CNC

Safety First

- The doors of the CNC enclosure must be closed while the machine is running.
- The **spindle must be turned off** whenever the doors are open, such as during tool changes or material setup.
- Drill bits are sharp so be sure to handle them carefully. Gloves are available if needed.
- Vacuum any sawdust at the end of your cut. Dust masks are available if needed.
- The **Emergency Stop Button** will halt the xyz axis from moving but will not stop the spindle from spinning. The **Spindle On/Off** switch is the left-most button on the panel.

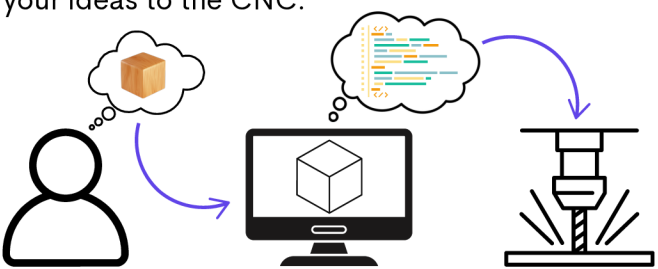
Table of Contents

Part 1: Collaborating with the CNC

CNC stands for **Computer Numeric Control** which is a manufacturing method that automates the movement of a tool with computer software.

You can think of this as **collaboration** with the CNC. **You and the machine will divide up the tasks involved in building a project based on each of your skills.**

As a designer, you have an abstract and conceptual understanding of spatial information, but the CNC understands information formatted in coordinate data files called G-Code. Software will act as a translator to communicate your ideas to the CNC.



Typically, a **CAD** (Computer Aided Design) software is involved in generating the digital design, and a **CAM** (Computer Aided Manufacturing) software is involved in telling the machine where and how to move.

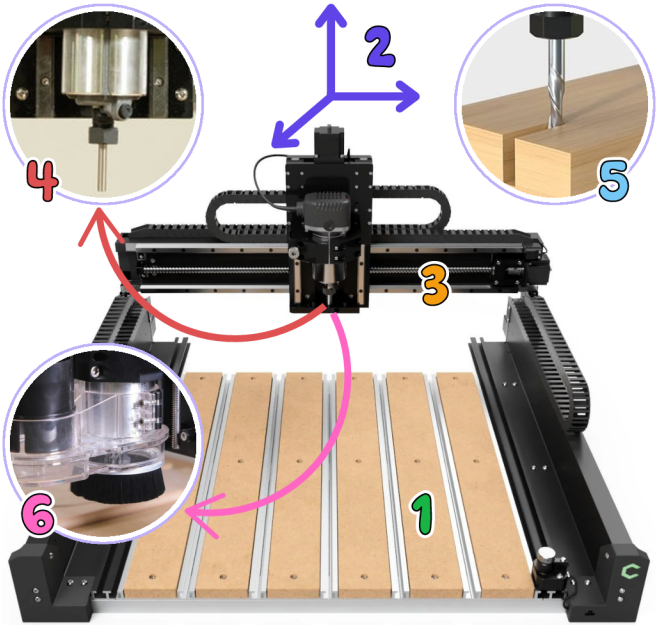
Our specific CNC uses two softwares:

- **Carbide Create** is a combined CAD and CAM software that will turn our design ideas into G-Code, which will give the CNC the coordinate data it needs to execute our project.

- **Carbide Motion** is a machine controller software which will allow us to load our file and drive the machine.

Part 2: Get to know the CNC

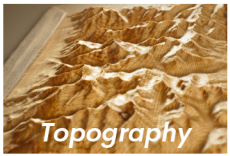
A CNC is a "subtractive manufacturing" machine that removes material. (An example of "additive manufacturing" is a 3D printer, which builds up material). It can carve wood, plastic, foam, wax, and more. Some parts include:



1. The base of the machine is called the **bed.** This is where we put our material that will be carved.
2. Our CNC has **3 axes** to move the spindle along in the **X, Y, and Z directions**. This means it can only move up/down, left/right, and forward/backward. Some other CNC machines have a fourth axis (or more!) that allows you rotate your material as it is carved
3. The crosspiece that holds the spindle is called the **gantry** and it moves back and forth along the guide rails at the sides.
4. A **spindle** which rotates interchangeable bits like a drill to carve your material.
5. Bits are sometimes called **tools** or **tooling.** As it cuts, it removes material the width of the tool.
6. The removed material is sucked through a vacuum tube. The **dust boot** is a removable end of the vacuum that surrounds the cutting tool to prevent stray debris.

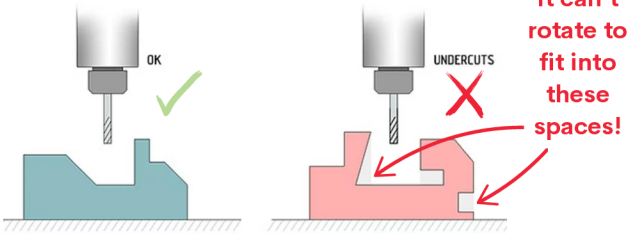
The anatomy of the CNC and the ways it can move means it's really great for some applications…

- It can cut through thicker materials than a laser cutter can.
- It can achieve high precision and detailed 3D contours in the places it can reach.

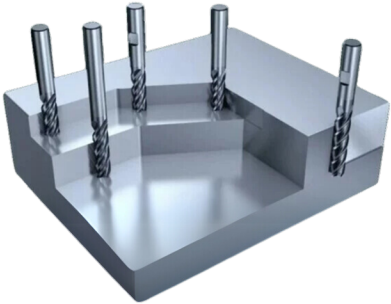


…but struggles with others:

- It can't reach everywhere! It only has 3 axis, so it can't do undercuts, or cut the underside of material.

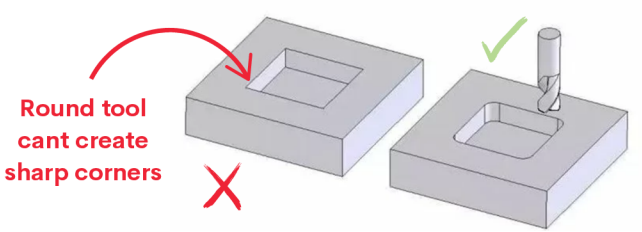


- Our CNC also can't do very deep cuts - The material can be up to 3 inches thick, but the cuts themselves can only go up to about 1 inch deep. The exact maximum cut depth will depend on the bit you are using.



How far down you can cut depends on the tool you use!

- It can't do sharp internal corners, because the corners will match the radius of whichever bit you are using.



Part 3: The CNC's part of the project

The CNC needs to know some information about its location and trajectory, such as:

- Where the spindle is currently positioned
- Where the starting point is (**origin** or **zero** point)
- Where to travel (**toolpaths**)
- How fast to travel (**feedrate**)
- How much to overlap with the previous pass (**stepover**)
- How fast to spin the drill bit (rotations per minute, or **RPM**)

Even with this information, **we will still need to watch the machine!** The CNC will not be able to tell if something is not cutting correctly, has become loose, or if the appropriate material decisions were made for the project.

Part 4: Our Part of the Project

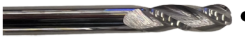
Two main factors will dictate most of the information we need to give the CNC: our **stock material**, and our **form** (the final geometry after carving). Knowing these, we can determine our bits, speed settings, and workholding methods.

What bits to use - bits come in different shapes and sizes to produce different effects. Many projects will use multiple bits. Their basic anatomy includes:

1. **Shank** - This cylindrical part is inserted into the collet of the spindle.
2. **Flutes** - These are the cutting edges. The tool shouldn't be submerged into the material beyond the length of the flutes.
3. **Cutting Point** - The shape of the tip will impact the contour of the cut.

Some common examples of bit types are:

- **End Mills** have a straight end, creating a channel with a flat bottom perpendicular to the sides.



- **Ball Nose Mills** have a rounded cutting tip, which is helpful for smoothing.

- **Vee Bits** are shaped like a **V** that allows for fine detail and sharper corners.

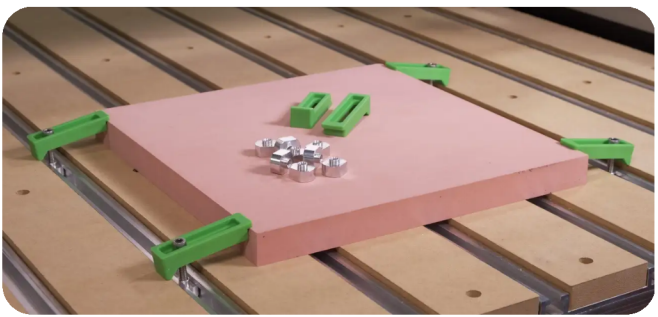
There are many more kinds of specialized bit types out there that you are welcome to research!

Remember: The CNC trusts the information you give it. If you plan to use one type of bit, it won't notice if you accidentally put in a different one.

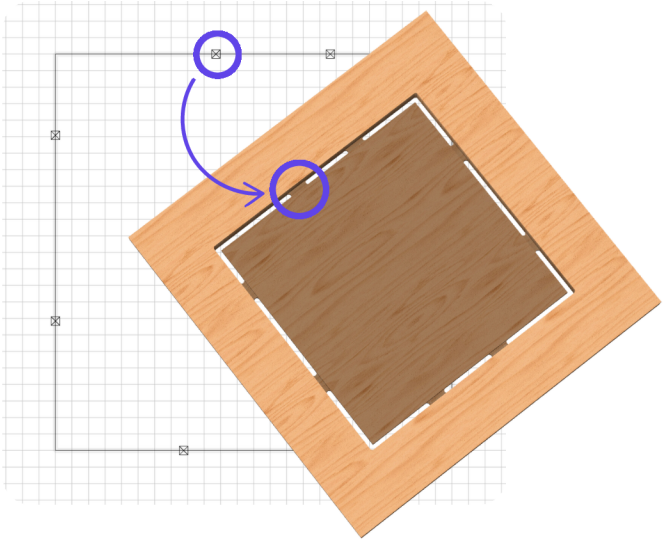
🦺 **Safety Tip!** 🦺 Remember that bits are sharp blades that carve material! Make sure to handle these carefully.

Speed Settings - The appropriate RPM, feedrate, and stepover rate are largely determined by the density of the material, in combination with the bit you are using. (For example, we typically need to go slower with denser materials!) Fortunately, our CNC has a built-in list of the appropriate speed settings based on our material and the bit.

Work Holding - This refers to the method used for keeping our material secure, so that it doesn't move around at all during the cut. Our standard method is using clamps that have T-Nuts that slot into the tracks of the CNC bed.



We also need to prevent our material from moving around once the cut is complete. We will create **tabs** - meaning the cut doesn't go completely through the material in a few locations.



Alternative methods of workholding include using specialized tapes, screws, other fasteners, or custom jigs.

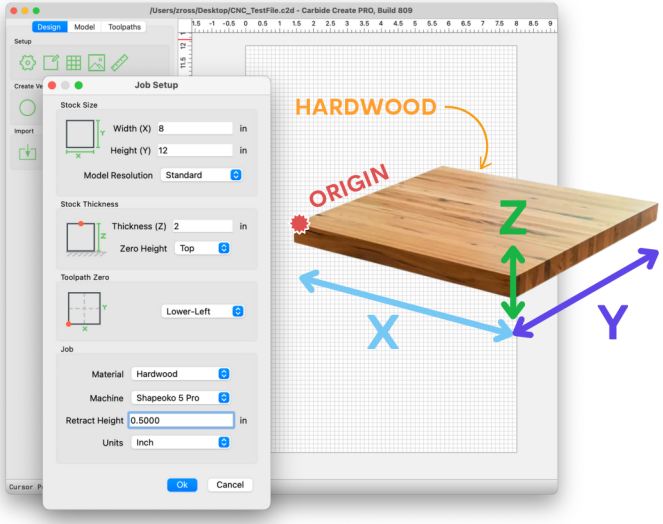
Part 5: Writing the Instructions

In Carbide Create we will:

1. Setup our stock
2. Design our 2D geometry
3. Design our 3D geometry
4. Generate our toolpaths

1. Stock Setup

This is where we describe our material:



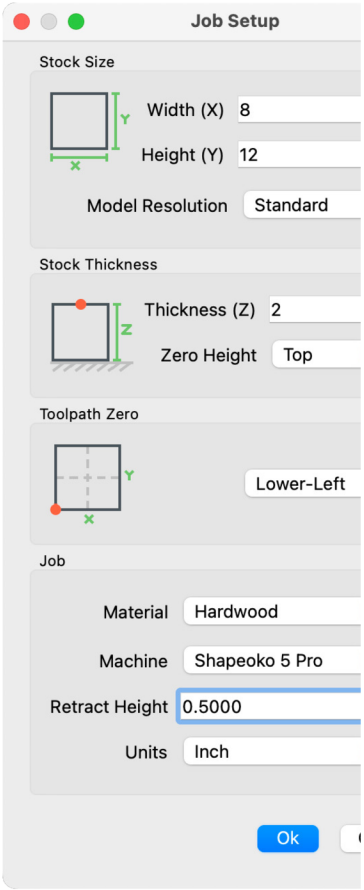
Dimensions as seen from above the bed

"Zero Height" should be "Top"

Select "lower left"

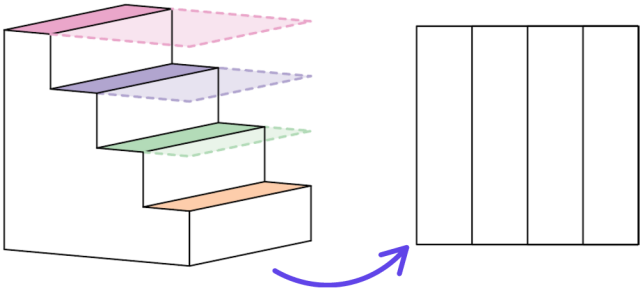
Enter material type

Retract height=0.5" to clear the clamps



Important Context - 2D vs. 3D Geometry

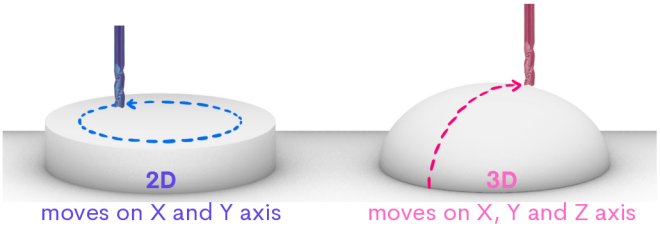
2D Geometry - In this context, a "vector" refers to a 2D path that the tool will follow along the X and Y axis. When we say geometry is "2D" it doesn't mean that the final object is flat. Rather, it means that the object can be thought of as stacked planes set to different depths.



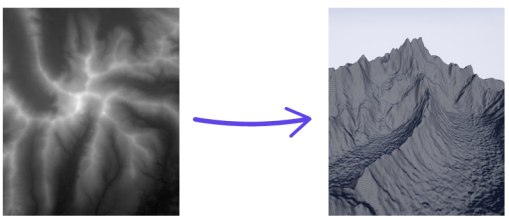
Cutting within each 2D vector at a different depth will result in the 3D form

2D vectors can be imported as SVGs or generated using image trace. You can also create vectors such as shapes, lines, and text in Carbide Create.

3D Geometry - In this context, when we say geometry is "3D", we mean that the bit must move along the X,Y, and Z axis simultaneously. STL files can be made in other 3D modeling softwares and imported into Carbide Create.

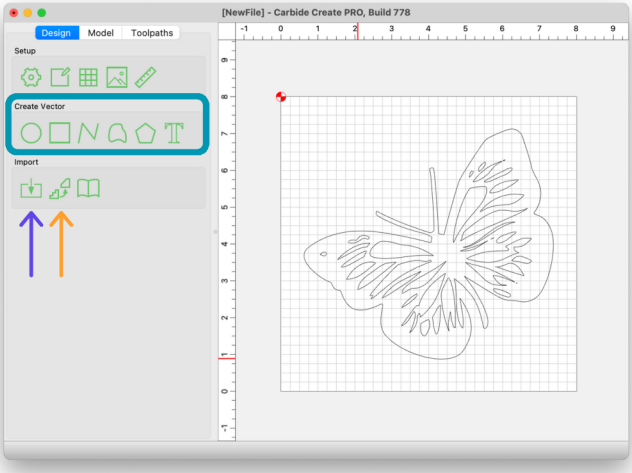


Alternatively, JPEG images can can also be imported to make a 3D height map. The software reads the image pixel by pixel, and interprets the height of each point based on its shading. The lighter the shade, the higher the point.

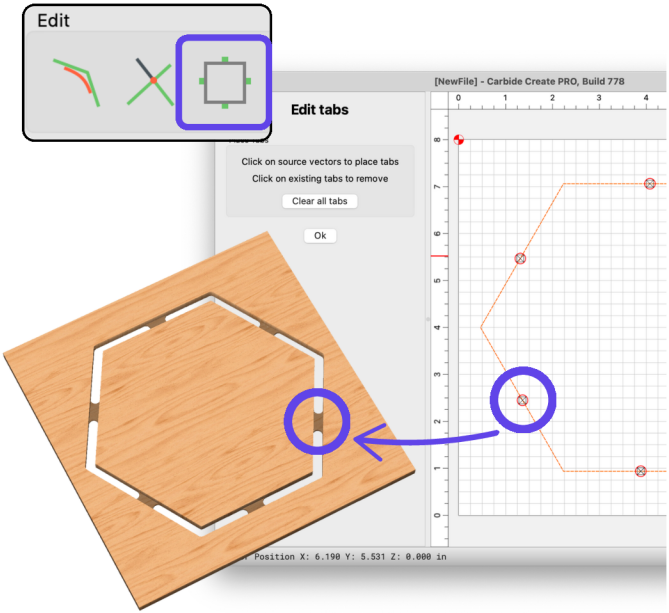


2. Designing Your 2D Geometry

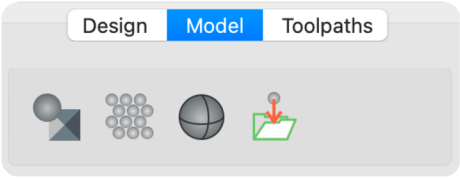
- If you already have a vector file from another software, you can **import it now as an SVG.**
- You can also use the **image trace function** to create vectors from an image.
- If you are designing vectors directly in Carbide Create, you can choose shapes from the **Create Vector** menu.



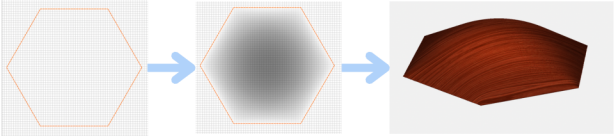
The **Edit** menu allows you to edit points on your lines, and also to place **tabs**. Tabs are recommended for any vectors that will be cut all the way through the material, to keep the piece from coming loose once it's cut. **Tabs will be removed manually once carving is complete.**



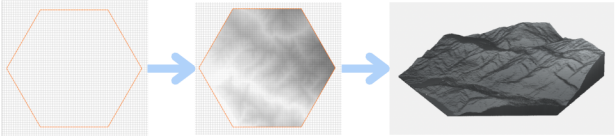
3. Generating Your 3D Geometry:



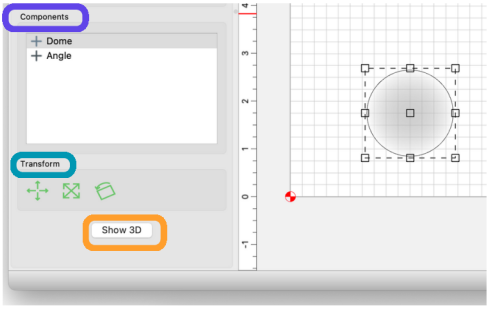
Add Shape converts 2D vectors into 3D non-planar geometry. Select the 2D vector, click this icon, and enter the relevant information about your desired shape.



Import Image converts .jpgs into height maps within a selected vector outline. (Darker pixels are higher than lighter pixels.)

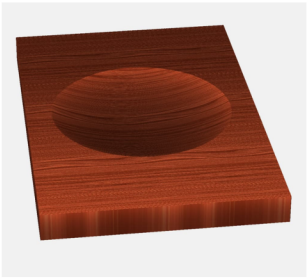


Import STL allows you to import 3D geometry. Select your file and enter the relevant parameters.



The **Components** list on the left contains each imported model. Click on the name of a model to select it. With a model selected, the **Transform** menu appears, which allows you to move and resize your model.

You can click on **Show 3D** to see a preview of what the file will look like in three dimensions. When you are happy with the settings and placement, click done.



Imported .STLs need an outline vector to contain the geometry. Right click on the component, **create outline vectors**, and set height to zero.

Height = 0 means the outline is at the base of the geometry

4. Generating tool paths

Each toolpath type carves material differently. Your file will likely use multiple toolpaths. These are some of the most common types:

2D Toolpaths

Contour - This toolpath carves your material along the vector line. You can select if you want the drill bit to carve inside, outside, or centered on the vector line.

Pocket - This toolpath removes material within the enclosed area of the vector, to your selected depth.

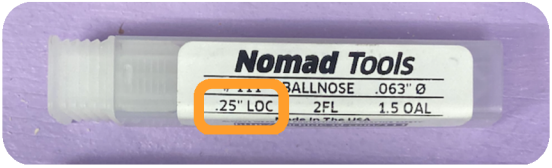
VCarve - This toolpath creates a V-shaped cut along the vector lines using a V-bit. It adjusts the depth of the cut dynamically based on the width of the vector, making it ideal for engraving detailed designs or text. The depth is determined by the width of the vector and the angle of the V-bit.

3D Toolpaths

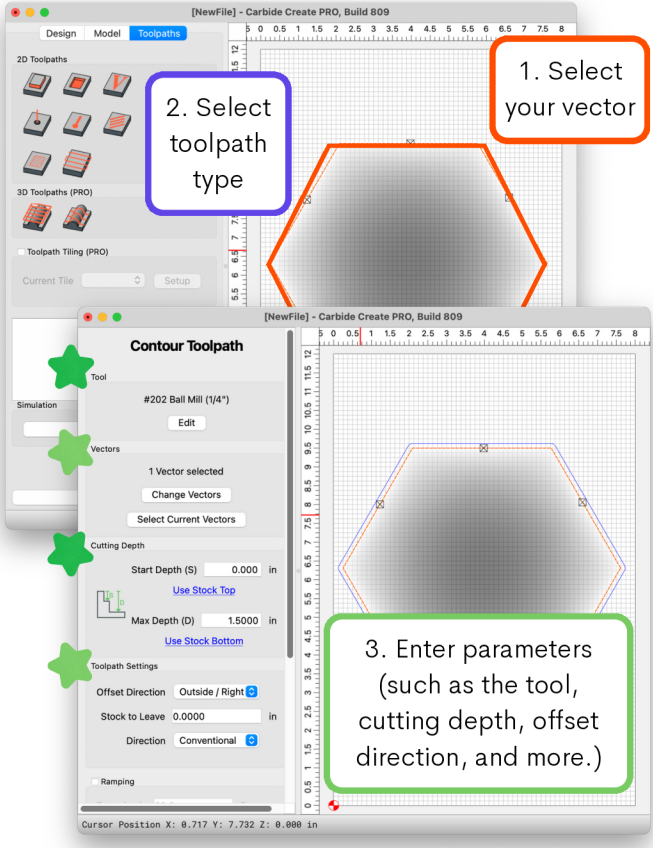
3D Rough – This toolpath clears out material in rough layers to reveal the 3D geometry contained within the vector outline.

3D Finish – This toolpath smooths out the details of the 3D geometry contained within the vector outline, after the roughing toolpath has first cleared out the bulk of the excess material. This type of toolpath travels along all 3 axes simultaneously and isn't contained to a single plane at a time.

Also keep in mind the maximum cut depth of each tool you use! The cut depth should not exceed the **Length of Cut (LOC)** as found on the side of the tool bit case:



To generate a toolpath:



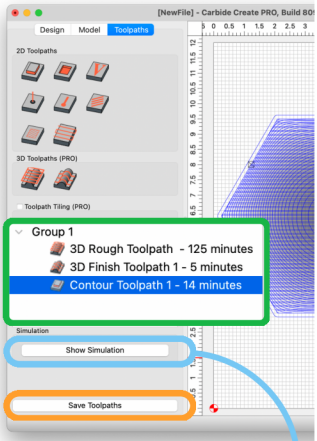
Here you can see **the sequence of your toolpaths**, and drag to reorder them.

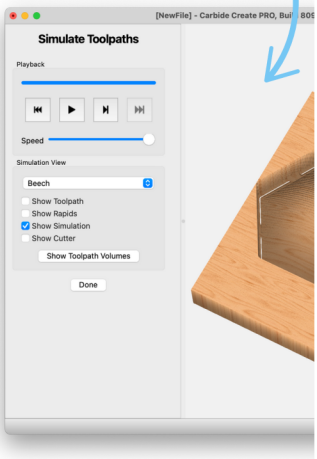
Make sure Roughing is before Finishing!

Also note **the duration estimates per toolpath.**

Click the **Show Simulation** button to see a preview of the cut. This is a good time to double check your tabs too!

When all the toolpaths are complete, click **Save Toolpaths** and select “save toolpaths in this file”



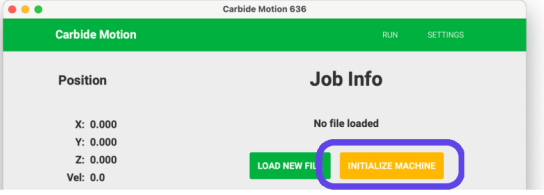


Part 6: Sending the Instructions

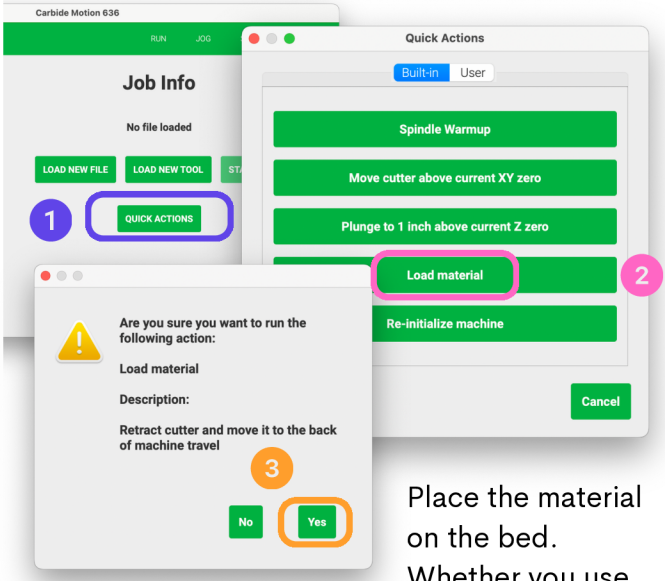
It's time to turn on the CNC! **The switch is located on the underside of the control box.**

Now we can open our controller software: Carbide Motion. **You cannot edit your design or tool paths at this point.** You can only control the CNC, which includes driving it to different locations, changing bits, loading files, etc.

Make sure the CNC bed is clear of stray items, and then click **Initialize Machine.** The spindle will move to a corner of the bed so that it can figure out where the spindle is currently in relation to the rest of the machine.



To move the spindle so we can load the material:

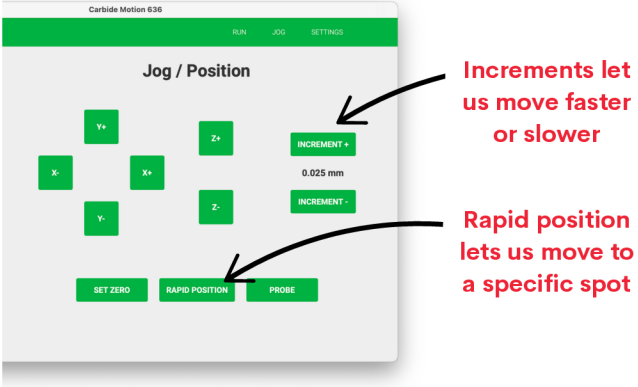


Place the material on the bed. Whether you use clamps or another workholding method, make sure that your material is secure in every direction!

Setting the origin

Now that we know where our material is, we need to tell the CNC. We will do this by using the **BitZero** and the **Probe** to set the origin.

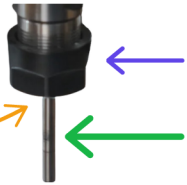
We can use the **Jog** tab to move the spindle to different locations along the X, Y, and Z axis.

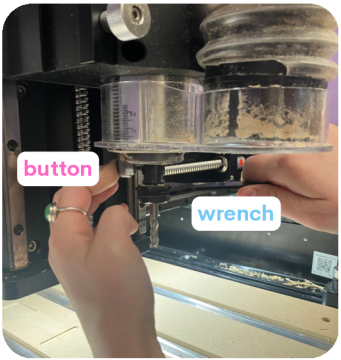


First, jog the spindle to the front of the machine so that you can insert the probe.

Inserting the probe

Insert the **probe**, the smooth, rod-like bit, into the **collet** inside.

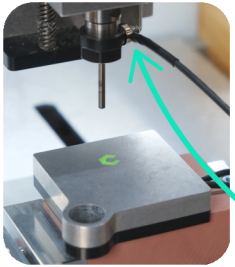




Press the **button** just above the **collet nut** to secure the bit from moving around.

Position the **wrench** on the **collet nut** and tighten it by *turning* counter-clockwise .

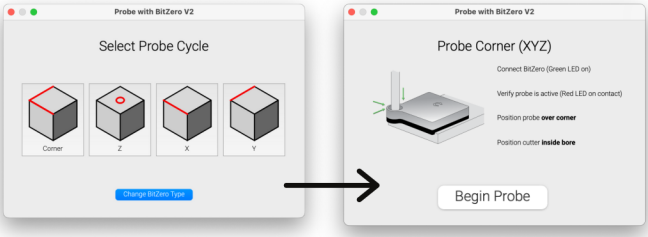
Probes and collets come in two diameters (1/8" and 1/4") to fit different bits. If needed, the collet nut can unscrew completely to change out the collet inside.



Place the **BitZero** where you set the origin in Carbide Create, typically the lower left corner.

Attach the **magnetic lead** to the **collet nut**.

Jog the machine so the probe is positioned into the **center of the BitZero hole** and *click* **Probe** from the **Jog tab:**

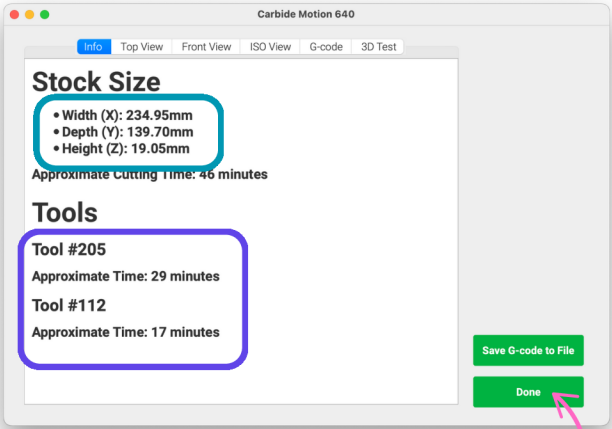


The machine with move between the hole and the **BitSetter**

Now the CNC is ready to read our instructions!
In the **Run tab**, *click* **Load new file**

Double check that all information is correct.

Pay attention to **stock size**, **tool(s) #**, and the approximate **time of each cut**.



If everything looks good, *press* **Done**.

Part 7: Time to Carve!

A safety reminder before we start:

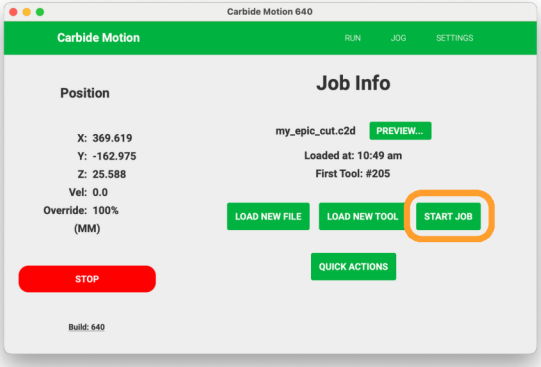
The **Emergency Stop** is located outside of the machine. It will stop the spindle from moving along the XYZ axis, but won't stop it from spinning. **You will not be able to resume your cut after using the Emergency Stop.**

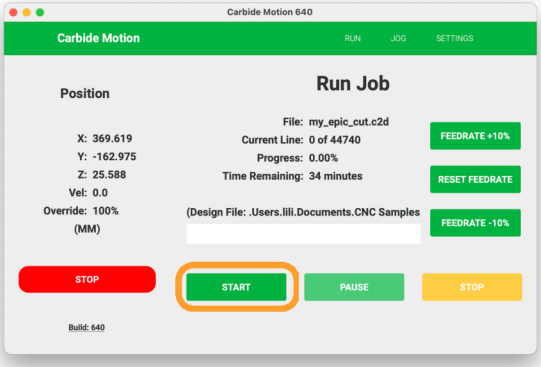
The leftmost button turns the **Spindle On/Off.** **Ensure the spindle is off whenever the doors are open, especially when you are changing out bits.** **Think light on = spindle on.**

Be sure to keep an eye on your cut while it's running, and alert a staff member of any issues!

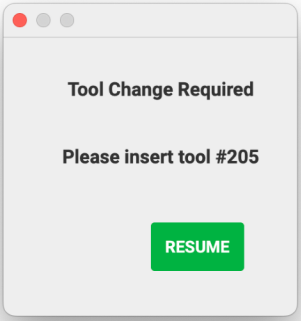
We're ready to start carving!

1. Click **Start Job** and **Start** on the next screens





2. You'll then be prompted to insert the first tool. **Make sure the spindle is off for this part!**

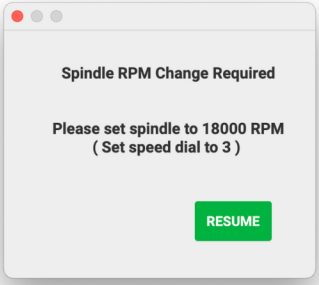


(Tools are inserted in the same way that the probe was.)

Each time you change the tool, it will move back to the **BitSetter**

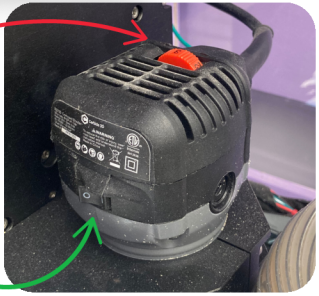
Carbide Motion will prompt you when it is time to change the tool bit between toolpaths. **Remember to turn off the spindle whenever you are opening the doors.**

3. You will now be prompted to set the RPM



Set the spindle speed using the **red dial** at the top.

Double check the switch on the side to ensure the spindle is activated.



4. Close the doors and turn on the spindle using the button on the outside of the machine **before** clicking resume.

When you click resume, your cut will start!

When you're done

1. Remove your material (you may need to "Jog" the spindle out of the way to reach it)

2. Return each tool to their container. Be sure to remove the tool from the spindle when you are done carving.

3. Sweep or vaccuum any sawdust.

4. Turn off the machine!

5. Any tabs will be removed manually with a knife or a saw. You can ask a staff member for guidance with this!

More Resources

For detailed instructional videos including software tutorials:

https://my.carbide3d.com/

Image Sources

- https://toolstoday.com/carbide-3d-shapeoko-pro-cnc-router-machine.html
- https://m.media-amazon.com/images/I/61Jb8IZ0kQL._AC_UF894,1000_QL80_.jpg
- https://shop.carbide3d.com/products/sweepy-2-0-dust-boot?_pos=1&_sid=cf89951c7&_ss=r&variant=32975720972349
- https://www.precisionpeaks.com/store-1
- https://www.create.bainbridgebarn.org/AssnFe/ev.asp?ID=5317993
- https://www.stylecnc.com/user-manual/CNC-router-machine-cut-quality-and-accuracy.html
- https://waykenrm.com/blogs/undercut-machining-for-cnc-machined-parts/
- https://waykenrm.com/blogs/machining-sharp-internal-corners/
- https://www.matterhackers.com/store/l/carbide-3d-shapeoko-5-pro-cnc-router
- https://community.carbide3d.com/t/fit-router-bits-into-collet-on-vfd/63579
- https://carbide3d.com/carbidemotion/

Design Center

BARNARD COLLEGE