

Nomad 3 CNC Quick Walkthrough Booklet

Vers: Feb 2025



THE UNIVERSITY OF BRITISH COLUMBIA

School of Biomedical Engineering

Faculties of Applied Science and Medicine



Table of Contents

Part File Design Tips: 3

File Preparation:..... 5

CNC Setup and Zeroing: 9

Cutting Execution:..... 12

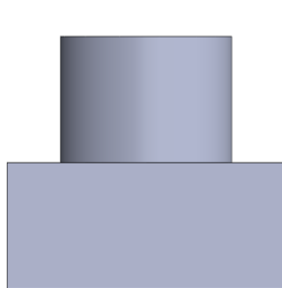
Using the Flip Jig: 12

Notes: 20

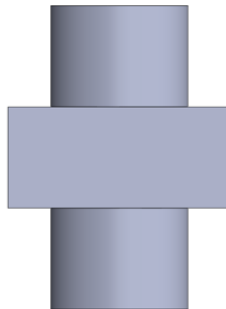
Nomad 3 CNC Training Manual

Part File Design Tips:

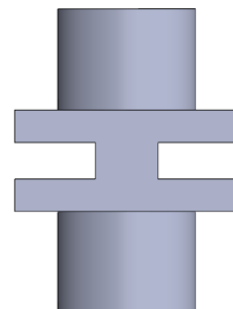
The Nomad 3 CNC is a 3-axis CNC with an optional flip jig attachment. As a 3-axis machine, the end mill (cutting bit), always approaches directly downwards towards the object surface. As such, all surfaces to be machined must be visible from a top-down view of the object. Any features not visible from this view cannot be machined on this CNC (unless they can be seen from a bottom-up view and the flip jig is used). While it may sometimes be possible to re-position the stock material in the device to allow for various operations to occur from multiple approach angles (i.e. one cut operation from top-down and then rotating the stock on to its side for another cut operation), ensuring proper alignment of the stock for the relative positions of the features to be accurate is difficult and should be avoided when possible.



Possible Without Flip Jig



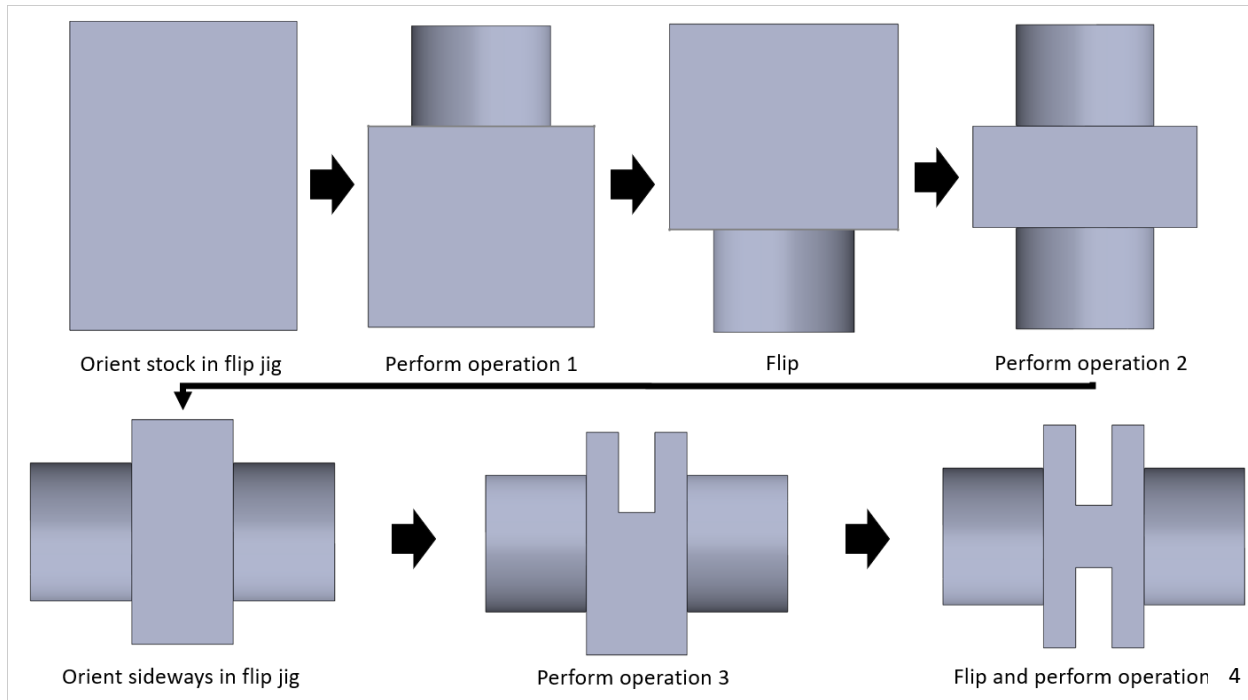
Possible With Flip Jig



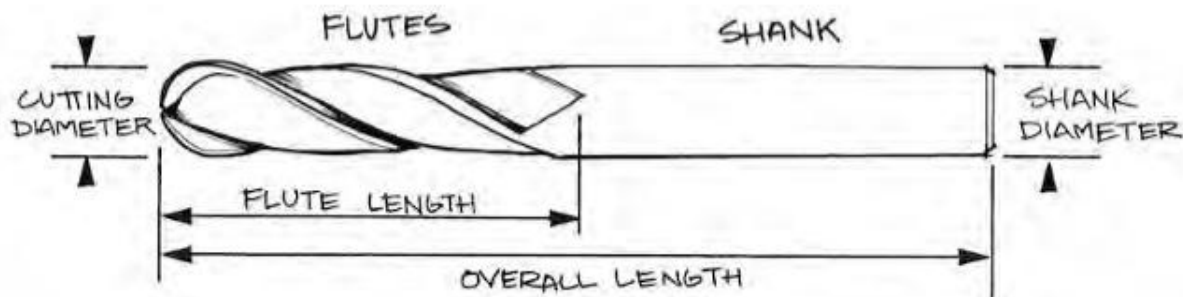
Not Possible With Flip Jig,
Would Require Second Flip
Operation with object
placed on side

With the 3rd scenario, the difficulty comes from the alignment of the cutting operation within the stock when moving to the side-view features. If performing CNCing this way, ensure to model the part with a distinct feature left or created from the top and bottom cut operations that can be used as an accurate zeroing point for the side cut operations. Otherwise, the side cut features may be some distance away from their intended position relative to the top and bottom features. To perform the side cuts accurately, you may need to edit the stock material dimensions to match the unmachined section of the stock. **As this is a unique and difficult**

application of the CNC, please see the Makerspace Technician for consultation.

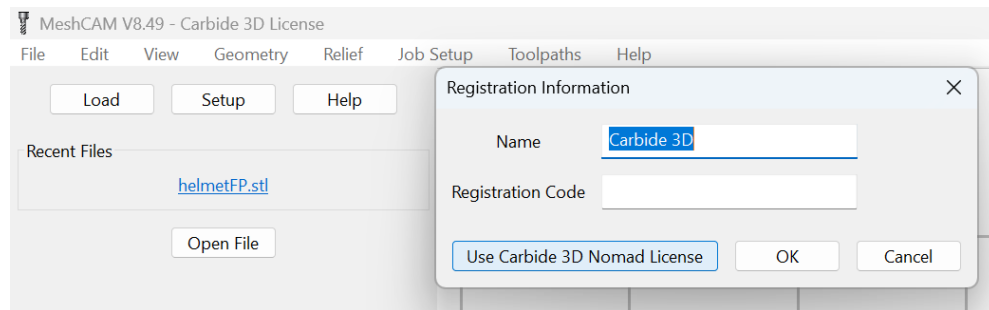


When creating parts to be CNCed, you must take into account the features of the end mills to be used. End mills have specific radii and depths of cut which can be achieved. Be sure to use an end mills with a radius less than the radius you intend to cut into your part (a 1in radius bit cannot cut a $\frac{1}{4}$ in radius internal corner). This also applies to depth of cuts. An end mill with a depth of cut of 0.5in cannot cut a 1in step into a part (consider ramping the material down to prevent sharp edges limiting max cutting depth). Generally, you should cut into the material only as far as your flute length, though some extra length may be possible depending on the cutting and shank diameters. If the shank and cutting diameters are the same, you may experience burnishing if cutting at a depth passed the flutes.

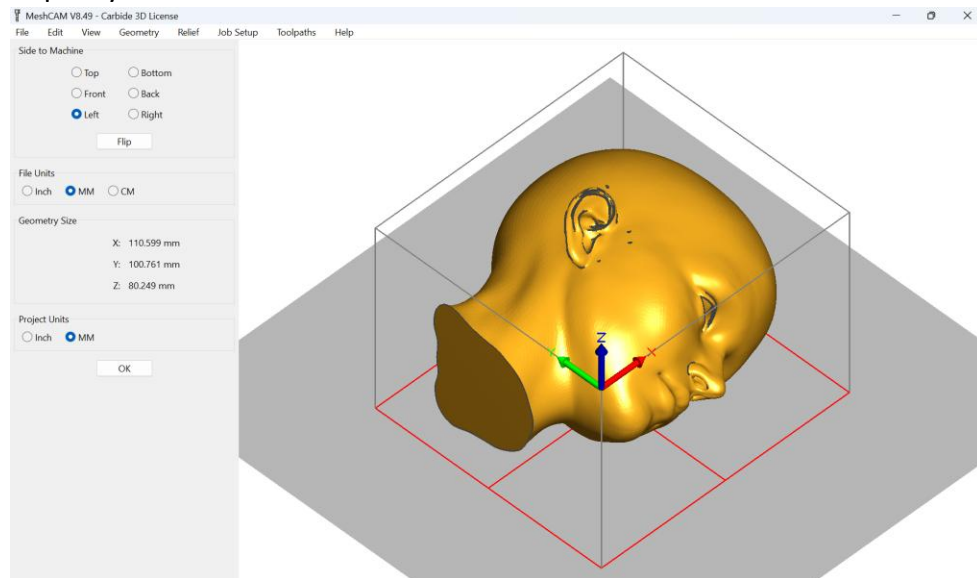


File Preparation:

1. Begin by designing your part in your CAD software of choice, such as SolidWorks. Once complete, export your part as an STL and import it into MeshCAM V8 (V8 allows for free use with the Nomad 3 CNC, V9 requires a subscription).
2. Open MeshCAM v8 and select → Help → Register → Use Carbide 3D Nomad License.

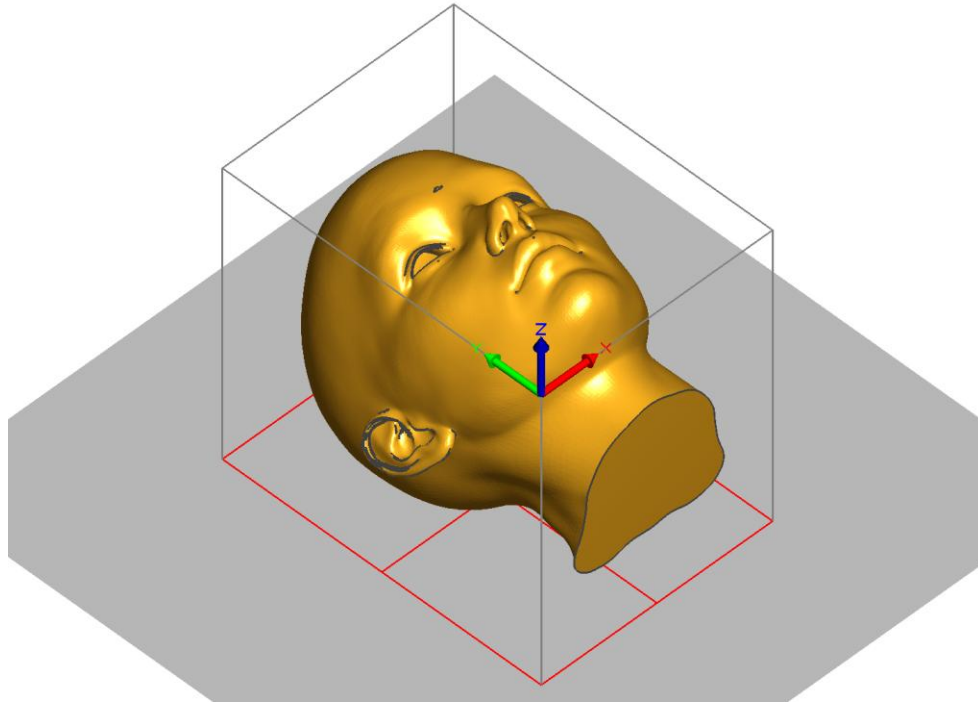


3. Import your STL into MeshCAM V8.



4. Choose which face points upward (towards the top of the CNC) and choose the project and file units.

5. Reorient the object if needed along the XYZ axes.



6. Choose your stock material and size, program zero (use the top face, bottom left corner for squared stocks) and define any supports (if needed).
- a. If not machining the full stock, set your max depth as well.
 - b. The position of the object in the stock (touching the top, bottom, etc) can also be edited in the “Stock” window.

SBME Jim McEwen Zone for Innovators, creators and
Entrepreneurs Training

7. Choose your job settings; job type (3-axis), tolerance, machine whole stock.

Part Setup

Rotate / Scale	Edit
Material	Wood - Soft
Stock	Edit
Program Zero	(SW Corner, Top Z)
Max Depth	100.7612 mm
Machine Regions	Edit
Supports	Edit

Job Setup

Job Type	3-Axis
Tolerance:	0.0250 mm
Area to Machine	Whole Stock

8. Under “setup” select or create the end mills you will use.

Select Tool

Tool Number	Tool Type	Diameter	Corner Radius	Flute Length	Taper Angle
1	Ball Mill	0.125 in		0.750 in	
102	End Mill	0.125 in		0.500 in	

Add

Delete

Copy

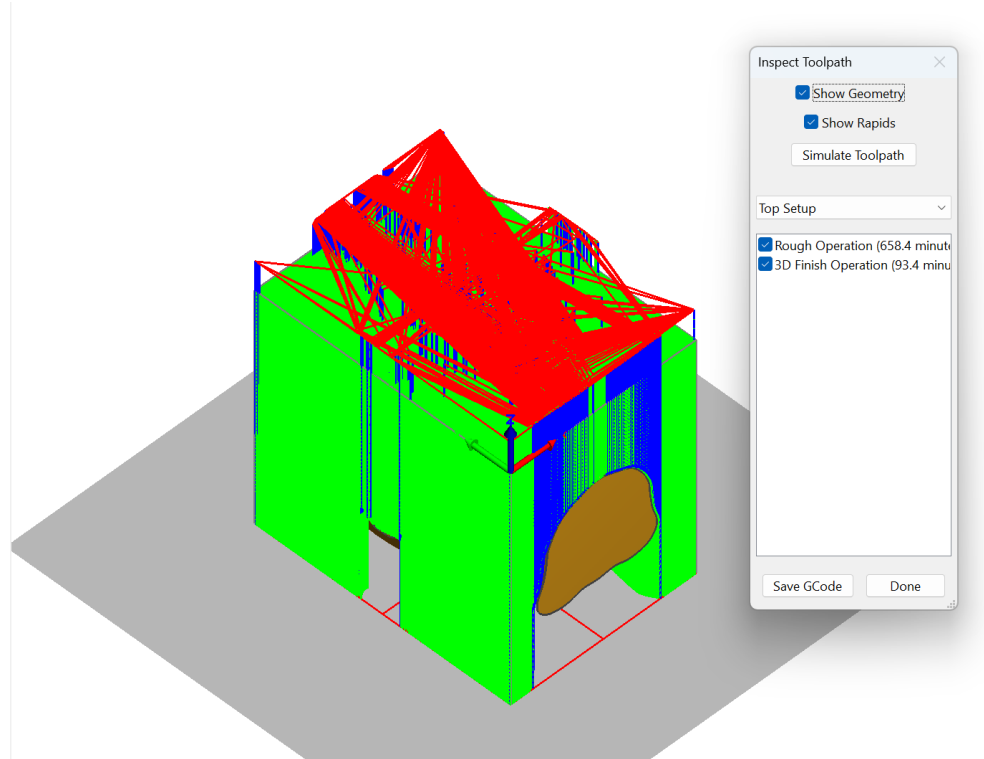
Edit

Done

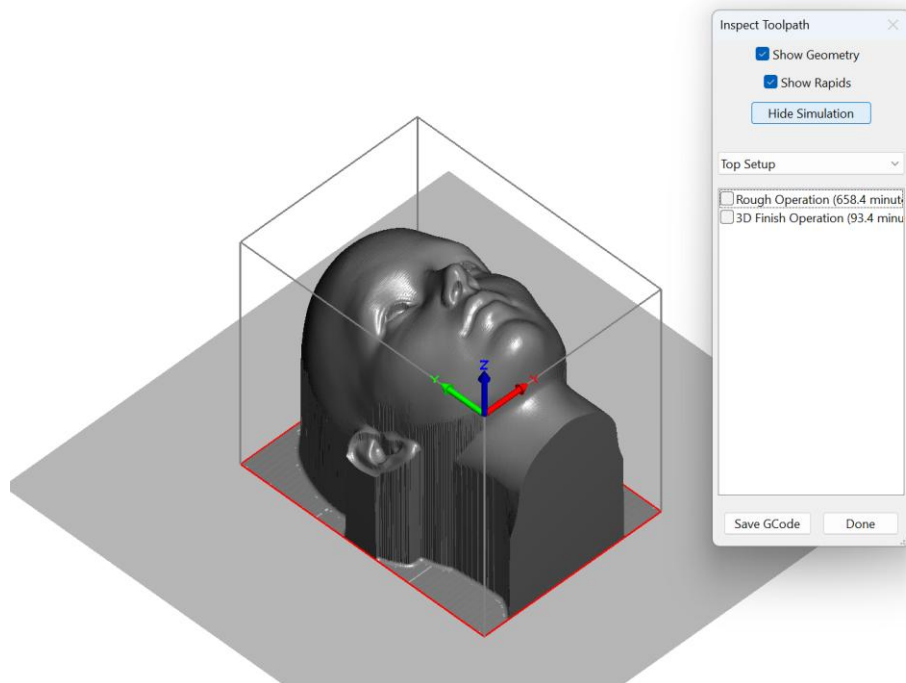
9. Select toolpaths → Roughing, select the correct tool and choose done.

10. Select toolpaths → Unified Finishing, select the correct tool and choose done.

11. Select Calculate Toolpaths.



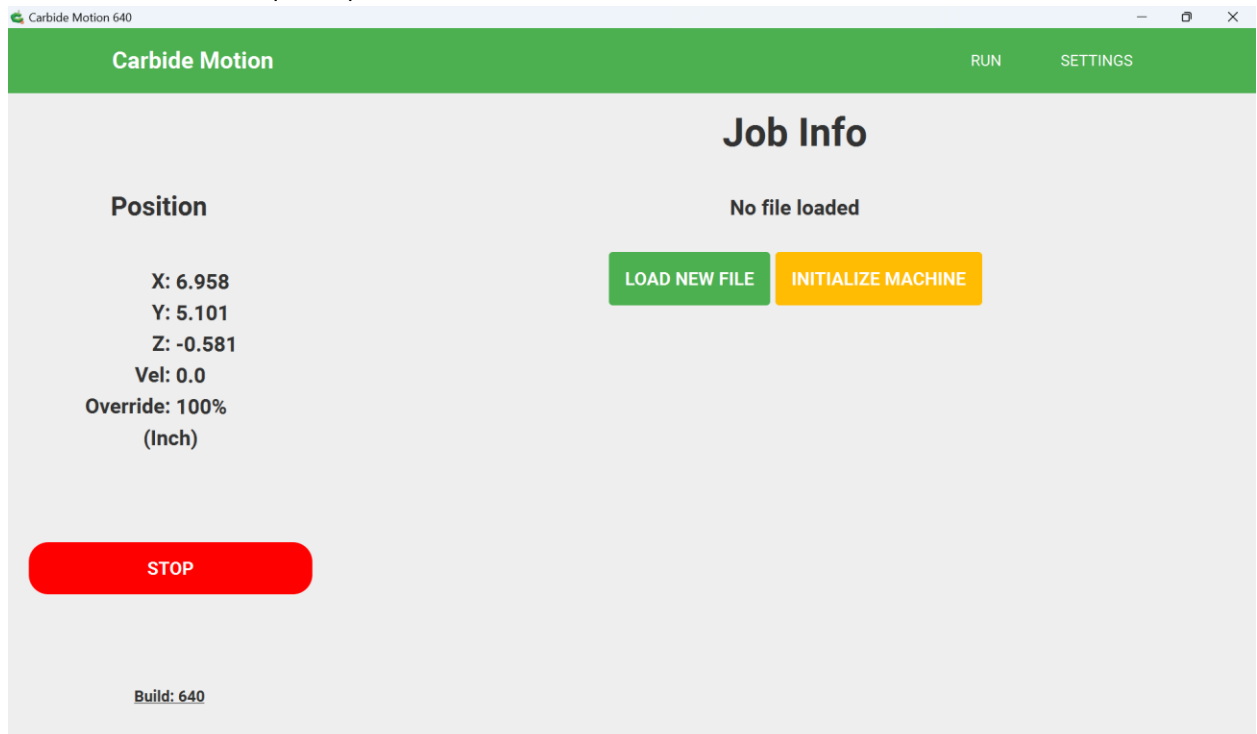
12. Simulate the toolpath and deselect the rough and finishing paths to view simulated result.



13. Once confirmed, reselect the toolpaths and save the GCode.

CNC Setup and Zeroing:

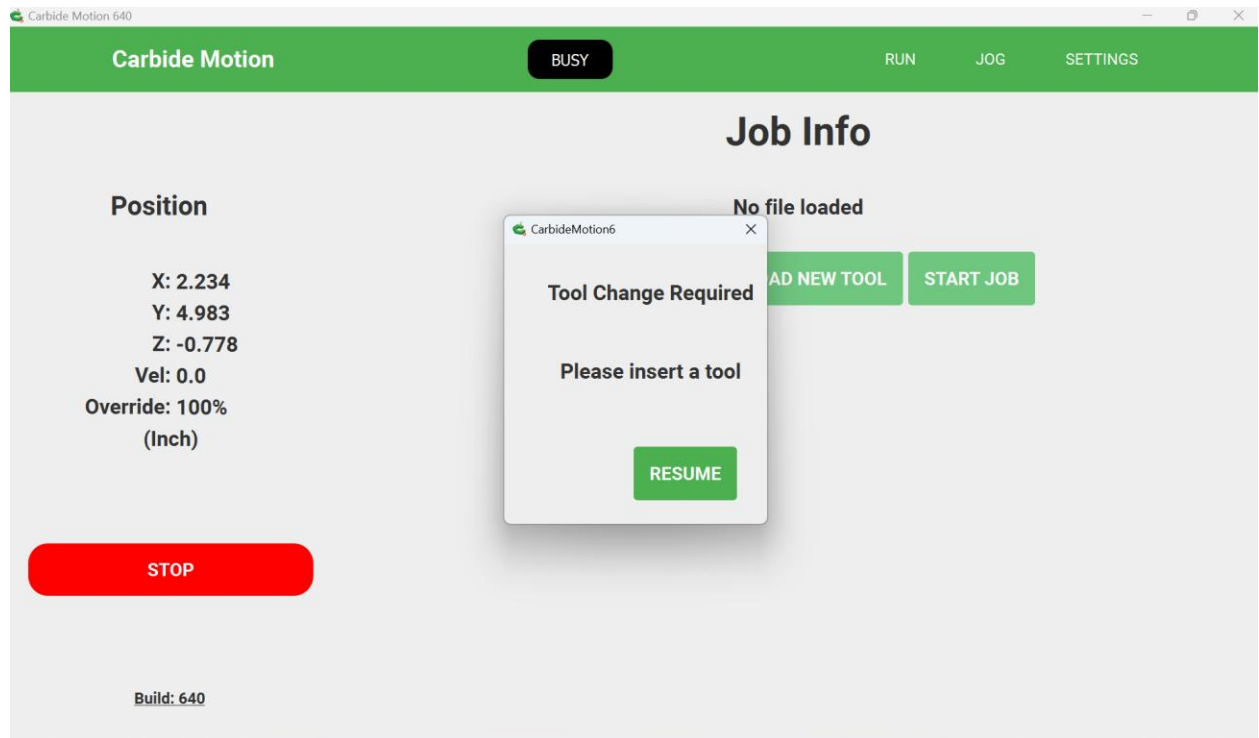
1. Insert the stock into the Nomad. Ensure the directional axes align with how it was defined in meshCAM (Y is forward/back and Z is up/down). Attach the stock to a waste board or other baseplate material.
 - a. If cutting to the bottom of the stock, use a wasteboard beneath it to prevent damage to the metal baseplates.
 - b. Ensure the stock is securely attached, use a combination of double sided tape, stops, and clamps.
2. Connect to Carbide Motion and the CNC.
3. Initialize the machine (home).



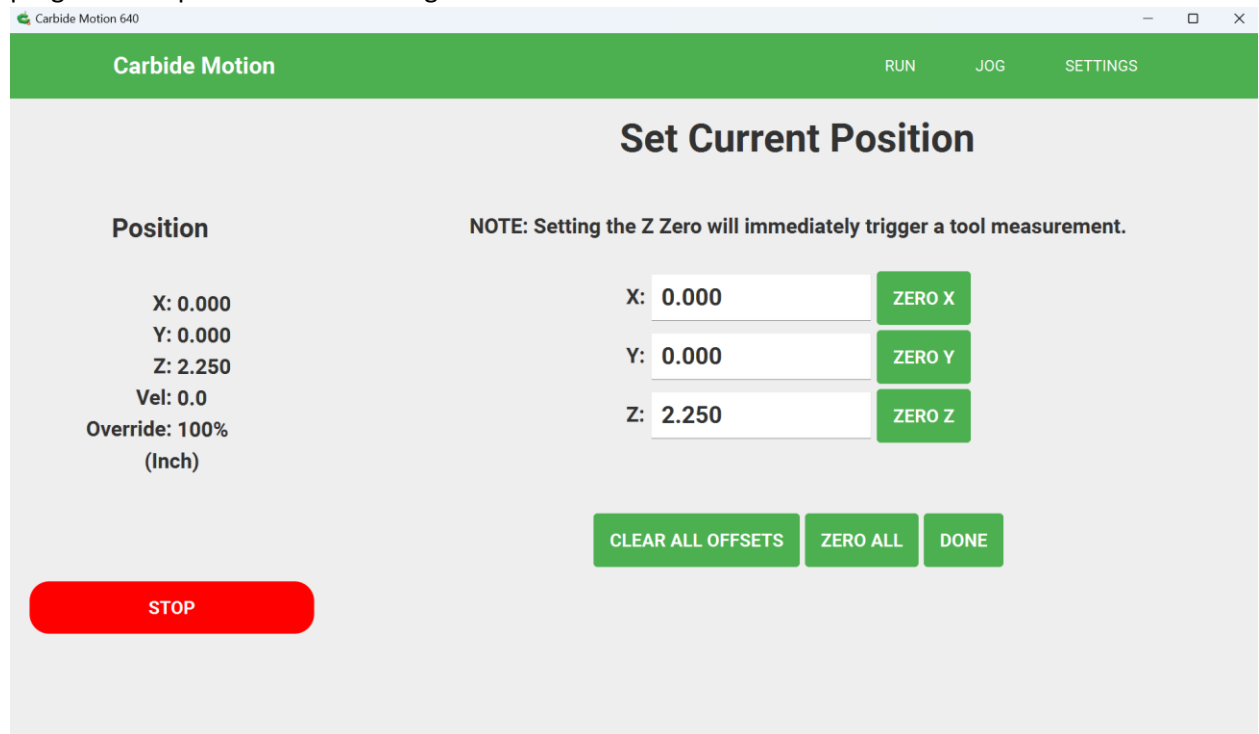
4. Choose Load New Tool and place the end mill to be used into the collet chuck. Ensure enough of the bit is in the collet for rigidity and enough is left out for the depth of cut needed. Never place

SBME Jim McEwen Zone for Innovators, creators and
Entrepreneurs Training

flutes in the collet.



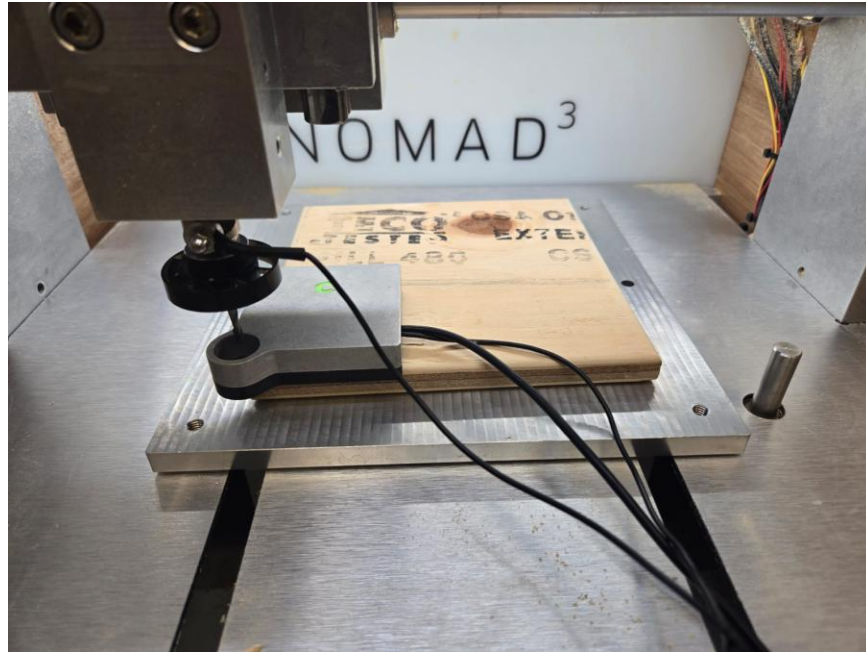
5. Set the CNC zero based on the defined program zero set earlier by moving the bit to the program zero position and selecting "zero all".



- a. If the chosen corner of the stock is square and the top is flat, the BitZero V2 can be used to zero the machine accurately.

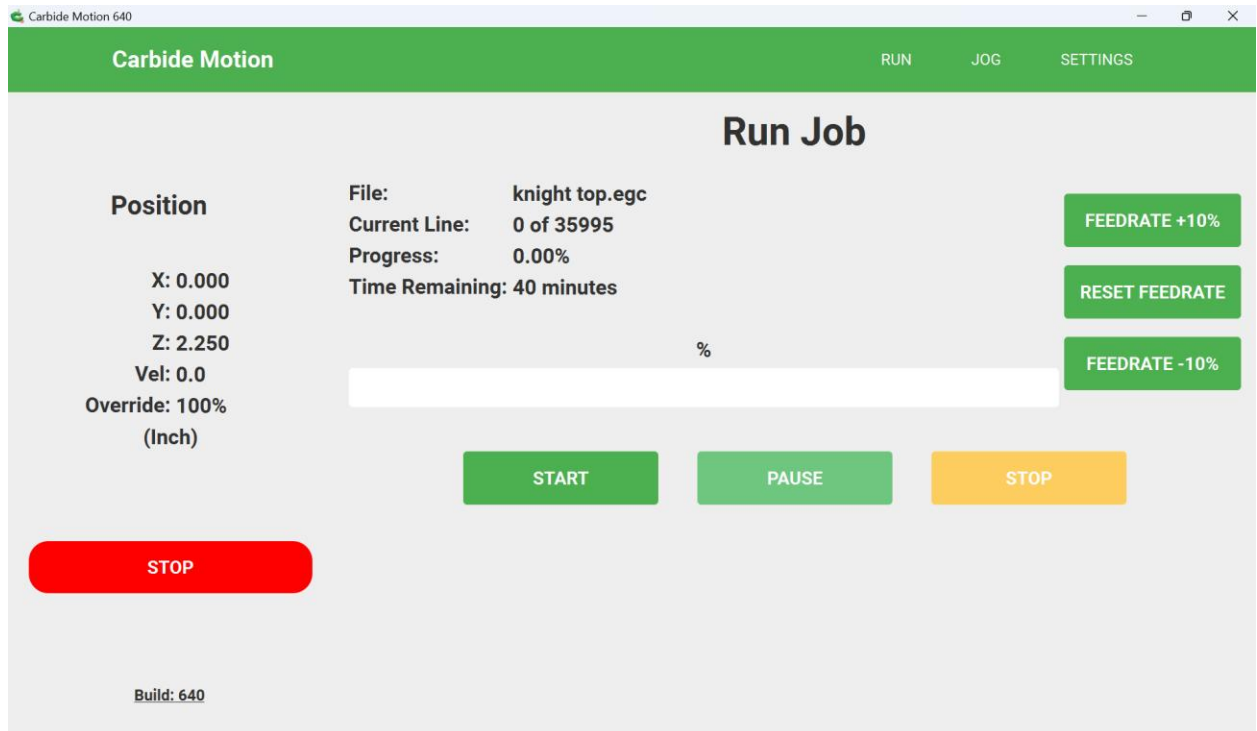
SBME Jim McEwen Zone for Innovators, creators and
Entrepreneurs Training

- i. With the BitZero probe in the collet, place the BitZero on the corner of the stock and place the ground clip on the metal collet.
- ii. Under the Jog tab, move the probe end so it is within the cylindrical section on the outside corner of the bitzero.
- iii. Select Probe and allow the movements to run (you may need to tape the BitZero down if you see excess movement during the process).
- iv. The machine is now zeroed.



Cutting Execution:

1. Load the GCode cutting operation file and choose start job.



2. Monitor progress, watching especially for collisions with the jig screws or walls.
3. While feedrate cant be overridden here, it is best to leave it default for now.

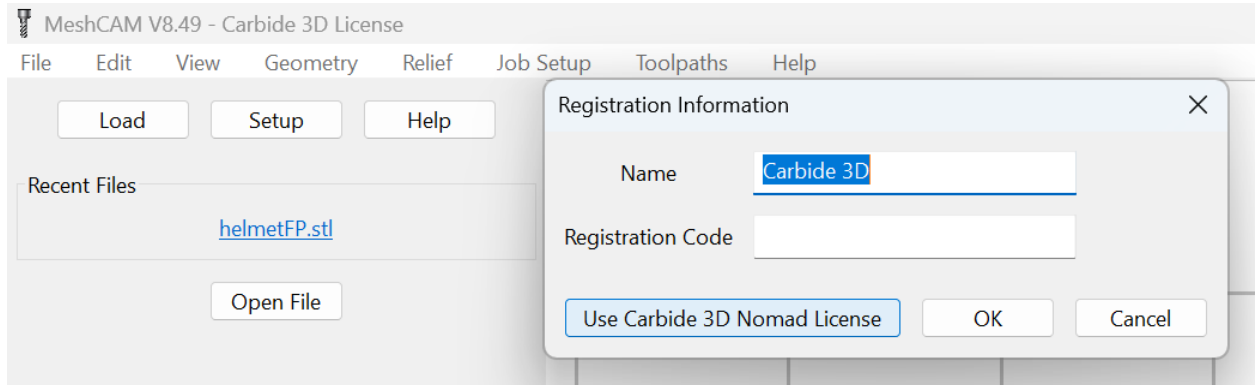
Using the Flip Jig:

The flip jig allows parts to be cut from both the bottom and top while maintaining alignment to increase accuracy over manual flip CNCing. The flip jig must be attached to the bottom metal plate of the CNC base and inset to the left side. Use all 4 corner bolts to secure the flip jig to prevent movement during cutting operations.

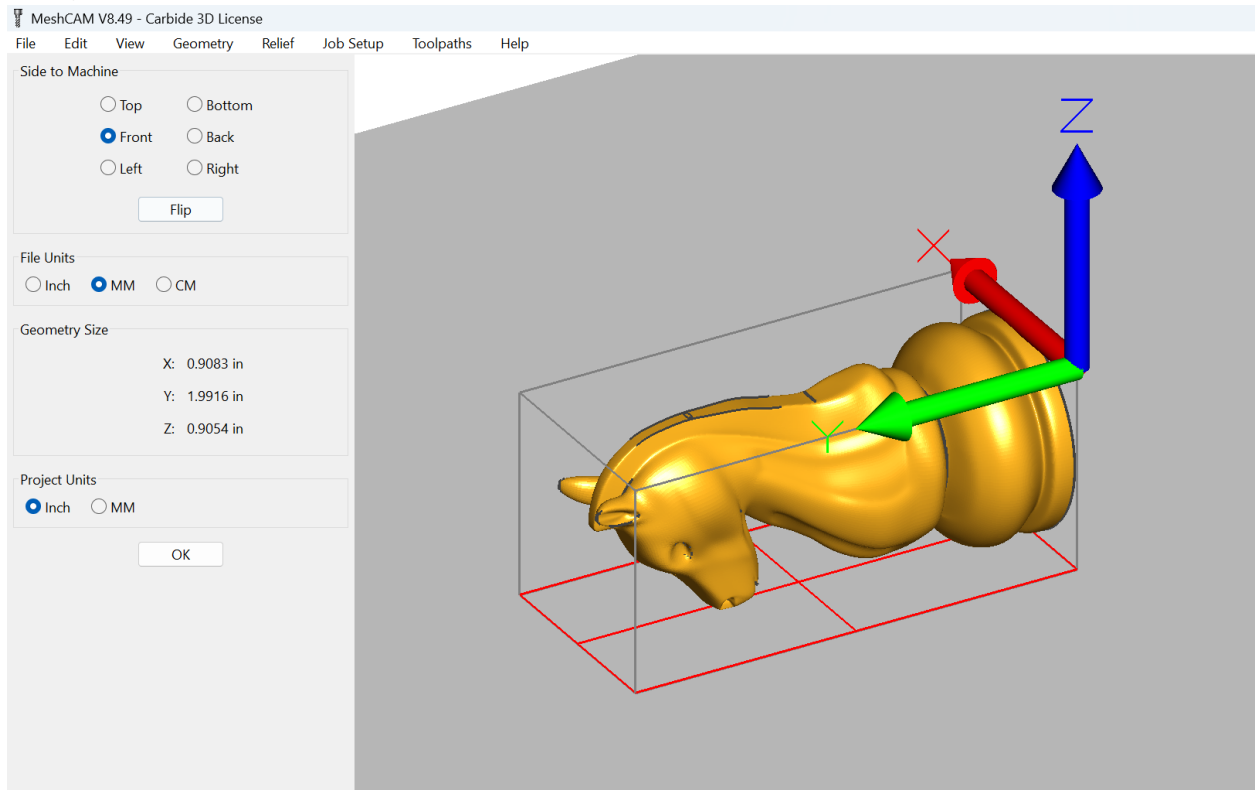
The flip direction will generally be about the X axis (i.e. the top edge of the jig becomes the bottom edge, rotated around an imaginary rod going left to right through the middle of the jig). Though, this should always be confirmed. When setting up a flip file, set the properties as previously in MeshCAM.

SBME Jim McEwen Zone for Innovators, creators and
Entrepreneurs Training

4. Open MeshCAM v8 and select → Help → Register → Use Carbide 3D Nomad License.



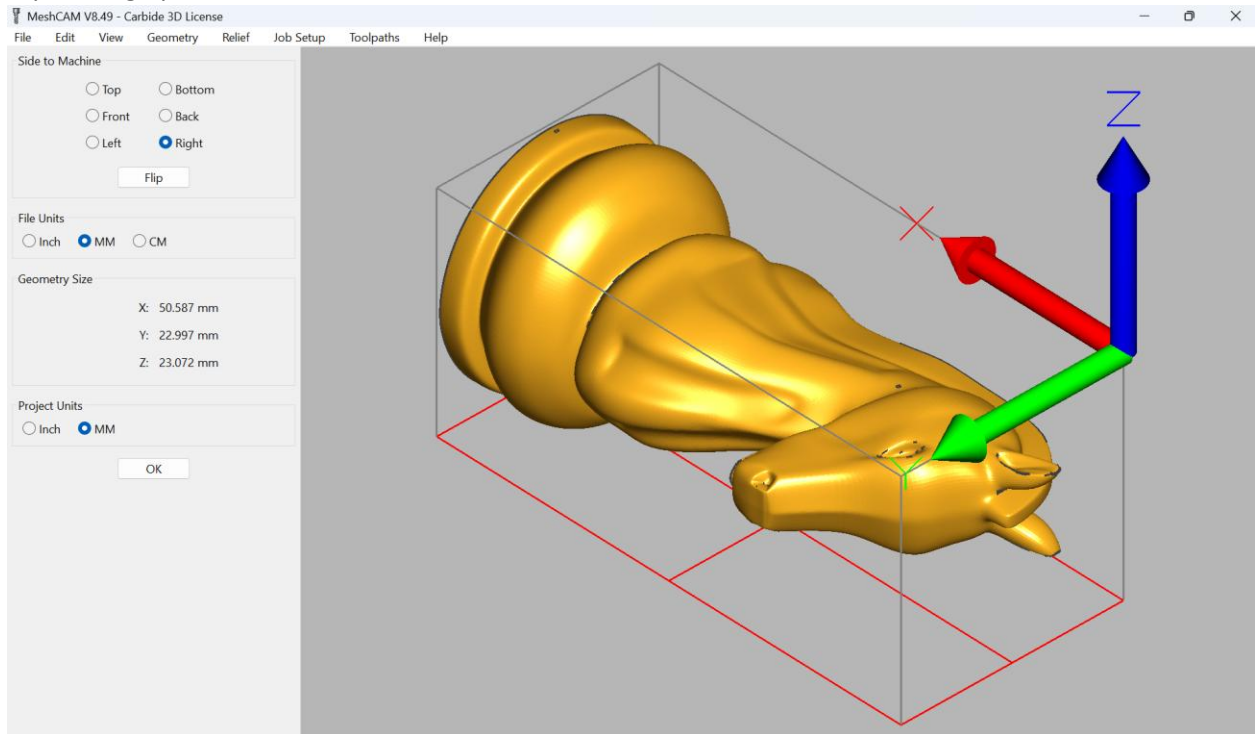
5. Import your STL. Select your correct file units and choose the project units (it is easiest to match them).



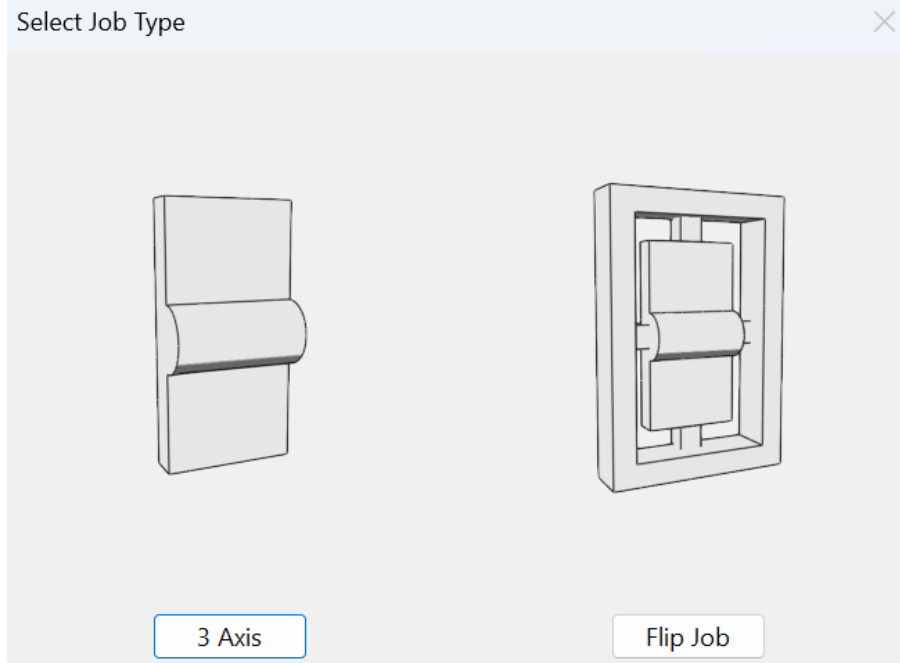
6. Choose an orientation for your part based on what features you most care about. For this job, as it is a flip job, I will place the side face of the knight upwards to have a symmetrical bottom and

SBME Jim McEwen Zone for Innovators, creators and
Entrepreneurs Training

top cutting operation.

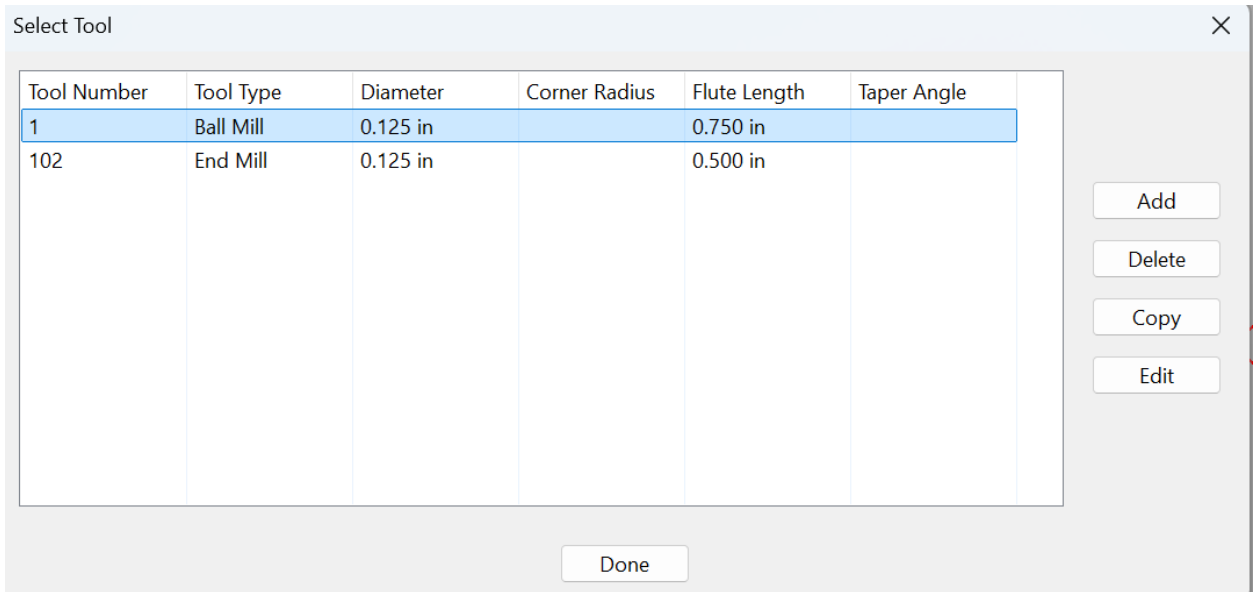


7. Choose flip job under job type.

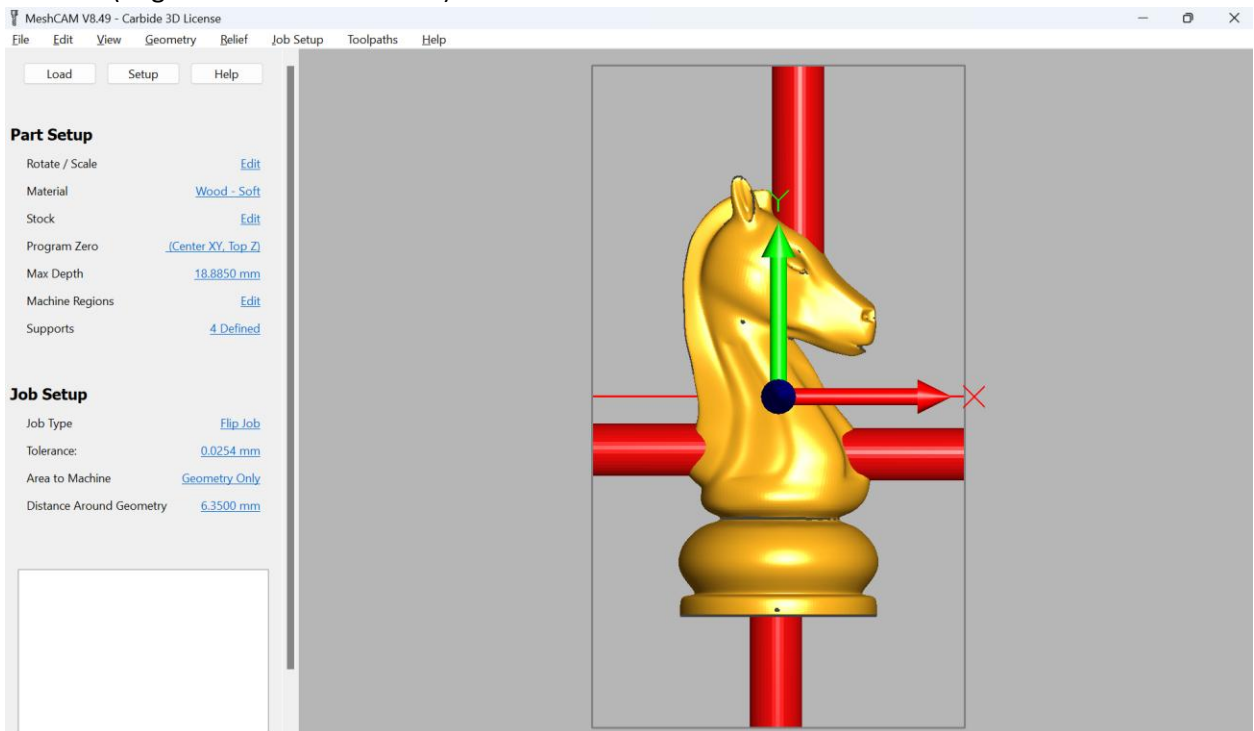


8. Under “setup → list tools”, choose or create the end mill(s) to be used in this project.

SBME Jim McEwen Zone for Innovators, creators and
Entrepreneurs Training



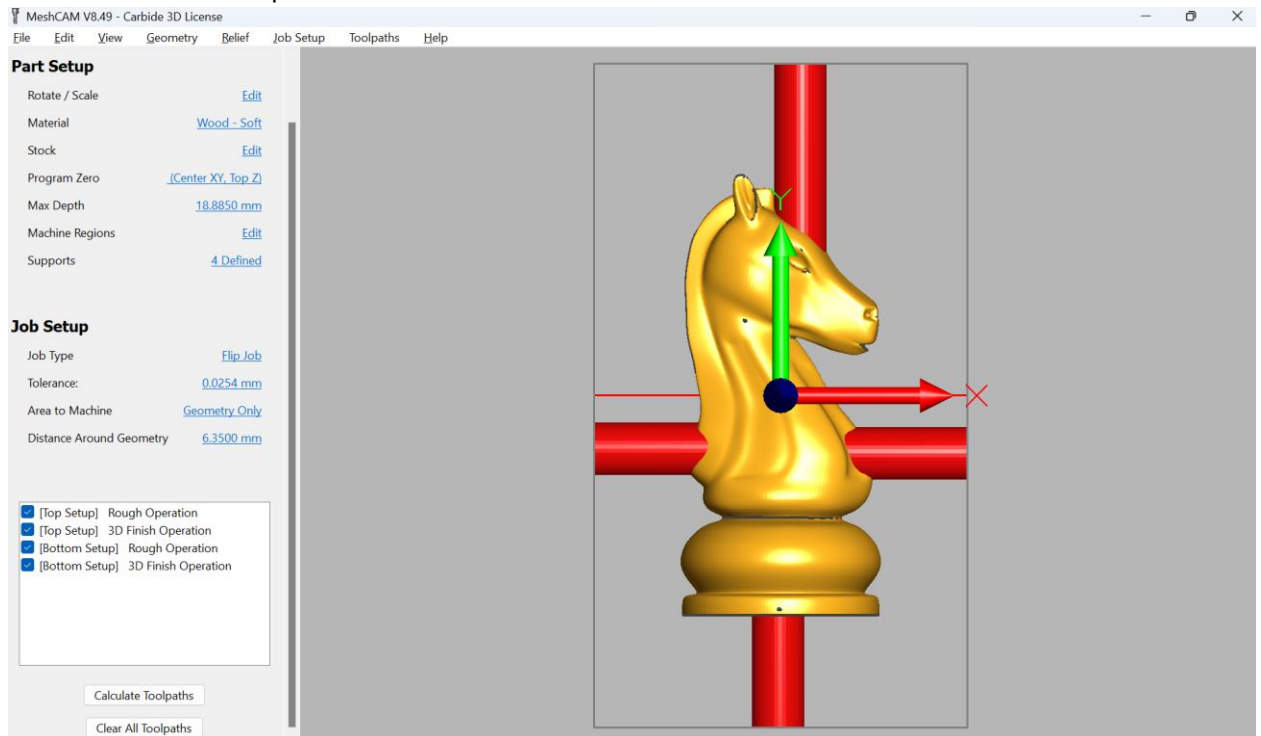
9. Set your part setting including stock size and material, program zero, and max depth. For max depth of a flip cutting operation, it is dependent on the object features. You need to be able to reach any upwards (or downwards) facing features when cutting from that face.
10. Build supports using the built in function. Set supports to be $\frac{1}{4}$ in or greater depending on material (larger for weaker materials).



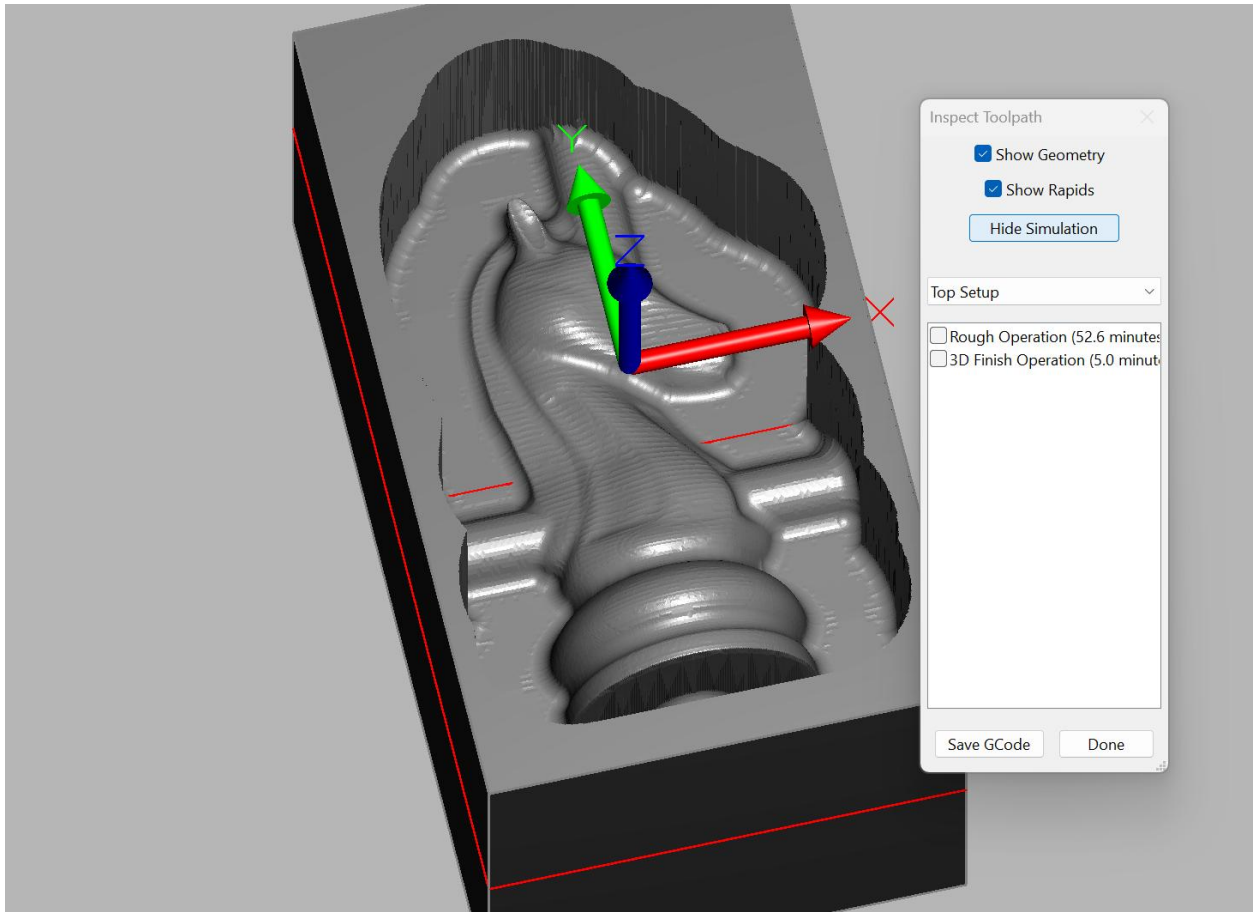
11. Set the job tolerance and choose Geometry only for the area to machine. Set the distance around geometry to be at least the diameter of your end mill.
12. Choose toolpaths → Roughing and select your tools. The information can be kept standard. Add another toolpath for Unified Finishing.

SBME Jim McEwen Zone for Innovators, creators and
Entrepreneurs Training

13. Choose Calculate Toolpaths.



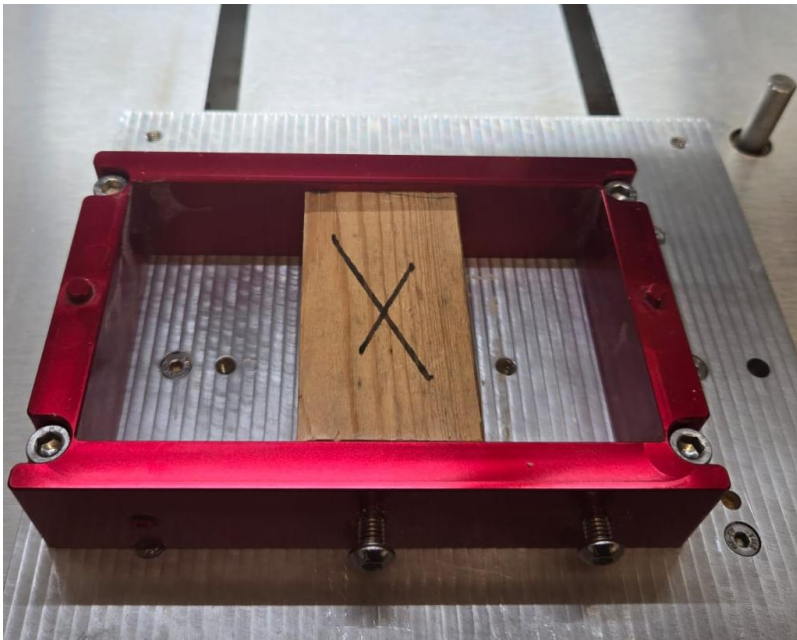
14. Simulate the toolpath and turn off the toolpath visibility to examine the resulting part. The same can be done for the bottom setup.



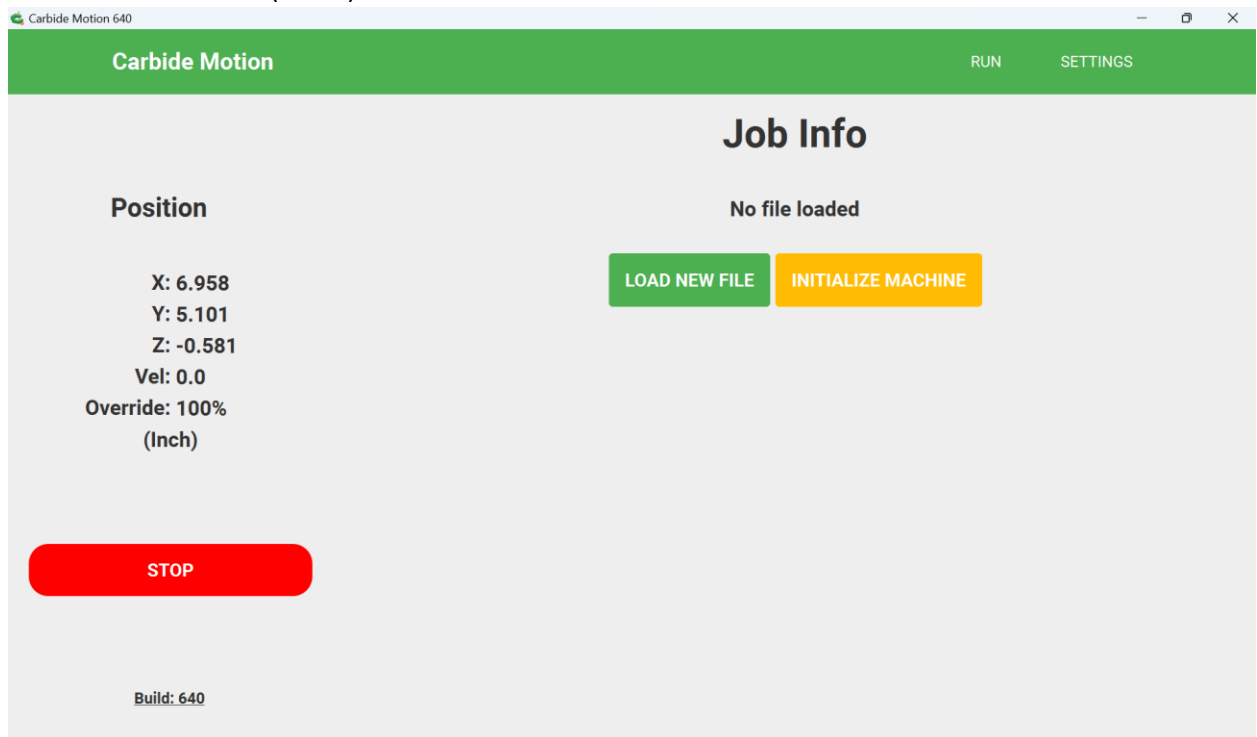
15. Once confirmed, re-enable the toolpaths and choose Save GCode. It will ask you to save both the top and bottom GCodes separately.

SBME Jim McEwen Zone for Innovators, creators and
Entrepreneurs Training

16. Insert the stock into the jig. Ensure the directional axes align with how it was defined in meshCAM (Y is forward/back and Z is up/down). Attach the jig to the baseplate.

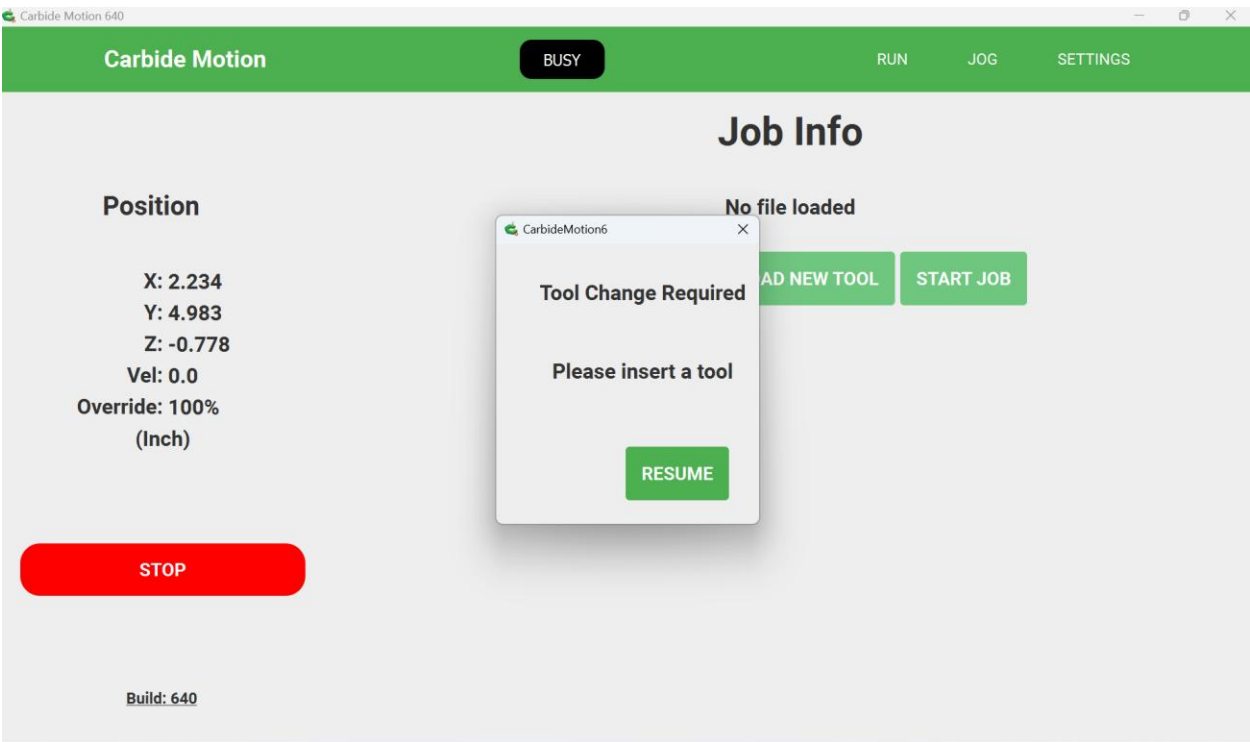


17. Connect to Carbide Motion and the CNC.
18. Initialize the machine (home).

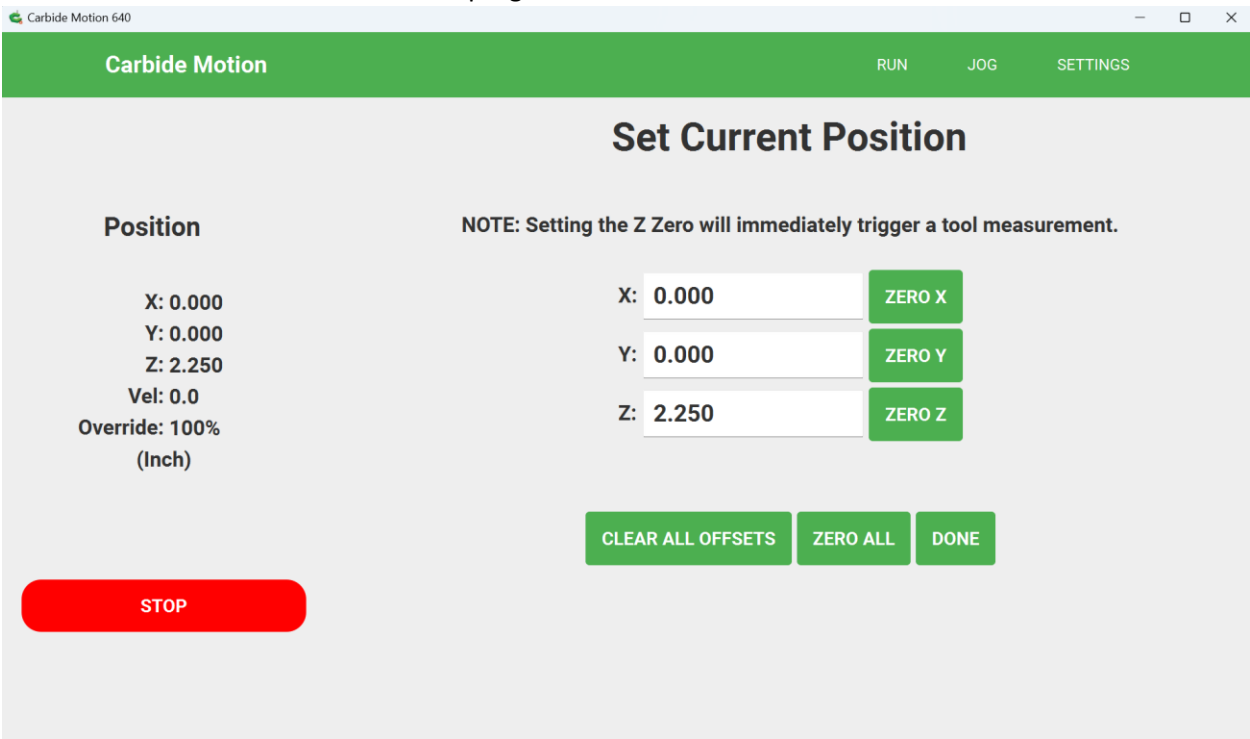


19. Choose Load New Tool and place the end mill to be used into the collet chuck. Ensure enough of the bit is in the collet for rigidity and enough is left out for the depth of cut needed. Never place

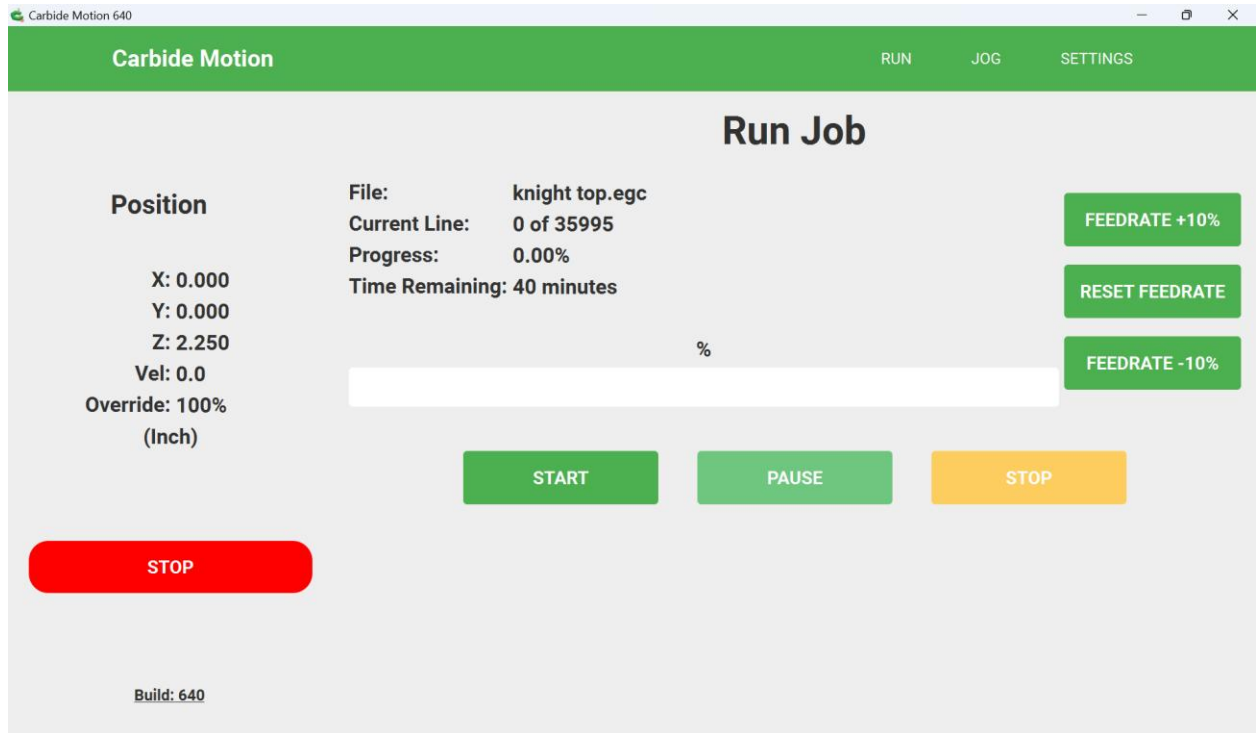
flutes in the collet.



20. Set the CNC zero based on the defined program zero set earlier.



21. Load the top cutting operation file and choose start job.



22. Monitor progress, watching especially for collisions with the jig screws or walls.

23. Once the top operation is complete, flip the jig about the appropriate axis, generally X, and begin the bottom cut operation.

- a. Vacuum the CNC bed while flipping the jig to prevent chips from getting under the jig and making it non-level.

24. Monitor progress, watching especially for collisions with the jig screws or walls.

25. Remove the support material around the object and sand or file down the residual material.

Notes:

- The Flip Jig is made to hold stock that's just under 3"x5" in size, and up to 1.2" thick. Therefore, the part being cut must be at most this large, accounting for needed support and remaining stock wall material.
- Ensure stock is centered in the flip jig in all axes, use shims if necessary.
- The door on the Nomad will oftentimes open just enough to break the interlock and stop the CNC due to the air pressure from the fan and the vibrations. If this occurs, push the door back in and select Continue or Start again.
- Be careful not to hit the baseplates with the cutter when machining down to the bottom of a stock, use a sacrificial board if necessary.
- Prior to machining more expensive materials, it is also best to do a test run in a cheap wood or substitute material to confirm first.

SBME Jim McEwen Zone for Innovators, creators and
Entrepreneurs Training

- Do not override feed or spindle rates, these will be automatically calculated based on material and the endmill chosen.
 - However, always confirm these speeds by checking a lookup table before machining.
- Maintain supervision of the machining at all times, especially when cutting metals, in order to turn off the machine in case of an emergency.
- Metal chips from the cutting process will be sharp, use gloves when removed parts from the CNC and deburr all metal parts following removal.
- Keep an eye on metal parts for discoloration during cutting, this may indicate excessive heating, you may need to let parts and the bit cooldown somewhat after a long period of cutting.
- Always cut with the fan attached.
- The main difference between a 3 axis and flip job type is that in a flip job you are performing 2x 3 axis operations, but only set the zero once. Therefore, it is imperative that the stock size is defined accurately, the stock is placed in the center of all axes of the jog, and the zero is set accurately according to the defined program zero.