Nomad 3: MeshCAM 3D Head Tutorial

This tutorial is an introduction to working with 3D projects in MeshCAM 8 and cutting 3D parts using your Nomad 3. For this project, we assume that you have already completed the Nomad 3 Starter Project. If you have not yet done so, we strongly recommend completing that tutorial before moving ahead with this tutorial.

If you run into problems as you are completing this tutorial or find any of the steps confusing, don't hesitate to reach out to our support team for help: support@carbide3d.com.

What You'll Need

- 2″ × 3″ × 1″ renshape
- #101 – ⅛″ ball mill
- BitZero V2
- ⅛″ probing pin
- Double-sided tape

Project Files

To run this project, you'll need to download the head.mcf project file from the Carbide 3D website.

Download the head.mcf project file here

Required Programs

MeshCAM 8

You'll need MeshCAM v8 to mill this 3D project on your Nomad 3.

MeshCAM is an easy-to-use 3D CAM program designed from the start to minimize the learning curve for new users, while allowing advanced users to complete complicated projects with ease.

Get MeshCAM 8 here

Carbide Motion

Carbide Motion lets you control your machine by jogging it around, setting zeroes, and loading and running G-code (the code that tells your machine when and how to mill). If you don't have Carbide Motion yet, or you need the latest version, go ahead and download it now.

Get Carbide Motion here

Important Safety Notes

The Nomad 3 has an interlock. When the spindle is running, opening the door will trigger the interlock and the spindle (and the job) will pause and move up to a parked position. When the interlock is triggered, Carbide Motion will display the pause screen. Once the door is closed again, you'll need to click the **Resume button** on the pause screen to continue the job.

The Nomad 3 enclosure is there for two reasons: to contain the mess and to protect you from broken cutters and flying stock.

The end mills are made of carbide, a very hard, very brittle metal. When they break, they snap suddenly and without warning. The window will help protect you from any broken cutters.

The second danger is that the stock material can break away during cutting. This can happen because you failed to use enough double-sided tape, because the material didn't adhere well enough using that tape, or because you were cutting too fast.

If either of these happens during a job, power the machine down (hit the emergency-stop if one is available on your setup) and fix the problem.

In summary, always close the door when the machine is running.

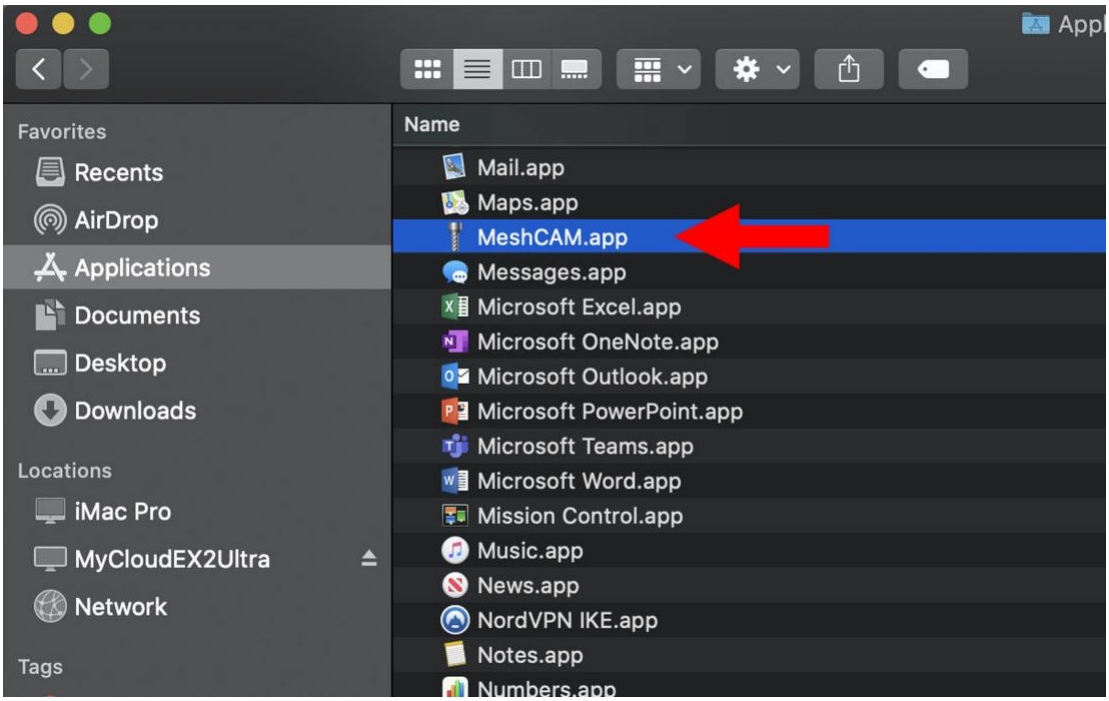
> **NOTE:** We also strongly recommend keeping the door closed when the machine is not in use. Leaving the door open for long periods of time could weaken the door hinge.

Download and Install Programs and the Project File

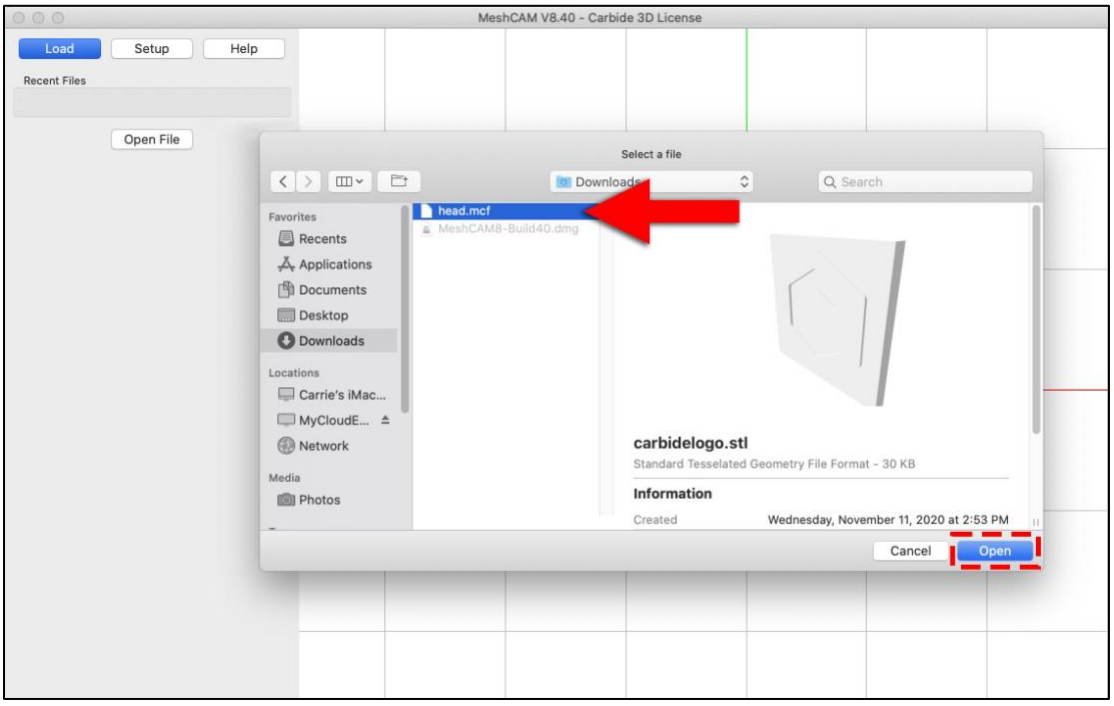
1. Download and install MeshCAM 8 and Carbide Motion, if you have not already done so.
2. Download the project file above.

Set up the Project in MeshCAM

1. Launch MeshCAM 8.



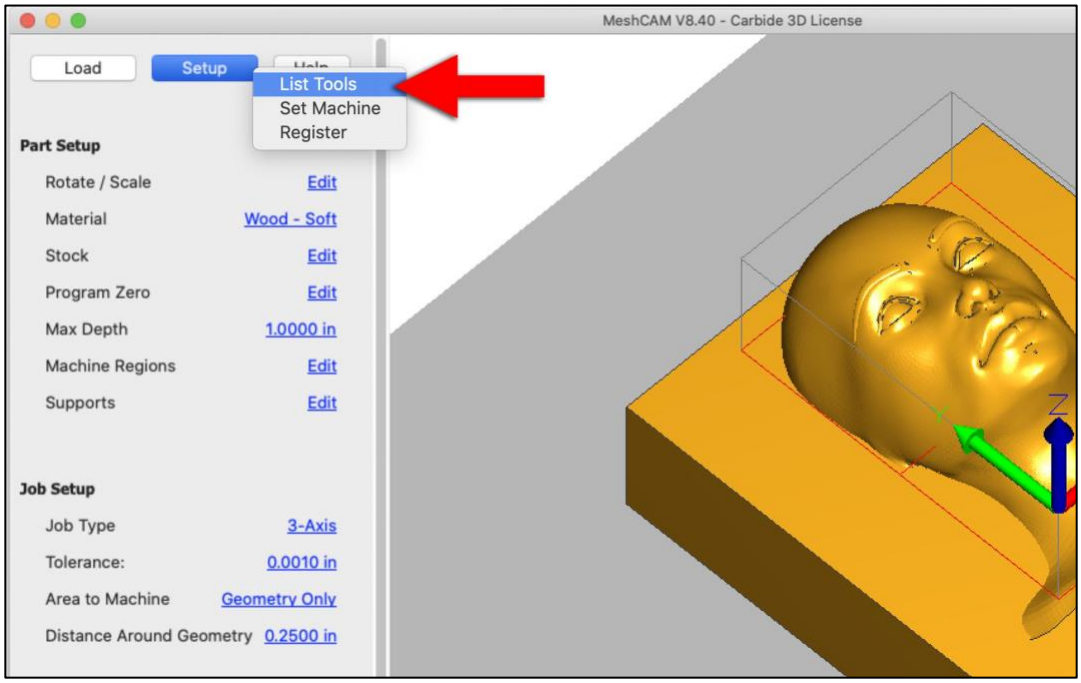
2. When prompted to enter your registration code, click the **Use Carbide 3D Nomad License button**.
3. We'll be starting the project with a CAD file. In MeshCAM, click the **Load button** and browse to where you saved the **head.mcf** project file.
4. Select the file and click **Open**.



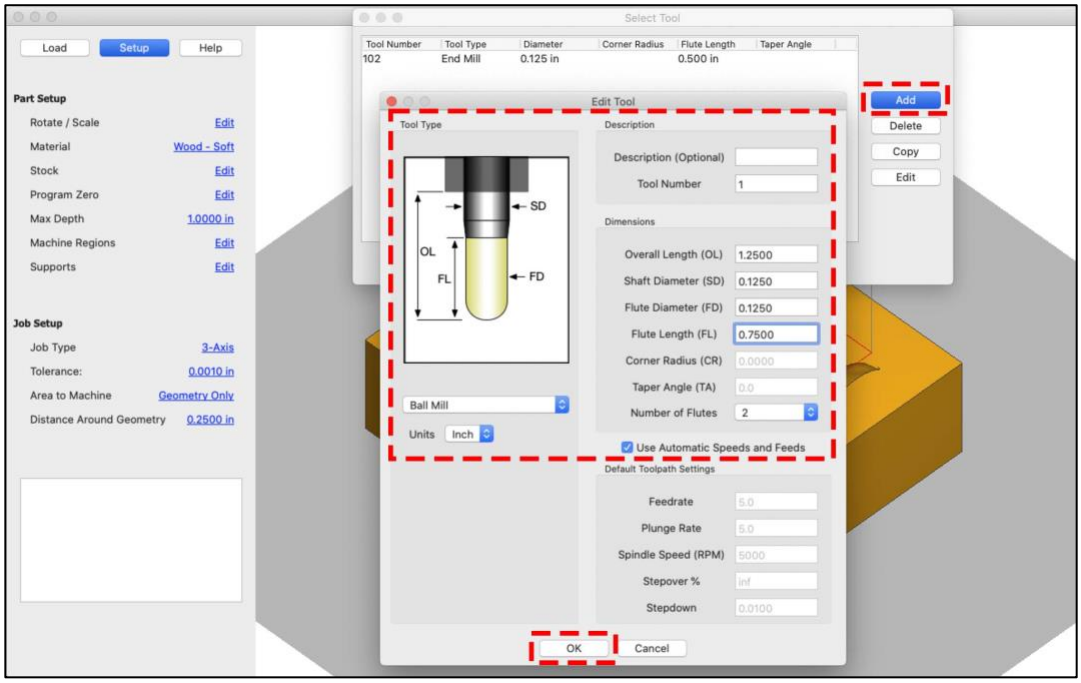
Add a Tool

MeshCAM needs to know what tool you plan on using to cut the geometry so it can properly calculate a toolpath.

1. Click the **Setup Button**, then select **List Tools**.



2. Click the **Add button** to add a new tool.
3. Enter the tool settings shown below.



4. Click **OK** to save and close the window, then click **Done** to close the *Select Tool window*.

Input Part and Job Setup Parameters

On the left side of your screen you can see the *Part Setup* and *Job Setup* panels. These parameters are global settings that need to defined for each toolpath generated.

In the *Part Setup panel*:

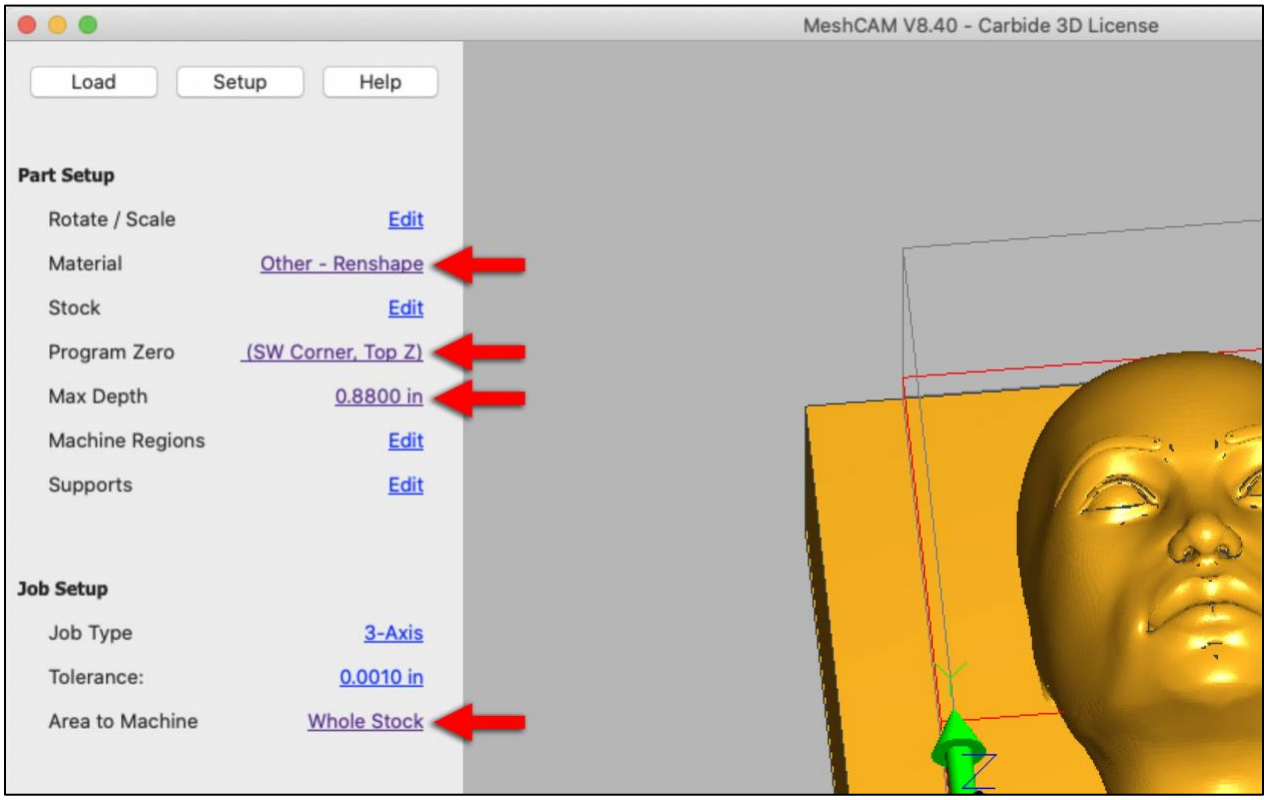
1. Click the **Material value** and select **Other – Renshape** from the list of materials.
2. Click the **Program Zero value**. In the *Z-Position section*, choose **Top** and in the *XY Position section*, select the **bottom-left corner**. Then, click **OK** to close the window.
3. Click the **Max Depth value** and enter **0.88"**. This will be the maximum depth of our toolpaths. We don't want to cut the head completely from the stock, so we're going to leave a small amount of stock at the bottom.

In the *Job Setup panel:*

1. Click the **Area to Machine value** and choose **Machine Whole Stock**.

The values for the remaining job setup parameters should be:

1. **Job Type:** 3-Axis. There are two job types: 3-axis and flip job. 3-axis is the most common type of job, and it machines the part from a single direction with a single setup. If a part requires machining from both the front and the back, you can use a flip (two-side) job which will automatically calculate the toolpaths from the top and bottom of the part.
2. **Tolerance:** 0.001"



MeshCAM will attempt to pick sensible speeds and feeds for your project based on the material and end mill you pick. It's fine if you want to do this tutorial on a different machine or use a different material than the renshape included in the Nomad 3 kit. Just be aware that if you do change the machine or material, you'll end up with different speeds and feeds values in the Create the Toolpaths section below.

Create the Toolpaths

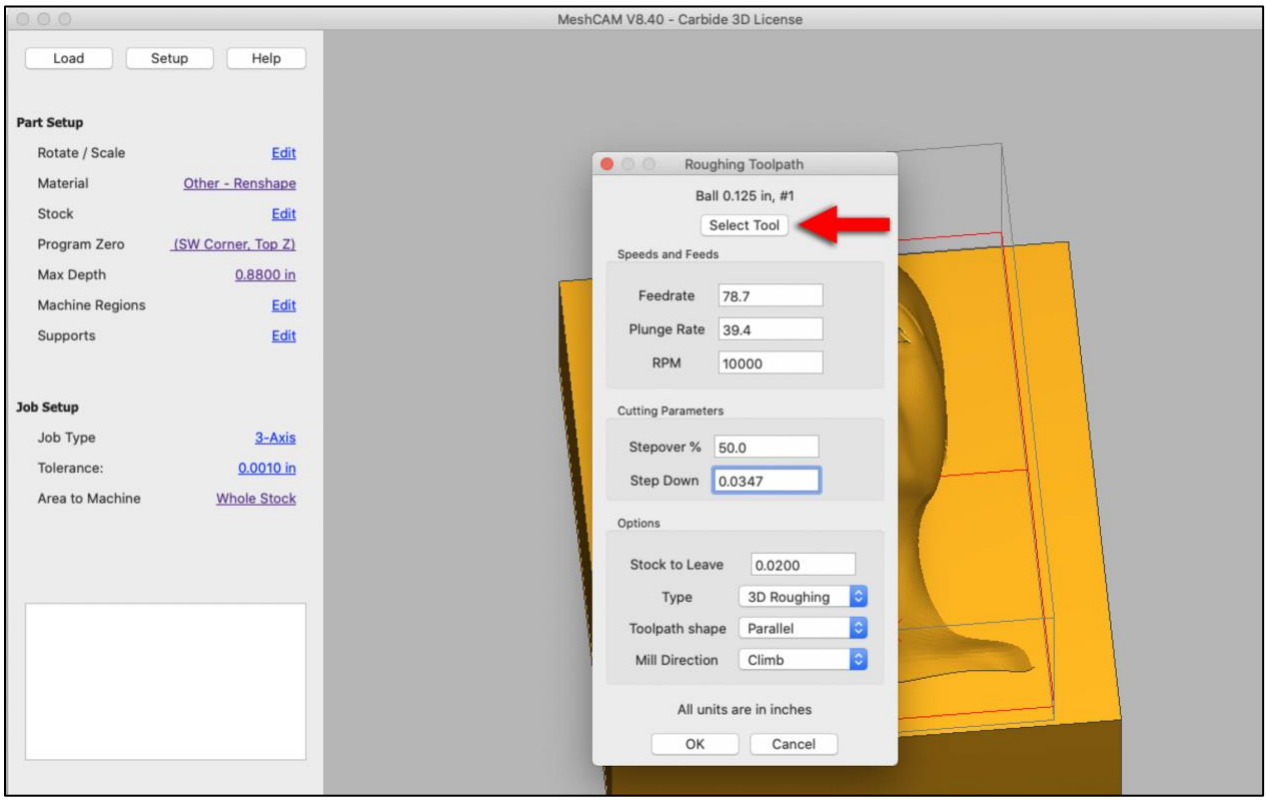
For a project like this, we'll create both a roughing toolpath and a finishing toolpath. Roughing has the task of removing material as quickly as possible while finishing has the task of making the machined surface match your model.

For the roughing toolpath:

1. In the menu at the top of the screen, go to **Toolpaths** > **Roughing**.
2. Click the **Select Tool button**.
3. Select the **#1 ball mill** you added above and click **Done**.

MeshCAM will pick approximate speeds and feeds based on your cutter, the machine, and the material you picked. Feel free to override these values if you feel comfortable doing so.

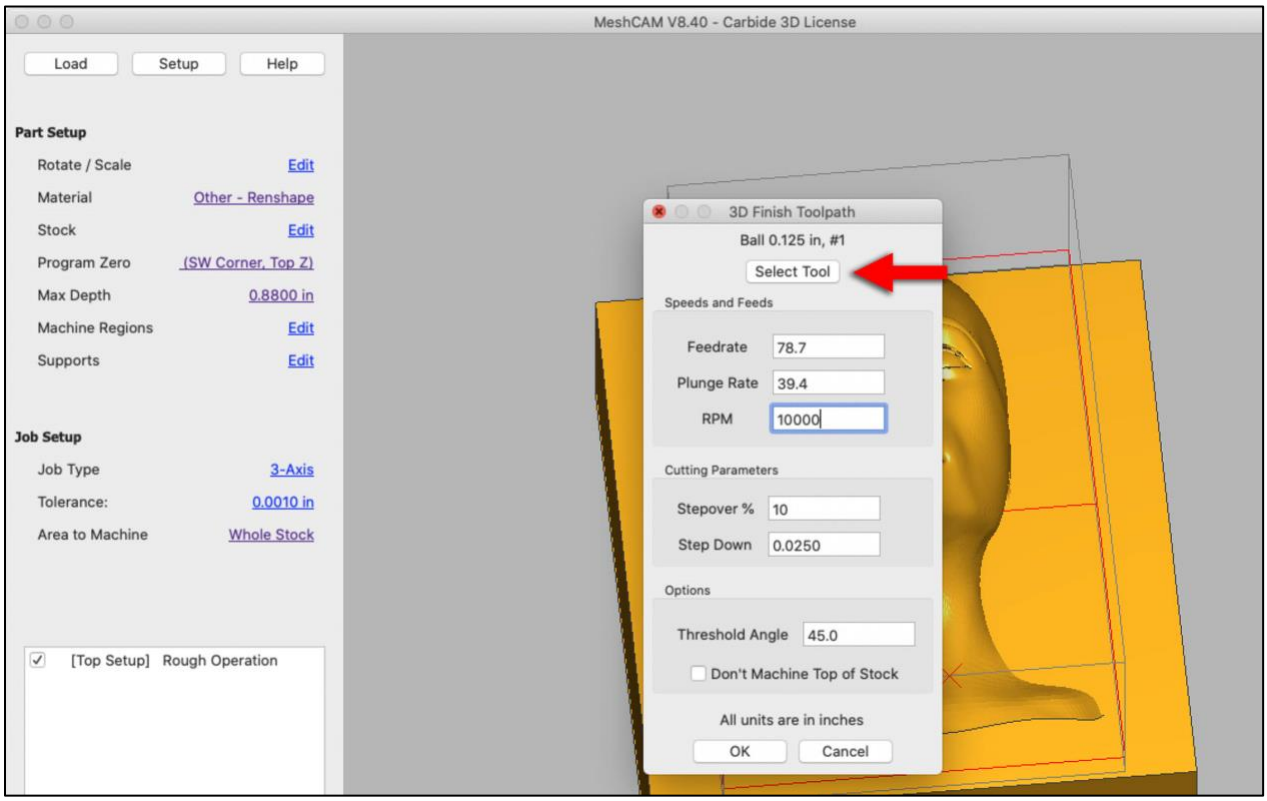
4. Enter the values shown below.



5. Click **OK** to save and close the toolpath.

For the finishing toolpath:

1. In the menu at the top of the screen, go to **Toolpaths** > **Unified Finishing**.
2. Click the **Select Tool button**.
3. Select the **#1 ball mill** again and click **Done**.
4. In the *Cutting Parameters section*, enter **10** for the **Stepover % value**.
5. Enter the values shown below.



6. Click **OK** to save and close the toolpath.

Calculate the Toolpath

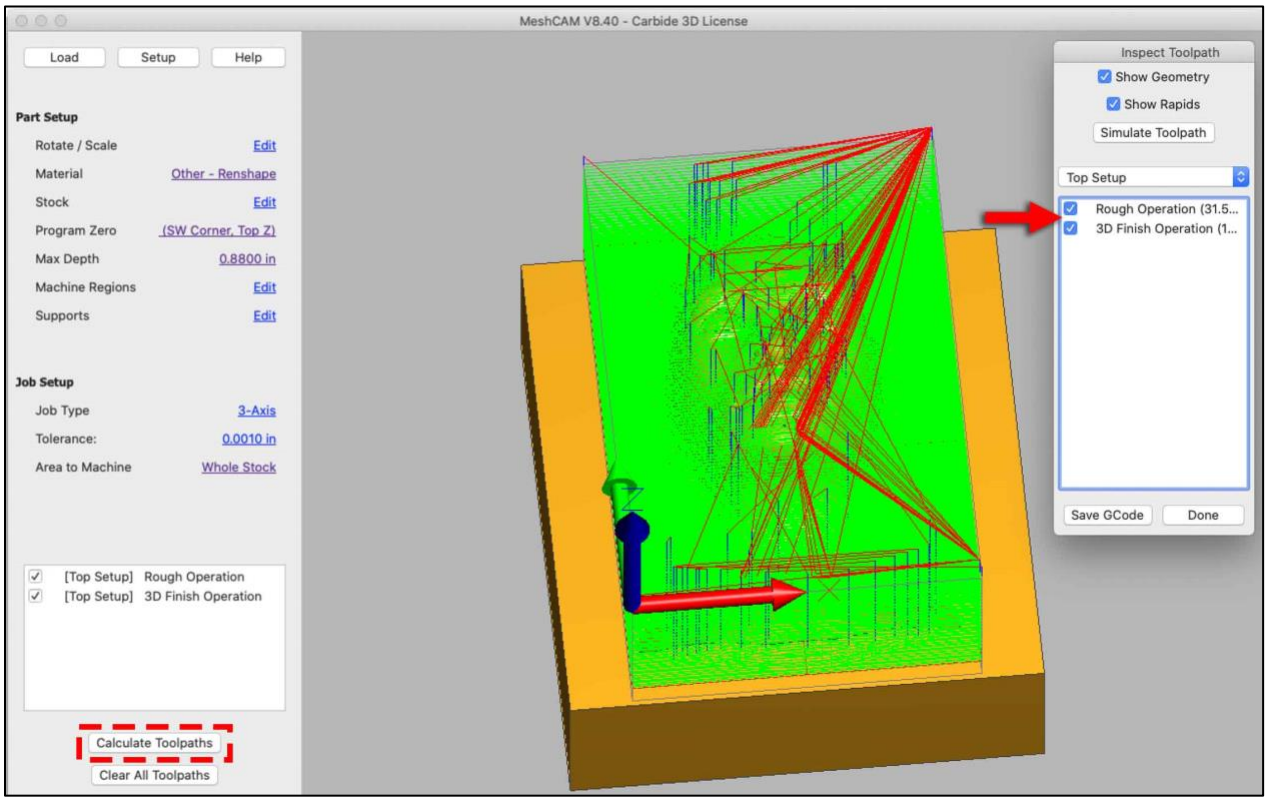
You should see both toolpaths in the *toolpath list* at the bottom of your screen. Now, we need to calculate the toolpaths.

1. Click the **Calculate Toolpaths button** below the list of toolpaths.

PRO TIP: You can edit a toolpath by double clicking its name in the toolpath list at the bottom-right of your screen. You can delete a toolpath by right-clicking on the toolpath name and selecting "Delete."

Preview the Toolpath

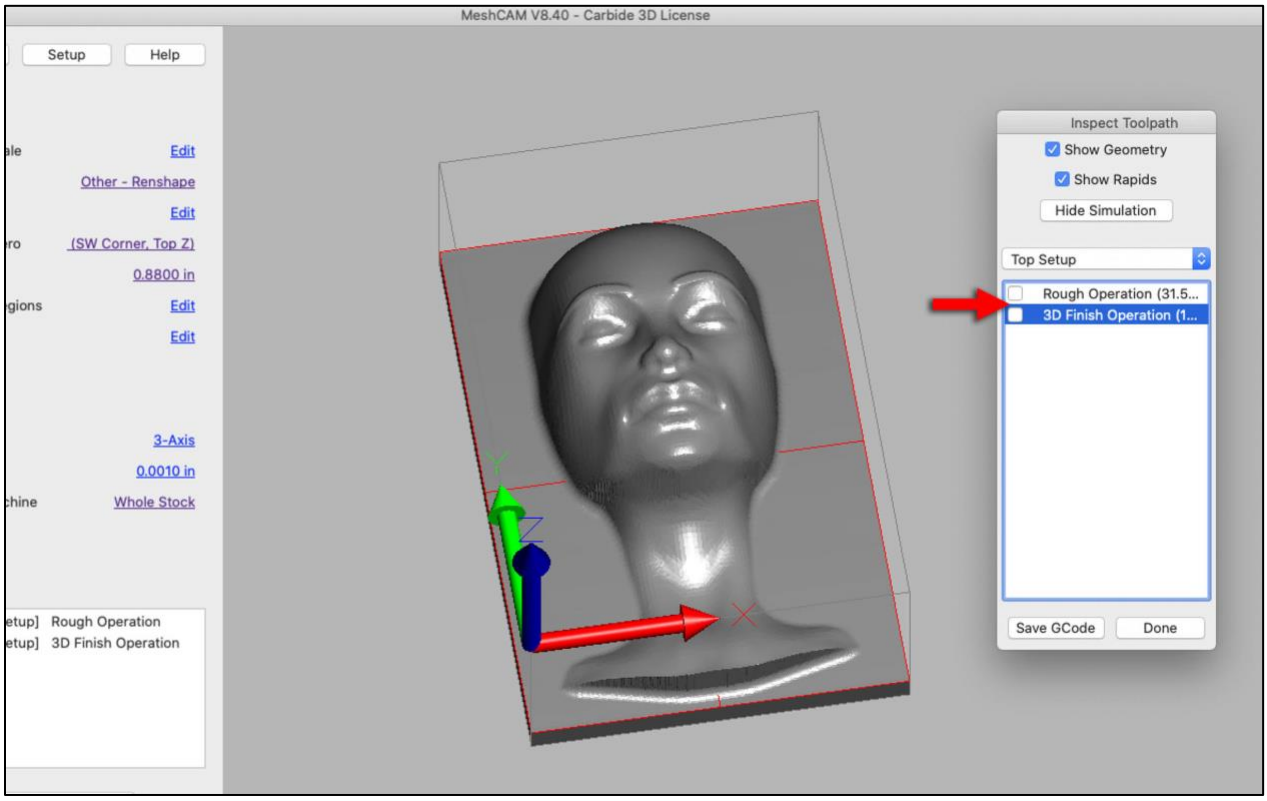
Once the toolpath appears in the 3D window, you can move the view to get a closer look at any parts you want to see in more detail. To get a better view, you can disable either the roughing or finishing toolpaths by unchecking them in the *Inspect Toolpath dialog box*. Make sure both the roughing and finishing toolpaths are checked before moving on or they will not be saved in the G-code file.



Simulate the Toolpath

Let's take a look at a full 3D simulation of the toolpath:

1. Click the **Simulate Toolpath button**.
2. If you would like to see the simulated part without the toolpaths in the way, you can disable them by unchecking them in the *Inspect Toolpath dialog box*.
3. Make sure to re-enable both toolpaths before moving on or they will not be saved in the G-code file.

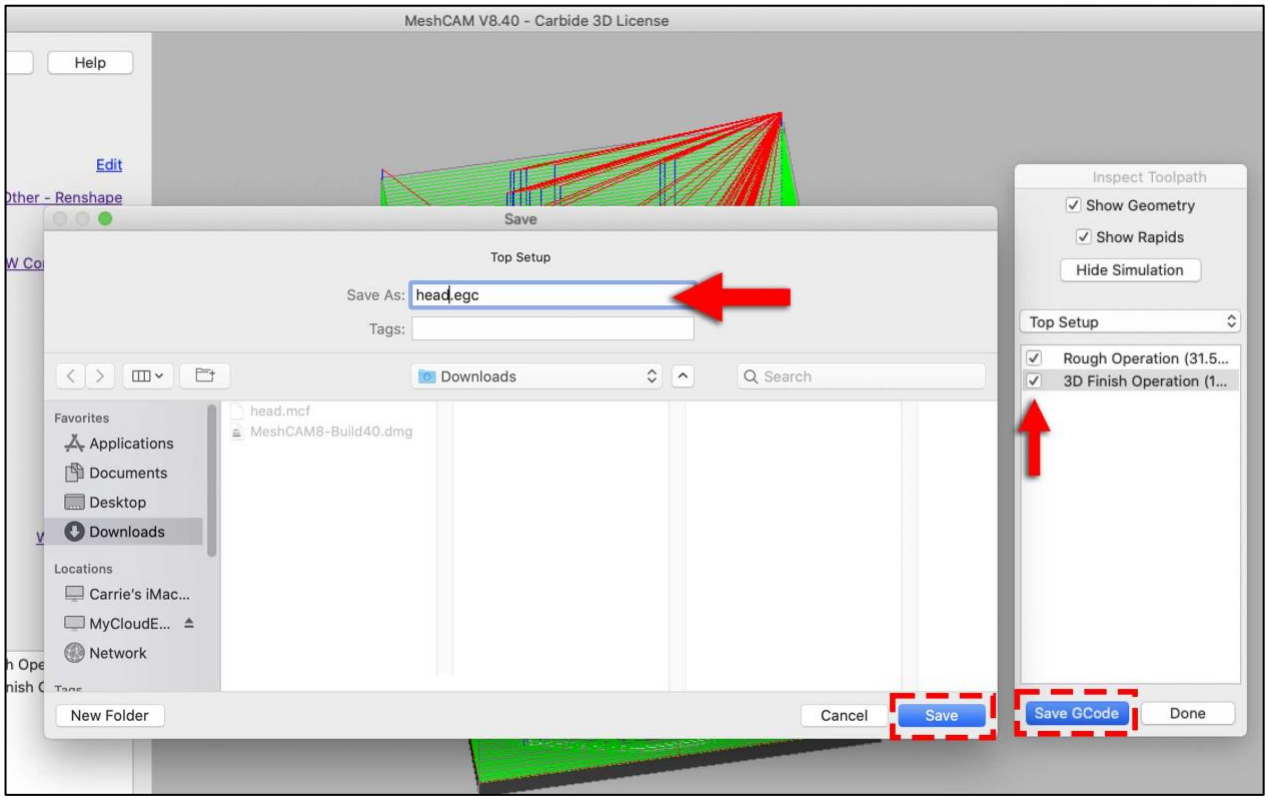


Save the G-code

Typically, G-code saves as a .nc file; however, when you register MeshCAM with the Carbide 3D Nomad License, your G-code will save as a .egc file which will only load on Nomad machines.

To save the G-code for your toolpaths:

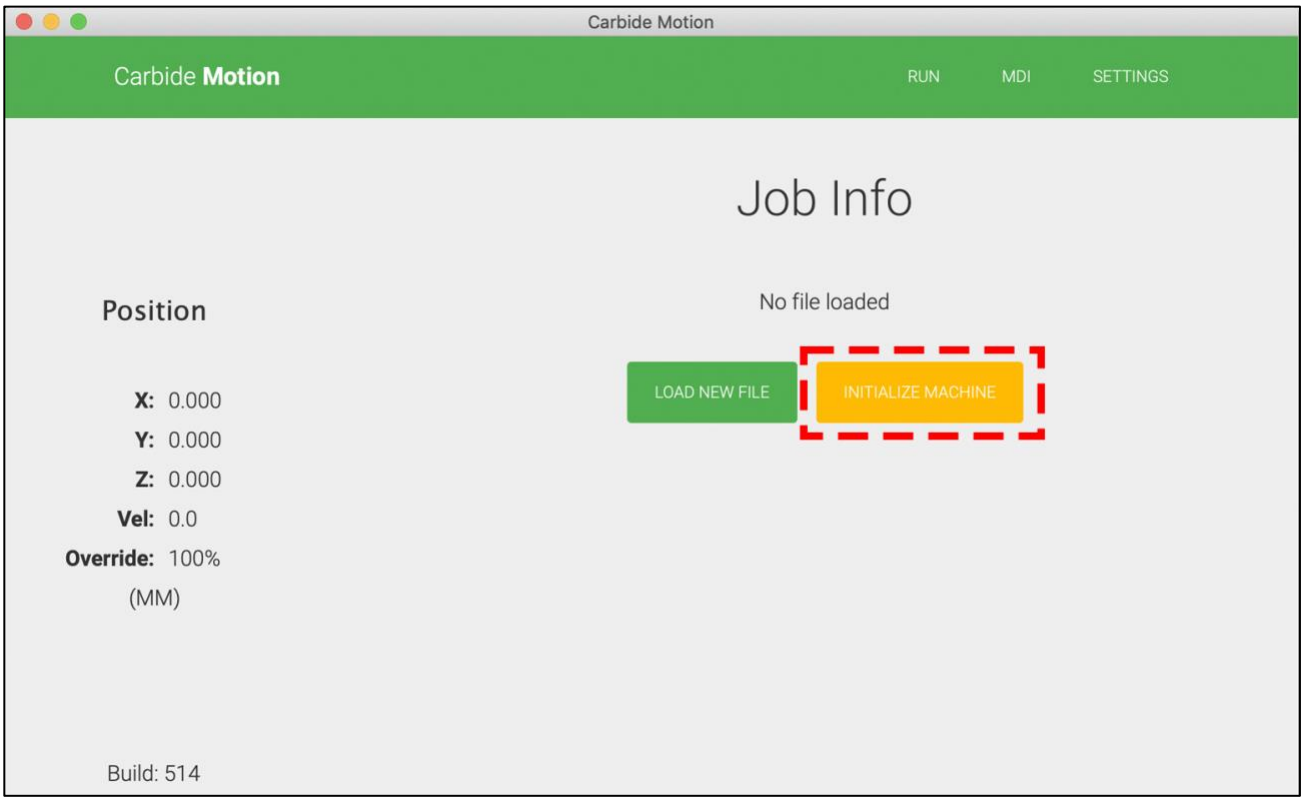
1. Click the **Save GCode button**.
2. Name your .egc file and click **Save**.



Connect to and Home the Machine

Connect your machine to Carbide Motion:

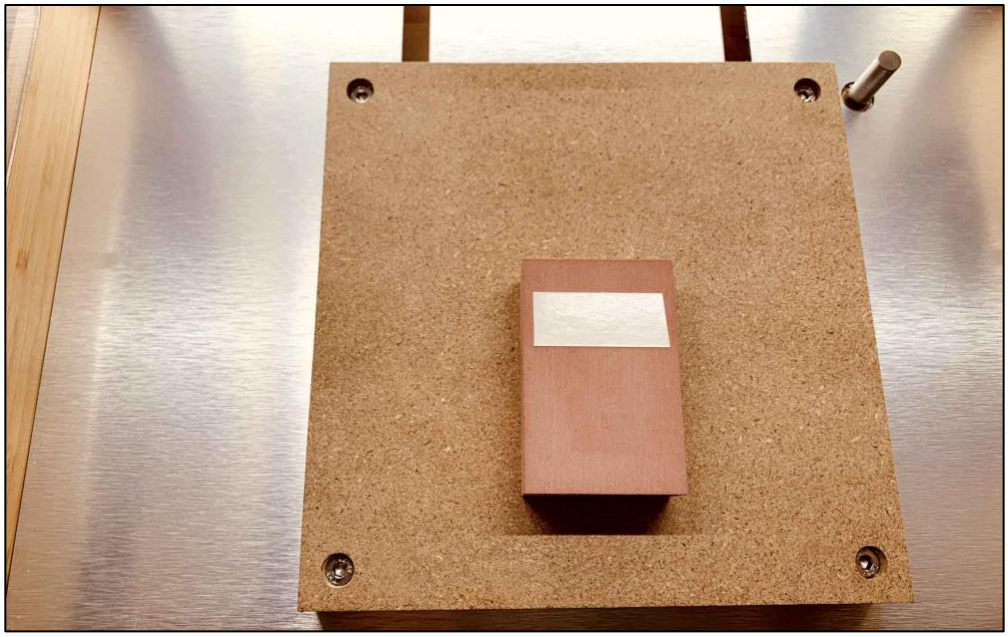
1. Make sure the power cable, USB cable, and BitZero V2 are plugged into the ports on the outside of the machine enclosure.
2. Power up your computer and connect the USB cable.
3. Start Carbide Motion, then power on your Nomad 3.
4. In Carbide Motion, click the **Connect to Cutter button**.
5. Once the machine homes, click the **Initialize Machine button**.



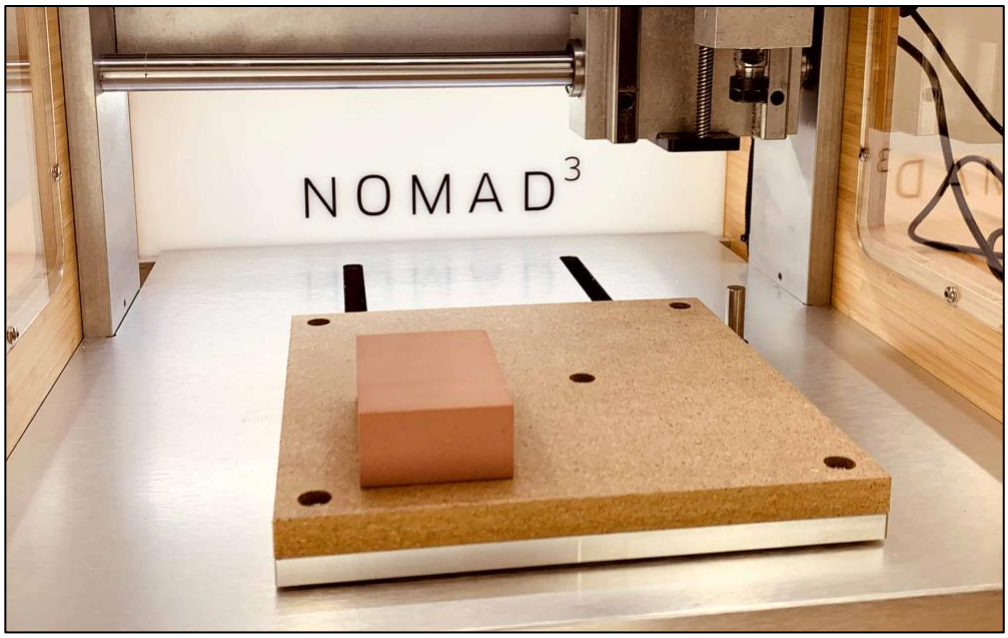
Secure Your Workpiece

Now, you need to secure the 2″ × 3″ × 1″ renshape to the wasteboard. Renshape machines like wood but has no grain, doesn't splinter, and holds incredible detail. Synthetic wood, such as renshape, is great for testing programs, learning how to use the machine, and for pattern or model making.

1. Apply a strip of double-sided tape to the top half of your stock. We're only putting tape on the top portion so we don't accidentally break the part when we pry it off the wasteboard.



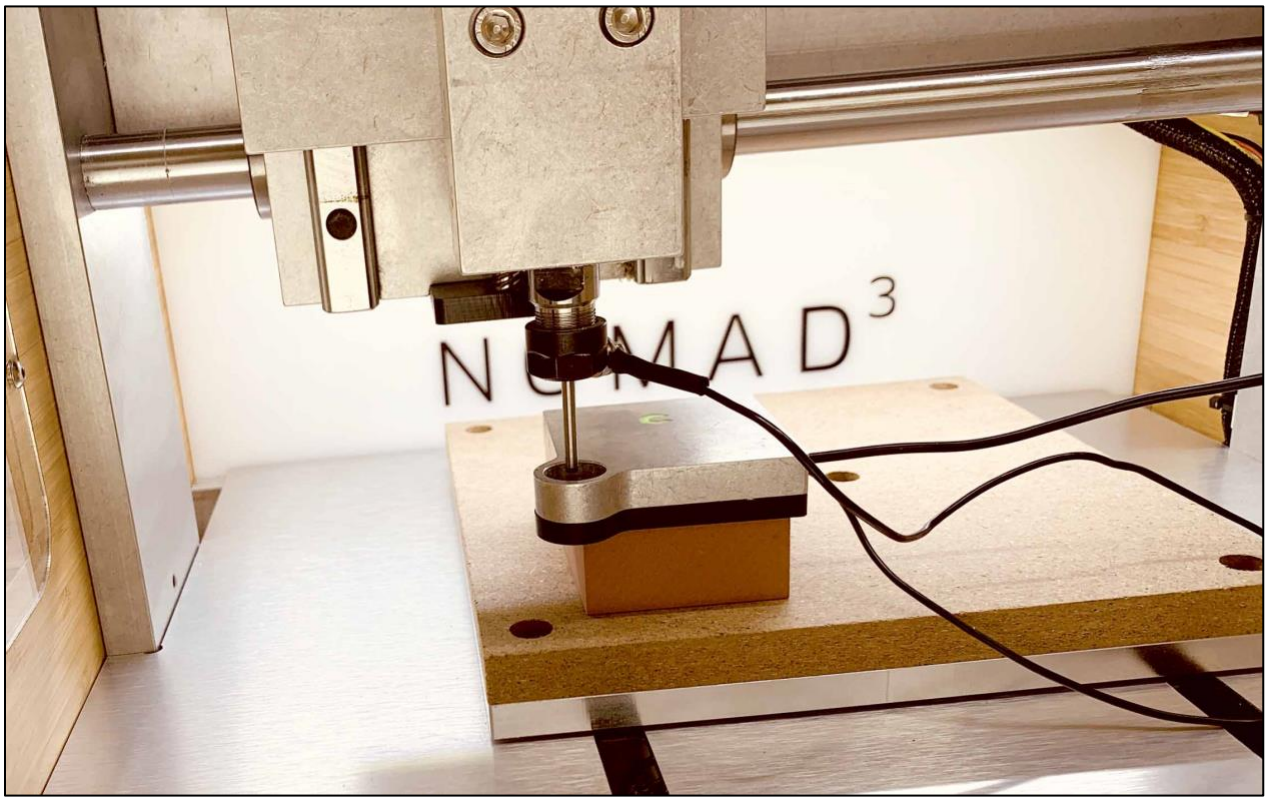
2. Ensure there is no dust or dirt on the surface of the wasteboard that could weaken the tape adhesion.
3. Align your stock to the wasteboard, making sure it is as square as possible.
4. Apply firm pressure to the stock for several seconds to secure it to the wasteboard.



Set Job Zero

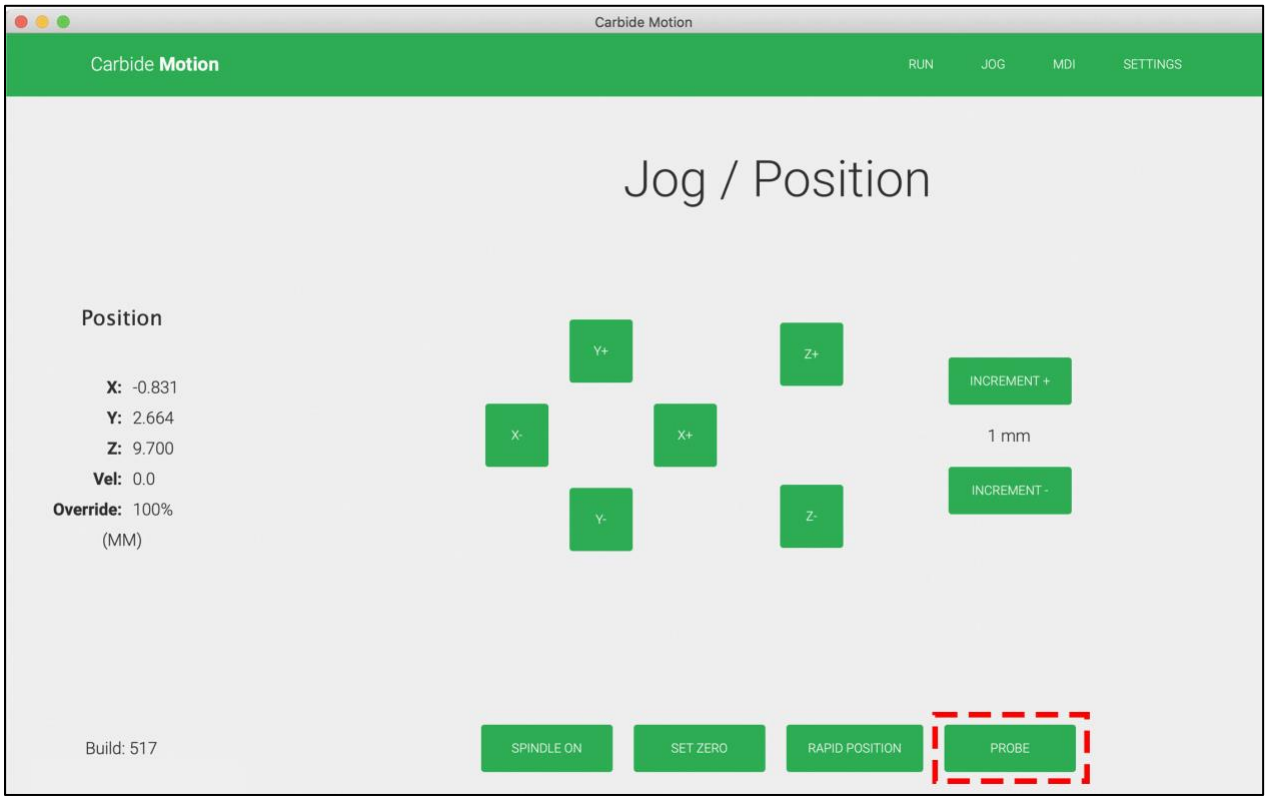
Now, it's time to set job zero using your BitZero V2.

1. After homing, Carbide Motion will prompt you to load a tool. Install the ⅛" probing pin into the collet.
2. After the length of the probing pin in the collet has been measured, set the BitZero V2 on bottom-left corner of your stock with the locating edges overhanging, but pressed up against the sides of your stock.
3. Click **Jog** in the top menu bar to open the *Jog screen*.
4. Jog the machine until the tip of the probing pin is just inside the circular pocket on the corner of the BitZero V2.



5. Attach the BitZero V2's magnetic grounding clip to either the collet nut or the probing pin in the spindle.

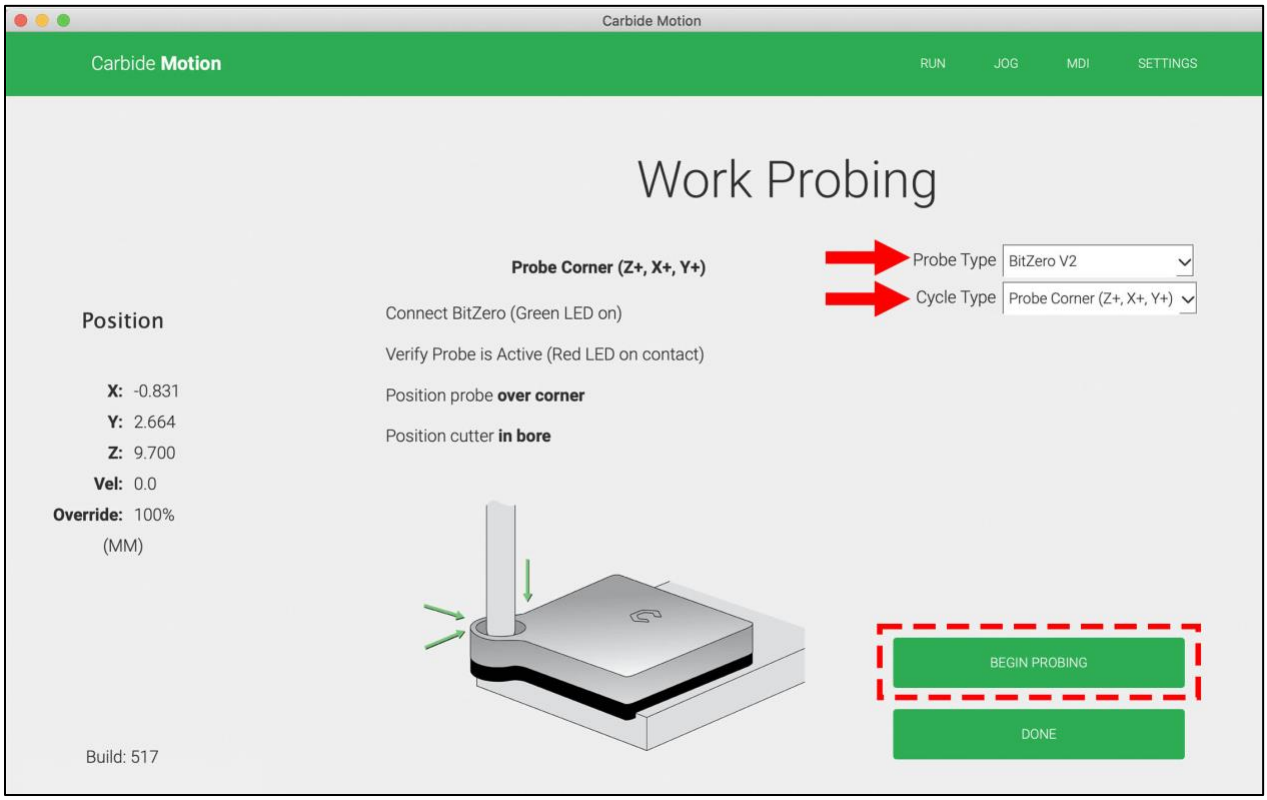
6. In Carbide Motion, click the **Probe button** to open the *Work Probing screen*.



7. In the *Probe Type dropdown list*, select **BitZero V2**.
8. In the *Cycle Type dropdown list*, select **Probe Corner (Z+, X+, Y+)**.

NOTE: Make sure your spindle/BitZero setup matches the image shown on the Work Probing screen for the **Cycle Type** you have selected. If your setup does not match the onscreen example, the calculated zero location will be incorrect.

9. Click the **Begin Probing button** to start the probing sequence and your zero position will be set for X, Y, and Z.



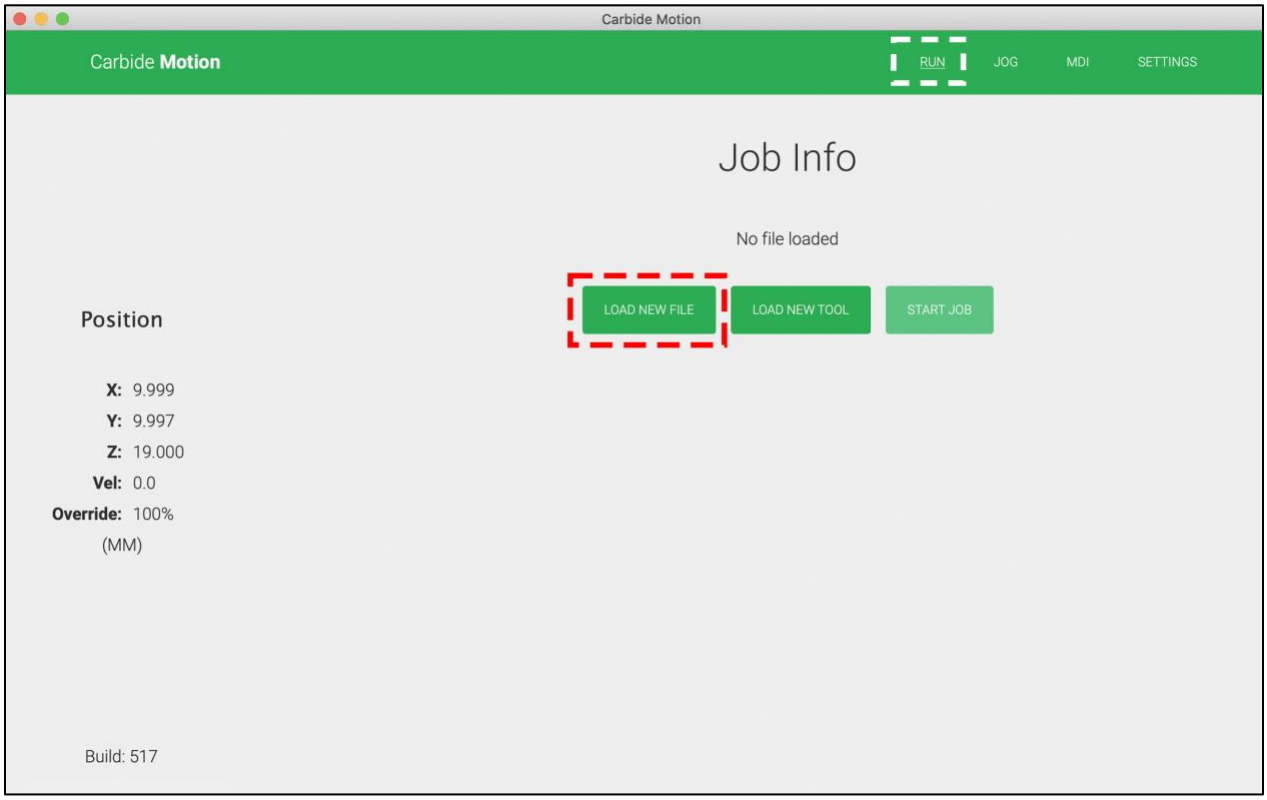
10. Once the probing sequence completes, click the **Done button** at the bottom of the *Work Probing screen* and Carbide Motion will set your zero location for X, Y, and Z.
11. Remove the BitZero V2 from your stock and the grounding clip from the spindle.

Now your Nomad 3 knows exactly where to begin machining from.

Load the G-code File

After setting job zero, you're ready to load the G-code file.

1. In Carbide Motion, click **Run** in the menu bar to open the *Run screen*.

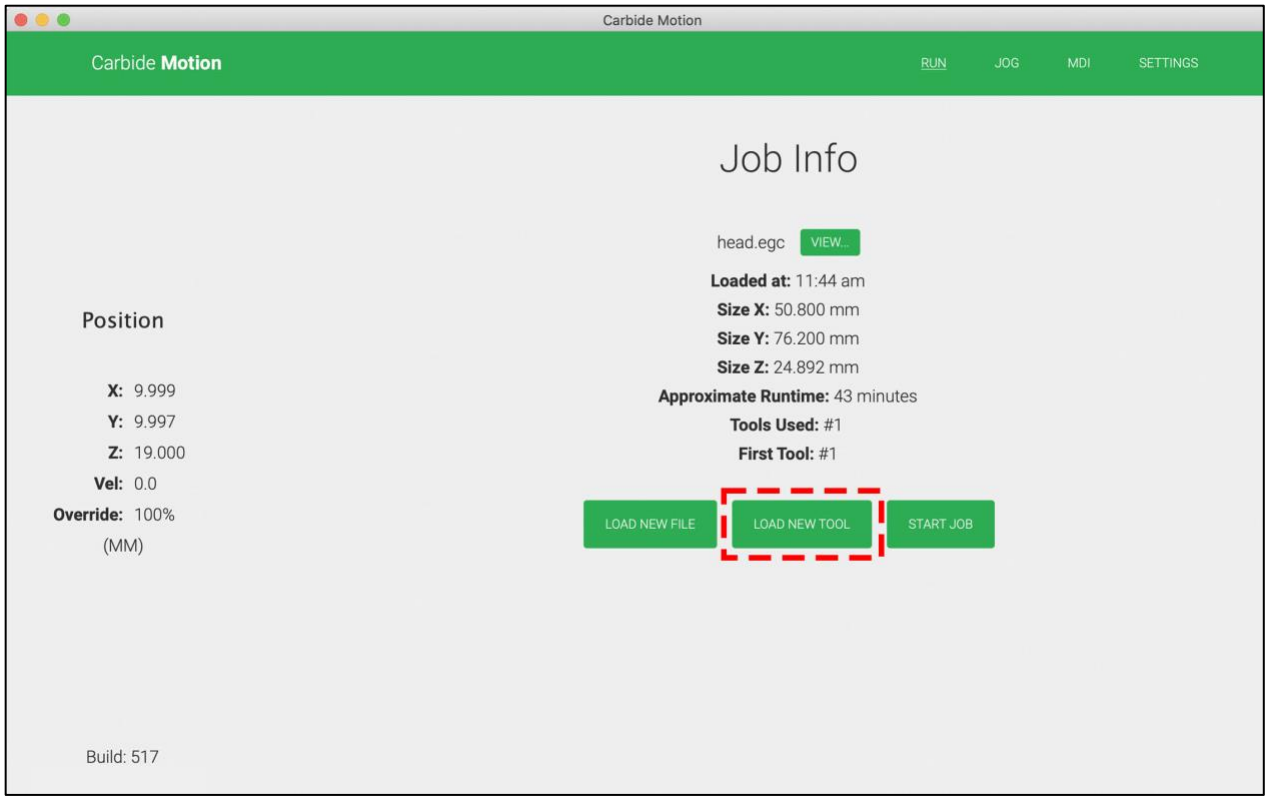


2. Click the **Load New File button**.
3. Browse to wherever you saved your **.egc file** and click the **Open button**.
4. Once the .egc file loads, you will see the approximate runtime, tools used, and other job info.

Install the End Mill

Now you need to load the end mill for this project, the ⅛" ball mill.

1. Click the **Load New Tool button**.



2. Install the ⅛" ball mill.

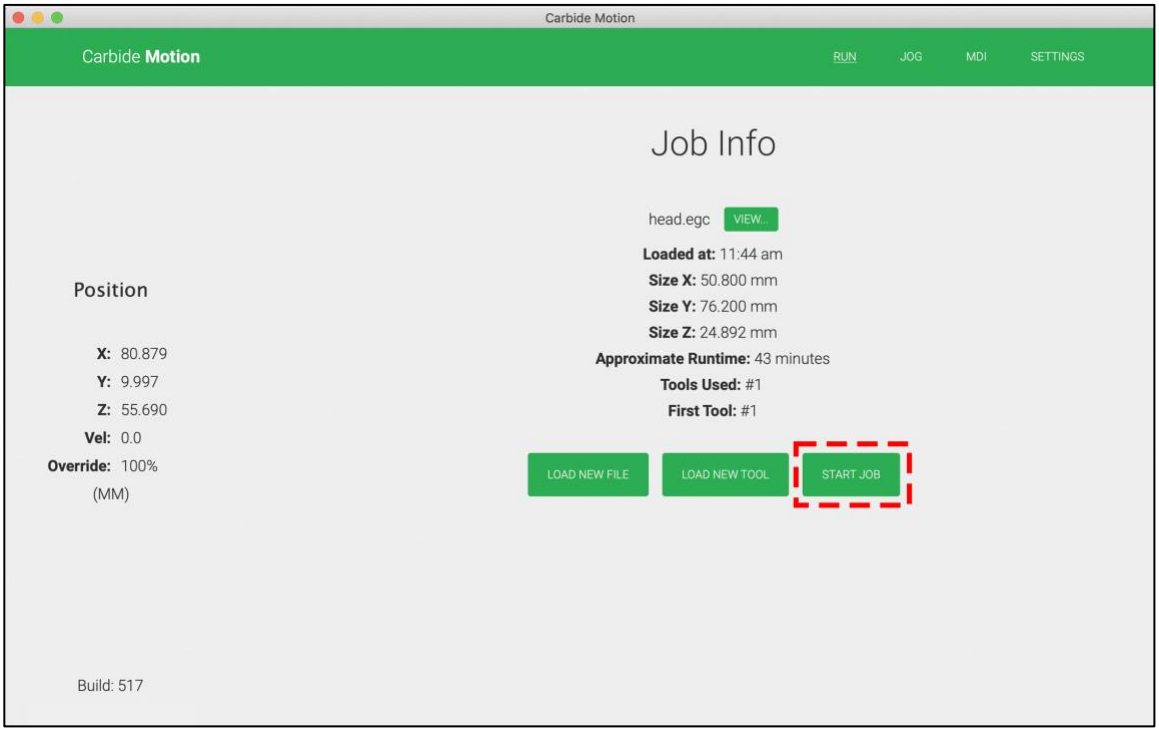
NOTE: Make sure not to insert the end mill any more than ¾" into the collet. Inserting the end mill too far into the collet could cause the collet nut or chip fan to crash into or ding the part, as the part is nearly 1" deep and ⅛" end mills are quite short.

3. Install the chip fan, if desired.
4. Close the machine door.
5. Then, click the **Resume button** in Carbide Motion.

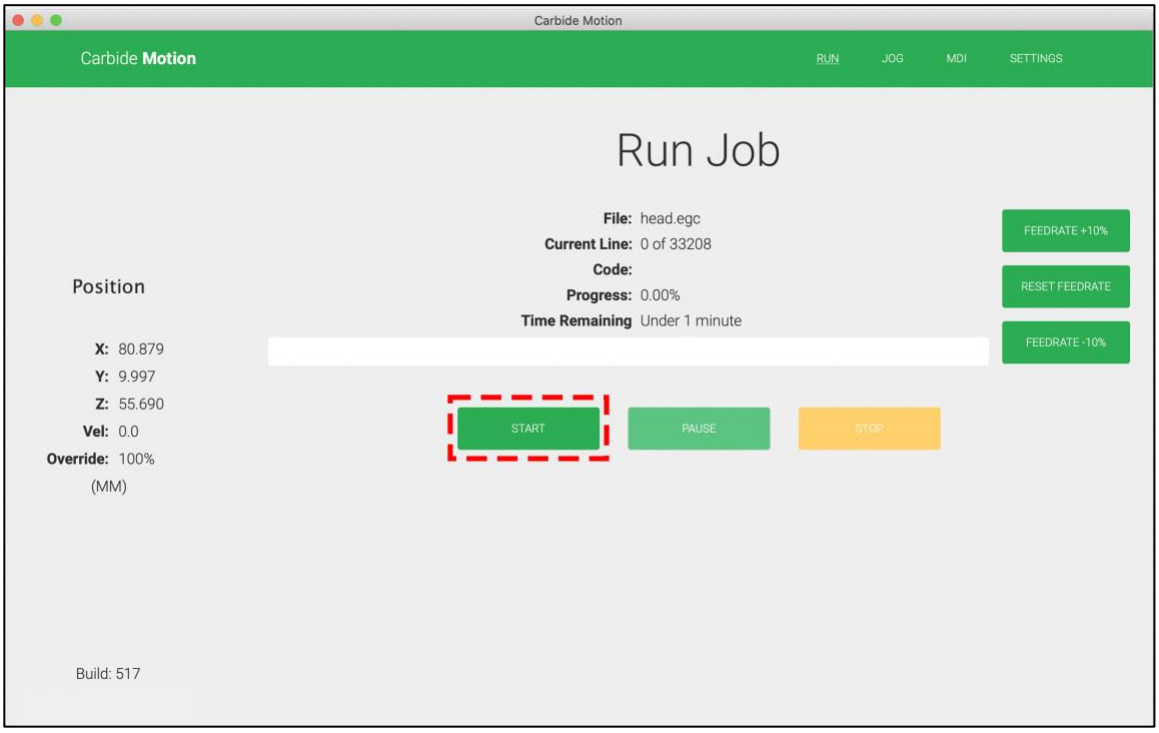
Run the G-code File

Now, we're ready to start running the job.

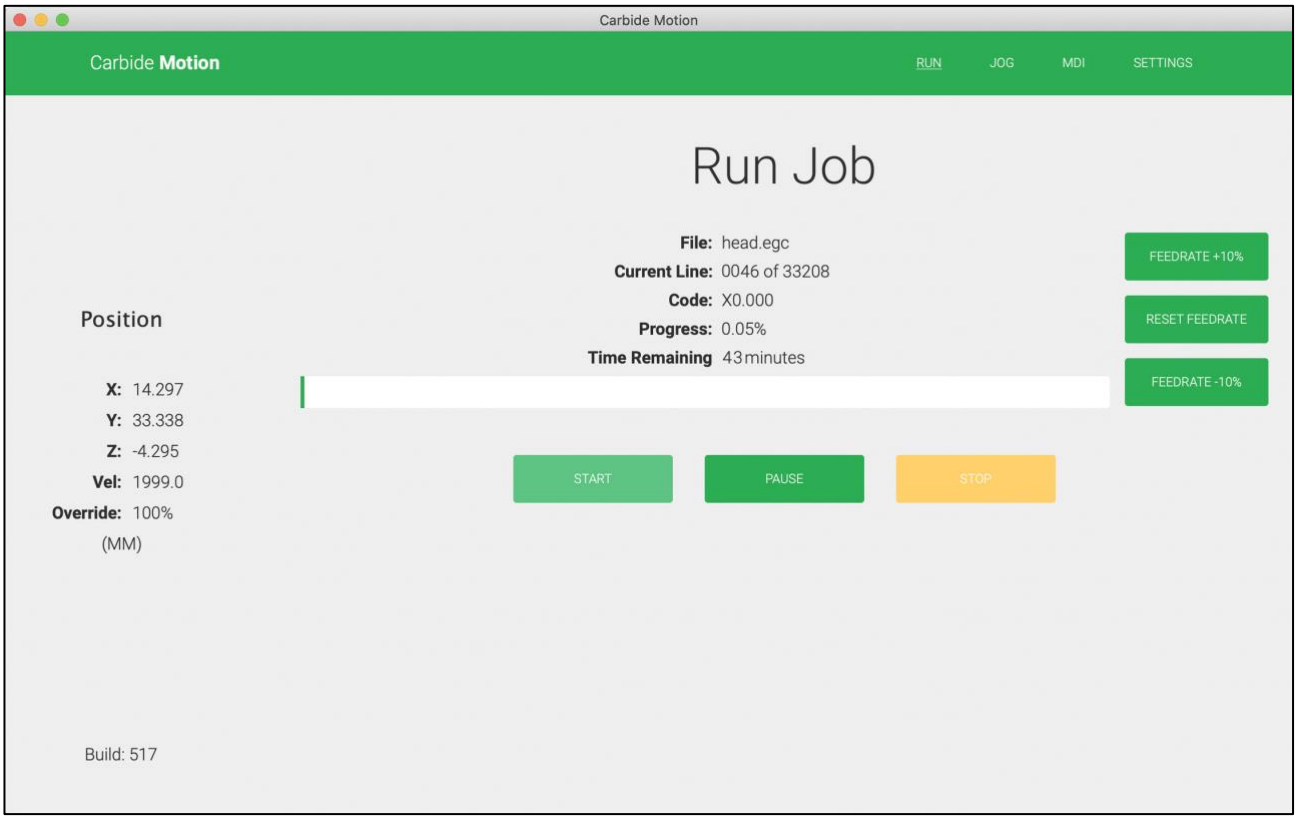
1. Click the **Start Job button**.



2. Then, click the **Start button** to begin the startup sequence.



3. During the startup sequence, Carbide Motion will prompt you to make sure the correct tool is installed in the spindle. If it's not, this is your last chance to correct that.
4. Click the **Resume button** once you have confirmed that the correct tool is installed and milling will begin. The job will take about 43 minutes from start to finish.



Remove the Part from the Wasteboard

When the program is complete, the spindle will stop automatically and the machine will home.

1. Once the table moves to the front of the machine, carefully pry the part off the wasteboard.
2. Remove the double-sided tape from the back of the part.

Conclusion

Hopefully you found it easy to generate a toolpath for a complex geometry.

If you're new to CNC machining then the best advice we can give is that you take the time to experiment with the toolpath values entered above and see how the toolpaths machine. A little experimentation will get you up the learning curve very quickly.