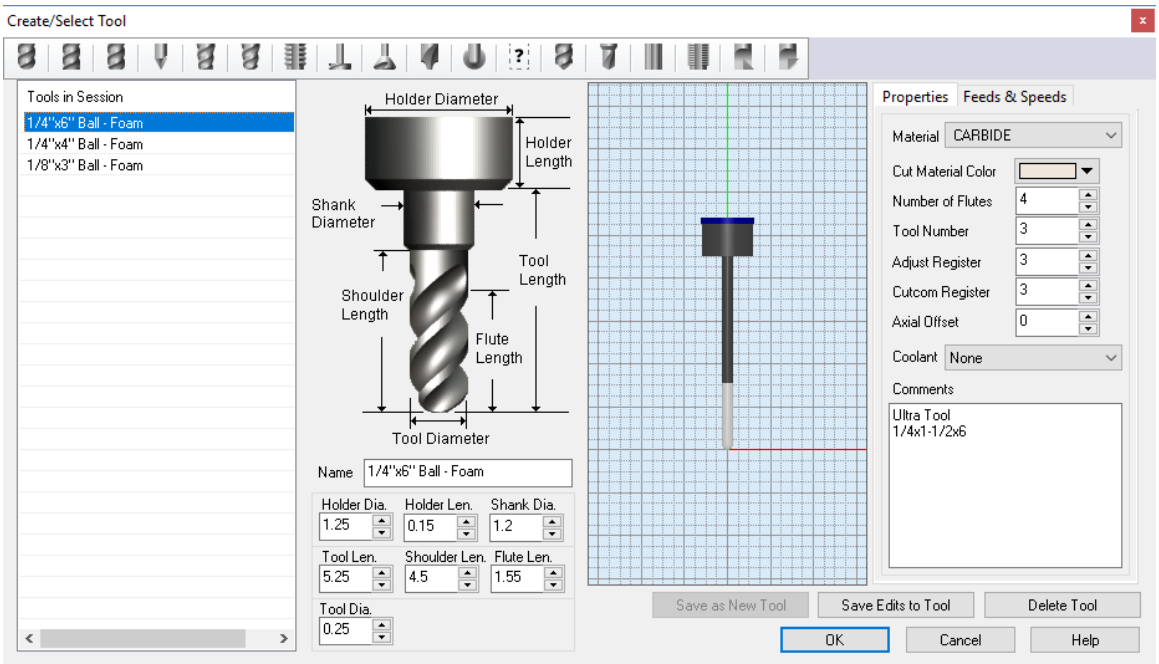
Hardware:
- K2 CNC Mill

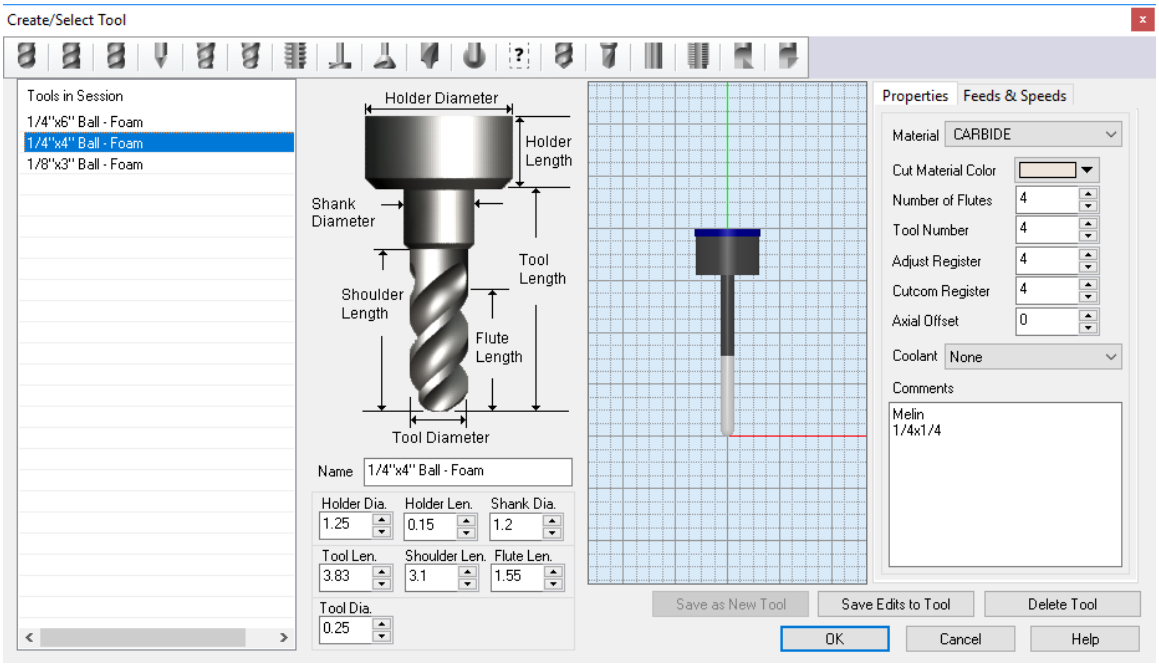
Tech Support:
- MecSoft: 949.654.8163

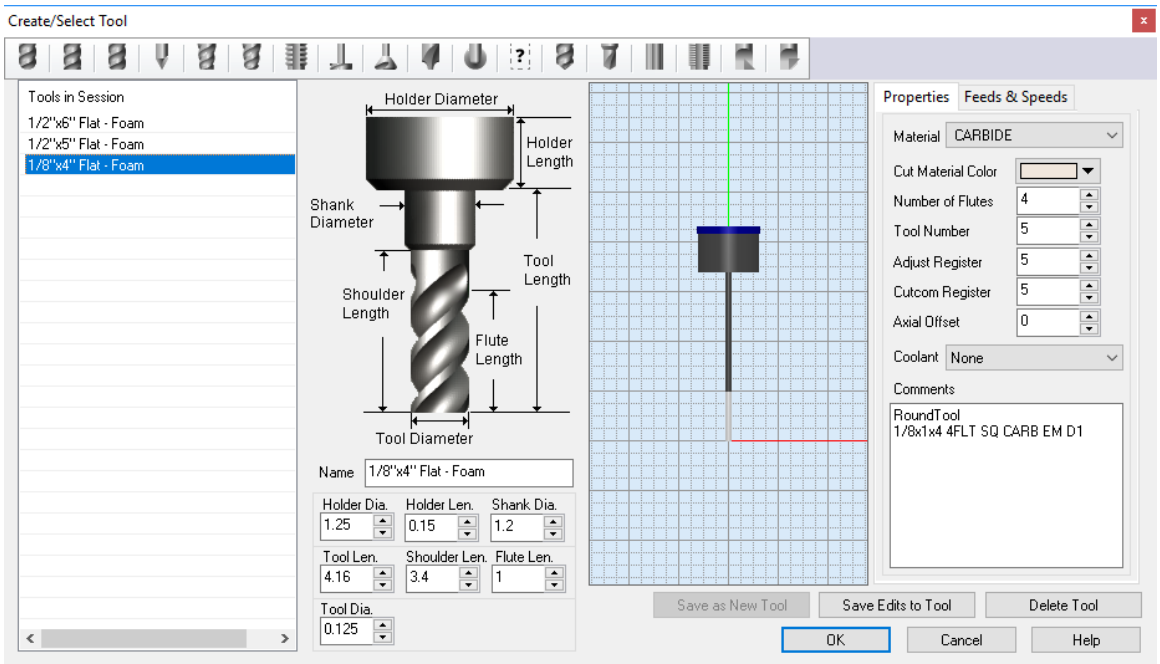


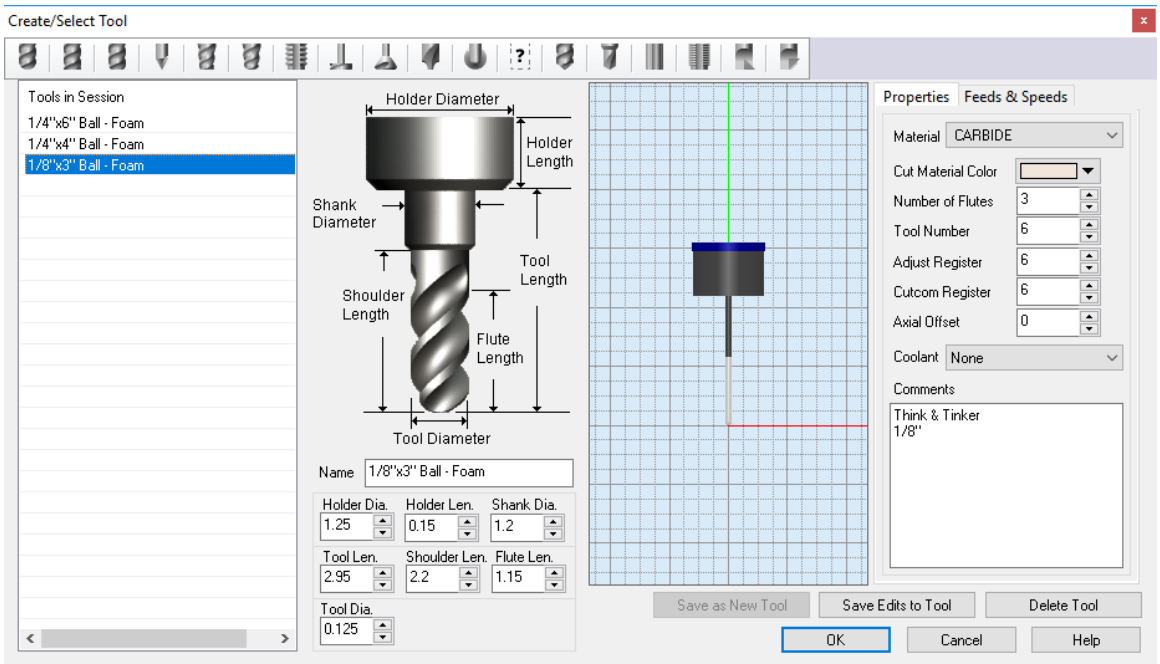

Create/Select Tool
Tools in Session
1/2"x6" Flat - Foam
1/2"x5" Flat - Foam
1/8"x4" Flat - Foam
Holder Diameter
Holder Length
Shank Diameter
Tool Length
Shoulder Length
Flute Length
Tool Diameter
Name
1/2"x6" Flat - Foam
Holder Dia.
1.25
Holder Len.
0.15
Shank Dia.
1.2
Tool Len.
5.47
Shoulder Len.
4.7
Flute Len.
4
Tool Dia.
0.5
Properties
Feeds & Speeds
Material
CARBIDE
Cut Material Color
Number of Flutes
4
Tool Number
1
Adjust Register
1
Cutcom Register
1
Axial Offset
0
Coolant
None
Comments
Niagara Cutter
HSCO 1/2x1/2x4x6
Save as New Tool
Save Edits to Tool
Delete Tool
OK
Cancel
Help


Create/Select Tool
Tools in Session
1/2"x6" Flat - Foam
1/2"x5" Flat - Foam
1/8"x4" Flat - Foam
Holder Diameter
Holder Length
Shank Diameter
Tool Length
Shoulder Length
Flute Length
Tool Diameter
Name
1/2"x5" Flat - Foam
Holder Dia.
1.25
Holder Len.
0.15
Shank Dia.
1.2
Tool Len.
4.57
Shoulder Len.
3.8
Flute Len.
3
Tool Dia.
0.5
Properties
Feeds & Speeds
Material
CARBIDE
Cut Material Color
Number of Flutes
4
Tool Number
2
Adjust Register
2
Cutcom Register
2
Axial Offset
0
Coolant
None
Comments
Niagara Cutter
HSCO 1/2x1/2x3x5
Save as New Tool
Save Edits to Tool
Delete Tool
OK
Cancel
Help


Create/Select Tool
Tools in Session
1/4"x6" Ball - Foam
1/4"x4" Ball - Foam
1/8"x3" Ball - Foam
Holder Diameter
Holder Length
Shank Diameter
Tool Length
Shoulder Length
Flute Length
Tool Diameter
Name
1/4"x6" Ball - Foam
Holder Dia.
1.25
Holder Len.
0.15
Shank Dia.
1.2
Tool Len.
5.25
Shoulder Len.
4.5
Flute Len.
1.55
Tool Dia.
0.25
Properties
Feeds & Speeds
Material
CARBIDE
Cut Material Color
Number of Flutes
4
Tool Number
3
Adjust Register
3
Cutcom Register
3
Axial Offset
0
Coolant
None
Comments
Ultra Tool
1/4x1-1/2x6
Save as New Tool
Save Edits to Tool
Delete Tool
OK
Cancel
Help


Create/Select Tool
Tools in Session
1/4"x6" Ball - Foam
1/4"x4" Ball - Foam
1/8"x3" Ball - Foam
Holder Diameter
Holder Length
Shank Diameter
Tool Length
Shoulder Length
Flute Length
Tool Diameter
Name
1/4"x4" Ball - Foam
Holder Dia.
1.25
Holder Len.
0.15
Shank Dia.
1.2
Tool Len.
3.83
Shoulder Len.
3.1
Flute Len.
1.55
Tool Dia.
0.25
Properties
Feeds & Speeds
Material
CARBIDE
Cut Material Color
Number of Flutes
4
Tool Number
4
Adjust Register
4
Cutcom Register
4
Axial Offset
0
Coolant
None
Comments
Melin
1/4x1/4
Save as New Tool
Save Edits to Tool
Delete Tool
OK
Cancel
Help


Create/Select Tool
Tools in Session
1/2"x6" Flat - Foam
1/2"x5" Flat - Foam
1/8"x4" Flat - Foam
Holder Diameter
Holder Length
Shank Diameter
Tool Length
Shoulder Length
Flute Length
Tool Diameter
Name
1/8"x4" Flat - Foam
Holder Dia.
1.25
Holder Len.
0.15
Shank Dia.
1.2
Tool Len.
4.16
Shoulder Len.
3.4
Flute Len.
1
Tool Dia.
0.125
Properties
Feeds & Speeds
Material
CARBIDE
Cut Material Color
Number of Flutes
4
Tool Number
5
Adjust Register
5
Cutcom Register
5
Axial Offset
0
Coolant
None
Comments
RoundTool
1/8x1x4 4FLT SQ CARB EM D1
Save as New Tool
Save Edits to Tool
Delete Tool
OK
Cancel
Help


Create/Select Tool
Tools in Session
1/4"x6" Ball - Foam
1/4"x4" Ball - Foam
1/8"x3" Ball - Foam
Holder Diameter
Holder Length
Shank Diameter
Tool Length
Shoulder Length
Flute Length
Tool Diameter
Name
1/8"x3" Ball - Foam
Holder Dia.
1.25
Holder Len.
0.15
Shank Dia.
1.2
Tool Len.
2.95
Shoulder Len.
2.2
Flute Len.
1.15
Tool Dia.
0.125
Properties
Feeds & Speeds
Material
CARBIDE
Cut Material Color
Number of Flutes
3
Tool Number
6
Adjust Register
6
Cutcom Register
6
Axial Offset
0
Coolant
None
Comments
Think & Tinker
1/8"
Save as New Tool
Save Edits to Tool
Delete Tool
OK
Cancel
Help

NOTES