

IOWA STATE UNIVERSITY

Computation & Construction Lab

Stinger CNC 2D Toolpaths Workflow



Hardware



Tool Chuck Holder + Spanner Wrench

ISO 30 Cone, Nut, ER 32 Collet

Wireless Pendant

Hardware



ShopSabre Pro 510 CNC

- Cut tool
- 2D and 3D cutting, drilling, engraving etc.
- Turn On once the files are ready to be cut.
- Does not need to be turned ON/OFF between cuts unless manually handling CNC parts.



Air Filtration System

- Filters dust particles from CNC
- Needs to remain ON at all times while working in the CNC room.



Variable Frequency Drive (VRD)

Hardware



Vacuum

- Used to suction materials to the CNC bed



Cyclone Dust Collector

- Collects dust from the CNC during job.
- Connected to the CNC router through suction ducts.
- Must be turned on before cutting.



Shop-vac

- Cleans saw dust or other material residue
- It is important to clean residue around hot machinery regularly to avoid fires.

Software



Design Setup

- RhinoCeros, Fusion 360, AutoCAD, etc.
- A range of software can be used to setup the initial design files to be milled.

3D Mill Setup

- VCarve
- Used to setup the toolpaths for the cut.
- For 2D cutting, drilling, etc.

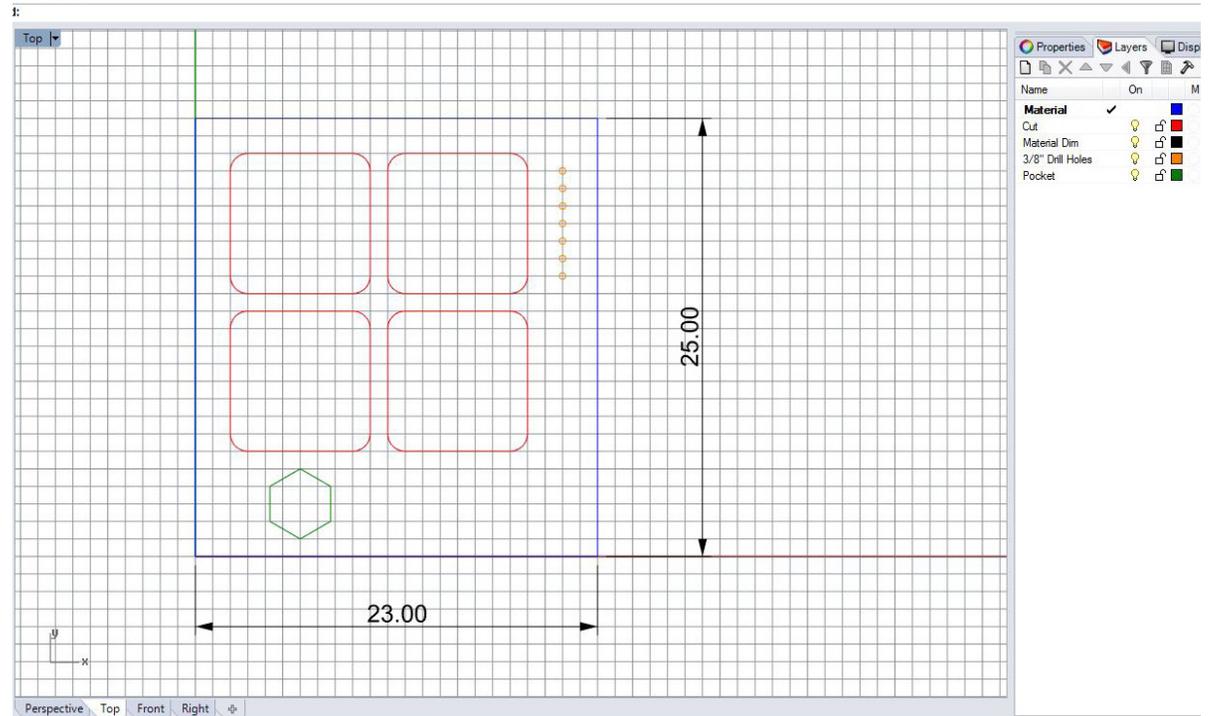
CNC Setup

- ShopSabre CNC
- Used to operate the CNC.

Design Setup

Rhinoceros

The 2D linework needs to be exported as .dxf. Work in inches. Be sure to put your different toolpath lines on separate layers, ie. one layer for cut toolpaths. The geometry to be cut needs to fit within a 5 ft. x 10 ft. boundary. If the geometry does not fit within the 5 ft. x 10 ft. boundary, divide the geometry and export as separate files. Make sure that the gap between geometries is the same size or larger than the diameter of the end mill bit you plan to use.



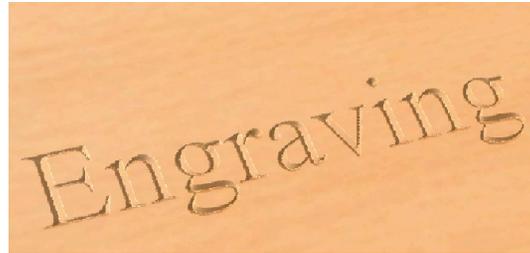
2D Toolpath Options

VCarve Pro

Profile, Pocket and Drilling Toolpaths will be covered in the following pages. There are other toolpath options. Below is a basic list for more information please look on VCarve's online manual or talk to the CCL Associate.

1. Engraving Toolpaths

This form is used specifically for calculating engraving and marking toolpaths. A nose cone is often used when engraving or marking material that is not flat. The nose cone is spring loaded forcing it to slide on the surface of the material.



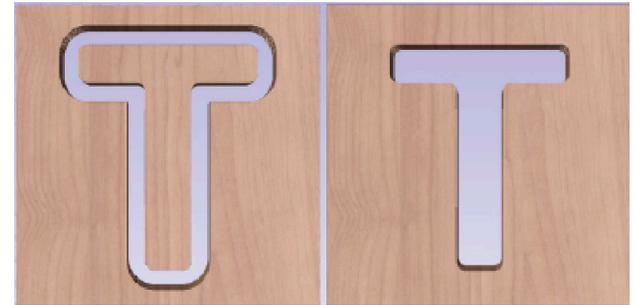
2. Inlay Toolpaths

The Inlay toolpath functionality can be used to calculate either profile or pocketing toolpaths with automatic compensation for the tool radius; this allows the cut-out parts to fit into the corresponding cavities.



3. Fluting Toolpaths

Fluting Toolpaths machine along vectors while varying the depth of the tool, creating extremely efficient machined decorative patterns.



4. Texture Toolpaths

The 3D Texture Machining functionality uses a specialized toolpath algorithm and the shape of the tool to generate a textured finish on the part. It should be noted this is a different technique than a 3D texture using the 3D model. Textures can be calculated inside any selected vector boundaries or if nothing is selected the complete job size. Large Ball nosed cutter and V-Bits typically give the best results when using this strategy.



5. Prism Carving Toolpaths

Prism carving gives a raised 'bevel' effect to shapes and letters similar to a 'hand carved' look. Prism Carving uses an angled tool to create a raised prism shape on the top of the selected vectors. The tool will profile at a given depth creating a nice sharp finished shape such as the lettering shown in the image below.



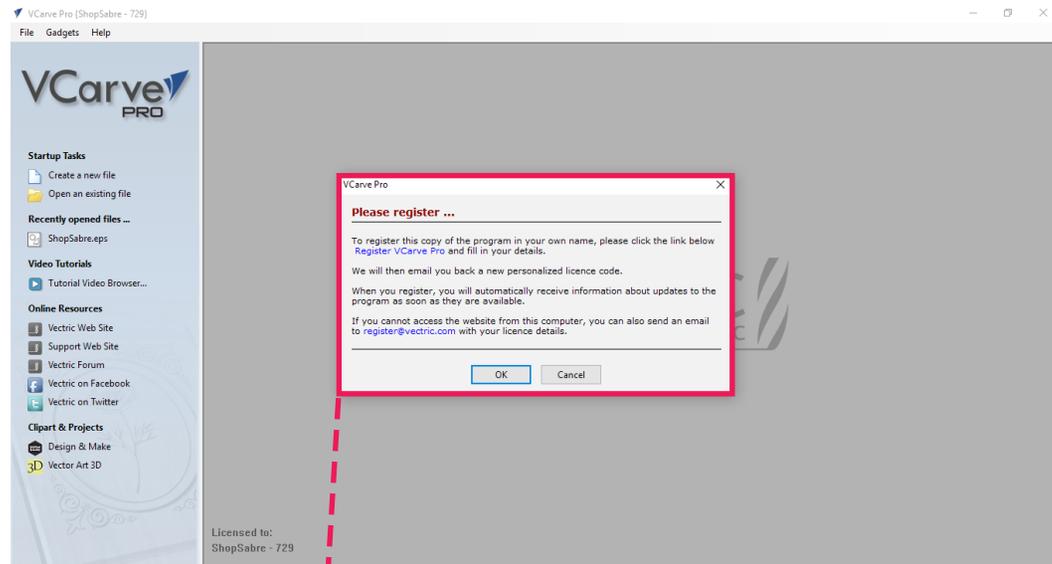
6. Moulding Toolpaths

This icon opens up the Moulding Toolpath Form. This form is used to create a toolpath from a drive rail and a profile. The result of machining to toolpath is the extrusion of the selected drive rail. Although strictly speaking the result of this is a 3D shape because it does not use a 3D model it is classified as a 2D Toolpath.

2D Profile Cut Setup - VCarve Pro

Open VCarve Pro

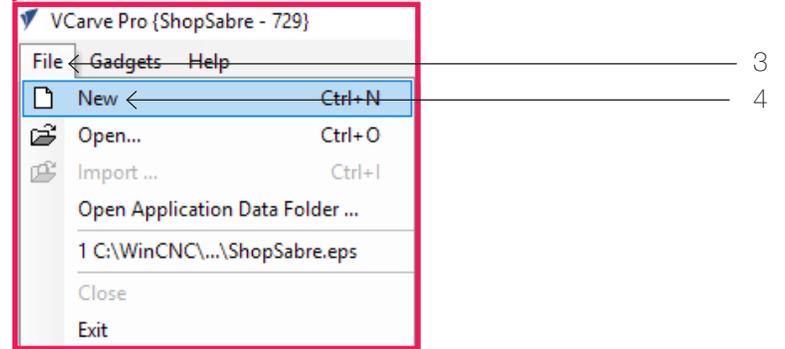
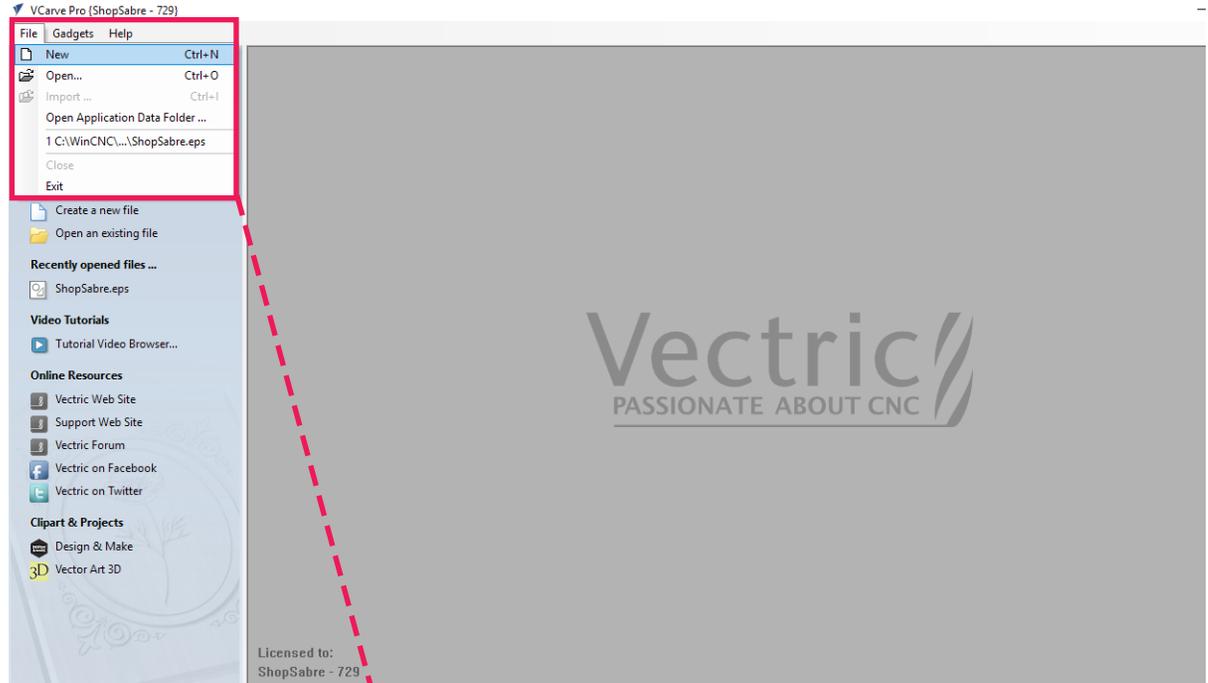
1. Click the VCarve Pro 8.5 icon on the desktop or taskbar
2. When prompted to register press Cancel.



2D Profile Cut Setup - VCarve Pro

Opening VCarve Pro

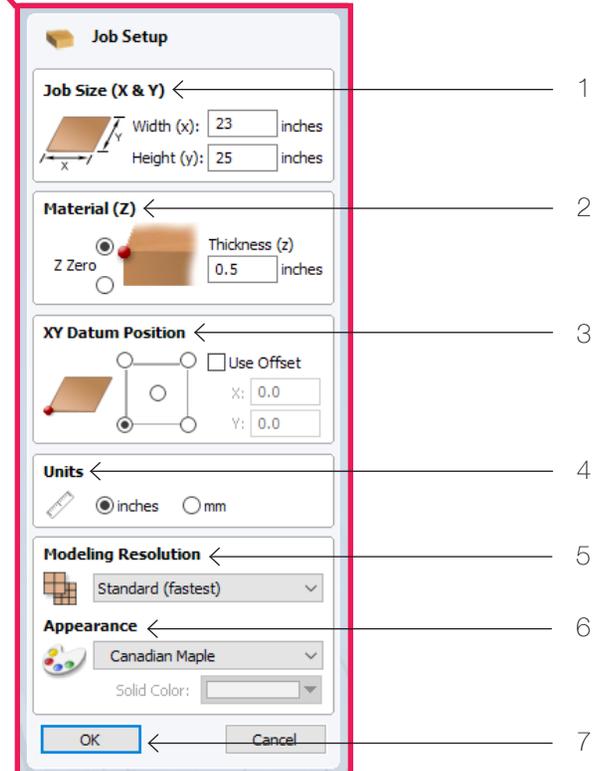
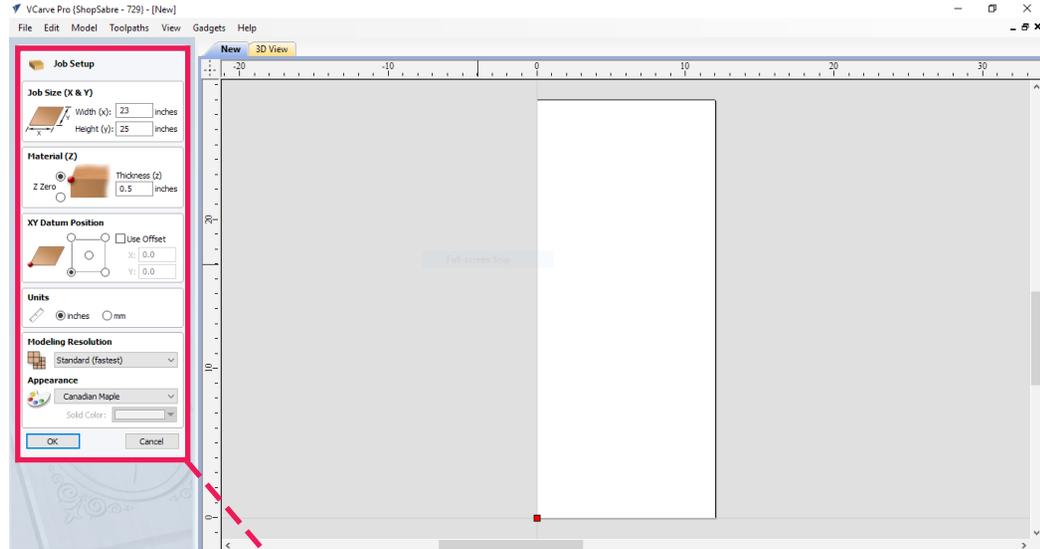
- 3. Click 'File'
- 4. Click 'New'



2D Profile Cut Setup - VCarve Pro

Job Setup/Material Dimensions

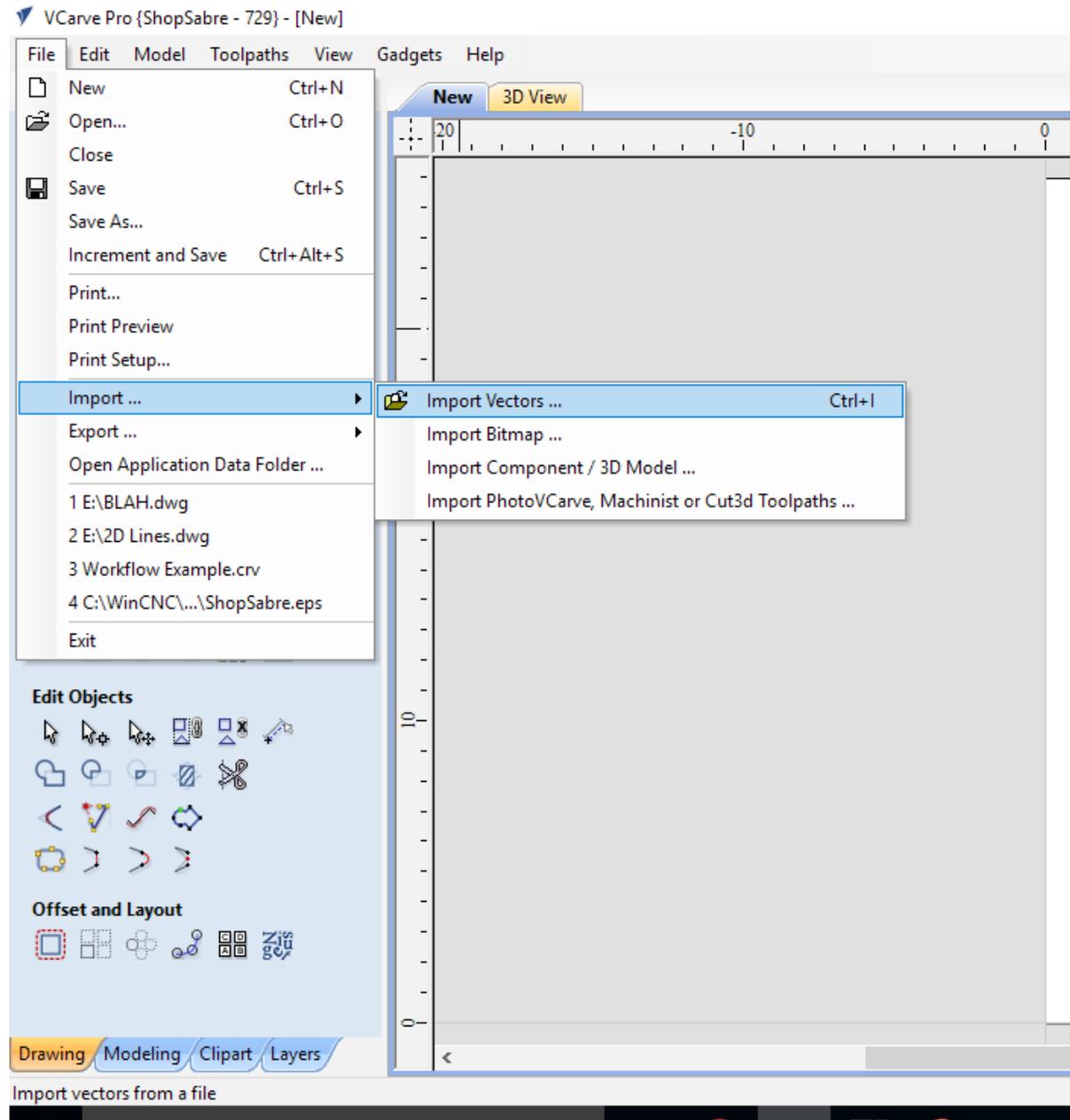
1. Job Size (X & Y): Insert Width (x) and Height (y) of material
2. Material (Z): Insert the material thickness. Be sure that Z Zero is at the top of the material.
3. XY Datum Position: Make sure that the Datum is at the bottom, right corner.
4. Units: Select inches
5. Modeling Resolution: Depends on the quality you desire.
6. Appearance: You can change the appearance of the material with the drop down.
7. Double check the settings and click 'OK'.



2D Profile Cut Setup - VCarve Pro

File Import

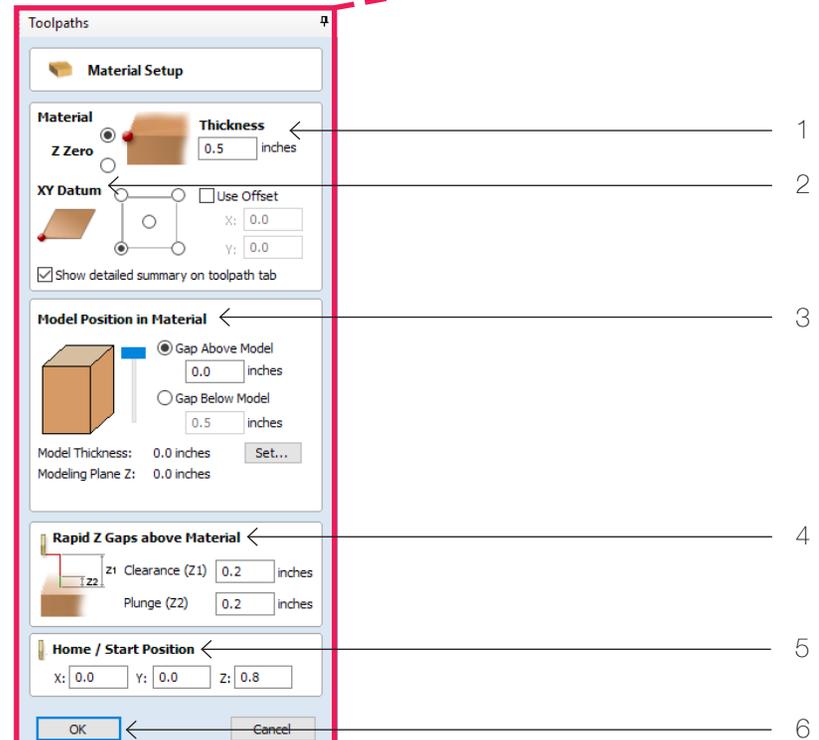
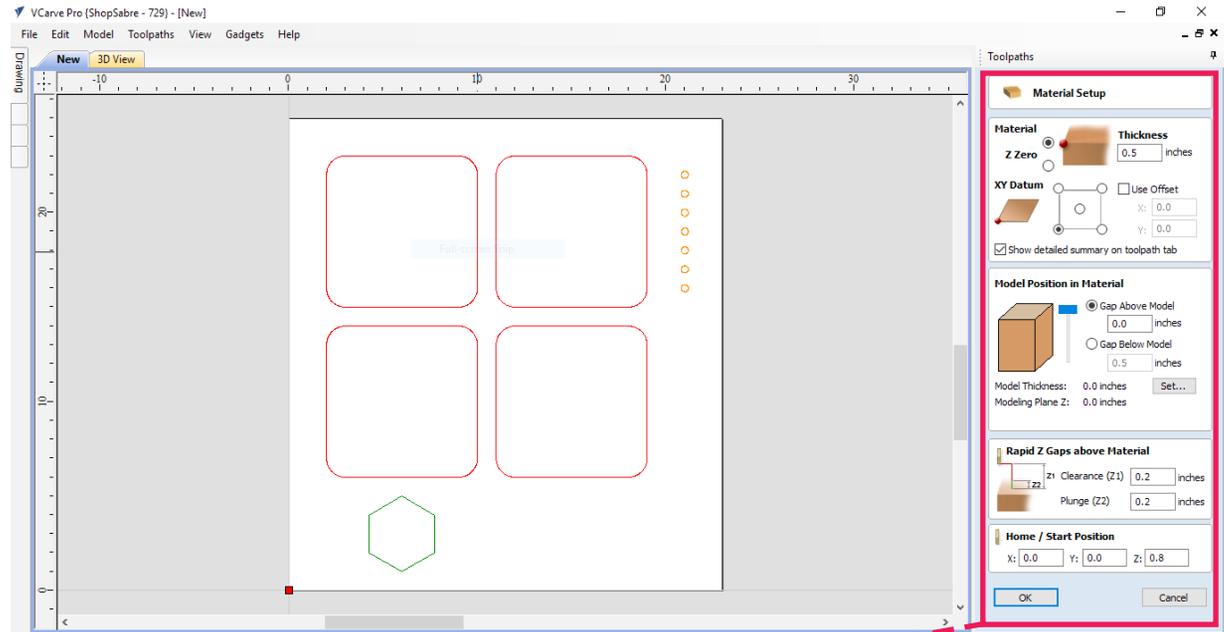
1. Click File > Import... > Import Vectors ... > (FileName).dxf.



2D Mill Setup - VCarve Pro

Material Setup

1. Material: Make sure that the Z Zero is at the top of the material. Type in the material thickness in inches.
2. XY Datum: Un-check "Use Offset"
3. Model Position in Material: Select "Gap Above Model"
4. Rapid Z Gaps above Material: Clearance (Z1) 0.2 inches Plunge (Z2) 0.2 inches
5. Home/Start Position: X: 0.0, Y:0.0, Z: 0.8
6. Click "OK"



2D Mill Setup - VCarve Pro

Drilling Toolpath- Setup

1. Click the Drilling Toolpaths Icon
2. Be sure to select the vectors you want to create pocket toolpaths with. When vectors are selected they are dashed.
3. Cutting Depths: Start Depth (D) 0.0 inches, Cut Depth (C) the material height or less.
4. Tool: See following page for more details. Click 'Select'.
5. Type a name into the text box. Be sure to include the toolpath type in the name.
6. Click 'Calculate'

Toolpath Operations

Drilling Toolpath

Cutting Depths

Start Depth (D) 0.0 inches
Cut Depth (C) .5 inches

Tool: Drill (0.250")
Select ... Edit ...

Use Peck Drilling

Retract above the cutting start depth
 Retract above the height of the previous pass

Retract Gap (R) 0.0 inches
Peck Depth (P) 0.25 inches
Note: Peck depth is controlled by the 'Pass Depth' for the tool

Dwell at the bottom of each drill pass
Dwell Time 0.0 seconds

Use Vector Selection Order

Safe Z 0.2 inches
Home Position X:0.00 Y:0.00 Z:0.80

Project toolpath onto 3D model

Vector Selection: Manual Selector ...

Name: Drill Toolpaths - 2D Milling Workflow

Calculate Close

Drilling Toolpath

Cutting Depths

Start Depth (D) 0.0 inches
Cut Depth (C) .5 inches

Tool: Drill (0.250")
Select ... Edit ...

Use Peck Drilling

Retract above the cutting start depth
 Retract above the height of the previous pass

Retract Gap (R) 0.0 inches
Peck Depth (P) 0.25 inches
Note: Peck depth is controlled by the 'Pass Depth' for the tool

Dwell at the bottom of each drill pass
Dwell Time 0.0 seconds

Use Vector Selection Order

Safe Z 0.2 inches
Home Position X:0.00 Y:0.00 Z:0.80

Project toolpath onto 3D model

Vector Selection: Manual Selector ...

Name: Drill Toolpaths - 2D Milling Workflow

Calculate Close

2D Mill Setup - VCarve Pro

Drilling Toolpaths- Tool Database

1. Select tool from the Tool List. Choose an End Mill for Drilling Toolpaths.
2. Make sure that the diameter matches the name and the bit you have chosen to use.
3. Change the Tool Number to match where it has been placed in the tool changer rack on the CNC. Refer to the page named Automatic Tool Changer.
4. Click 'Apply'
5. Click 'OK'

Tool Database

Tool List ← 1

- Imperial Tools
 - End Mills
 - End Mill (0.25 inch)
 - End Mill (0.375 inch) ← 2
 - Ball Nose
 - Ball Nose (0.25 inch)
 - Ball Nose (0.125 inch)
 - V-Bits
 - Form Tools
 - Engraving
 - Specialist
 - Drills
- Metric Tools
 - End Mills
 - Ball Nose
 - V-Bits
 - Form Tools
 - Engraving
 - Specialist
 - Drills

Tool Info

Name: End Mill (0.375 inch)

Tool Type: End Mill

Notes:

Geometry

Diameter (D): 0.375 inches ← 2



Cutting Parameters

Stepover: 0.15 inches 40.0 %

Feeds and Speeds

Spindle Speed: 12000 r.p.m

Feed Rate: 100.0 inches/min

Plunge Rate: 30.0 inches/min

Tool Number 2 ← 3

Apply ← 4

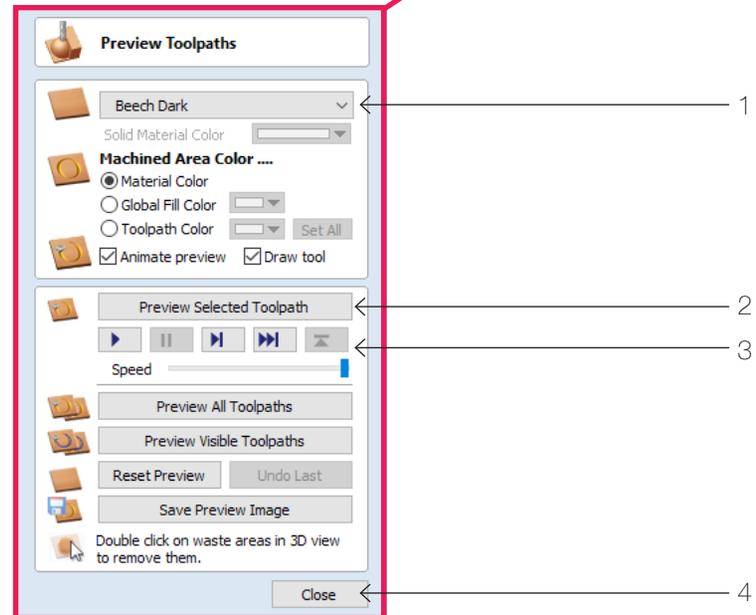
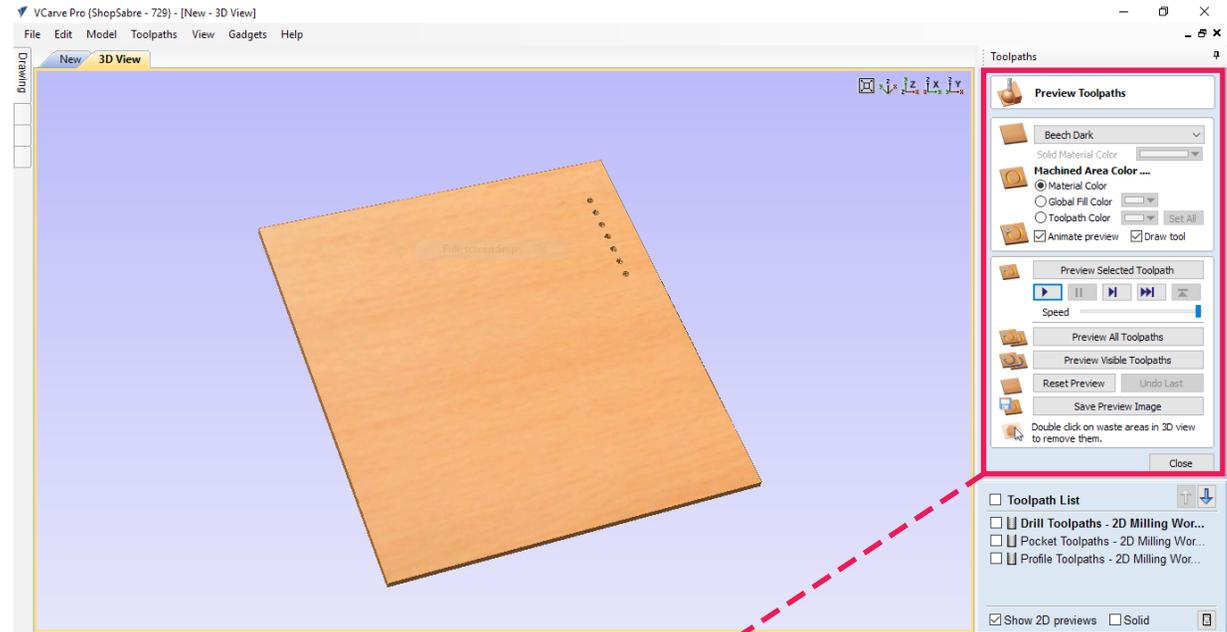
OK ← 5 Cancel

2D Mill Setup - VCarve Pro

Drilling Toolpaths- Preview

Drilling allows the centers of selected closed vectors to be drilled to a specified depth.

1. The first section allows one to change the type of wood, colors, etc.
2. Allows one to preview selected tool paths.
3. One can play, skip, change the speed, etc.
4. Click 'Close' when finished.

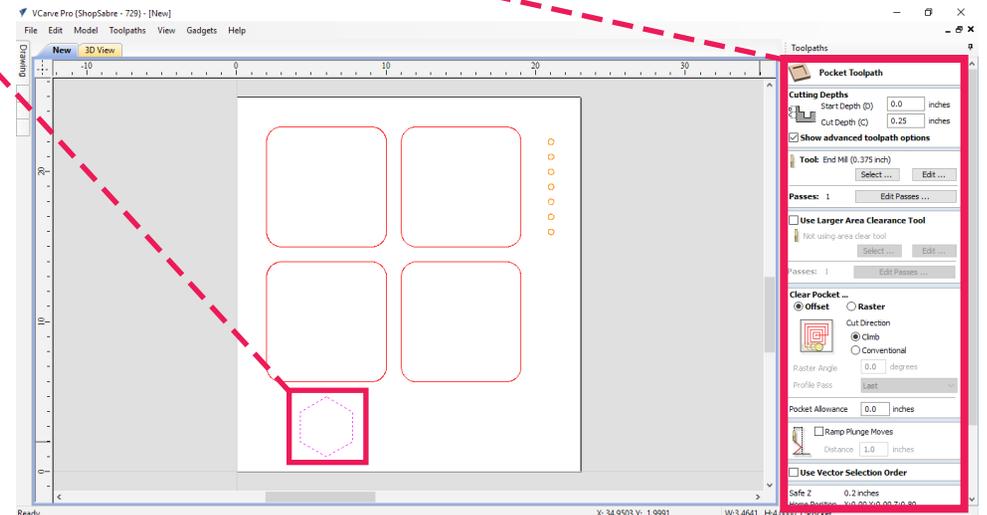
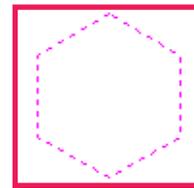
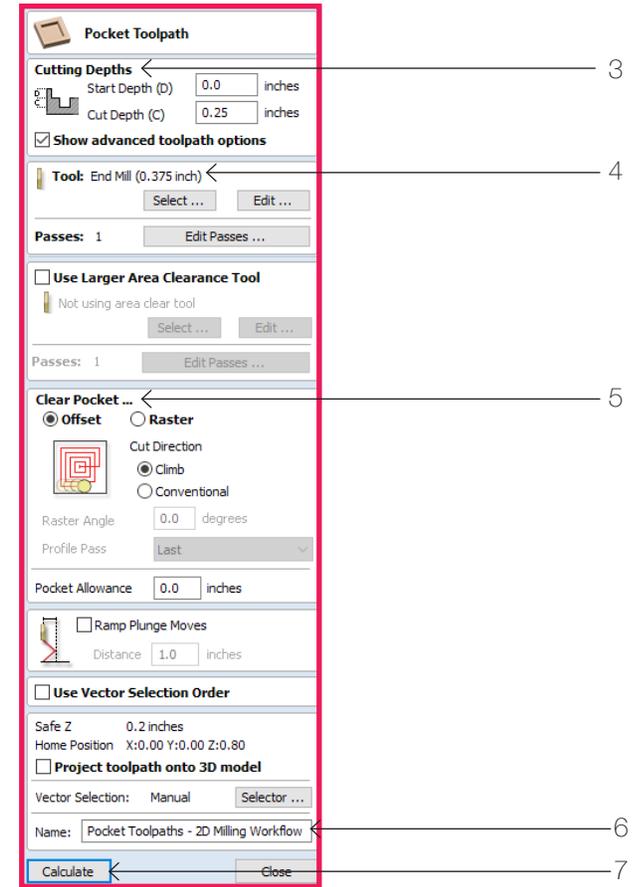
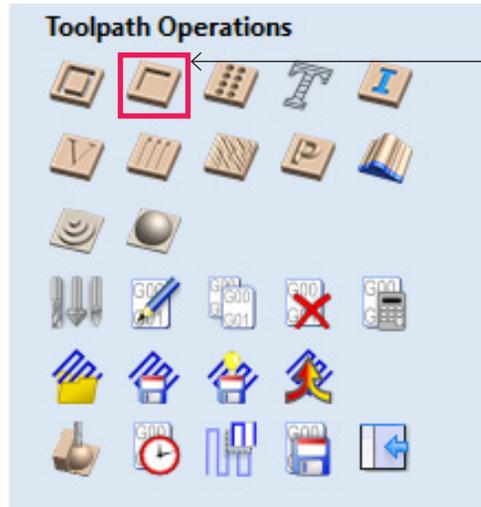


2D Mill Setup - VCarve Pro

Pocket Toolpath- Setup

Carves 2D pockets

1. Click the Pocket Toolpaths Icon
2. Be sure to select the vectors you want to create pocket toolpaths with. When vectors are selected they are dashed.
3. Cutting Depths: Start Depth (D) 0.0 inches, Cut Depth (C) 0.5 inches (or the material height or less)
4. Tool: See following page for more details. Click 'Select'.
5. Choose Offset or Raster under clear pocket.
6. Type a name into the text box. Be sure to include the toolpath type in the name.
7. Click 'Calculate'



2D Mill Setup - VCarve Pro

Pocket Toolpaths- Tool Database

1. Select tool from the Tool List. Choose an End Mill for Drilling Toolpaths.
2. Make sure that the diameter matches the name and the bit you have chosen to use.
3. Change the Tool Number to match where it has been placed in the tool changer rack on the CNC. Refer to the page named Automatic Tool Changer.
4. Click 'Apply'
5. Click 'OK'

Tool Database

Tool List ← 1

- Imperial Tools
 - End Mills
 - End Mill (0.25 inch)
 - End Mill (0.375 inch) ← 2
 - Ball Nose
 - Ball Nose (0.25 inch)
 - Ball Nose (0.125 inch)
 - V-Bits
 - Form Tools
 - Engraving
 - Specialist
 - Drills
- Metric Tools
 - End Mills
 - Ball Nose
 - V-Bits
 - Form Tools
 - Engraving
 - Specialist
 - Drills

Tool Info

Name: End Mill (0.375 inch)

Tool Type: End Mill

Notes:

Geometry

Diameter (D): 0.375 inches ← 2



Cutting Parameters

Stepover: 0.15 inches 40.0 %

Feeds and Speeds

Spindle Speed: 12000 r.p.m

Feed Rate: 100.0 inches/min

Plunge Rate: 30.0 inches/min

Tool Number 2 ← 3

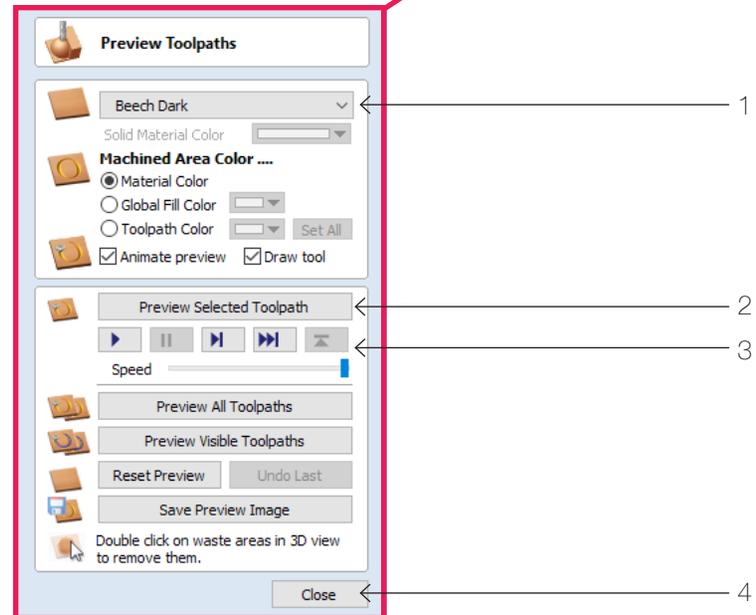
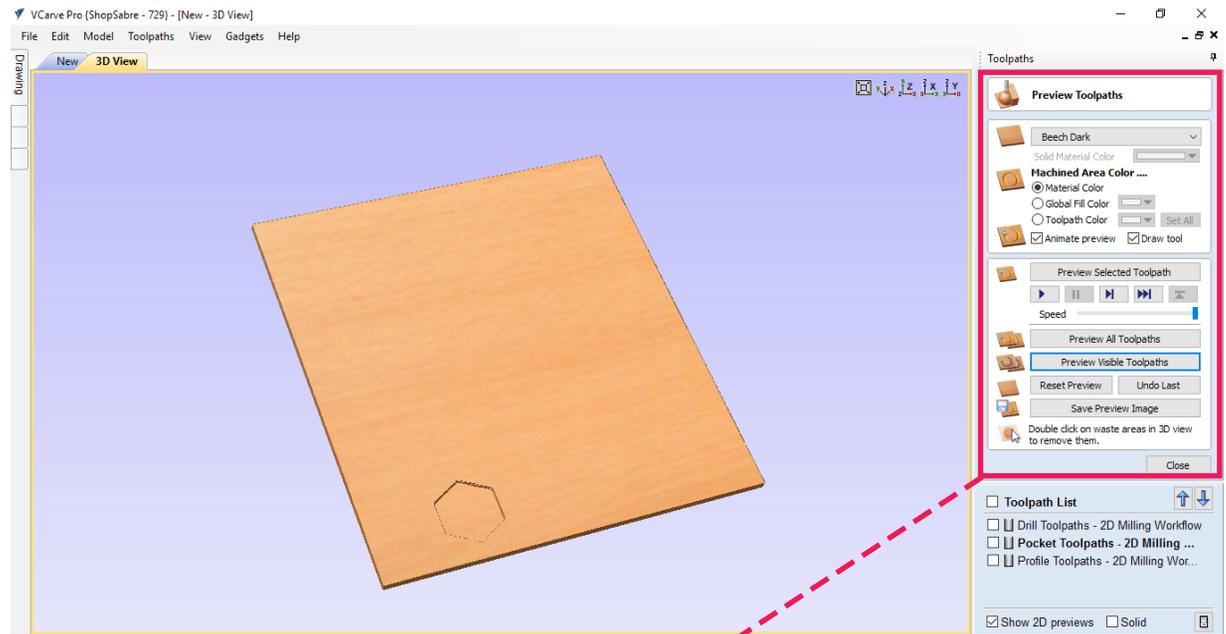
Apply ← 4

OK ← 5 Cancel

2D Mill Setup - VCarve Pro

Pocket Toolpaths- Preview

1. The first section allows one to change the type of wood, colors, etc.
2. Allows one to preview selected tool paths.
3. One can play, skip, change the speed, etc.
4. Click 'Close' when finished.

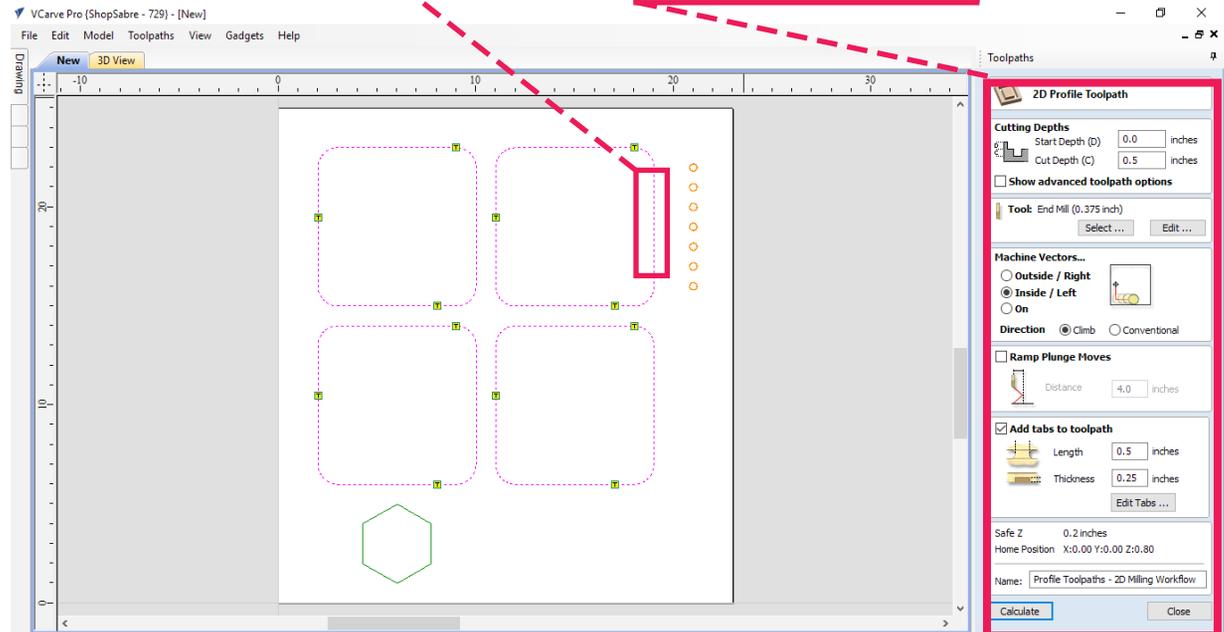
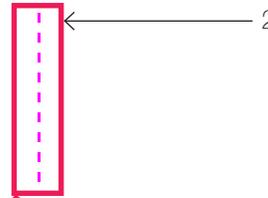
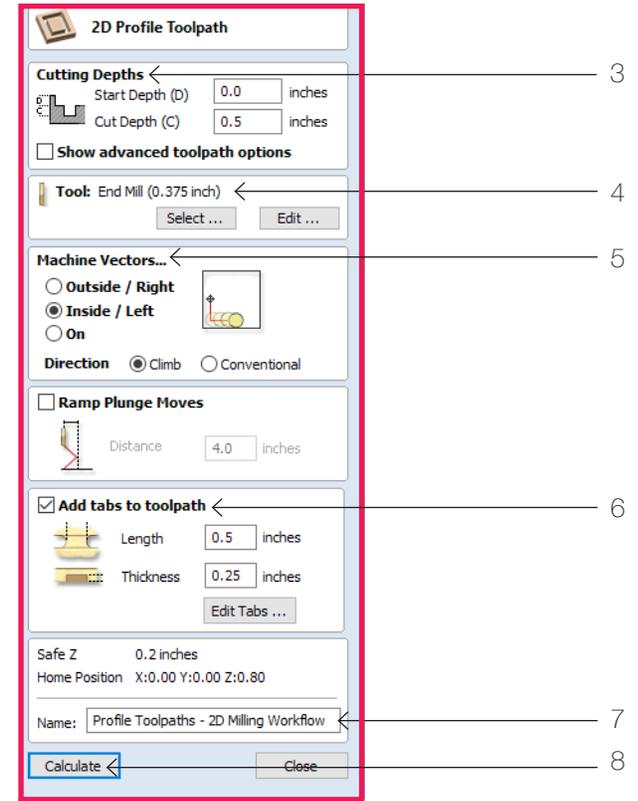
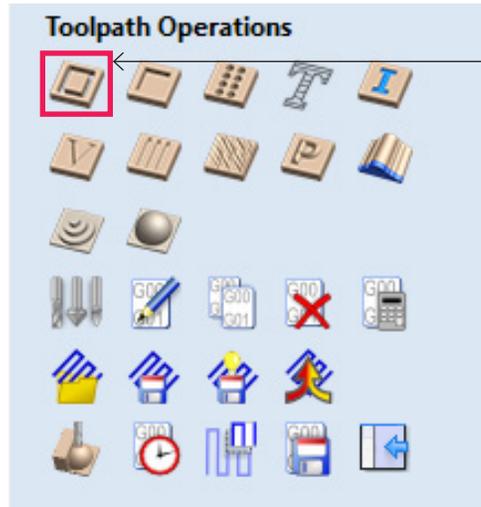


2D Mill Setup - VCarve Pro

2D Profiles/Cut Out Tool Paths- Setup

Profile Machining is use to cut around or along a vector.

1. Click the 2D Profile Toolpaths Icon
2. Be sure to select the vectors you want to create pocket toolpaths with. When vectors are selected they are dashed.
3. Cutting Depths: Start Depth (D) 0.0 inches, Cut Depth (C) 0.5 inches (or the material height or less)
4. Tool: See following page for more details. Click 'Select'.
5. Machine Vectors...: Select 'Outside / Right', Direction 'Climb'
6. Check 'Add tabs to toolpaths'. See following page for more details.
7. Type a name into the text box. Be sure to include the toolpath type in the name.
8. Click 'Calculate'



2D Mill Setup - VCarve Pro

2D Profiles/Cut Out Tool Paths- Tool Database

1. Select tool from the Tool List. Choose an End Mill for Drilling Toolpaths.
2. Make sure that the diameter matches the name and the bit you have chosen to use.
3. Change the Tool Number to match where it has been placed in the tool changer rack on the CNC. Refer to the page named Automatic Tool Changer.
4. Click 'Apply'
5. Click 'OK'

Tool Database

Tool List ← 1

- Imperial Tools
 - End Mills
 - End Mill (0.25 inch)
 - End Mill (0.375 inch) ← 2
 - Ball Nose
 - Ball Nose (0.25 inch)
 - Ball Nose (0.125 inch)
 - V-Bits
 - Form Tools
 - Engraving
 - Specialist
 - Drills
- Metric Tools
 - End Mills
 - Ball Nose
 - V-Bits
 - Form Tools
 - Engraving
 - Specialist
 - Drills

Tool Info

Name: End Mill (0.375 inch)

Tool Type: End Mill

Notes:

Geometry

Diameter (D): 0.375 inches ← 2



Cutting Parameters

Stepover: 0.15 inches 40.0 %

Feeds and Speeds

Spindle Speed: 12000 r.p.m

Feed Rate: 100.0 inches/min

Plunge Rate: 30.0 inches/min

Tool Number 2 ← 3

Apply ← 4

OK ← 5

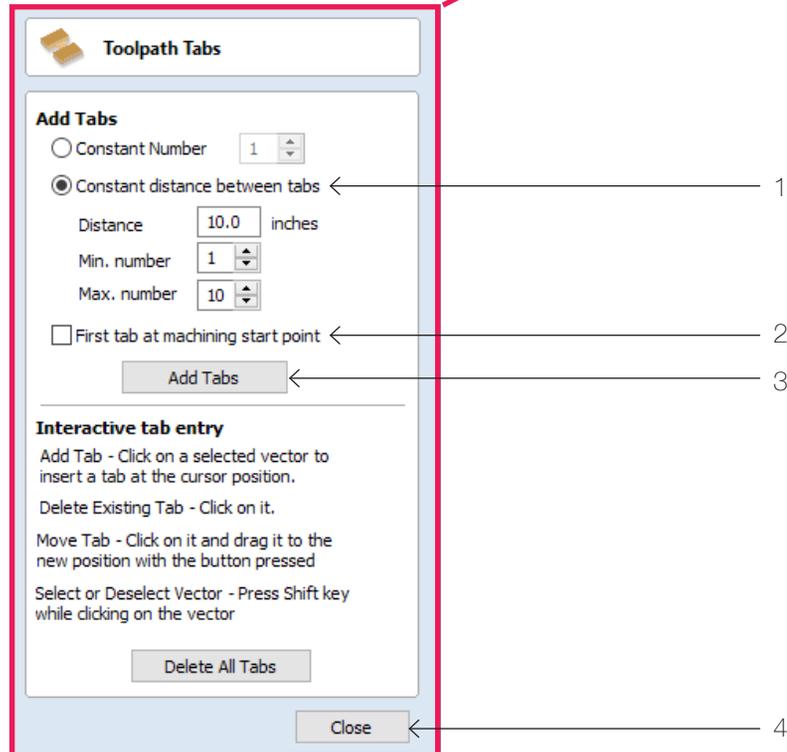
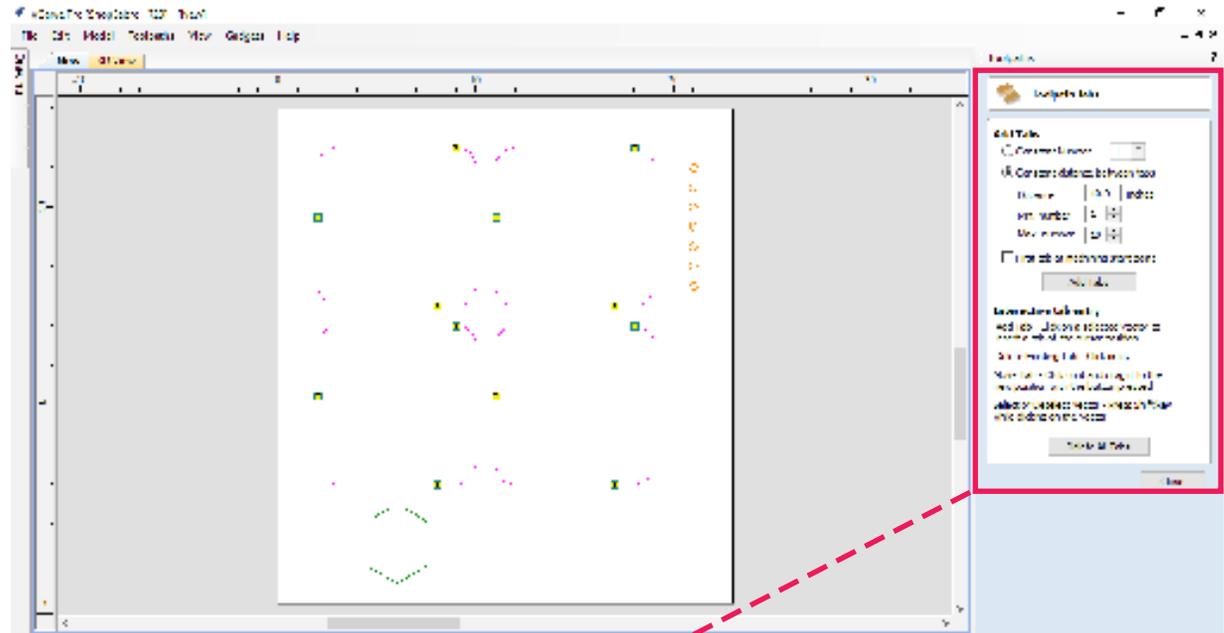
Cancel

2D Mill Setup - VCarve Pro

2D Profiles/Cut Out Tool Paths- Toolpath Tabs

Tabs can be added to 2D Profiles/Cut Out pieces to avoid displacement.

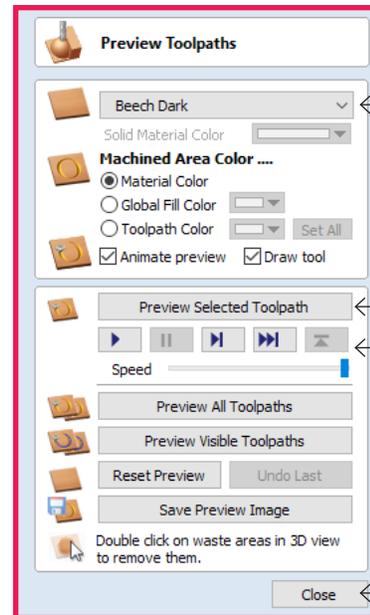
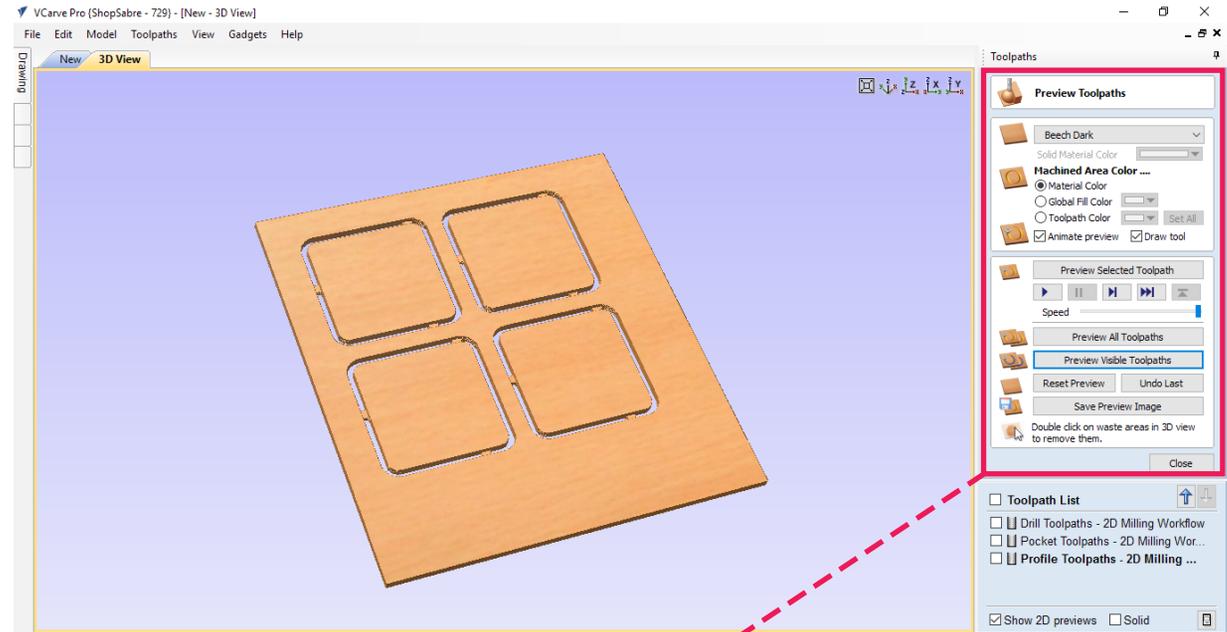
1. Constant distance between tabs: really depends on the size of the model. Reference the yellow squares in the drawing to get an idea of where the tabs will be.
2. Easier with sharp corners not to check the box first tab a machining start point.
3. Click 'Add Tabs'
4. When finished click 'Close'



2D Mill Setup - VCarve Pro

2D Profiles/Cut Out Tool Paths- Preview

1. The first section allows one to change the type of wood, colors, etc.
2. Allows one to preview selected tool paths.
3. One can play, skip, change the speed, etc.
4. Click 'Close' when finished.

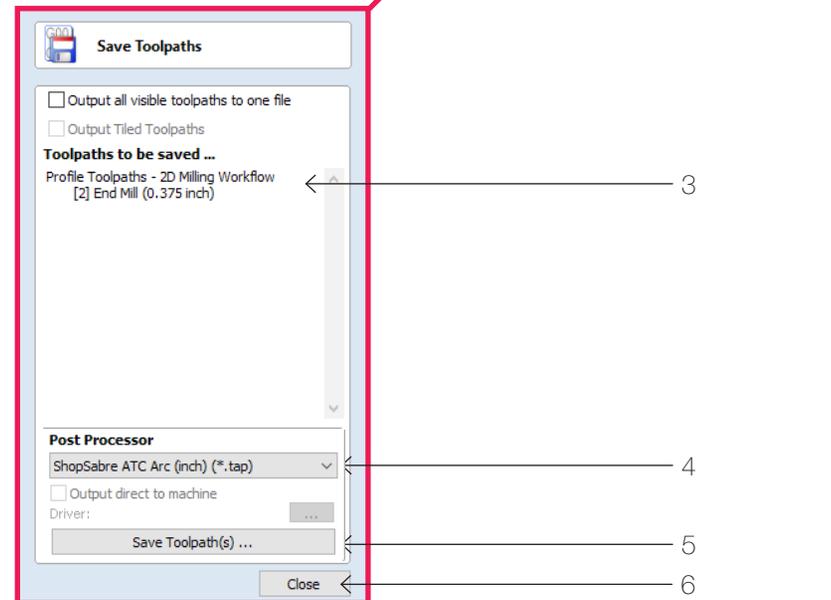
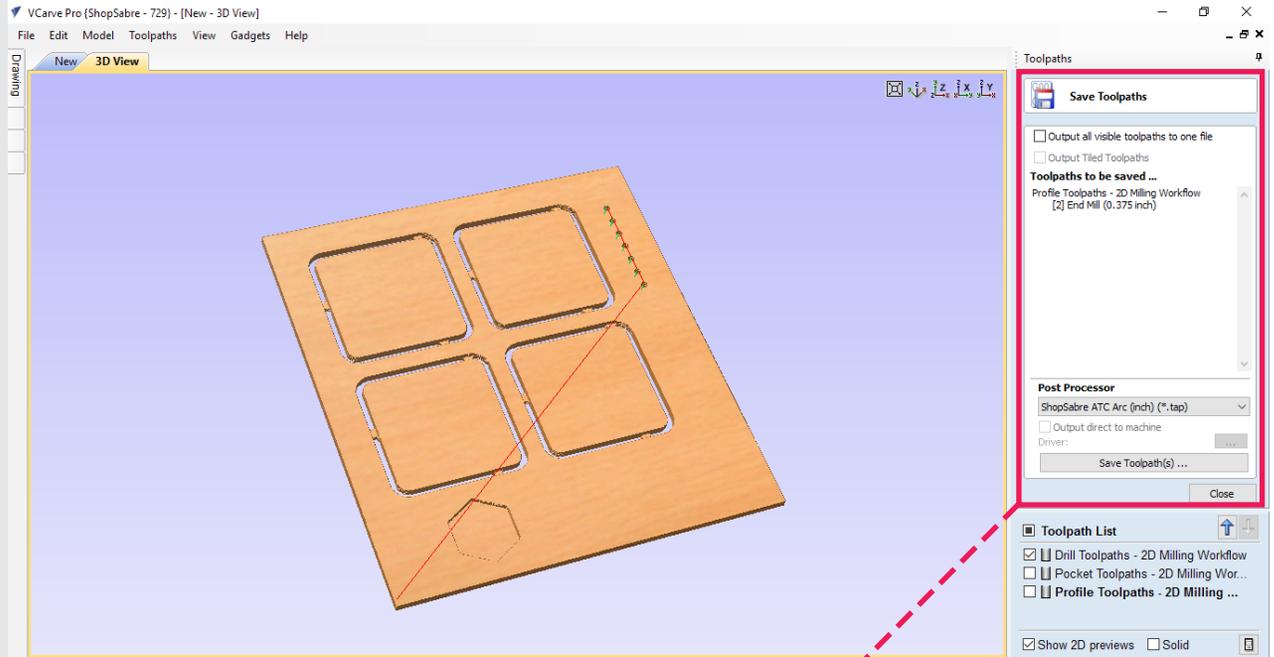


2D Mill Setup - VCarve Pro

Saving Toolpaths

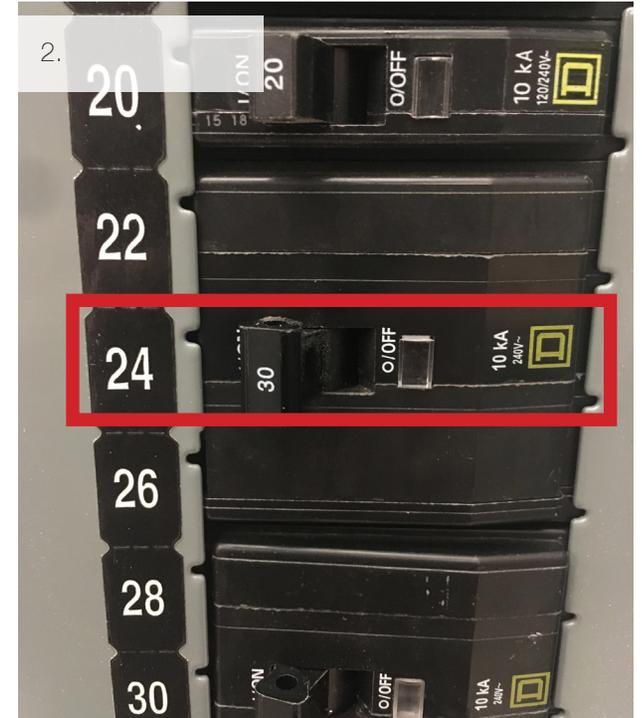
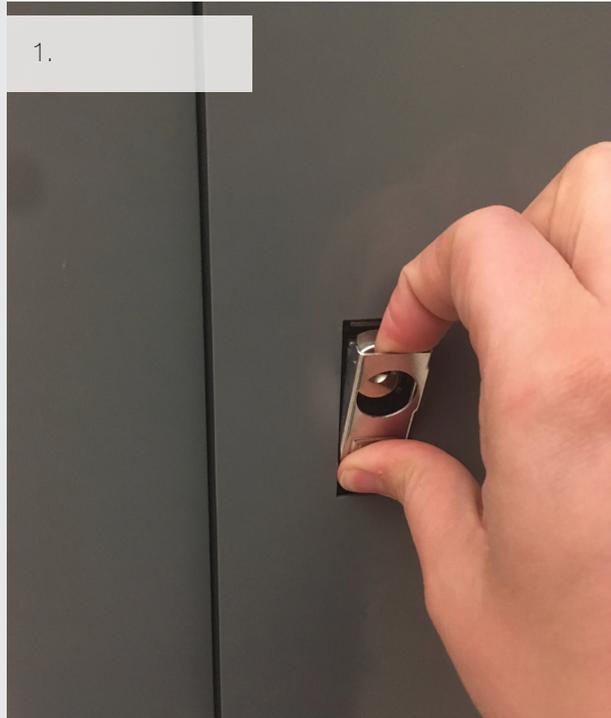
3. Double check that you have selected the correct toolpath.
4. Under Post Processor select ShopSabre (inch) (*.tap) from the drop down menu.
5. Save Toolpath(s): Name file as Filename - Toolpath Type
6. Close
7. Repeat this step for all toolpaths.

The files are now ready to be cut. IMPORTANT NOTE: Cut 2D Profile/Cut Out Toolpaths last!



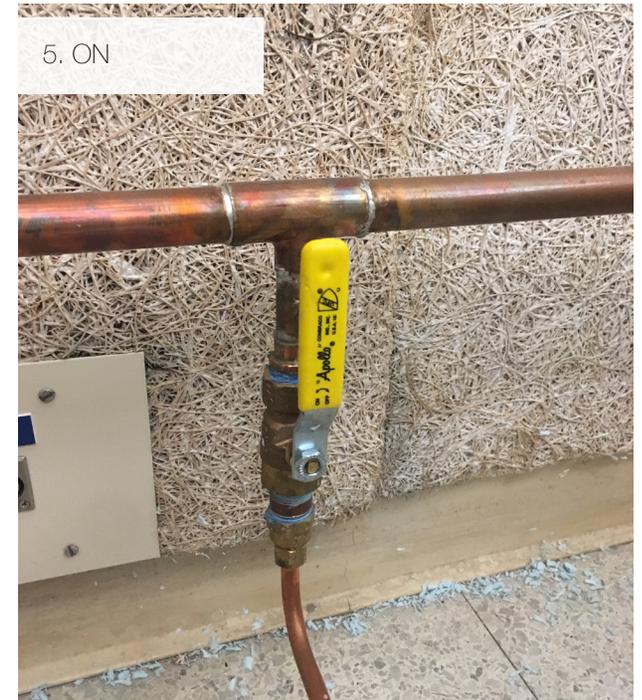
Turning on the Machine

1. Open the circuit breaker box by pushing the bottom with your thumb while pulling the top with your index finger.
2. Make sure that the switch labeled 30 is switched to the left towards 24.
3. Make sure that the Variable Frequency Drive (VFD) is on. If it is on it will have a number displayed on the front.
4. Turn the machine's power switch on.



Turning on the Machine

5. Turn the air on by pulling the handle up.
6. Turn on the dust collector by twisting the knob 90 degrees clockwise. Wait to turn this on until you start your cut. It is quite loud.



Turning on the Machine

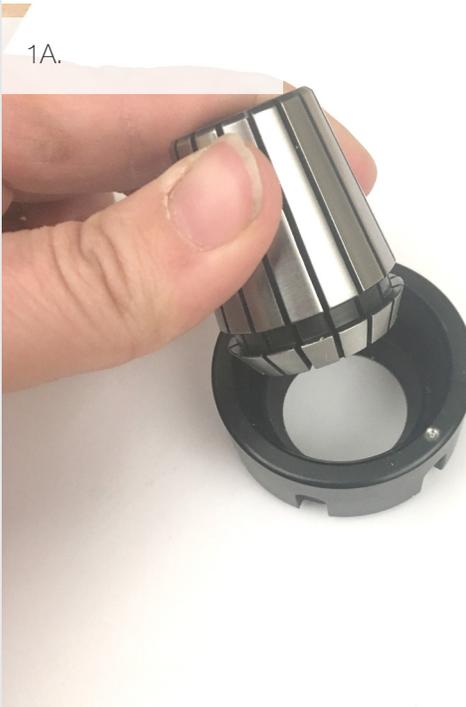
7. Turn on the air filtration system using the remote velcroed to the corner of the desk. Wait to turn this on until you start your cut. It too is quite loud. See image 7A. Press 'ON/OFF' and 'SPEED' twice. You can see the setting on the side of the air filtration system closest to the ShopSabre CNC. A red light will indicate the chosen settings. See image 6B. Return to velcro immediately after using. If the remote is not working, tell the CCL Associate so they can provide you with new batteries.
8. To open Shop Sabre Controller click the icon on the desktop or taskbar labeled ShopSabre CNC.



Automatic Tool Changer

Changing the Bit

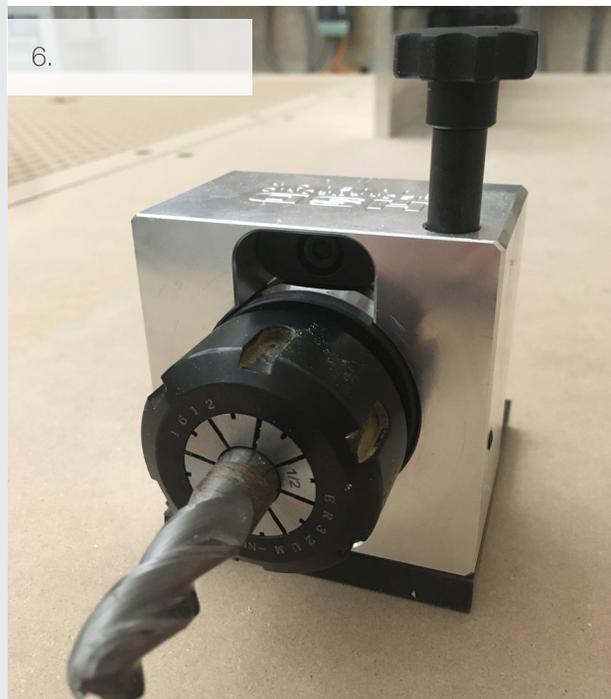
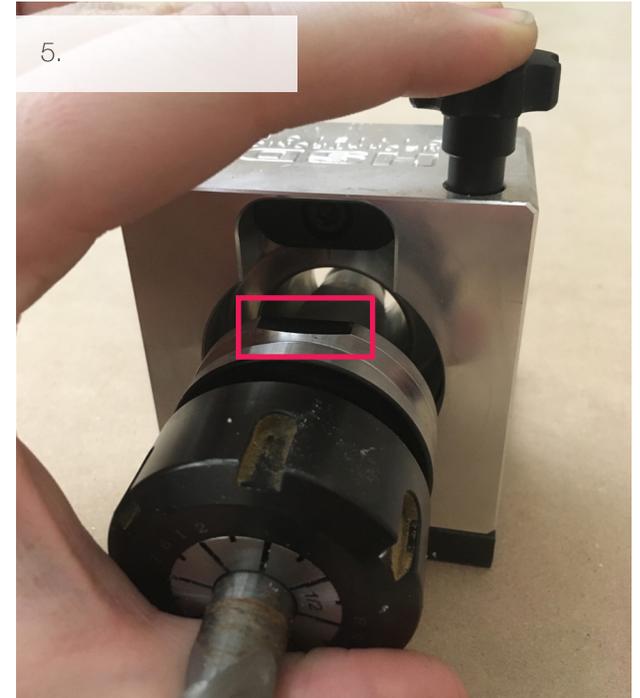
1. Take the nut of the ISO 30 Cone. Press the collet into the nut. It should snap together. Make sure that you choose the collet that is the same size as the bit you plan to use.
2. Put the proper bit into the collet.
3. Hand screw the ISO 30 Cone to the nut.



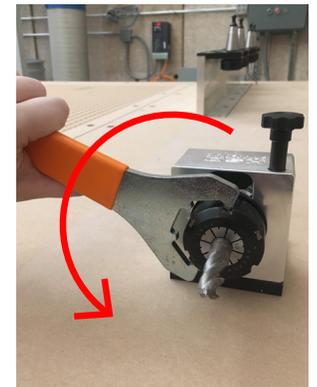
Automatic Tool Changer

Changing the Bit

4. Press down on the button on the Chuck Holder.
5. Put the cone of the Iso 30 Cone through the hold in the Chuck Holder. Line up the Iso 30 Cone so that one of the chamfers is facing up.
6. Release the button on the Chuck Holder to lock the Iso 30 Cone.
7. Tighten the nut with the Spanner Wrench; Clockwise- to tighten the collet, Counter clockwise to loosen.



Clockwise



Counter Clockwise

Wireless Pendant

This remote should be on the table with the ShopSabre Pro 510 CNC Router's desktop. If the batteries need replaced please let the CCL Associate know. Be sure that the USB receiver is plugged into the back of the desktop prior to trying to use the remote.

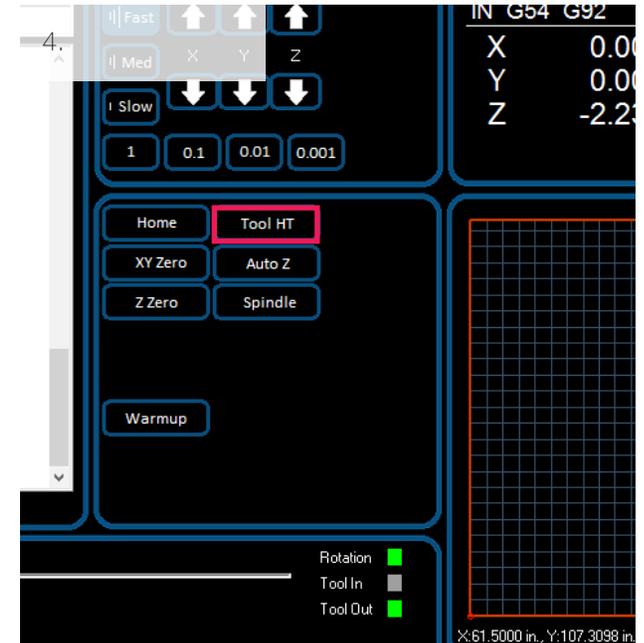
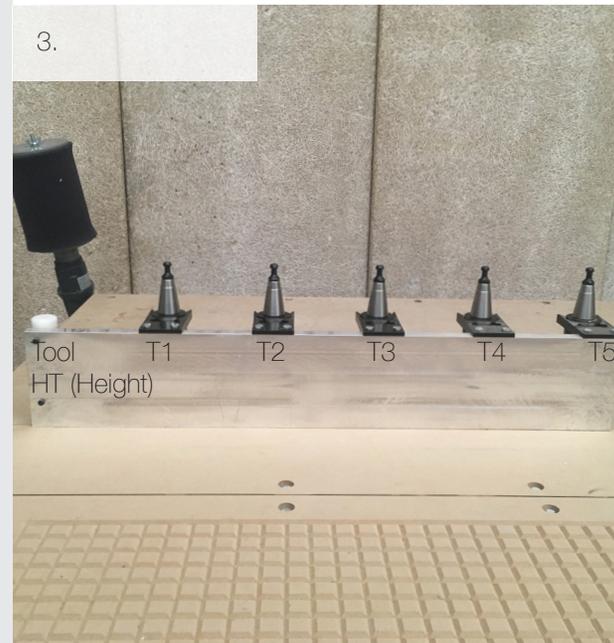
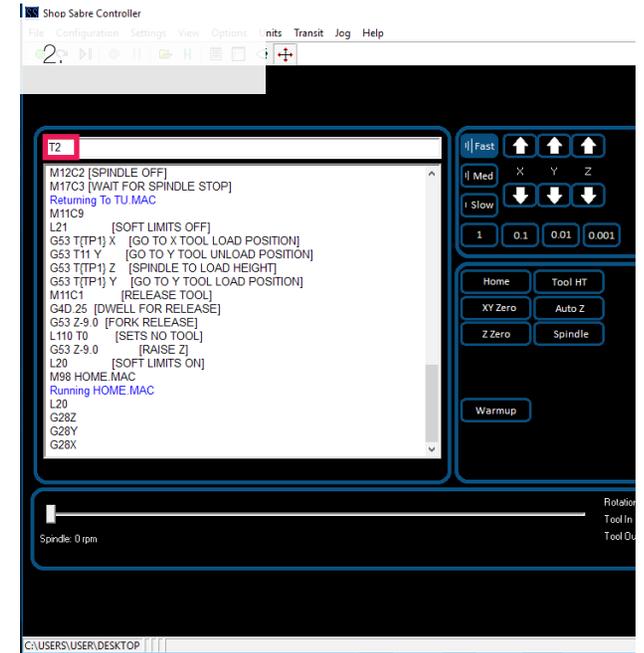
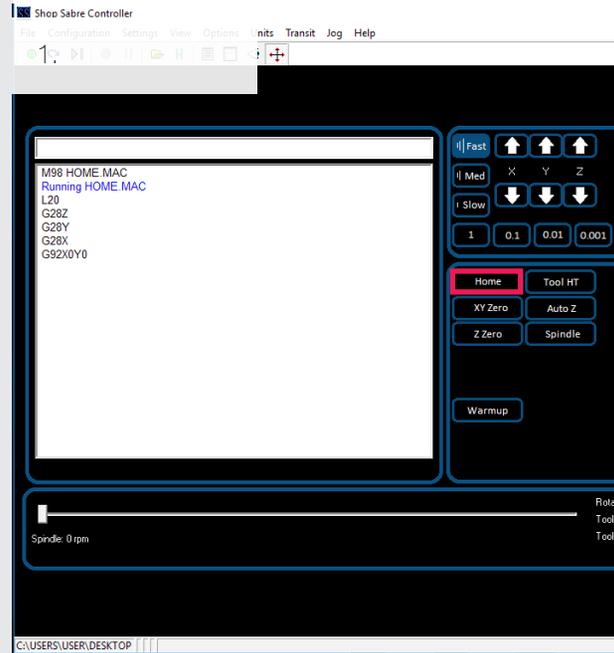
1. Press down to turn on the remote.
2. Abort- Aborts job
3. Pause Cont.- Pauses job
4. Change the axis to move the CNC router.
5. Wake- the pendant's screen will turn off after a time of not being used. This turns it back on..
6. Home- homes the CNC Router
7. Go XY- Moves the machine to your current XY work zeros at rapid speed.
8. Trans- Multiple presses cycle between speeds: Fast, Medium and Slow
9. Spin- Will trigger the spindle to start spinning.
10. Set XY- Sets the current bit location as an XY work zero.
11. Clear- Clears work zeros
12. Turning this dial choose which axis the remote is controlling.
13. Turning this dial clockwise or counterclockwise moves the axis selected by the dial above.



Automatic Tool Changer

Using with Shop Sabre Controller

1. Always 'Home' the CNC before using the automatic tool changer.
2. To automatically change a tool, in the input bar, type the letter 'T' with the desired tool number after with no spaces. At this point the machine will automatically move and change the tool.
3. To match the correct tool on the rack with the tool number on your ShopSabre screen simply count from the front of the CNC from left to right. If you would like to remove the bit and replace it with another type T and the number of the bit you would like to use. If you are done using the CNC and want no bit type T0 (no tool). **Always remove the bit before shutting off the CNC and move to the home position.** When operating the CNC without a bit you can type M12C1 to close the jaw to stop the loud blowing air.
4. Once you have loaded a new tool into the jaw of the CNC, measure that length of the by clicking the button labeled "Tool HT" on the Shop Sabre CNC Controller. At this time the machine will automatically move over to the white tool switch (see image 3) and touch off measuring and storing the new cutting bit. This operation will need to be preformed every time a new bit is inserted into the chuck.



Automatic Tool Changer

Using with Shop Sabre Controller

- 5. To see what tool the CNC has look at the bottom left hand corner of the Shop Sabre Controller software.

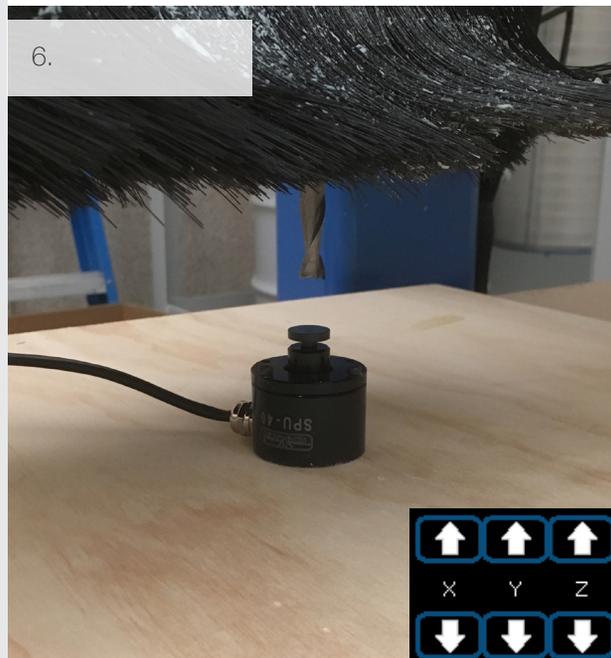
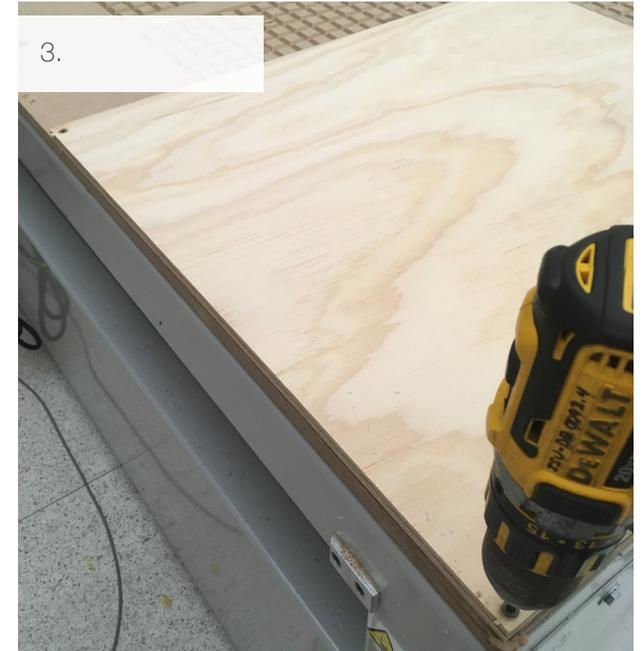
5.



No Tool

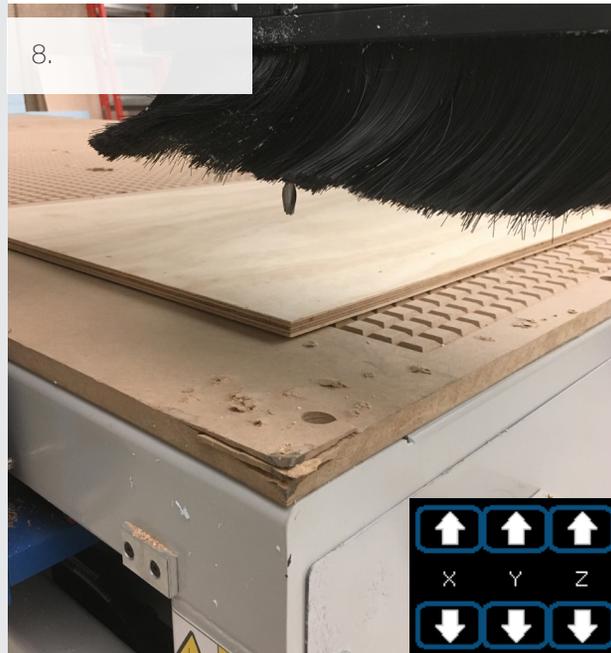
Preparing Material

1. Cut material to the proper dimensions.
2. Place the stock on the corner of the bed where the CNC homes near the emergency stop button.
3. Use a drill and screws to secure the stock to the CNC bed.
4. Home the CNC.
5. Use the automatic tool changer to get the first bit you will use to cut your rough toolpaths, for more information go the section titled Automatic Tool Changer.
6. Next use the X and Y arrows to get the bit over your stock. Then use the Z arrows to lower the Z axis to about 4" off the stock. Place the Material Height Touch Pad under the bit.
7. Click 'Auto Z.' At this time the Z axis will lower, touching off the pad and setting and storing the material height.



Preparing Material

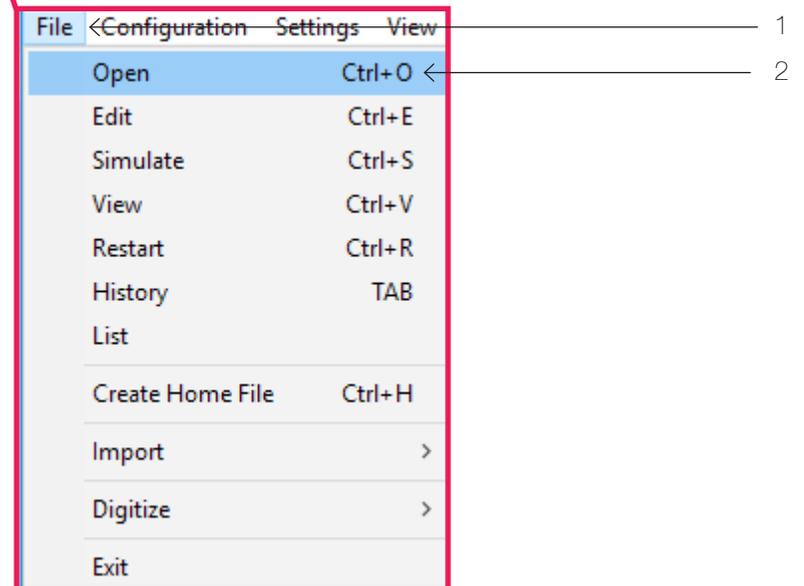
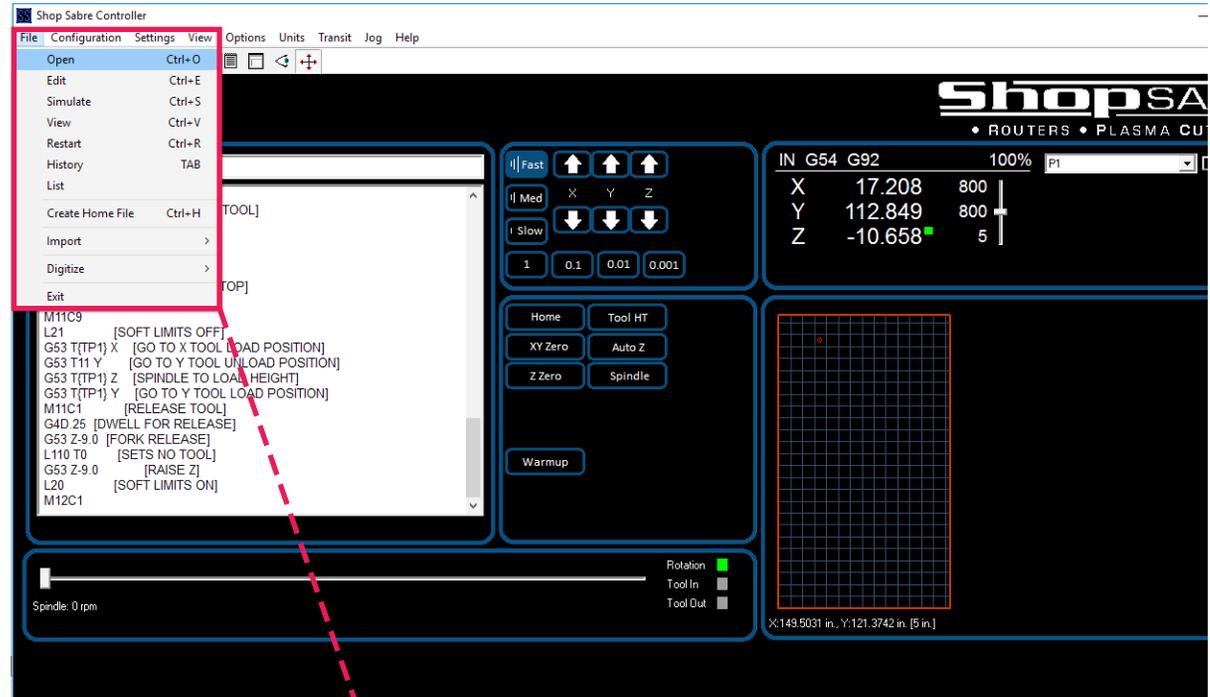
8. If your material is not at the corner of the bed or if you do not want the X and Y axis at homed zero. You can use the X, Y and Z arrows to find a new home. When you get to the location that you would like to use as zero click 'XY Zero.'



Shop Sabre Controller

Opening File

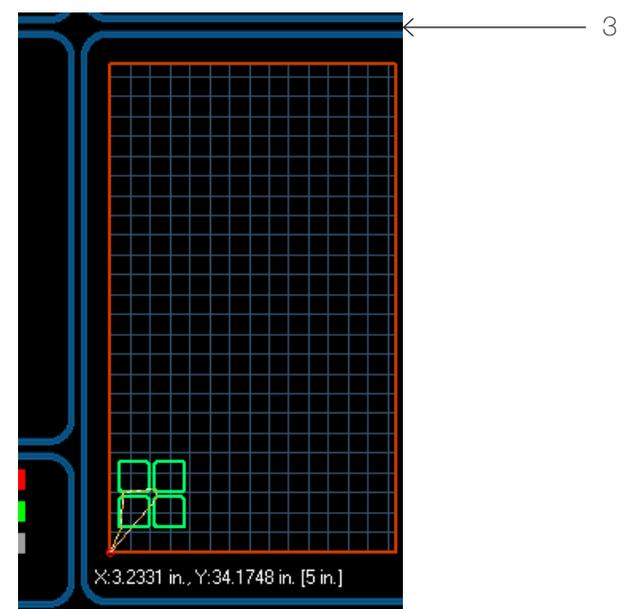
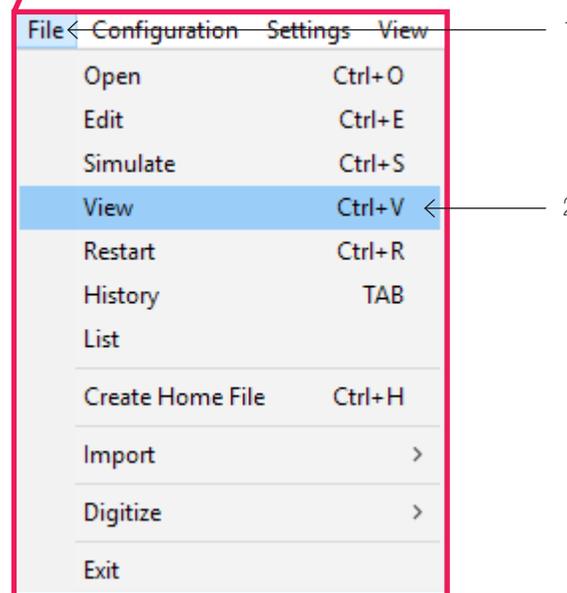
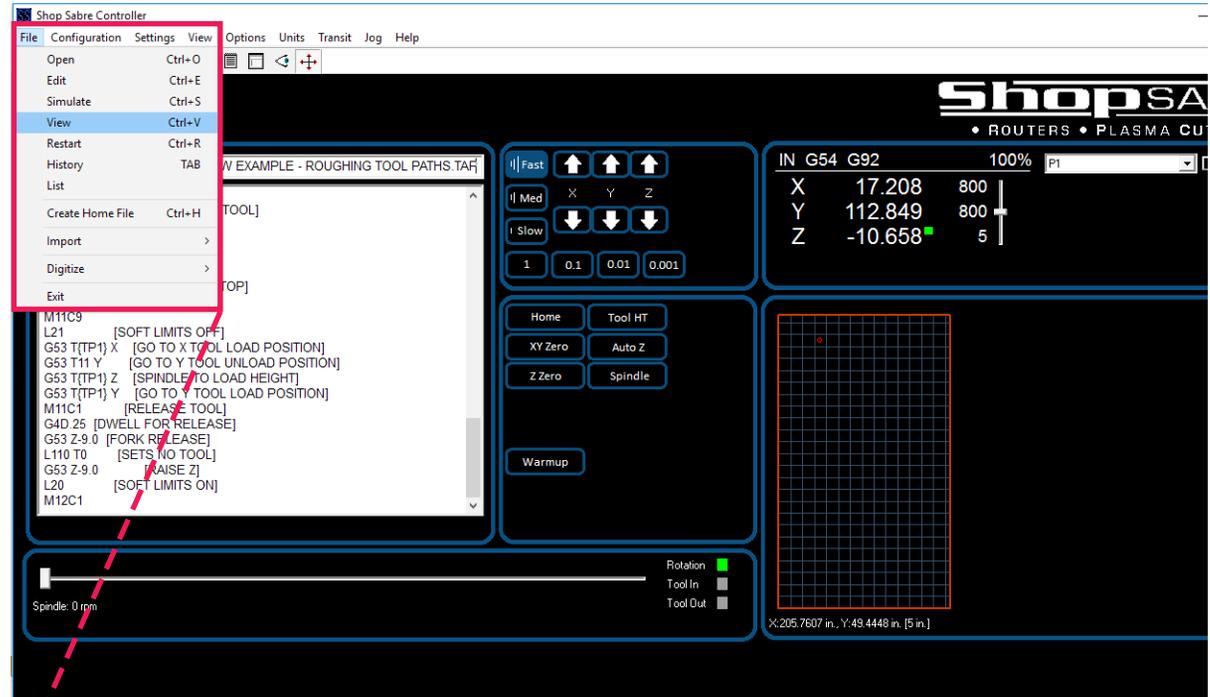
1. Click 'File'
2. Click 'Open'
3. Find file and click 'Open'



Shop Sabre Controller

Viewing File and Starting Job

1. Click 'File'
2. Click 'View'
3. A rendering of toolpaths will appear in the bottom right of the Shop Sabre Controller screen inside the gridded, red box representing the CNC bed. If the image is somewhere else on the bed rehome the CNC before starting your cut. The bit is represented by the red dot.
4. Then press enter on the keyboard to start cut.
5. Repeat steps 1-5 for each saved toolpath.



Workflow Example- Drilling



Workflow Example- Pocket



Workflow Example- 2D Profile

