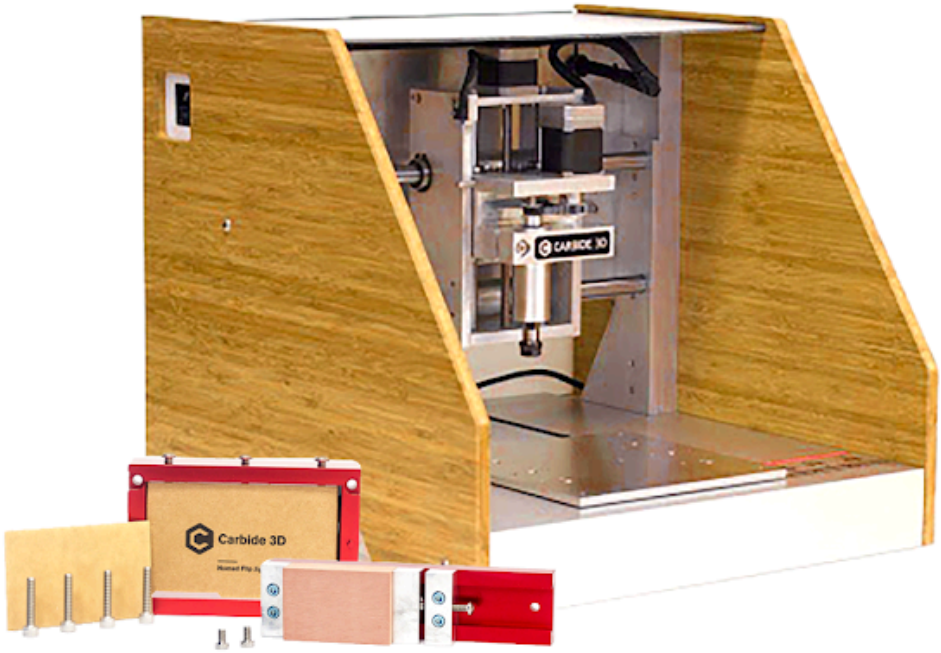
NOMAD 883
USER GUIDE & TUTORIAL



PCC Sylvania MakerLab 2024

Table of Contents

1) Table of Contents

2-16) Machine Start Guide

17-27) First Project Tutorial: Bottle Opener

28-29) Introduction to Endmills and Bits

30-31) Feeds and Speeds

32) Common Problems

33) Additional Resources and Project Ideas

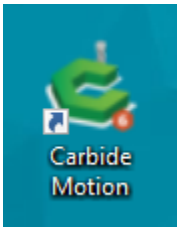
Machine Quick Start Guide:

To start a project on the Nomad 883 CNC Router please ensure that you have the following prepared

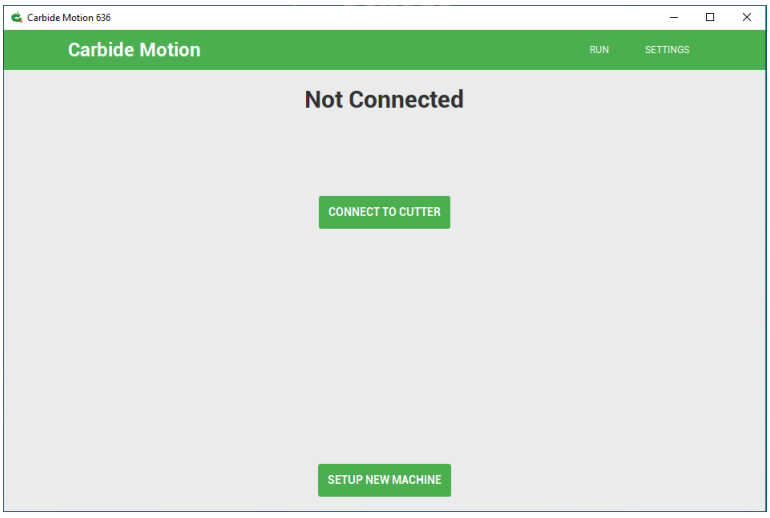
1. PPE properly fitted to your person, safety glasses are mandatory, hearing protection is optional depending on the material.
2. Material for your project, cut down to a maximum size of 8x8x3 inches
3. Workholding (such as clamps, screws, or carpet tape)
4. A prepared Project file in the .nc or .c2d file formats (these can be created in Carbide Create or Fusion 360)
5. Your required endmills and engravers
6. Both collet wrenches (as pictured below)

Step One: Open Carbide Motion from the desktop on the computer connected to the CNC router you intend to use. The computer should be located directly next to each respective machine on a shared table.

Click on this Icon →



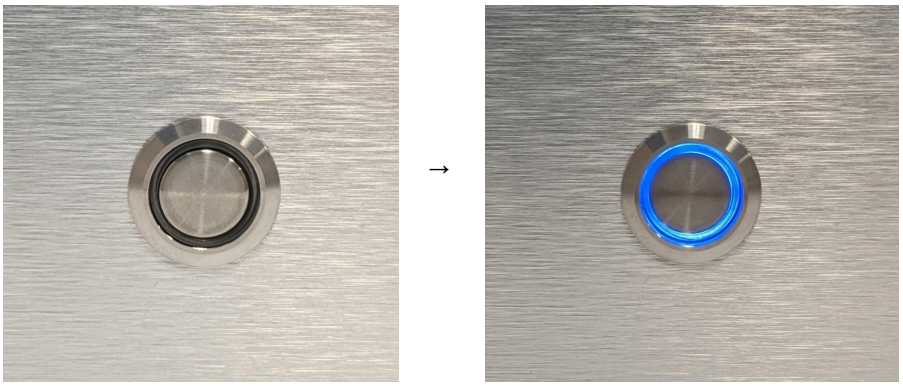
It will bring you to a window that looks like this →



Do not press anything yet

Step Two: Turn on the Nomad by pressing the metal button located on the bottom right of the machine's front face.

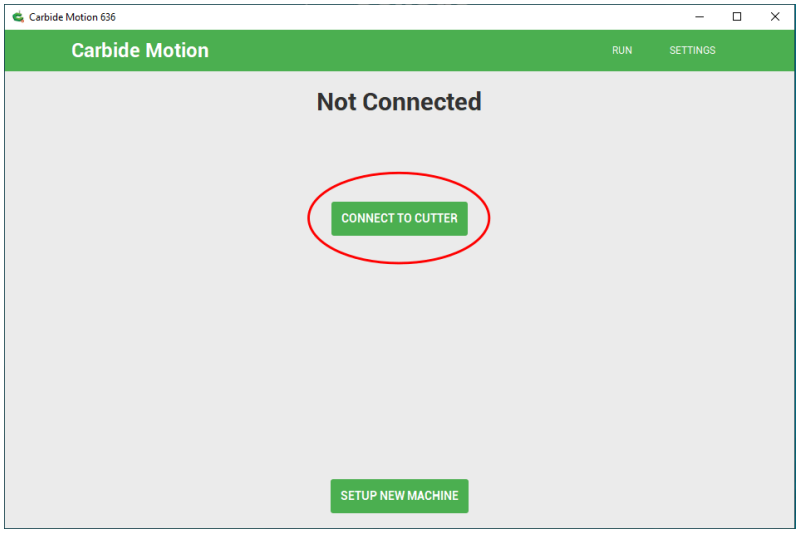
The button should light up blue and you may hear a short and sudden jolting sound, this is normal and expected.



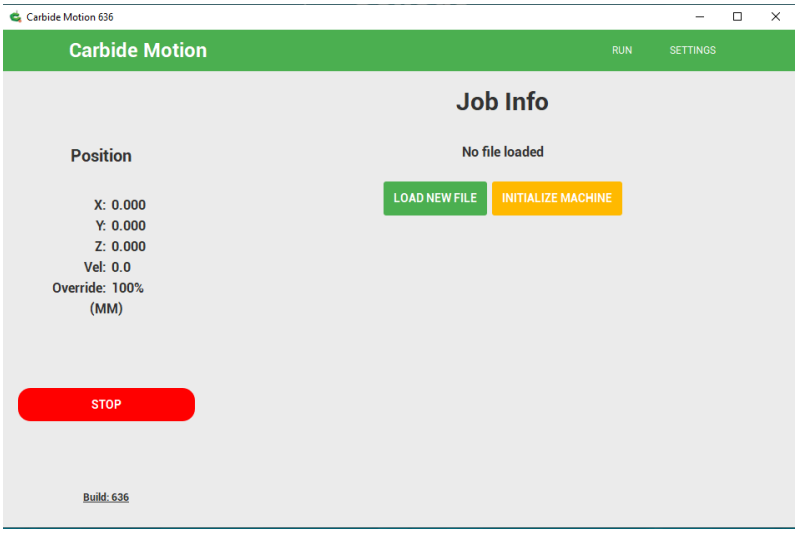
You can always tell the state of the machine by the button color, blue indicates that it is turned on, no light at all indicates that it is turned off.

Step Three: Return to your opened Carbide Motion window and press the “connect to cutter” button which should now be green

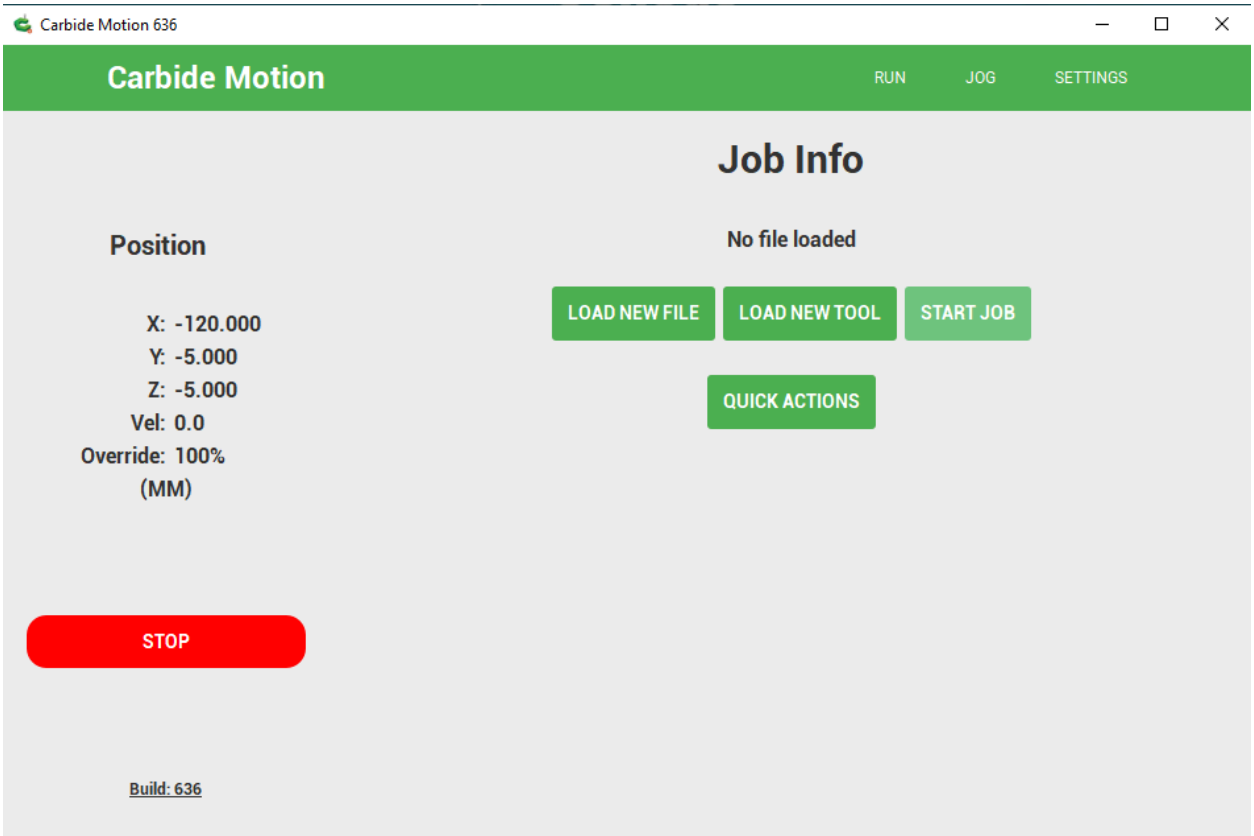
This will bring you to a second screen with a few options, “load new file” and “initialize machine.”



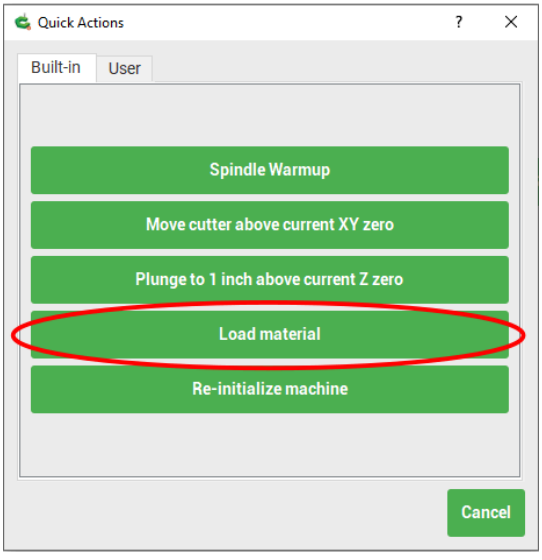
Before proceeding make sure everything is **clear of the machine**, including hands, tools, or stray work materials. **The machine is about to move.** Once the machine is clear, press “Initialize machine” this will cause the carriage to move and test its limit switches.

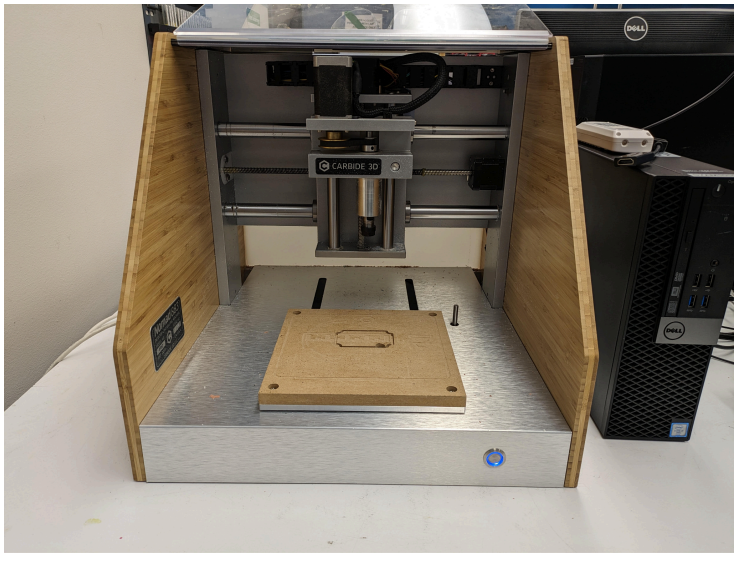


Step Four: After step three is complete you should be looking at the below screen.

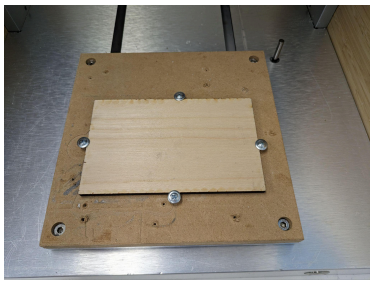
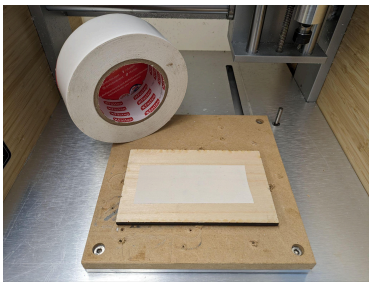
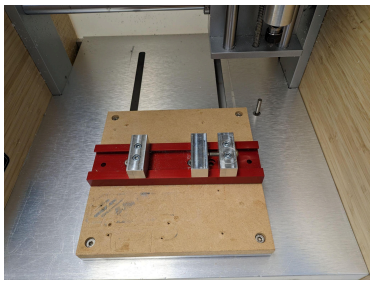


From this screen select “quick actions” this will bring up a menu with several commands, which you will then select “load material” from. This will send the bed of the machine all the way to the front.





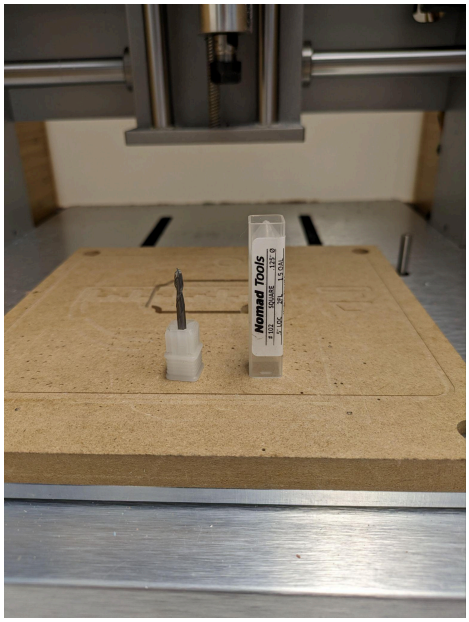
Step Five: Attach your material firmly to the bed,this can be done in a variety of ways using screws, carpet tape (this is comparable to double sided duct tape), clamps, or vises

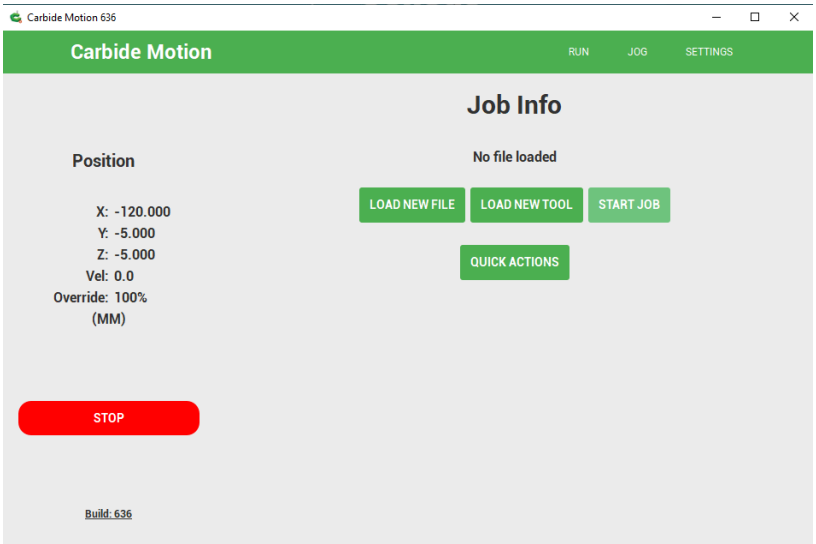
Step Six: Now that you have loaded your material it is time to install your first tool. From the wall you should be able to find 2 wrenches, one bigger and one smaller. As well as whatever tooling you will be using, each one should be labeled.

If you are using multiple tools it can be convenient to locate all of them now, during this step and setting them aside in order for further use.

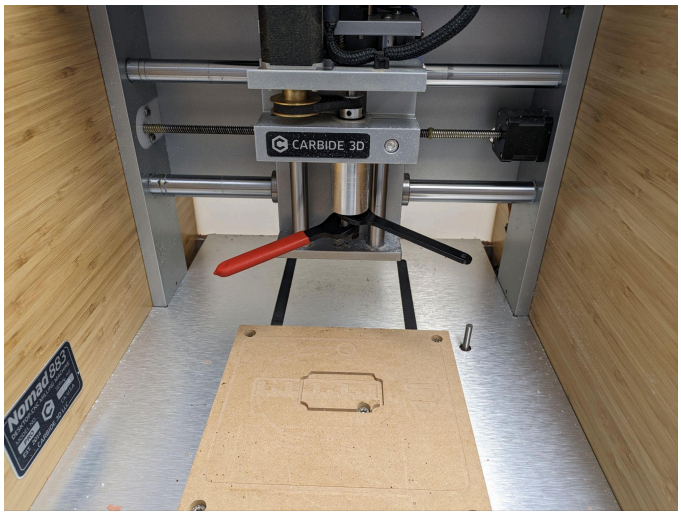
Remove your first tool from its protective cover by grabbing the white base and pulling it out like a cork. Take care to place the cover somewhere you will remember, they are individually labeled to each tool and it is important that you return the right tool to its cover after every use. Then you can remove the tool from the base, **be careful** while grabbing the tool as some may have **sharp edges or points**. Finally set the tool to the side for now.



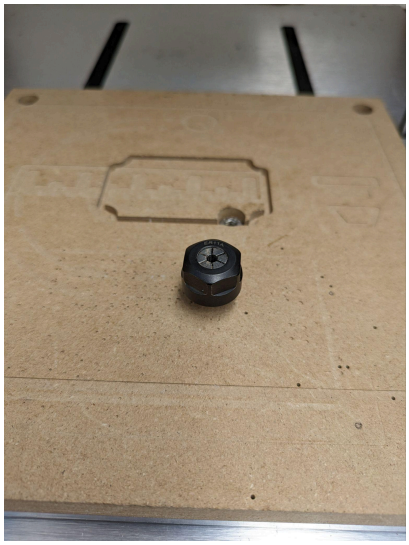
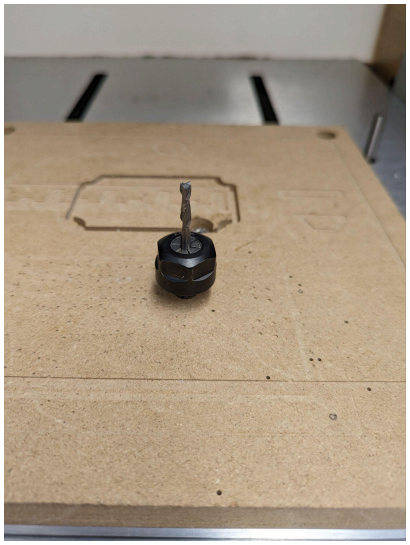
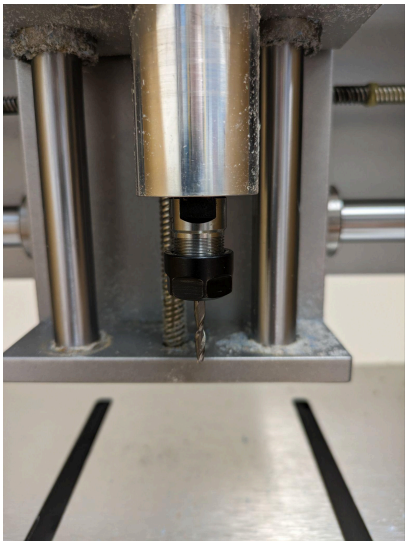
After this, return to carbide motion and select “load new tool” this will cause the machine to move the spindle to the center of the machine, so be careful not to have anything loose within the enclosure.



Next, take the two wrenches and use them to loosen the collet, this is the black nut located at the bottom of the spindle. Place the wrenches as seen below with the black wrench on the top shaft where it has corresponding flats and the red wrench on the bottom most piece. Push away from yourself moving the wrenches further apart to loosen the collet. It is a standard thread with left to loosen and right to tighten.

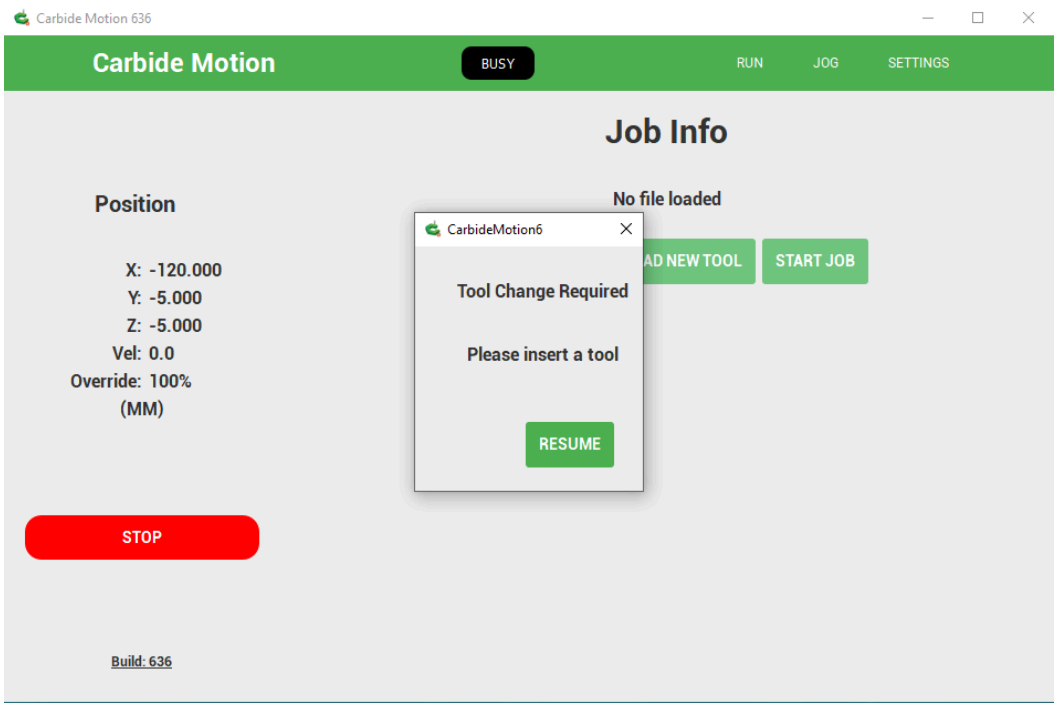


Once the collet is loosened you may remove any tools that may be in the machine (taking care to return them to the correct spot) before inserting your own, this can be done either on the spindle or while the collet is entirely removed like it is shown here


→

→


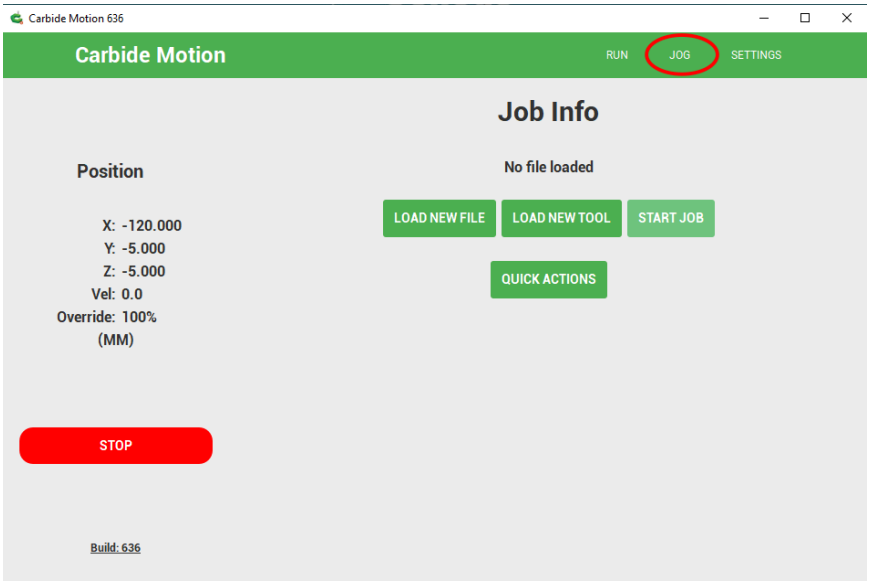
Then tighten repeating the above step for loosening in reverse, making sure to get a very snug hold on your tool. This is important for the safe operation of the machine.

Now you may close the lid of the machine making sure to **remove the wrenches** and any other loose parts before selecting “resume” in carbide motion. This will prompt the machine to move to the far right and probe the tool against a button to measure its length. Don't worry it is supposed to do this a few times. Once the machine has finished moving the tool has been successfully loaded, double check that you have loaded the **correct tool for the job** you are about to start.



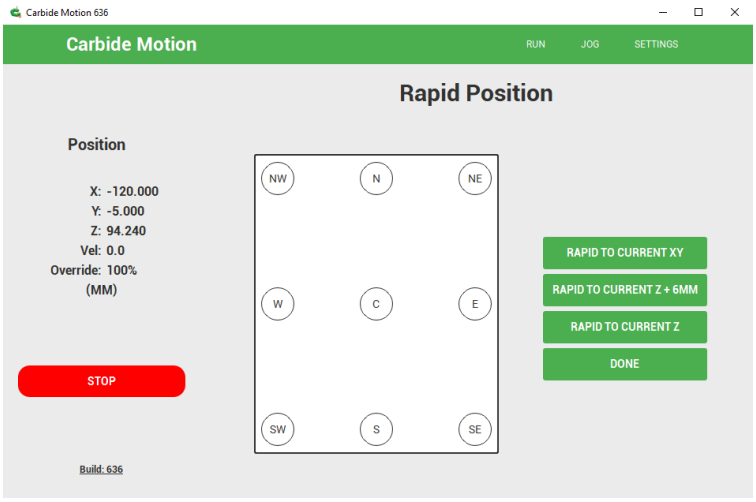
Step Seven: Select the “Jog” tab in the top right of Carbide Motion

This will bring you to a control board for positioning the Nomads gantry

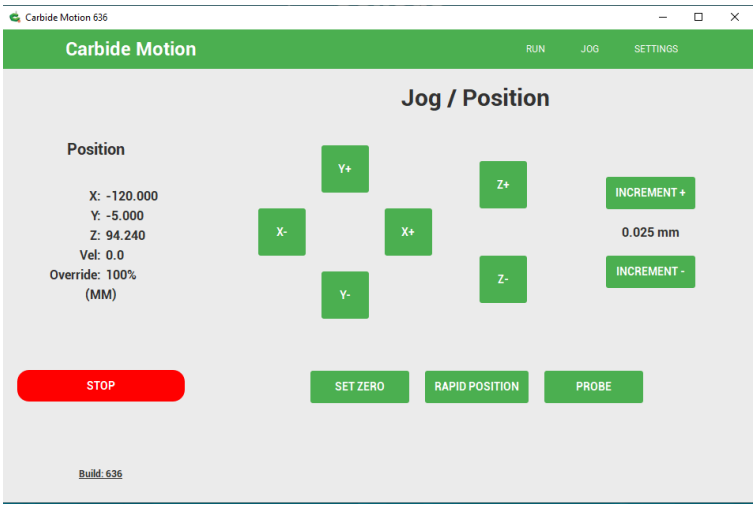


From this control board you will be able to position your origin on the work piece, the origin is what the machine will consider the starting point for your project. As long as the stock is bigger than your project the origin does not need to be exact in the X and Y directions.

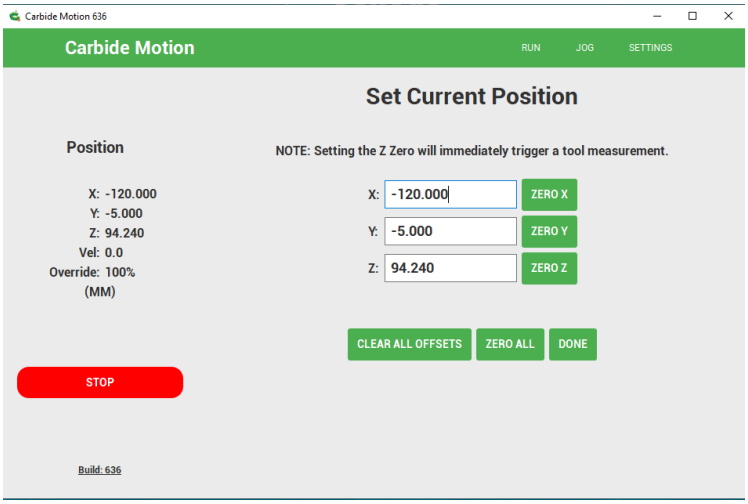
Using this page, move the tip of your endmill to your project's origin in the bottom left corner relative to the front of the machine. To quickly reach an approximate position you can select “rapid move” on the bottom of the jog screen and then select a corner from the screen.



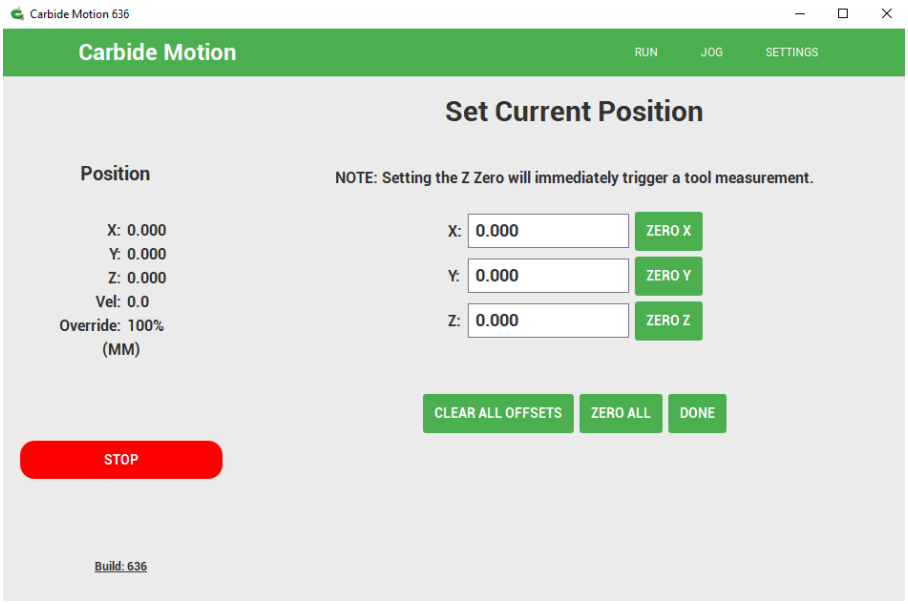
You can adjust the speed the machine moves at by increasing or decreasing the value located on the right side of the screen. Do this by using the buttons labeled “increments ( )” above and below it respectively. All axes can be positioned visually for most projects, decreasing to lower increments of movement to achieve a more precise result. For the most accurate result on the z axis you can lower the z axis onto your material with a sheet of paper on top until you can pull the paper out with slight resistance.



Once you have the tool positioned exactly where you want it in the x,y, and z directions you can press “set zero.” this will bring you to a page listing the machine coordinates, these are the coordinates that the machine sees from its own origin point, but they are not important for now.



Next select “zero all” this will set all the values to zero and it may prompt you to re-measure the tool, you can ignore this and press ok. When you are done your screen should look like the picture below.

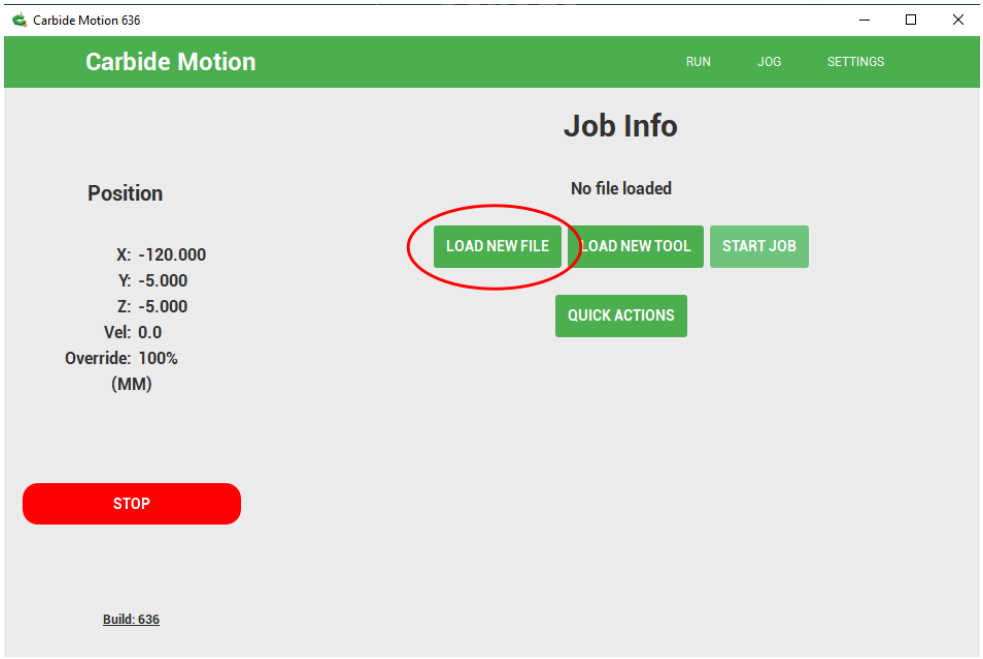


When everything is correctly positioned and your Origin point is set (showing all zeros at current position) you may return to the run screen by selecting it in the top right toolbar.

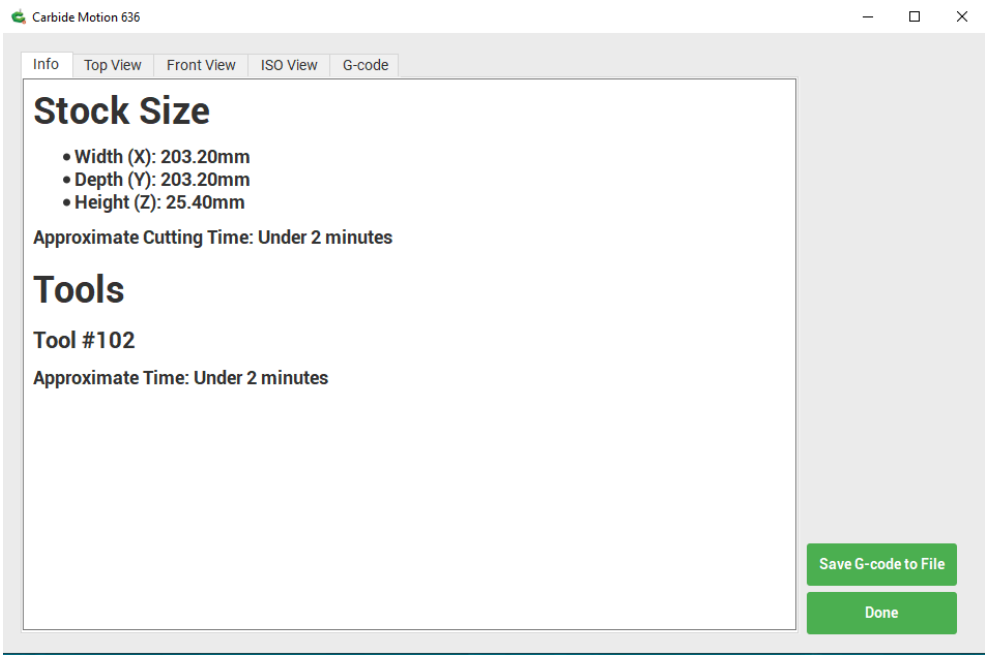
Checkpoint: Find a staff member to verify:

- ☐ Correct tool for the job
- ☐ Tool and colette are installed correctly
- ☐ Material hold down technique
- ☐ Verify Zero for X, Y and Z
- ☐ Review CAM file
- ☐ Verify Feed, Speed, Plunge Rate
- ☐ Verify Stock Height
- ☐ Verify Model Position

Step Eight: Next we will load your toolpath file (you can learn to make one further into this document if you do not have one already.) Start by selecting “load new file”



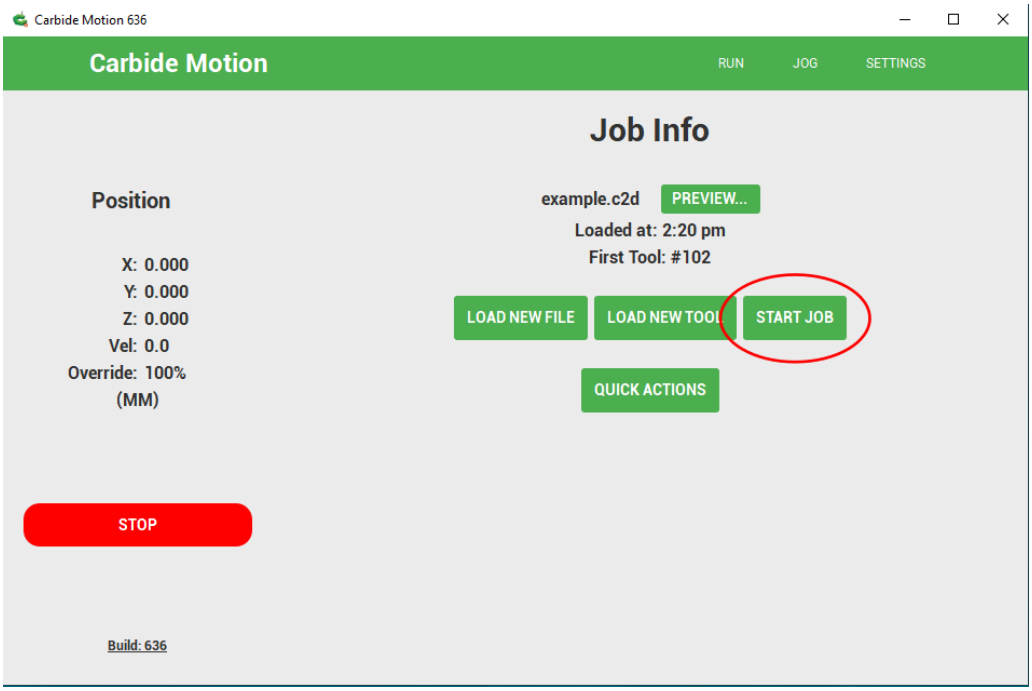
This will bring you to a file explorer window from which you can navigate to your file and select it. This should load the file into Carbide Motion and automatically bring you to an information page displaying different stats about your project.



Read over this page and make sure to confirm all of the information including the tools used, the tool order, and the stock size. The stock size does not need to be the exact size of your actual stock but it must be able to fit inside of the stock you are using in the x and y directions. The height, Z, must be the same or very close between the displayed and actual stock. Once you have verified everything, select the **“done”** button from the lower right corner.

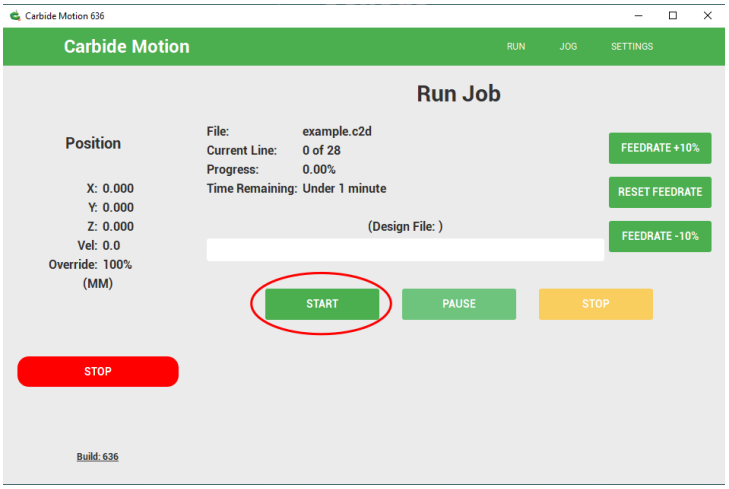
Please note that if you are running a toolpath generated in fusion 360 it is possible that the listed “approximate time” may be very inaccurate.

Step Nine: Once you have verified that everything is correct, your origin is in the right starting position, your stock is firmly affixed to the bed, and you have the right tool loaded. **Close the lid** of the machine and hit “start job.” this will bring you to the job progress screen, where you can monitor certain aspects of your project.

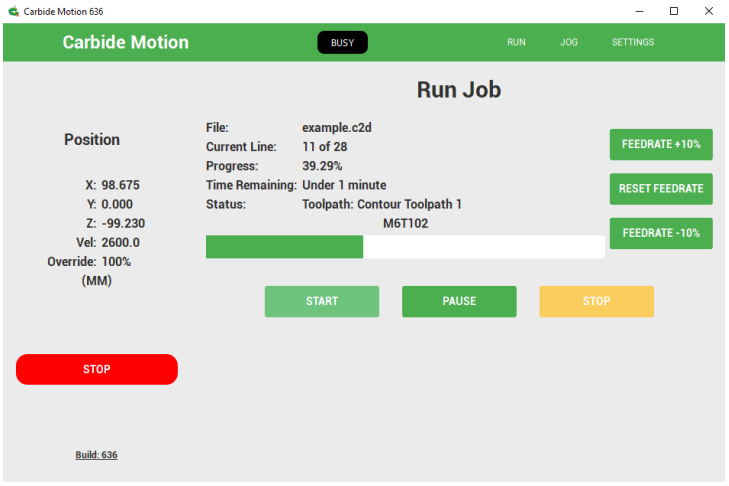


From this page hit “start” it may prompt you to load another tool, assuming the correct one was loaded in the previous step you can ignore this and hit “resume” this will reprobe the tool before starting your job, once you hit “Start” the machine will start the spindle, this should start quiet but will become very loud when cutting some materials. This is normal. At this point the job has started and the machine will **start moving**. Make sure you monitor throughout the entire cutting process. After a few minutes if

nothing has gone wrong you may move about the room and monitor the machine by sound, but you must be able to hit stop within a few seconds of any errors.



During the Job carbide motion will display a few settings and controls, importantly **“STOP”** and “pause” which you may need to use in the event of any failures. The most common of which is breaking a tool or your stock becoming loose. These things happen and it is ok to restart. This page also shows your progress bar for the entire job.



Step Ten: When the machine has finished cutting, turn off the machine, vacuum all remaining dust from the enclosure, making sure to move the bed and vacuum under it. Then you may remove your finished product. Please return all tools and materials to where you found them and to return the machine to the state you found it in.

Final tips:

Tool Changes, if your project uses multiple tools the machine will pause and you may be prompted to change the tool, follow the procedure from step seven and then hit resume.

Hitting pause instead of stop will let you resume the job, hitting stop will cause you to need to restart the job.

Pausing at various points in the job to vacuum off the stock material is sometimes necessary to keep a good surface finish and to keep the tool from overheating, if you are doing a long job with the same tool feel free to do this as often as you'd like but its only really necessary if you notice material sticking to the tool.

First Project Tutorial: Hardwood Bottle Opener

This can be started outside of the lab. You will need Carbide Design software.

Download here: https://carbide3d.com/carbidecreate/download/. However,

make sure to read through what is needed below.

In order to complete this project you will need the following

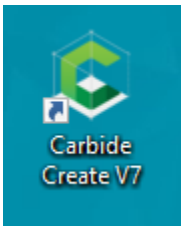
1. Hardwood material blank, this can be procured from any makerspace staff member on request
2. Bottle Opener Metal insert, also available on request
3. Carpet tape, available in the “Adhesives” drawer of the blue tool cart
4. Measuring device, calipers and a ruler
5. Anything listed in the previous section “*Machine Quick Start Guide*”

Recommended:

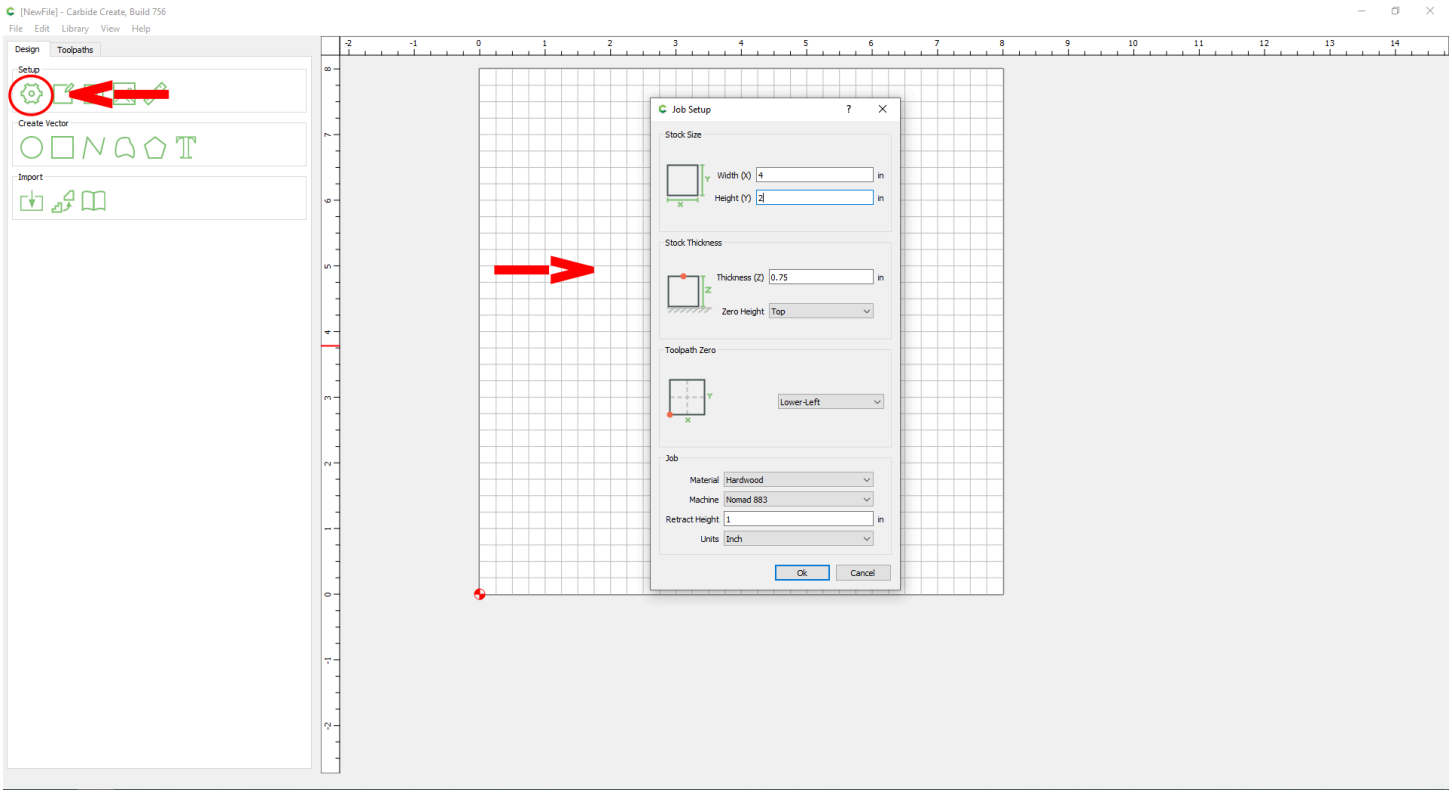
1. Basic understanding of Inkscape or Illustrator
2. Basic understanding of calipers
3. Laser Training (this is optional but allows you to further customize your project)

Step One: gather your materials and download the Bottle Opener Blank svg

Step Two: locate and open carbide create, this should be found on the desktop and looks like this →



Step Three: upon opening carbide create you will see a blank work space, select the gear shaped button in the top right, this will open a menu where you can input the dimensions of your stock, select the material you are working on, and set the work origin point



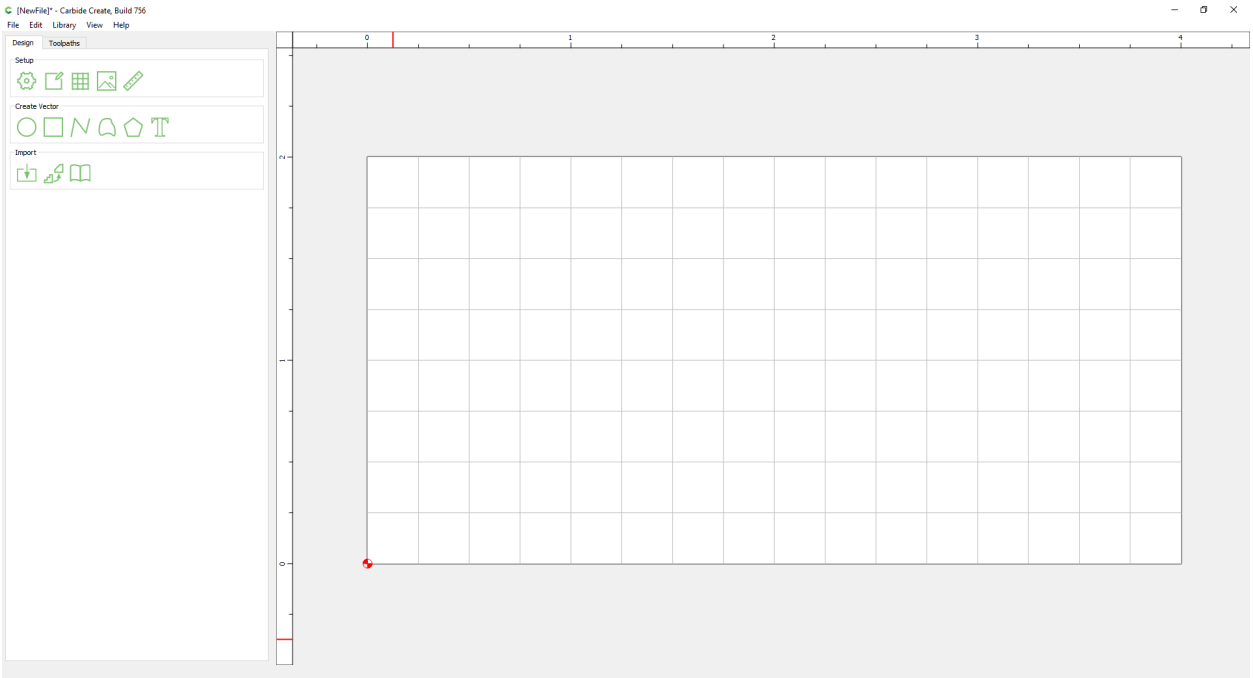
For this project you will need to measure your selected stock, it should be an unknown width (which is what you will measure and put in for x) a known height (Y) of 2in and an unknown thickness (Z).

Set your selected **Toolpath Zero** to lower-left if it is not already and verify that your "**Zero Height**" is set to top.

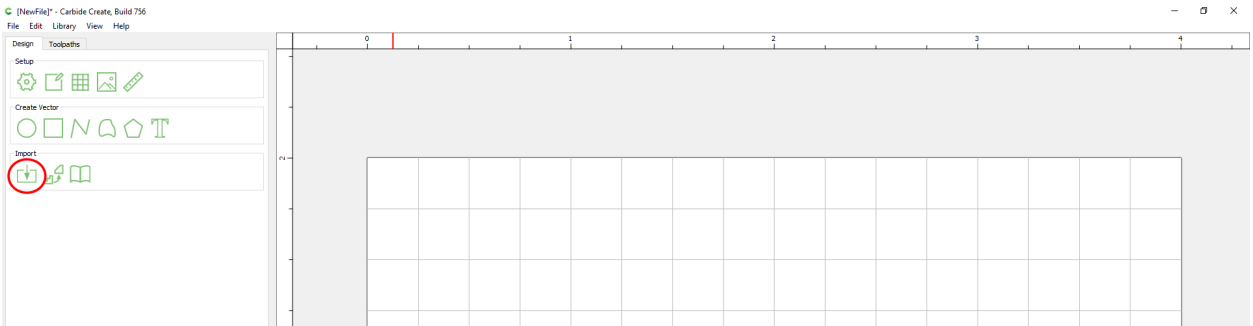
Finally change your material to "hardwood", your machine to the "Nomad 883", and make sure your units are set to inches

Then hit "OK"

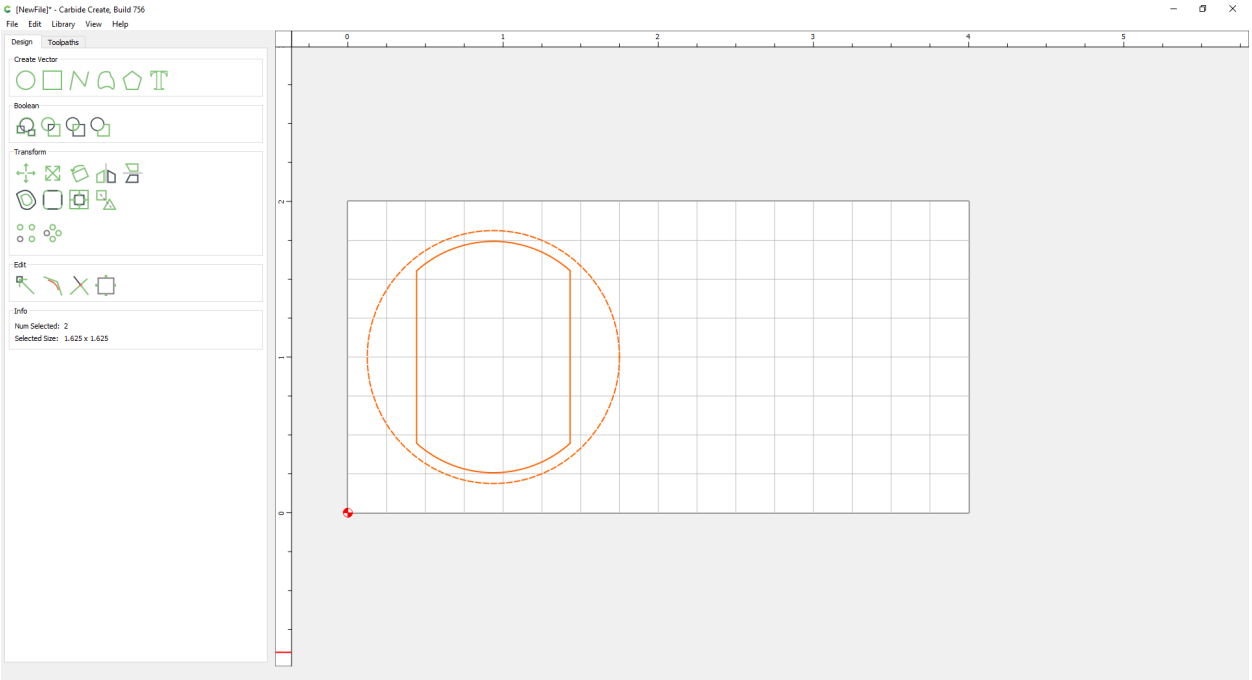
This should change your work area (the white grid space) to be a smaller rectangle



<u>Step Four:</u> select the import icon under "import" and navigate to the svg you downloaded earlier, selecting it, and hitting "Open."



This should open up the svg within or near your work space. The geometry of the bottle opener graphic is important for it to fit the hardware, so **do not alter** it. However, it should be easy to click and drag the graphic to anywhere you want within the work area (grid), just make sure that all parts of the graphic are moving together. If you make a mistake, you can go to Edit tab and Undo, or ctrl + Z on the keyboard.

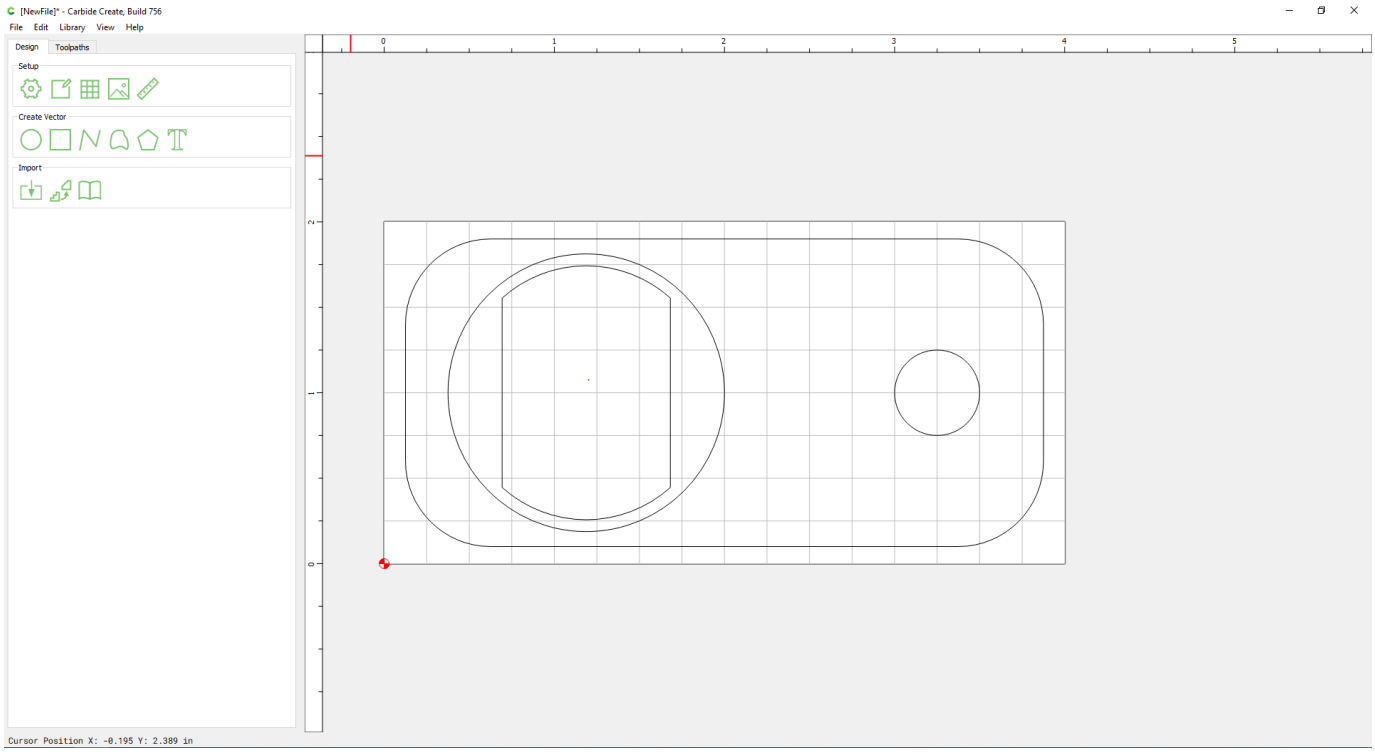


<u>Step Five:</u> Design your bottle opener, this can be done in Carbide create or in Inkscape (saved as an SVG and imported like the last step.) as long as it fits within your measured stock. Please try to avoid holes tighter than 0.5in

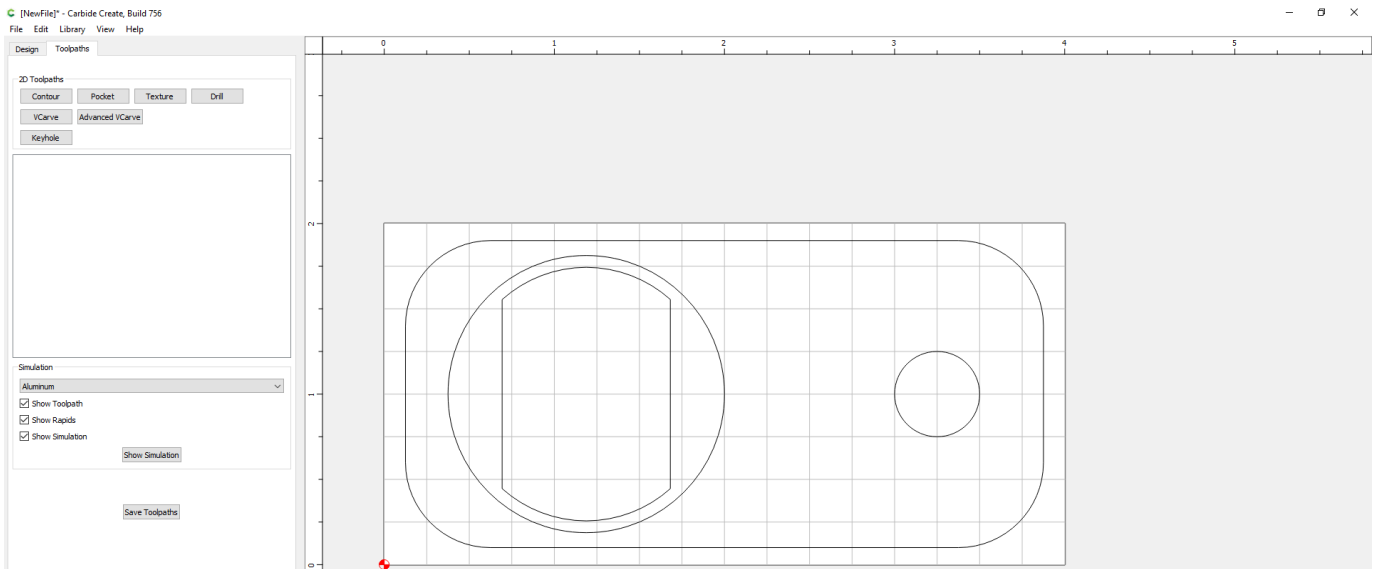
Keep in mind when designing your bottle opener that all features will be cut out as a top down silhouette from a single side. For example the bottle opener insert from the last step will be cut as a half inch deep hole with a 1/8in deep hole around it. These different

depths allow us to create 3D geometry. This style of design is often called 2.5D design because you are limited in the third dimension.

Here is an example of a possible design, while it is entirely fine to replicate it, you do have the freedom to design anything you wish within the constraints of your stock.



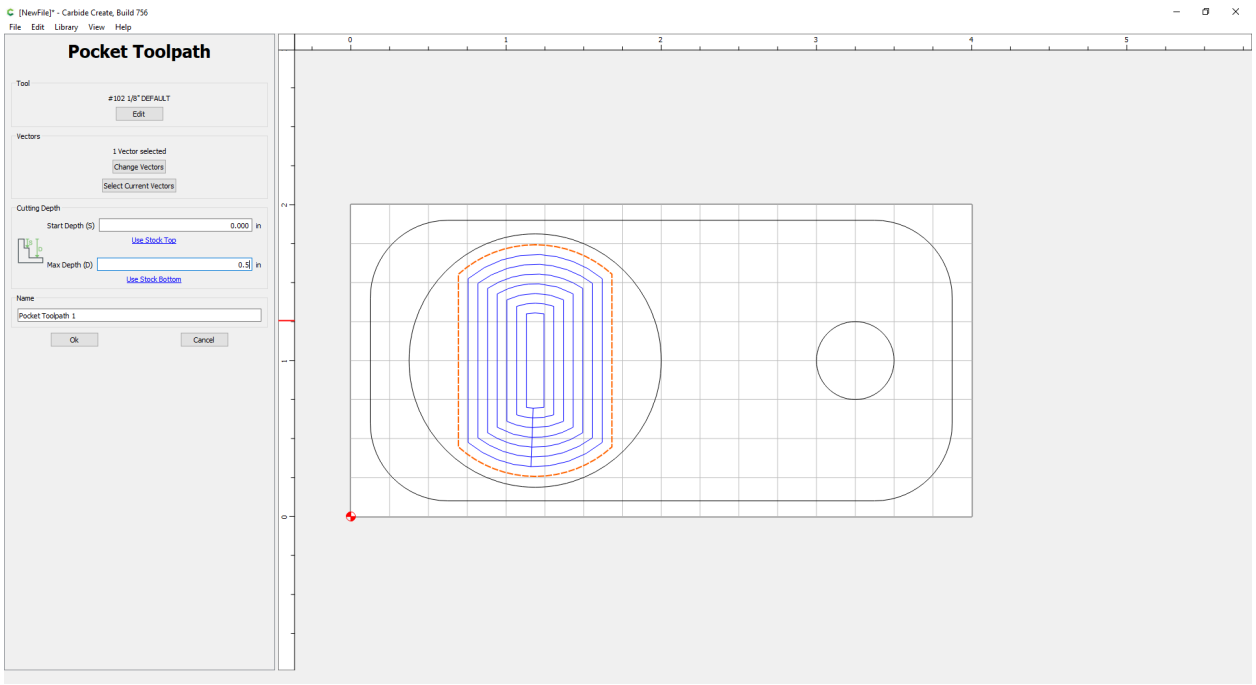
Step Six: Next we will set up the toolpaths, these are what the machine will look at in order to carve out your design. Click on the toolpath tab in the top left



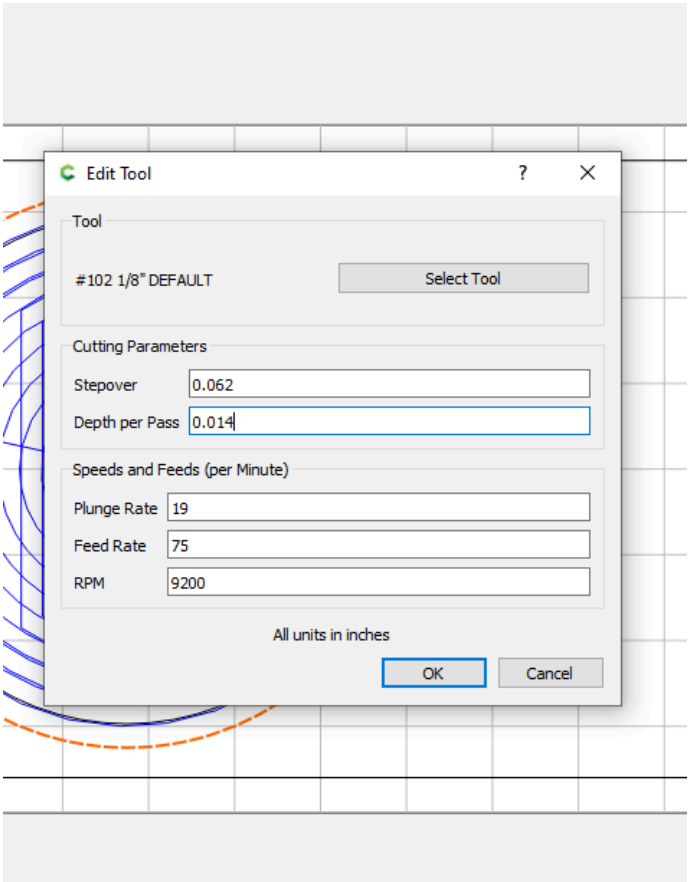
then select the first outline you would like to cut, working from shallowest to deepest is usually advantageous but this order is not a necessity and should be determined by each project's unique requirements. When the outline you wish to cut is selected click on one of the seven toolpath options, finally select “use current selection” from the pop up, the most basic one are:

- Contour: this toolpath will cut only along the selected line this is great for cutting out the final shape of your project, or borders around your designs
- Pocket: this toolpath will cut everything within a selected closed shape, if you select two shapes it will cut everything between the inner and outer shape. This is great for roughing out geometry, creating inset sections, and any part of a design that needs to be flat and at a different level from the rest.
- Texture: this toolpath will fill everything within the selected shape with a rough texture reminiscent of gouged wood grain. It can be used to provide contrast between the background and raised elements of a design such as text on a sign.
- V-Carve: This toolpath will take a pointed engraver bit and try to carve out the space between two selected lines or within a thin selected shape, resulting in an engraving with sloped sides. This toolpath is mostly used to carve small text or images into a project.

For this example the pocket toolpath was selected, from this screen you can adjust the start depth and max depth of your project. For the bottle opener make sure that this innermost pocket is set to a starting depth of 0.0in and a max depth of 0.5in, under the **Cutting Depth** section.



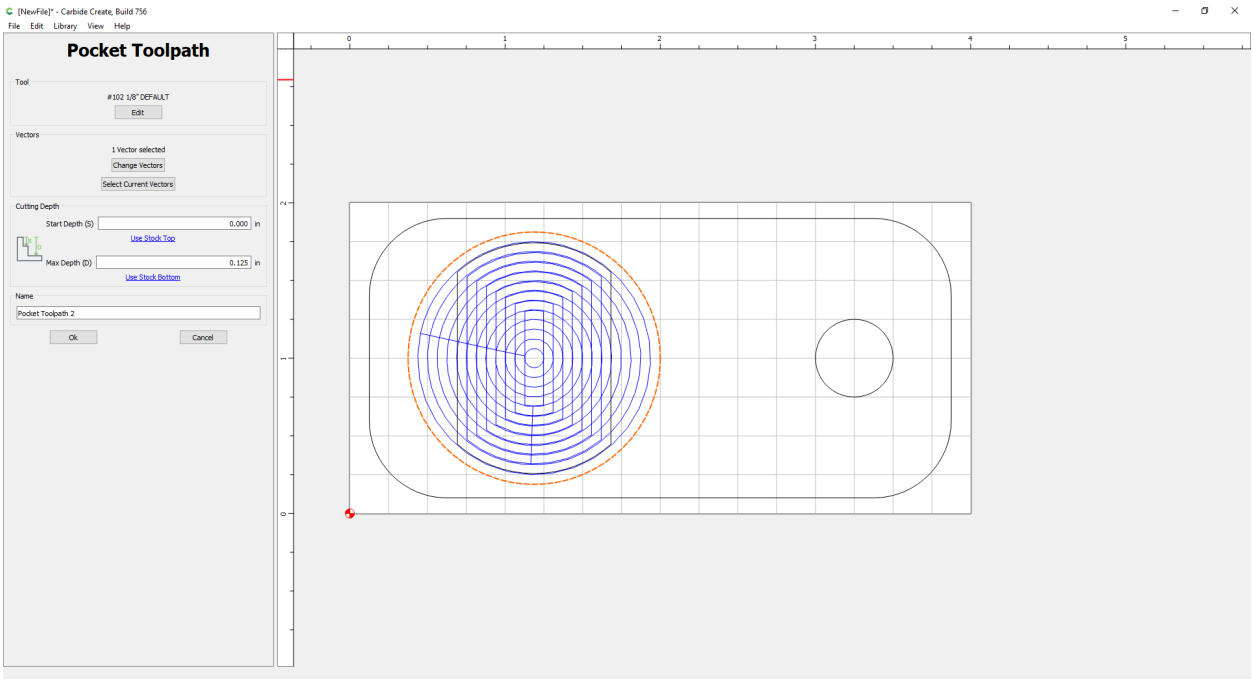
To select the correct tool type, select “Edit” under the **Tool** section. This will bring you to a popup window, for this project you can leave the tool as the default #102 1/8in endmill, but you will need to change the settings of the toolpath. These settings are usually referred to as “feeds and speeds”. It is very **important that your settings match** this image exactly. These are



specific for hardwood (a full list provided on a later page). Click ok.

Double check that all of your depths are correct, your settings are right, and the correct tool is being used, then hit “OK”. This will exit you to the previous screen but with your new toolpath listed.

Step Seven: Next select the outer ring of the bottle opener insert pocket. Then select the pocket toolpath once again.

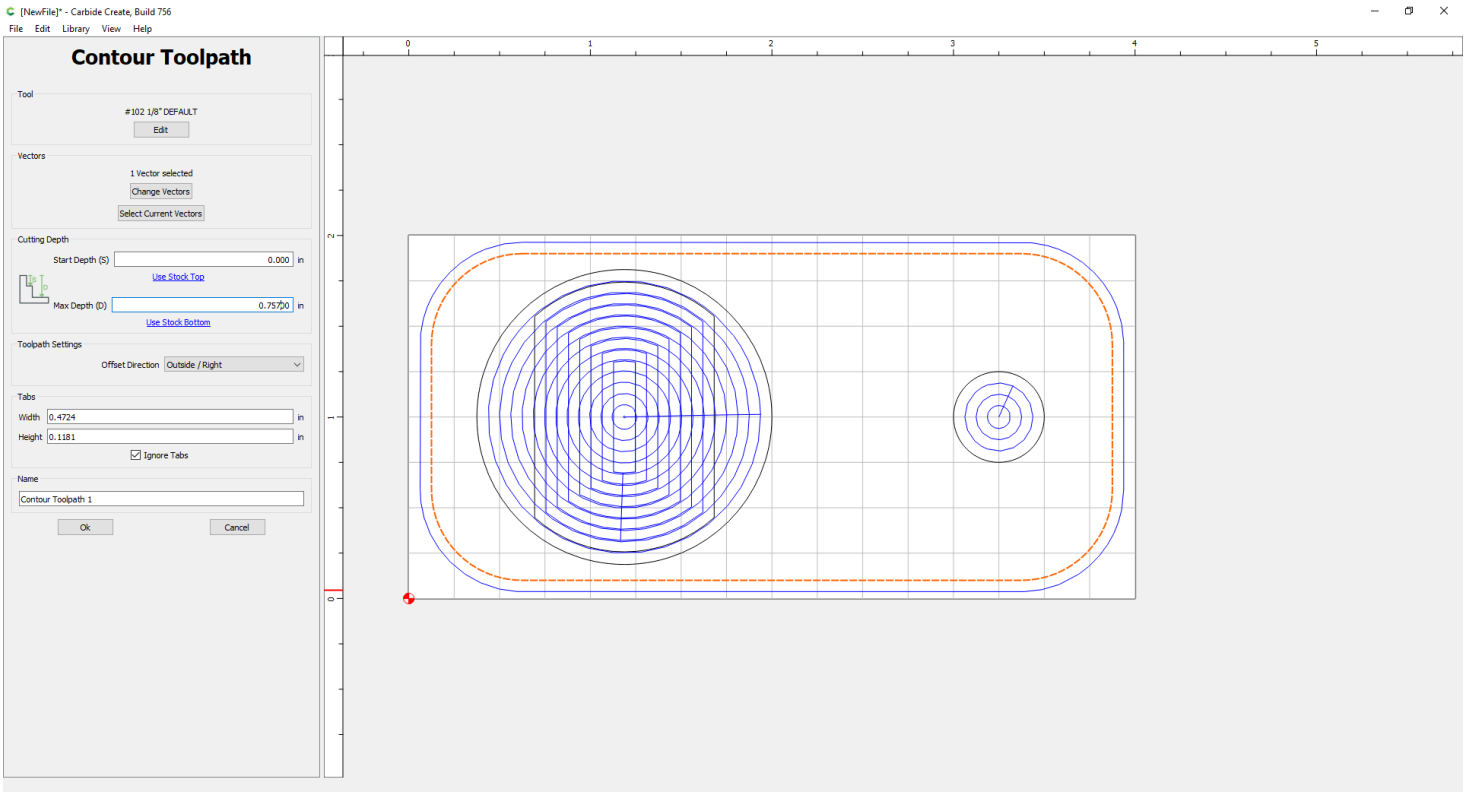


For this pocket, the starting depth remains 0.0in, but the max depth is now going to be 0.125in. Make sure to change your feeds and speeds to be the same as the previous step. You will use these **settings** for every toolpath in this project so make sure to adjust them on **every toolpath**.

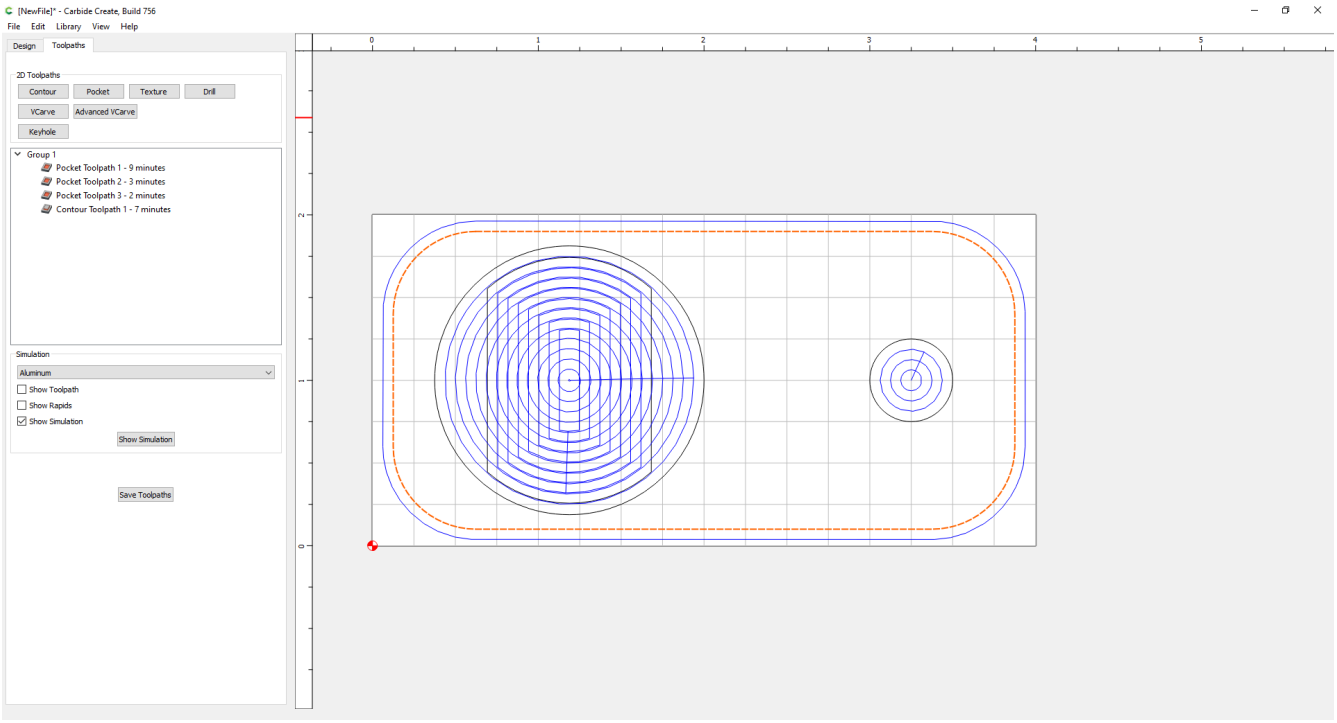
Note! If you don't use the correct feeds and speeds the machine may end up going too fast and breaking a tool or too slow and burning your project material.

Step 8: You may now add toolpaths for the custom designs that you decided. Make sure your always select the correct tool, depths and feeds and speeds. **Refer to steps six through seven as needed.**

Step 9: Cutting out your premiter. To do this select your overall outline and select “contour” cut. Then edit all of the settings in the same manner from the previous steps. The outside contour should be done last in order to keep the workpiece stable until the last step. When selecting the max depth you can also select “use stock bottom” just underneath it to automatically fill in the full depth of your part. Add 0.01in to this depth so that the bit cuts through the entire piece of material.

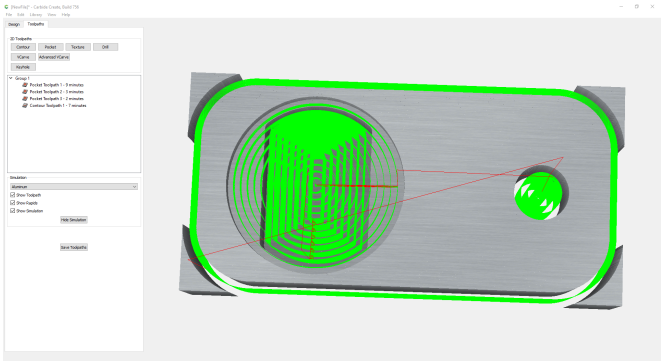


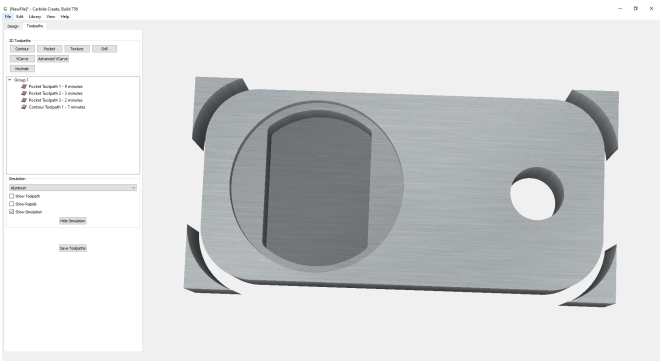
Step 10: Now you should be done with designing your part, and your screen should look something like this, notice the list of toolpaths on the left side.



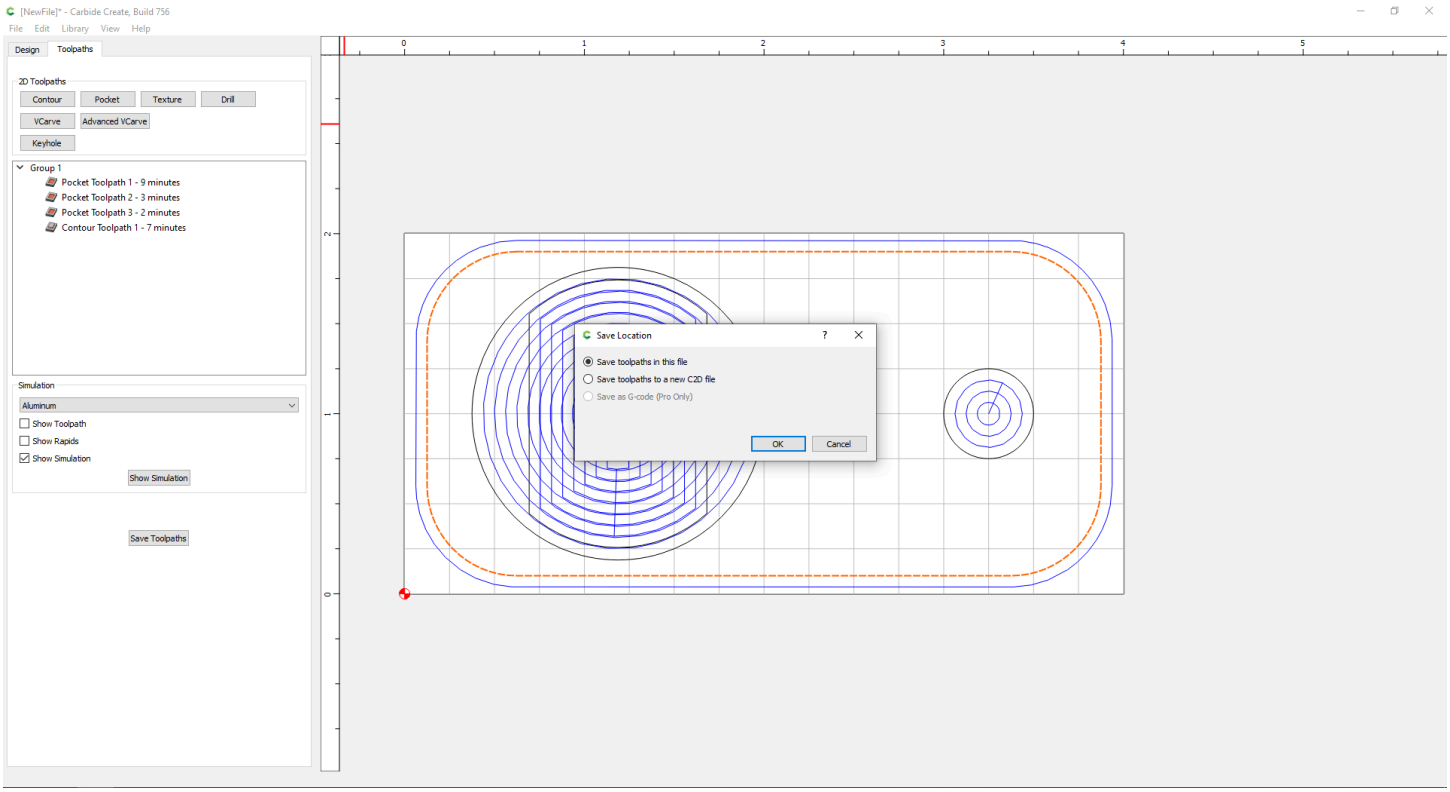
You can check to make sure your design is doing what you expect by selecting “show simulation” on the left this will show you a digital representation of your 3D part, you can select and deselect various aspects of the design in order to show or hide them, such as toolpaths and retractions. Note, text/v-carving will not display correctly in some designs.

Note: This simulation will let you know about any unexpected issues on your finished part so make sure to check it!





Step 11: If everything checks out you can now export your toolpaths by selecting “Save Toolpaths” and saving to the same file by making sure “Save toolpaths in this file” is checked in the popup.



Step 12: Now your file is complete! You can either save it for later use or transfer it to one of the computers located next to the Nomad CNC Router. Each machine is located on the same table as the computer that controls it. From here you can start "Machine Quick Start Guide" located on Page 2.

Introduction to Endmills and Bits:

CNC Router bits come in all shapes and sizes but there are three common tool types that you will encounter while using the Nomads, or any type of CNC router. Each one has its own specialty and making sure to use the proper tool for the proper job is essential to success.

1. *Square end mills:* sometimes called flat end mills, square end mills are characterized by a parallel bit ending in a flat edge. This is the most common starter tool for most jobs. They are good at making flat geometry with sharp

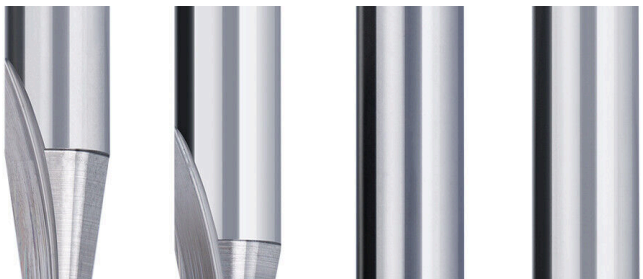
inside edges, or for removing the bulk of material before using another tool to create details

Square End Mill

2. *Ball Nose End Mill:* Unlike its square counterpart, the ballnose endmill has a rounded bottom edge, the radius of which is half the tool's width. The tools are used most often in adding in details to surfaces such as art carvings, or to do finishing passes on curved or sloped surfaces.

Ball Nose End Mill

3. *Engraving Bits:* Also known as PCB Engravers, or sometimes V-Bits, these are V shaped tools specifically for engraving. Engraving bits come in a variety of angles (this affects how narrow the tip becomes) in addition to several styles and can be used to add text or fine lines to a project. Most commonly these are used to make signs and circuit boards. Be careful as these can be very sharp!



Feeds and Speeds:

Feeds and speeds are unique to every bit and every material, and can be calculated through a complicated process; many great tutorials can be found on the subject online. However for the Nomad 883 here is a handy quick guide to get you started. This should cover all common materials you may encounter.

ALL UNITS IN INCHES

Material	**Depth of Cut**	**RPM**	**Feed Rate**	**Plunge Rate**
Plastic				
ABS	0.015”	9000	42	15
Acrylic	0.019”	9000	44	14
Delrin	0.02”	5900	65	18
HDPE	0.02”	6250	80	21
Linoleum	0.03”	3600	75	9
PVC	0.03”	6000	40	15
Styrene	0.03”	6000	40	15
Polycarbonate	0.013”	9000	55	18

Composites				
AL-PVC Panel	0.065”	9200	35	17
G10	0.015”	7300	60	28
Foamcore	0.1”	7500	75	35
Carbon Fiber	DO NOT CUT	N/A	N/A	N/A

Material	**Depth of Cut**	**RPM**	**Feed Rate**	**Plunge Rate**

Metal				
6061 aluminum	0.01”	9200	8	1
360 Brass	0.01”	9200	8	1

Wood				
Bamboo	0.04”	9000	55	23
Pine	0.03”	4500	72	32
Hardwoods	0.014”	9200	75	19
Plywood	0.034”	7800	50	22
MDF	0.017”	4000	75	18

Other/Exotic				
Cork	0.03”	9000	20	15
Graphite	0.02”	6000	20	15
HD Wax	0.04”	5000	80	40
Lead	0.013”	8000	15	3

Phenolic	0.01”	8000	40	15

For a more expanded list please see

https://docs.carbide3d.com/support/supportfiles/Nomad883_feeds_125.pdf

Consult staff before using any uncommon material or those listed under “*Other/Exotic” in the table above*.

Common Problems:

“*[Blank] axis has triggered an endstop.”*

This problem is most often caused by one of the carriages being too far to one side, this can happen while moving the machine when it is offline or after a crash. To fix, turn *off the machine, manually move all axes to the center (this should take some effort but not a ton of force) and turn the machine back on.* It should now initialize without an error.

“Machine has disconnected” or “No connection available”

This means the machine is turned off, turning it on should fix the issue, if it disconnects in the middle of a cutting operation, this likely means that the machine has reset due to crashing, or stalling. In this scenario you should check your file for any incorrect values (look for cuts that are set to deep or speeds that are too fast.) Then return your machine to its zero position and restart the operation.

You notice the material moving during the cutting process.

This usually happens when the material you are cutting has broken loose from the machine's bed. Work holding can fail for a number of reasons but once it

does it is likely you will need to restart the project, paying attention to use stronger hold down methods such as screws or clamps, this can be challenging on smaller pieces of material. If you catch the error early or the piece has received only minor damage, you can reattach it to the machine and complete the job.

Additional useful information may be located at the links below.

https://my.carbide3d.com/#Design_with_Carbide_Create

https://www.youtube.com/@carbide3d

https://community.carbide3d.com/c/cnc-machines/nomad/11

https://carbide3d.com/3d-print/

https://docs.carbide3d.com/support/supportfiles/Nomad883_feeds_125.pdf

https://www.youtube.com/watch?app=desktop&v=obwwoEAbBjc

https://www.speedtigertools.com/solution/ins.php?index_id=97