NEXT WAVE CNC

SHARK

SD110/SD120

Owners Manual

© 2023 Next Wave CNC - All Rights Reserved - PDF website version

To Our Customers

Thank you for purchasing a **SHARK SD110/SD120** from **Next Wave CNC**! Your CNC opens up a world of creative possibilities for your ideas and designs. Following the step-by-step instructions, your **SHARK SD110/SD120** machine will be up and running in no time.

Whether you're new to CNC technology or a seasoned expert, your **SHARK SD110/SD120** machine provides a lot of flexibility for programming and operation. Our specially designed **LCD Pendant** is an auxiliary touch-screen that provides hands-on control over many of the functions of the **SHARK SD110/SD120** right at the machine, without the need for a connection to a personal computer. The **LCD Pendant** includes a USB port to transfer files from your PC.

For additional control and programming options using your PC, we've included our **Ready2Control** software. **Ready2Control** allows you to run toolpath files and control your **SHARK SD110/SD120** from the PC. It operates in a similar way to the **LCD Pendant** but includes additional machine control tools. See the **Ready2Control** User's Manual for more information.

Vectric VCarve Desktop software is a world-wide standard for designing parts and programming the toolpaths for your **SHARK SD110/SD120** machine using an intuitive interface. **Vectric VCarve Desktop** software includes a broad set of CNC clip art files and a library of tools for cutting and shaping the part. Design tutorials and software support can be found at *Vectric.com*.

System Requirements

All **SHARK SD110/SD120** machines plug into a standard 110v receptacle for power. A 15 amp or higher circuit is required.

Ready2Control software requires at minimum, a computer with Microsoft Windows 10 or higher, a 2 Ghz Quad-Core processor, a minimum of 8Gb RAM, 300 Mb Disk space (Program), a 1280 x 720 Display monitor, and a USB port.

Vectric VCarve Desktop software requires a 2 Ghz Multi-Core x86 or x64 processor (Intel or AMD) with Microsoft Windows 10 or higher, a minimum 4Gb RAM, 300 Mb Disk space (Program) + 7.7GB for optional tutorials and clipart, and a 1024 x 768 Display monitor.

SD110

SD120

Please Read This Manual Carefully

This manual provides important setup and operational information for your **SHARK SD110/SD120** machine. Using your **SHARK SD110/SD120** machine requires experience with basic computer operation and the technical knowledge to safely operate power tools.

Next Wave® CNC warrants your new **SHARK SD110/SD120** to be free from defects in material and workmanship for TWO YEARS from the date of purchase. The warranty applies only to the original retail purchaser of the **SHARK SD110/SD120** when purchased from an authorized *Next Wave CNC* distributor. This warranty covers the parts and labor to correct the defect. It does not cover the cost of shipping the machine and/or parts to **Next Wave CNC** for evaluation or repair. This warranty does not apply to problems arising from normal wear and tear, misuse, abuse, negligence, accidents, unauthorized repairs, alterations, or lack of maintenance. This warranty is void if the **SHARK SD110/SD120** or any portion of it is modified without the prior written permission from **Next Wave CNC**, or if the machine is located or has been used outside of the country where the machine was purchased.

Please contact **Next Wave CNC** to take advantage of this warranty. If **Next Wave CNC** determines that your **SHARK SD110/SD120** is defective in material or workmanship, **Next Wave CNC** will at its expense and upon proof of purchase send replacement parts to the original retail purchaser necessary to cure the defect. **Next Wave CNC** will repair your **SHARK SD110/SD120** provided the machine or affected components are returned to **Next Wave CNC**, shipping prepaid, with proof of purchase and within the warranty period.

Next Wave CNC disclaims all other express or implied warranties, including fitness for a particular purpose. **Next Wave CNC** shall not be liable for death, injuries to persons or property, or incidental, consequential, contingent or special damages arising from the use of the **SHARK SD110/SD120**.

Ready2Control software **Next Wave CNC** warrants **Ready2Control** and **Virtual Zero Unlimited** to perform as intended and will provide customer support to the original purchaser when purchased from an authorized retail distributor. Warranty only applies to the current version or the support needed to update a past version. The cost of the software upgrade (if any) is not covered by the warranty.

Lifetime Technical Support is provided to the original purchaser.

Technical support for the **Vectric VCarve Desktop** software furnished with your **SHARK SD110/SD120** is provided by **Vectric**. You can find a variety of training videos located at *www.vectric.com*. If you have questions or need software support, please get in touch with **Vectric** directly at *support@vectric.com*

Copyright 2023 **Next Wave CNC**
All Rights Reserved.

CNC SHARK is the registered trademark
of **Next Wave CNC**.

Ready2Control is copyrighted by
Next Wave CNC.

Virtual Zero Unlimited is patented by
Next Wave CNC.

Version: 1/30/2023

All other trademarks are the property of their respective owners. Information in this manual is subject to change without notice. The most recent version of this manual can be found at *www.NextWaveCNC.com/downloads-links*

Contact Us

If you need technical assistance with your **SHARK SD110/SD120** or software, please visit our Support webpage at:

NextWaveCNC.com/support
or email Customer Support at:
Support@NextWaveCNC.com.

Please include your product model number, date of purchase, and other pertinent information associated with the issue such as .tap files, VCarve files, screen captures, or photos of your setup or the problem.

Support Email: support@NextWaveCNC.com
Available: 9 am – 5 pm (ET) Monday-Friday

Serial Number and Software License Information: For easy reference and record keeping, enter your **SHARK SD** and **Vectric** information at right. To locate the information, refer to the pages below.

Controller Box Serial Number

_____

LCD Pendant Serial Number

(page 7)_____

LCD Pendant Unlock Code

(page 7)_____

VCarve License Code

(page 15)_____

Your VCarve User's Name

(page 15)_____

NEXT WAVE® CNC

SHARK®
World Class. Made in USA.

Next Wave CNC, LLC,
600 W. Boundary St., Perrysburg, Ohio 43551
USA
Main Office Phone (419) 318-4822
info@NextWaveCNC.com

CONTENTS

Let's Assemble Your CNC Router
Machine Overview 5
Connecting the Cables 6

Machine & Pendant Registration 7

Safety Tips 8
Stop Your CNC Fast 9

Basic Pendant Functions
Jog & Speed Settings 10
Zero the X & Y-Axes 10
Move to Origin 10
Zero Z-Axis - Manually 11
Zero Z-Axis - With Touch Plate 11

Install Vectric VCarve
Software Installation 12
Opening an Account 18
Updating 18
VCarve Workflow 20

1st Build–A Simple Project
Tool Holder 26

2nd Project–Through Cuts
Cutting Board 28

3rd Project–Carving
Trivets 30

Maintenance 33
Resources 33

Upgrade Accessories 34

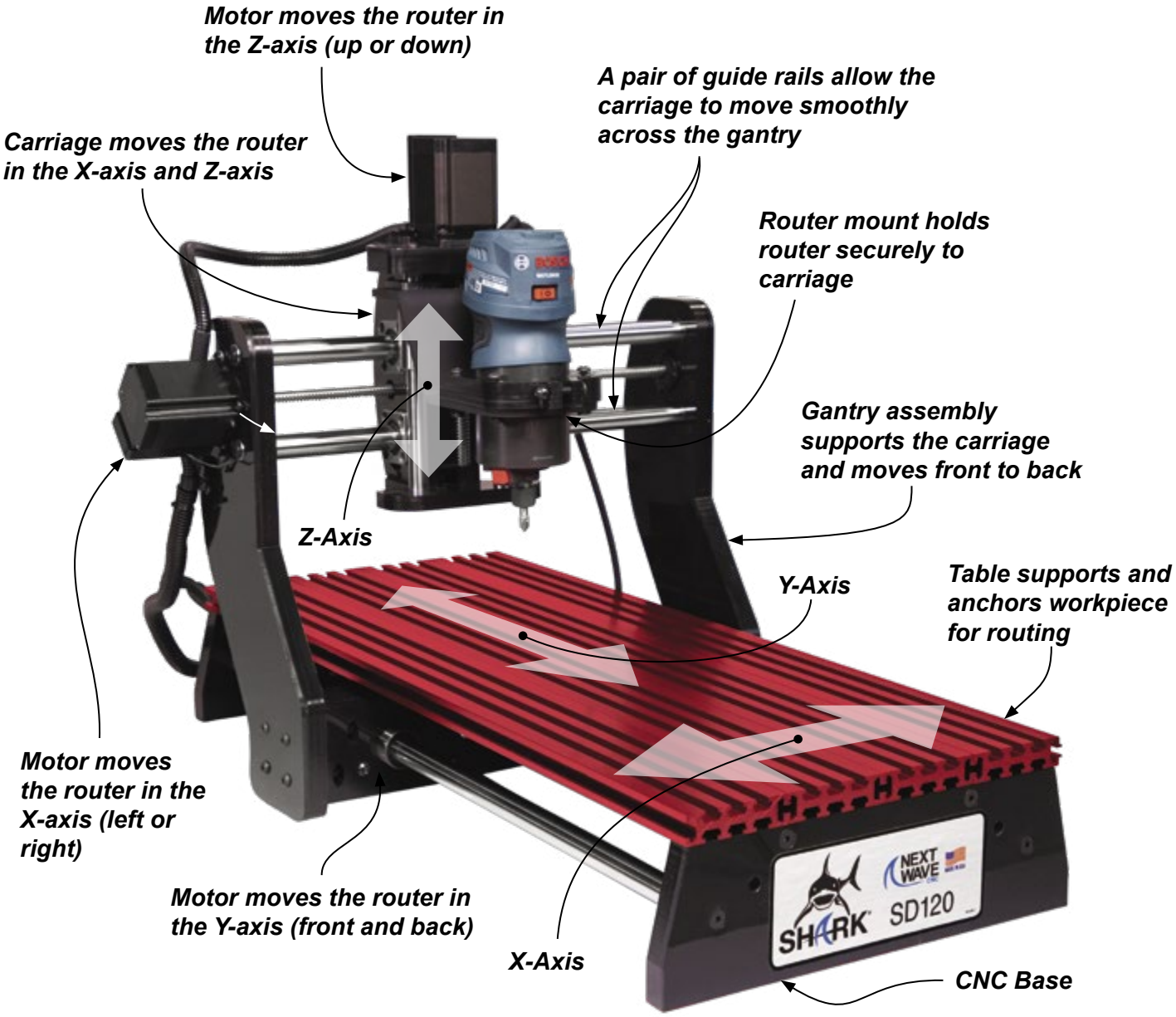
For videos and tips on getting started with your CNC visit:
www.NextWaveCNC.com/gettingstarted
or scan the QR code below.

Machine Overview

Your **SHARK SD110/SD120** machine comes preassembled. All you need to do is install a router and connect the cables.

When removing the CNC unit from the shipping box, we recommend you enlist some help. The unit is heavy, so make sure to have an area clear to set the unit when removed from the box.

See the illustration below to get familiar with all of the parts of your CNC. You'll find a variety of videos covering your **SHARK SD110/SD120** machine online at *https://www.NextWaveCNC.com/video*.



Setting Up Your Machine

Unpack the **SHARK SD110/SD120** machine and verify that all items are present. Make sure when removing the **SHARK SD110/SD120** machine from its box that you lift it from the base or table. Lifting it by the gantry can cause misalignment.

- **SHARK SD110/SD120** machine
- CNC Control Box
- 15 pin LCD Pendant Cable
- Touch Screen Pendant
- Power Supply
- Power Cord
- 2 Hold Down Clamps
- Router ON/OFF Controller
- 3 Router Clamp Options
  - B1 Bosch 1 HP Colt
  - B2 Bosch 1.25 HP Colt
  - D DEWALT Trim router

To set up your **SHARK SD110/SD120** machine, begin by identifying the cables coming from the motors. The plugs are labeled for the X-, Y-, and Z-axis motors. Identify the X, Y and Z plug ports on the back of the control box. Plug in the the X, Y, and Z cables and tighten the locking screws.

Connect the **LCD Pendant** to the front of the control box using the included cable and tighten the locking screws. Plug the Transformer cord into the front of the control box and plug the power cord into the transformer. Verify that all three axis drive cables are connected to their respective Controller Interface leads.

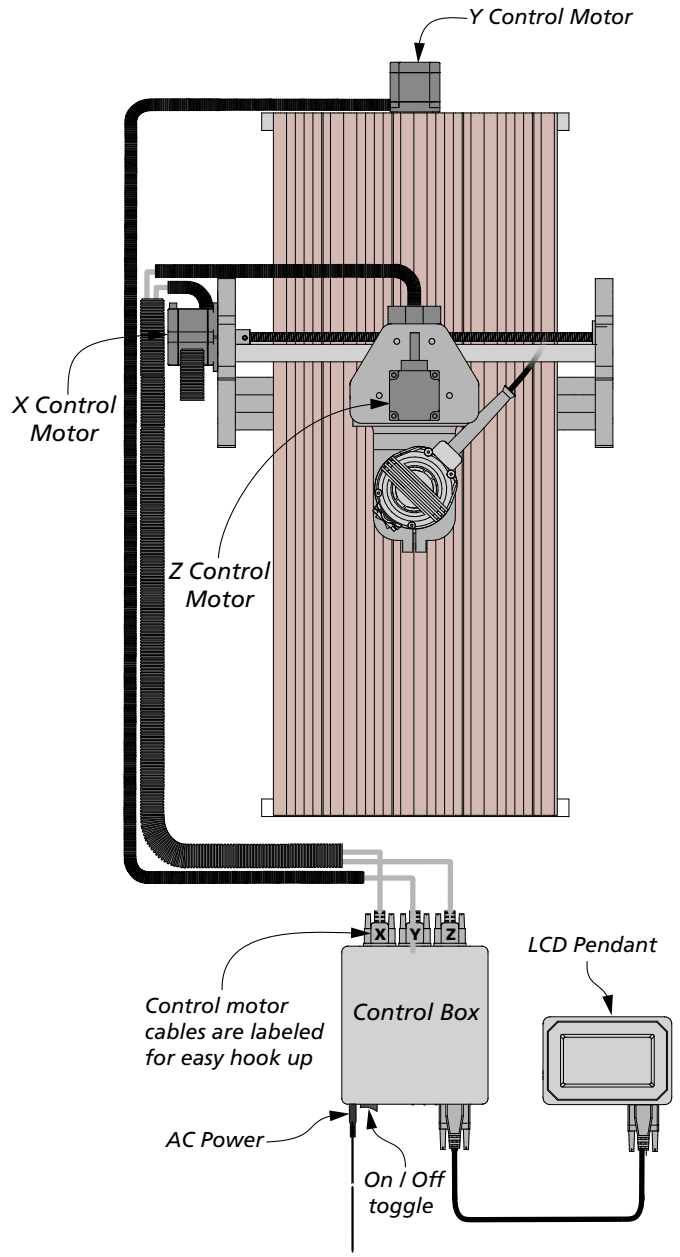
To minimize signal interference, arrange the drive motor cables, power cables, and USB cable in a manner that minimizes overlap.

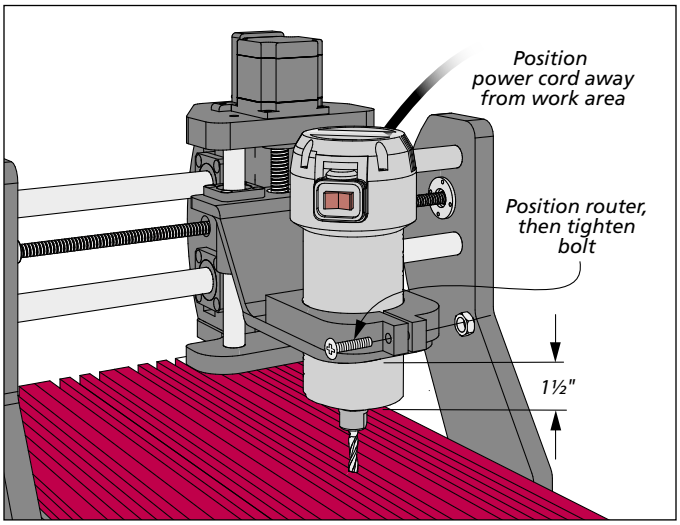
Plug the **SHARK SD110/SD120** Power Supply into a 120V AC wall outlet, power strip, or surge protector. We recommend a switched power strip. When the power is switched on, you will hear a slight bumping or clicking noise coming from each of the motors. This noise is momentary and will occur every time.

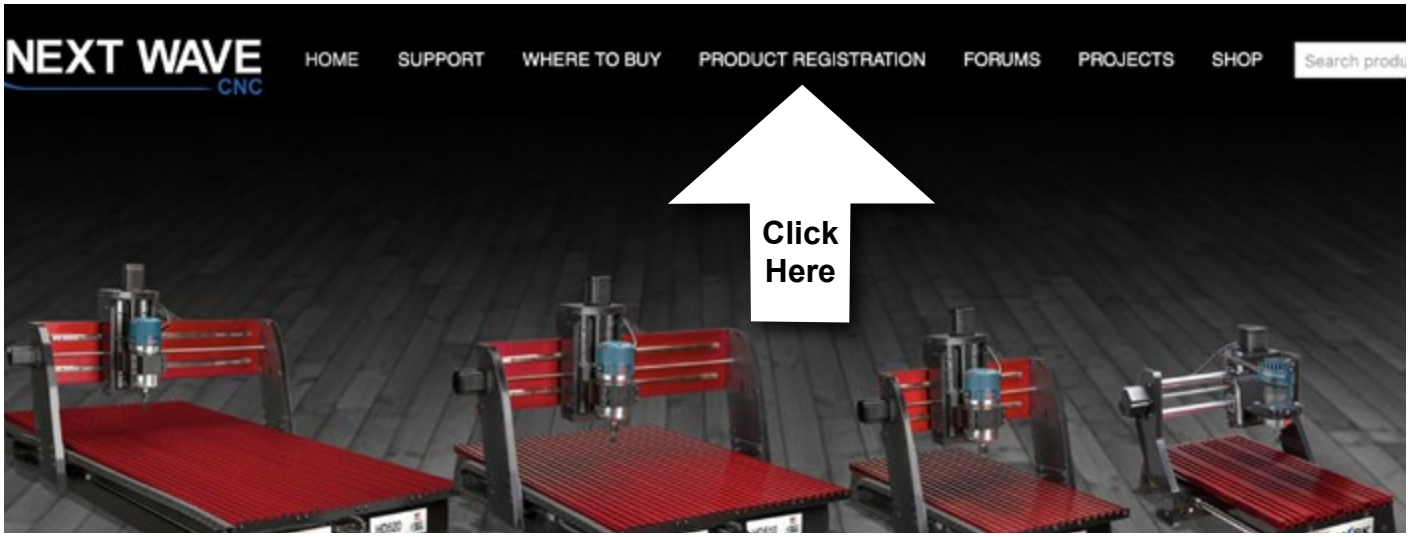
Install the router onto the router mount by loosening the clamp ring. Insert the router until the motor protrudes about 1½" below the clamp ring or so the tip of your shortest bit is slightly above the table.. Tighten all bolts securely. See lower drawing at right.

The machine's hardware is now set up and is ready for basic testing. You must register your **SHARK SD110/SD120** machine software before you can completely test your machine.

Wiring Schematic







LCD Pendant Registration

Before continuing the setup of your **SHARK SD110/ SD120** machine, register it online by navigating your web browser to **NextWaveCNC.com**. Click on the PRODUCT REGISTRATION tab at the top of the screen (see above).

Once you complete the registration, create a customer account and follow the instructions to generate a unique code that will unlock your **LCD Pendant**. You will also receive the code in your email inbox. Use the number keys on the **LCD Pendant** to enter your Unlock (Access) Code. Press Submit. This unlocks the **LCD Pendant**.

For complete and detailed instructions on using the **LCD Pendant**, refer to the **LCD Pendant** Owner's manual.

Registering the Pendant

Unlock the **LCD Pendant** by entering the access code you received upon registration.

After successfully unlocking the **LCD Pendant**, you're ready to control your **SHARK CNC SD Series** machine.

Our 12 Golden Rules for Safety

1. Read, understand, and follow the safety and operating instructions before using your **SHARK CNC**.

2. Take time to fully understand how to safely operate your **SHARK CNC**.

3. Set up your **SHARK CNC** following the instructions in this manual.

4. Always wear appropriate eye and hearing protection when operating your **SHARK CNC** and accessories.

5. Machine only non-ferrous metals with your **SHARK CNC**.

6. Never attempt to adjust the workpiece or move the **SHARK CNC** while it is running.

7. If needed, use the Cancel or Pause buttons to Stop or Pause your **SHARK CNC** during an operation.

8. Never leave your **SHARK CNC** unattended while it is running.

9. While operating your **SHARK CNC**, keep a multi-purpose, dry-chemical fire extinguisher nearby. It must be rated for both A & C fires.

10. For added safety and convenience, connect your **SHARK CNC** to a 110-115V surge-protected power strip with an on/off switch. This provides an additional way to turn off the machine in case of an emergency.

11. Follow all Safety instructions provided with your **SHARK CNC** and accessories.

12. Follow all accepted and recommended safety precautions and practices for woodworking and machining.

Stop Your CNC Router Fast!

Every CNC user experiences times when the machine seems to have a mind of its own. This can be caused by errant tool paths in the design file, power surges, broken tools, or any number of other reasons. There are three ways to shut down the system in an emergency:

1. Use the **LCD Pendant** to stop the machine by pressing the red STOP button. This stops the CNC's movement, but not the router, so you must turn off the router separately.

2. Use the power switch on the rear of the control box.

3. If the control box and router are plugged into a switched power strip, turn off the power strip. This stops the CNC's movement and turns off the router at the same time.

Methods 2 or 3 also work to turn off your machine at the end of the day.

IMPORTANT NOTE: If you stop your CNC using any of these methods, you will need to restart the cutting from the beginning of the file. The cutting progress and machine positions are lost during an emergency shut-down.

Once you are familiar with the locations of these switches, you'll be ready to use them if and when you need them in the middle of a routing project.

3 Ways to Shut It Off

1

2

3

Basic LCD Pendant Functions

The **LCD Pendant** has many great functions to control the **SHARK SD110/SD120** machine. But you only need to learn a few basic operations to get started.

JOG AND SPEED SETTINGS. Let's start with learning how to move (jog) the router. Jogging the router allows you to move it out of the way or to another location. This comes in handy for locating or removing clamps on the workpiece, changing router bits, or setting up start points for the router to run a routine.

First, look at the main screen (Figure 1) and identify the jog keys. They are quick to identify when standing in front of your CNC and are intuitive to use. You can jog the router right or left in the X-axis, toward you or away from you (Y-axis), or up and down in the Z-axis. You control the speed of the jog from fast to slow by using the speed key on the right side of the **LCD Pendant** screen (red arrow). Pressing this key repeatedly cycles between Fast, Medium, and Slow jog speeds.

Take some time to experiment with moving the router around. Jog the router to get a feel for the different speeds. Take special note of the position readouts at the top of the **LCD Pendant** screen. Try to stay within the limits of the machine. If the router travels and bumps against the limits, you may hear a "grinding" noise. Don't be concerned—the noise is the sound of the internal motor magnets electronically skipping over each other. This does not cause any damage, but it's good practice to avoid hitting the travel limits of your machine.

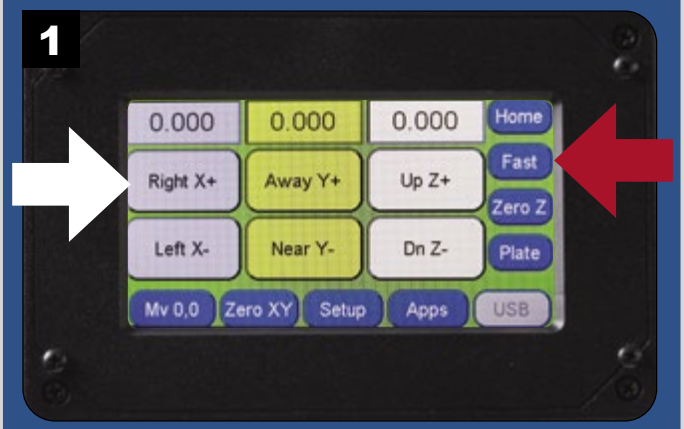
ZERO X- & Y-AXIS. Another function of the **LCD Pendant** is the "Zero XY" function, as shown in Figure 2. This sets the current position of the router as a zero. This location is also referred to as the origin or datum point for the X- and Y-axes.

Origin points are typically used as a starting point to run a cutting routine. Often the center of a workpiece or the lower left corner are used as a starting point for the X- and Y- axis. The Z-axis origin is usually set from the top of the workpiece. We'll cover setting the Z-axis on the next page. For now, practice setting up the origin (datum) point by moving the router to a number of locations and zeroing the X- and Y-axis.

MOVE TO THE ORIGIN. Another useful **LCD Pendant** function for moving the router is the "MV 0,0" key (Figure 3). Pressing this key with a short press or single tap automatically moves the spindle to the X- and Y-origin point you have previously set. Give it a try, move the router away from your X,Y zero (origin/datum) point and then press the Mv 0,0 button and watch the router automatically move back to the X,Y zero point. Using a quick double tap moves the spindle to the original X, Y, and Z position.

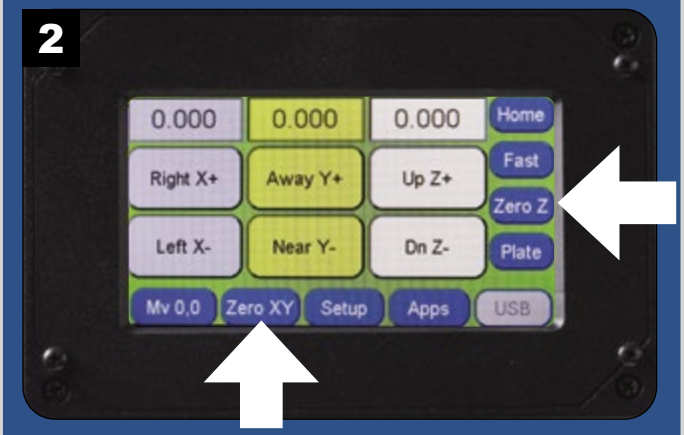
Make It Move

1



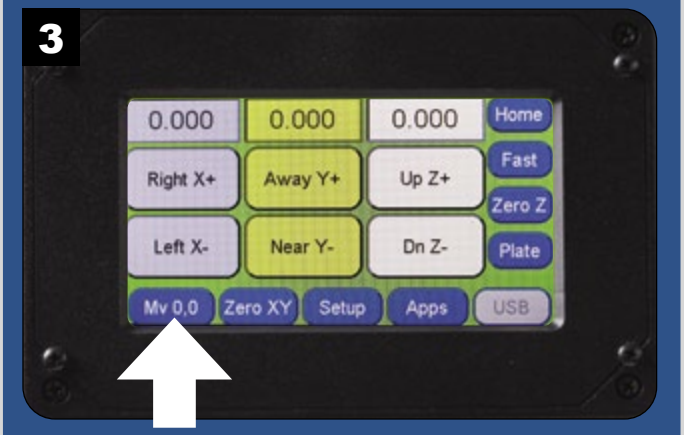
Changing the location of the router is easy with the easy-to-use **LCD Pendant** interface.

2



The **LCD Pendant** makes it easy to set a new X, Y, and Z origin for the router's location.

3



Use the MV 0,0 button to move the router back to the previously set origin.

Most of all, have fun! Experiment with the basic **LCD Pendant** functions to get familiar with how your **SHARK SD110/ SD120** machine moves and sounds.

Learning to zero the X-, Y-, and Z-axis is an important skill for accurate work. You can zero the Z-axis manually or automatically with the help of the **Next Wave CNC** Touch Plate, available as an add-on accessory (see figure 6 below).

ZERO THE Z-AXIS MANUALLY. Setting the Z-axis zero point manually is an easy process. First, install the router bit you'll use for the project into the collet of the router. Begin by moving it to the marked center of your workpiece using the jog functions shown on the opposite page. Change the jog speed on the **LCD Pendant** to "Slow." Carefully move the bit over the center mark on the workpiece by using the X and Y jog keys. With the bit centered over the workpiece, press the "Zero XY" key to zero the X- and Y-axis.

To zero the Z-axis, lower the bit to within about ½" of the top of the workpiece and set the jog speed to slow. Slide a piece of paper underneath the bit and move it back and forth as you tap the "Dn Z" key to gradually lower the bit. When the bit starts to "grab" the paper, stop. Press the "Zero Z" key again, then raise the router bit approximately ½". At this point, all three axes are calibrated.

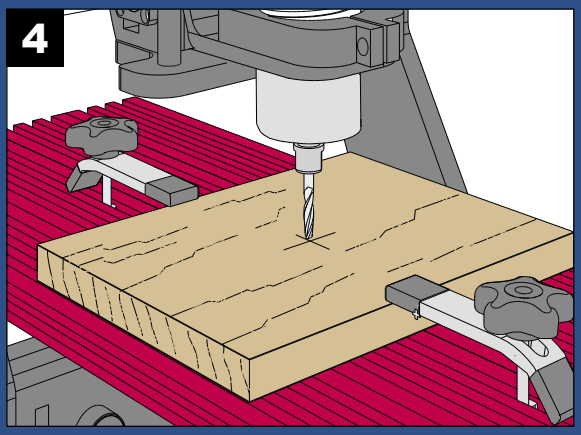
USING THE CNC TOUCH PLATE. Using the CNC Zero Touch Plate accessory is a fast and accurate way to zero the Z-axis, as shown in figure 6. Begin by plugging in the lead into the **LCD Pendant**. Position the Touch Plate under the router bit. Attach the magnet on the end of the lead to the collet or bit.

Press the "Plate" key on the main screen of the **LCD Pendant** and follow the steps listed. When the router bit comes into contact with the Touch Plate, this completes an electrical connection to indicate the zero position of the Z-axis. The bit automatically retracts to a safe height.

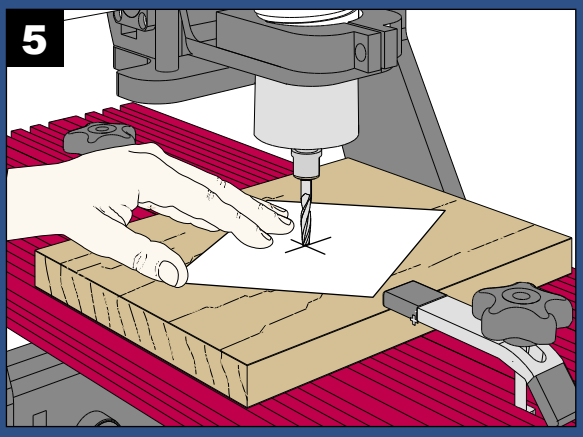
For videos and tips on getting started with your CNC visit:
www.NextWaveCNC.com/gettingstarted
or scan the QR code below.

CNC Getting Started
SCAN for More Info

Manual Setup

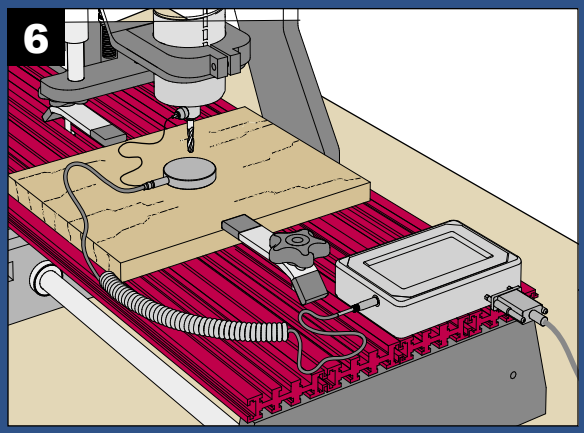


Zero the X- and Y-axis first by jogging the bit until it's positioned over the center of the workpiece.



Gradually lower the bit until it just starts to touch a piece of paper placed on the workpiece.

Optional Zero Touch Plate



Plug the cord into the side of the **LCD Pendant**. Position the Touch Plate under the router bit and attach the magnet to the collet nut or side of the bit. Then press the Plate button on the **LCD Pendant** and follow the prompts.

Installing VCarve Software

If you want to jump right into cutting a project, you can temporarily skip this section and go to pages 26-32. On these pages you will find three pre-designed projects that don't require VCarve. Once you've cut those and have a better feel for your machine, come back to this section, which will lead you through the installation of **Vectric VCarve Desktop**,which is the program you will use to design your own projects.

Using a CNC router is unlike any other powered woodworking tool—you can't just power it on and put to use right away. Making projects with the CNC is a little more involved. The CNC must be programmed with instructions that define the router bits, toolpaths, and cutting speed to be used for your project. You have already experienced running project cut files on your CNC router by making the organizer, cutting board and trivet.

For your own creations, you need to use VCarve to design your project which also involves creating the toolpath files that you then run on your **SHARK SD110/SD120** machine to cut your project.

You're **SHARK CNC** comes with a license for **Vectric VCarve Desktop** software. This powerful and user-friendly software unleashes your creativity. For system requirements, visit:
vectric.com/support/system-requirements

There are a number of steps involved in installing, registering, and configuring VCarve so be patient, go slow, and remember that you only have to do this once. Step-by-step instructions start on the next page.

NOTE: During the installation process,you're required to register VCarve twice. The first time, you'll use the Username and License Code printed inside the box lid. Then you'll need to register again using your personal Username, and a new License Code that is emailed to you from **Vectric** after the first registration. This process is shown in the steps that follow.

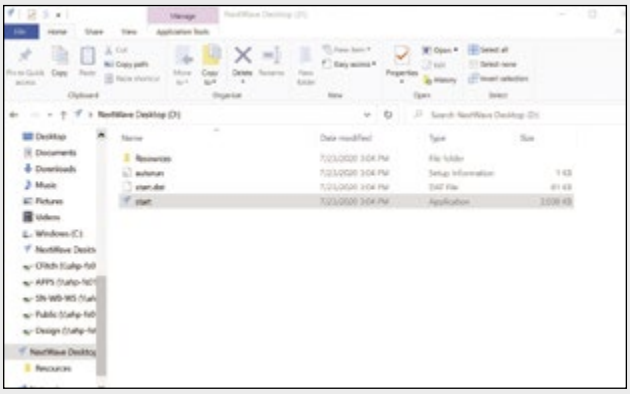
Vectric offers excellent instructional tutorials and customer help at *Vectric.com*. Take advantage of this valuable resource.

Important –Please Read

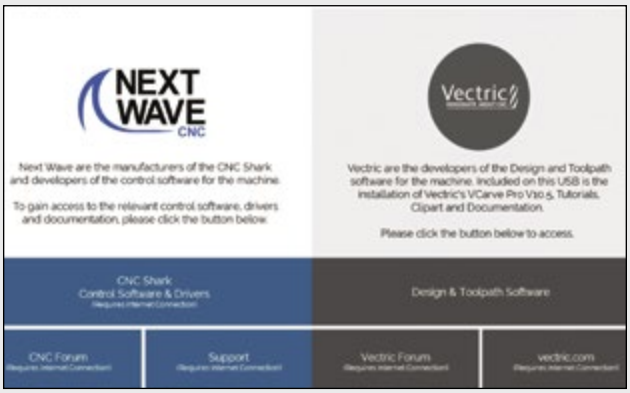
Installing and registering the VCarve software requires several steps. The instructions in this section show the typical installation and registration process. Following these steps will result in a successful installation. However, variations in security settings, virus protection and operating systems can prevent a smooth installation. The process may also vary due to any changes to the installation process occurring after this manual was printed. If you experience problems installing VCarve on your computer, first try repeating the process. If the problem persists, email *Register@Vectric.com* for assistance. VCarve is supplied and supported by Vectric, LTD. They are best suited to help you with software installation problems.

Software Installation Process

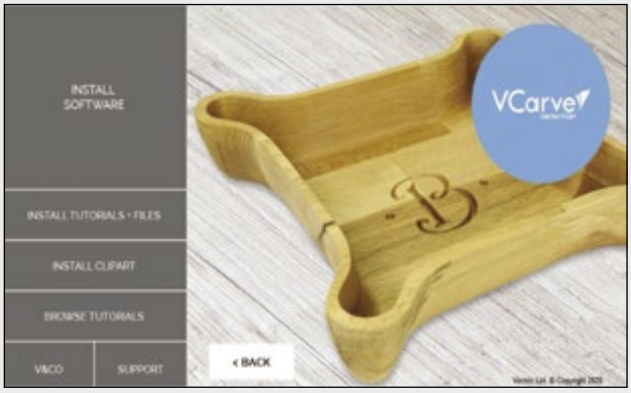
STEP 1. Insert the **VCarve** USB drive into a USB port on your computer.



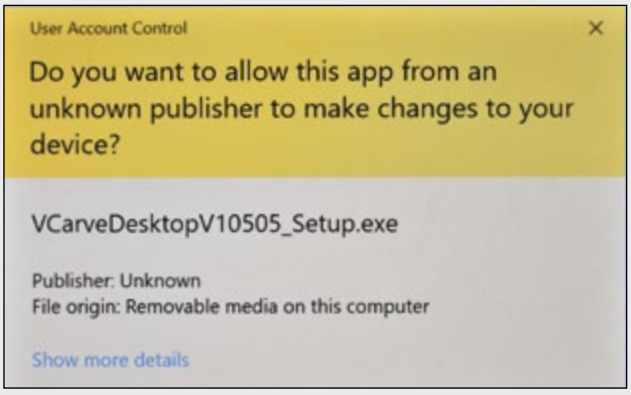
STEP 2. Using File Explorer, navigate to the USB drive and double-click on the file *setup.exe*.



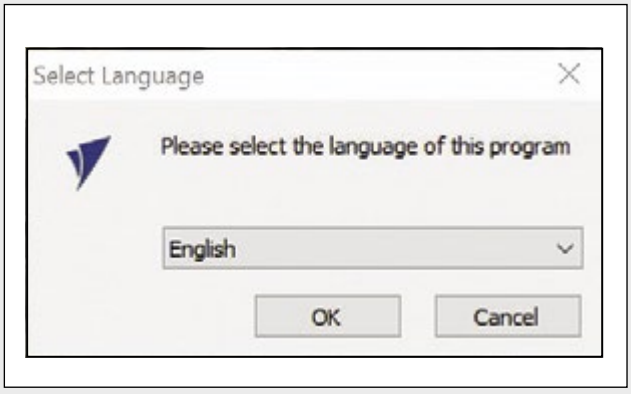
STEP 3. Click on *Design & Toolpath Software* to install **Vectric VCarve** on your computer.



STEP 4. Click on *INSTALL SOFTWARE* to begin the installation.



STEP 5. A box will pop up (appearance may vary) asking if you want to allow this software to make changes on your computer. Click *OK*.

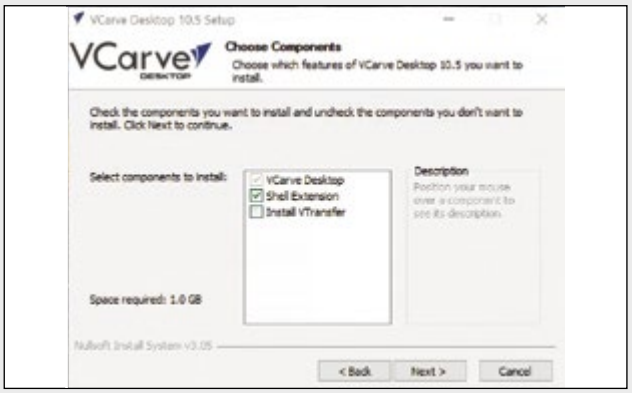


STEP 6. Choose your preferred language from the pull-down list and select *OK*.

Software Installation Process *continued*

STEP 7. Follow the instructions on this pop-up and select *Next*.

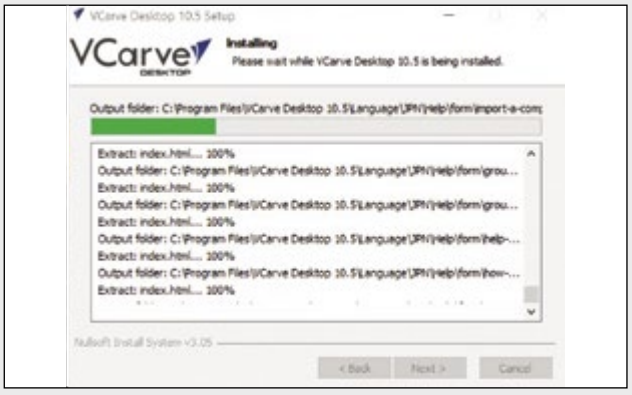
STEP 8. Read through the license agreement and select *I Agree* to continue the software installation.



STEP 9. Ensure that *Shell Extension* is selected then click *Next*.

STEP 10. Using the default location on your computer is recommended, but you can use the *Browse* button to select a different location. Click *Next* to continue.

STEP 11. Accept the default *Start Menu Folder* by clicking on *Install*.

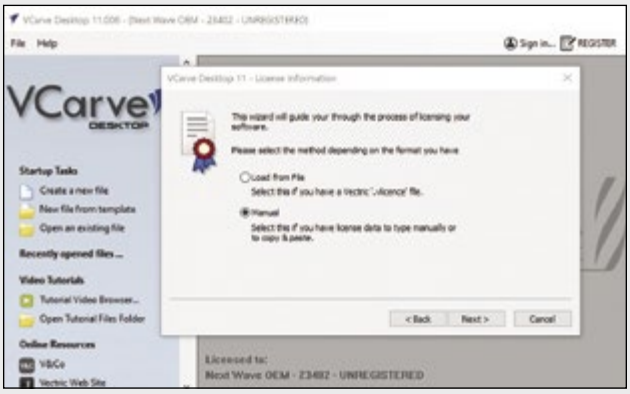


STEP 12. Wait until the installation process is complete before clicking on *Next*.

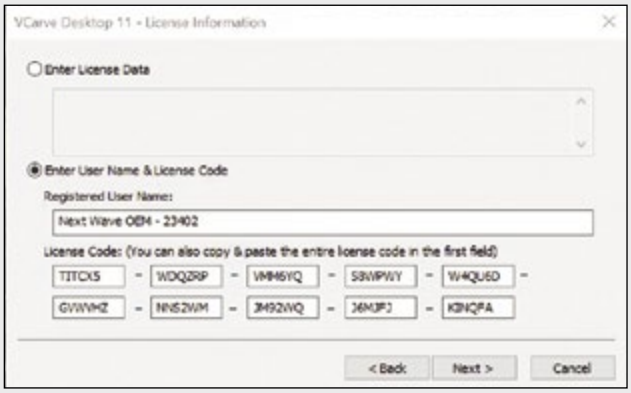
Software Installation Process *continued*

STEP 13. With the box labeled *Run VCarve Desktop* checked, press *Finish* to start *VCarve*.

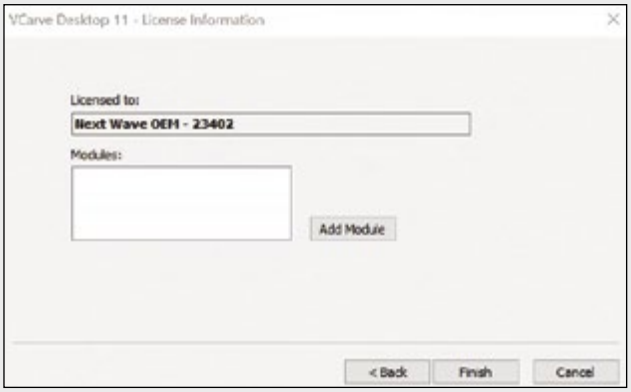
STEP 14. To begin registering the software, choose *Offline* then click *Next*.



STEP 15.. You'll be entering your license information from the box lid, so select *Manual* and press *Next*.



STEP 16. Choose *Enter User Name & License Code*, then enter the *Registered User Name* and *License Code* from inside the box lid.



STEP 17. Click *Finish* to proceed to the *Personal Registration* screen.

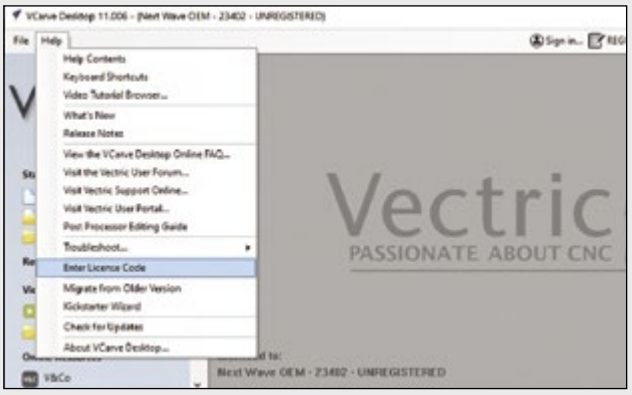
STEP 18. You've registered the **Next Wave** version of the software, now you need to enter your personal registration information by clicking *Register Now*.

Software Installation Process *continued*

STEP 19. Select the option for personal use under *Registration Type* and enter your *Contact Details*. Click *Submit Registration Request*.

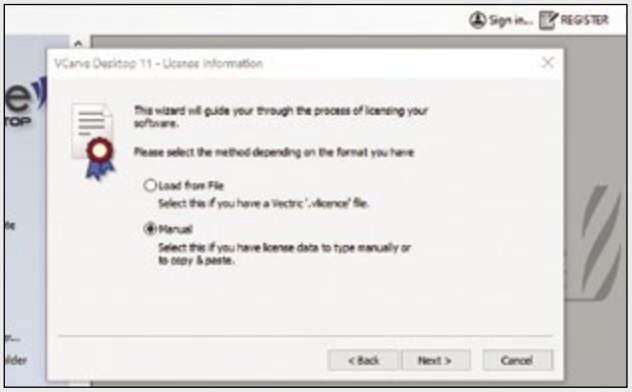
STEP 20. If you have successfully registered the software, you will see this screen.

STEP 21. You will receive an email from **Vectric** that contains the registration information you need to license the software under your personal user name.



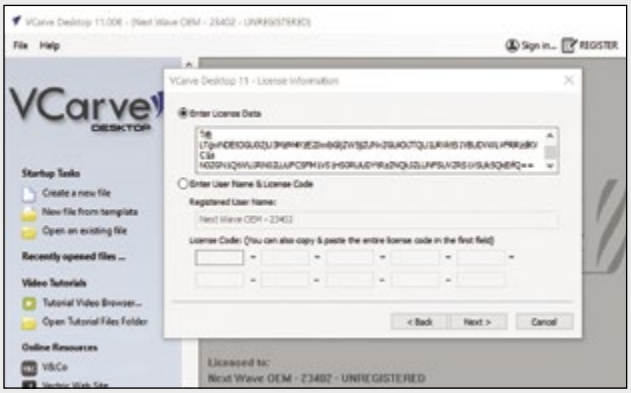
STEP 22. In **VCarve**, you may see a pop-up dialog box asking for your license code. You can also select *Enter License Code* from the *Help* drop-down menu.

STEP 23. Since you'll be entering the license information from the email you received from **Vectric**, select *Offline* then click *Next*.



STEP 24. You'll be able to copy and paste your license code from the email, so choose *Manual* and click on *Next*.

Software Installation Process *continued*



STEP 25. Choose the option to *Enter License Data* then cut and paste the license information from the email you received. Click *Next*.

STEP 26. The license information shows your user name. Check the box *Restart Now* then click *Finish* to restart **VCarve**.

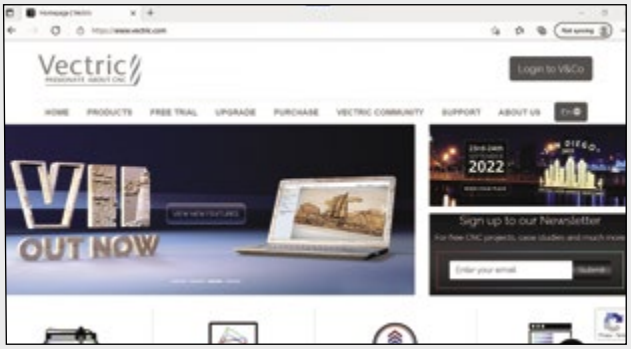
STEP 27. If you click *Finish* in Step 26 without checking the box *Restart Now*, you can simply choose *Exit* from the *File* pull-down menu to exit **VCarve**.

STEP 28. After restarting **VCarve** you should see your username at the top of the screen and under *Licensed To* in the opening window.

Important –Please Read

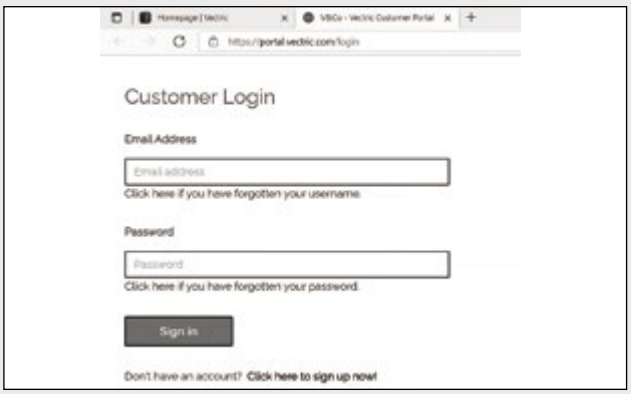
Installing and registering the VCarve software requires several steps. The instructions in this section show the typical installation and registration process. Following these steps will result in a successful installation. However, variations in security settings, virus protection and operating systems can prevent a smooth installation. The process may also vary due to any changes to the installation process occurring after this manual was printed. If you experience problems installing VCarve on your computer, first try repeating the process. If the problem persists, email *Register@Vectric.com* for assistance. VCarve is supplied and supported by Vectric, LTD. They are best suited to help you with software installation problems.

Opening a Vectric Account



STEP 1. At *Vectric.com*, click *Login to V&Co* to create a portal account for access to special content downloads, software updates, and technical support.

STEP 2. If you have an existing **Vectric** account, log in here.

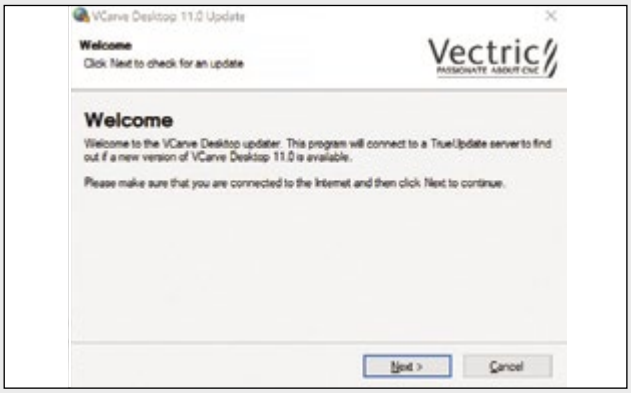


STEP 3. If you have an account at *Vectric.com*, enter your user name and password. If you're a new user, scroll down and select *Click here to sign up now!*

STEP 4. For setting up a new **Vectric** account, enter your contact information and password, accept the *Terms & Conditions*, and click *Sign Up*.

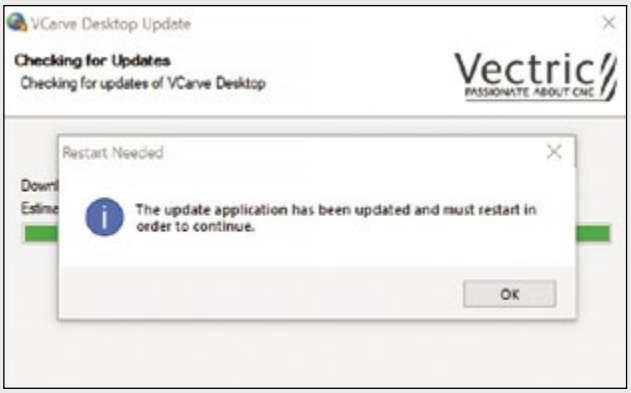
Updating Vectric VCarve

STEP 1. If software updates are available, an update notice appears in the upper right corner of the screen when you open **VCarve**. Click it to install the update.

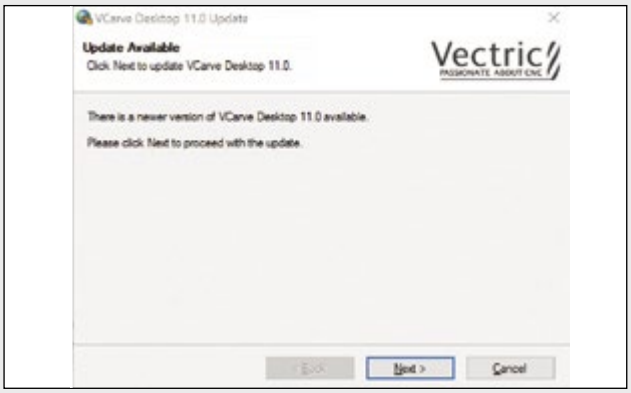


STEP 2. Your system must be connected to the internet to download updates. When you click *Next*, the system checks for any available updates for **VCarve**.

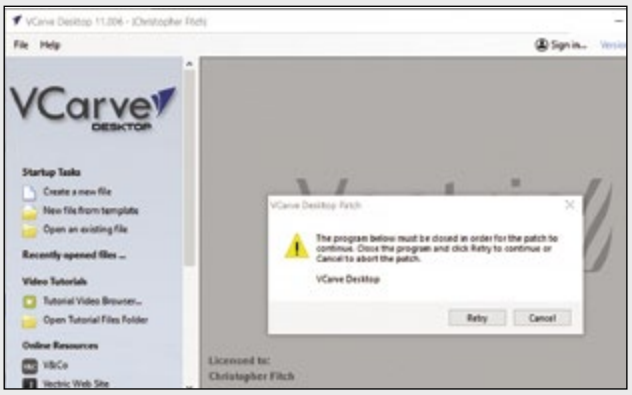
Updating Vectric VCarve *continued*



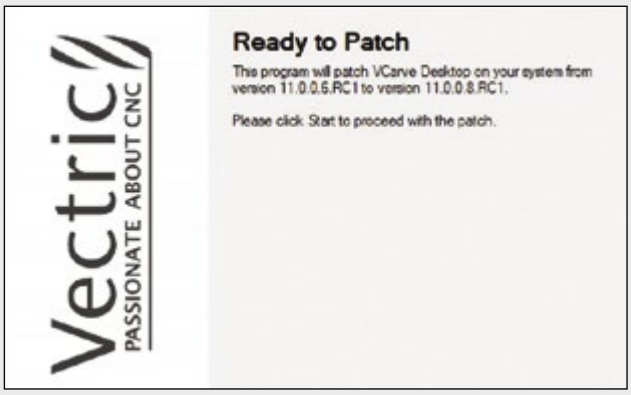
STEP 3. If there is an update for the software that checks for and delivers the **VCarve** updates, it will be installed first. Click *OK* to continue.



STEP 4. Any updates for **VCarve** will be displayed. Click *Next* to install the update.

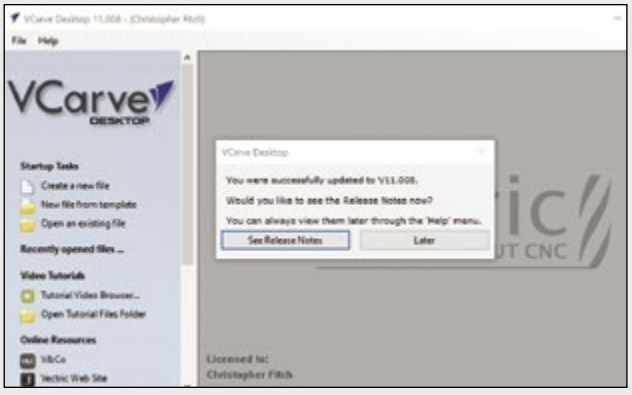


STEP 5. In order to properly install the update, **VCarve** must be shut down. Click *Retry* to continue installing the software update.



STEP 6. This informational dialog will appear. Simply click *Start* to continue installing the software update.

STEP 7. After the software upgrade is installed, press *Finish* to continue using **VCarve**.



STEP 8. If you're interested in all of the features or software bugs that were addressed with the update, click on *See Release Notes*. Otherwise, click *Later*.

A Productive VCarve Workflow

If you cut the projects on pages 26-32, then you've had experience using .tap files, which are also referred to as G-code files or cutting files. They use the information from a design in VCarve and tell your CNC where and how to move. Now it's time to learn how to create your own projects and the .tap files needed to cut them using the powerful tools in VCarve. Let's take a look at how the basic process works.

To create a new design in VCarve, select Create a new file from Startup Tasks (Figure 1). A Job Setup dialog box allows you to set up parameters for the project (Figure 2).

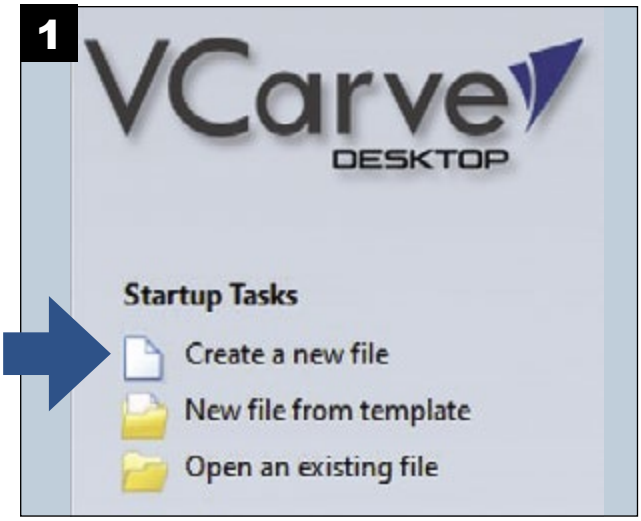
Let's begin by entering the width, length and thickness of the project. Be sure to think about how the workpiece will be secured to your machine, as you may need to make the material larger to accommodate hold downs.

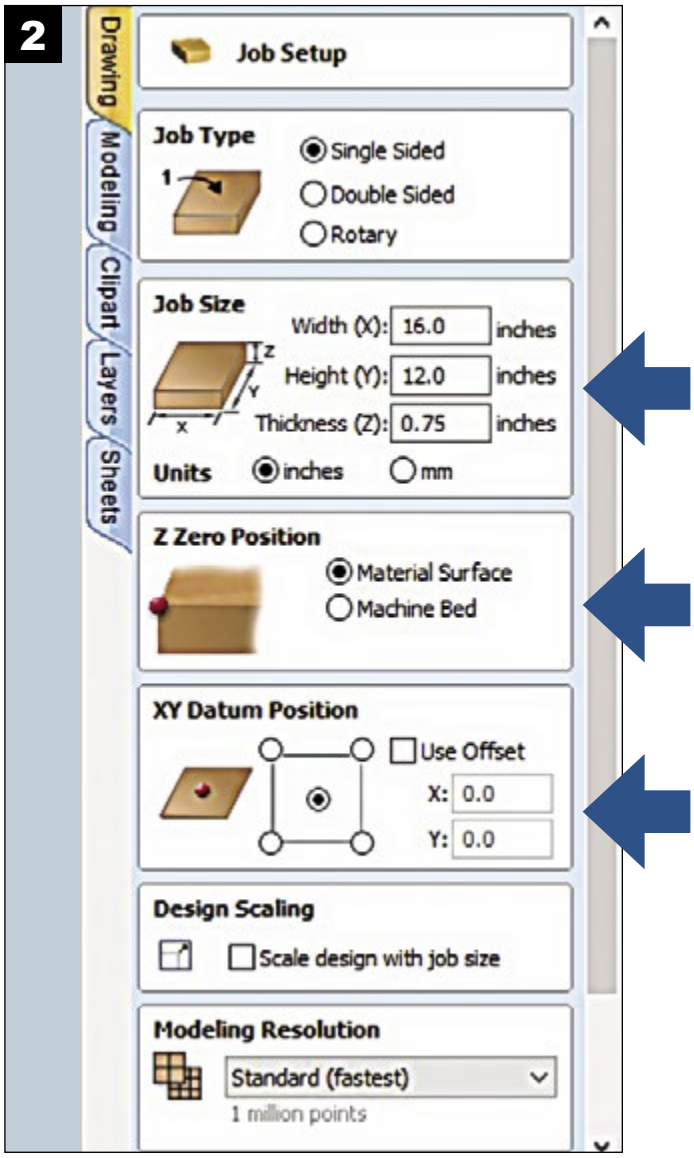
The Z Zero and XY Datum Points are important references to orient the CNC machine correctly to the workpiece. Setting these zero points on your machine are covered on pages 10-11 of this manual. For this project example the Z Zero is on top of the materials surface and the XY Datum Point is in the center of the project. This is how the projects on pages 26-32 were set up.

Click OK to save and exit the Job Setup window.

For videos and tips on getting started with your CNC visit:
www.NextWaveCNC.com/gettingstarted
or scan the QR code below.

SCAN for More Info

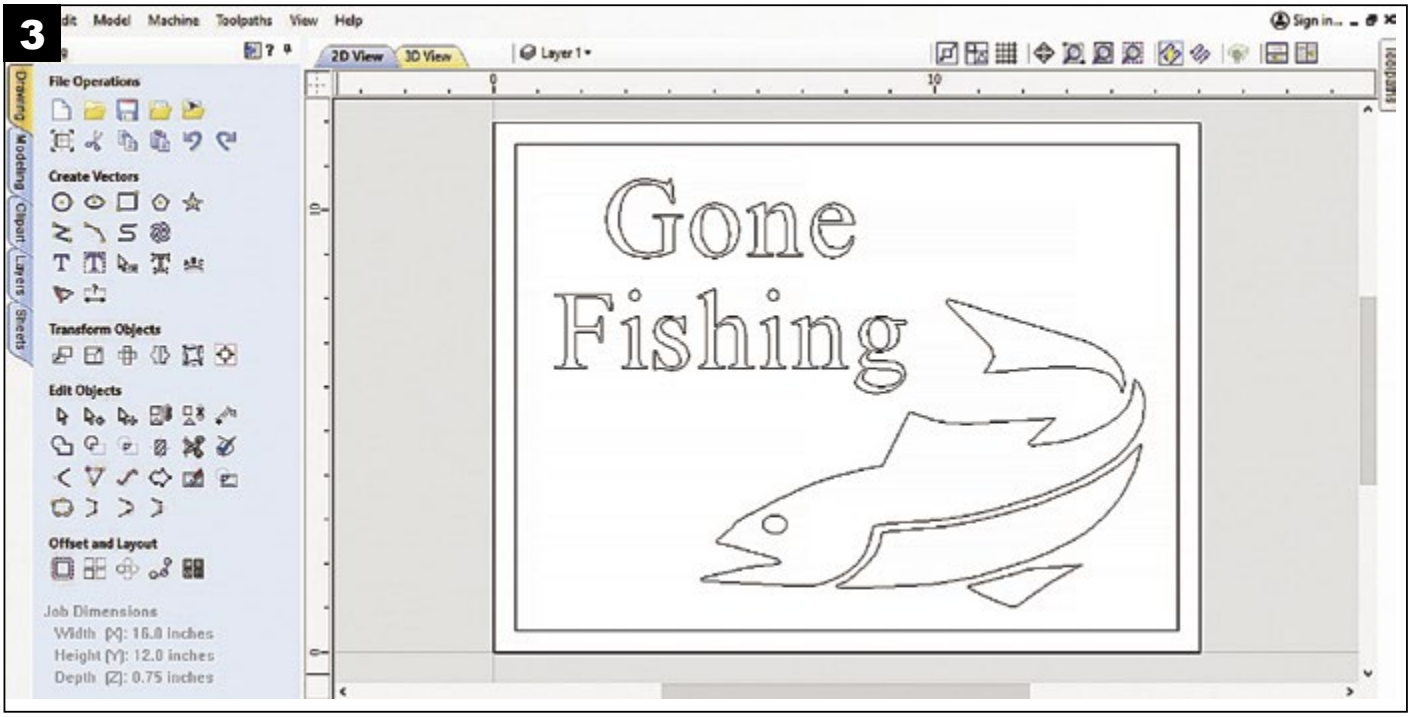




Drawing Tools

Understanding how to use drawing tools is one of the biggest parts of the CNC learning curve, but there are many resources available to help you master VCarve. You can find a complete listing of resources at Vectric.com and NextWaveCNC.com/gettingstarted.

Once you have a basic understanding of the drawing tool groups, dive in and start working with them. If you need a place to start as you learn, one way is to use the clip art that comes with VCarve, or to import a drawing that you can trace over. Add a frame and some text and you've got a project (Figure 3 below).



Toolpaths

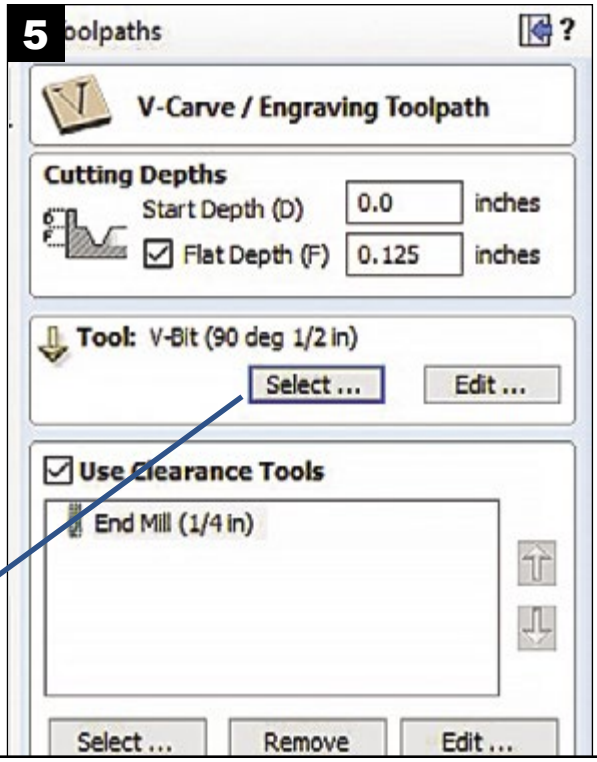
Use the toolpath tools to assign router bits to the lines in your drawing. This process begins by highlighting the lines on the drawing which will become the toolpath (Figure 4 at right), choosing the type of cut (profile, pocket, or v-carve for example, (Figure 5 next page) and selecting the router bit that will be assigned to the cut.

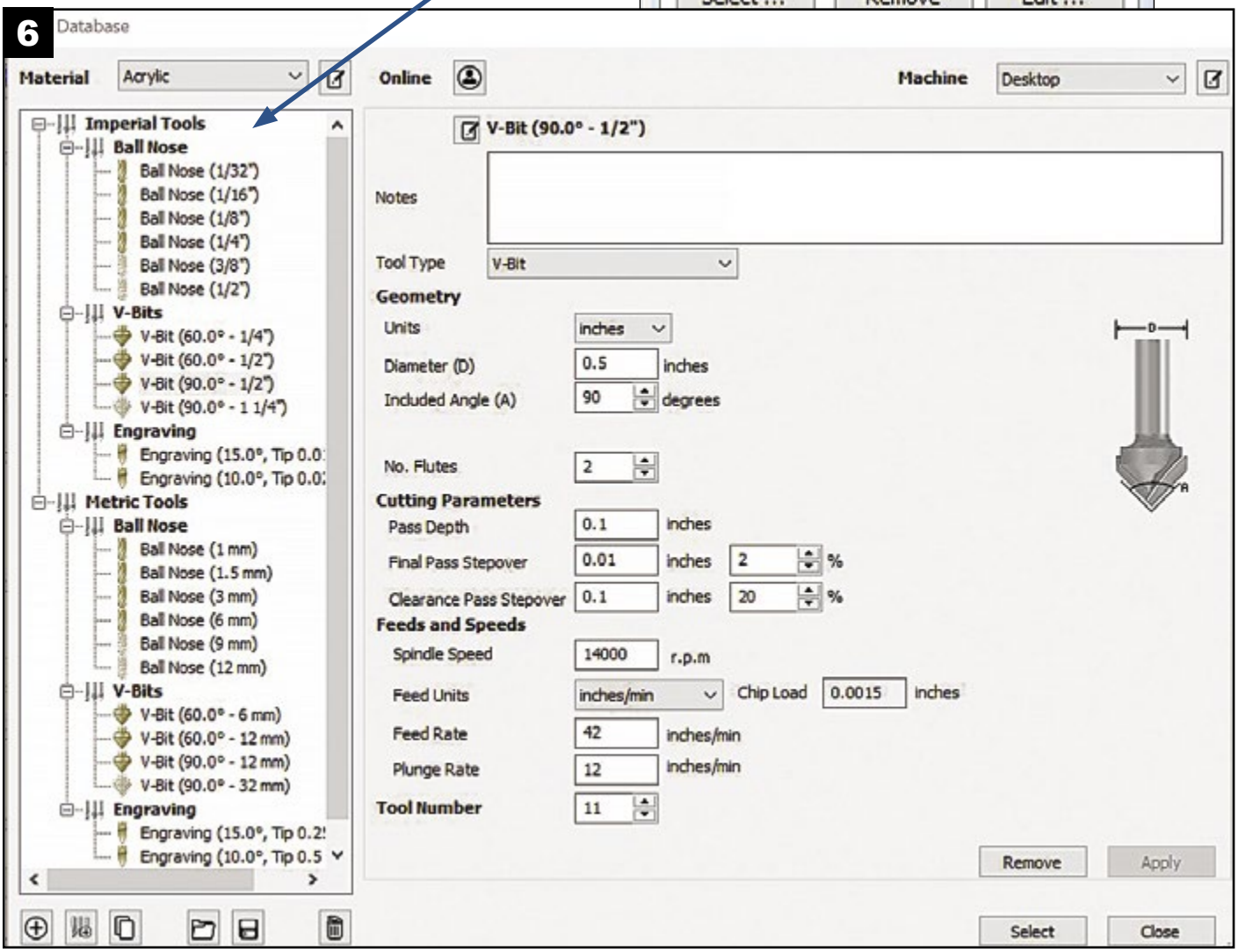
Learning to set up the various parameters for router bits is the other big challenge in learning to use a CNC router. VCarve has a Tool Database with many common router bits listed and default settings for each (Figure 6 next page), however every project is different and may require you to change the parameters.

The settings listed below work well as a starting point for most bits and materials. For some bits and materials, you may want to use more or less aggressive settings than shown below. Practice on spare material to fine tune these numbers to ones that work best for you.

All bits - Pass Depth 0.20 to 0.05 in.
All bits - Feed Rate 50-80 ipm
All bits - Plunge Rate 50-80 ipm
Straight bit stepover 10% - 40%
V-bits - Final pass stepover 2%
V-bit - Clearance pass stepover 20%
Ballnose bit - 3D finishing pass stepover 7-11%

More info at www.NextWaveCNC.com/gettingstarted

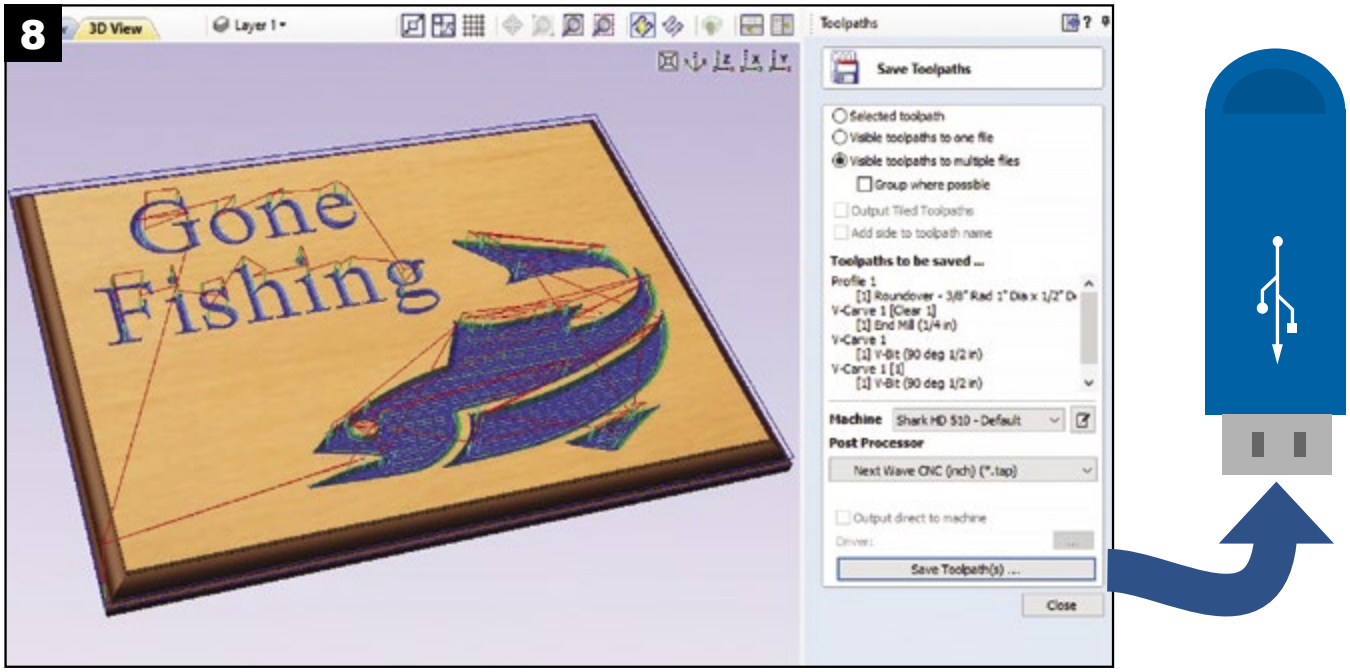




Preview & Save Toolpaths to Thumbdrive

The preview window in VCarve is an important tool in the process to make sure that the toolpaths you have created are cutting where you want them to cut. The window displays the completed project in 3D and allows you to rotate and inspect it (Figure 7 below). If something doesn't look right in the Preview Toolpath window it means that something is probably wrong with one or more of your toolpaths. To correct the problem you may need to adjust the settings in the toolpath or make changes to your drawing in the 2D window.

Check the toolpaths you wish to save and select the post processor that matches your machine. For your Shark CNC router select the Next Wave CNC (*.tap) post processor. The post processor converts the VCarve drawing and toolpath selections into a file that tells your CNC machine where to move. The code file has a .tap extension and is commonly referred to as a .tap, G-code or cutting file. Finish by pressing the Save Toolpath key and saving the file to a thumb drive (Figure 8 below).



Setting Up Your Machine for Cutting

Mark the XY Datum position on your material and secure it to the machine bed of your router. In our example the XY Datum point is specified as the center of the material, but you can choose any of the four corners as well.

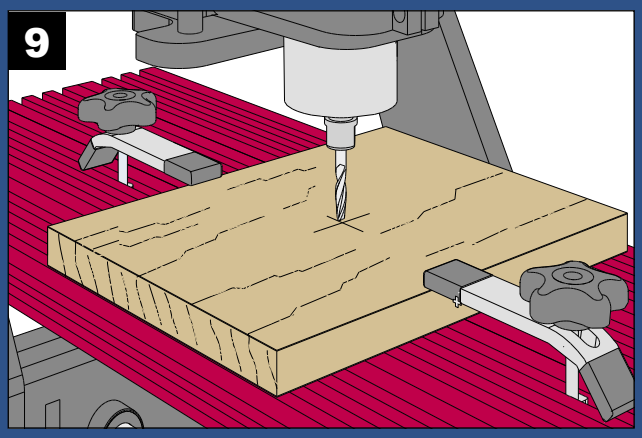
After installing your preferred router bit for the first cut, use the X and Y jog keys in the jog function of the Pendant to move the router close to the XY Datum mark. Change the jog speed to "slow' and position the router bit directly over the mark (Figure 9 below). Press the "Zero XY" key to zero the X- and Y-axes.

To zero the Z-axis, lower the bit to within ½" of the top of the workpiece and set the jog speed to slow. Slide a piece of paper underneath the bit and move it back and forth as you tap the "Dn Z" key to gradually lower the bit. When the bit "grabs" the paper, stop. Press the "Zero Z" key and raise the router bit approximately ½". Now all three axes are calibrated.

Now insert the thumbdrive into the **LCD Pendant** and press the USB key to open the file selection menu. Press on the project .tap file you want to cut (Figures 11 & 12 below).

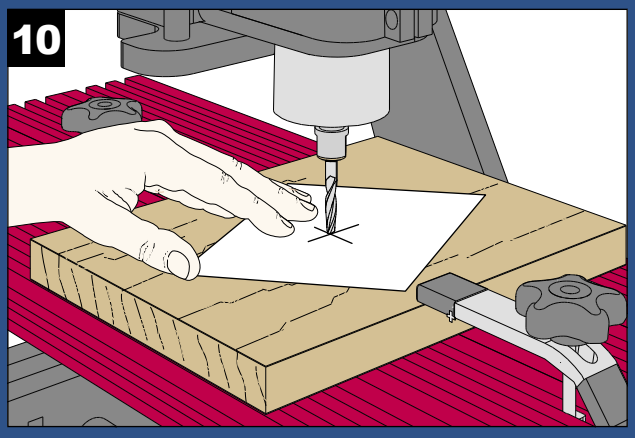
CNC Setup

9



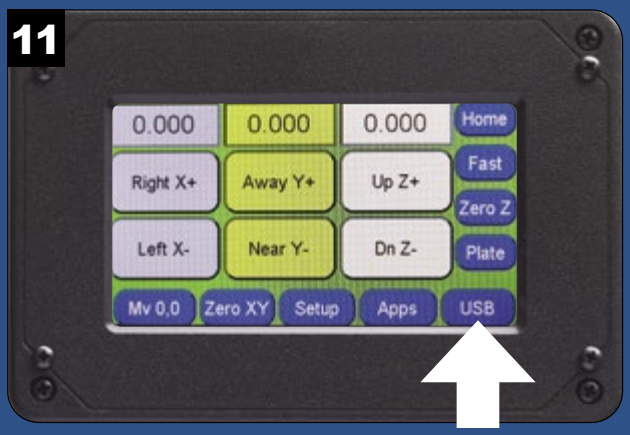
Zero the X and Y axis first by jogging the bit until it's positioned over the center of the workpiece.

10



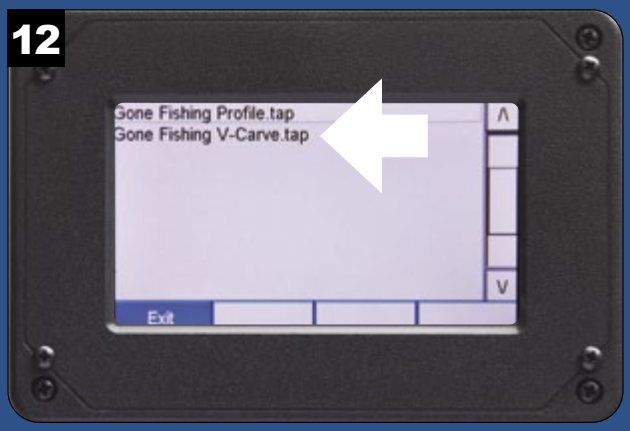
Gradually lower the bit until it just starts to touch a piece of paper placed on the workpiece.

11



After inserting the flash drive into the **LCD Pendant**, press the "USB" button to select the project file.

12



From the list of available projects, choose the Gone Fishing V-Carve.tap file to procede.

If you experience issues with your VCarve software please contact Vectric for help at *support@vectric.com*

Finishing the Job

Preview the settings to make sure you've selected the right .tap file, and that the settings match how you have prepared the VCarve file and your CNC machine (Figure 13). If everything checks out, press Next.

Review the safety check lists from the project pages, turn on your router and press Start. Your **Shark CNC** router will begin cutting the project (Figure 14). Once the first toolpath is cut, change router bits and re-zero the Z-axis. Now return to the **LCD Pendant** main screen and repeat steps 12, 13 and 14.

When all of the .tap files have been run, remove your project, sand and finish.

Review, Then Cut

13

Verify all of the set up information for this job, review the safety checklist, then press "Next".

14

To begin the CNC operation, simply press "Start".

Project 1: Tool Holder

The first project you'll make on your CNC is a practical one: a router bit and wrench organizer, shown at right. This project demonstrates pocket cuts, which don't cut all the way through the material.

The first thing you'll need to do is download the project cutting file by navigating your web browser to *NextWaveCNC.com/cncprojectplans*. Copy the .tap file for the Tool Holder SD project to a USB flash drive. The .tap file contains the G-code that tells your machine how to move.

For this project you will need a piece of solid wood, plywood, or MDF (medium-density fiberboard) 8½" x 11" x ¾" thick. Mark the center of the workpiece to use as a reference for zeroing the X-, Y-, and Z-axes. Using the hold-down clamps that came with your CNC router, securely clamp the workpiece to the table of the CNC, as shown in Figure 1 below.

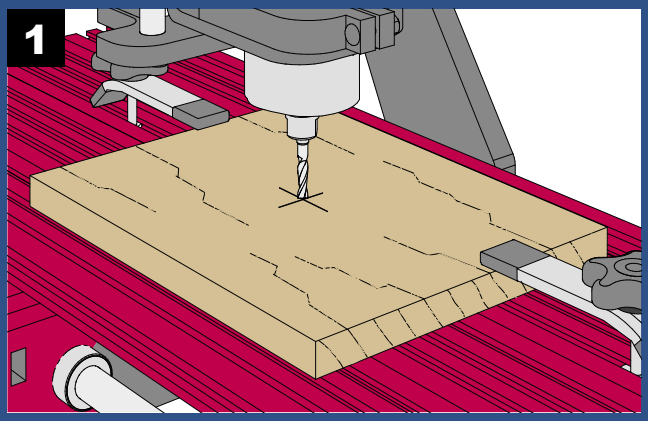
Now you'll install a router bit and zero the X-,Y- and Z-axes, as shown on pages 10 and 11 of this manual. Install a ¼" straight router bit in the collet of your router and set the speed to approximately 90% of full speed. Use the **LCD Pendant** to zero the X- and Y- axes and then zero the Z-axis.

To load the project .tap file that feeds instructions to the CNC, insert the thumb drive into the **LCD Pendant**. Press the "USB" button (Figure 1 next page) to open the file selection window. Select the Tool Holder HD.tap file (Figure 4 next page).

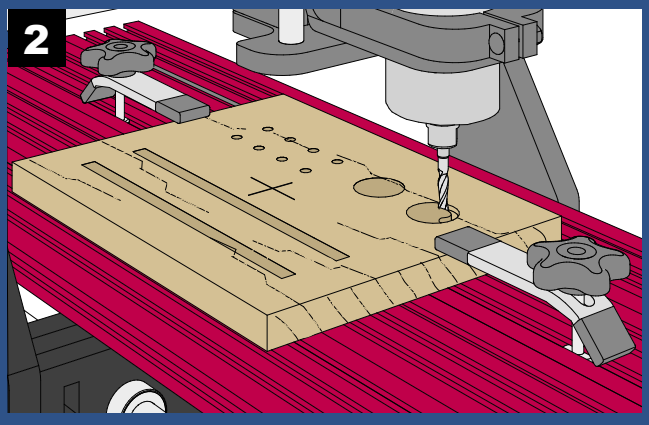
The next screen that pops up (Figure 3) displays set-up information about the file. Once you have reviewed the setup press "Next."

Before going any further, now is a good time to review a safety checklist. You should go through this checklist for every project you make on your CNC.

Workpiece Setup & Routing



Mark a center point on the workpiece before securely clamping it to the CNC table with hold-down clamps. Zero the X-, Y-, and Z-axes before proceeding.



The project file contains the instructions for the CNC to make all of the pocket cuts in the workpiece.

- ☑ Is the workpiece securely clamped to the table?
- ☑ Have you zeroed the X- and Y- axes to the center of the workpiece?
- ☑ Have you zeroed the Z-axis to the top surface of the workpiece?
- ☑ Is the router or set to approximately 90% of full speed?
- ☑ Have you reviewed the emergency stop procedures?
- ☑ Are you wearing hearing and eye protection?

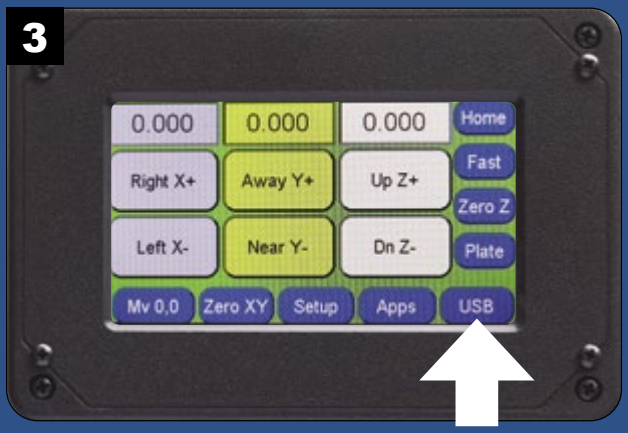
Press “Start” and watch your CNC router create your first project, as shown in the lower right drawing, previous page.

NOTE: You can press the pause button on the **LCD Pendant** screen to stop the movement of the router if you need to make a minor adjustment. You must also turn off the router or spindle to avoid injury. Once you’ve made all of the adjustments needed, restart the router and press “Resume.”

Once the project is routed, power off the router and remove the workpiece. All that’s left is to sand and apply a finish to the organizer. Now you’ve got a convenient storage place for the router collets, wrenches, and bits.

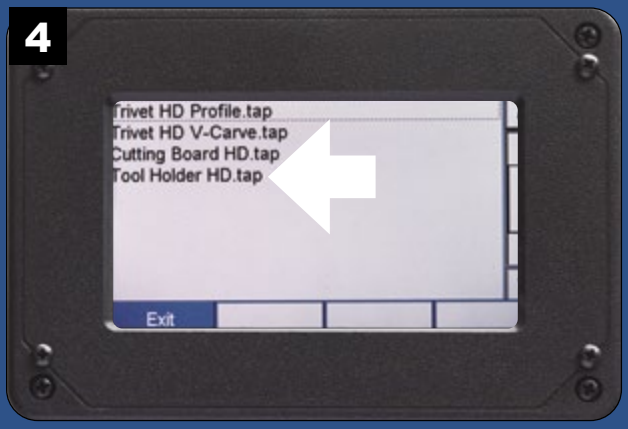
Verify, Then Create!

3



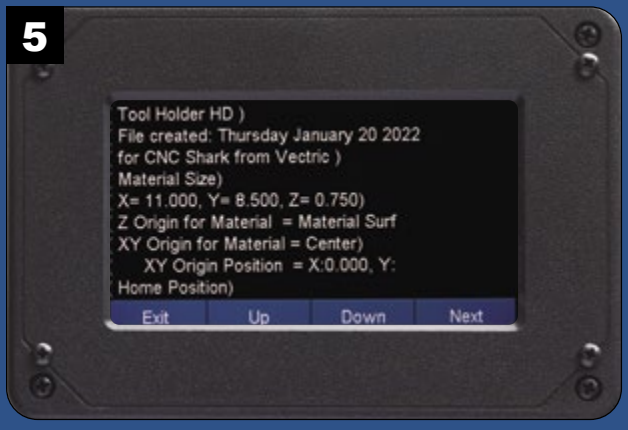
After inserting the flash drive into the **LCD Pendant**, press the “USB” button to select the project file.

4



From the list of available projects, choose the Tool Holder HD.tap file to proceed.

5



Verify all of the setup information for this job, review the safety checklist, then press “Next.”

6

To begin the CNC operation, simply press “Start.”

Project 2: Cutting Board

This project will find a practical use in your kitchen. The cutting board demonstrates a “through profile” cut where the finished piece is separated from the parent material. Think of it like a bandsaw.

You may be wondering how the cutting board can remain secure on the table if the router cuts the complete profile around it. There’s a trick CNC users employ called tabs that keep the project connected to the rest of the board during cutting. .

Tabs are small, connecting pieces of material that span the kerf left by the router bit. To create these tabs, the CNC doesn’t cut all the way through the thickness of the material in certain areas. After the the job is done, you can cut through the tabs with a saw or chisel, then sand the profile smooth.

Begin by going downloading the Cutting Board SD.tap file from *NextWaveCNC.com/CNCprojectplans*. Copy the file onto a USB flash drive.

Attractive, tight-grained hardwoods are best for a cutting board. Prepare a blank 7½” x 11” x ¾”-thick. Now is a good time to mark the center point of the workpiece for zeroing out the axes.

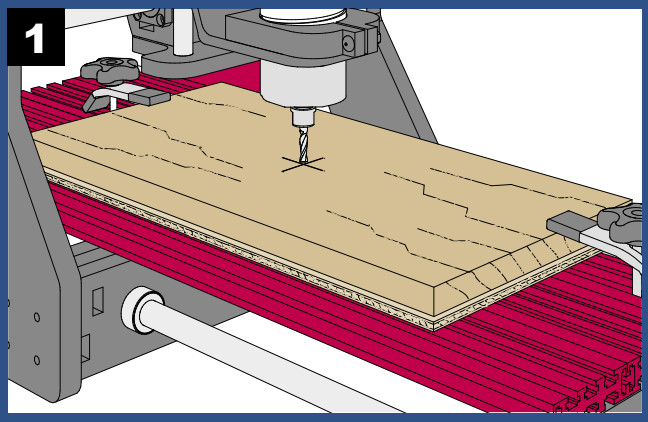
Since the CNC cuts through the entire thickness of the blank, you need to place a waste board underneath the blank. This prevents the router bit from cutting into the CNC table. Use a waste board that’s at least a ¼” thick and the same size or larger than the cutting board blank.

Place the waste board under the workpiece blank and clamp them to the CNC table using hold-down clamps. Install a ¼” straight bit in the collet of your router and set the speed to approximatley 90%. Zero the X-, Y-, and Z-axes.

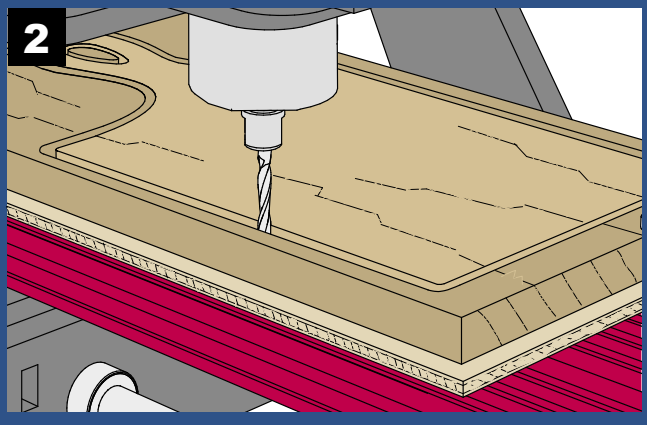
Insert the thumb drive into the Pendant to load the cutting file. Press the “USB” button (Figure 1) on the main screen to open the file selection window. Locate and select the Cutting Board HD.tap file, as shown in Figure 4 at right.

As shown in Figure 5, setup information for the job pops up for your review and verification. If everything looks okay, press “Next.”

Workpiece Set -Up



Mark a center point on the workpiece before securely clamping it and the waste board to the CNC table with hold-down clamps. Zero the X, Y, and Z axes.



Watch the router cut around the profile of the cutting board while leaving tabs to keep it secure on the CNC table.

Review your safety checklist:

- ☑ Is the workpiece securely clamped to the table?
- ☑ Have you zeroed the X- and Y- axes to the center of the workpiece?
- ☑ Have you zeroed the Z-axis to the top surface of the workpiece?
- ☑ Is the router set to approximately 90% of full speed?
- ☑ Have you reviewed the emergency stop procedures?
- ☑ Are you wearing hearing and eye protection?

Press “Start” and watch your CNC router cut the profile of the cutting board, as illustrated in the lower right drawing, previous page. Note the tabs created during the routing process that hold the cutting board in place.

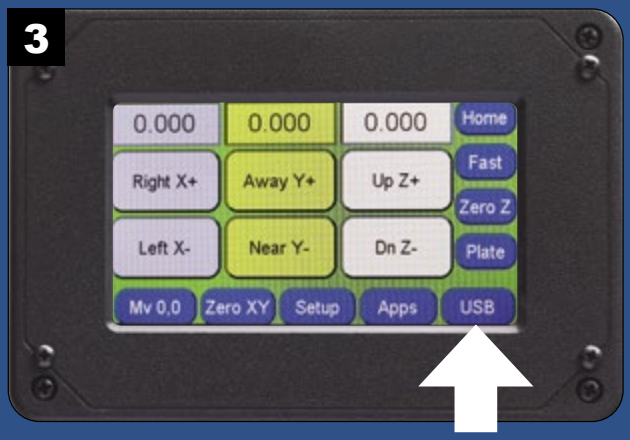
NOTE: You can press the pause button on the Pendant screen to stop the movement of the router if you need to make a minor adjustment. You must also turn off the router or spindle to avoid injury. Once you’ve made all of the adjustments needed, restart the router or spindle and press “Resume.”

When the CNC stops, power down the router before removing the workpiece and waste board. Use a chisel or small hand saw to cut through the tabs to separate the cutting board. Clean up the remnants of the tabs with rasps and sandpaper. Sand the entire cutting board smooth before applying a food-safe finish.

As a word of caution, you can damage your cutting board by placing it in the dishwasher. Instead, simply wipe it clean with a damp cloth and mild detergent between uses.

Verify, Then Create!

3

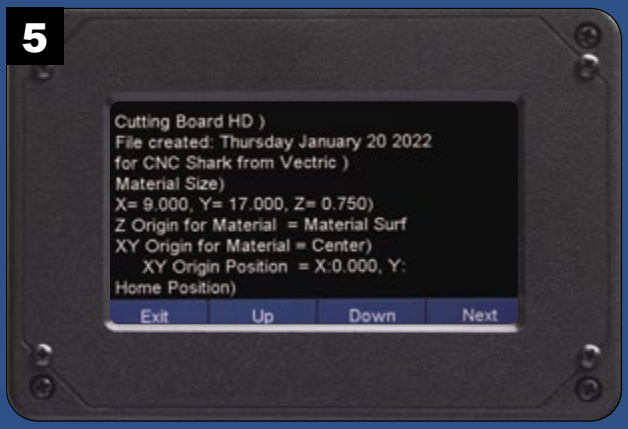


After inserting the flash drive into the **LCD Pendant**, press the “USB” button to select the project file.

4

From the list of available projects, choose the Cutting Board.tap file to proceed.

5



Verify all of the setup information for this job, review the safety checklist, then press “Next”.

6

To begin the CNC operation, simply press “Start”.

Project 3: Carved Trivet

This project, a trivet, features a technique that you may not be familiar with: V-carving. This versatile cutting method uses a V-shaped router bit to generate appealing designs by using cuts of varied widths. You can see these intricate cuts in the photo at right.

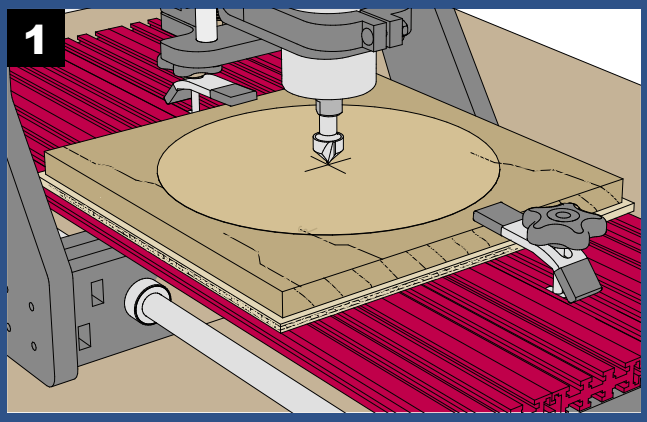
During operation, the V-bit gradually rises and lowers as the Z-axis moves in sync with the X- and Y-axes. This creates a tapered look unique to V-carving. With this technique, you can create beautiful lettering and amazing patterns.

The trivet is made with two different cutting techniques. First, you'll create the decorative pattern and lettering using a V-carving cut and a 90 degree V-bit. The second operation is a profile cut that creates the round shape of the trivet in the same way that you made the cutting board, using a ¼" straight bit, a waste board, and utilizing tabs to keep the project from moving. When different router bits or types of cuts are used for a project, a separate cutting file is required for each.

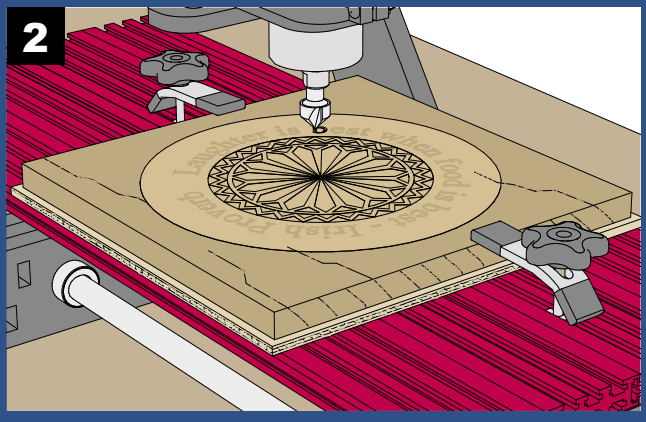
Get started by downloading the project cutting files from *NextWaveCNC.com/cncprojectplans.* Copy the TAP files onto a USB flash drive.

For this project, you will need a piece of solid wood that measures 11" x 11" x ¾" thick. Hardwoods like maple or cherry work best. You will also need a waste board to place under the trivet material. Mark the center of the blank, then clamp the blank with the waste board underneath to the CNC table using the hold-down clamps, as shown in Figure 1. Install a 90° V-bit bit in the router collet and set the speed to about 90% of maximum.

Workpiece Setup



Mark a center point on the workpiece before securely clamping it to the CNC table with hold-down clamps. Zero the X-, Y-, and Z-axes.



The V-bit makes unique cuts that create an eye-catching pattern in three dimensions.

Once you've zeroed the X-, Y-, and Z-axes, insert the thumb drive into the **LCD Pendant**. To select the file for processing, press the "USB" key (Figure 3) to open the file selection window. Select the file Trivet Vcarve HD.tap as shown in Figure 4.

Figure 5 represents the setup information about the file that you should review and verify. If everything checks out, press the "Next" button.

Review your safety checklist:

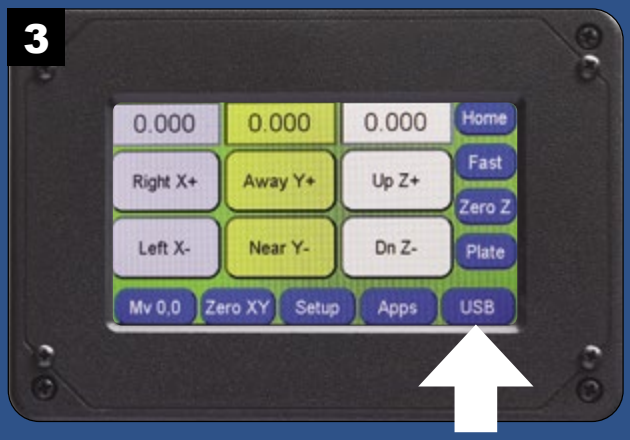
- ☑ Is the workpiece securely clamped to the table?
- ☑ Have you zeroed the X- and Y- axes to the center of the workpiece?
- ☑ Have you zeroed the Z-axis to the top surface of the workpiece?
- ☑ Is the router set to approximately 90% of full speed?
- ☑ Have you reviewed the emergency stop procedures?
- ☑ Are you wearing hearing and eye protection?

Press "Start" and watch your CNC router carve an intricate pattern on the workpiece.

NOTE: You can press the pause button on the **LCD Pendant** screen to stop the movement of the router if you need to make a minor adjustment. You must also turn off the router to avoid injury. Once you've made all of the adjustments needed, restart the router and press "Resume."

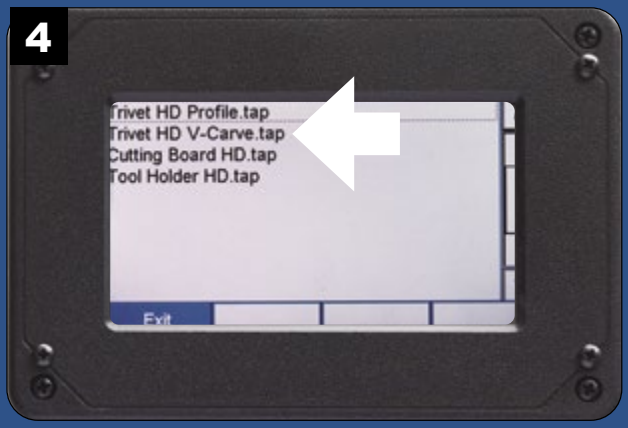
Verify, Then Create!

3



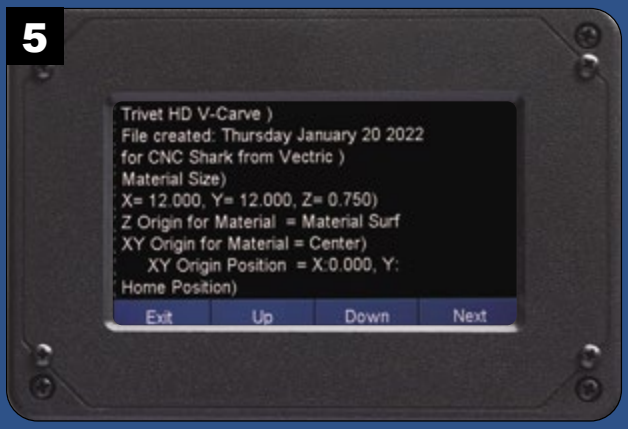
After inserting the flash drive into the **LCD Pendant**, press the "USB" button to select the project file.

4



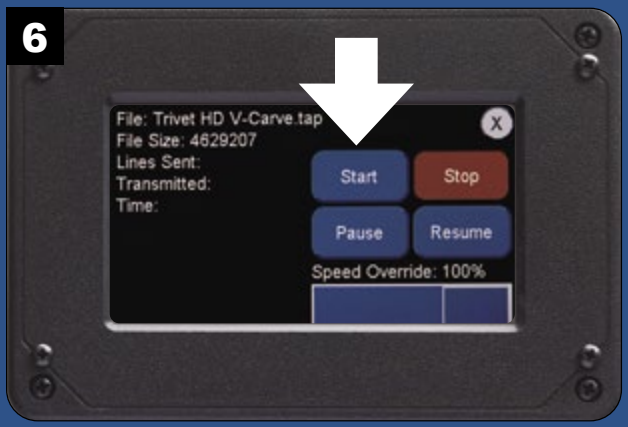
From the list of available projects, choose the Trivet HD V-Carve.tap file to proceed.

5



Verify all of the setup information for this job, review the safety checklist, then press "Next".

6

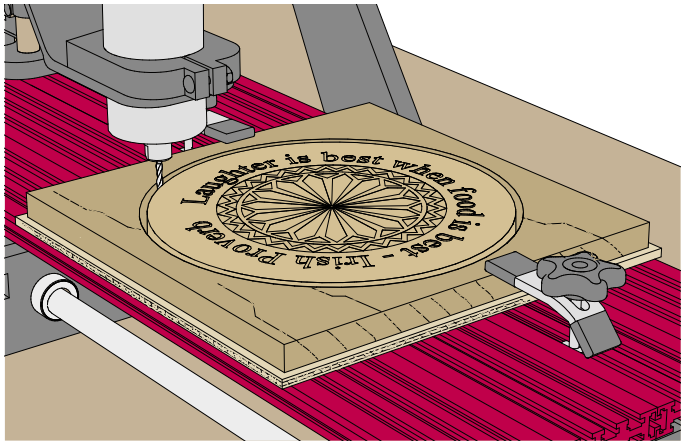


To begin the CNC operation, simply press "Start".

When the CNC stops, power down the router before swapping out the V-bit to a ¼" shank straight bit. Once you've re-zeroed Z- axis, select the new file for processing by pressing the "USB" key (Figure 7) to open the file selection window. Select the file Trivet Profile HD.tap as shown in Figure 8.

Review new setup information (Figure 9), then press "Next." Press "Start" (Figure 10) to cut the profile of the trivet, as illustrated in the drawing at right, Note the tabs created during the routing process that hold the Trivet in place.

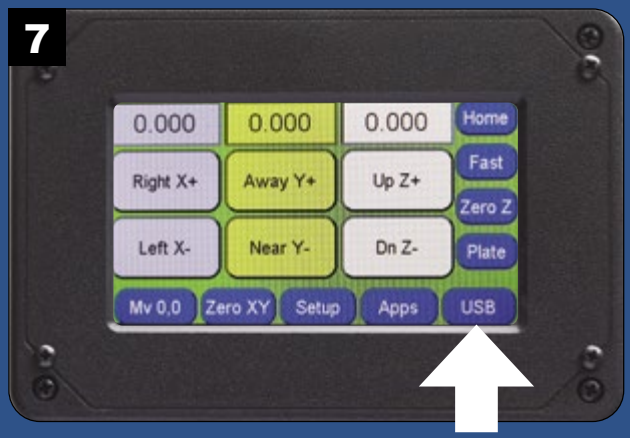
When the CNC stops, power down the router before removing the workpiece and waste board. Use a chisel or small hand saw to cut through the tabs to separate the trivet. Clean up the remnants of the tabs with rasps and sandpaper. Then sand and finish.



Watch the router cut around the profile of the trivet while leaving tabs to keep it secure on the CNC table.

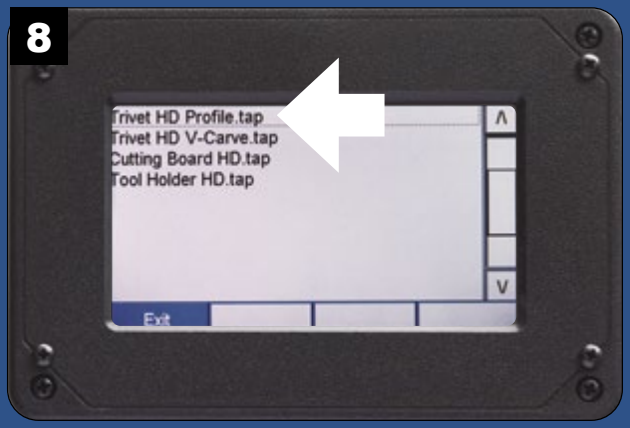
Finish the Trivet

7



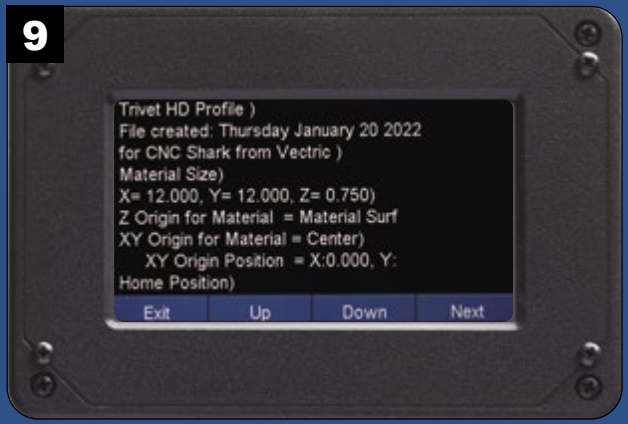
After inserting the flash drive into the **LCD Pendant**, press the "USB" button to select the project file.

8



From the list of available projects, choose the Trivet Profile HD.tap file to proceed.

9



Verify all of the setup information for this job, review the safety checklist, then press "Next".

10

To begin the CNC operation, simply press "Start".

Maintenance

Periodic maintenance and lubrication keeps your **SHARK CNC** running efficiently and smoothly.

If the **LCD Pendant** buttons stop responding correctly,- clean around the screen with a piece of paper (see photo at right).

Use a dry lubricant for the bearings, round bars, and lead-screws. Dry lubricant doesn't attract dust. Purchase Du-pont Silicon Teflon at your local **Next Wave CNC** retailer or online at: NextWaveCNC.com/shop. You will find similar dry lubricant products at your local hardware store.

Daily

- Check for damaged wires or components. Repair or replace them as needed.
- Check for loose parts. Tighten or adjust as needed.
- Vacuum or dust off machine and components, includ-ing the controller and router.
- Wipe down the bars with a soft cloth.
- Clean the leadscrews with a soft brush or vacuum.
- Apply dry lubricant to the guide rods and leadscrews, wiping off the excess with a soft rag.

Monthly or Every 40 Hours of Use

- Deep clean the guide rails and leadscrews with mineral spirits or other mild solvent to remove wood resin, dust and dirt.
- Apply a light coat of dry lubricant to the guide rails and leadscrews. Wipe off excess with a soft rag.

Guide Rail & Bearing Maintenance

X, Y and Z Rails

Dust and debris build-up on the guide rails can have a negative impact on the performance of your machine, so it is important to keep them clean and lubricated. To maintain optimum performance, clean and lube the X, Y and Z rails at least once every 8 hours of operation. While the machine is stopped, wipe off the rails with a clean cloth or soft brush. After cleaning the rails, lubricate the bearings with a dry lubricant, such as CRC Ultra-LIte 3-26 (available at www.NextWaveCNC.com), Dupont Non-Stick Dry-Film Lubricant or Blaster Dry Lube.

Wipe the **LCD Pendant** screen with a soft cloth and remove sawdust buildup between the case and screen with a folded piece of paper.

Resources

Next Wave CNC
NextWaveCNC.com
Manufacturer of the **SHARK HD & SD110/SD120 CNC** machines and accessories. The website offers software and documentation downloads, educational content, as well as information about the full line of **Next Wave CNC** products.

Next Wave CNC Technical Support
For questions about your SHARK, Ready2Control soft-ware, or other Next Wave CNC products, please contact our support team at support@NextWaveCNC.com

NWA Owners Only Forum
forums.NextWaveCNC.com/
Join our group exclusively for Next Wave CNC owners. You'll find projects, ideas, project showcases, and valu-able support from users of all levels.

CNC SHARK Talk User Forum
www.cncSHARK talk.com
This discussion group is a valuable resource open to users of any CNC machine.

Vectric
vectric.com
Vectric produces VCarve and other CNC design software. Here, you'll find product information, FAQs, on-line tutori-als, and an excellent user forum. Vectric does not provide technical support for **Next Wave** products.

SCAN for More Info

For videos and tips on getting started with your CNC visit:
www.NextWaveCNC.com/gettingstarted
or scan the QR code at left.

Upgrade Accessories

Authentic SHARK Accessories

STANDARD 2 WATT SOLID STATE LASER

- Compatible with any of our CNCs
- Powerful enough to engrave just about any species of wood
- Cuts fabric and thin veneers
- Works with Vectric VCarve and Aspire software programs
- Free raster program allows quick conversion of photos to laser art

SKU 20024

NEXT WAVE CNC

Authentic SHARK Accessories

TOUCH PLATE

- Compatible with all Next Wave CNC machines.
- Required for using Virtual Zero Unlimited available on select machines
- Solid Aluminum Construction
- 4 ft Cord (6’ extension cable also available)
- Sets Z axis at exact zero for precision cutting
- Lays flat on the material surface or machine bed

SKU 20141

NEXT WAVE CNC

Upgrade Accessories

SIX PIECE STARTER BIT SET

We've put together a set of the most commonly used bits to help you get started carving. This 6-piece Starter Bit Set will help you complete any common projects right from the get-go. All have 1/4" shanks.

- 1/16" x 1.0" - 5.5 degree tapered solid carbide four fluted ball nose bit
- 1/8" x 1.0" - 3.8 degree tapered solid carbide four fluted ball nose bit
- 1/4" x 1.0" - 0 degree straight solid carbide four fluted ball nose bit
- 1/4" x 1.0" straight solid carbide two fluted up-cut bit
- 3/8" x 1/2" 60-degree carbide-tipped v-groove bit
- 1/2" X 1/2" 90-degree carbide-tipped v-groove bit

SKU 20119

BIT-SAVING HOLD DOWN CLAMPS

These plastic hold-down clamps are less likely to damage your expensive bits if you accidentally run into them. They feature the same functionality as our aluminum hold-down clamps and are specifically designed with three adjustable pivot points that allow adjustment to the thickness of the material.

SKU 200197

NEXT WAVE® CNC

SHARK®
World Class. Made in USA.

SD110/SD120
User's Manual

Version: 1/30/2023
PDF web version

205025