

Nomad 3 Starter Project: Tool Organizer Tray

This project will help you get familiar with your new Nomad 3 and walk you through the steps for creating a bamboo tool organizer tray. The tray fits 20 end mills in their plastic tool packaging, 15 loose end mills or other hardware, two ER collets, and an assortment of tools in the open pocket in the front of the tray.

What You'll Need



- 6" x 7" x 1/2" bamboo plywood
- #102 – 1/8" square end mill
- Wasteboard
- M6 x 10mm screws (x5)
- 6mm hex key
- 17mm collet wrench
- 13mm spindle wrench
- Double-sided tape

Project Files

All of the files used to create this project in Carbide Create and run it in Carbide Motion can be downloaded below. We've also included files to create and run this project using Fusion 360, if you would like to use that instead; however, the instructions in this guide only cover Carbide Create and Carbide Motion.

The zip file contains:

- Bamboo Tray.svg — Saved design ready to create toolpaths in Carbide Create **OR** you may use this file to edit the tray design in a vector graphics editor (*optional*)
- Bamboo Tray (Create).c2d — Saved design **and** toolpaths in Carbide Create
- Bamboo Tray (Create Gcode).nc — Saved G-code to run the job in Carbide Motion
- Bamboo Tray (Fusion).f3d — Saved design and toolpaths in Fusion 360 (*optional*)
- Bamboo Tray (Fusion Gcode).nc — Saved G-code to run the job in Fusion 360 (*optional*)
- Bamboo Tray Vectors.dxf — For editing the design in a vector graphics editor (*optional*)

Download the Nomad3StarterProjectFiles.zip file here:

<https://my.carbide3d.com/files/Nomad3StarterProjectFiles.zip>

Required Programs

Carbide Create

This project was made with Carbide Create v5—if you haven't downloaded the program yet or don't have the latest version, head over to the download page and grab a copy.

Carbide Create is a cross platform CAD/CAM program that is free to use and highly compatible with most CNC machines on the market.

[Get Carbide Create 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](#)

Optional Programs

Inkscape

If you'd like to edit the tray design SVG, you are welcome to do so in a vector graphic program such as Inkscape or Adobe Illustrator. Inkscape is a free, open-source graphics editor for vector files. Use Inkscape to create and edit vector files that can then be used in Carbide Create.

[Get Inkscape 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's enclosure is there for two reasons: to contain the mess and to protect the user from broken cutters and flying stock.

The end mills, or cutters, 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.

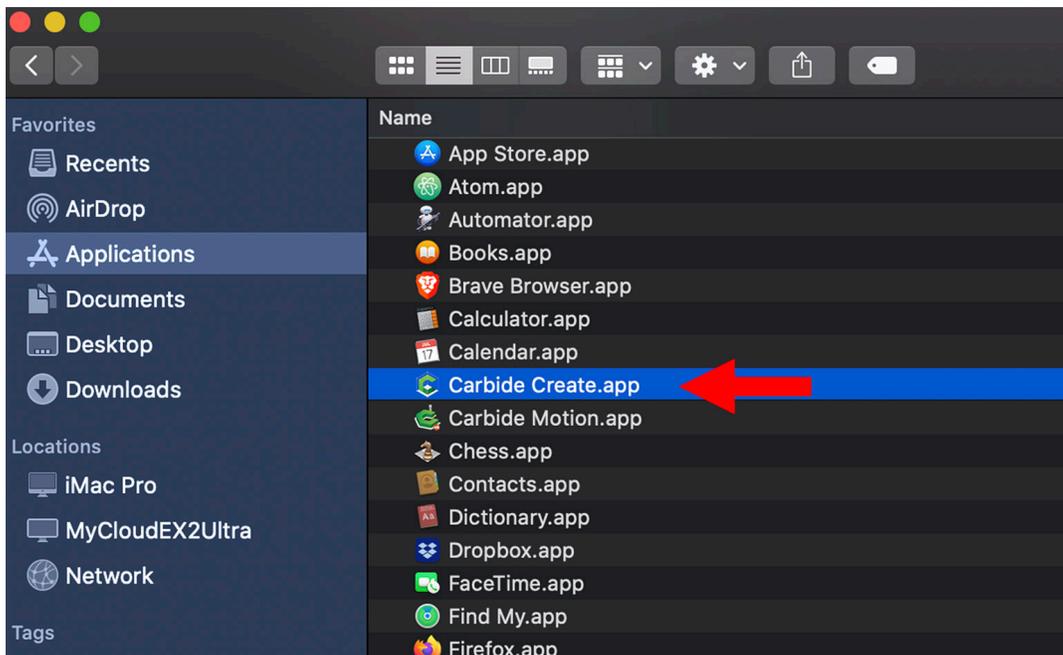
In summary, always close the cover when the machine is running.

Download and Install Files and Programs

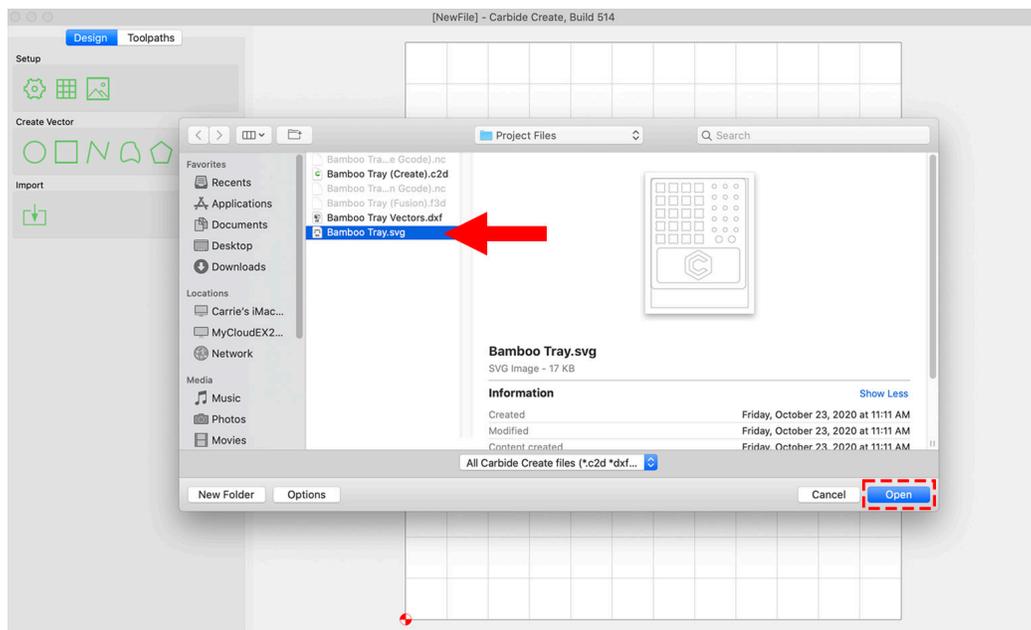
1. Download and install Carbide Create and Carbide Motion.
2. Download the project zip file (in Project Files above).
3. Download Inkscape, if you would like to edit the tray design.

Set up the Project in Carbide Create

1. Launch Carbide Create.



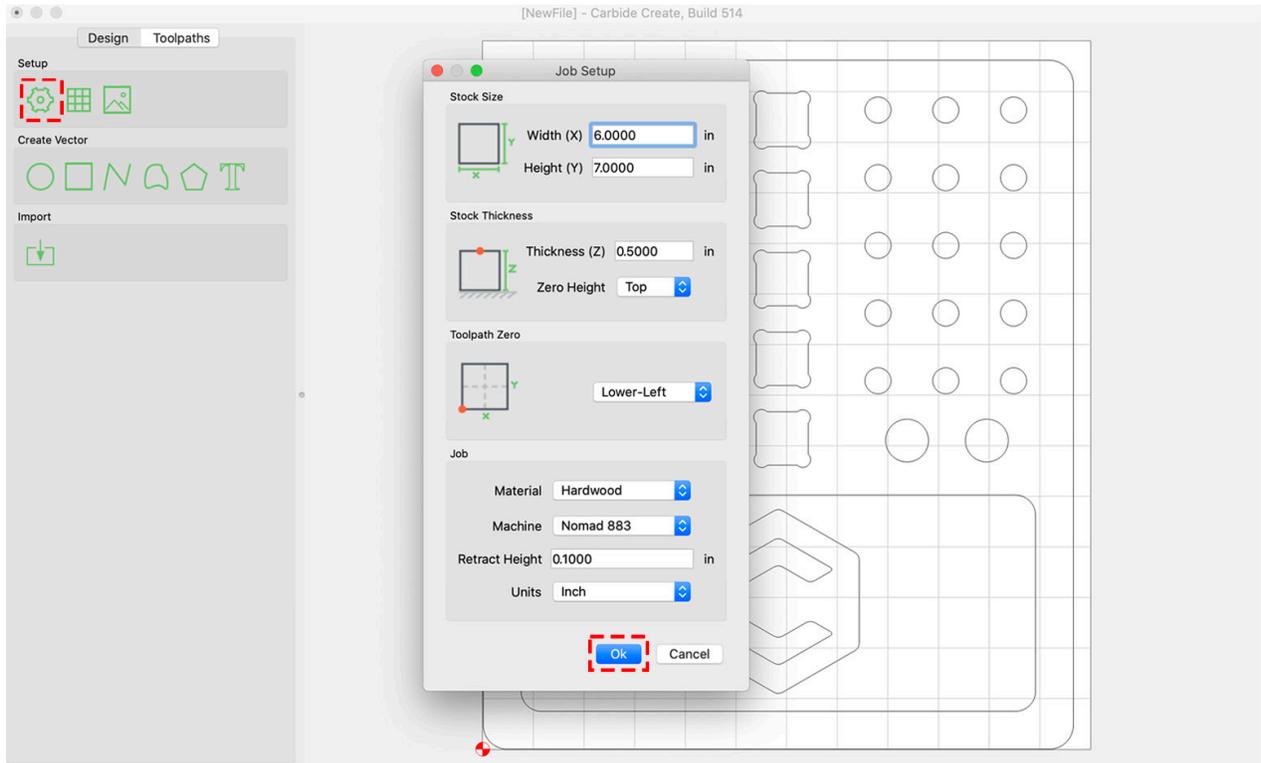
2. In Carbide Create, go to **File > Open** and browse to where you saved the project files.
3. Select the **Bamboo Tray.svg** file and click the **Open** button.



4. When the file loads, you will see the design for our tool tray.

Designs can be created here in Carbide Create with the drawing and transform tools located in the *Design tab*. Or, designs can be created in a vector graphics editor, such as Inkscape, and imported into Carbide Create as an SVG file.

5. Click the **gear icon** at the top-left of your screen to open the *Job Setup window*. Here you can see all of the parameters for the job, which were set when the design was originally created in Carbide Create:



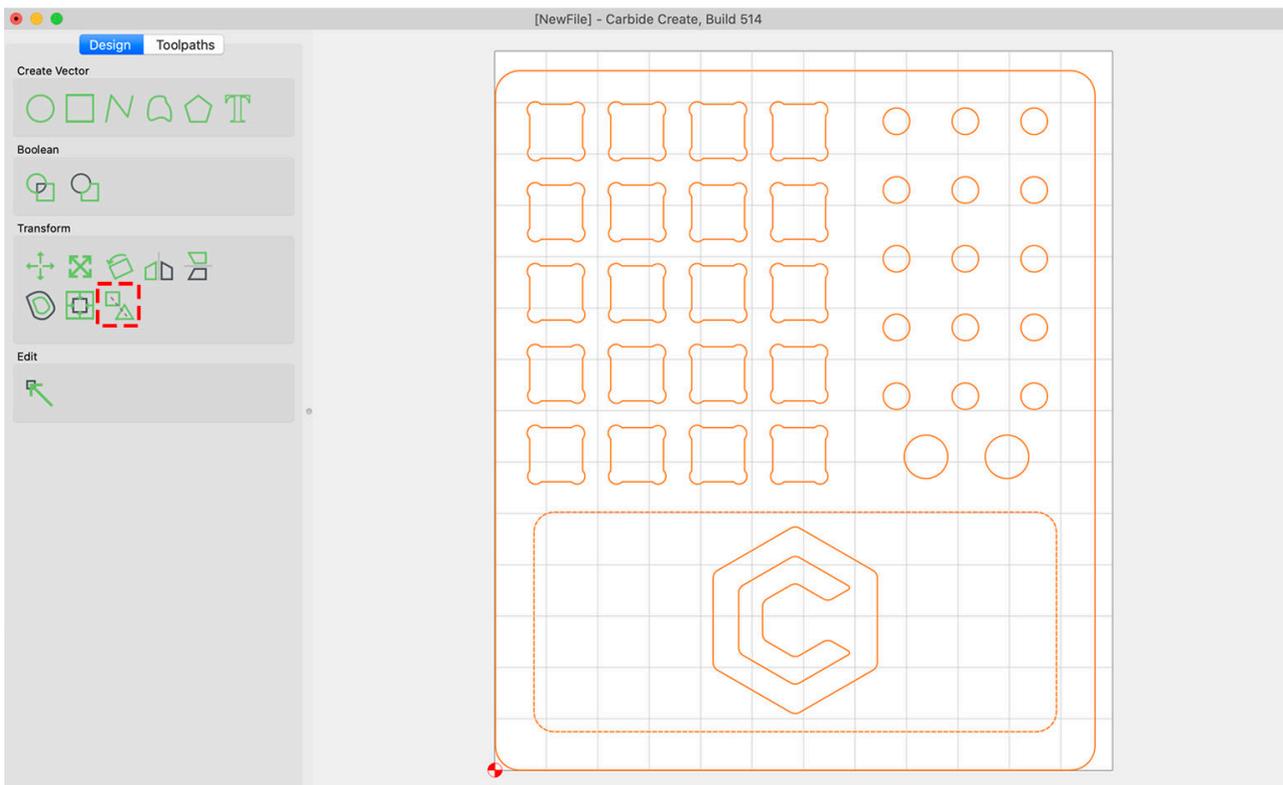
- a. Stock Size is set to **6" x 7"** to match our stock.
 - b. Stock Thickness is set to **1/2"** to match our stock.
 - c. Zero Height is set to **Top**.
 - d. Toolpath Zero (the origin or starting point of your project, marked on the canvas by a red and white circle) is set to **Lower-Left**.
 - e. Material is set to **Hardwood**.
 - f. Machine is set to **Nomad 883**.
 - g. Retract Height (how high above the material the machine will retract during rapid moves. Set to a height which clears all clamping, if any) is set to **0.1000"**.
 - h. Units are set to **Inch**.
6. If you want to cut this project using a different piece of wood than the bamboo plywood included in your kit, you'll need to change the job settings. Otherwise, all of the parameters are correct, so go ahead and click **Ok** to close the Job Setup window.

Before we start creating toolpaths, we want to center the boundaries of the tray in the workspace, not butted up against any of the walls of our canvas. This will give us a little margin for error when machining our tray since the tray is slightly a narrower than our stock.

7. Select all of the design features.

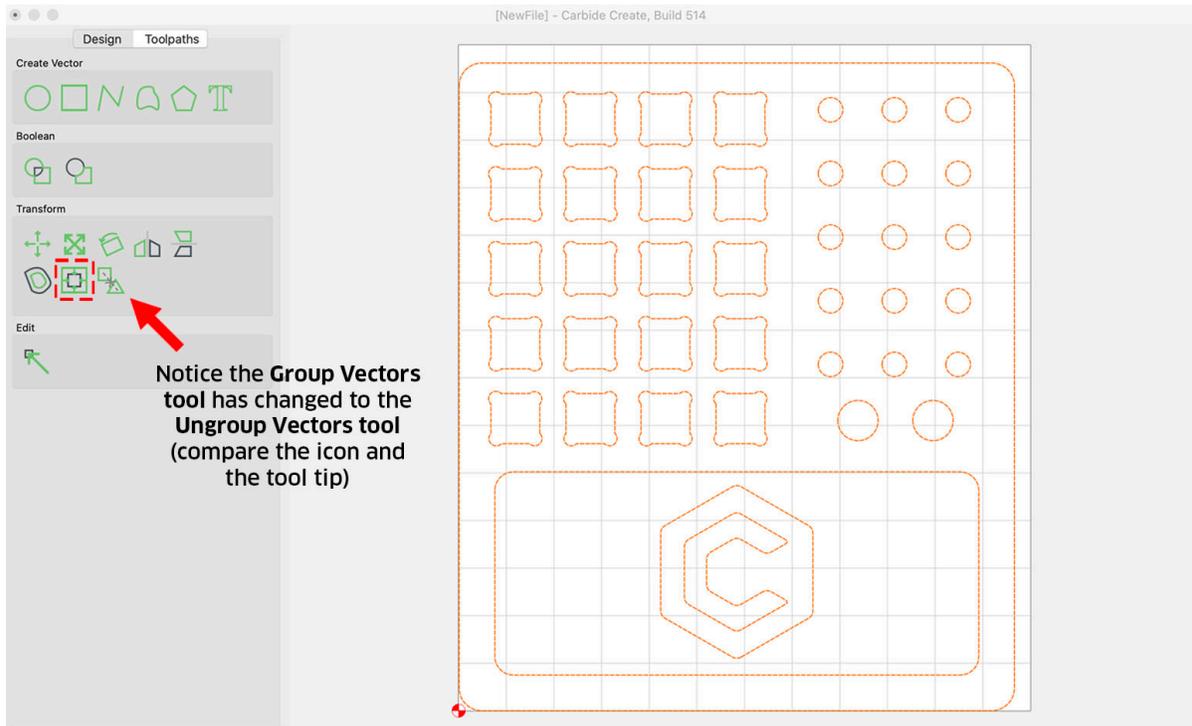
Notice when a feature is selected, its outline becomes orange.

PRO TIP: To select multiple features (shapes/text/vectors in the design), click and drag around all of the features you want to select. If you need to add a feature or two to your bulk selection, hold the shift key while clicking on each new feature.

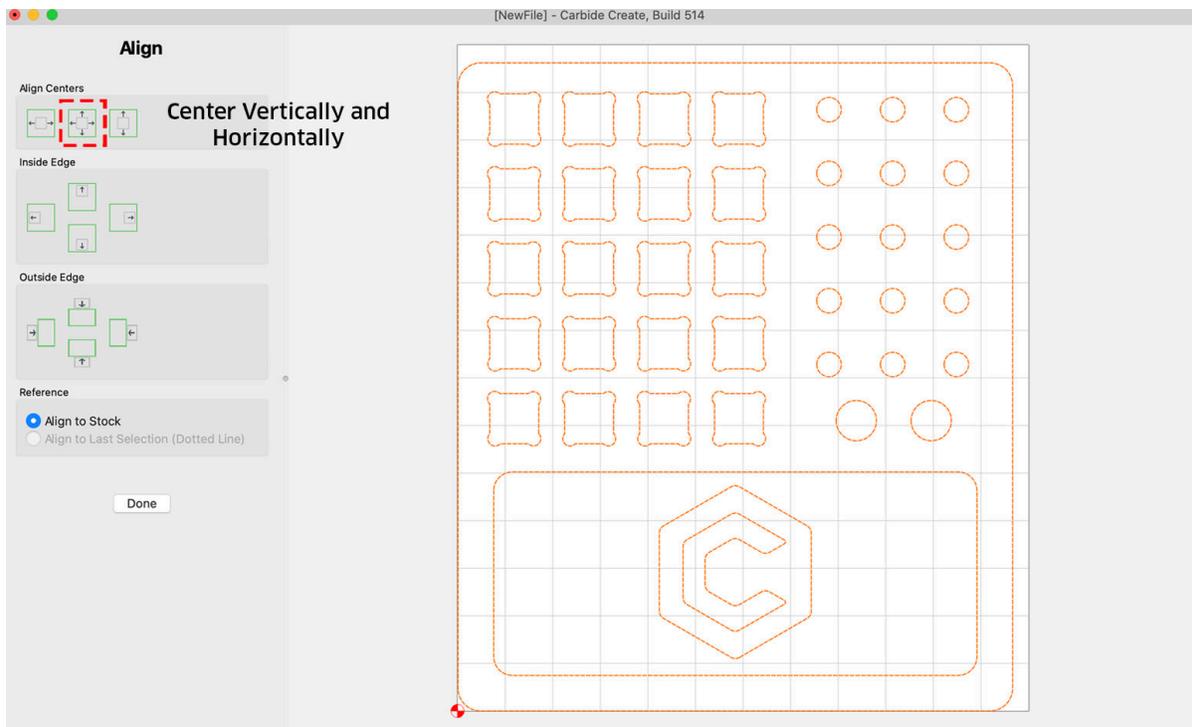


8. Click the **Group Vectors** tool to group all of the features together.
Notice as soon as you click the tool icon, that both the icon itself and the tool tip change from the "Group Vectors tool" to the "Ungroup Vectors tool".

9. Click the **Align Vectors** tool.



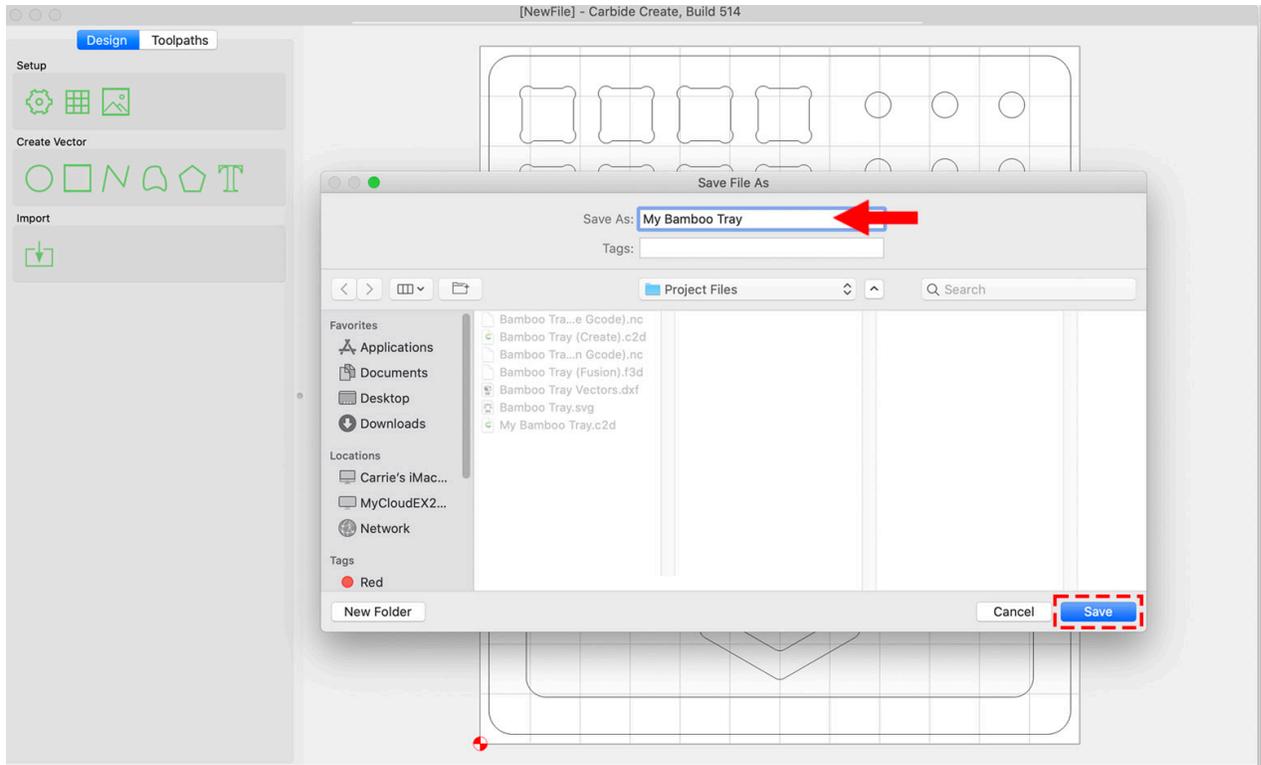
10. Click the **Align Vertical and Horizontal Centers** tool, which will center the design on the canvas.



11. Click the **Ungroup Vectors** tool, to ungroup the features.

- Click anywhere on the canvas to unselect the selected features.
- Now is a great time to save your file, in case you need to change something in the design later. Go to **File > Save As**, input a file name, such as My Bamboo Tray, and click **Save**. Carbide Create will save your file as a .c2d file.

NOTE: .c2d is the format that Carbide Create uses to save design and toolpath information for future editing. Once we're finished setting up our toolpaths, we'll save the generated G-code as a .nc file.



- The design is now ready to go, unless you wanted to make any adjustments, such as changing the logo or adding text.

A Quick Aside About Carbide Create Shortcuts

As with any program, shortcuts can help you quickly and easily navigate Carbide Create. Here are a few to make your first project go smoothly.

To adjust your view of the canvas, go to **View** in the menu bar:

- **Reset View:** Re-centers the canvas on the screen.
- **Snap to Grid:** Forces all control points (nodes) to snap to the nearest grid point. This can be helpful when aligning features with the grid or each other.

Mouse and Trackpad Shortcuts

With a mouse or trackpad, you can zoom in/out and move the canvas around. You can also quickly select features.

Mouse	Trackpad
Scroll wheel: Zoom in and out	Drag with two fingers: Zoom in and out
Right-click and drag: Pan/move the canvas	Click and drag with two fingers: Pan/move the canvas
Left-click: Select a feature	Click/tap with one finger: Select a feature
Left-click + SHIFT: Select multiple features	Click/tap with one finger + SHIFT: Select multiple features
Left-click and drag: Create a selection box to select any feature that falls within the box boundary	Click and drag with one finger: Create a selection box to select any feature that falls within the box boundary

Keyboard Shortcuts

With keyboard shortcuts, you can quickly and easily duplicate, select, and delete features and more.

Windows	Mac
CTRL + S: Save file SHIFT + CTRL + S: Save file as...	CMD + S: Save current file SHIFT + CMD + S: Save file as...
CTRL + Z: Undo	CMD + Z: Undo
CTRL + X: Cut/delete selected feature(s)	CMD + X: Cut/delete selected feature(s)
CTRL + C: Duplicate selected feature(s)	CMD + C: Duplicate selected feature(s)
CTRL + A: Select all features	CMD + A: Select all features
SHIFT + Click: Select multiple features	SHIFT + Click: Select multiple features

Set Up the Toolpaths

Now we need to create the toolpaths that tell the Nomad 3 how to cut out our project.

Quick Overview of Toolpaths

When your CNC machine is running, it is executing G-code. G-code is created by defining toolpaths in Carbide Create. Think of toolpaths as your visual guide to “programming” your CNC machine. There are 4 basic types of toolpaths:

- **Inside Cut:** Offsets the cutter to the inside of the feature
- **Outside Cut:** Offsets the cutter to the outside of the feature
- **Pocket:** Clears everything inside the feature
- **Engrave:** Cuts directly on the line (no offset)

We use the term *offset* to describe the toolpaths. In this case, offsetting means to move the cutting edge of the end mill to one side of the line or the other.

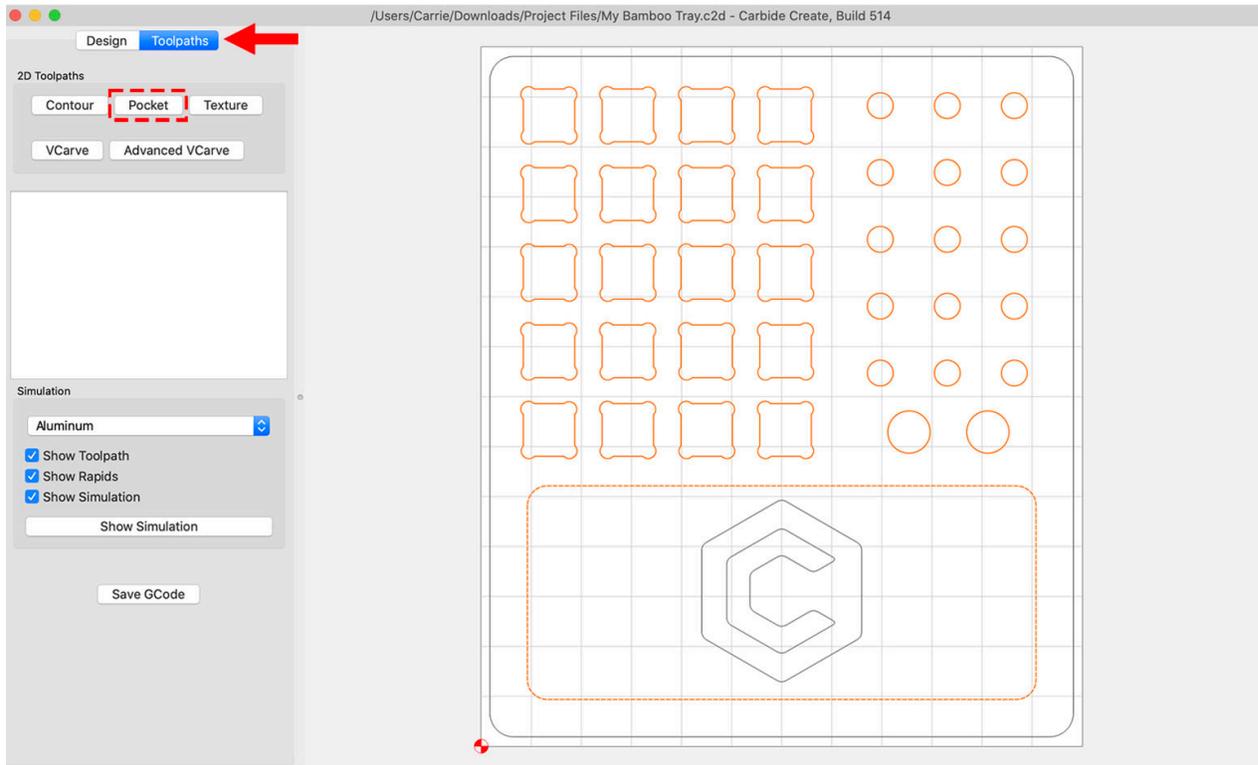
Setting Up the Toolpaths

NOTE: The Bamboo Tray (Create).c2d file included in the project files is the completed design and toolpaths in Carbide Create, in case you want to reference, or work from, the completed design.

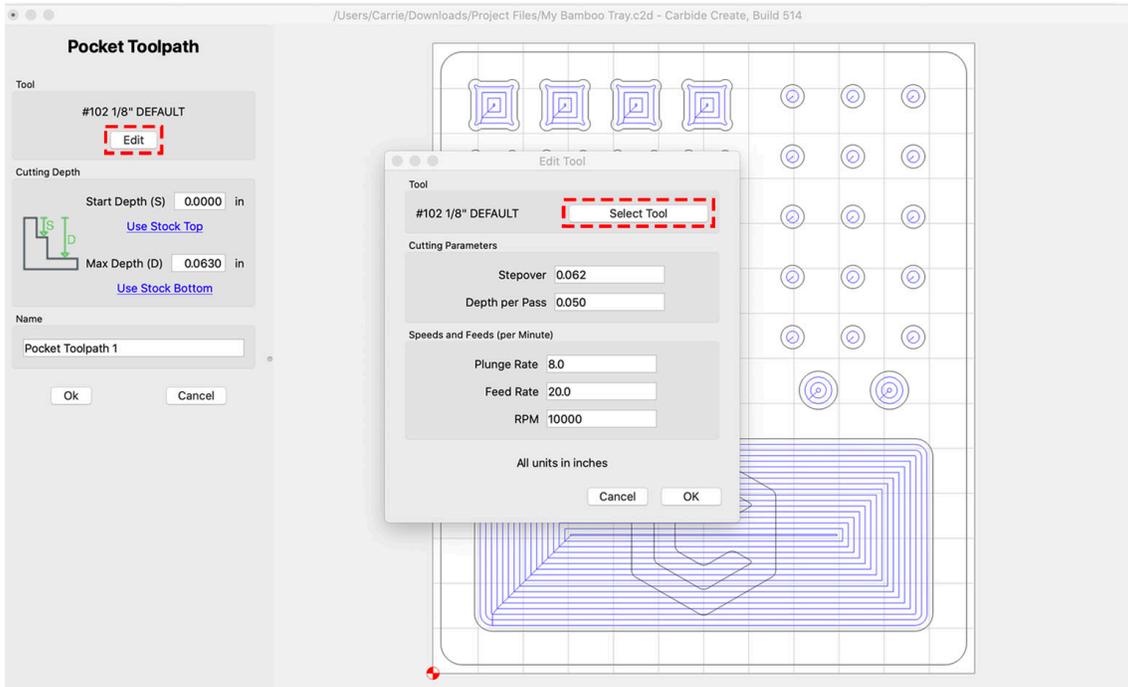
First, we want to create a toolpath for all of the tray features, *except* the outer-most feature and the logo. All of these features will be pocketed to the same depth using an $\frac{1}{8}$ ” end mill:

1. Select all of the features except the logo and the perimeter rectangle.
2. Click the **Toolpaths** button at the top of your screen to open the *Toolpaths* tab.

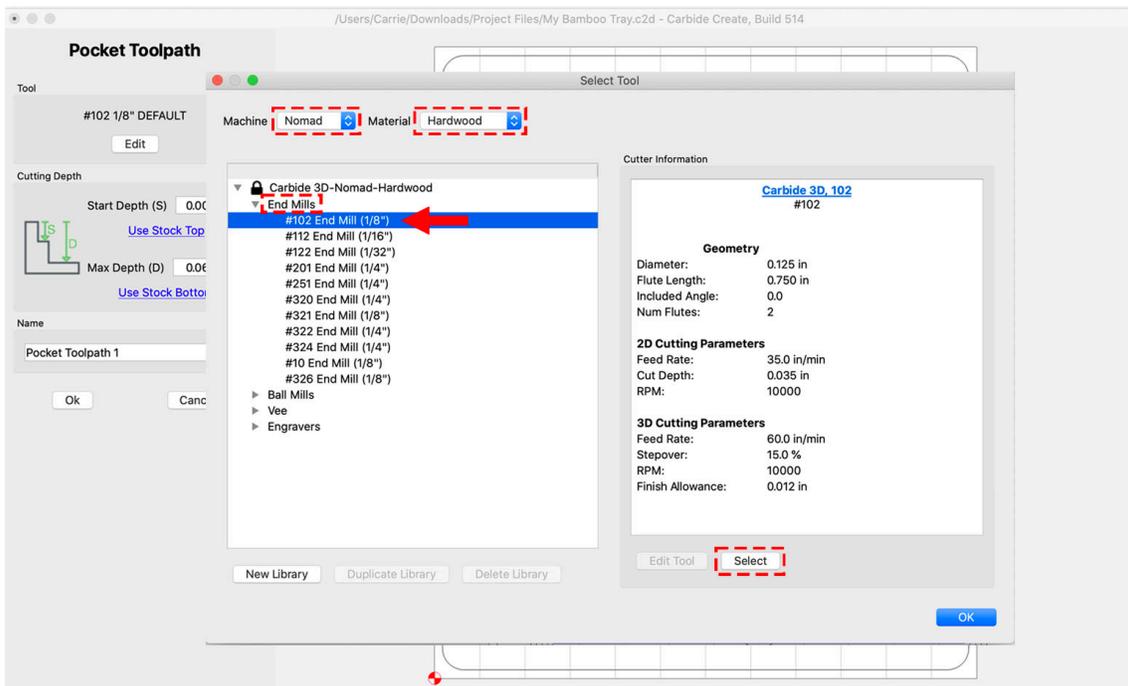
3. Click the **Pocket** button to create a pocketing toolpath.



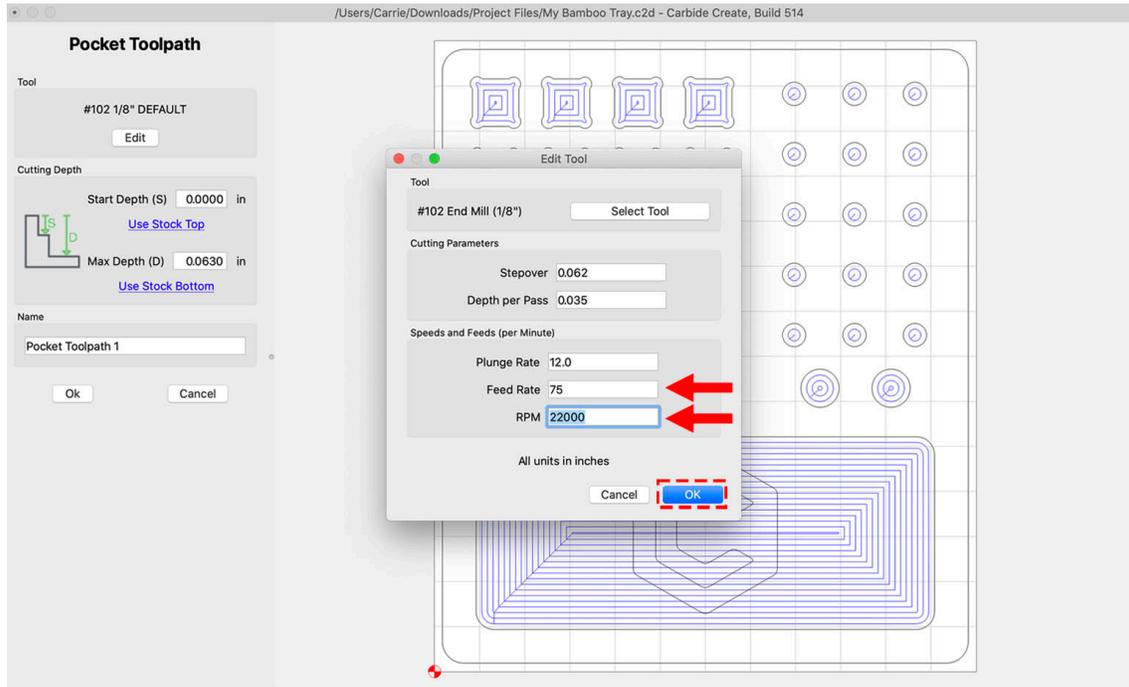
4. In the *Tool panel* at the top of the screen, click the **Edit** button.



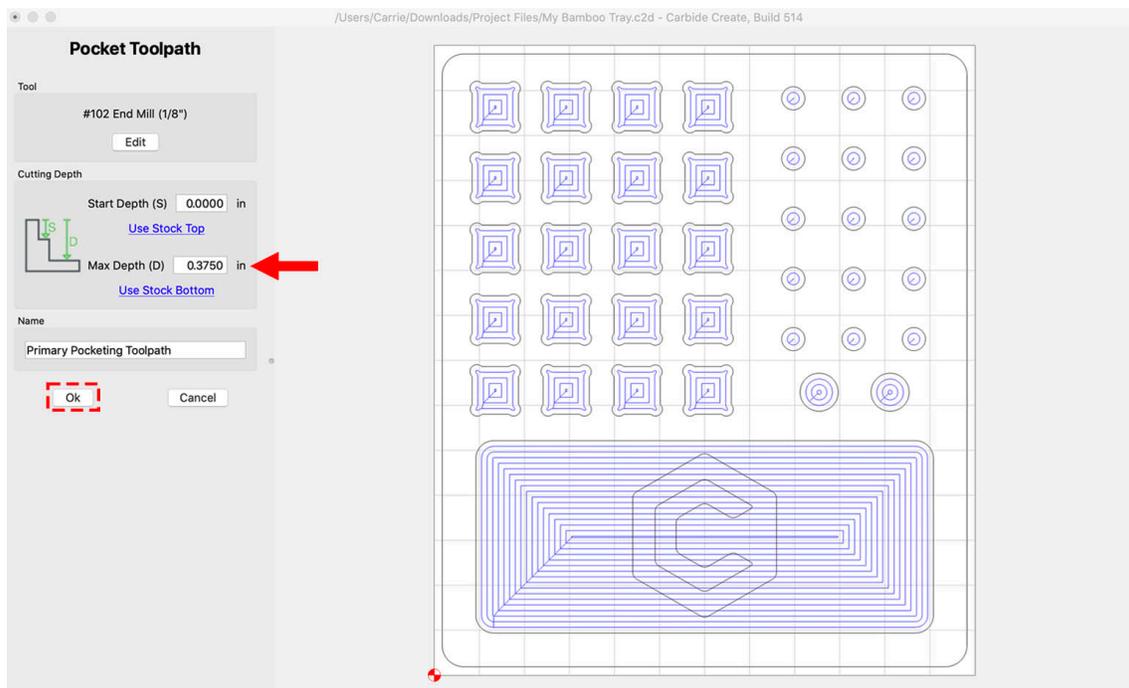
5. The *Edit Tool window* is where you can change the cutting parameters and edit the selected cutting tool. Click the **Select Tool** button to choose a new end mill.
- Select **Nomad** and **Hardwood** from the dropdown lists at the top of the *Select Tool window*.
 - Click **End Mills** to open the dropdown list, select **#102 End Mill (1/8")**, and click the **Select** button.



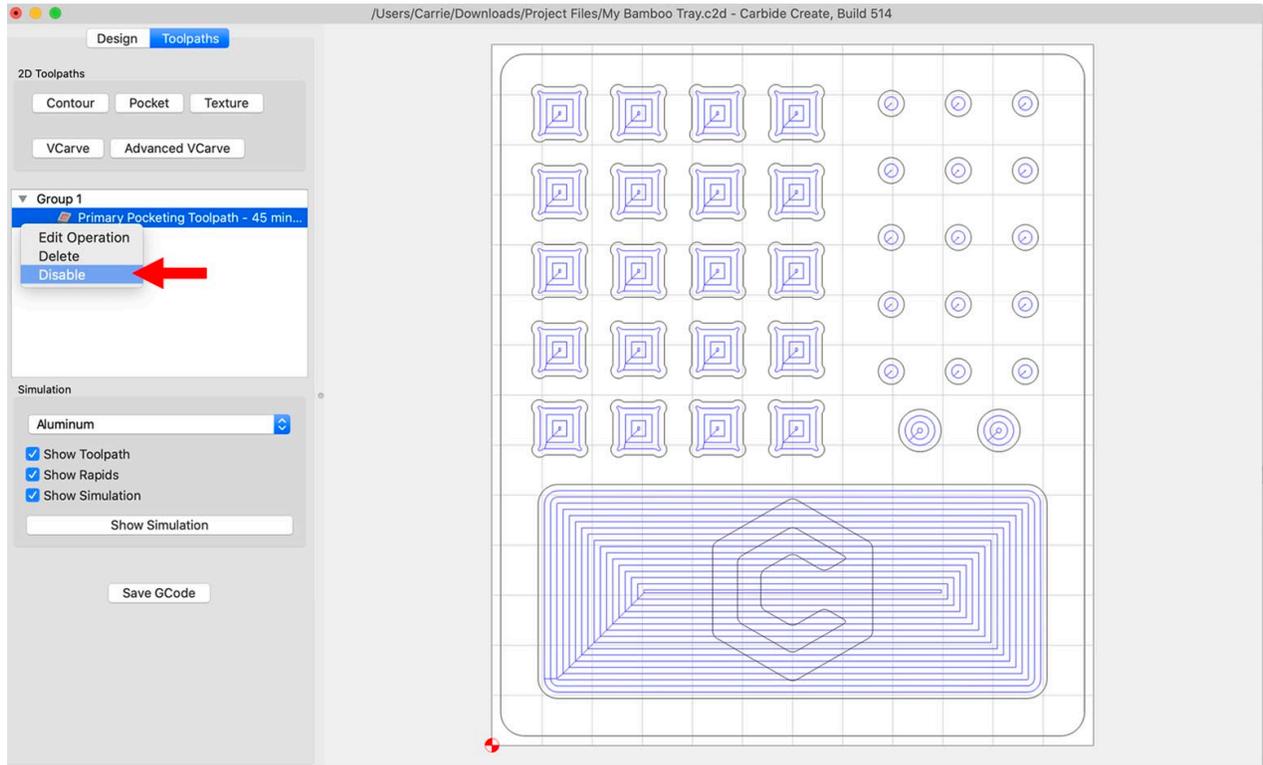
6. Change the Feed Rate to **75** and RPM to **22000**, and click **OK** to save the new cutting parameters.



7. We want the pockets cut down to a depth that is less than our material's thickness—so anywhere between from 0.3 – 0.4 inches will work great. We have set our Max Dept (D) to **0.3750**". Keep in mind, deeper pockets will take marginally longer to machine.
8. Name the toolpath **"Primary Pocketing Toolpath"**. Naming toolpaths helps remind us what each one does.
9. Click the **Ok** button to save and close the toolpath settings.



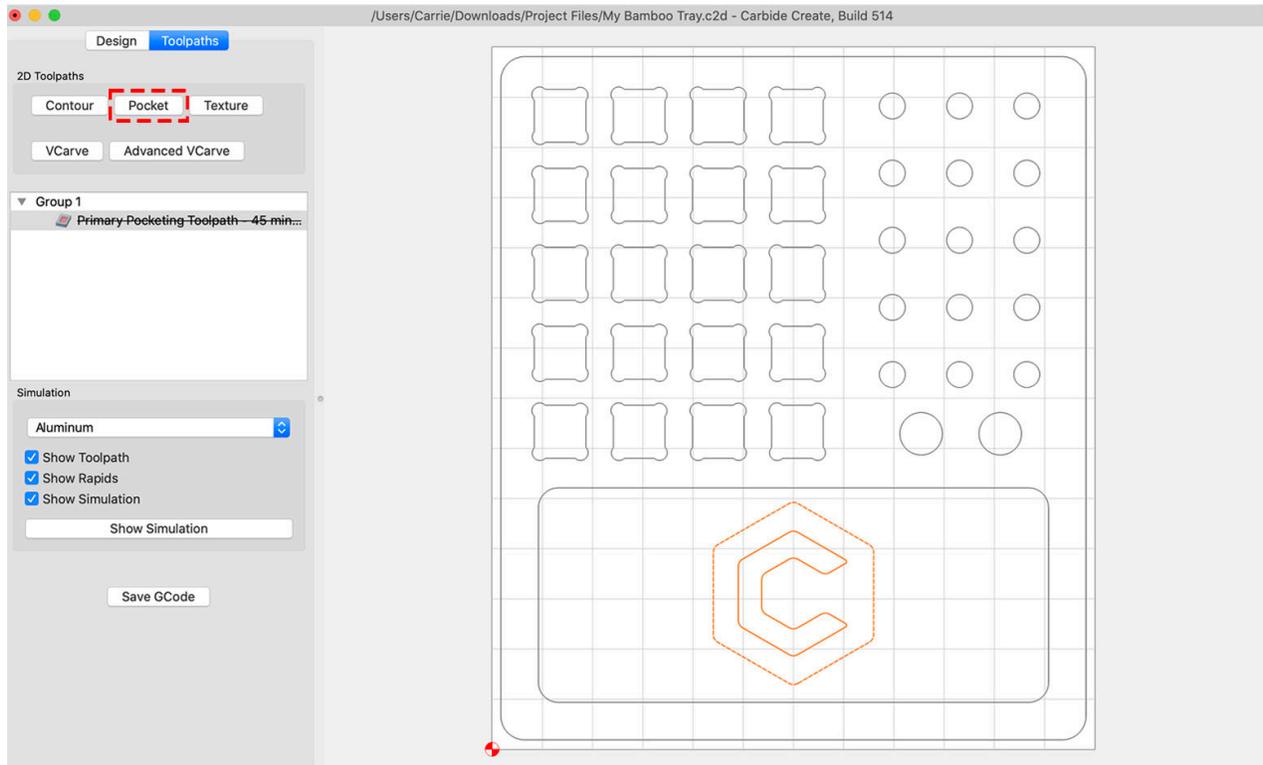
10. You'll see that our new Primary Pocketing Toolpath now appears in the *Toolpath List*, along with an approximate cutting time.
11. We want to disable the toolpath temporarily so we can easily see what we're working on when we create the next toolpath. **Right-click** on the toolpath name in the list and choose **Disable**.



Next, we want to create a pocketing toolpath for the Carbide 3D logo in the large tray pocket:

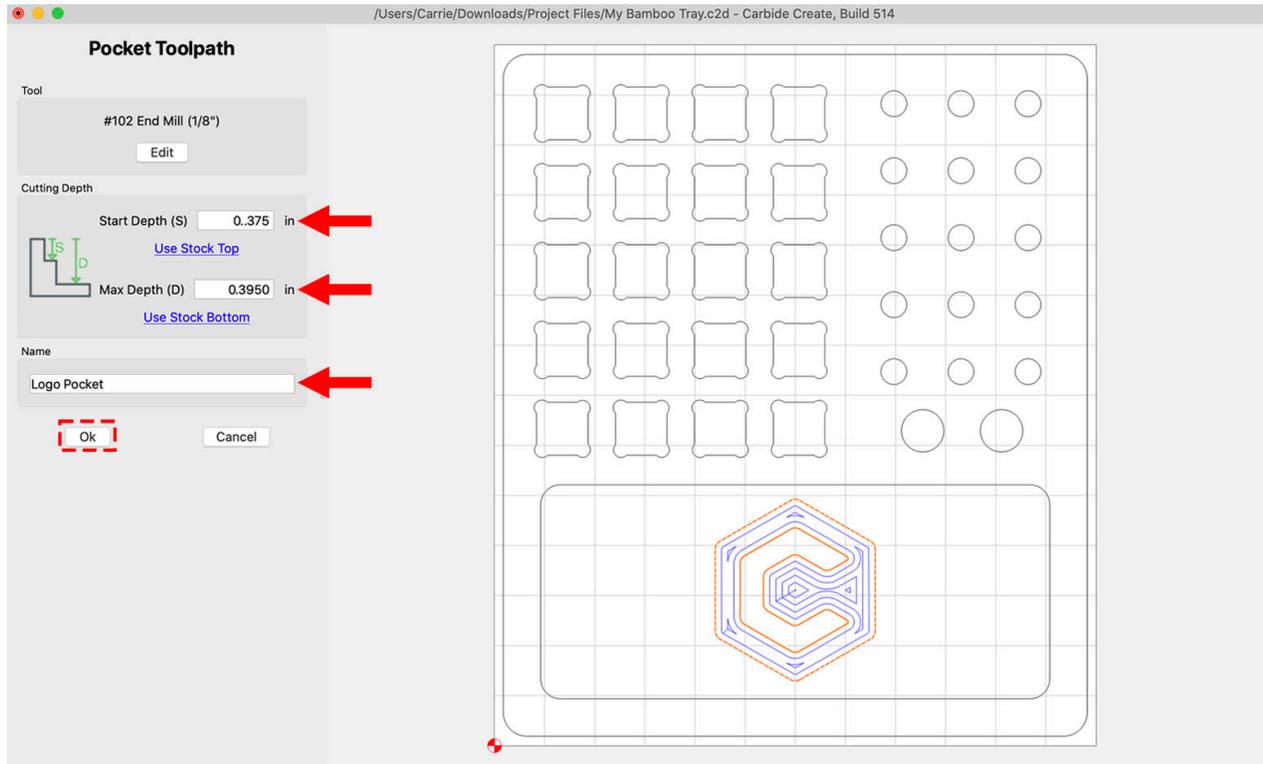
1. Select the C-shaped profile and the surrounding hexagon.
2. Click the **Pocket** button to create a pocketing toolpath.

Selecting two different contours where one is inside the other and then adding a pocket toolpath to them will machine away the area between the two contours.



NOTE: Carbide Create selects the previously used tool by default, so there's no need to change the tool for our final two toolpaths, since all three toolpaths use the $\frac{1}{8}$ " end mill.

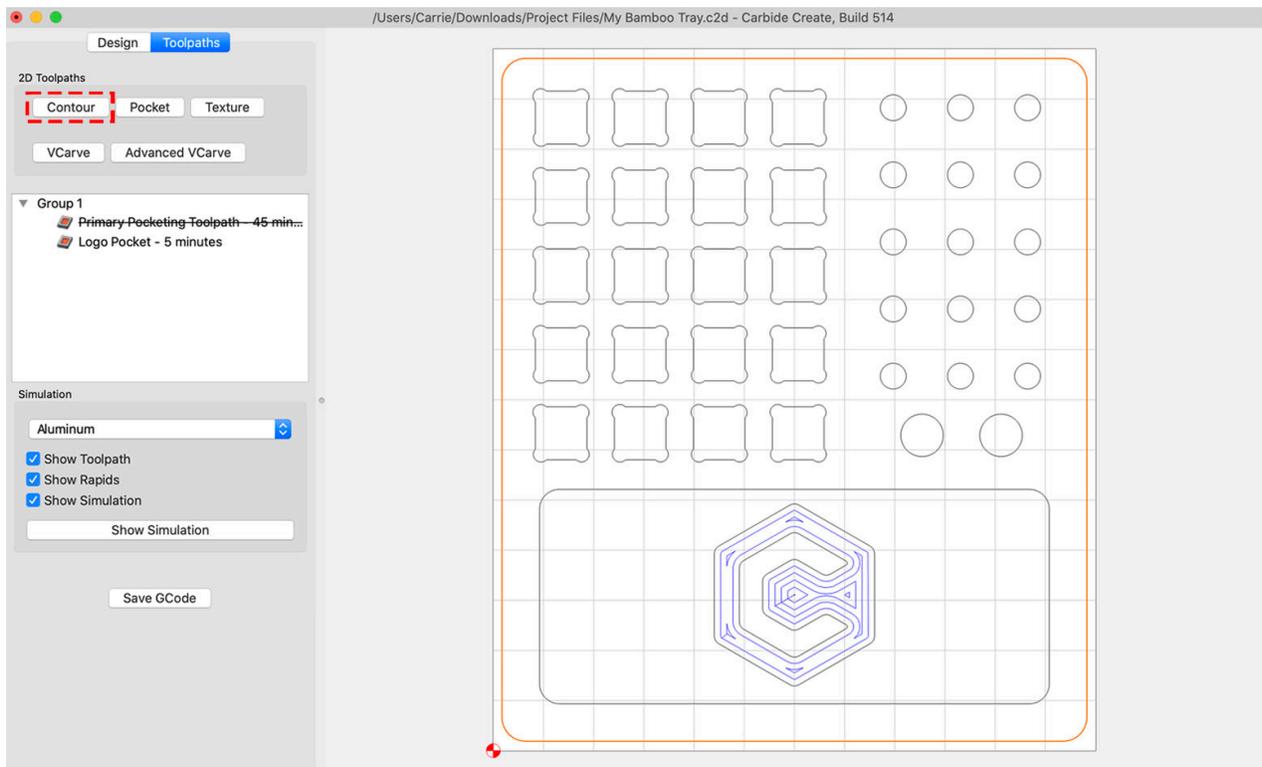
3. We want to cut the logo pocket to 0.02" deeper than the large main pocket it's within. So, we've set the Max Depth (D) to **0.3950"**.



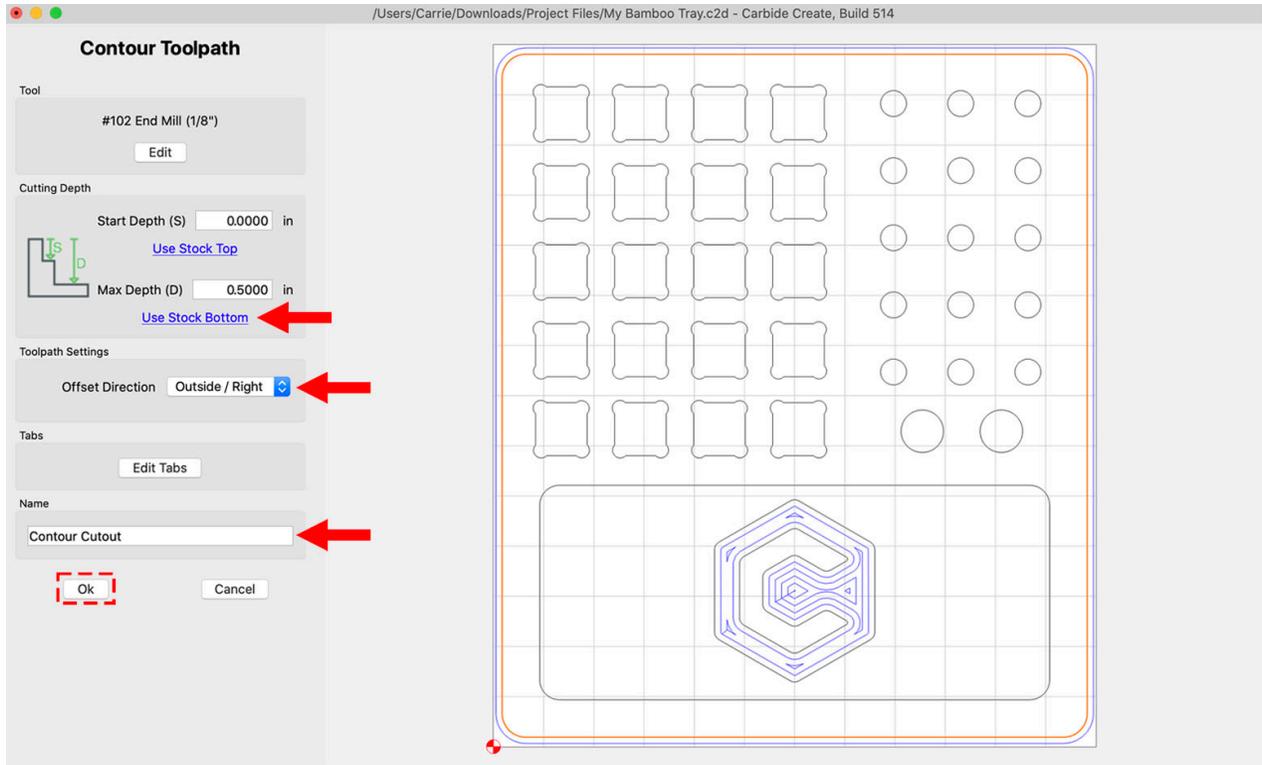
4. We also need to change the Start Depth (S) to the depth of the main pocket, which is **0.375"**.
This lets Carbide Create know that it doesn't need to worry about the material that was machined away previously. Note that these values are absolute, not relative. They don't add up, so we're not going to create a pocket that's 0.77" deep and goes all the way through our stock with these settings.
5. Name the toolpath **"Logo Pocket"**.
6. Click **Ok** to save the toolpath.

Finally, to cut out our tray we need to select the tray outline and apply a contour toolpath on the outside of it:

1. Select the perimeter feature and click the **Contour** button to create a contour toolpath.



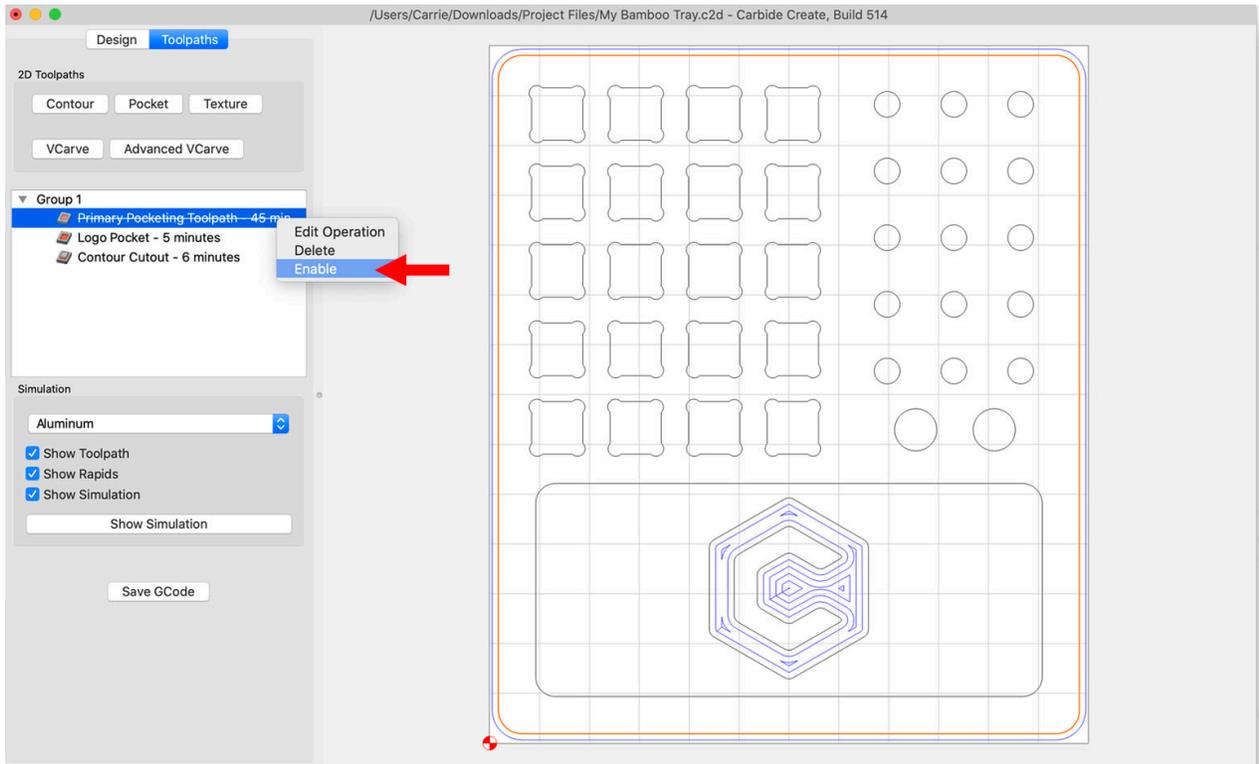
- We need to make sure that this toolpath will cut all the way through our stock, or 1/2" down. For the Max Depth (D) you can either input **0.5000"** or click **Use Stock Bottom**.
- Set the Offset Direction to **Outside/Right**, name the toolpath **"Contour Cutout"**, and click **Ok**.



Previewing the Simulation

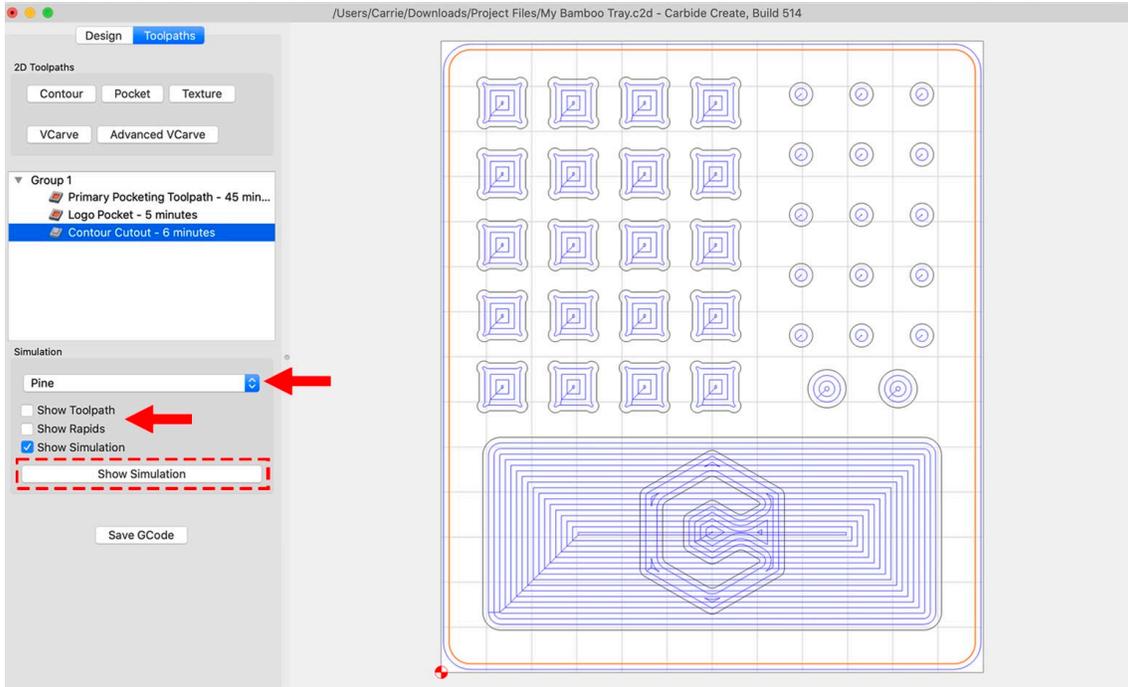
Don't forget we disabled the first toolpath so that it wouldn't visually interfere with the creation of the last two toolpaths. To re-enable the toolpath:

1. **Right-click** on the Primary Pocketing Toolpath and choose **Enable**.

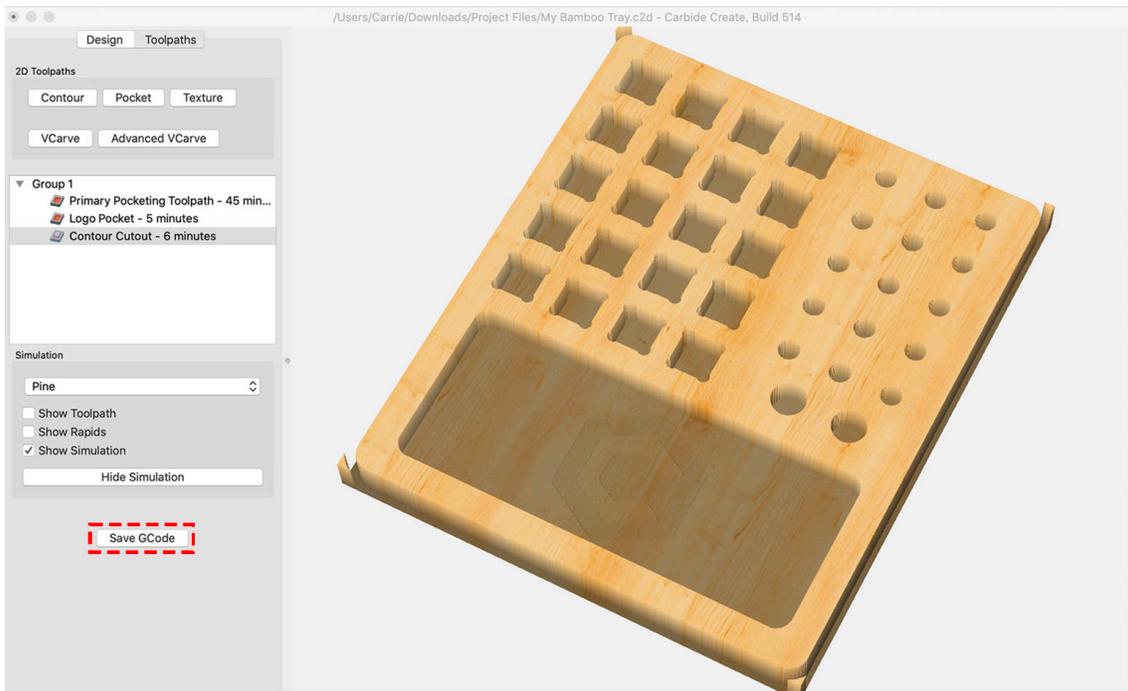


Now we can preview a simulation of the final tool tray:

1. Choose **Pine** from the material dropdown list.
2. Uncheck **Show Toolpath** and **Show Rapids** and click the **Show Simulation** button.



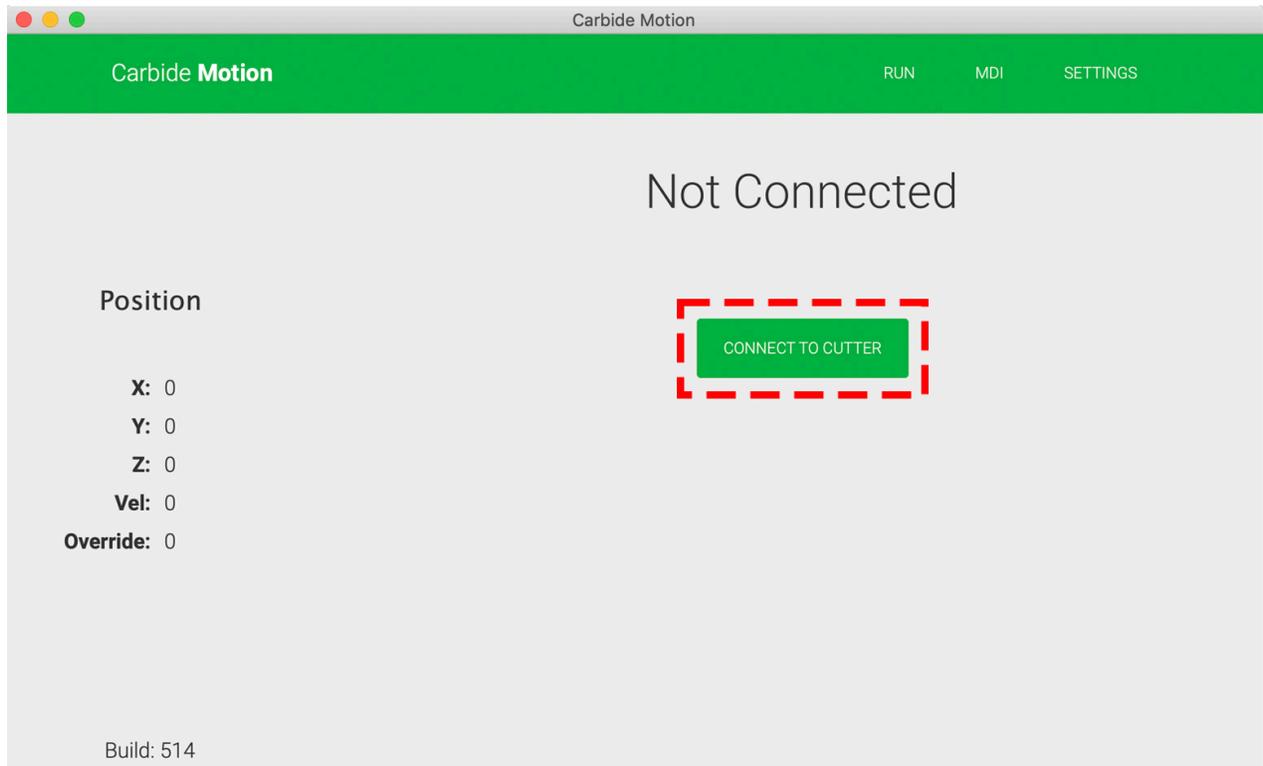
3. You can click and drag the simulation to angle the image for better viewing. If the simulation looks good, click the **Save G-code** button to export your G-code, .nc file, which we'll load into Carbide Motion later.
4. Press Ctrl+S (CMD+S on a Mac) to save your .c2d design file in case you need to make changes to it later.



Connect to the Nomad 3

The Nomad 3 is controlled by Carbide Motion. To connect your machine to Carbide Motion:

1. Power up your computer.
2. Connect the USB cable to your computer and to the port on the outside of the Nomad 3 enclosure (at back-right).
3. Start Carbide Motion 5 on your computer.
4. Power on your machine.
5. In Carbide Motion, click the **Connect to Cutter button**.

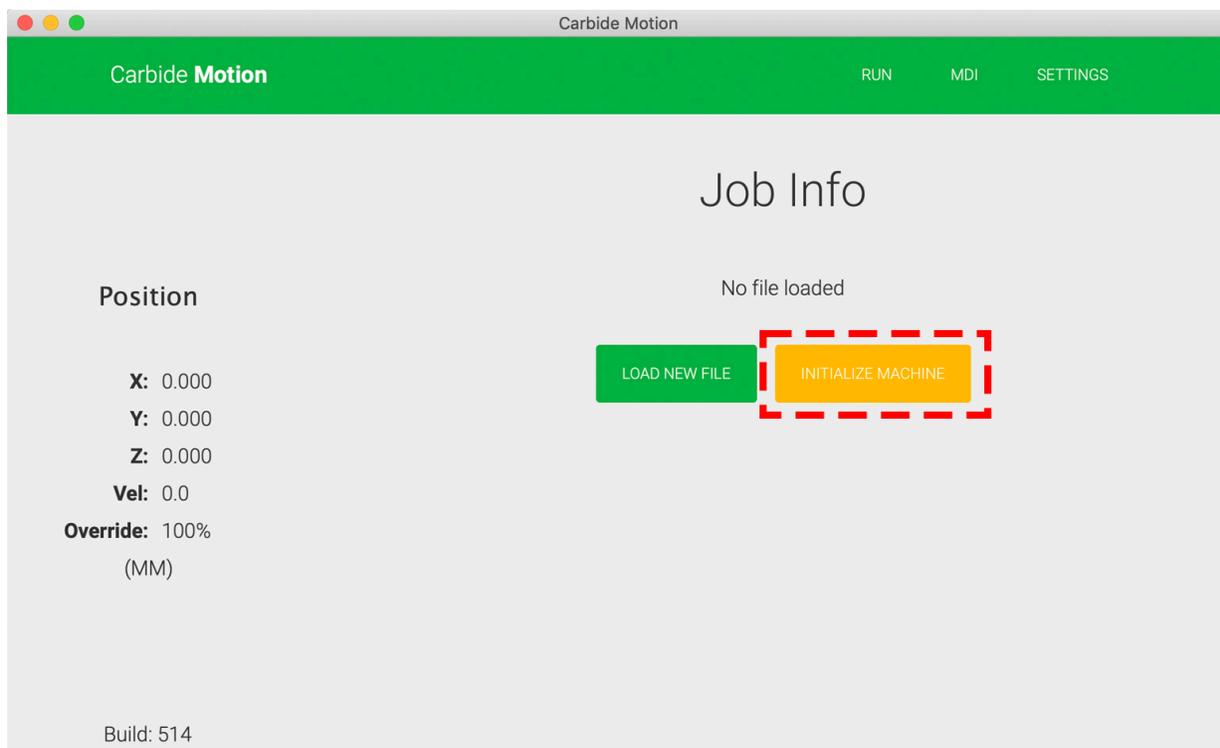


Home the Nomad 3

Now that you are connected to your machine, it's time to home it. Homing your CNC machine is the process of sending it to a known, fixed, repeatable location. This means that every time you home the machine, it will move to EXACTLY the same position. Every. Single. Time. This allows you to move your machine to ANOTHER position, relative to the home position, with great precision. On your Nomad, the home position is the back-right corner.

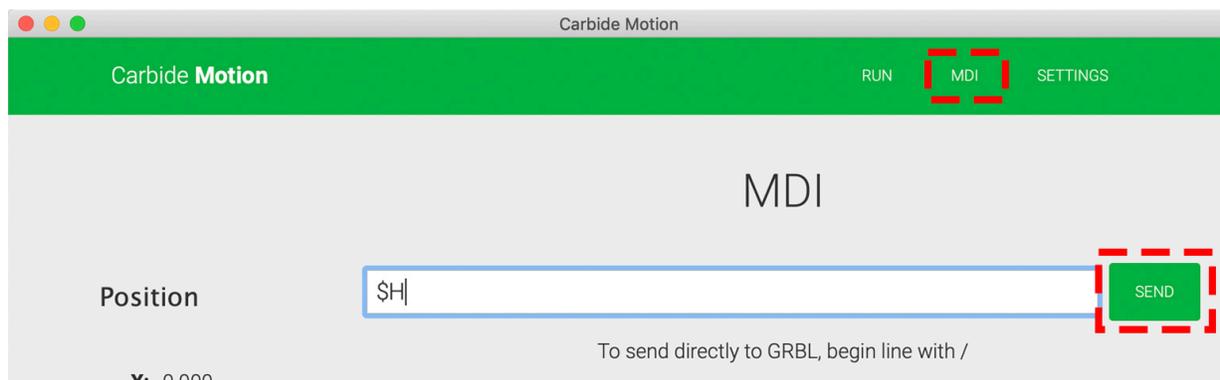
To home the Nomad 3:

1. Click the **Initialize Machine** button.



During homing, the table will move to the front of the machine and the spindle will move to the back-right corner.

PRO TIP: Sometimes, later on in a project you will want to home the machine again. To do so, click MDI in the top menu bar to open the *MDI screen*. Enter `$H` into the text field and click the Send button. This will re-home the machine.



Set Up Your Wasteboard

Now, that the table is all the way to the front of the machine, it's time to install the wasteboard. The wasteboard is used to protect the machine's aluminum table from damage. While you can attach stock directly to the table, chances are that you'll accidentally cut through the stock material and into the table at some point, so we highly recommend using the wasteboard. To install the wasteboard:

1. Place the wasteboard on the table with the countersunk holes facing up.
2. Use a 6mm hex key and the five (5) M6 × 10mm screws to secure the wasteboard.

Secure Your Stock

Now, you need to secure your stock to the wasteboard. This process is called workholding. There are many different methods of workholding, including double-sided tape, clamps, direct fastening with nails or brads, low-profile vise, and machinable wax. Which workholding solution you choose for each project will depend on several factors, including the type and size of your material and the kind of part you are milling.

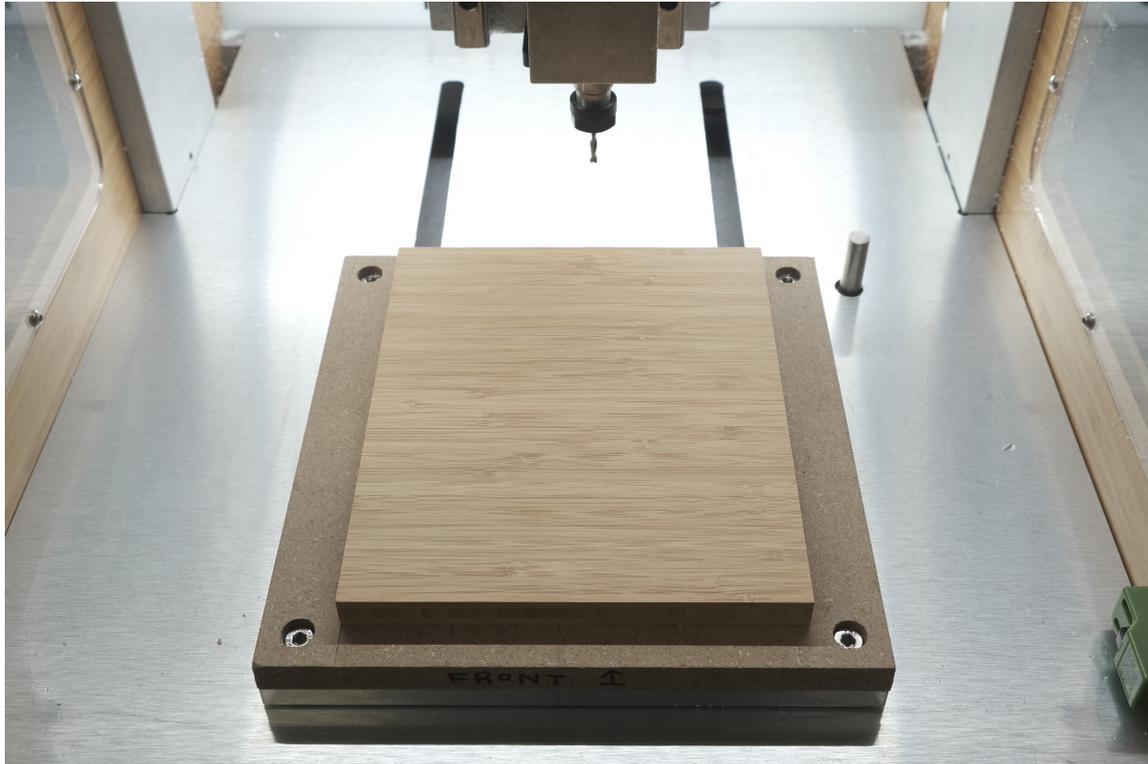
Since we'll be using almost the entire surface area of our stock, there's no room around the perimeter to hold the material down with clamps. Instead, we'll be using double-sided tape, which is a simple, cheap way to temporarily secure your material.

1. Cut a few strips of double-sided tape. We recommend using at least two strips long enough to span nearly the long edge of the material (approximately 5.5–6" long). We also added a third shorter strip of tape in the center to assure the stock doesn't come loose.
2. Place the two long strips of tape on the sides of the bamboo plywood, about one inch from the outside edge. Place the smaller strip in the middle.



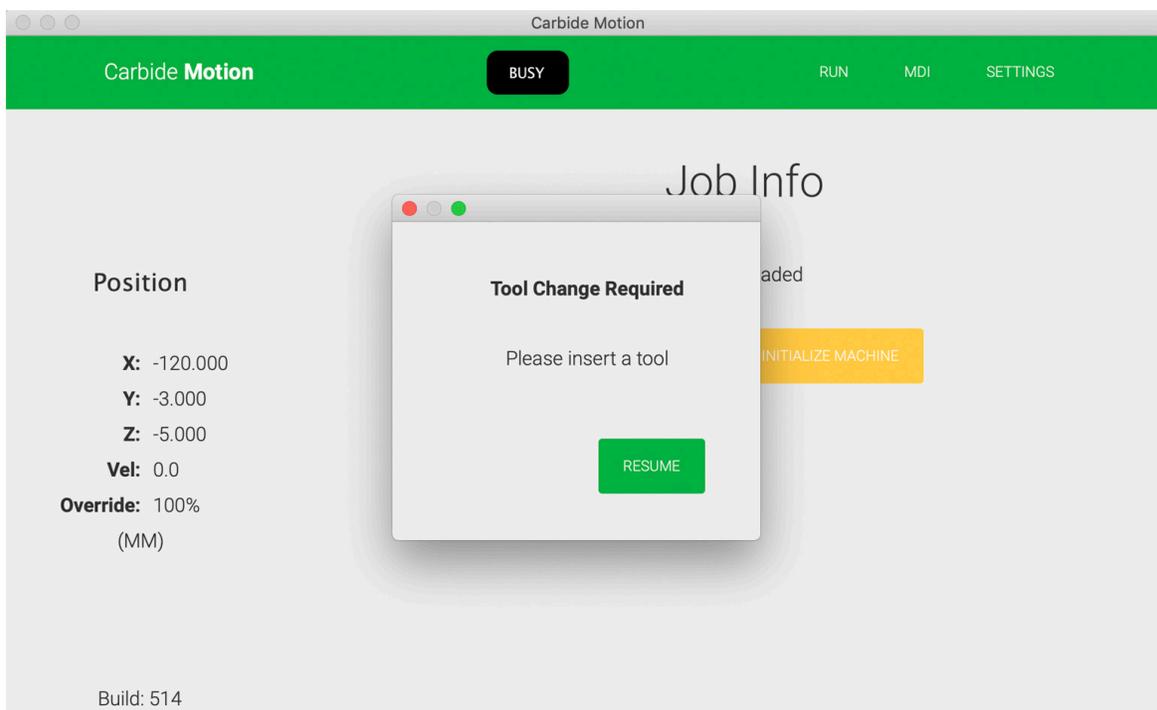
If you place the tape too close to the edge of the stock, you run the risk of your cutter getting all gunked up as it mills the outside edge of the tray.

3. Ensure there is no dust or dirt on the surface of the wasteboard that could weaken the tape adhesion.
4. Align the bamboo to the wasteboard. Make sure the plywood is as close to parallel with the edges of the wasteboard as possible (don't be afraid to draw a few marks or a line on the wasteboard if you need to).
5. Apply firm pressure to the bamboo for several seconds to secure it to the wasteboard.

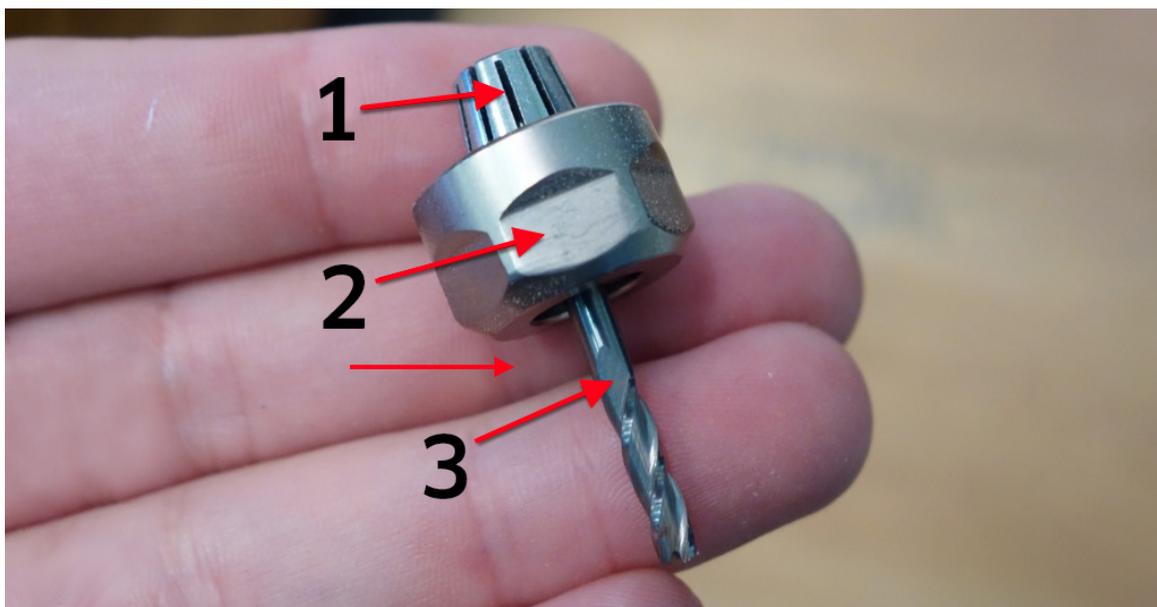


Install the End Mill

After homing your Nomad 3, Carbide Motion will prompt you to load an end mill.



The collet (1) holds the end mill (3) when the collet nut (2) is tightened in the spindle. Watch our End Mill Installation Tips video: <https://www.youtube.com/watch?v=gOFUHPcKkIE>.



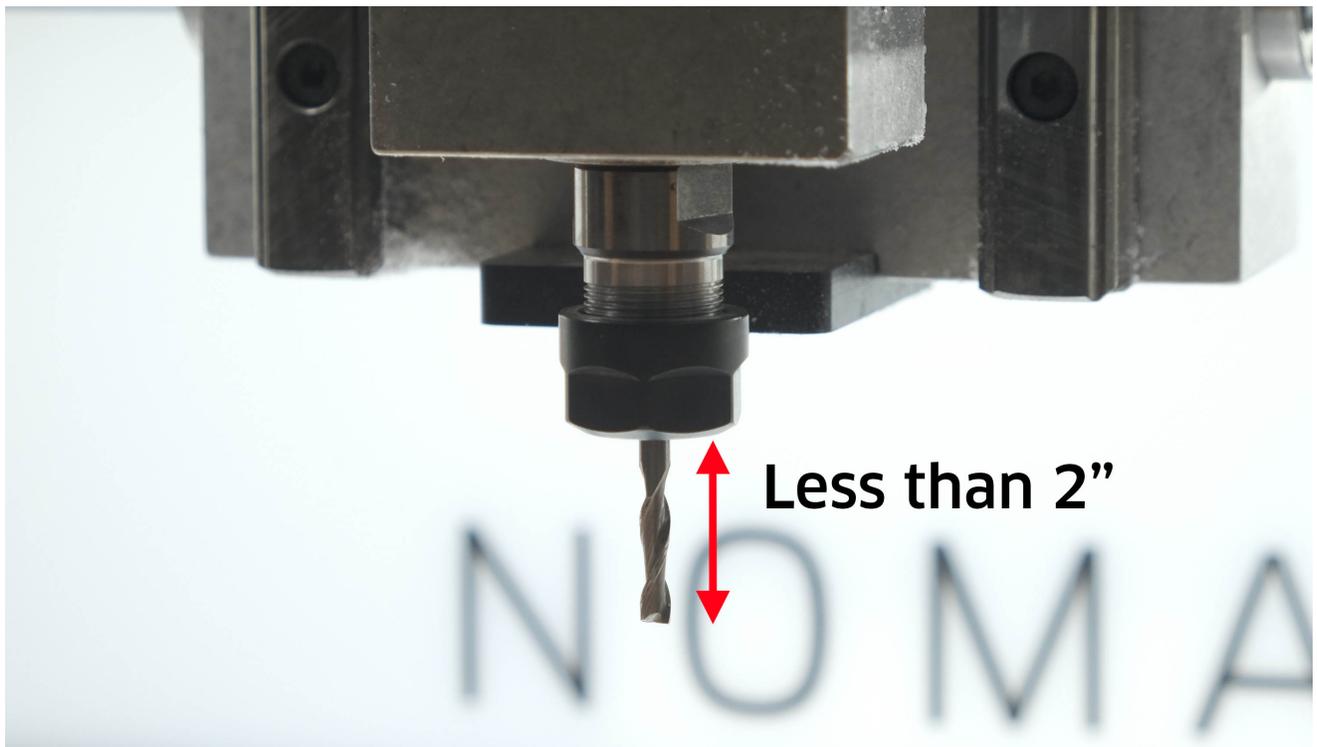
NOTE: A collet is sized to hold one diameter of cutter. Your Nomad 3 comes with a 1/8" ER-11 collet which is capable of holding any 1/8" cutter, even if you did not buy it from Carbide 3D. You can buy additional ER-11 collets to hold different diameter cutters from Carbide 3D or other vendors.

To install your end mill:

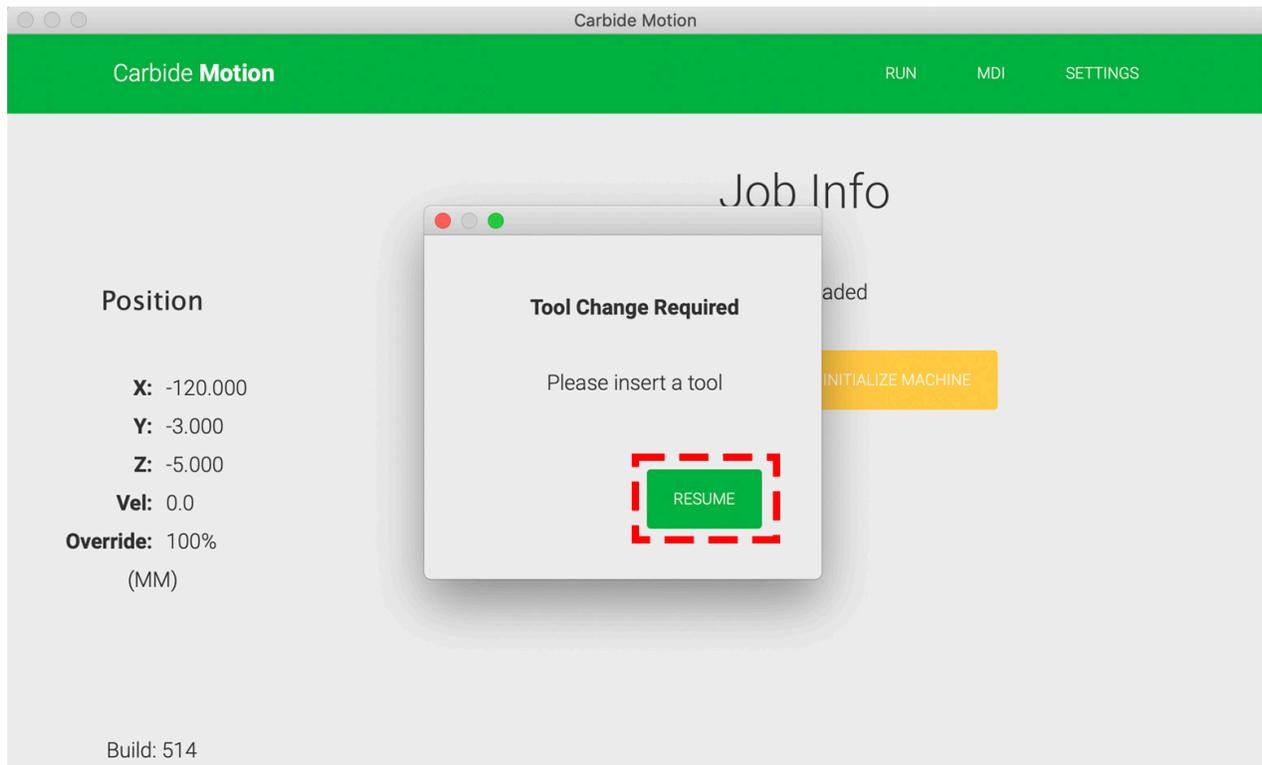
1. Insert the shank of the end mill into the collet.
2. Use the 17mm collet wrench to tighten the collet nut securely as you hold the spindle shaft in position with the 13mm spindle wrench.

WARNING: Never tighten the collet nut without an end mill inserted into the collet.

NOTE: When tightening the collet nut, the tool should not extend more than 2 inches from the collet nut.



3. Now that the end mill is installed, click the **Resume** button in Carbide Motion.



4. Carbide Motion will position the cutter above the BitSetter, which will measure the end mill length.

Set Job Zero

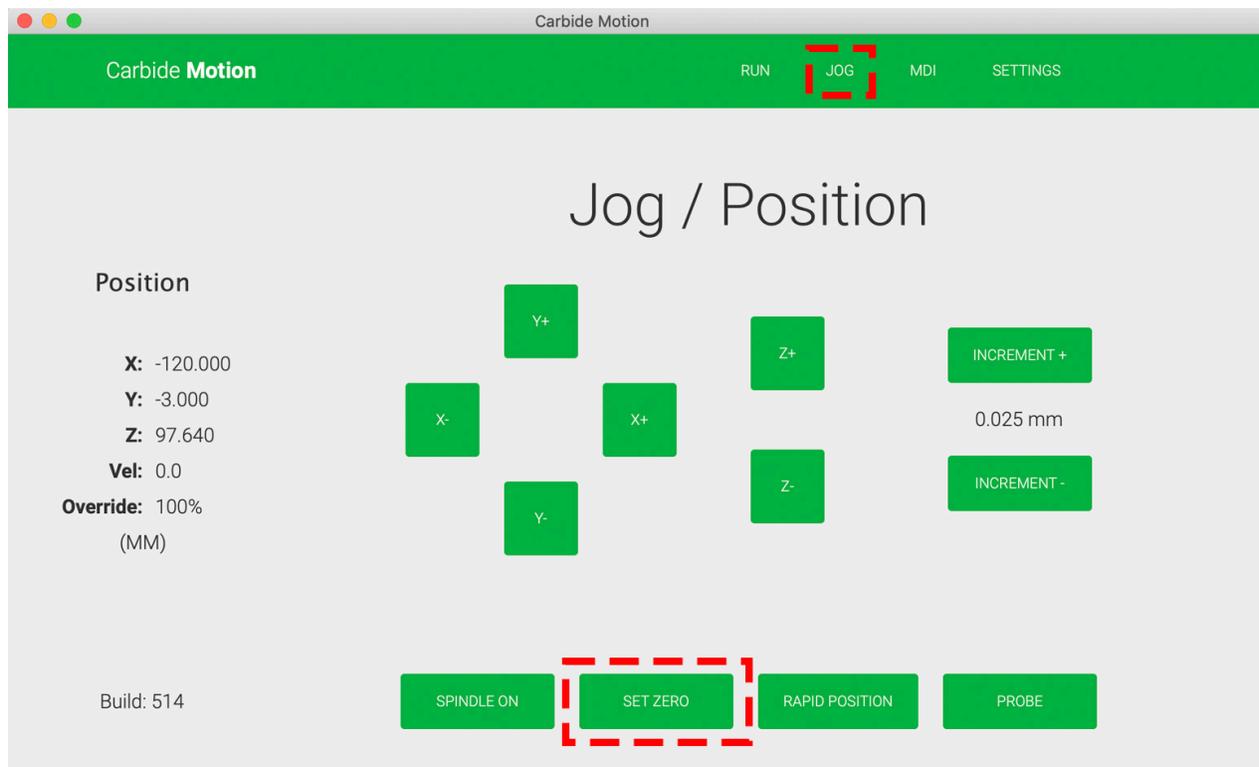
Once your machine is homed, you can jog the machine (controlling the X, Y, Z axes movement of the machine) or load G-code.

Before we load the G-code for our project, we need to tell the Nomad 3 where the origin point of the project is. If you remember from the Carbide Create section above, the origin point is in the lower-left corner of our stock.

PRO TIP: Carbide Motion uses millimeters by default. If you're more comfortable working in inches, click Settings in the menu bar and choose "Inches" from the Units dropdown list.

To set job zero (the origin point), first we need to move the end mill so it's centered over the corner of our stock:

1. Click the **Jog** button in the menu bar.



2. Use Carbide Motion to jog your machine to the lower-left corner of your stock:

- a. Use the **Y- / Y+ buttons** to position the Y-Axis (the table) and the **X- / X+ buttons** to position the X-Axis (the spindle, moving side-to-side).
- b. Use the **Increment+** and **Increment-** buttons to speed up or slow down the machine's movements.
- c. You may need to use the **Z- / Z+ buttons** to position your Z-Axis closer (the spindle, moving up/down) to the top of your stock so you can set X and Y more accurately. **Make sure to always move slowly while approaching the top of your stock.** Get the cutter close enough to determine your X/Y Zero **without** actually touching the top of the stock just yet.

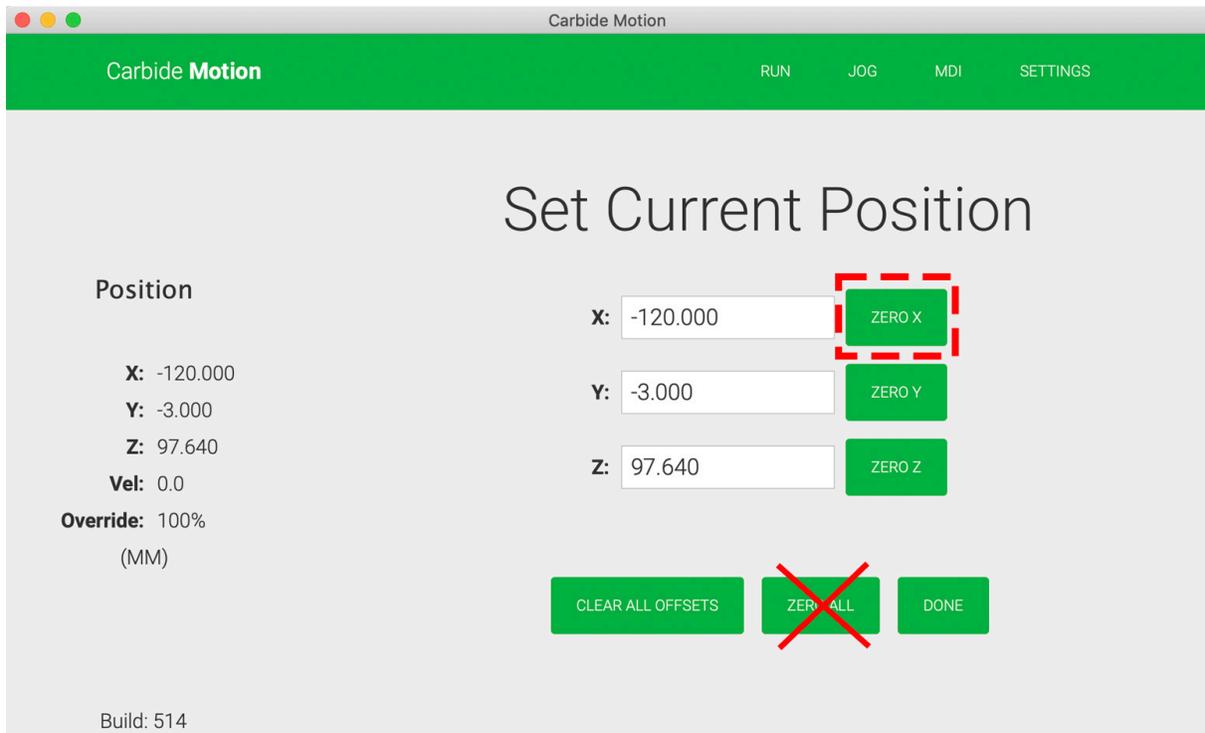
Jog Shortcut Keys

- Up Arrow:** Move Y-Axis toward the back.
- Down Arrow:** Move Y-Axis toward the front.
- Left Arrow:** Move X-Axis to the right.
- Right Arrow:** Move X-Axis to the left.
- , (Comma)** : Move Z-Axis down.
- . (Period)** : Move Z-Axis up.
- 1,2,3,4** : Select Move Increment

3. Once the machine is in position over the lower-left corner, you'll need to set job zero:

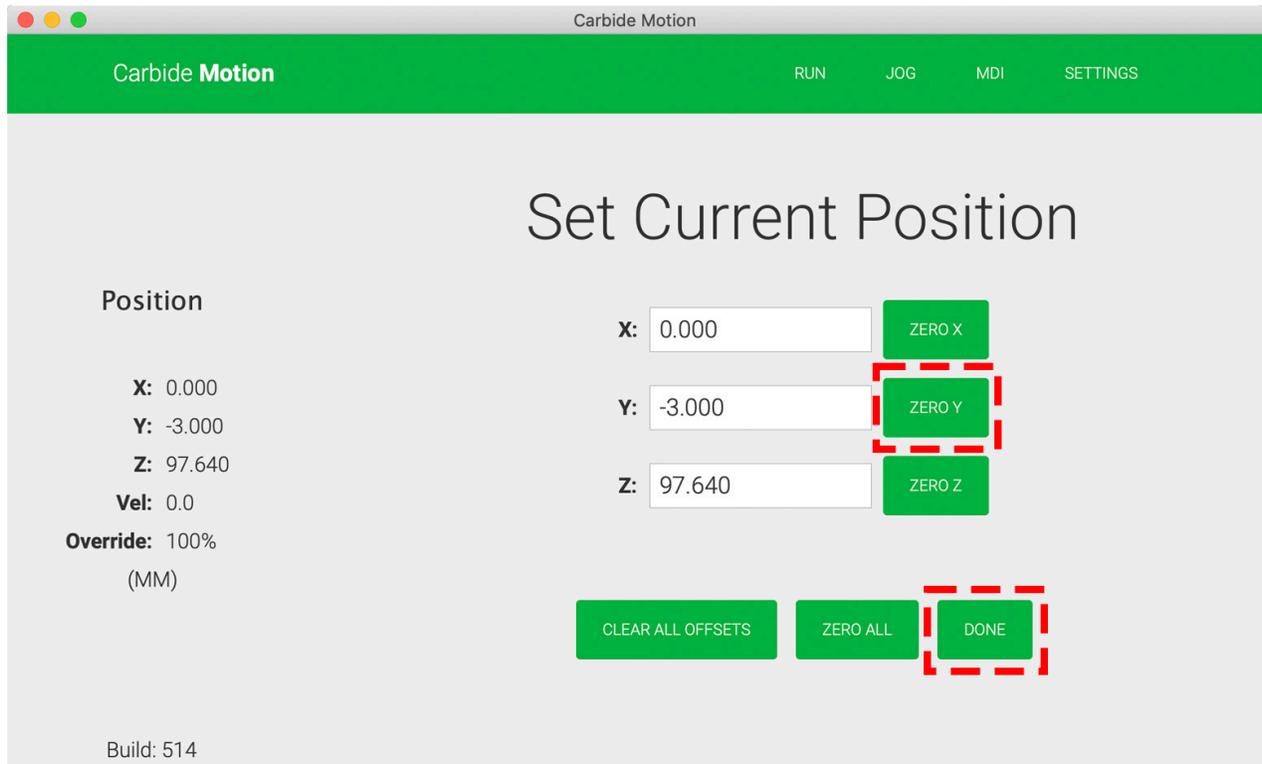


a. Click the **Set Zero** button to bring up the *Set Current Position* screen.



b. Click the **Zero X** button to set your X-axis to 0.00. You're telling the controller that your machine is now where you want ZERO to be for the X. **Do NOT** click the **Zero All** button.

- c. Click the **Zero Y button** to set your Y-axis to 0.00.
- d. Then, click **Done** to return to the *Job screen*.



Set Z-Zero

Now we need to set our Z-Axis starting point (Z-zero for short). The goal of setting Z-zero is to lower the cutter until it touches the very top of the material. Here is a video of the process: <https://youtu.be/1FgMjg98cyM>.

Do NOT turn on the spindle yet! With your spindle in position over your stock material, your cutter installed (following the steps above), and with X-zero and Y-zero set (following the steps above), set Z-zero by:

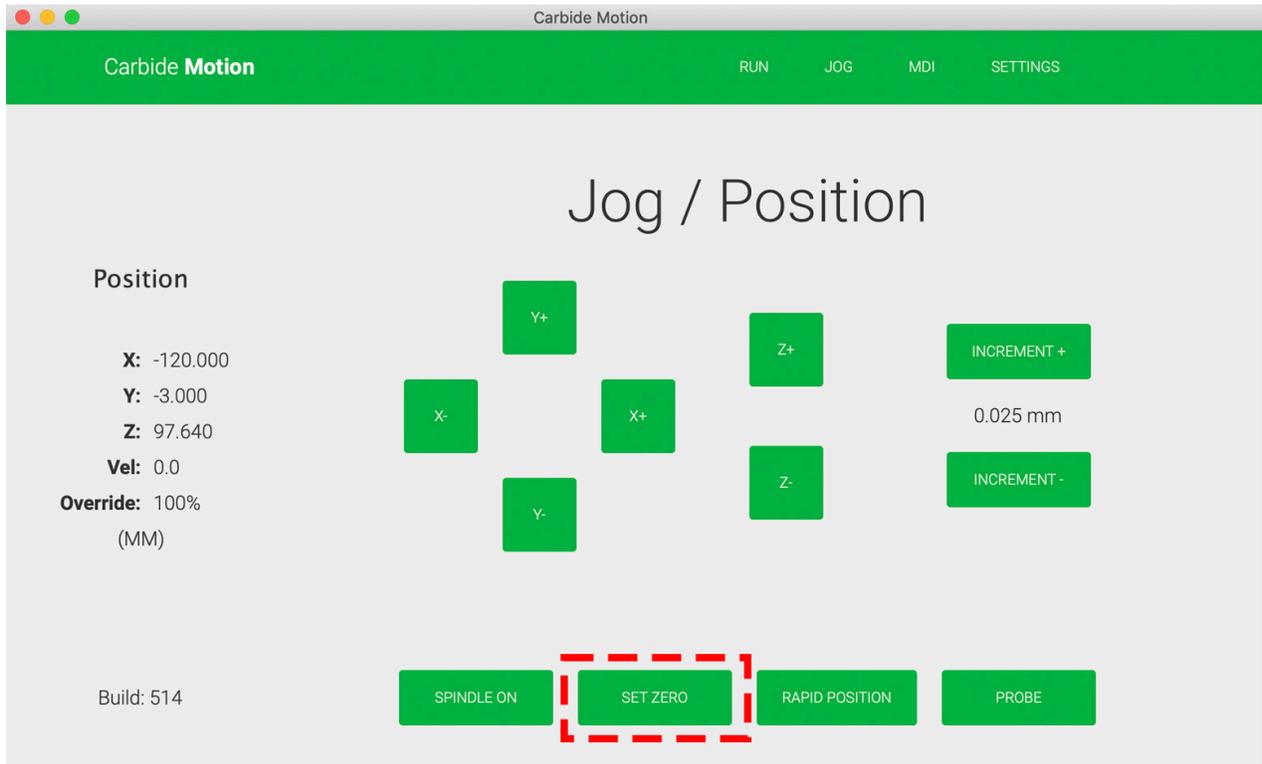
1. It's easier to set Z-zero when the end mill is fully over the stock, so jog your machine a few millimeters inward from the corner (on both the X- and Y-Axes).
2. Use the **Z- button** to begin lowering your cutter.
3. As the cutter moves closer to the material, reduce your increment, by clicking the **Increment-** button, accordingly. You do **NOT** want to plunge the cutter directly into the material. We are trying to get the cutter to touch the very top of the material, but no further.
4. Continue moving the cutter down until it is within about 1mm of the material.

5. Slide a small scrap of regular paper between the cutter and the material.



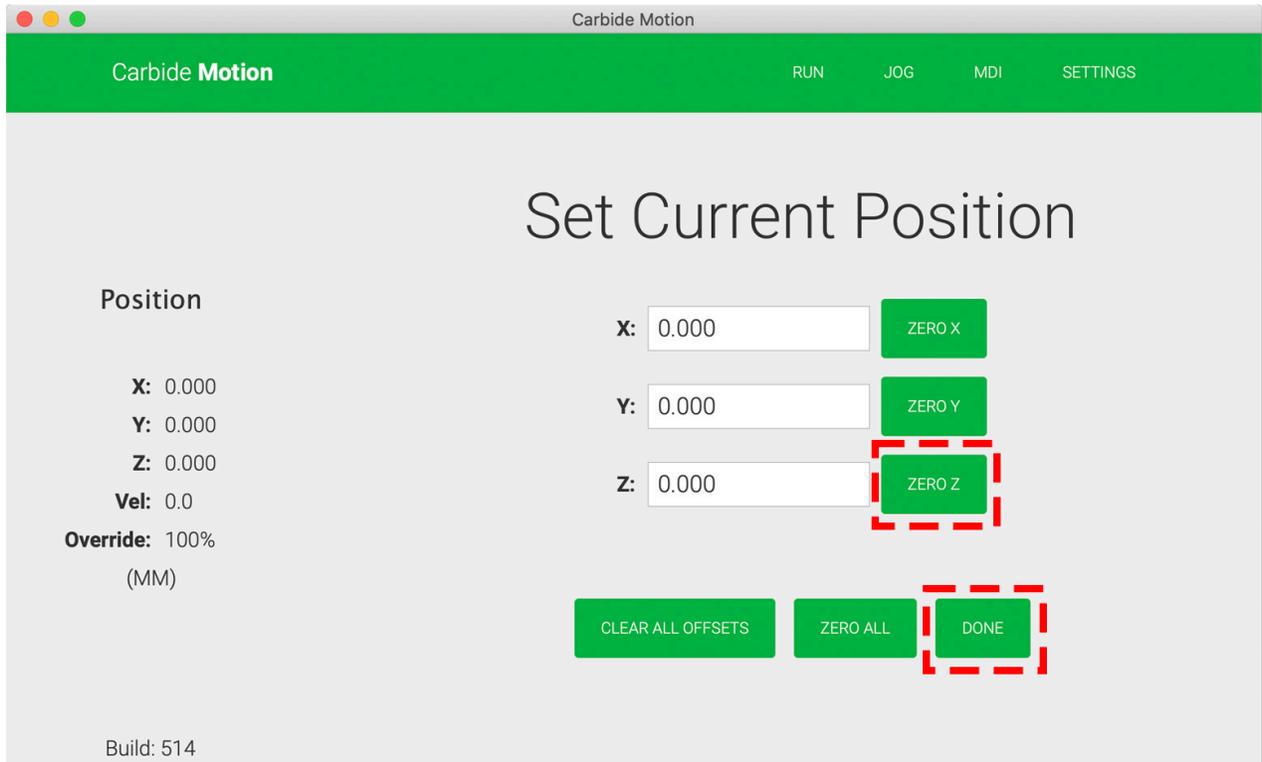
6. Very slowly, and with a very small increment, resume lowering the cutter towards the material. Simultaneously, begin moving the paper back and forth across the top of the material.
7. When the cutter grabs the paper and you can no longer wiggle it back and forth, stop lowering your cutter; you have found Z-zero.

8. In Carbide Motion, click the **Set Zero** button.



9. Click the **Zero Z** button to set your Z-axis to 0.00.

10. Click **Done** to return to the *Jog* screen.



11. In the *Jog screen*, use the **Z+ button** to raise the cutter to release the paper.

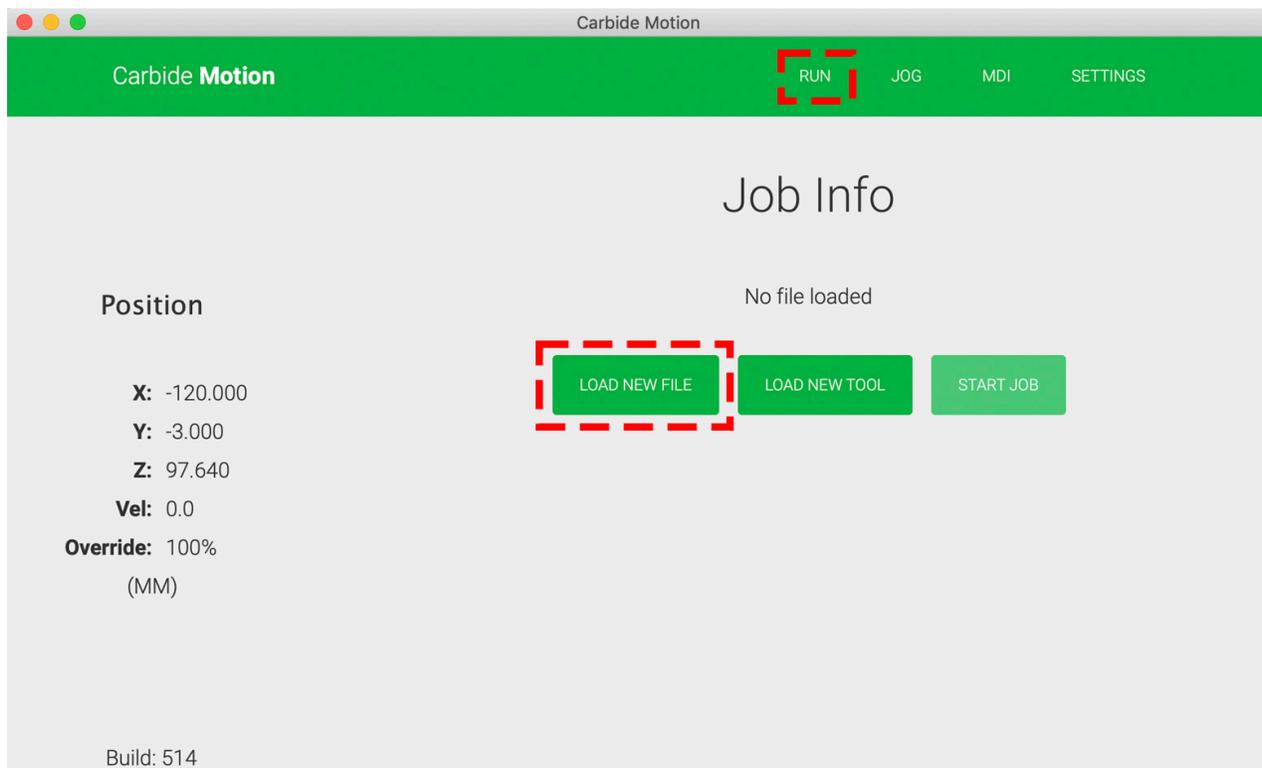
Now your Nomad 3 knows exactly where to begin machining from.

NOTE: Forgetting to set job zero (particularly Z-zero) is the number one cause of a project cutting in the wrong location.

Load the G-code File

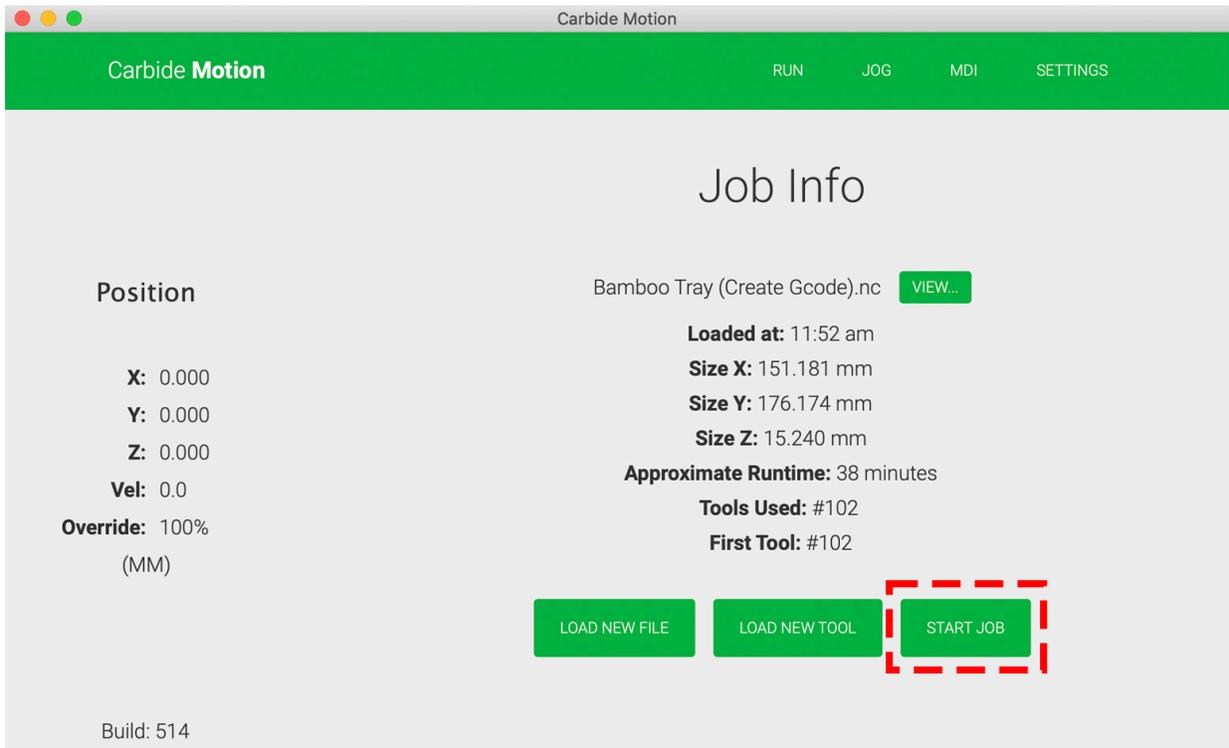
The Nomad 3, as well as most other CNC machines, runs files called G-code. G-code is a format that goes back decades to the original CNC machines in the 1960s. G-Code is basically just a set of motion commands that tells a machine what cutter to use, a series of coordinates to move to, and the information about how fast to go. G-Code contains only this motion information, it does not contain your original CAD data anymore.

1. Close the Nomad 3's protective cover.
2. In Carbide Motion, click the **Run button** in the menu bar to open the *Run screen*.

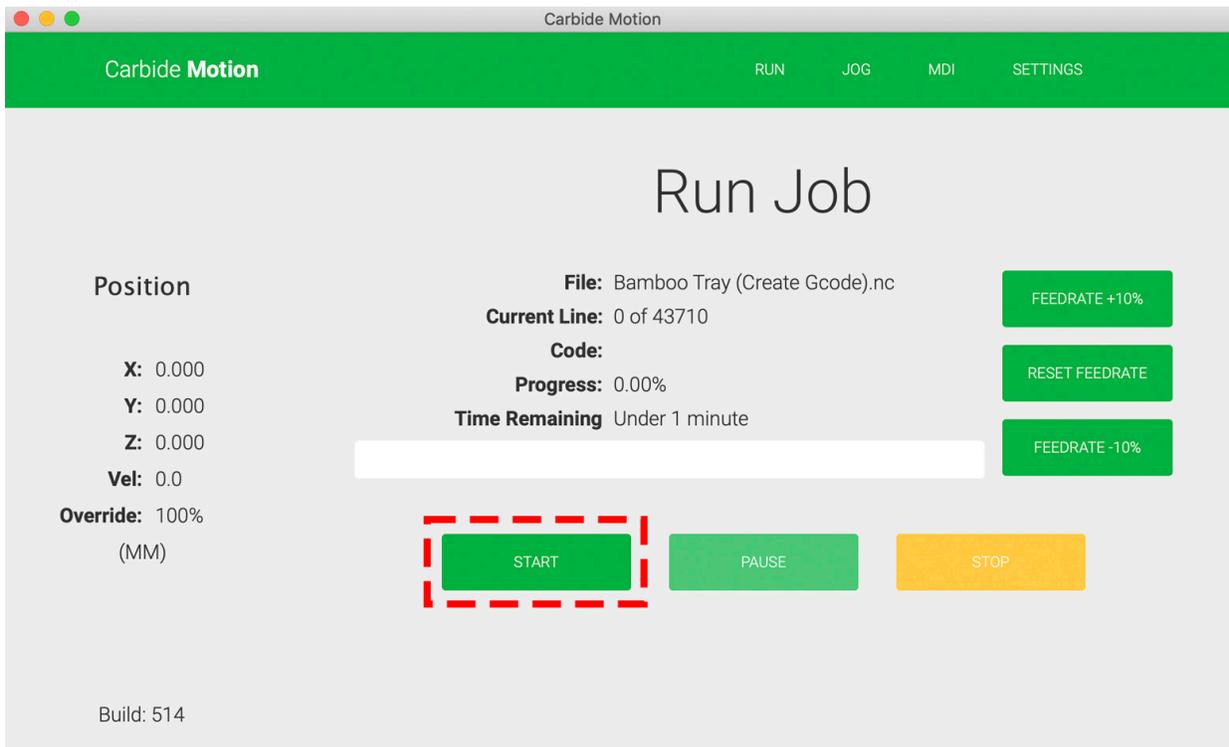


3. Click the **Load New File** button.
4. Browse for the **Bamboo Tray (Create Gcode).nc** file and click the **Open** button.

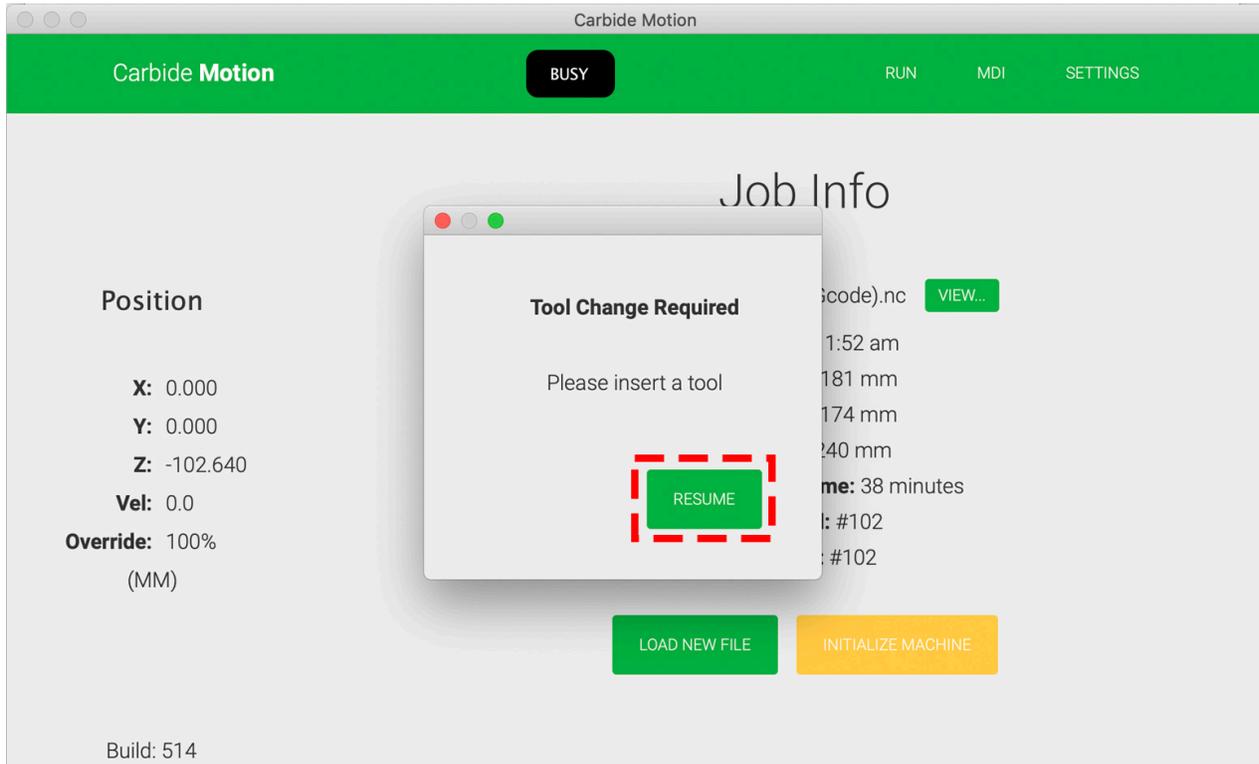
- Once the .nc file loads, you will see the approximate runtime, tools used, and other job info.
- Click the **Start Job** button.



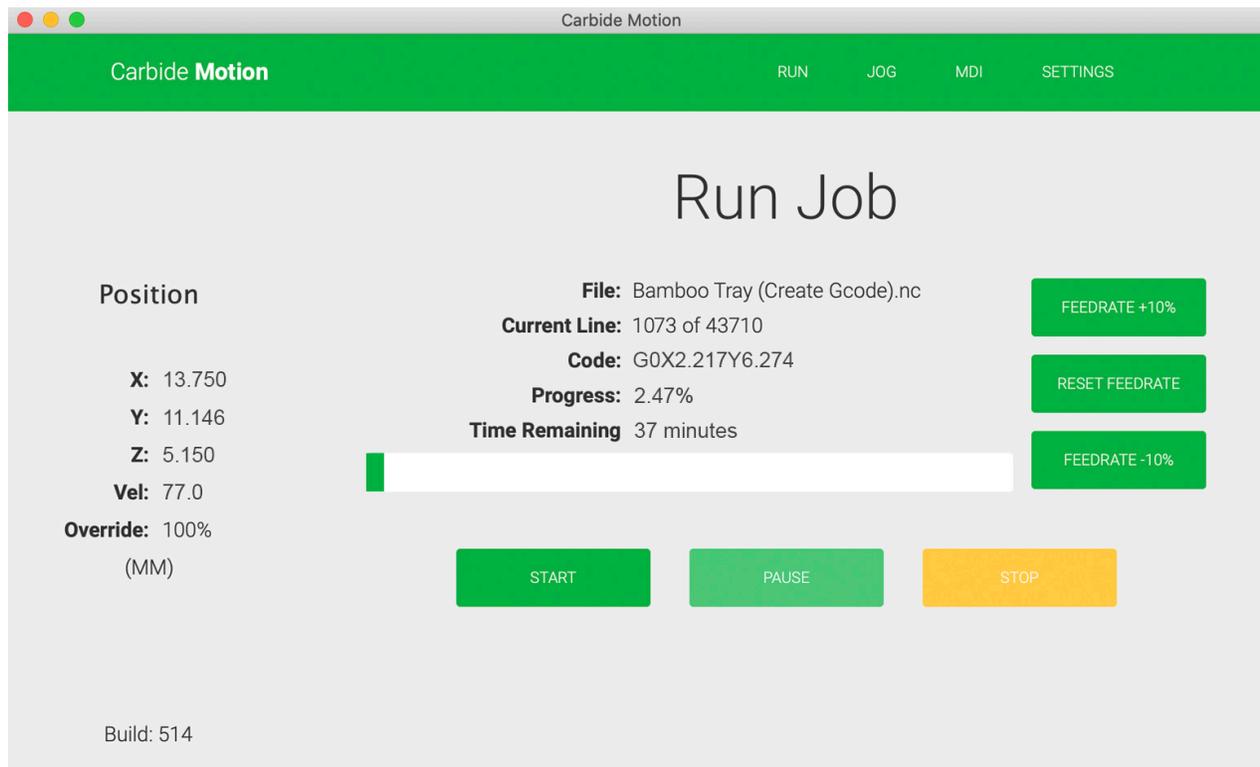
- Then, click the **Start** button.



8. During the initial 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.
9. Click the **Resume** button once you have confirmed that the correct tool is installed and milling will begin.



If all goes to plan, the Nomad will machine out all of the pockets and holes in the tray, and then trim up the outside of the bamboo. The program will take about 37 minutes to run.



NOTE: If a lot of wood chips and dust are collecting on your workpiece, you can hit the Pause button in Carbide Motion at any time, open the door, and vacuum up the milling debris.

Finish Up the Project

When the program is complete, the spindle will automatically stop, park, then home and the table will move all the way to the front of the machine so you can access the part.

1. Pry the tray off the wasteboard.
2. Remove the double-sided tape from the back of the bamboo.

PRO TIP: If you peel the tape against the grain, some of the bamboo's surface fibers could break free. If you want a smooth finish on the back of your tray, peel with the grain.

3. Since bamboo has a moderately coarse fiber structure, there will be some fuzz left along the walls of the tray. To clean things up, you can use some 220-grit sandpaper on the edges.



Video Tutorial

Watch the video tutorial for this project here: <https://youtu.be/cD7HhA6mdEw>.