NEXT WAVE® CNC

SHARK® HD500/510/520

Owners Manual

To Our Customers

Thank you for purchasing a *SHARK HD CNC* 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 CNC will be up and running in no time.

Whether you're new to CNC technology or a seasoned expert, your *SHARK HD CNC* provides a lot of flexibility for programming and operation. Our specially designed *LCD Pendant* is an auxiliary touchscreen that provides hands-on control of many of the functions of the *SHARK CNC* right at the machine, without the need for a connection to a personal computer. The *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 HD* CNC 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 is a world-wide standard for designing parts and programming the toolpaths for your *SHARK HD CNC* using an intuitive interface. *V-Carve* 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 CNCs* 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-A port.

Vectric VCarve 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.

HD500

HD510

HD520
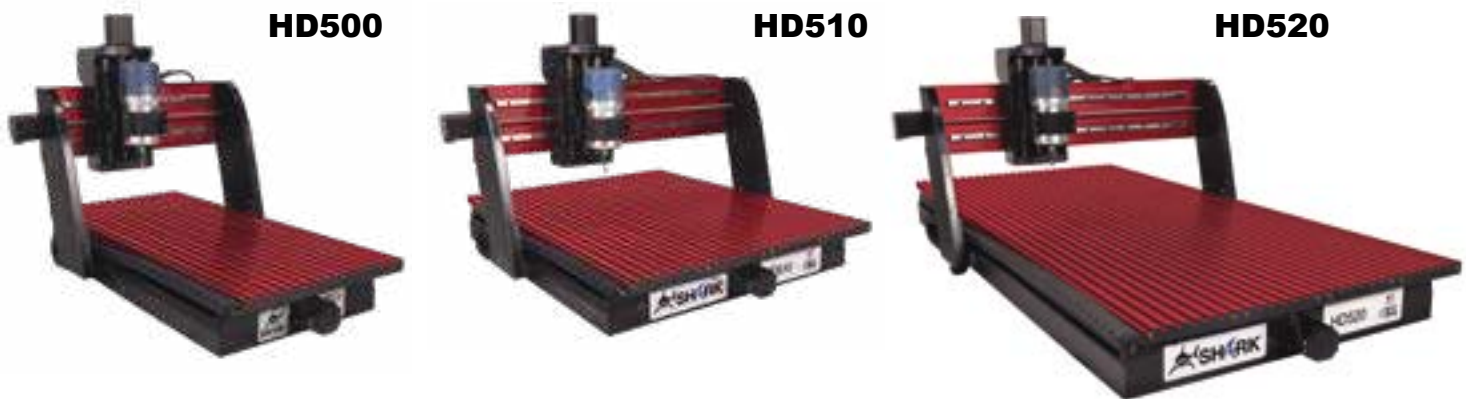
Please Read This Manual Carefully

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

Next Wave CNC warrants your new *SHARK HD series CNC* 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 HD series CNC* 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 HD CNC* 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 HD series CNC* 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 HD CNC* 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 HD CNC* .

Ready2Control software Next Wave CNC warrants *Ready2Contol* 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 *VCarve* design software furnished with your *Shark CNC* 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*

Contact Us

If you need technical assistance with your *SHARK HD CNC* 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 HD* and *Vectric* information at right. To locate the information, refer to the pages below.

Controller Box Serial Number

_____

LCD Pendant Serial Number

(page 13)_____

LCD Pendant Unlock Code

(page 13)_____

VCarve License Code

(page 21)_____

Your VCarve User's Name

(page 22)_____

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
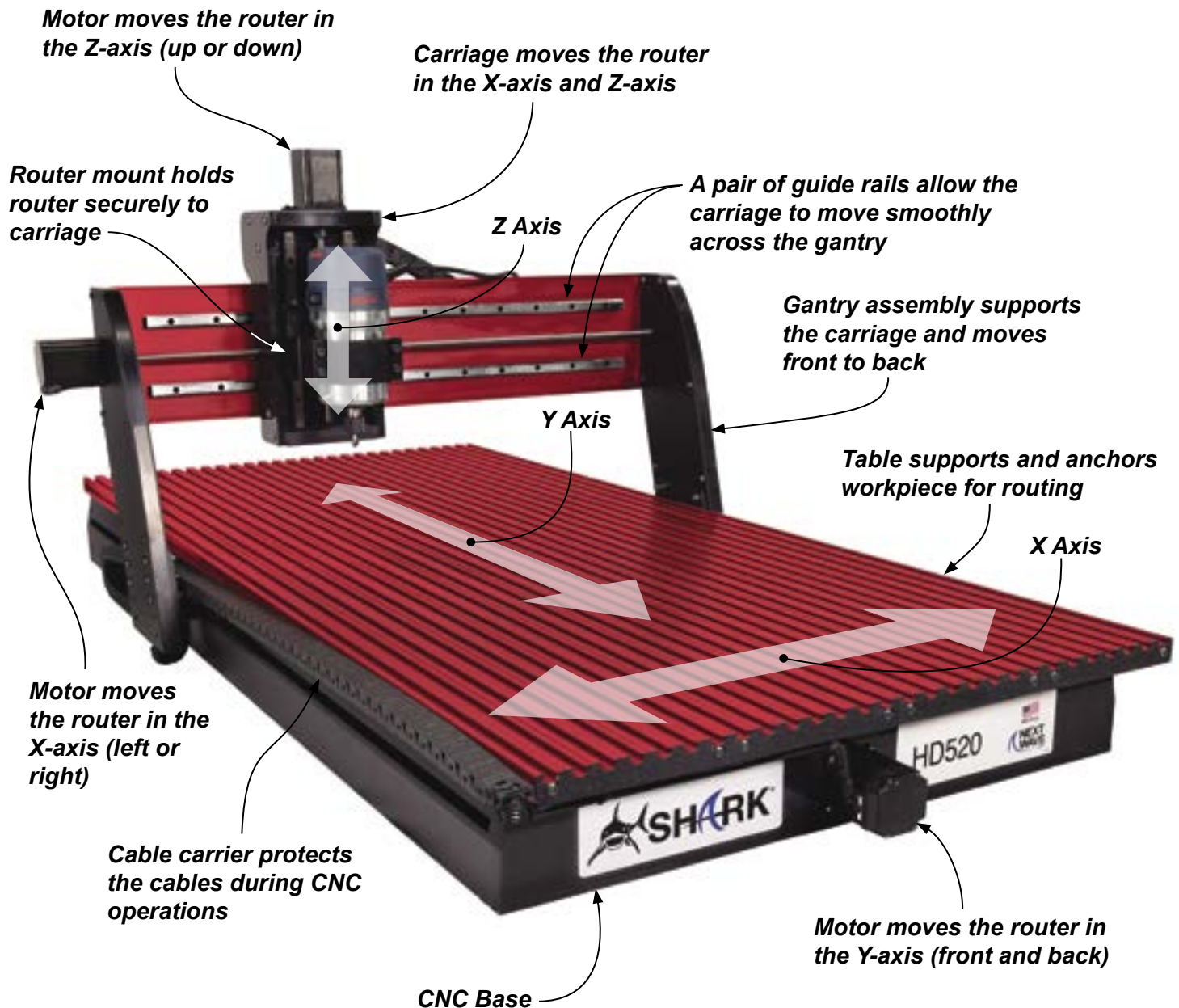info@nextwaveCNC.com

CONTENTS

Machine Overview

Before assembling and using your *SHARK CNC*, we strongly recommend viewing the series of assembly videos with detailed instructions at *https://www.nextwavecnc.com/support.*

Determine where in your shop it will be located. Assembling your CNC at or near this area will save some hassle later if the CNC needs moved to its final home.

See the Assembly Overview illustration below to get familiar with all of the parts of your CNC. You'll find a variety of videos covering the assembly of your *Shark CNC Router* online at *https://www.nextwavecnc.com/video.*

As you are aware, many components of your CNC are heavy. To make assembling and moving the CNC easier, it's best to have the assistance of a friend or coworker.

Motor moves the router in the Z-axis (up or down)

Carriage moves the router in the X-axis and Z-axis

Router mount holds router securely to carriage

Z Axis

A pair of guide rails allow the carriage to move smoothly across the gantry

Gantry assembly supports the carriage and moves front to back

Y Axis

Table supports and anchors workpiece for routing

X Axis

Motor moves the router in the X-axis (left or right)

Cable carrier protects the cables during CNC operations

CNC Base

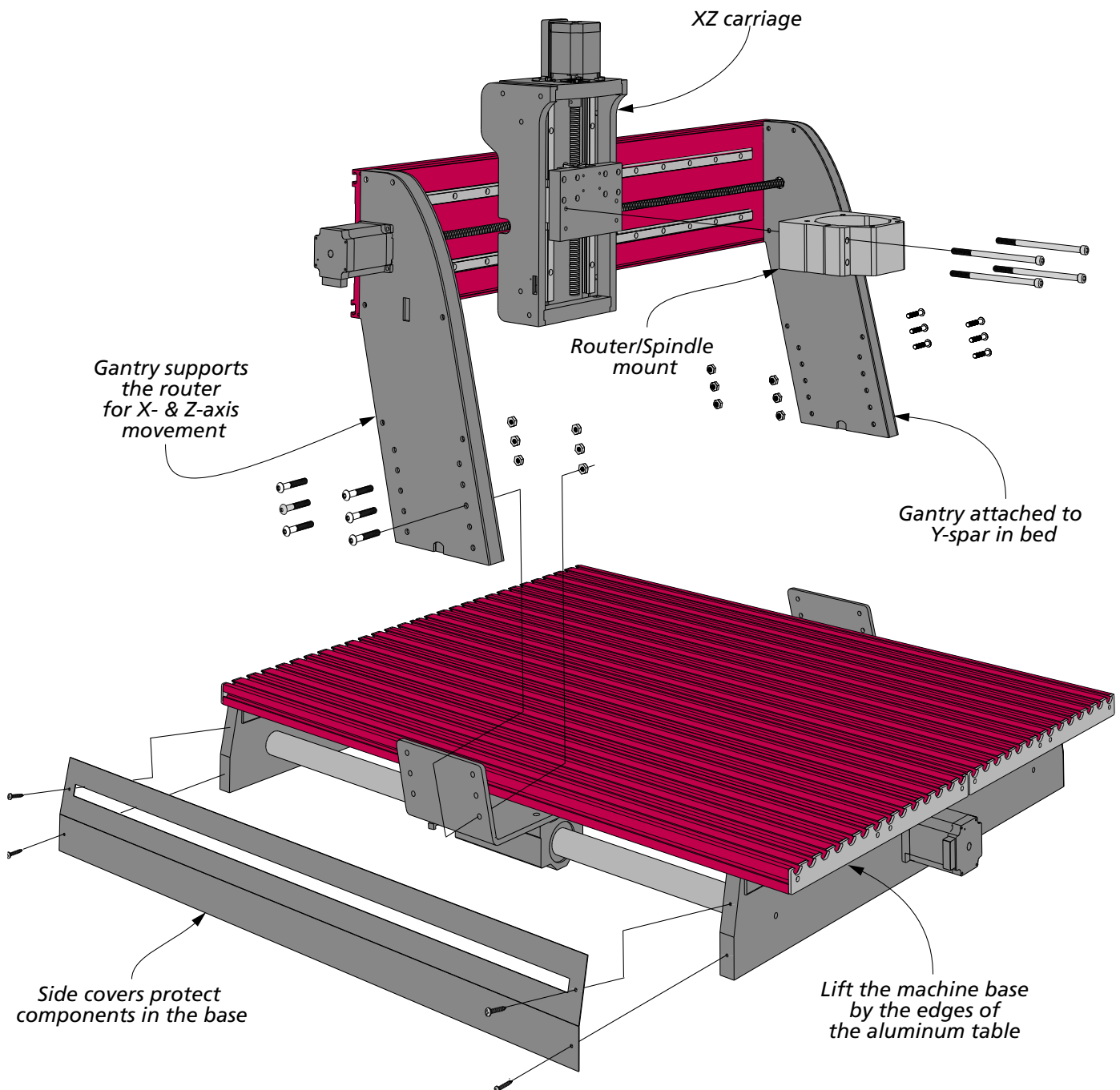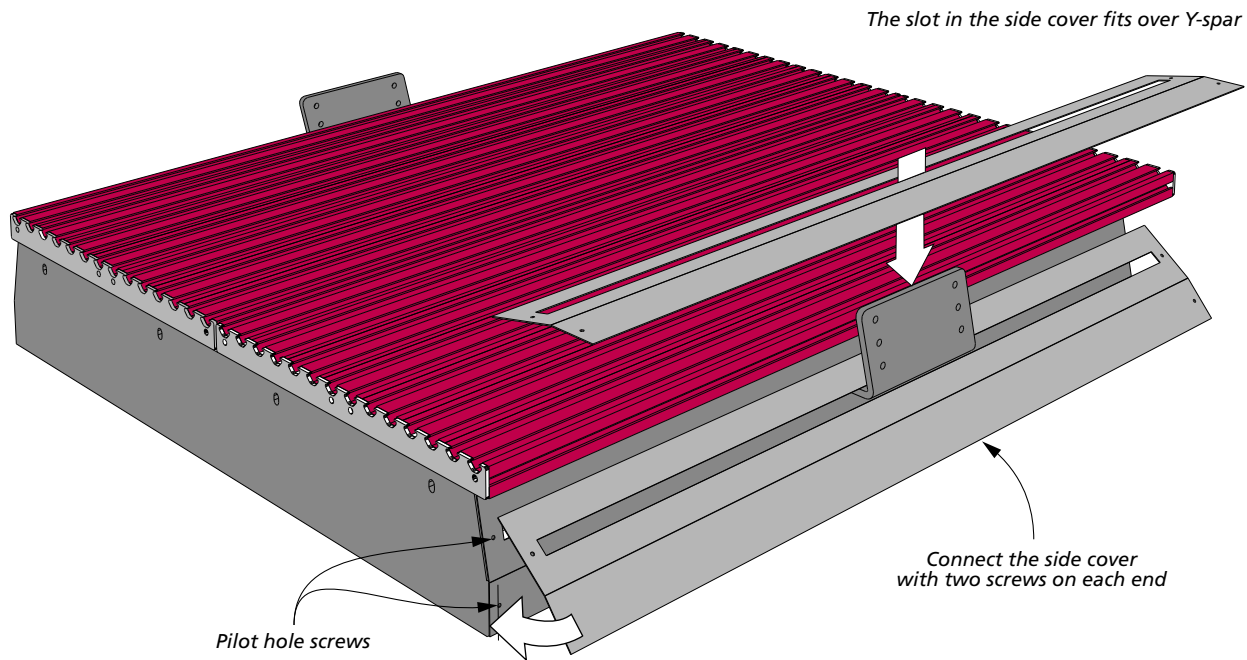*Motor moves the router in the Y-axis (front and back)*

HD520

SHARK

Assembly Overview

The foundation of your CNC machine is the base. It is specifically engineered to support the other main components. The top of the base also provides a solid, even surface for securing the workpiece before routing or carving.

The gantry is a bridge-like assembly attached to the base. The gantry supports and moves the router. The entire gantry moves front to back on rails in the base while the router rides on a carriage that moves left and right. Your CNC operates on three axes: X, Y, and Z. Standing at the front of your machine, the X-axis runs left and right. The Y-axis runs front-to-back. The Z-axis runs vertically from the table of the CNC. Understanding these basic elements will go a long way toward helping you understand how to design and make projects with your CNC.
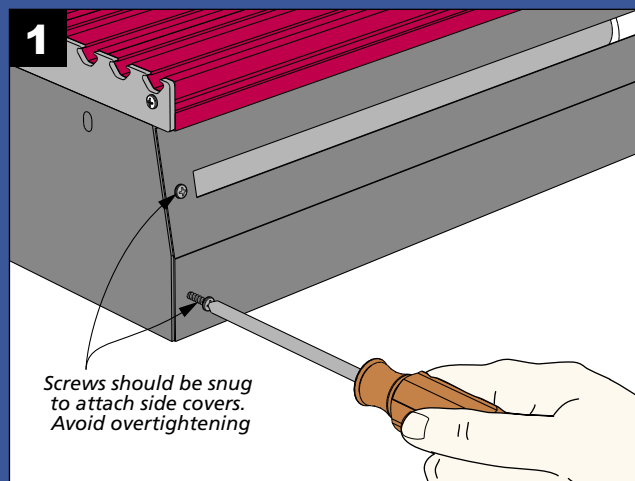
XZ carriage

*Gantry supports
the router
for X- & Z-axis
movement*

*Router/Spindle
mount*

*Gantry attached to
Y-spar in bed*

*Side covers protect
components in the base*

*Lift the machine base
by the edges of
the aluminum table*

*Connect the side cover
with two screws on each end*

Pilot hole screws

Adding the Gantry

SIDE COVERS. A Y-spar bracket extends from each side of the CNC. The Y-spar moves the gantry in the Y-axis. As shown in the illustration above, slip the slot of the side covers over the Y-spar brackets. Attach the covers to the base with four screws, being sure not to overtighten them (Figure 1).
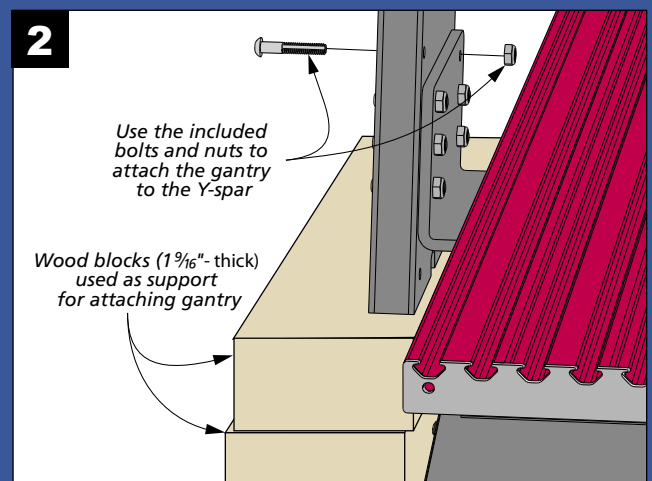
GANTRY. As mentioned previously, the gantry supports the router or spindle, moving it in the X and Z directions.

The gantry assembly is heavy, so you'll want to enlist some help. Also, positioning support blocks under each side of the gantry makes the job easier, as illustrated in Figure 2. Attach the gantry to the Y-spars with six bolts and lock nuts on each side (Figure 2). Use the top six holes in the gantry. This is the "low" gantry position. If you need to cut thick material in the future, you can raise the gantry using the lower holes for the "high" gantry position.

Side Cover & Gantry Install

1

*Screws should be snug
to attach side covers.
Avoid overtightening*

2

*Use the included
bolts and nuts to
attach the gantry
to the Y-spar*

*Wood blocks (1 9/16"- thick)
used as support
for attaching gantry*

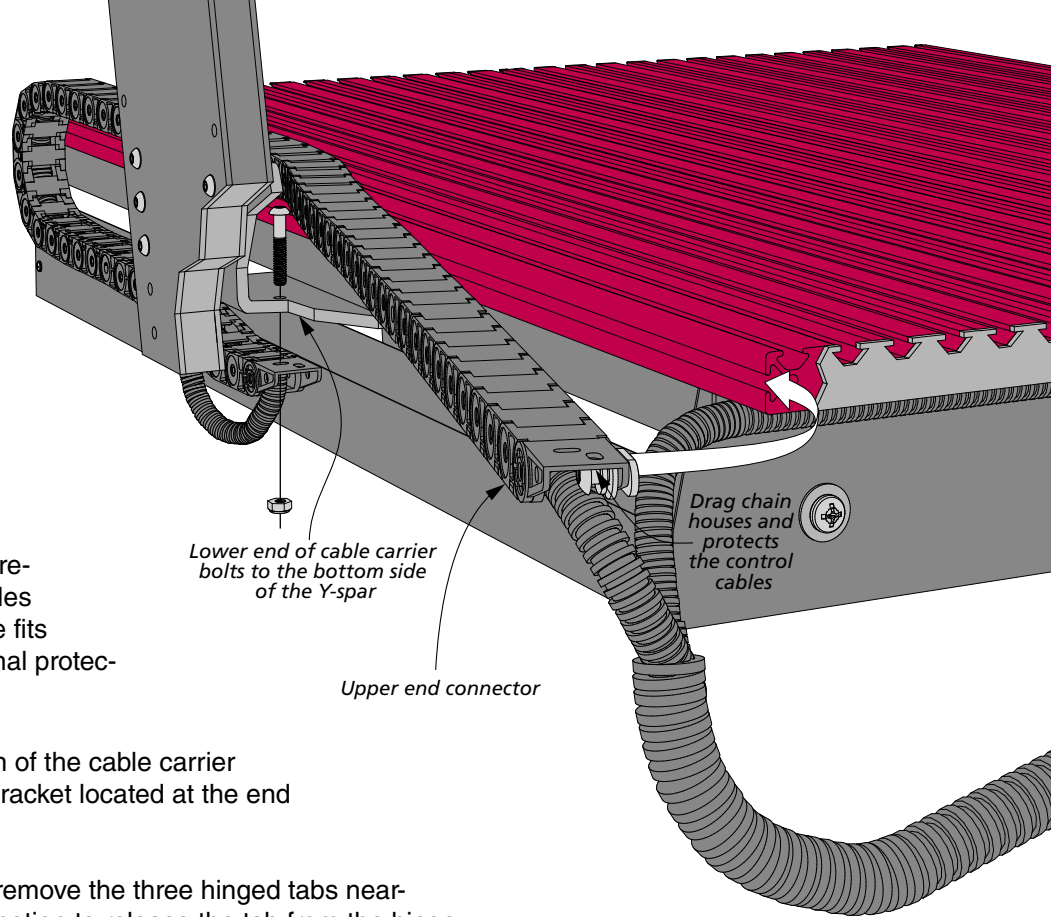Fasten the side covers with screws, just snugging them up tight.

Support blocks help align the holes in gantry and Y-spar brackets before fastening it with nuts and bolts.
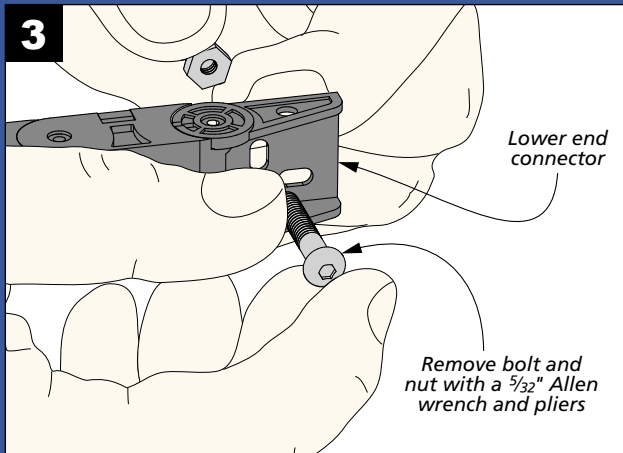
Installing the Cable Carrier

The cable carrier is a hollow, flexible plastic track that resembles a bike chain. Its job is to protect the cables that connect the controller to the gantry. As the gantry moves, the joints in the cable carrier allow it to flex, preventing sharp bends and kinks in the cables that pass through it. A plastic conduit tube fits inside the cable carrier to provide additional protection for the cables.

As shown in Figure 3, start the installation of the cable carrier by removing the screw for the mounting bracket located at the end of the chain.

Using a small screwdriver, pry open and remove the three hinged tabs nearest the mounting bracket with a twisting motion to release the tab from the hinge point (Figure 4). Pop open the remaining tabs on the cable carrier but do not remove them.

Lower end of cable carrier bolts to the bottom side of the Y-spar

Upper end connector

Drag chain houses and protects the control cables

Preparing The Cable Carrier

3

Lower end connector

Remove bolt and nut with a 5/32" Allen wrench and pliers

Remove the bolt and nut from the lower end connector of the chain assembly.

4

Tab hinge side of chain

Tab latch side of chain

Gently pry tabs with a flat blade screwdriver to avoid damaging chain
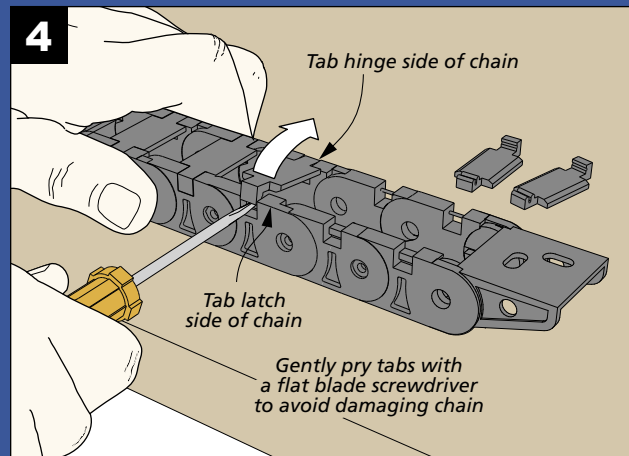
Gently pop open tabs and remove the first three from the chain assembly.

The cable that extends from the bottom of the gantry contains wiring that connects the CNC controller. This cable is housed in and protected from damage by the cable carrier. To make installing the cable into the cable carrier easier, remove the mounting bracket from the end of the cable carrier with a slight sideways twist, as shown in Figure 5.
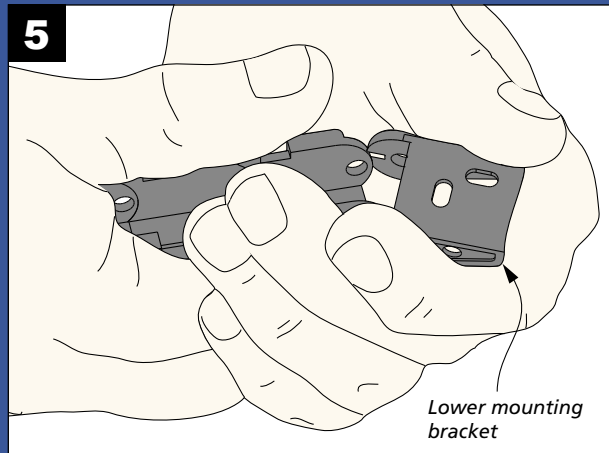
With all of the tabs open on the cable carrier (Figure 6), align and insert the cable. Snap the mounting bracket back in place in preparation for securing the end of the cable carrier to the Y-spar bracket.

Figure 7 shows you how the mounting bracket fastens to the Y-spar. Insert the screw from the top through pre-drilled hole in the horizontal leg of the Y-spar and through the mounting bracket. Install and tighten the nut securely.

SHARK®
World Class. Made in USA.
Assembly Video Series

How to attach the Drag Chain to the Base

Watch the how-to video at:
www.nextwavecnc.com/support

Chain & Cables

5

Lower mounting bracket

A slight sideways twist is all that is required to remove the lower mounting bracket.

6

Cable assembly nestles within drag chain
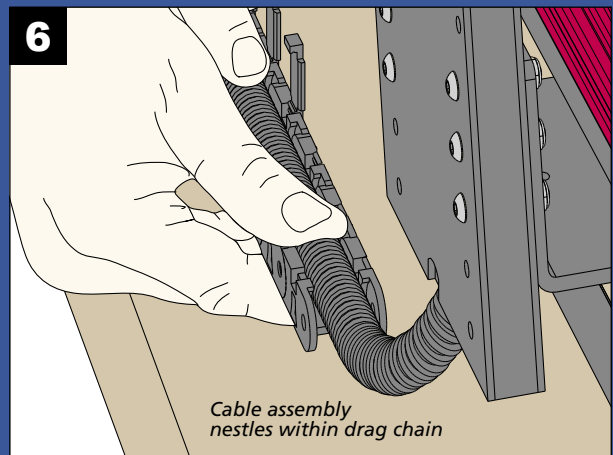
Fit the cable that extends from the bottom of the gantry into the cable carrier.

7

Mounting bracket attaches to bottom of Y-spar

FRONT VIEW
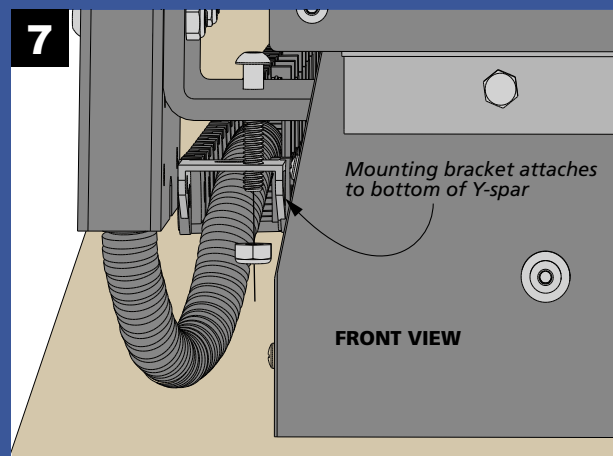
Secure the mounting bracket to the Y-spar with the screw and nut that was removed in Step 3.

Now that the cable carrier mounting bracket is secured to the Y-spar, go ahead and snap the open tabs closed around the cable, as you can see in Figure 8.

Next comes the task of routing the cable carrier inside the gantry arm and securing it to the CNC base. To do this, flip the cable carrier over itself so that it creates a "C" shape, as in Figure 9. The cable carrier should fit between the base and gantry arm.

To make it easier to secure the cable carrier to the base, remove the screws from the trim piece at the front of the

base closest to the cable carrier (Figure 10). This allows you to slide the slot nuts into the T-track on the side of the CNC table. While holding the loose end of the trim piece away from the base, slide the slot nuts into the track on the side of the base. Reattach the front trim piece, then tighten the bolts on the cable carrier using a Phillips screwdriver.

Tidy up the cabling by placing the two smaller cable bundles into the large cable managment tube and secure each end with a couple zip ties (Figure 11).

Attaching the Cable Carrier

8

Snap the brackets closed over the cable to secure the tube
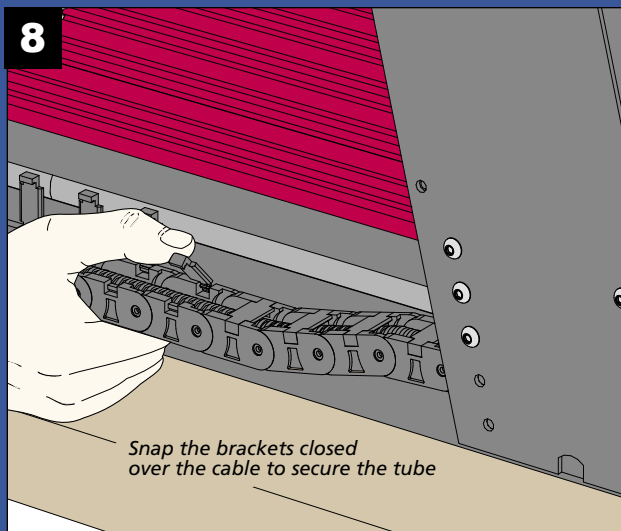
Snap all of the remaining tabs closed to secure the cable.

9
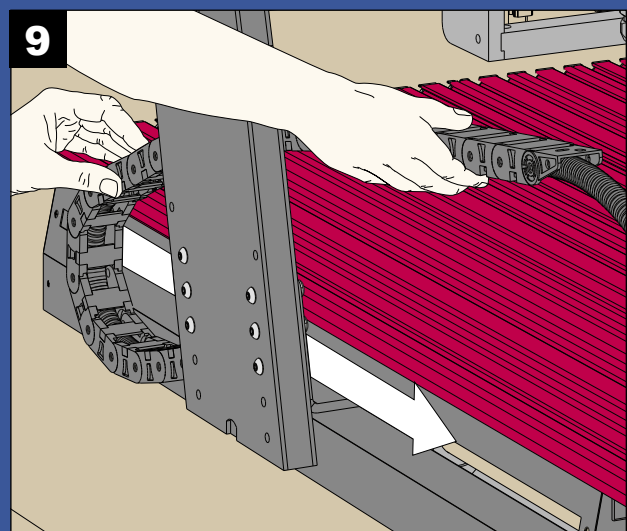
Route the cable carrier to the inside of the gantry arm to position it for final attachment to the CNC base.

10

Tighten bolts once they're installed

Retract trim piece in order to slide slot nuts into the base
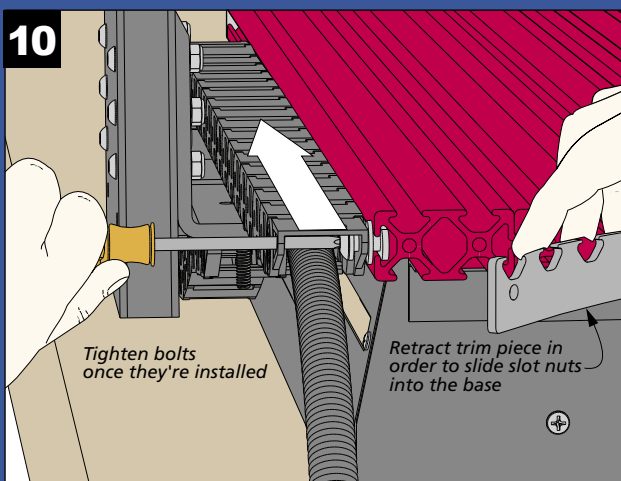
Secure the cable carrier to the slot in the side of the base with bolts and slot nuts.

11

After tightening the zip ties, snip the end off for a clean, finished look
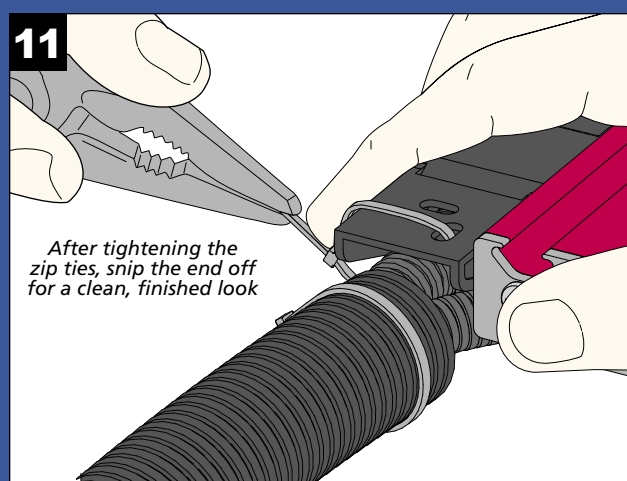
Insert the two cable bundles into the large cable management tube and clamp tight.

Installing the Router

The workhorse of the *Shark CNC* is the router or spindle. Your CNC comes with a mount designed to hold a router motor. A 2hp or 3hp liquid-cooled spindle kit is available separately from *Next Wave*. The spindle provides the same functionality as a router, but is quieter and designed for continuous use. The router and spindle both hold the cutting bits that remove and shape the material according to the tool-paths you create in your designs. Whether you're mounting a router motor or a spindle motor, the procedure for attaching the mount to the gantry is the same.

To install the router/spindle, you'll first need to attach the mount to the carriage on the gantry using four bolts, shown in the illustration at right. Tighten the two bolts on the closed (right) side of the mount, but leave the pair of bolts on the slotted (left) side loose for now.

Now insert the router into the mount. Position the collet about ¾" above the table. This position ensures most router bits will reach the table. You can adjust this height as needed for long bits or thick projects.

Tighten the remaining bolts to ensure the router doesn't move in the mount. If you need to adjust the cutting height of the router later on, simply loosen the two bolts on the slotted side and adjust the position of the router, then tighten the bolts. Zip-tie the power cord to the rear cable carrier to keep it out of the work area. Make sure to leave enough slack at the top to allow the router to move up and down freely.

Position power cord away from work area
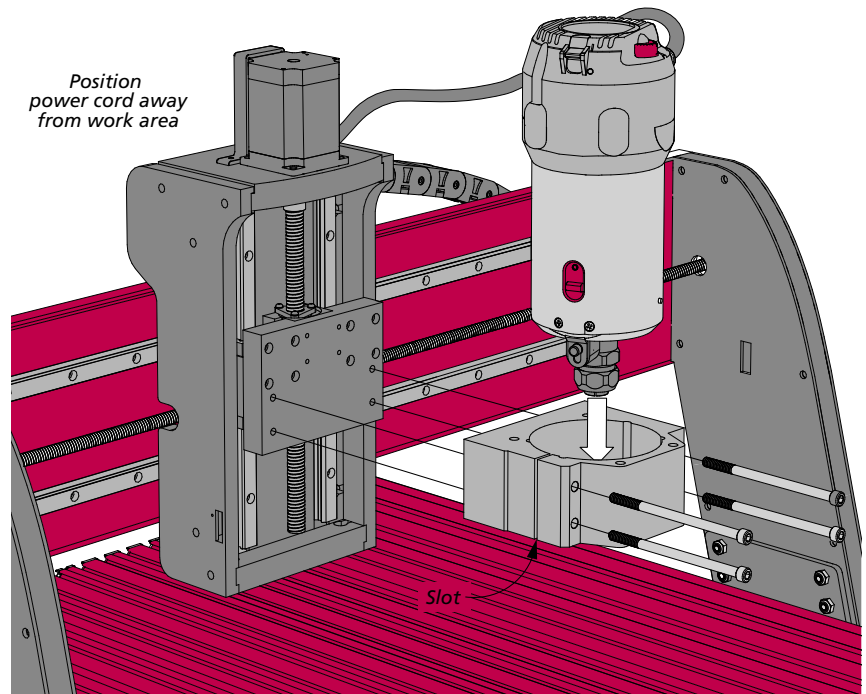
Slot

To install the router/spindle mount on the carriage, fully tighten the bolts on the right side, then insert the router/spindle and tighten the bolts on the slotted side.

IMPORTANT: READ CAREFULLY

DO NOT PLUG IN AND TURN ON THE POWER UNTIL AFTER ALL OF THE MOTOR CONNECTIONS HAVE BEEN MADE. DO NOT EVER ATTACH OR DETACH THE MOTOR CABLES WHEN THE CONTROL BOX IS POWERED UP, AS THIS WILL DAMAGE THE ELECTRONICS AND IS NOT COVERED BY THE WARRANTY!

Connecting the Cables

As you have discovered by now, and as is shown in the drawing below, there are several electrical connections that need to be made before you can power up your *Shark CNC*. The control box acts as an interface for all of the connections.

Begin by attaching the three motor cables to the control box. Each cable controls a separate motor. The ends of the cords are labeled accordingly. Insert the end of the Z, Y, and X cables into their respective port on the rear of the control box. Secure the cables by tightening the two screws on each plug finger tight. Plug the white RCA cables into the round RCA jacks. These cables control the "homing" function of the CNC, which you'll learn about later.

Your router's power plugs into the back of the control box. The electronics in the control box will power up the router and turn it off as needed. Finish by plugging in the primary AC power cable for the control box into the female connector on the rear of the control box.

NOTE: If you have the spindle kit, follow the connection instructions that come with your spindle.

The *Pendent* provides an easy way to interface with the *Shark CNC* for controlling many of its functions. Plug one end of the Pendant cable into the port on the Pendant. Connect the other end to the port on the front of the control box.

It's best to use a power strip with surge protection for your CNC. This protects it from power spikes that might interrupt the operation of the CNC. If you need to use an extension cord, we recommend a heavy-duty 12/3 cord. Plug the control box into the power strip. Now you can power up the control box with the switch on the rear of the box. To ensure that power is flowing to the machine, double check that the emergency stop is turned to the left and the button is popped out (pushing it turns it off). You can power down the CNC using either of these switches.

Wiring Schematic
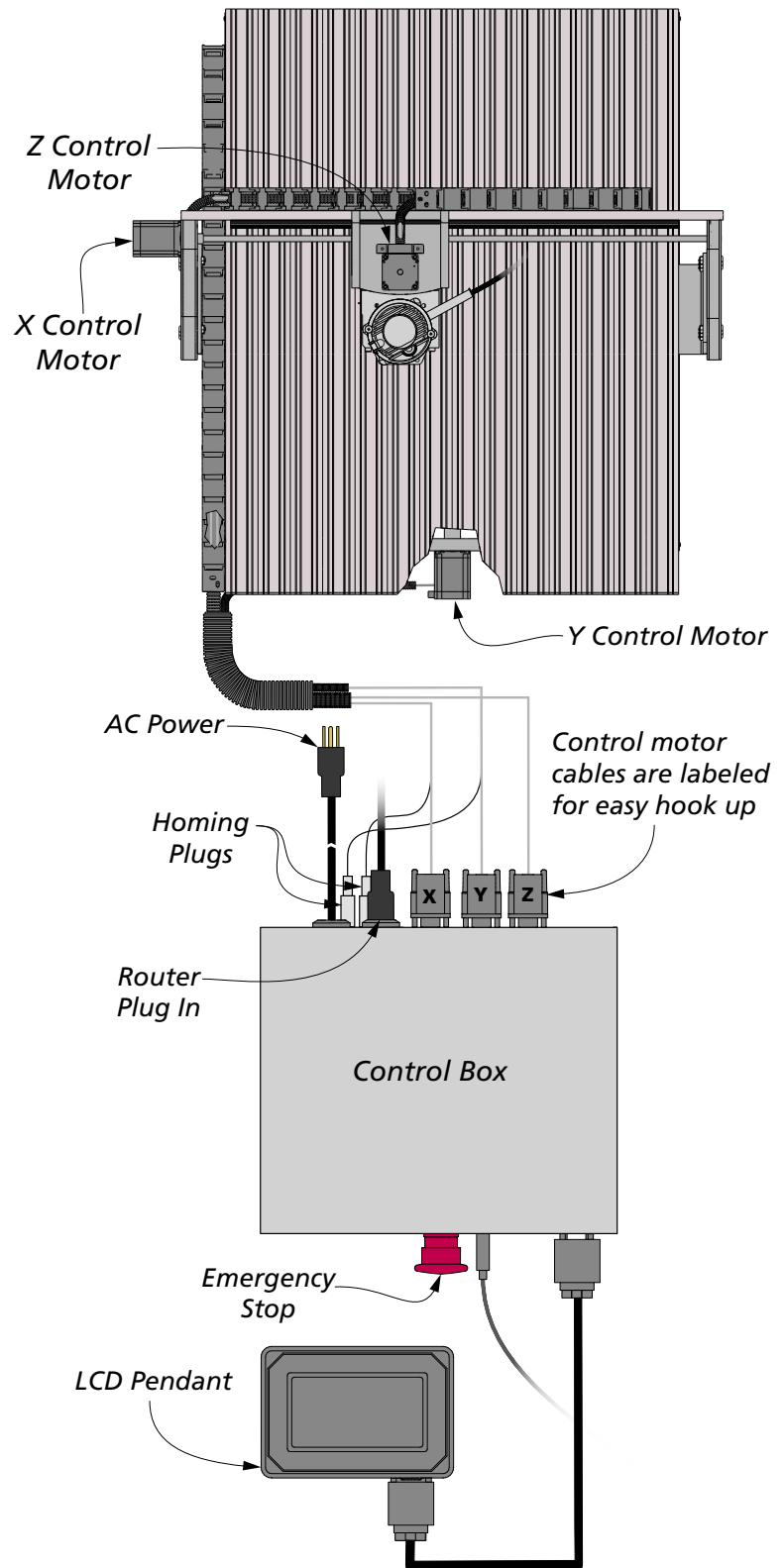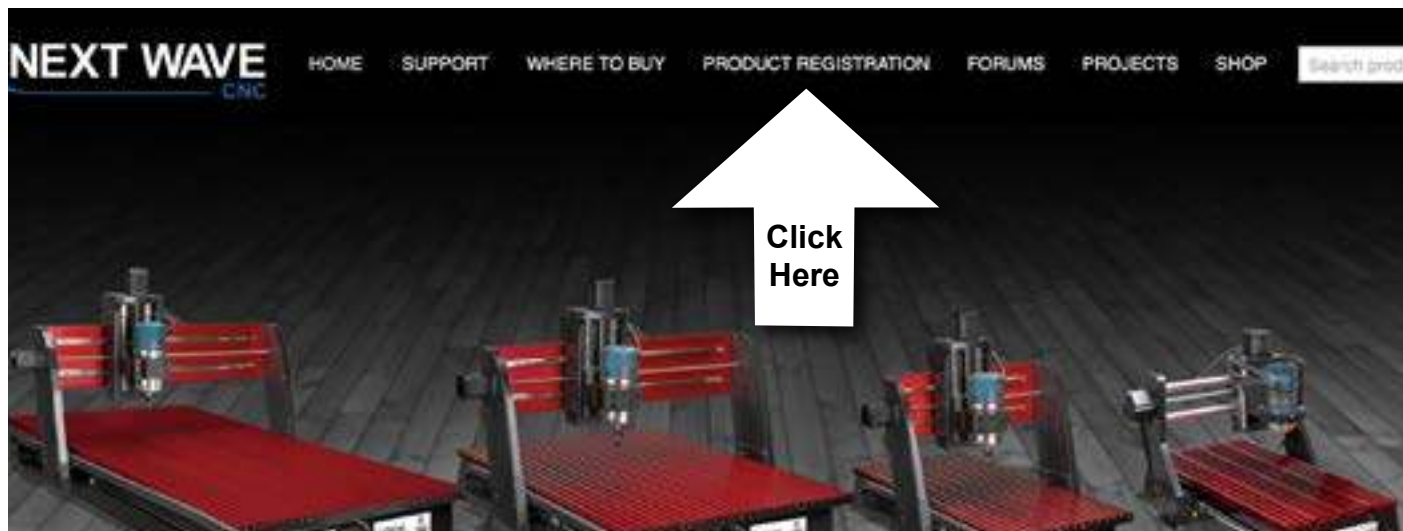
Z Control Motor

X Control Motor

Y Control Motor

AC Power

Control motor cables are labeled for easy hook up

Homing Plugs

X Y Z

Router Plug In

Control Box

Emergency Stop

LCD Pendant

Click Here

Pendant Registration

Before continuing the setup of your *SHARK HD CNC*, register your machine 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 the unlock code will display on your computer screen. (An account confirmation will also be sent to you email.) Use the number keys on the Pendant to enter your Unlock (Access) Code.

For complete and detailed instructions on using the Control Pendant, refer to the Pendant User's manual.

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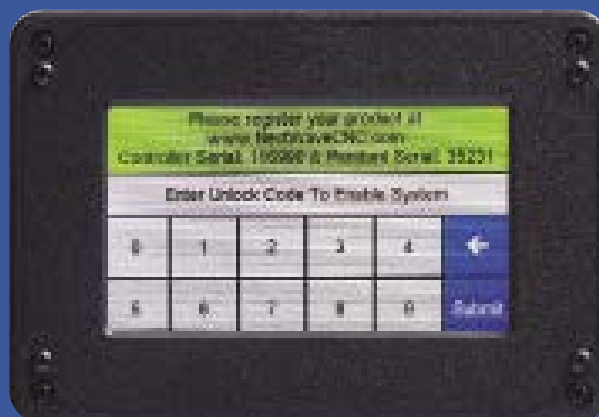
Unlock the Pendant by entering the access code you received upon registration.

After successfully unlocking the Pendant, you're ready to control your *SHARK HD CNC*.

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. Setup 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* 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.

1. First, you can use the Pendant to stop the machine. Simply press the red STOP button on the Pendant screen. This stops the CNC's movement and turns off the router/spindle.

2. The second option for quickly stopping the CNC operation is to press the large, red Emergency switch on the front of the control panel.

3. You can also use the power switch on the rear of the control box to shut down the system.

4. If you have the control box plugged into a switched power strip, you can also power down the machine in an emergency by turning of the power strip.

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, since 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.

Methods 2, 3, or 4 also work to shut off your machine at the end of the day.

4 Ways to Shut It Off

1

File: Next Wave Cutting Board.tap
File Size: 145368
Lines Sent:
Transmitted:
Time:

Start
Stop
Pause
Resume
Speed Override: 100%

2

3

4

Basic Pendant Functions

The Pendant has many great functions to control the *SHARK HD CNC*. 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 Pendant screen (red arrow). Pressing this key repeatedly cycles between Fast, Medium, and Slow jog speeds.

Take some time to experiment with moving the spindle around. Jog the router to get a feel for the different speeds. Take special note of the position readouts at the top of the Pendant screen. Try to stay within the limits of the machine. If the router travels and bumps against the limits, you may here 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-AXES. Another function of the Pendant is the "Zero XY" function, as shown in Figure 2. This sets the current position of the router as a zero, or origin 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- axes. 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 for a cut by experimenting with moving the spindle to a number of locations and zeroing the X- and Y-axes.

MOVE TO THE ORIGIN. Another useful 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 router to the X- and Y-origin point you have previously set. Set a new origin point and give it a try. Using a long press/double tap moves the spindle to the original X-, Y-, and Z position.

Make It Move

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

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

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 Pendant functions to get familiar with how your *SHARK HD CNC* moves and sounds.

Learning to zero the X-, Y-, and Z-axes 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 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- axes.

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 OPTIONAL 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 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 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.

Manual Setup
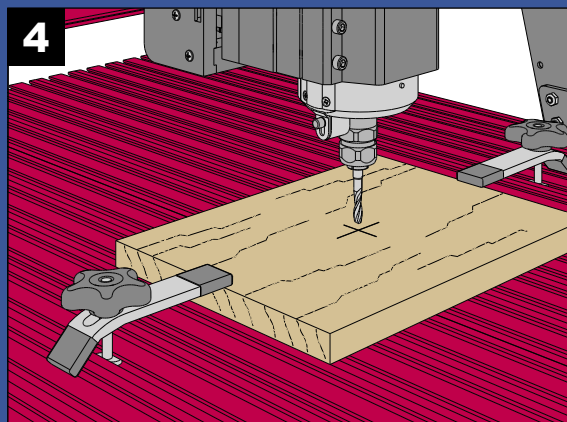
Zero the X- and Y- axes first by jogging the bit until it's positioned over the center of the workpiece.
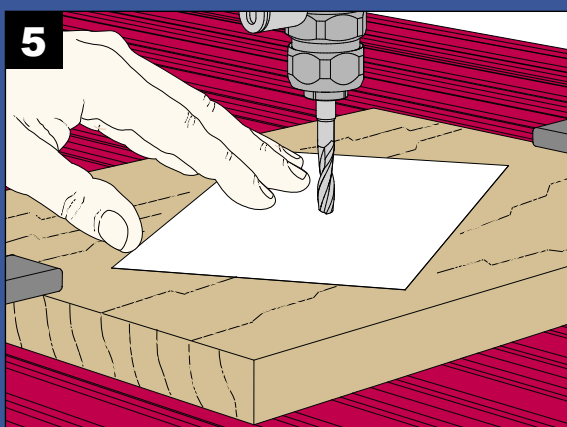
Gradually lower the bit until it just starts to touch a piece of paper placed on the workpiece.

Optional Zero Touch Plate
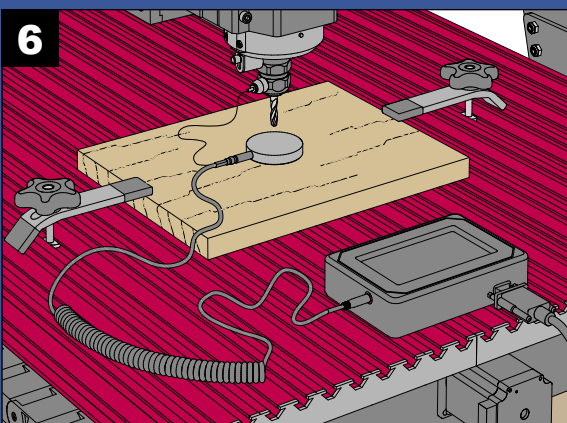
The CNC Touch Plate has a single lead on one end of the cord. Plug this into the side of the Pendant. Position the CNC Touch Plate under the router bit and attach the magnet to the collet nut or side of the bit.

VCarve

Installing VCarve Software

Using a CNC router is unlike any other powered wood-working 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.

For your own creations using the *SHARK CNC*, the first thing to do is design your project. Then you define the tools and toolpaths that will be used by the CNC to cut the workpieces

VECTRIC VCARVE. Your *SHARK CNC* comes with a license for *Vectric VCarve* design 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. For design support and training with *VCarve*, visit www.vectric.com and click on the Support Tab for information and video tutorials. You can also email your *VCarve* questions to *support@vectric.com*.

Software Installation Process

STEP 1. Insert the *VCarve* USB-A drive into a USB port on your computer.
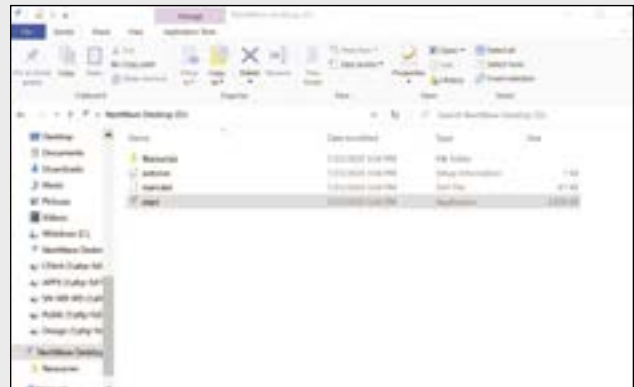
STEP 2. Using File Explorer, navigate to the USB drive and double-click on the file *setup.exe*.

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

STEP 4. Click on *INSTALL SOFTWARE* to begin the installation.
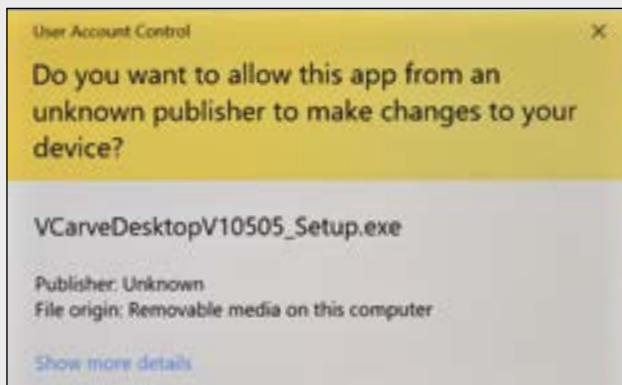
STEP 5. A box will pop up (appearance may vary) asking if you want to allow this software to make changes on your computer. Click *OK*.

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

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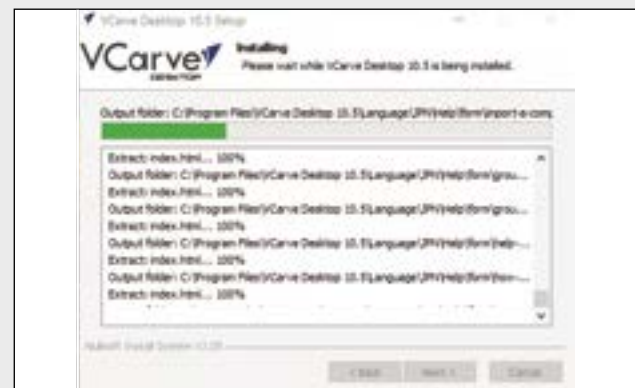
STEP 12. Wait until the installation process is complete before clicking on *Next*.

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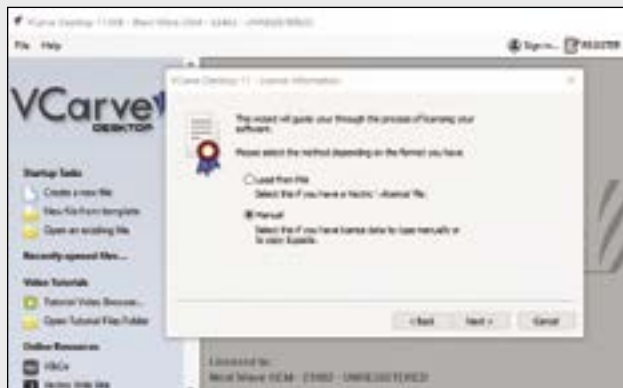
STEP 15.. You'll be entering your license information from the box lid, so select *Manual* and press *Next.*

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.*

STEP 19. Select the option for personal use under *Registration Type* and enter your *Contact Details*. Click *Submit Registration Request*.
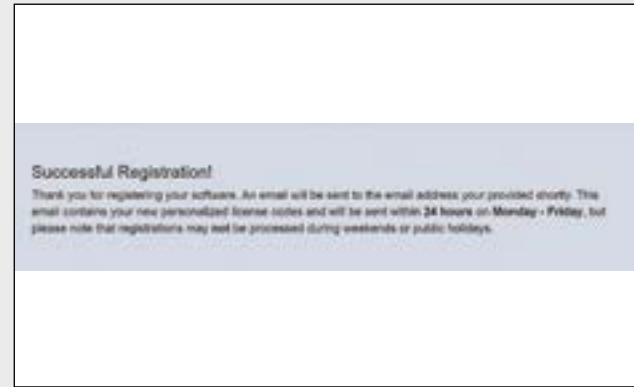
STEP 20. If you have successfully registered the software, you will see this screen.

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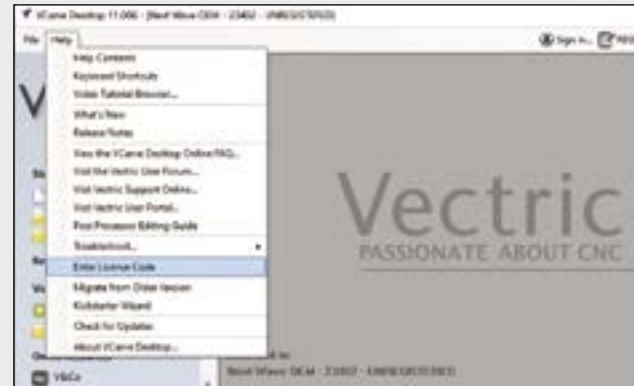
STEP 22. In *VCarve*, you may see a pop-up dialog box asking for your license code. You can also select *Enter License Code* from the *Help* drop-down menu.

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*.

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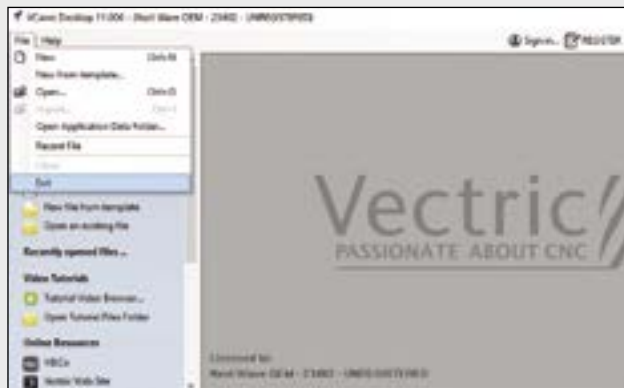
STEP 27. If you click *Finish* in Step 26 without checking the box *Restart Now*, you can simply choose *Exit* from the *File* pull-down menu to exit *VCarve*.

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

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 (See Step 3).

STEP 3. If you have an account at *Vectric.com*, enter your username 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.
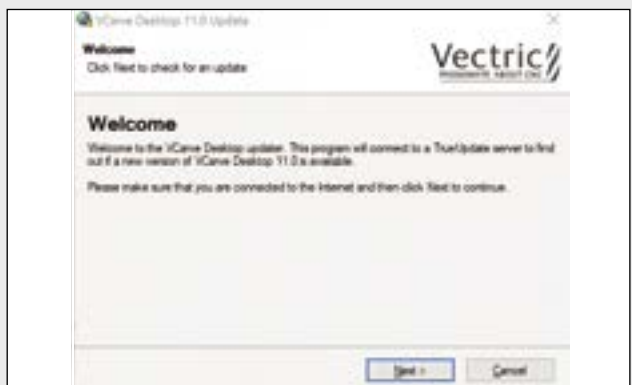
STEP 2. Your system must be connected to the internet to download updates. When you click *Next*, the system checks for any available updates for *VCarve*.
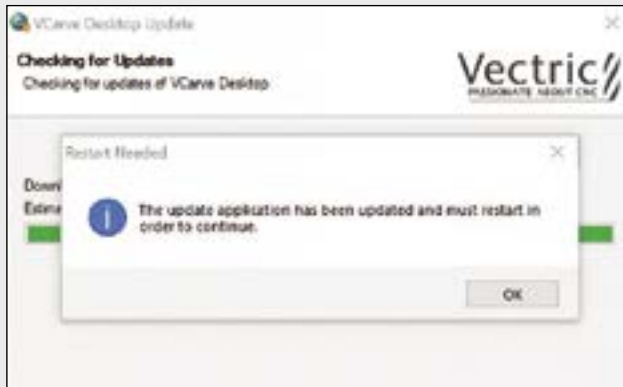
STEP 3. If there is an update for the software that checks for and delivers the *VCarve* updates, it will be installed first. Click *OK* to continue.

STEP 4. Any updates for *VCarve* will be displayed. Click *Next* to install the update.
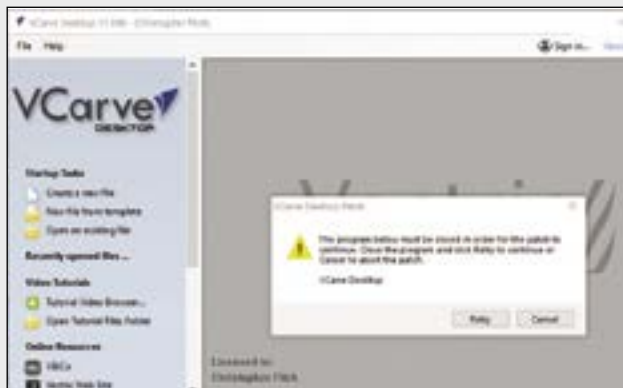
STEP 5. In order to properly install the update, *VCarve* must be shut down. Click *Retry* to continue installing the software update.

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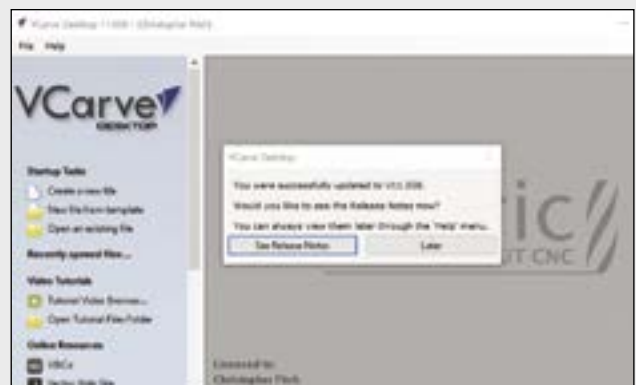
STEP 8. If you're interested in all of the features or software bugs that were addressed with the update, click on *See Release Notes*. Otherwise, click *Later*.

A Productive VCarve Workflow

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 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. 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 example projects listed later in the manual are set up.

Click OK to save and exit the Job Setup window.
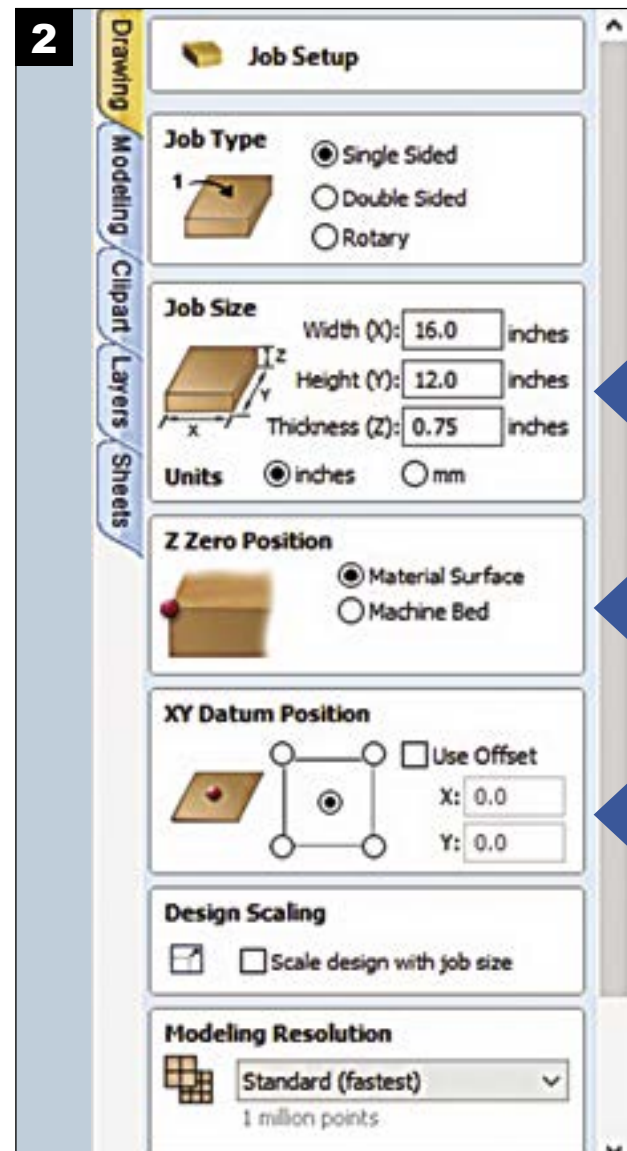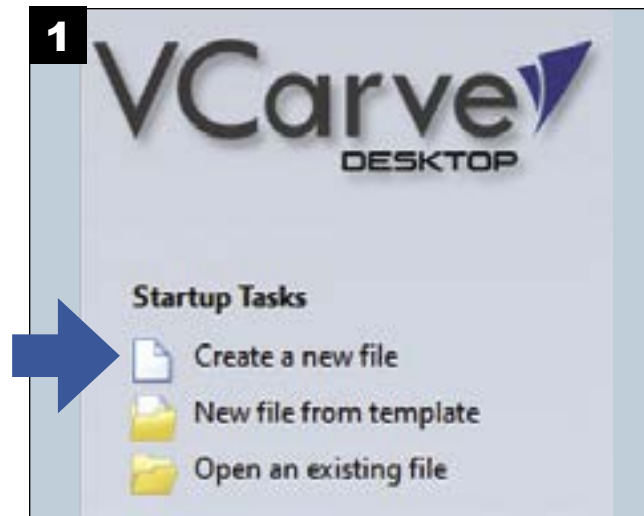
Drawing Tools

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/Webinars.*

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 setup 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.

27

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 0.05 to 0.125 in.
> All bits - Feed Rate 50-120 ipm
> All bits - Plunge Rate 50-80 ipm
> Straight bit Step over 10% - 40%
> V-bits - Final pass 2%
> V-bit - Clearance pass 20%
> Ballnose bit - 3D finishing pass 7-11%

5 oolpaths

V-Carve / Engraving Toolpath

Cutting Depths
Start Depth (D) 0.0 inches
☑ Flat Depth (F) 0.125 inches

Tool: V-Bit (90 deg 1/2 in)
Select ...   Edit ...

☑ **Use Clearance Tools**
End Mill (1/4 in)

Select ...   Remove   Edit ...

6 Database

Material Acrylic   Online   Machine Desktop

⊟ **Imperial Tools**
  ⊟ **Ball Nose**
    Ball Nose (1/32")
    Ball Nose (1/16")
    Ball Nose (1/8")
    Ball Nose (1/4")
    Ball Nose (3/8")
    Ball Nose (1/2")
  ⊟ **V-Bits**
    V-Bit (60.0° - 1/4")
    V-Bit (60.0° - 1/2")
    V-Bit (90.0° - 1/2")
    V-Bit (90.0° - 1 1/4")
  ⊟ **Engraving**
    Engraving (15.0°, Tip 0.0:
    Engraving (10.0°, Tip 0.0:
⊟ **Metric Tools**
  ⊟ **Ball Nose**
    Ball Nose (1 mm)
    Ball Nose (1.5 mm)
    Ball Nose (3 mm)
    Ball Nose (6 mm)
    Ball Nose (9 mm)
    Ball Nose (12 mm)
  ⊟ **V-Bits**
    V-Bit (60.0° - 6 mm)
    V-Bit (60.0° - 12 mm)
    V-Bit (90.0° - 12 mm)
    V-Bit (90.0° - 32 mm)
  ⊟ **Engraving**
    Engraving (15.0°, Tip 0.2!
    Engraving (10.0°, Tip 0.5

☑ **V-Bit (90.0° - 1/2")**

Notes

Tool Type   V-Bit
Geometry
Units   inches
Diameter (D)   0.5   inches
Included Angle (A)   90   degrees

No. Flutes   2

Cutting Parameters
Pass Depth   0.1   inches
Final Pass Stepover   0.01   inches   2   %
Clearance Pass Stepover   0.1   inches   20   %
Feeds and Speeds
Spindle Speed   14000   r.p.m
Feed Units   inches/min   Chip Load   0.0015   inches
Feed Rate   42   inches/min
Plunge Rate   12   inches/min
Tool Number   11

Remove   Apply

Select   Close

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).
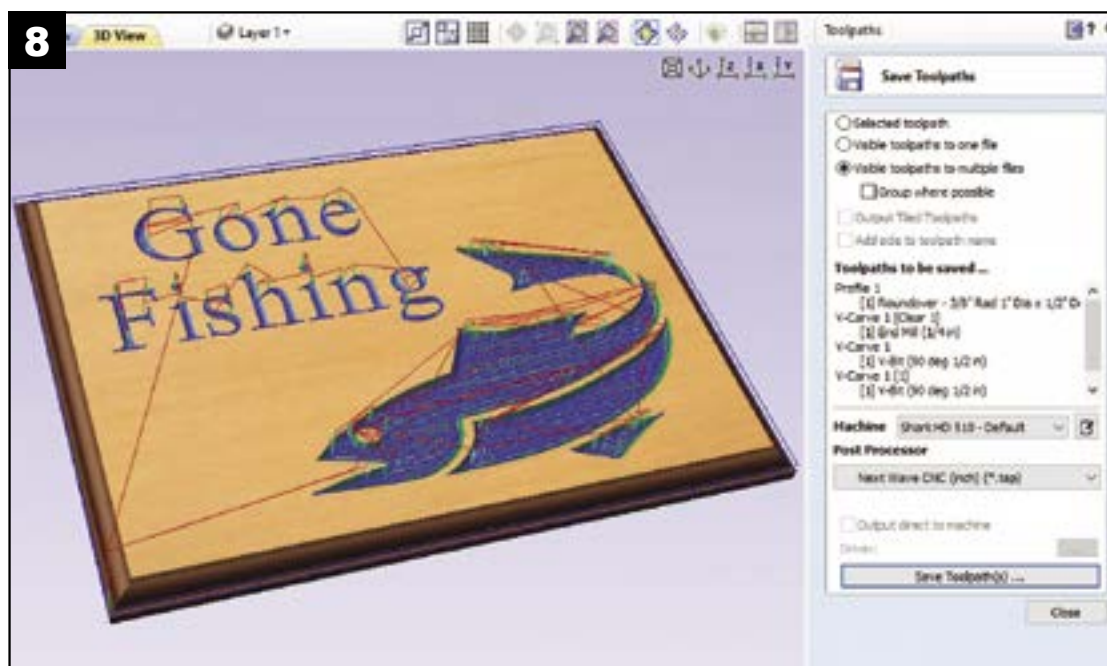
Setting Up Your HD Machine for Cutting

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 *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
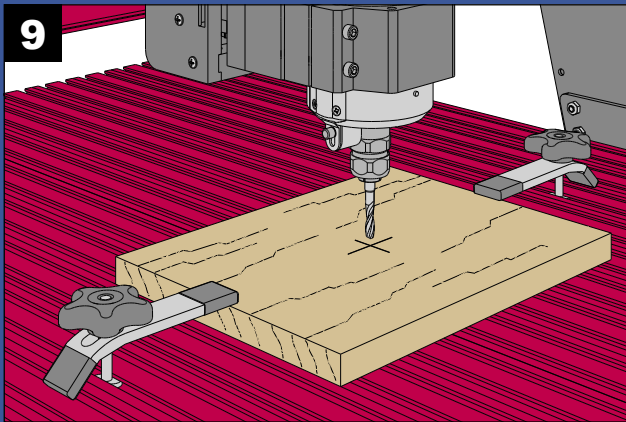
Zero the X- and Y- axes first by jogging the bit until it's positioned over the center of the workpiece.
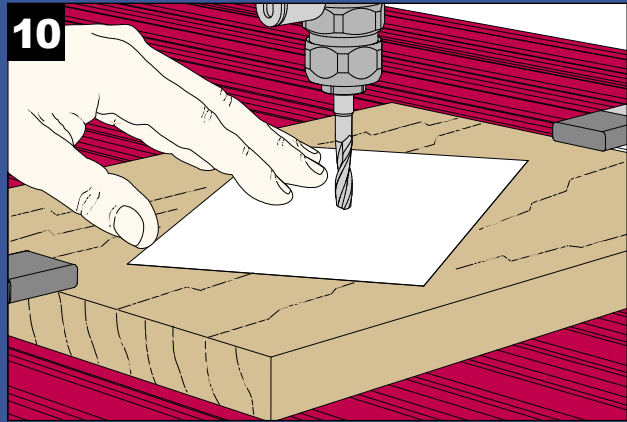
Gradually lower the bit until it just starts to touch a piece of paper placed on the workpiece.
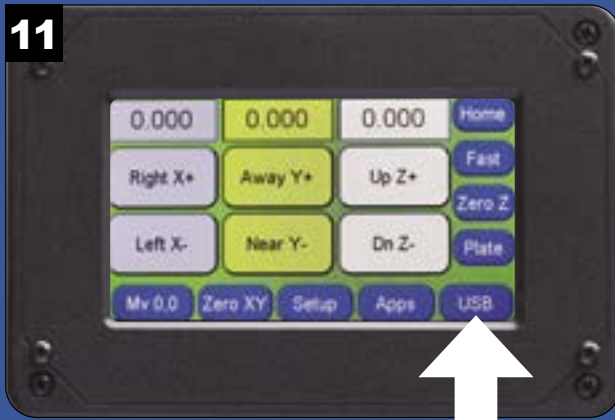

After inserting the flash drive into the Pendant, press the "USB" button to select the project file.

From the list of available projects, select your chose .tap file to proceed.

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 *VCave* file and your CNC machine (Figure 13). If everything checks out, press Next.

Review the safety check lists from page 33, 35, or 37, turn on your router and press Start. Your *Shark CNC* rout-er 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 *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

```
VECTRIC POST REVISION)
47050B966FEC1778F1CBCD9BC4FA35FC)
Gone Fishing Profile )
File created: Friday January 28 2022 -
for Next Wave Automation from Vectric
Material Size)
X= 16.000, Y= 12.000, Z= 0.750)
Z Origin for Material  = Material Surf
XY Origin for Material = Center)
```
Exit   Up   Down   Next

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

14

```
File: Gone Fishing Profile.tap
File Size: 1234
Lines Sent:
Transmitted:
Time:
```
Start   Stop
Pause   Resume
Speed Override: 100%

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 HD 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 16 and 17 of this manual. Install a ¼" straight router bit in the collet of your router or spindle and set the speed to approximately 17,500 RPM (Spindles equipped with a spindle interface cable will automatically set the speed). Use the Pendant to zero the X- and Y- axes and then zero the Z-axis.

Download project plans at
www.NextWaveCNC.com/cncprojectplans

To load the project .tap file that feeds instructions to the CNC, insert the thumb drive into the 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 Set-Up & Routing
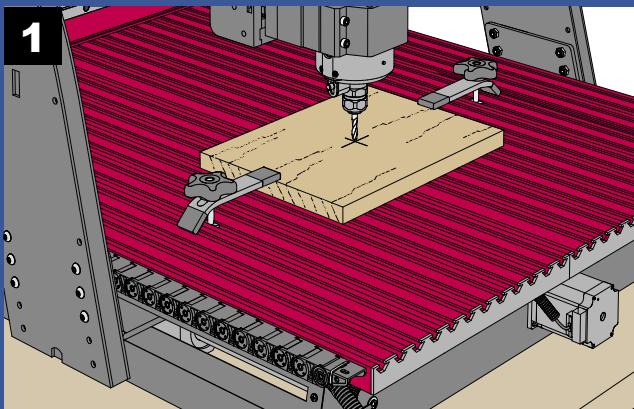
1 Mark a center point on the workpiece before securely clamping it to the CNC table with hold-down clamps. Zero the X-, Y-, and Z-axes before proceeding.
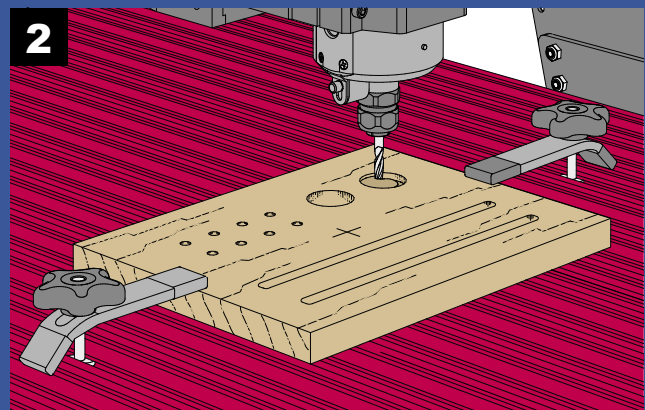
2 The project file contains the instructions for the CNC to make all of the pocket cuts in the workpiece.

☑ 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 spindle set to approximately 17,500 RPM?

☑ 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 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."

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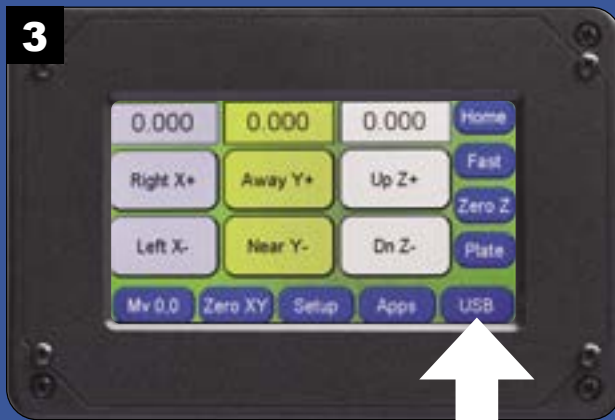
After inserting the flash drive into the Pendant, press the "USB" button to select the project file.

4
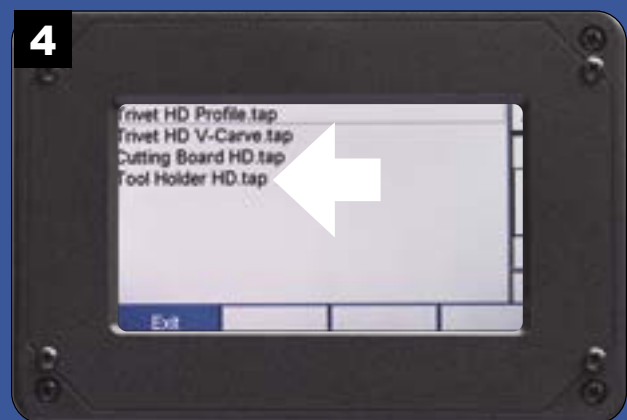
From the list of available projects, choose the Tool Holder HD.tap file to procede.
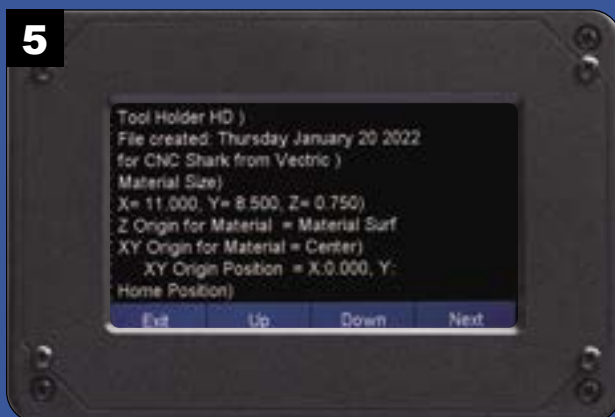
5

Verify all of the set-up 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, as shown at right. 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 HD.tap file from *NextWaveCNC.com/CNCprojectplan*. Copy the file onto a USB flash drive.

Attractive, tight-grained hardwoods are best for a cutting board. Prepare a blank 9" x 17" 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

Download project plans at
www.NextWaveCNC.com/cncprojectplans

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 or spindle and set the speed to 17,500 RPM. 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, set-up information for the job pops up for your review and verification. If everything looks okay, press "Next."

Workpiece Set -Up

1
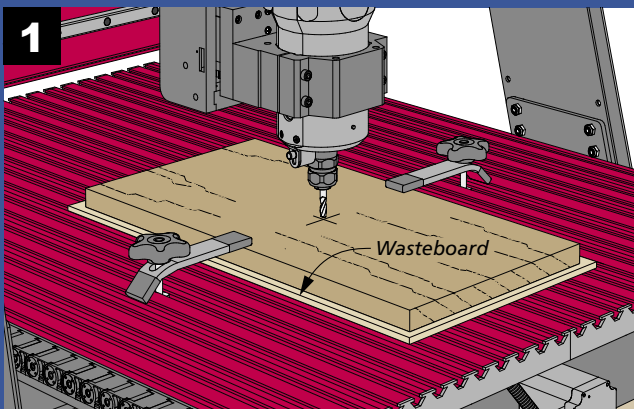
Mark a center point on the workpiece before securely clamping it and the waste board to the CNC table with hold-down clamps. Zero the X-, Y-, and Z-axes.

2
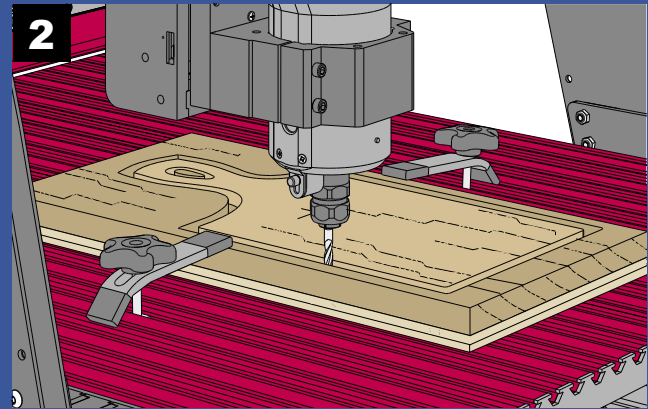
Watch the router cut around the profile of the cutting board while leaving tabs to keep it secure on the CNC table.

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 or spindle set to approximately 17,500 RPM?
- ☑ 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!
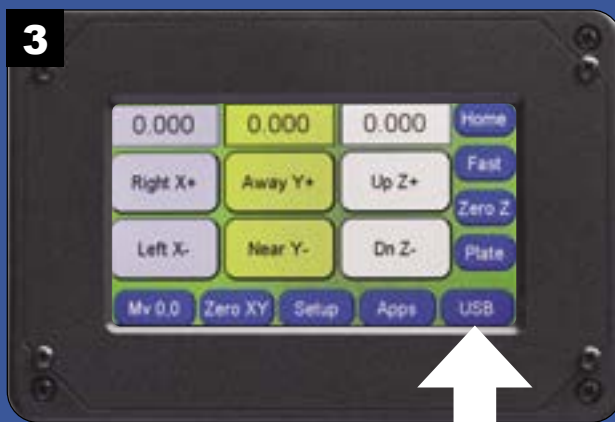
After inserting the flash drive into the Pendant, press the "USB" button to select the project file.

From the list of available projects, choose the Cutting Board.tap file to proceed.
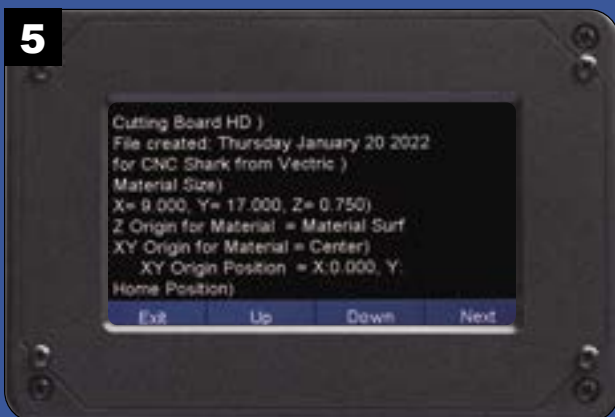
Verify all of the setup information for this job, review the safety checklist, then press "Next".

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

35

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 *Next-Wave.com*: Trivet Vcarve HD.tap and Trivet Profile HD.tap. Copy the files onto a USB flash drive.

Download project plans at
www.NextWaveCNC.com/cncprojectplans

For this project, you will need a piece of solid wood that measures 12" x 12" 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 17,500 RPM.

Workpiece Set -Up
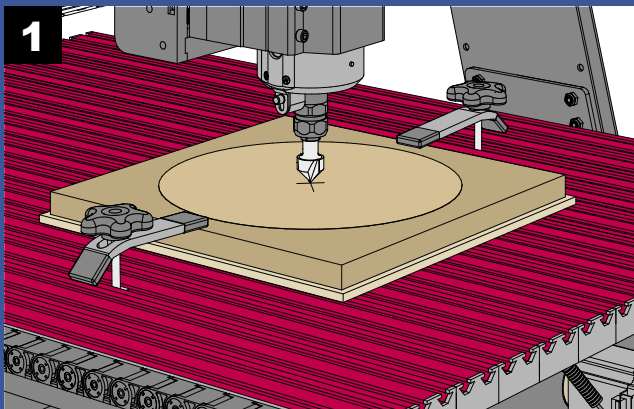
1

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.

2
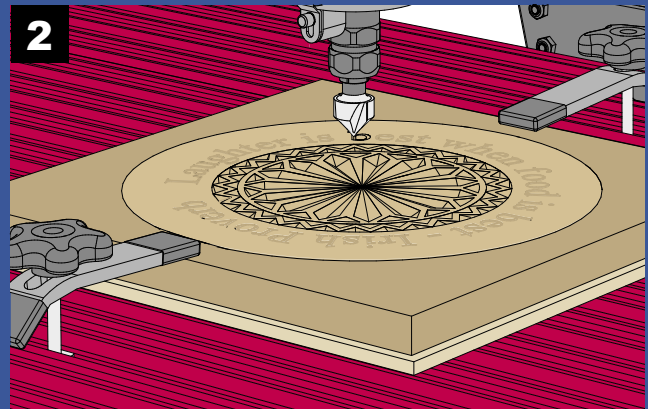
The V-bit makes unique cuts that create an eye-catching pattern in three dimensions.

Once you've zeroed the X-, Y-, and Z-axes, insert the thumb drive into the 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 set-up 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 or spindle set to approximately 17,500 RPM?
☑ 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 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."

Verify, Then Create!
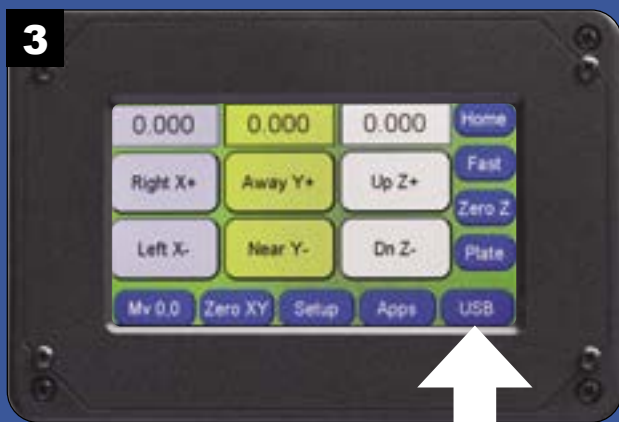
After inserting the flash drive into the Pendant, press the "USB" button to select the project file.
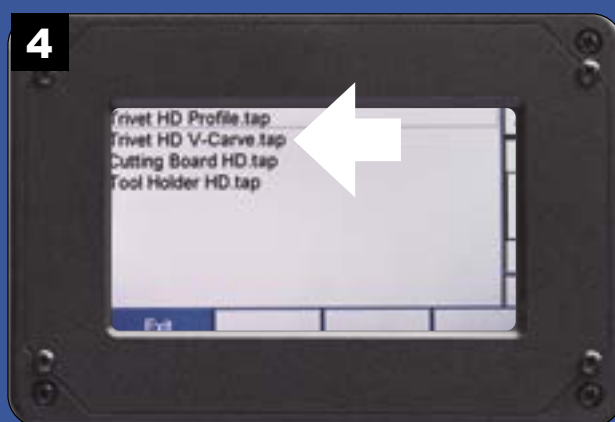
From the list of available projects, choose the Trivet HD V-Carve.tap file to proceed.

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

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

37

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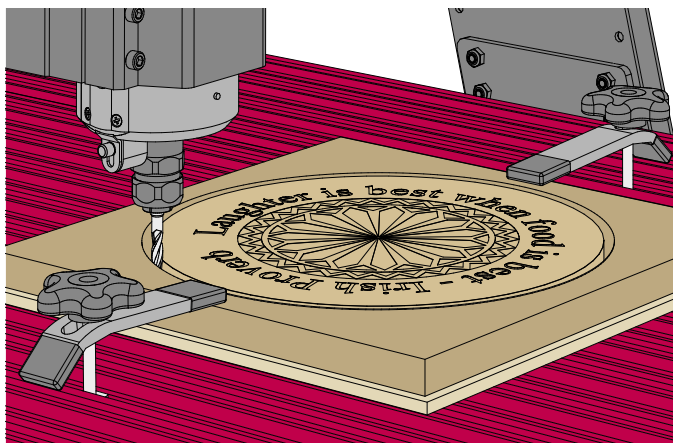
Watch the router cut around the profile of the trivet while leaving tabs to keep it secure on the CNC table.

Finish the Trivet

7
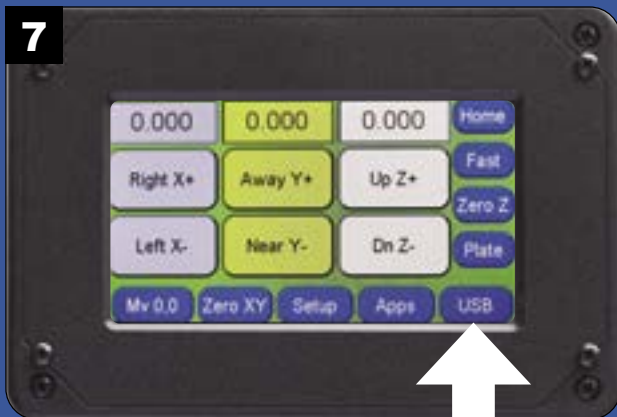
After inserting the flash drive into the Pendant, press the "USB" button to select the project file.

8
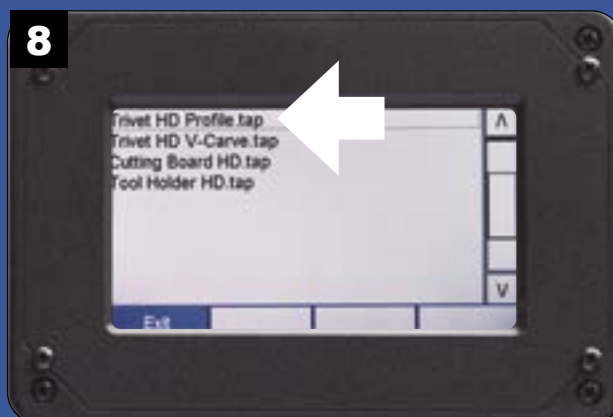
From the list of available projects, choose the Trivet Profile HD.tap file to proceed.

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 *Pendant* buttons stop responding correctly, clean around the screen with a piece of paper.

Use a dry lubricant for the rails, round bars, and leadscrews. Dry lubricant doesn't attract dust. Purchase *Dupont 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.

To lubricate the linear bearings, use the linear bearing grease and nozzle kit from *Next Wave*. It includes a light lithium grease and a nozzle to attach to a mini grease gun you supply:
https://www.nextwavecnc.com/shop/LINEAR-BEARING-GREASE-AND-6MM-NOZZLE-KIT-p208340135

Find additional maintenance information at:
https://nextwaveautomation.zendesk.com/hc/en-us/articles/4402720442253-HD5xx-Series-Lubrication-Guide

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, including 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 rods and leadscrews with mineral spirits or other mild solvent to remove wood glue and resin.
- Apply a light coat of dry lubricant to the round Y-bars and leadscrews. Wipe off excess with a soft rag.
- Grease the linear bearings (more often if needed) with the Next Wave grease and nozzle kit.

Guide Rail & Bearing Maintenance

X 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 and Z rails at least once every 8 hours of operation. While the machine is

Clean & Grease

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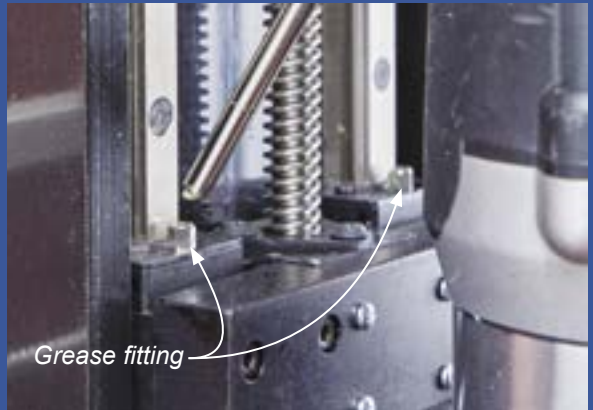
Grease fitting

Place the grease gun's nozzle over the grease fitting located on the bearings. Firmly push the nozzle against the bearing's grease fitting and use the trigger to apply the grease. It takes only a couple pulls for each application.

Grease fitting

To lubricate the Z-axis bearings, you'll find the grease fittings behind the spindle mount.

stopped, wipe off the rails with a clean cloth or soft brush. After cleaning the rails, lubricate the bearings with a dry lubricant.

X and Z Bearings

The two Z-axis bearings are located behind the router/spindle mount. The two X-axis bearings are located behind the Z carriage. These four bearings require periodic lubrication with light lithium grease included in the linear bearing grease and nozzle kit from *Next Wave*. Linear Bearing Grease and 6mm Nozzle Kit Available at *www.NextWaveCNC.com/shop.*

Y Rails (under the machine)

Clean the round Y rails under the machine in the same manner as the X and Z rails. After cleaning the Y rails, apply a dry lubricant then run the machine Y-axis back and forth a few times to distribute the lubricant evenly along the rails.

Y Bearings (under the machine)

The round linear Y bearings under the machine do not require lubrication. Applying dry lubricant to the round rails is sufficient.

Resources

Next Wave CNC

nextwaveCNC.com

Manufacturer of the *SHARK HD* and *SD* series 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* software, 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 valuable 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

For design support and training with *VCarve*, visit *www. vectric.com* and click on the Support Tab for information and video tutorials. You can also email your *VCarve* questions to *support@vectric.com.*

SHARK®
World Class. Made in USA.

2HP WATER COOLED SPINDLE SYSTEM

- Powerful 2HP motor
- ER11 collet system includes 1/4" & 1/8" collets
- Water pump, hose and connection kit included
- 110V uses standard 20 amp circuit
- 6,000—24,000 rpm (rotations per minute) variable speed Inverter
- Spindle clamp adapts to existing router mounts
- Less than .005 mm runout Superior quiet operation Outer dimensions 80 mm x 180 mm

SKU 20157

NEXT WAVE CNC

SHARK®
World Class. Made in USA.

3HP WATER COOLED SPINDLE SYSTEM

- Powerful 3HP motor
- Digital Speed Control
- ER20 collet system includes 1/4" & 1/2" collets
- Enclosed Cooling System
- In line Flow Indicator

SKU 20160

NEXT WAVE CNC

SHARK®
World Class. Made in USA.

DUST BOOT SYSTEM

- Quick Remove Brush attachment allows for easy bit change
- Accepts standard dust connectors
- Replaceable Brushes - Available for all Machines

HD SKU 20170 • SD SKU 20165

NEXT WAVE CNC

SHARK®
World Class. Made in USA.

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

![Next Wave CNC logo] **NEXT WAVE** CNC — **Online Webinars & Training**

Getting Started
with your SHARK CNC

SHARK LCD Pendant
Set-Up & Operation

READY2CONTROL NEXT WAVE CNC

Ready2Control
Set-Up & Operation

VCarve Design
Tool Path Essentials

NEXT WAVE CNC   VCarve   CNC

And More:
Laser Setup and Operation
Rotary 4th Axis Setup and Operation
Spoil Board Setup and Maintenance

SCAN ME

Register Now
https://www.nextwavecnc.com/webinars

NEXT WAVE
CNC

SHARK®
World Class. Made in USA.

SHARK HD500/510/520
User's Manual

800001

V. 01 06 2023 rj

SHARK Pendant
10.1" Touch Screen Display
Owners Manual

Watch Getting Started with your Pendant training video at:
https://www.nextwavecnc.com/webinars

Technical Support

If you need technical assistance with your *SHARK Pendant*, 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: 9am – 5pm Monday – Friday (Eastern Time).

Pendant System Requirements – The Pendant is compatible with all current *SHARK* HD and SD series CNC machines as well as past machines that use a control box with a Pendant cable input port.

Some (but not all) older machines that lack the Pendant input port can be upgraded by switching to the new style Control Box that has the Pendant input port. For information on upgrading an older *Next Wave CNC* machine contact Customer Service at *Support@NextWaveCNC.com*

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

Pendant Overview

The Pendant serves as the primary interface that allows you to operate and control your *SHARK CNC.* The touch screen allows you to control the CNC using your finger, or a soft stylus.

You will also find three connectors on the Pendant. The multi-pin connector on the bottom of the Pendant fits the cable from the CNC's control box. Connect this cable first. The optional Z touch plate accessory plugs into the jack on the left side of the Pendant. The touch plate zeros the Z-axis prior to cutting. The USB ports accept a USB flash drive containing the TAP/G-code files to cut your project on the CNC.

Left End View of Pendant

USB Ports

Bottom View of Pendant

Control Box Connector

Z Touch Plate
Accessory Port

Warning

DO NOT plug in and turn on the control power until all the motor and Pendant cables have been attached. DO NOT attach or remove the motor or Pendant cables when the control box is powered up as this can damage the electronics and is not covered by the warranty.
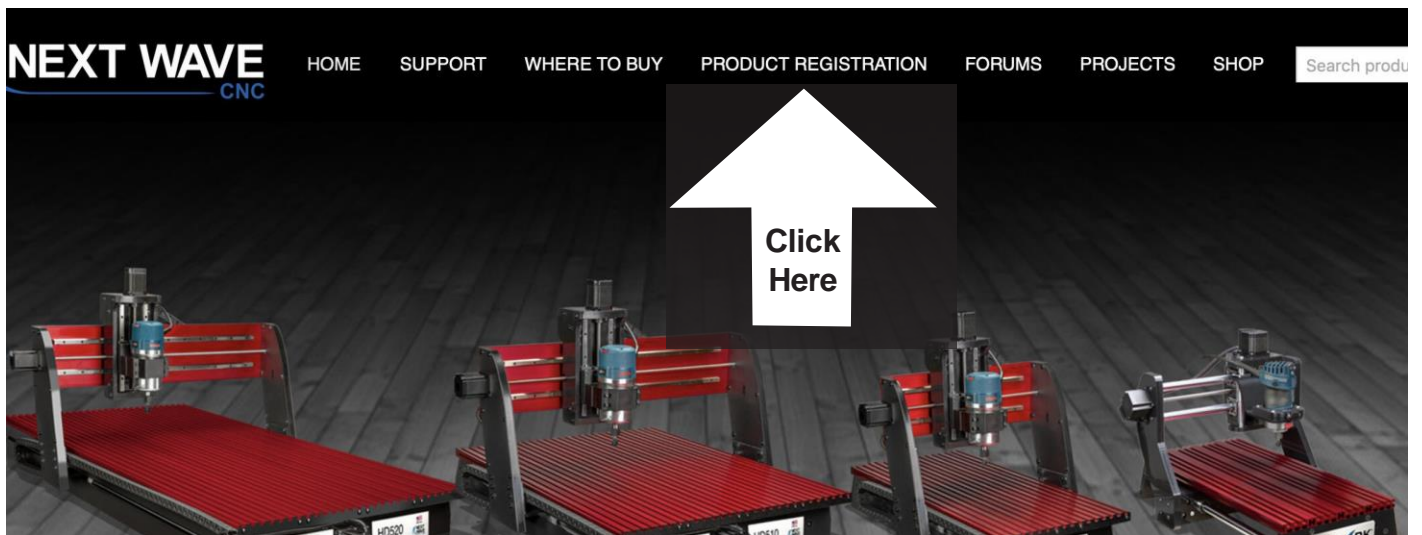
Click Here

Pendant Registration

Before continuing the setup of your *SHARK HD CNC*, register your machine 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 to unlock your Pendant. You will also receive the code in your email inbox. Use the number keys on the Pendant to enter your Unlock (Access) Code.

Your *SHARK CNC* is now unlocked, and ready to use.

Serial Number and Software License Information:
For easy reference and record keeping, enter your *SHARK* Pendant and Controller information below.

Pendant Unlock Code

_____

Pendant Serial Number

_____

Controller Box Serial Number

_____

Registering the Pendant

To register the Pendant, scan the QR Code in Registration Info (located in Settings).

Enter the unlock code provided by the website. After successfully unlocking the Pendant, you're able to run files from your *SHARK Pendant*.

Quick Guide to Basic Pendant Function Keys

CNC Top View

CNC Front View
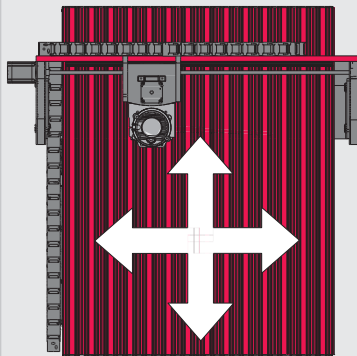
Axis position fields- These three fields show the current X, Y, and Z location of the router bit. Keep in mind that the location is always relative to where the zero points of the X-, Y-, and Z-axes have been set . Pressing on an Axis position field button opens the Axis edit screen for that axis (see page 12 for more information).

NO FILE LOADED
NOT HOMED

0.000   0.000   0.000

| Jog | Max | Up |
| Zero X, Y | Zero Z | Down |

Y+

X-   +X

Y-

SHARK®

World Class. Made in USA.

Apps   Home   Move to Zero
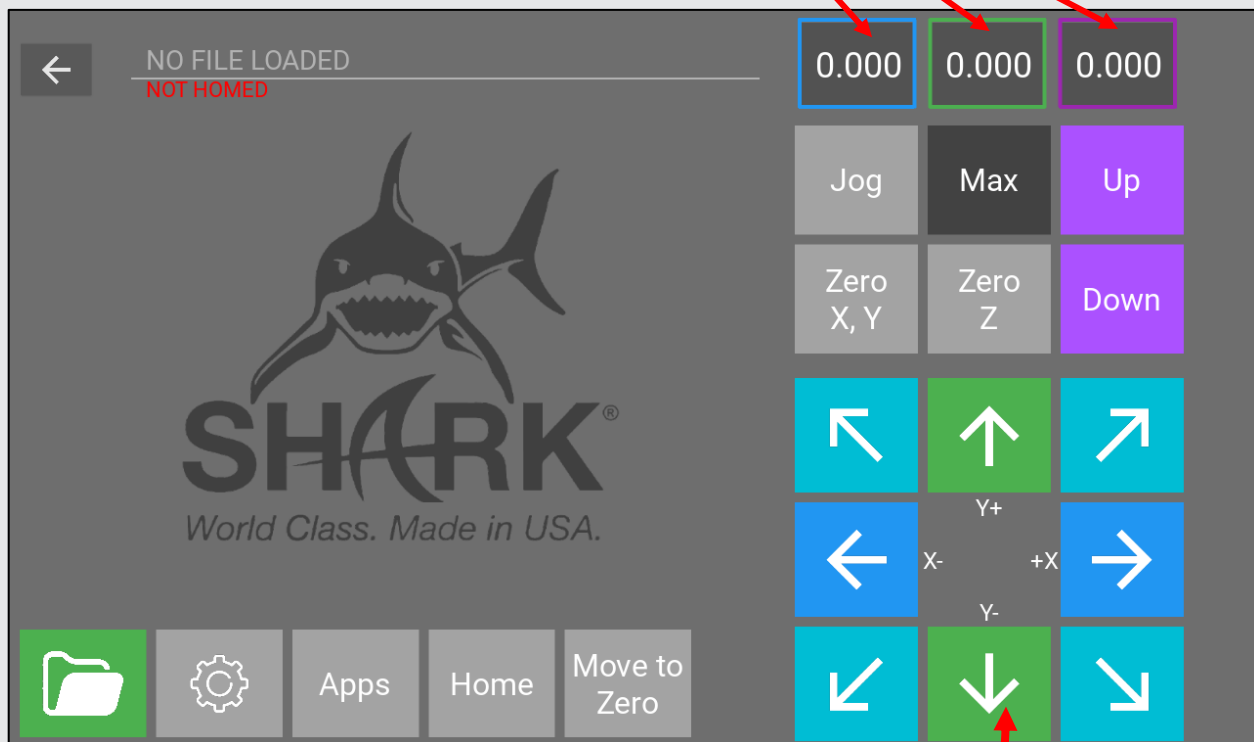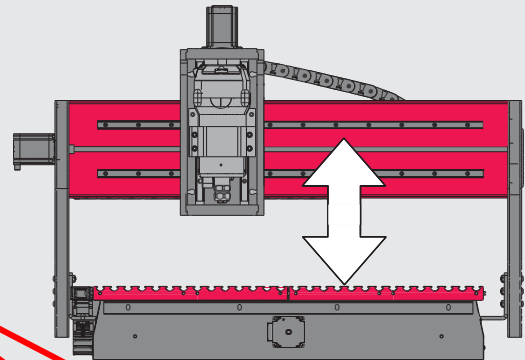
Axis jog buttons – These eight buttons control the movement of the router in the specified direction. Use them to jog the router to set a zero point, move it out of the way while securing or removing material, or move it to a convenient spot to change the bit. Change the speed of the movement by toggling the jog speed button.

Main Screen Function Keys

Work Offset Indicator – Will display the 'Work Zero' after the machine has been homed. The Work Zero is the X/Y Zero set away from the machine's home position.

Home - The Home key uses sensors located on the X- and Y- axes to Home (zero) the machine. The Home position serves as an origin for setting up repeating operations **(see page 9)**.

Slow, Med, Fast, and Max Jog speeds – Press to toggle between jog speeds. You can use the Max speed to quickly jog the unit into a general position and switch to med or slow for the fine control needed for setting a zero point.

NO FILE LOADED			0.000	0.000	0.000
NOT HOMED			Jog	Max	Up
			Zero X, Y	Zero Z	Down
	Apps	Home	Move to Zero	Y+	
			X-	+X	
				Y-	
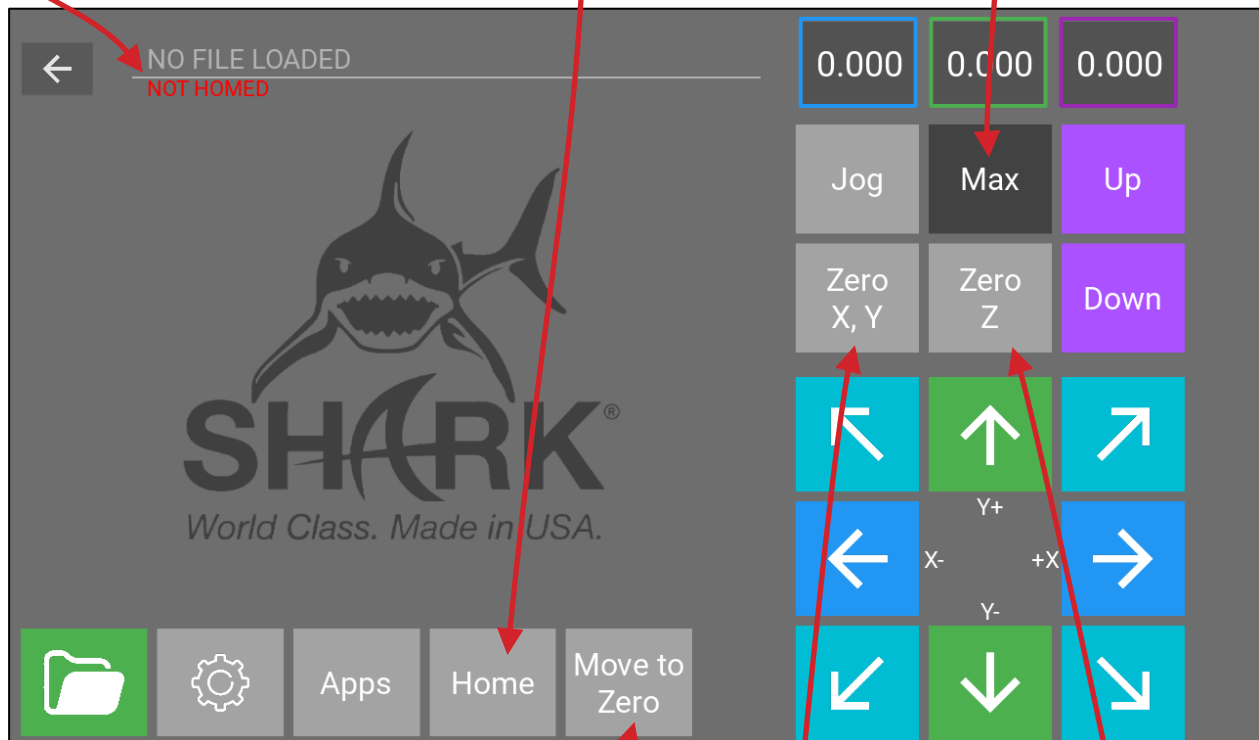
Move to Zero – This button allows you to rapidly move the machine to the recorded X/Y, or XYZ Zero position.

The Move to Zero function is handy for quickly moving the router back into position to start a cut. In addition, if you are in the process of cutting many parts but need to shut your machine down at the end of the day you can bring the router back to the 0,0,0 point. The next time you start your machine the axes will already be zeroed out to that location and you are ready to pick up where you left off

Zero XY– Press this button to zero the X- and Y-axes to the current position of the router. The X- and Y- position fields will reset to 0.000. Use this when positioning the router bit over the datum (XY zero) point on a project.

Zero Z – Pressing this key zeros out the Z-axis at its current position. The Z- position field will reset to 0.00. Use this function to manually set the zero position of the Z- axis.

Load File – Press this button to select a cutting (.tap) file stored in the Pendant's internal storage or USB Drive. **(see page 13)**.

Jog/Step Toggle – Press to toggle between 'Jog' and 'Step' mode.
Step mode will allow you to jog the machine a set distance.

Axis Edit Screen Access – press the Axis Position button to open the corresponding Axis Edit Screen **(see page 12)**

NO FILE LOADED
NOT HOMED

0.000   0.000   0.000

Jog   Max   Up

Zero X, Y   Zero Z   Down

Y+
X-   +X
Y-

SHARK®
World Class. Made in USA.

Apps   Home   Move to Zero
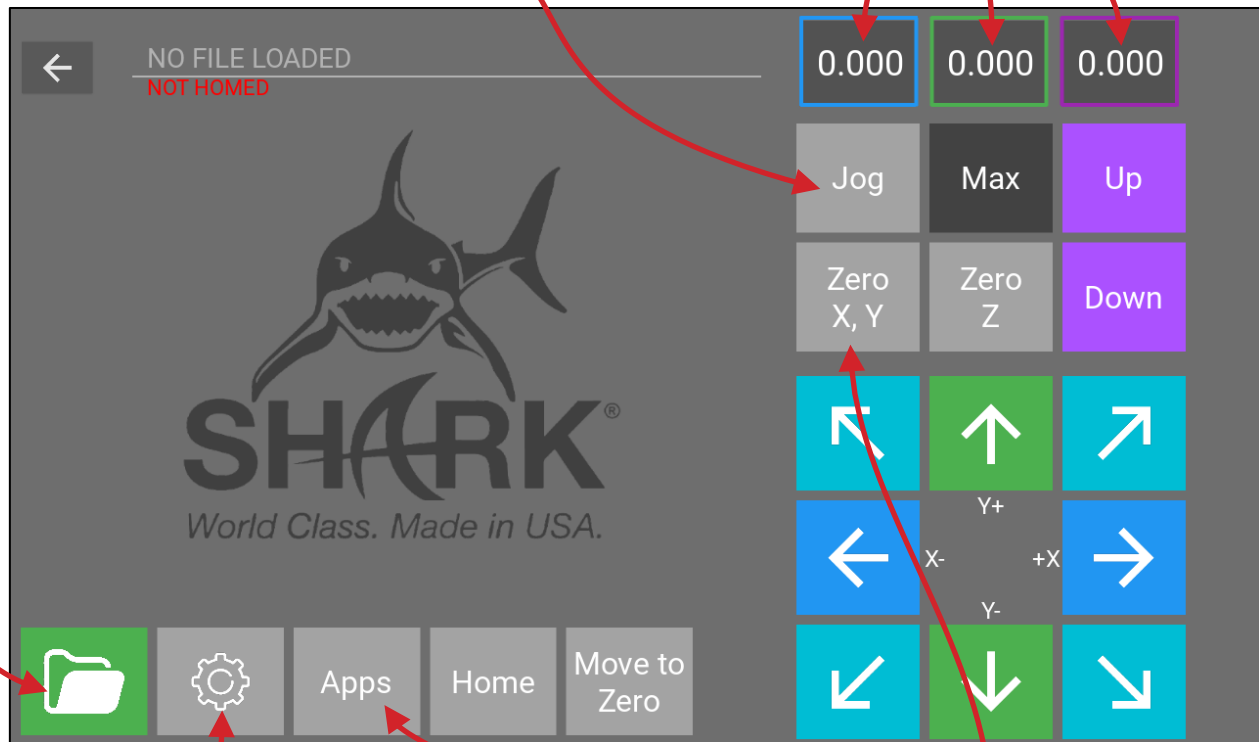
Settings - Press to access the Settings sub-menu. The Settings sub-menu contains options that control how the Pendant functions **(see page 16).**

Apps – Press Apps to view the Apps submenu. It contains special functions for calibrating the operation of the CNC. It's also used with optional accessories **(see page 19)**.

Zero XY– Press this button to zero the X- and Y-axes to the current position of the router. The X- and Y-position fields will reset to 0.000. Use this when positioning the router bit over the datum (XY zero) point on a project.

Homing the machine

The Homing function calibrates and zeros out the X and Y - axes to the machine's Home location in the front-left corner of the table for Shark HD5 series machines and the rear-left corner on a Shark SD120 machine.

NOTE: The SD101 and SD110 do not have the required sensors, so the Home app does not apply to those machines.

X-Y Home can be calibrated each time you power up your Shark or at any time before you run a project file. Using the Homing function provides a consistent, repeatable reference point that is particularly useful when using jigs, or when you need to run a cutting file from a precise location.

To access the Homing function press the Home button on the Main Screen (Figure 1).

Select Home X & Y Axes from the list to set its home location (Figure 2).

The router will home the X axis first, and then the Y axis. A message will show when the process has completed.

Homing the Z axis (only available for HD544 models) can be done by pressing the Home Z Axis button, or the Home X,Y, & Z Axes button in the homing menu.

When homing, the Z axis will move quickly up until the top travel limit is reached. After detecting the travel limit, the Z axis will then position itself 2" below the top travel limit.
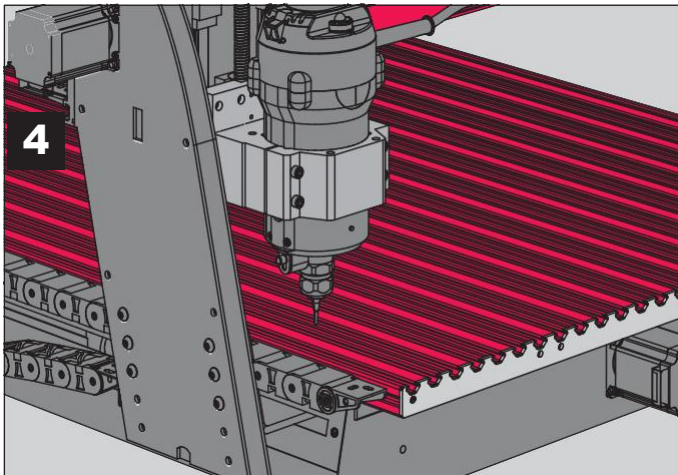
When the Homing routine completes, the Z-axis will be centered over the machine's Home location and the Pendant X & Y Position fields will read 0.000.

Homing X & Y Axes

1
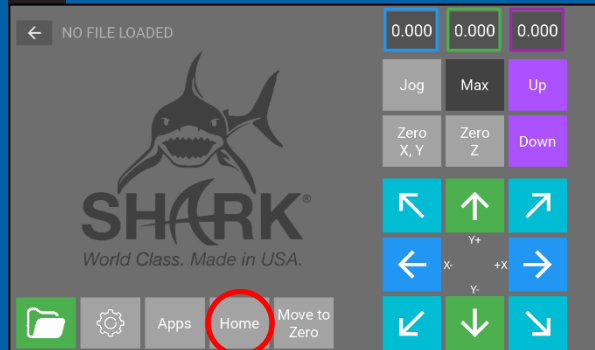
Access the Homing function by pressing the Home button on the Main screen (page 7).

2
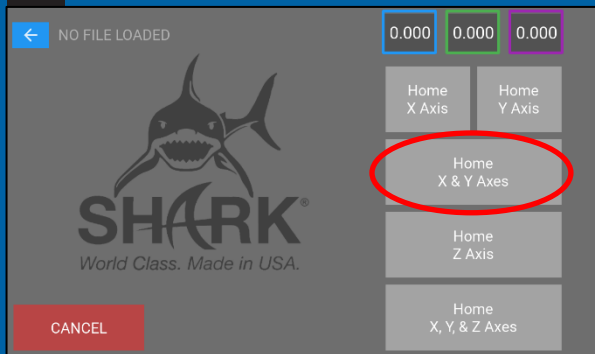
Select Home X & Y Axes to calibrate the zero points for the X and Y-axis home positions.

3
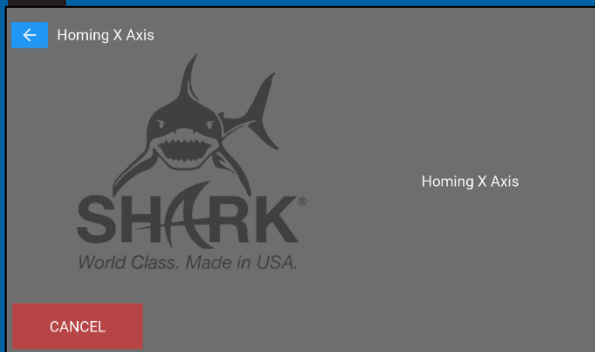
Make sure the router bit is within 4" of the lower left corner in the X- and Y-directions.

Shark Z Zero Touch Plate

The optional Shark Z Zero Touch Plate accessory (see page 32) provides a fast and accurate way to zero the Z-axis to your cutting material. It's a useful upgrade for your Shark CNC router that makes this process easier. Simply place the touch plate on top of the material to be routed before completing the process.

The touch plate assembly includes a ⅜"-thick aluminum disc and a cable assembly. One end of the cable is fitted with an ⅛" plug that fits into the jack on the side of the Pendant. The opposite end of the coiled cable features two leads: an aluminum touch plate, and the other lead has a small magnet that you attach to the router bit. This completes the circuit when the bit touches the plate during the zeroing process.

To zero your Z axis with the Touch Plate:
Press the Zero Z button on the Main Screen (Figure 1), then press Zero Z with Touch Plate (Figure 2) to confirm touch plate connection. Once the connection is confirmed, you may then proceed to record your Z position.

Raise the touch plate to touch the router bit. While the plate is touching the bit, you should see a blue 'Connected' box as shown in Figure 3. Replace the touch plate below the bit and press Proceed to begin the movement of the router in the Z- axis.

The bit will move downward until it contacts the touch plate (Figure 4) and immediately raise to a safe height.

Zero with Touch Plate

1
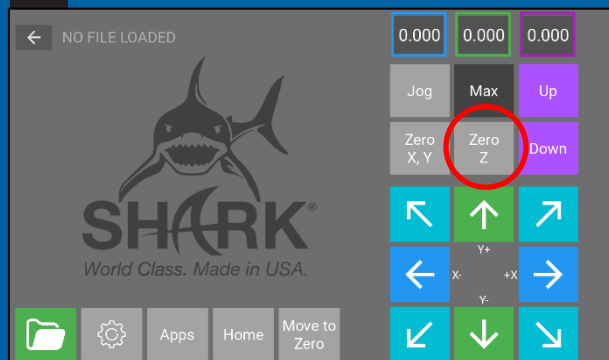
Press the Zero Z button on the Main screen to access the option to zero with the touch plate.

2
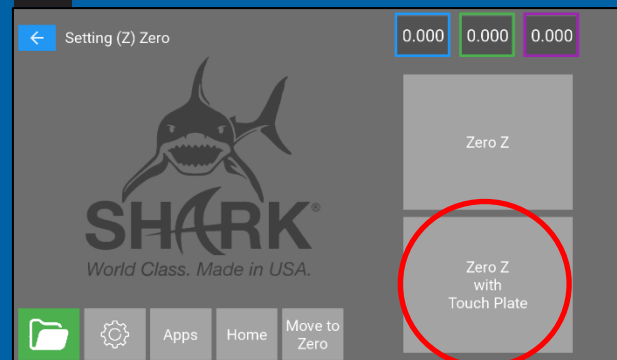
The Run Z Calibration option begins the routine to zero the Z- axis using the touch plate.

3
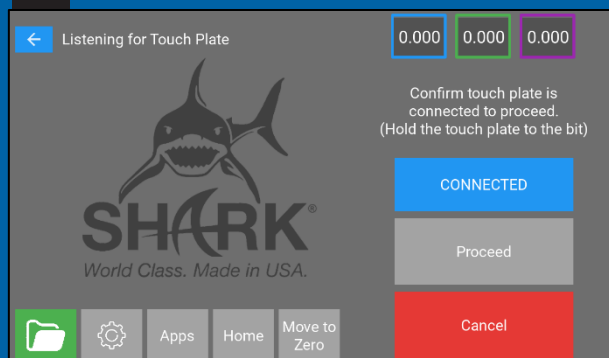
Raise the touch plate until it touches the router bit to confirm a proper connection.

4
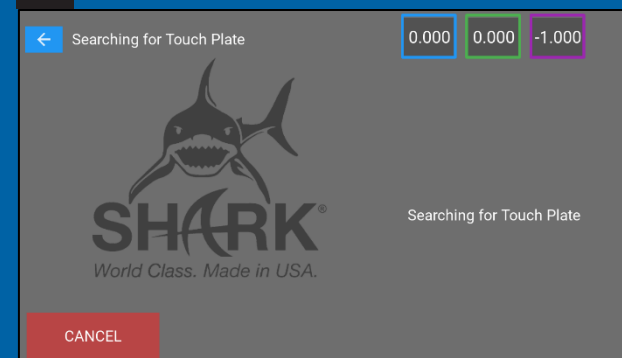
Upon pressing proceed, the router will slowly descend until the Touch Plate is detected or the operation times out.

Z Zero Touch Plate *continued*

Touch plate function settings (Speed, and Touch Plate Thickness) can be adjusted by selecting 'Touch Plate Calibration' in the Apps submenu.

Adjust Touch Plate Options
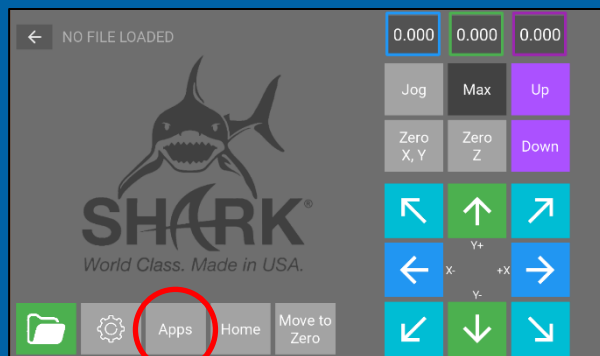
To adjust options for the Touch Plate function, tap the Apps button on the Main Screen, and then select Touch Plate Calibration.
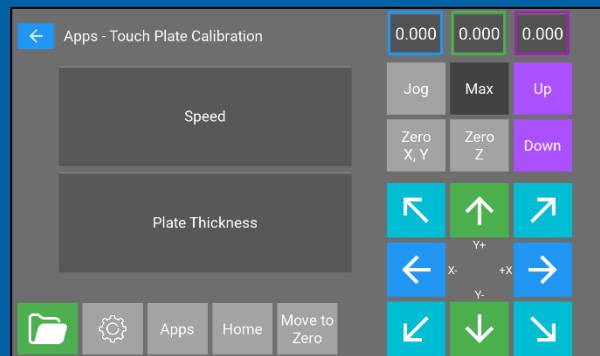
Choose the option you would like to edit.
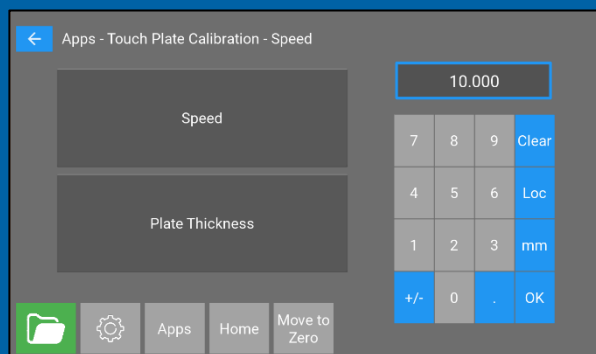
When the keypad is displayed, the active option is displayed at the top of the screen.
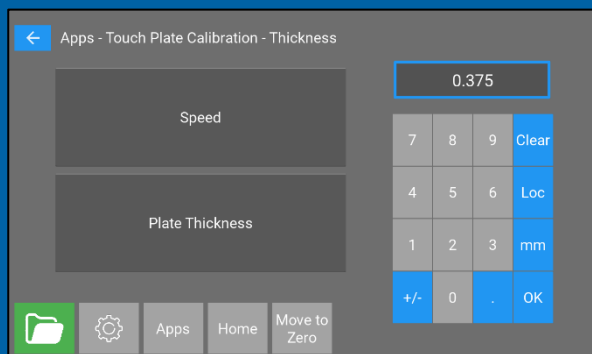
Enter the desired value in the keypad and press OK to submit.

Axis Edit Screens
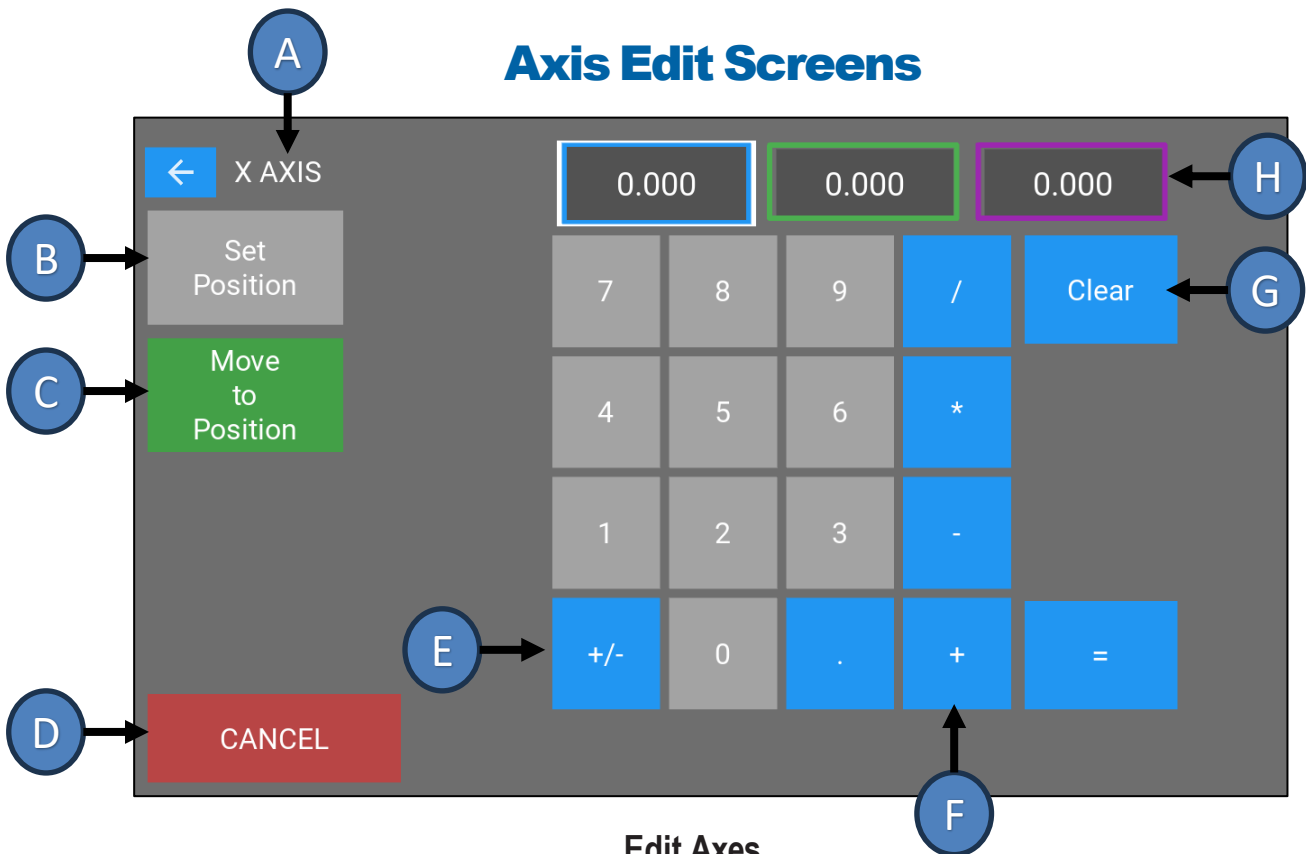
Edit Axes

On the main screen of the Pendant, pressing the Position field for the X-, Y-, or Z-axis at the top of the screen allows you to make adjustments to the selected axis. You can also calculate values using the numeric and math function keys.

A. Active Axis: Displays the axis that is currently open for editing. The active axis can be switched by tapping any of the other position fields.

B. Set Position: Sets the new location entered in the Position field. This also changes the value on the Main Screen.

C. Move to Position: Moves the active axis to the value input.

D. Cancel: Stops movement

E. +/- button: Toggles the position field between a positive and negative value.

F. Function keys: Similar to those on a typical math calculator

G. Clear: Clears (deletes) the information in the position field. Displays the current position of the active axis.

H. Position field: Displays the current or edited position of the axis. Use the keypad to edit this field.

Edit A Axis

Editing the A-axis uses the same functions as above.

This page can only be accessed when 4th Axis Controls are enabled in the Settings – Accessories menu.

Pendant Workflow

Routing projects with your *Shark CNC* consists of a few basic steps, a summary of these steps is listed below and detailed on the following pages:

1. Save toolpaths from the *VCarve* design file to a USB flash drive.
2. Insert the flash drive into the Pendant.
3. Mount the project material to the bed of the CNC.
4. Install a router bit.
5. Zero the X- and Y-axes.
6. Zero the Z-axis.
7. Press the Load File button on the Pendant to access the toolpath file.
8. Select the toolpath file.
9. Verify the setup parameters shown on the Pendant screen.
10. Press Start to begin routing your project.

1. SAVE TOOLPATH FILES. After completing the project design and generating the toolpaths in *VCarve*, save the toolpaths to a USB flash drive so you can load the files into the Pendant. To do this, select the toolpaths you wish to save then select the Post Processor that matches your machine. For the *Shark CNC* router select the *Next Wave CNC* (*.tap) post processor (Figure 1). The post processor converts the *VCarve* toolpath selections into instructions for the CNC to rout the project. 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 button and saving the file to a USB flash drive (Figure 1).

2. INSERT USB DRIVE. To enable the Pendant to load the toolpath (.tap) file, switch on the control box and insert the USB drive into the USB port on the left side of the Pendant. You'll select the required toolpath later.
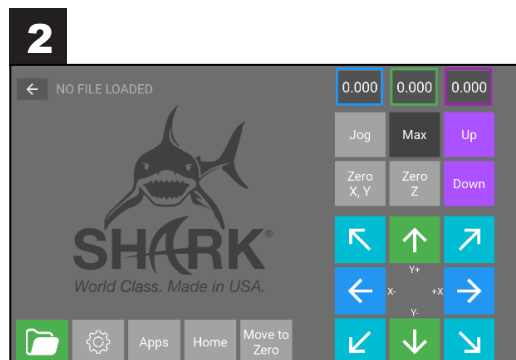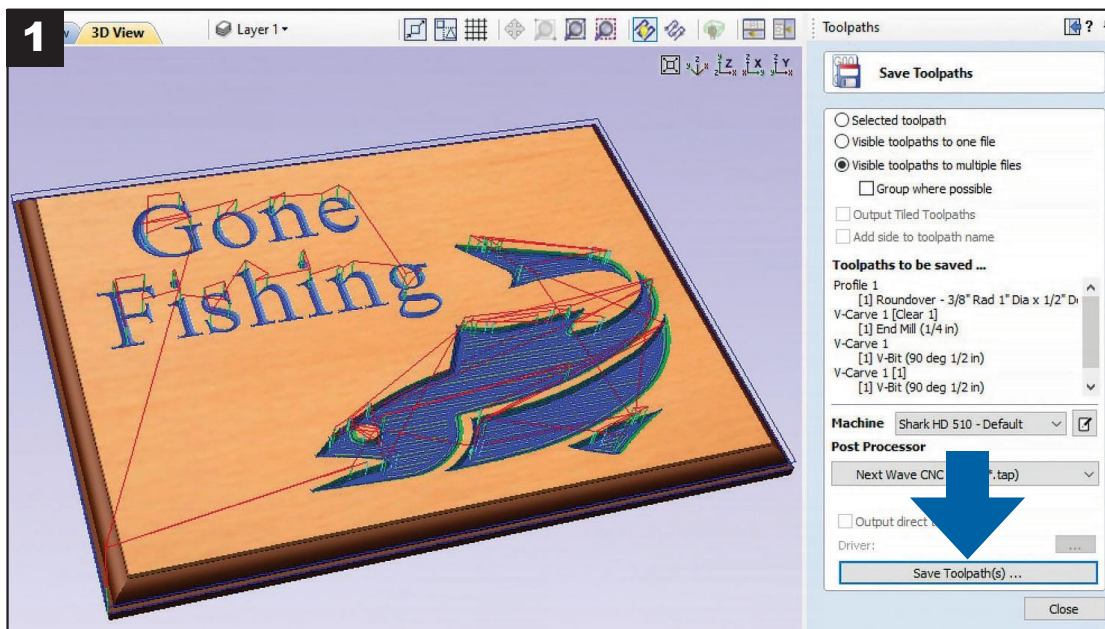
3. MOUNT THE WORKPIECE. To set up the origin of the X-, Y-, and Z-axis on your workpiece, first mark the XY datum (zero) position on your material. (Figure 3). The marked datum point should match the location on the workpiece you used in your *VCarve* design file. In our example, the XY datum point is specified as the center of the material.

Securely clamp the workpiece to the bed of the CNC (Figure 3). Use enough clamps to ensure the workpiece won't shift during the routing process. It's also important to position the clamps so they're not in the way of the toolpaths to be routed into the workpiece. A little careful planning here will go a long way toward ensuring success with your project.

4. INSTALL THE ROUTER BIT. Once the workpiece is secured, properly install the router bit you specified in *VCarve* for the toolpath you'll be using (Figure 4).

5. ZERO THE X- AND Y-AXES. You'll find an overview of the main Pendant screen on pages 6-8. Use the four X and Y buttons (Right X+, Left X-, Away Y+, Near Y-) of the Pendant to jog the router close to the XY datum mark. Notice the values in the Position boxes at the top of the screen change values as you jog the router. Change the jog speed to Slow and position the router bit directly over the datum mark (Figure 5). Press the "Zero XY" button to zero the X- and Y-axes. The values for the X- and Y-axes should read 0.000 in the position boxes.

6. ZERO THE Z-AXIS. 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 'Down' button to gradually lower the bit (Figure 6). When the bit "grabs" the paper, stop. Press the "Zero Z" button. The Position value for the Z-axis should read 0.000. Now, all three axes are calibrated.

Raise the router bit approximately ½" to prepare for cutting the project.

NOTE: A fast, easy, and accurate way to zero the Z-axis is with the optional *Z Zero Touch Plate* accessory. See page 10 and learn more at *NextWaveCNC.com*.

CNC Setup

3
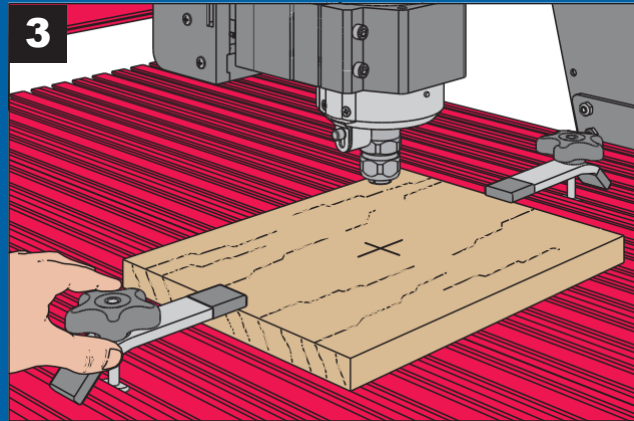
Zero the X and Y axis first by jogging the bit until it's positioned over the center of the workpiece.

5

4
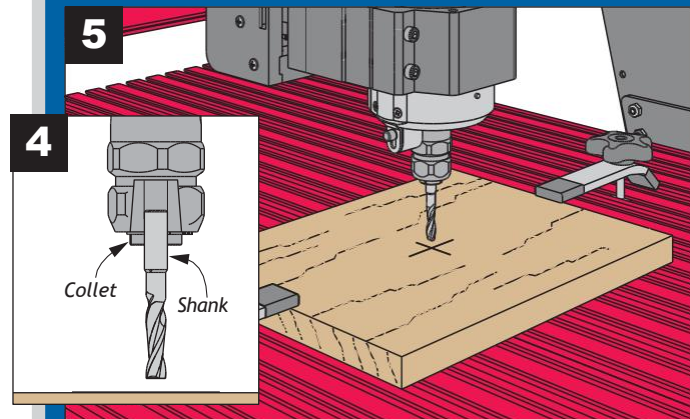
Collet

Shank

Insert the bit so that a small amount of smooth shank shows between the collet and the cutting edges, but also make sure the bit isn't bottomed out in the collet.

6
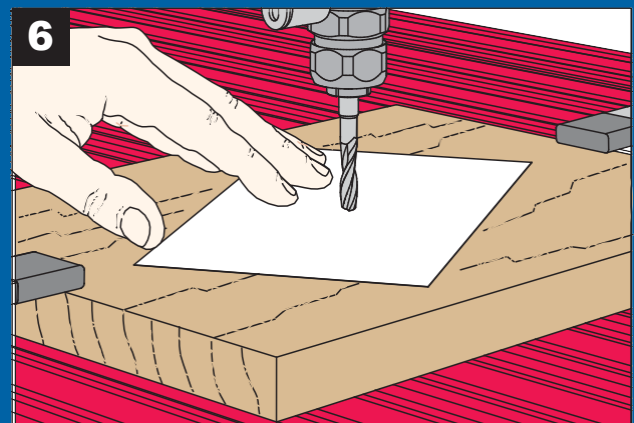
Zero the Z axis by lowering the bit until it just touches a piece of paper on the workpiece.

7. PRESS THE USB BUTTON. With a USB drive inserted into the Pendant, press the Load File button (Figure 7) to open the file selection menu.

8. SELECT THE PROJECT FILE. From the displayed list, select the project .tap file you want to cut (Figure 8).

9. VERIFY JOB SETUP. The next screen (Figure 9) displays an image of the GCode with the option to view the GCode (File Info), Virtual Zero Setup, and Continue. The name of the file will be displayed at the top of the screen. Press the Continue button to proceed to the Running screen.

10. PRESS START! Review the safety check lists in your CNC manual, then press the Start button (Figure 10). Your *Shark CNC* will automatically turn on the router and begin cutting the project.

If your project has more than one toolpath file, repeat steps 4 through 10 for the remaining toolpath (.tap) files for the project. When all of the .tap files have been run, remove the project, sand off the rough edges, and apply a finish.

Review, Then Cut

7
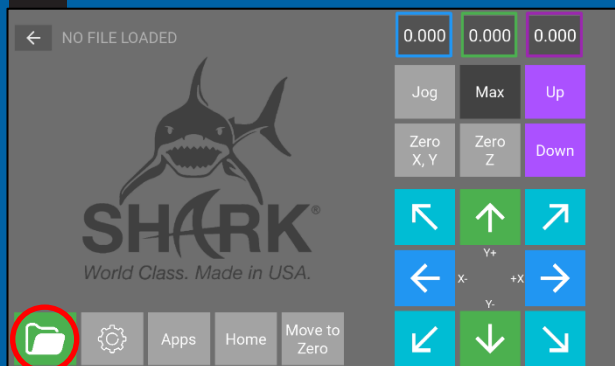
After inserting the flash drive into the Pendant, press the Load File button to select the project.

8
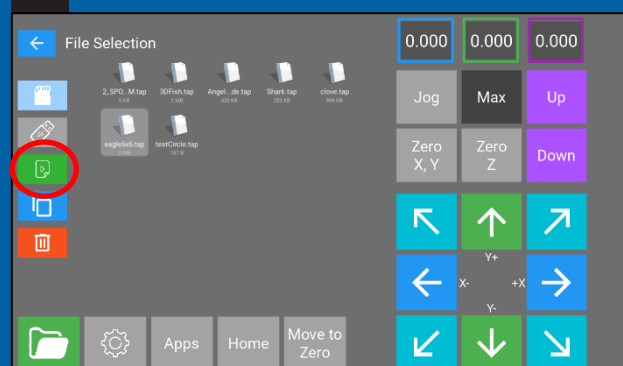
From the list of available projects, choose the appropriate .tap file you wish to use. Then tap the Open File button.

9
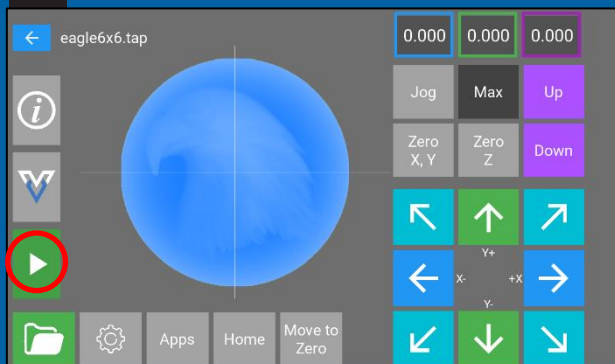
Verify all of the set-up information for this job, review the safety checklist, then press the Start button.

10
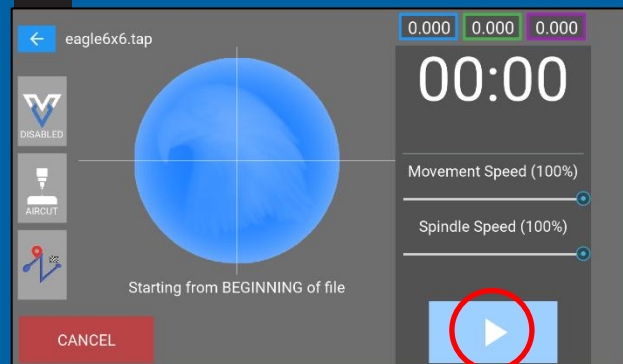
To begin the CNC operation, press the Start button. If needed, the Stop button shuts down the CNC.

Settings Menu
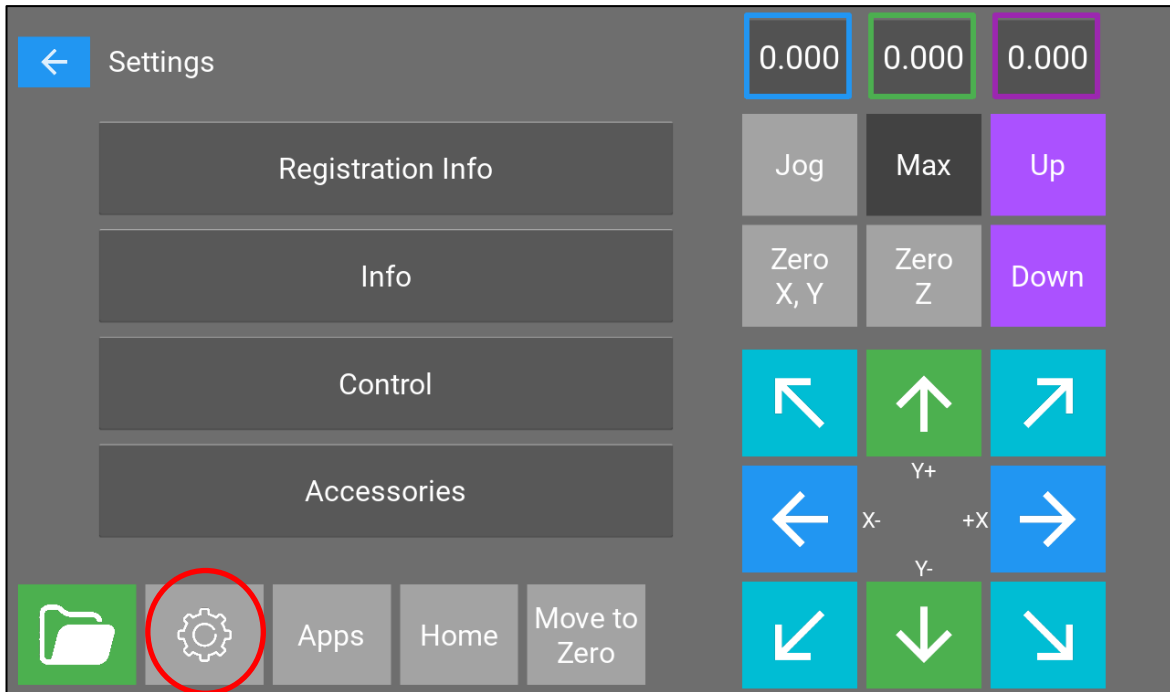
Settings Menu Screen: Press the Settings button on the Main Control screen to open the Settings submenu. The Setup menu is broken up into categories that contain options for controlling specific Pendant functions. Press a menu item to open the submenu for that option. To return to the Main Control Screen, press the back button in the upper left corner.
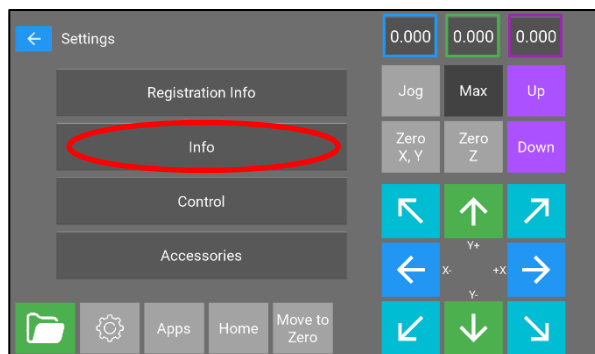
Info: Press Info to view your machine's: Serial number, Model Number, Firmware Version, etc.
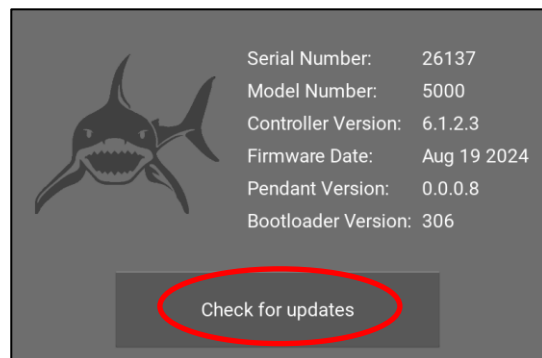
You may also update the Pendant's software from this screen by connecting a USB with the proper update files and tapping 'Check for Updates'.
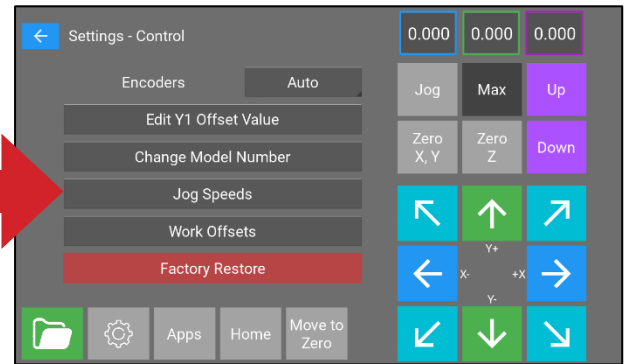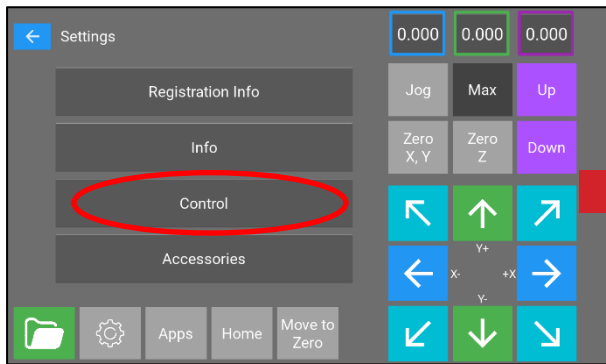
Control: Press to adjust various settings related to the operation of the machine. Such as Jog Speeds, Model Number, Safe Z Height, and Encoder settings.

Encoders: Adjust the behavior of the closed-loop encoders.
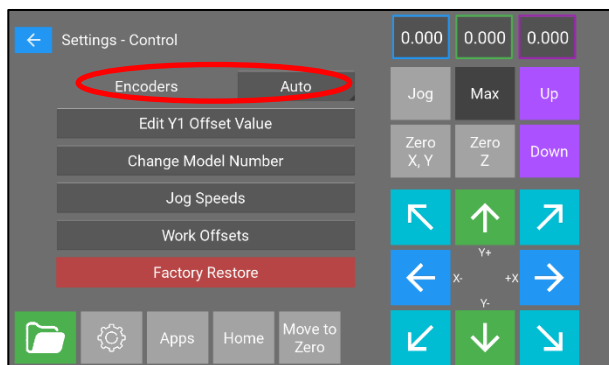
Encoder Options	
Auto (Default)	Sets Encoder behavior to Model number default
Force ON	Enables all encoders (if applicable)
Force OFF	Disables all encoders (if applicable)
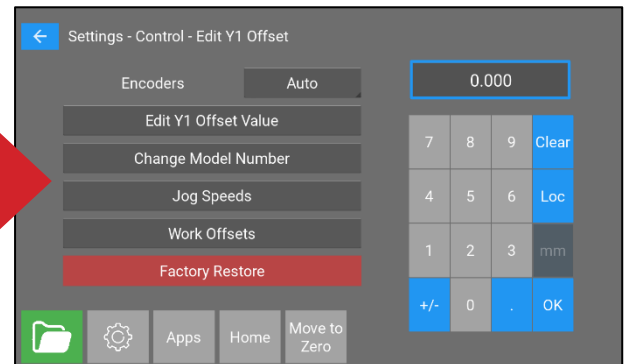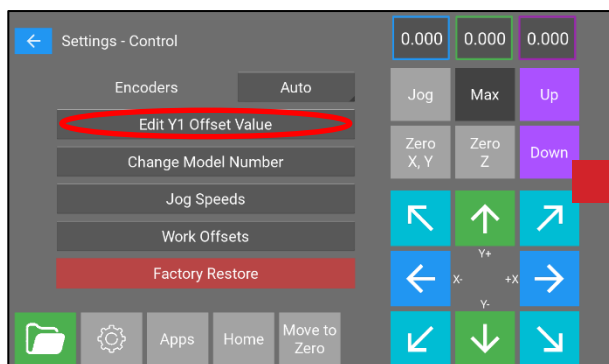
Edit Y1 Offset Value: Adjust the offset of the Y1 axis (HD544). This value is automatically adjusted when using the Square Machine App.
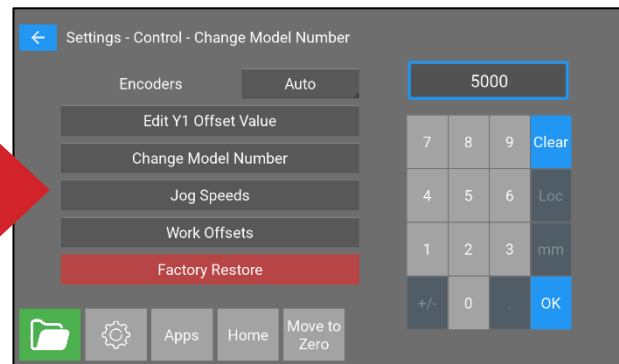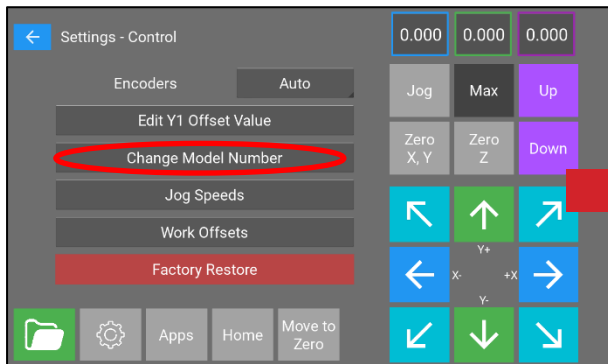
Model Number: This function controls several settings on your *Shark CNC* such as motor rotation and XY machine orientation. Verify the model number for your machine using the chart at right. Verify this setting after performing a firmware update for the Control Box or Pendant.

Machine Type	Model Number
SD101	400
SD110	102
SD120	202
HD500, 510, 520	4003
Shark II, HD4	4004
HD544	5000
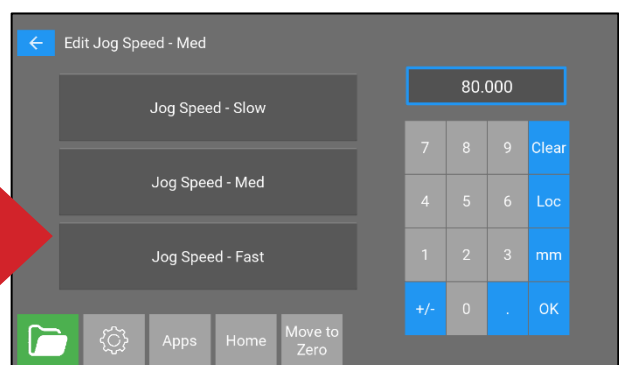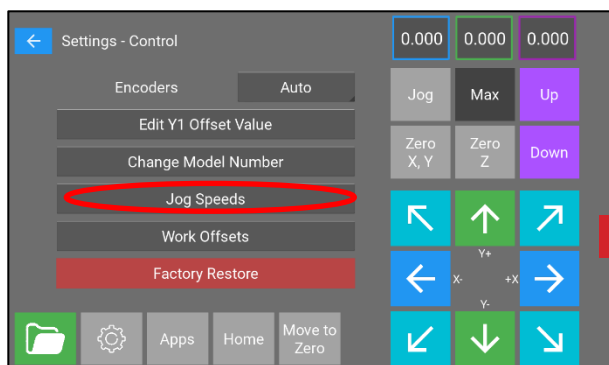
Jog Speeds: Allows editing of the Fast, Med, and Slow Jog Speeds. Choosing an option opens a numeric pad for entering a new value. Press OK to save the setting.
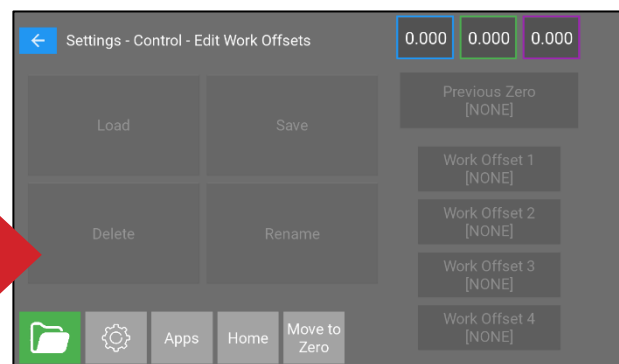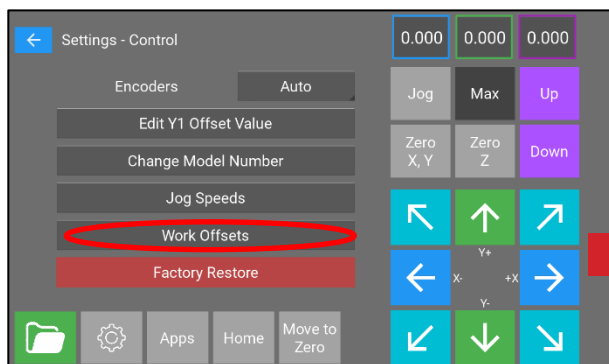
Work Offsets: Allows the saving, naming, and loading of multiple X/Y Zero positions that have been saved after homing the machine after powering on.

The machine must be homed prior to saving, or loading any X/Y Zero positions made separately from the machine's Home position.

Apps Menu
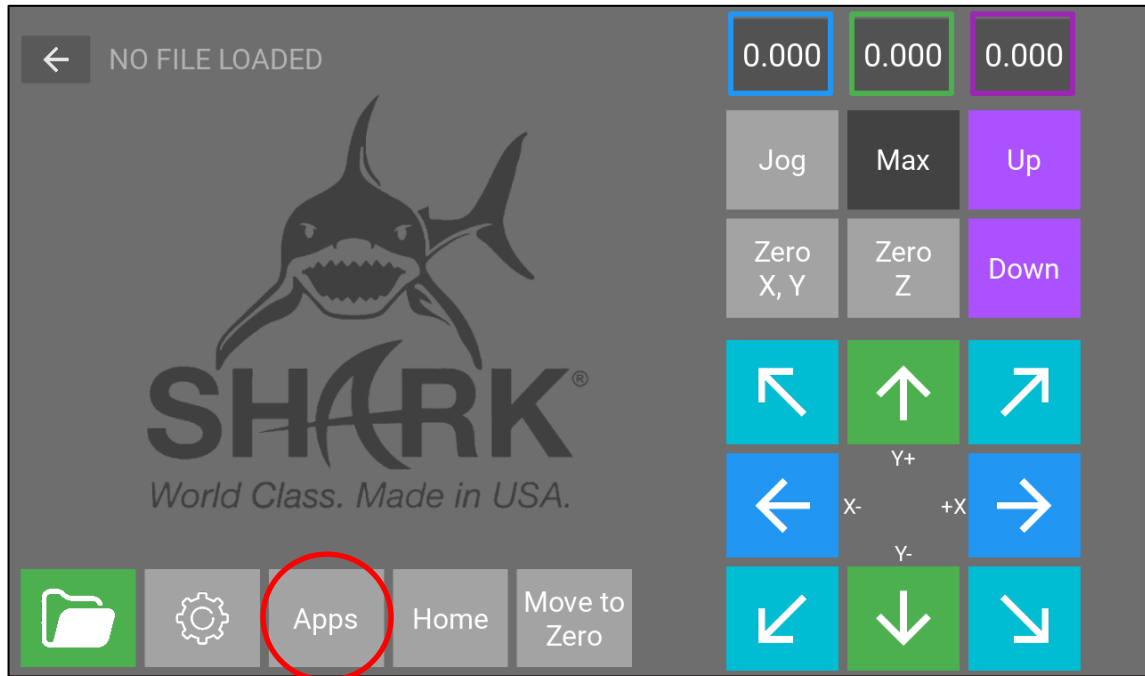
Apps Menu Screen: The Apps submenu contains special functions for calibrating and operating your Shark CNC and optional accessories. Select an item from the list of apps to open the settings screen for that option. Press the Back button to return to the Main Screen.
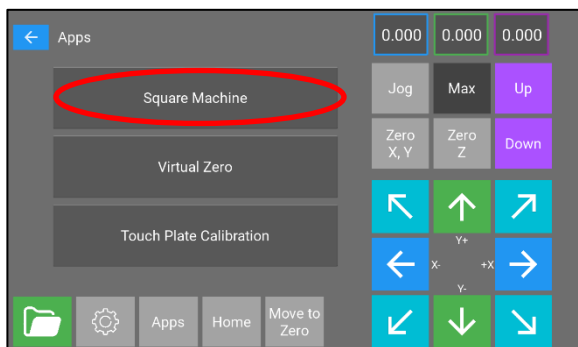
Square Machine (HD544): Select this app to automatically adjust the Y1 offset by measuring the distance between two points made by the machine (see next page).
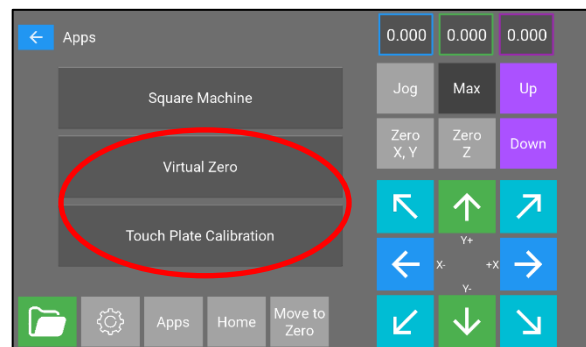
Virtual Zero / Touch Plate Calibration: Select these options to adjust settings for the Virtual Zero App (see page 23) and Touch Plate.

Square Machine App
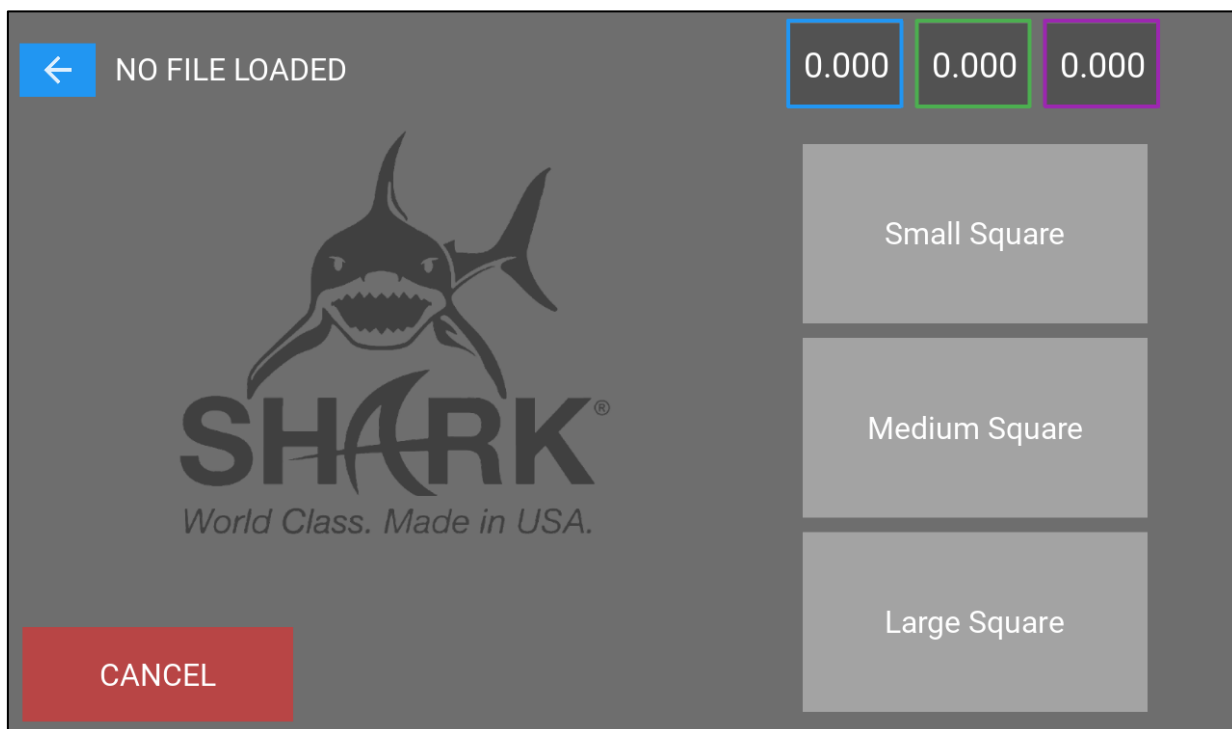
Square Machine (HD544 ONLY): This app will determine whether your HD544 machine's Y axis motors are positioned out of square. A Touch Plate will be required for this application.

There are three squaring procedures representing lengths travelled by the X and Y axes:

Small – 10.606"
Medium – 21.213"
Large – 42.426"
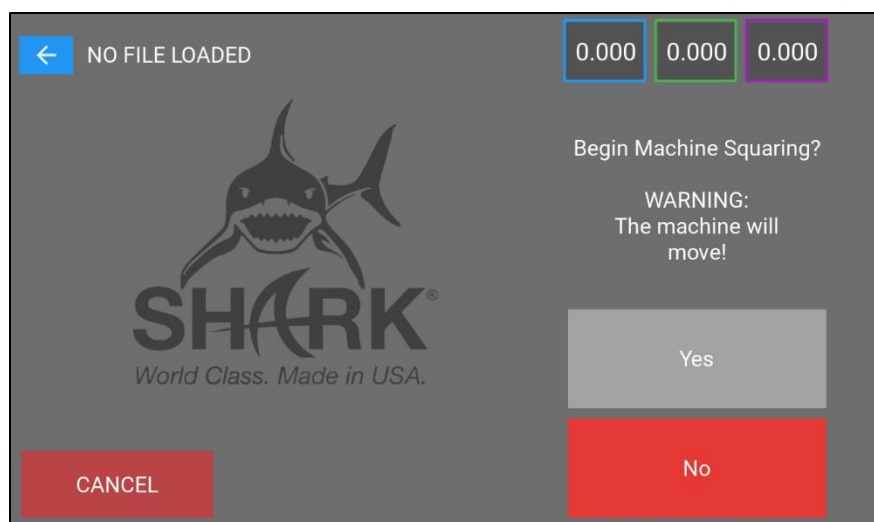
Once a procedure length is chosen, the Pendant will ask for confirmation to start the procedure.
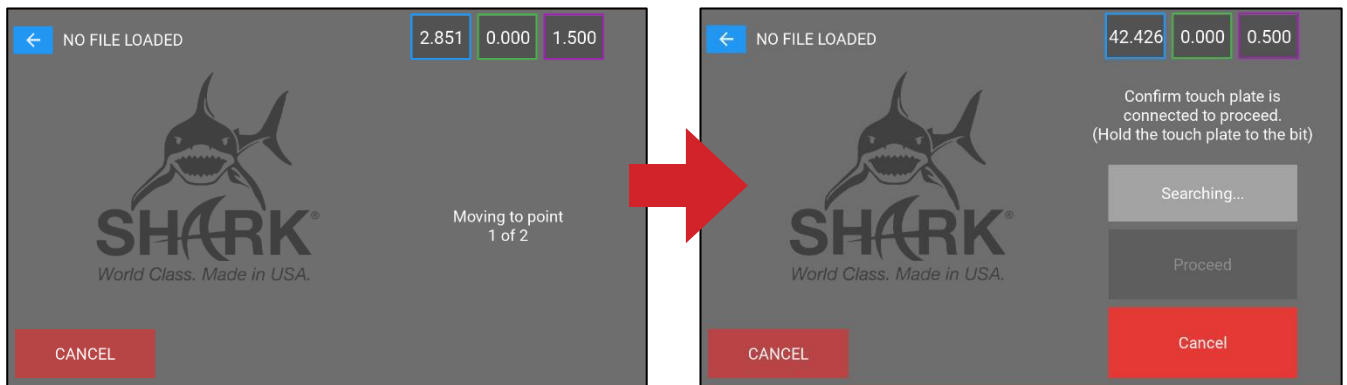
Upon confirmation, the machine will immediately move to the first point in the procedure.

When the machine is in position, the Pendant will then wait for a Touch Plate to record the height of the marking position.

Lift the Touch Plate to the router bit to proceed, the machine will briefly lift to acknowledge the Touch Plate before descending. Once the machine lifts, place the Touch Plate onto the material.
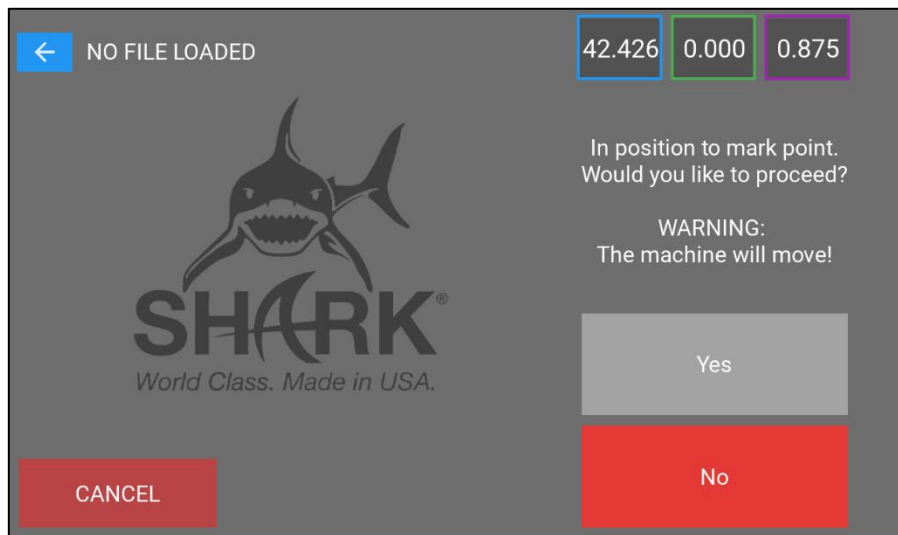
After measuring the marking area's height, the pendant awaits confirmation.
Press the Yes button to proceed. The machine will then:

- Slowly descend to indent the material
- Mark the point
- Move to the next point
- Repeat the process

Square Machine (cont.)

After marking both points, the gantry and router return to the starting position.

NO FILE LOADED   41.000 25.000 1.500

Movements Complete.

CANCEL   OK

Check Distance

0.000

Measure, and enter the distance between the two marked points.

7	8	9	Clear
4	5	6	Loc
1	2	3	mm
+/-	0	.	OK

CANCEL
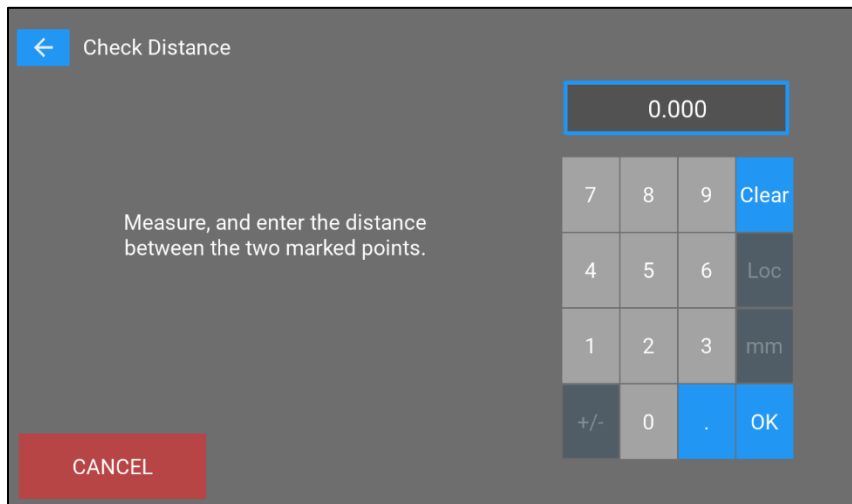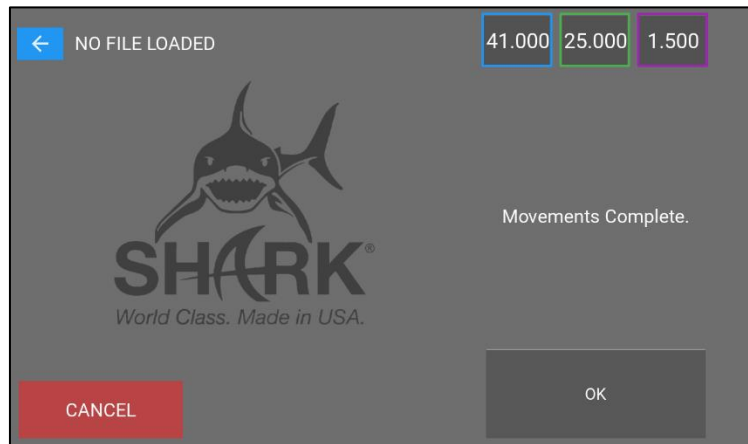
Measure the distance between the marks made by the machine and enter the value into the keypad shown.

The Pendant will use this measurement to determine the offset needed in the Y1 motor to square the machine Gantry to the Y axes.

NO FILE LOADED

In order to apply squaring adjustment, the machine must be homed.

Home the machine now?

Yes

No

CANCEL
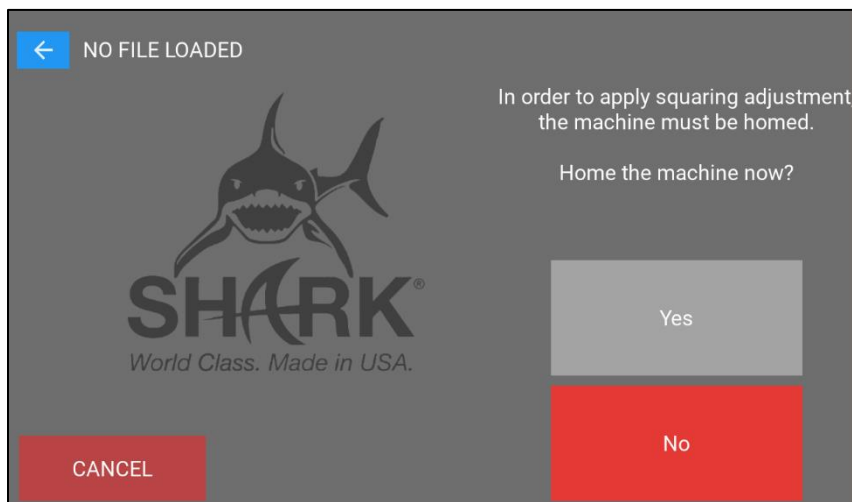
To apply the calculated offset to the Y1 motor, the machine will need to be homed.

Upon confirmation, the Y axis will start the homing process with the offset applied.

Virtual Zero App

The Virtual Zero app makes adjustments for project material that may not be flat. This app allows you to define five Z zero locations to account for the uneven material: each corner and the center. It maps the uneven surface of the material and makes adjustments to the Z-height values in the .tap file.

The Virtual Zero profile automatically makes the adjustments to every toolpath for the cut file when different router bits are needed to complete the project. The Virtual Zero map remains in the Pendant memory until you reset it or turn off the control box.
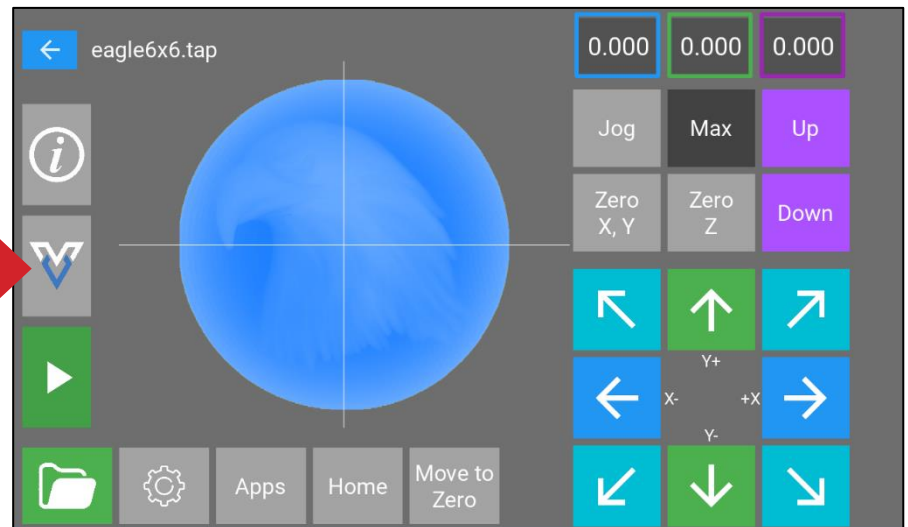
Mapping the surface of your material with Virtual Zero requires several steps, so, take some time to read through the steps and practice with your machine before setting up to cut your project.
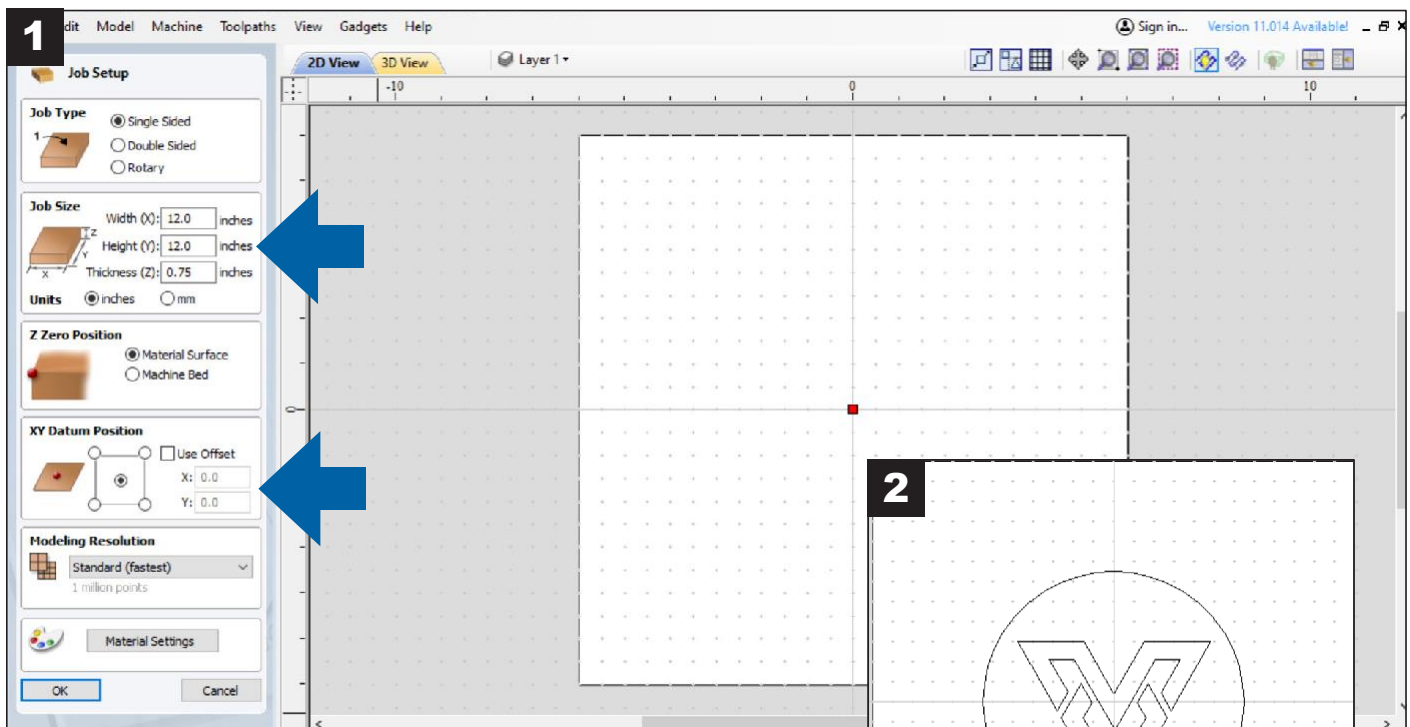
To prepare the design to use the Virtual Zero function, set the Z Zero Position to the Material Surface and the XY Datum Position to the center of your project (Figure 1).

Create your project design as you normally would.

Create and Save the project toolpaths to a USB thumb drive. In the example in Figure 3, two toolpaths are saved: one for carving the letter using a 90° V-bit and another toolpath to cut the circle using a straight bit.

Setting Up Virtual Zero

4
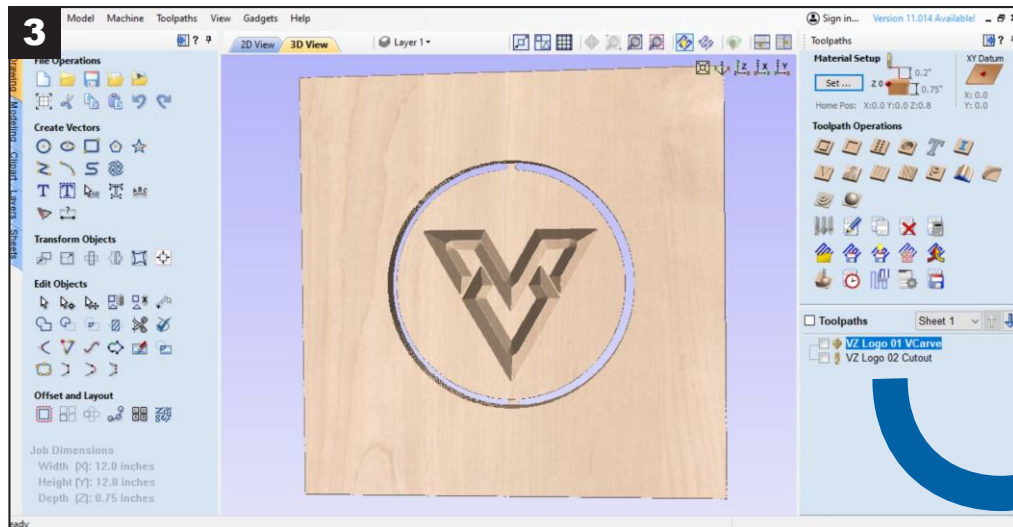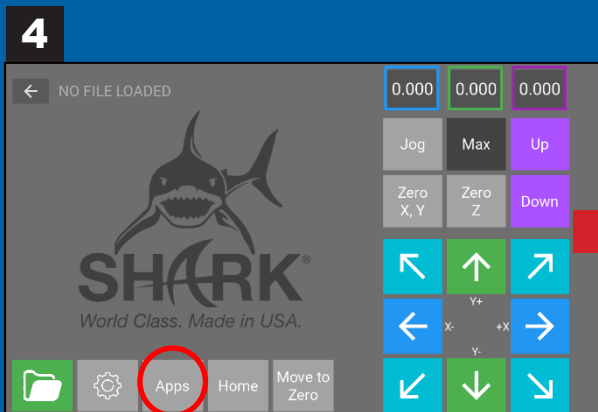
Press the Apps button and select the Virtual Zero option to display the Virtual Zero settings screen (Figure 5).

5

6
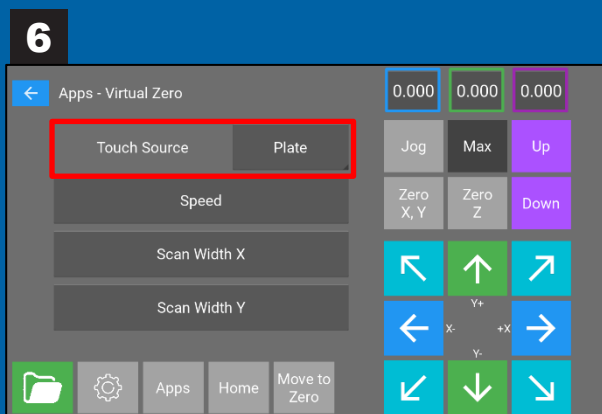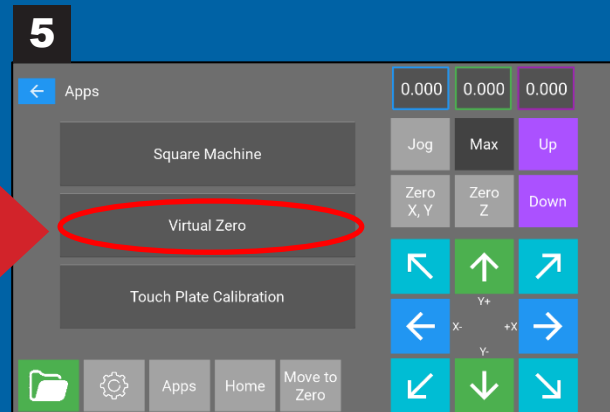
The Touch Source defines the method used to set the Z zero point. Many *Shark CNC* users will use the *Shark Z Zero Touch Plate*. You may also use the optional digital probe accessory.

7
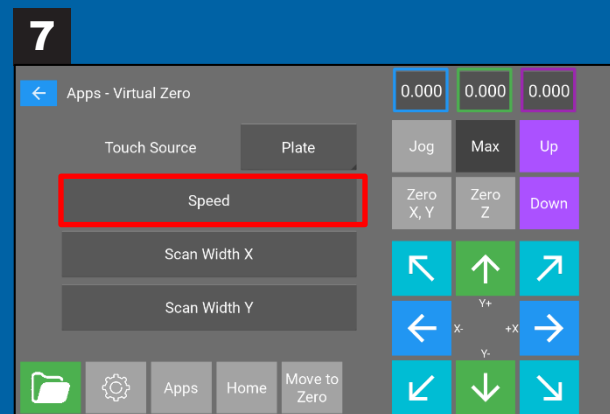
The Speed setting controls the Z-axis plunge rate. The default setting of 10 works well for most situations. Change it by pressing the settings field in the upper right and enter a new value using the numeric keypad.

Setting Up Virtual Zero *continued*

8
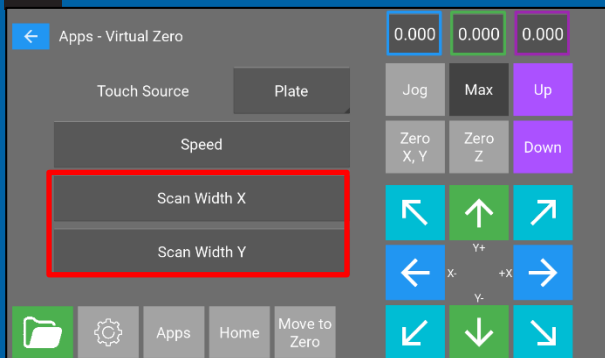
The scan area defines where on the material you will be setting the five Z zero points. You can set this to any size, but it's usually best to limit it to the approximate size of the cutting area. You can also set points that best represent the
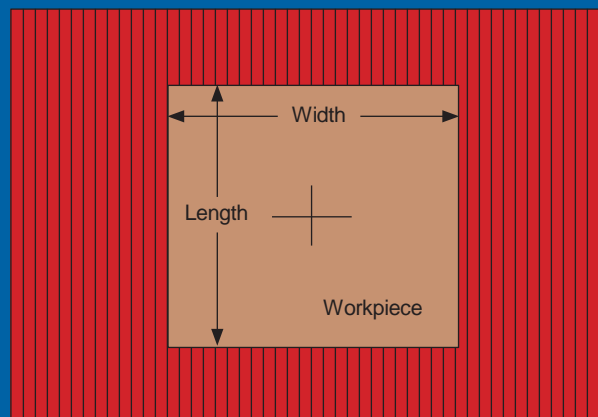
area that will be carved. Another option is to bypass this step and use the File Specifies width and length derived from the *VCarve* cutting file.

9
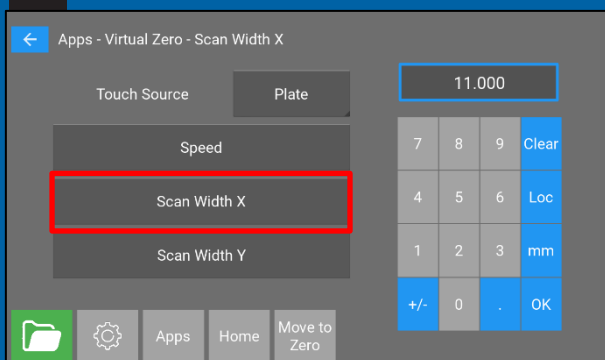
After choosing "Scan Width X," enter the width in the upper right hand field using the numeric keypoad, then press OK to continue.

10
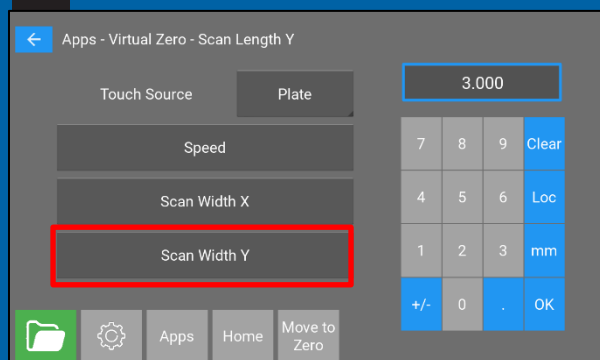
Repeat the process for the "Scan Length Y" option.

11
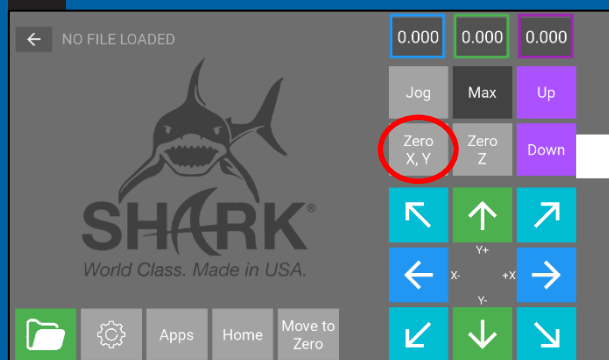
Mark a center point on the workpiece. Install the appropriate router bit. Move the router bit to the center mark. Press Zero XY to zero the X- and Y-axes to the center of the material.

25
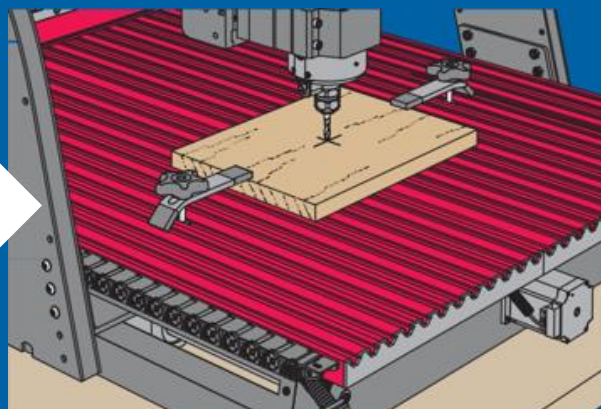
Setting Up Virtual Zero *continued*

12
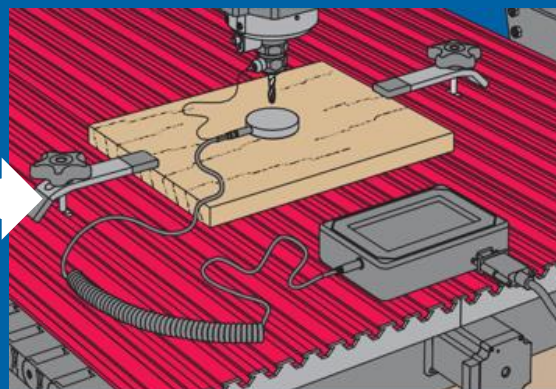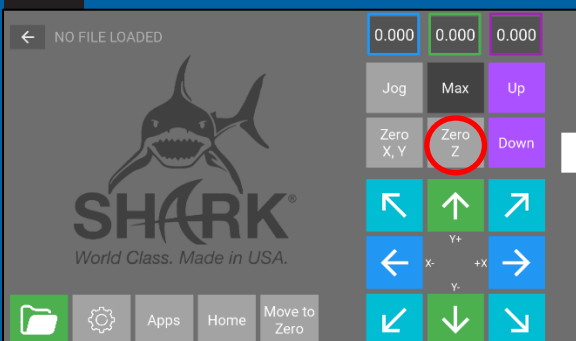
Zero Z with Touch Plate to zero the Z- axis to the top of the material.

13
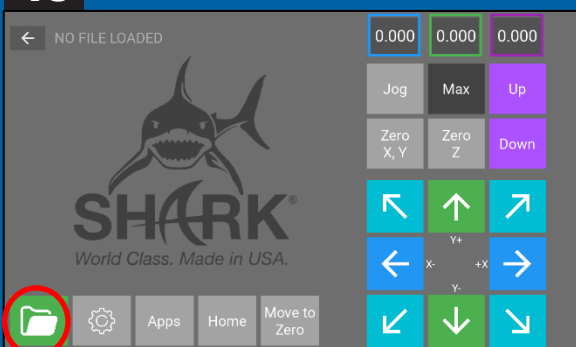
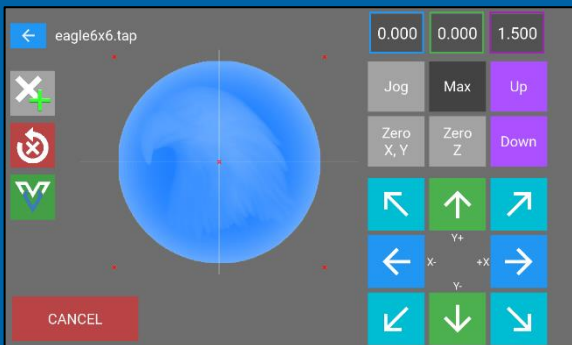Press the Load File button to bring up the list of .tap files (Figure 14).

14
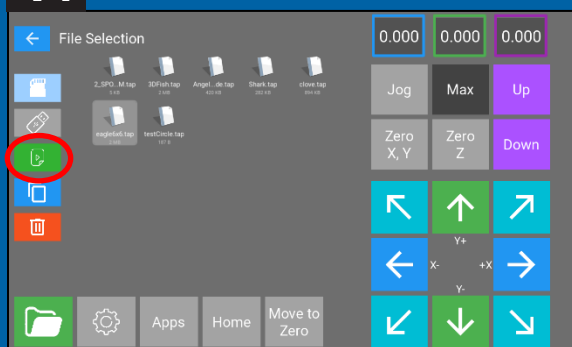
Select the toolpath file that you want to cut and open it.

15
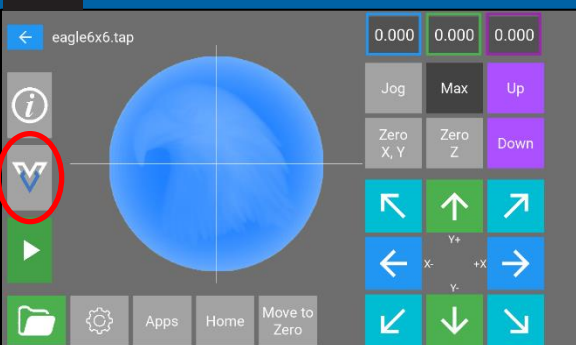
To define Virtual Zero coordinates for your file, press the Virtual Zero button located on the left (Figure 15).

Virtual Zero uses the Z Zero Touch Plate routine to capture the scanned points. Virtual Zero will go through the Z Zero Touch plate routine five times to capture the four corners and the center point of the area chosen to scan.
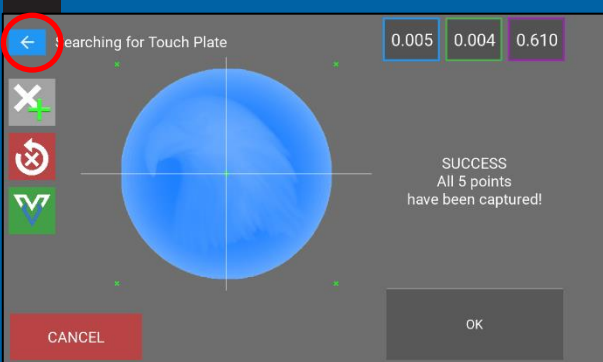
Setting Up Virtual Zero *continued*

17
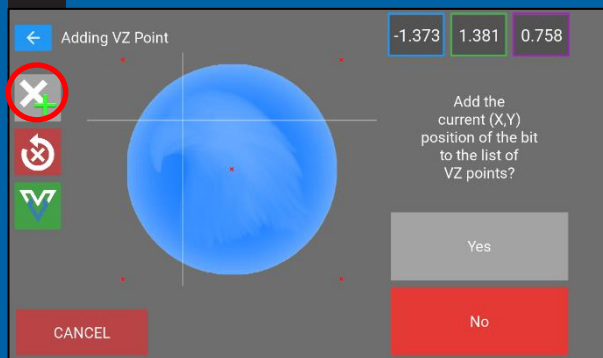
You may also add extra Virtual Zero points to record by pressing the Add Point button (shown above).
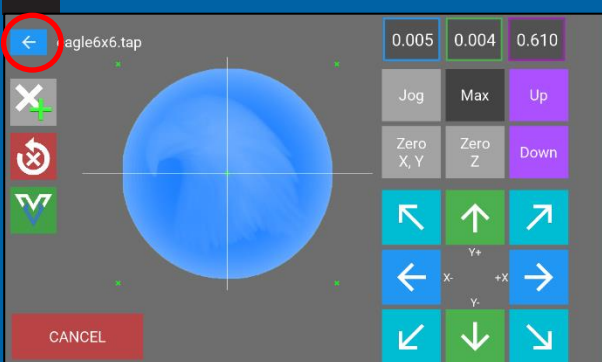
18
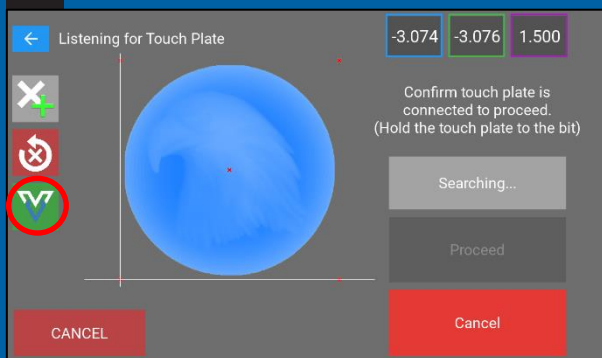
Virtual Zero uses the Z Zero Touch Plate routine to capture the scanned points (Figure 18). Virtual Zero will go through the Z Zero Touch plate routine for each point.
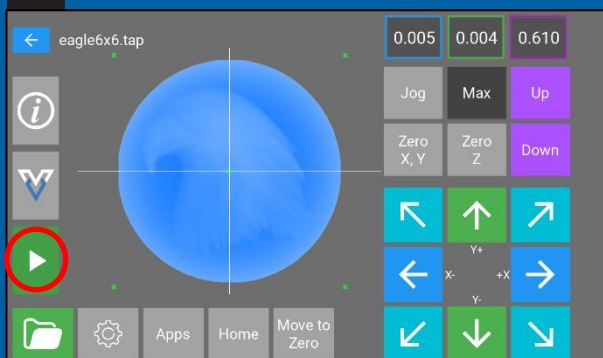
19

Once all points have been captured, press the back button at the top left, or the ok button to return to the previous page.
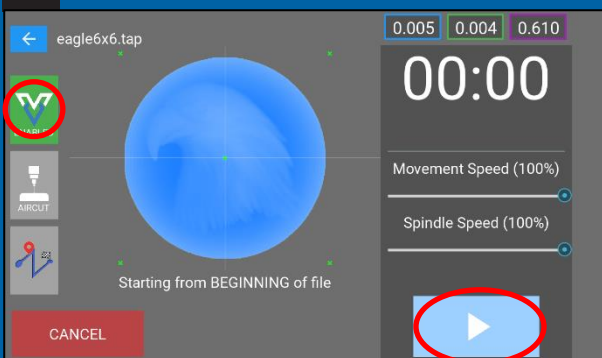
20

Press the back button again to leave the Virtual Zero application.

21

Press the Continue button to proceed to the File Run screen.

22

Press the Virtual Zero button on the left side of the screen to apply the Zero points you have taken to your project.
Then press Play to start the file.

Power Loss Recovery

Utilizing the Work Offsets feature can allow you to resume an interrupted file or cancelled file.

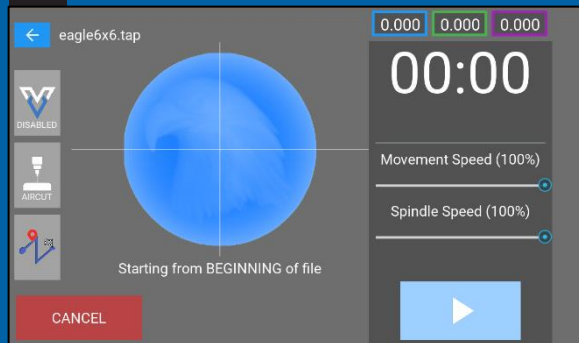The machine must be Homed after startup for your Work offsets and most recent X/Y zero to be saved.

After homing the machine, the most recent X/Y Zero position recorded will be automatically saved in the event of a power loss.

If a previous X/Y Zero is detected, the Pendant will ask if you would like to set the last known Zero position again.
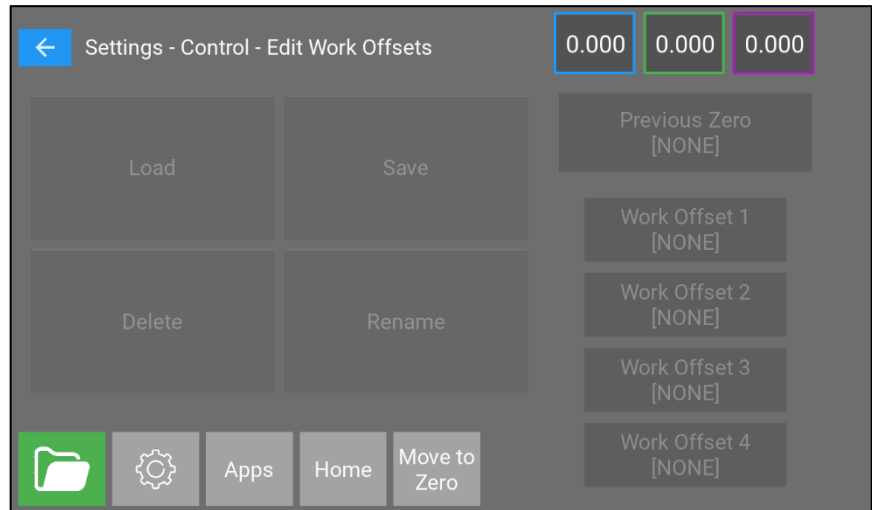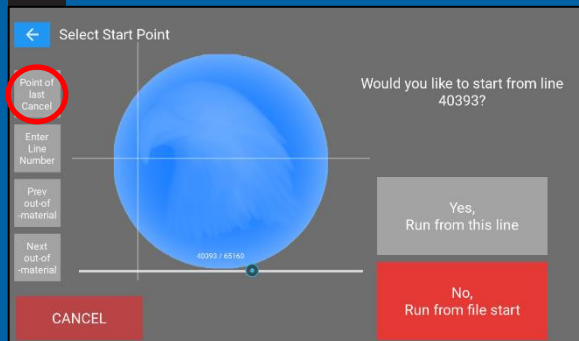

Resuming a Project

1

Set, or Load your X/Y Zero position and reload the interrupted file. Continue to the Run Screen.
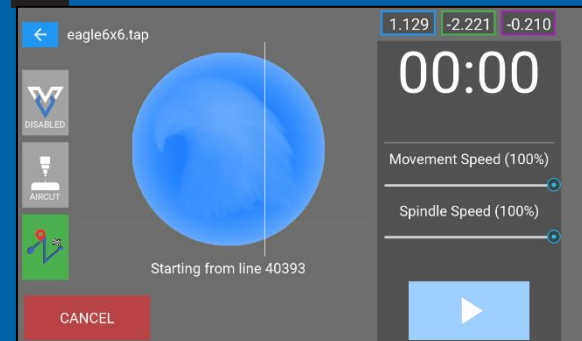
2

The Select Start Point button is located on the left side of the screen under the Air-cut toggle button (Figure 2). Press this button to choose the file's starting position.

3

Press the 'Point of Last Cancel' button located on the left side under the Back button (Figure 3). The slider will snap to the last know point in the file. Press the Yes button to proceed.

4

Press the run button to resume the file. The Pendant will check all lines up to the Start Point for errors before resuming.

Accessory Settings

The Accessory settings submenu contains various settings for add-on accessories such as the Laser and 4th Axis rotary systems.
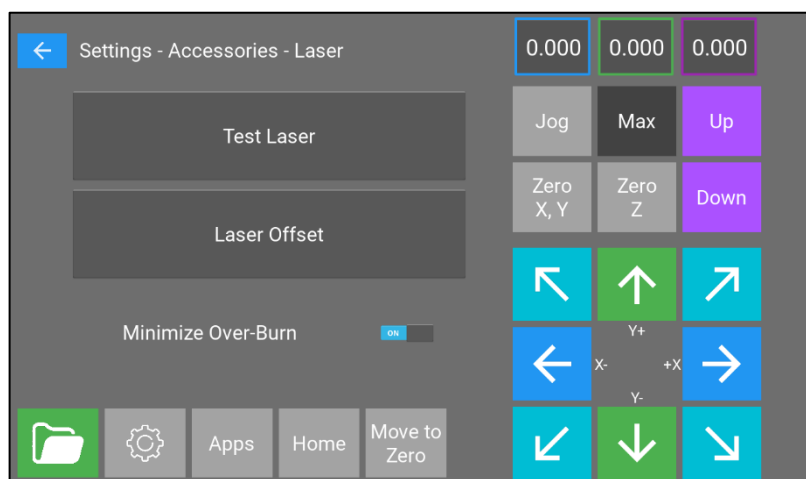
In this menu, you may toggle your 4th Axis Controls and Position display, change your 4th Axis Model, and adjust various settings for the Laser accessory such as Offset, Minimize Over-Burn, and fire the laser as a test.
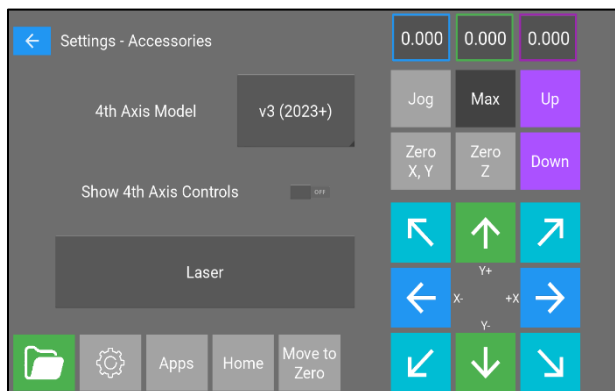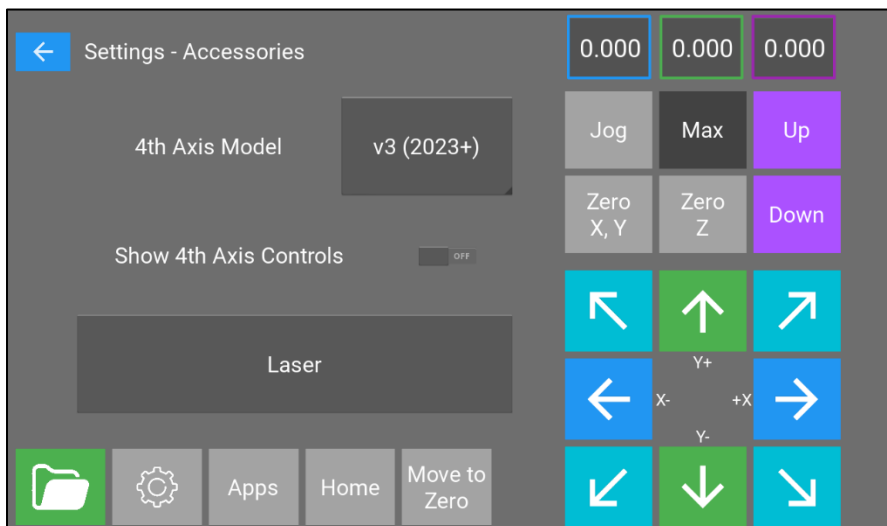
4th Axis Model: Change the model type of the 4th Axis. This changes the step value of the motor (See the 4th Axis Owner's Manual)
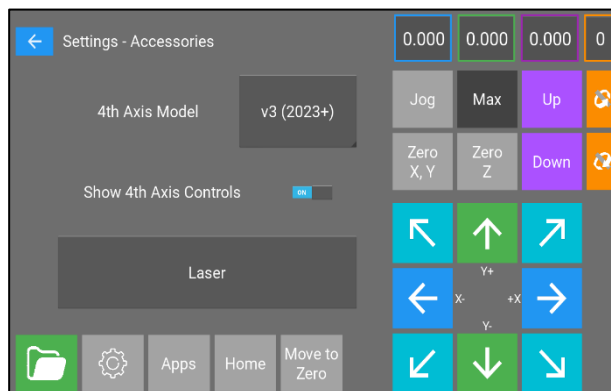
Show 4th Axis Controls: Toggle the display of the A-Axis location and Control buttons

Test Laser: Manually fire the laser at low power.

Laser Offset: Change the Offset values for applicable Laser accessories.

Minimize Overburn: Reduce the laser intensity when the machine slows down around corners. (Does not apply when burning raster files.)

Laser: Test and adjust the configuration of your Laser Accessory

Maintenance

The Pendant is a durable and modern tool for controlling your *Shark CNC,* but the dust created by the CNC can sometimes cause problems.

To keep the Pendant operating trouble-free, it's important to keep it clean. To clean the Pendant case and screen, use canned air to blow off dust.

Firmware updates are available at
NextWaveCNC.com/firmware

Wipe the Pendant screen with a soft microfiber cloth to prevent scratches and remove sawdust.

Find more maintenance and setup tips and answers at:
https://www.nextwavecnc.com/
frequentlyaskedquestions

Upgrade Accessories

SHARK®
World Class. Made in USA.

2HP WATER COOLED SPINDLE SYSTEM

- Powerful 2HP motor
- ER11 collet system includes 1/4" & 1/8" collets
- Water pump, hose and connection kit included
- 110V uses standard 20 amp circuit
- 6,000—24,000 rpm (rotations per minute) variable speed Inverter
- Spindle clamp adapts to existing router mounts
- Less than .005 mm runout Superior quiet operation Outer dimensions 80 mm x 180 mm

SKU 20157

NEXT WAVE CNC

SHARK®
World Class. Made in USA.

3HP SPINDLE KIT w/ eCOOL XT

- Powerful 3HP motor
- Digital Speed Control
- ER20 collet system includes 1/4" & 1/2" collets
- Enclosed Cooling System
- In line Flow Indicator

SKU 20160

NEXT WAVE CNC

MINI-TOUCH PLATE

- Use the 1" diameter mini touch plate in tight spots, especially useful for Virtual Zero Unlimited
- Solid Aluminum Construction
- 12" patch cord with banana receiver
- Requires standard NWCNC touch plate to work (SKU 20141)

SKU 20146

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

SHARK®
World Class. Made in USA.

SHARK PENDANT
User's Manual
Version: 09/17/2024

800039