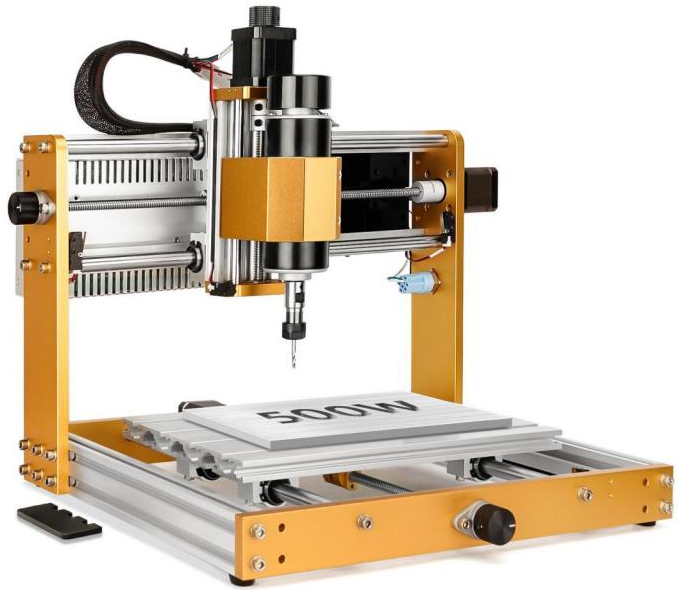
LUNYEE INDUSTRIES

# LUNYEE 3018CNC ROUTER MACHINE

## Tutorial For Beginner

Version 1.0 • January 2024



# Contents

Part 1: Software Installation----------3
Part 2: Software Settings----------9
Part 3: Recommended Settings-Router----------29
Part 4: FAQ &Precautions----------33
Part 5: Default GRBL Settings----------35
Part 6: Best Practices----------41
Part 7: Terminology----------42

# Part 1 Software Installation

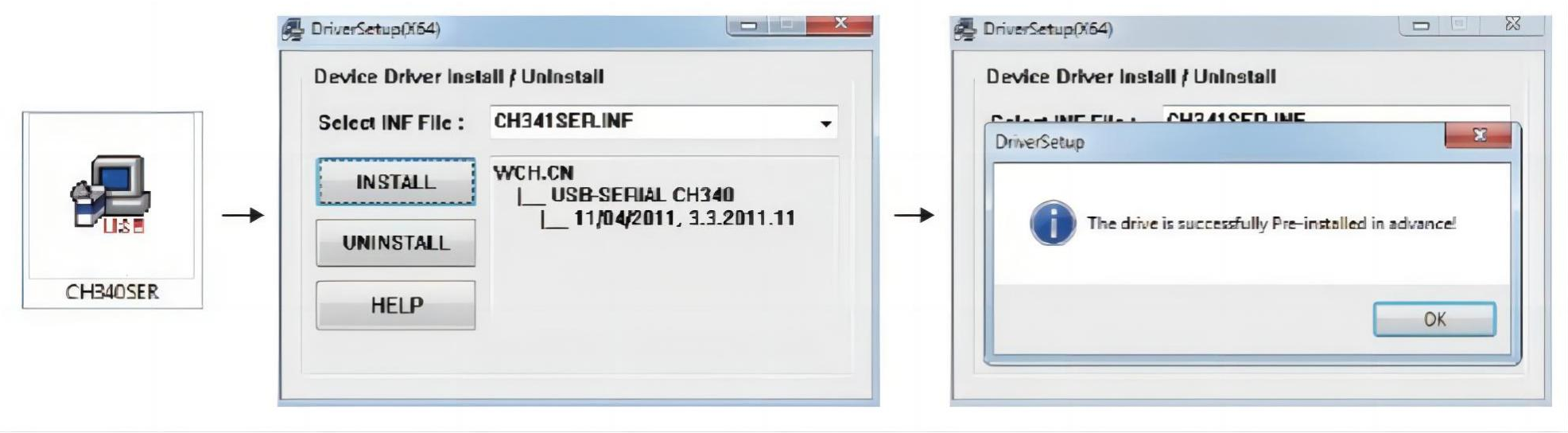
## Step1-Install the machine's driver(CH340SER)

If not available on the USB drive,you may download it here: https://sparks.gogo.co.nz/ch340.html

NOTE FOR MAC USERS:If your Mac is running Mojave OS or higher,do NOT install the driver,as the OS has native support for the CH340.If you are running Sierra or High Sierra,please contact LUNYEE Technical Support for additional guidance.

If the driver installation fails,please clic kUninstall and then select Installagain.

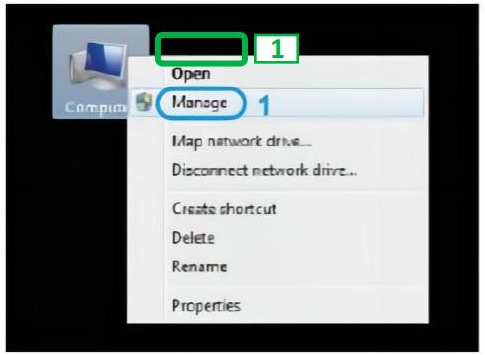
The CH340 driver is installed to ensure that the CNC engraving machine or controller can communicate stably and reliably with the computer through the USB interface.

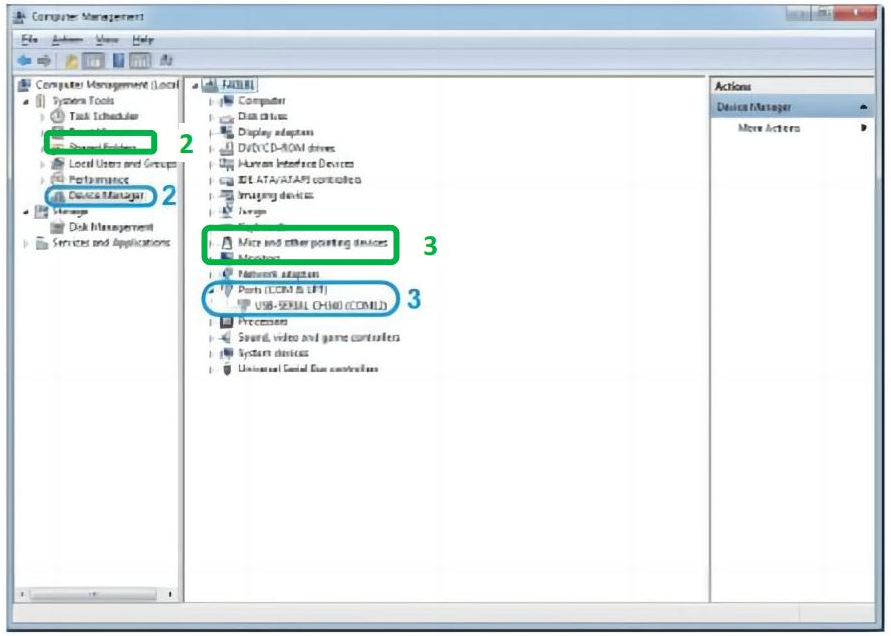


# Part 1 Software Installation

## Step 2 - Determine your machine's COM port(Windows computers)

· **WindowsXP** : Right click on My Computer,select Manage,then Device Manager.
· **Windows7** : Click on Start on the task bar,right click on Computer,select Manage,then Device Manager.
· **Windows10** : Inthesearch box on the task bar,type Device Manager,then select it from the menu.




In the tree, expand *Ports (COM & LPT)*. Your machine will be identified by the USB Serial Port (COMX),where the "X" represents the COM number, for example COM3.

If there are multiple USB serial ports, right click on each one and check the manufacturer, the machine's should be "CH340".

**Note - Machine must be connected to your computer in order for a COM port to be assigned.**

# Part 1 Software Installation

## Step 3 - Login to Easel

We recommend that our customers use a web application called Easel,which is developed by Inventables for their X-carve machines.




Easel is an easy-to-use online CNC (computer numerical control) software for designing creating and controlling CNC engraving and cutting projects.

It works great with the the LUNYEE 3018 too. It is free (the pro version is not really needed) powerful and easy to use to design and carve your projects.
Go to easel.inventables.com and click on theblue Sign in through inventables button.

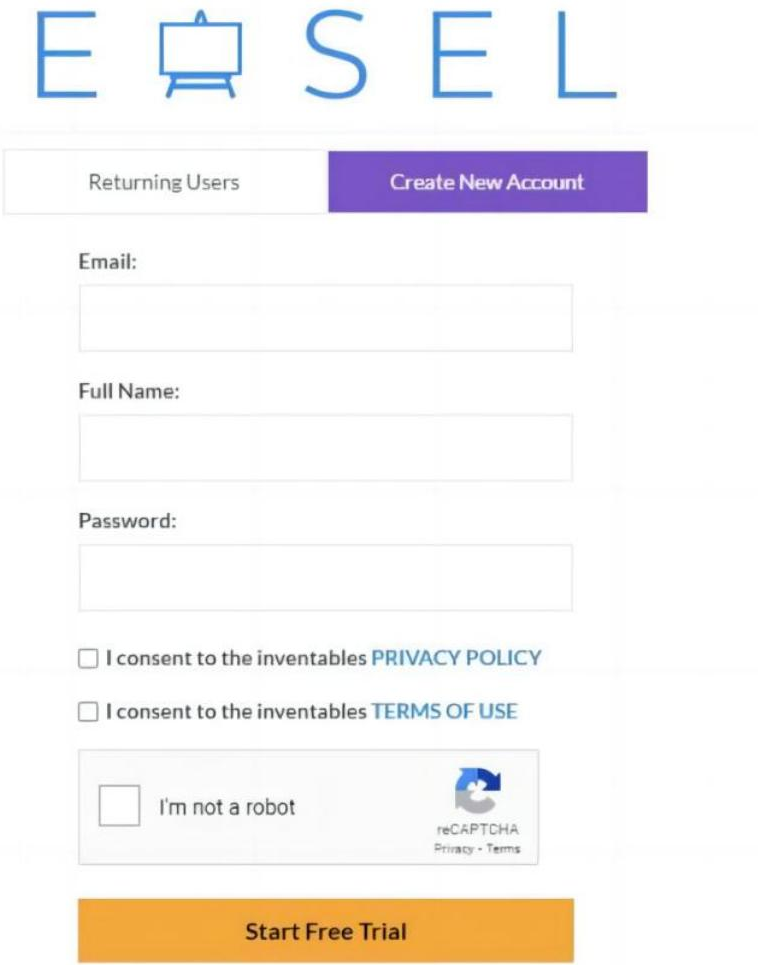
# Part 1 Software Installation

## Step 4 - Create your account in Easel

·Click on the“Create New Account"option ·Enter your personal information.

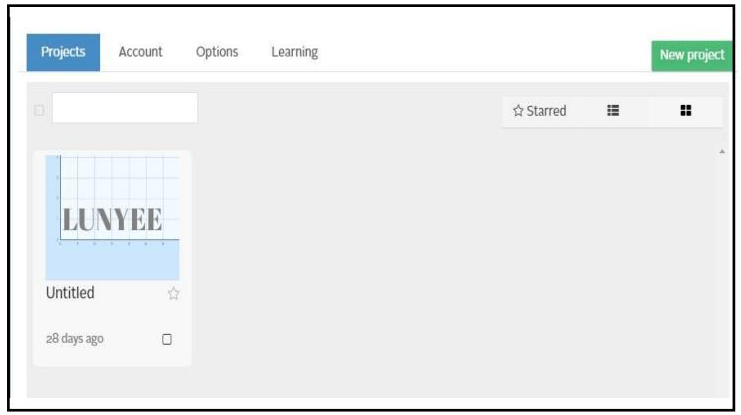
·When you are done, click on the orange Start Free Trial button.

**Note：**
**Inventables defaults all new accounts to a free 30-day trial of Easel Pro. At the end of that trial, you can continue with a paid Pro subscription, or revert to a FREE Easel account.**

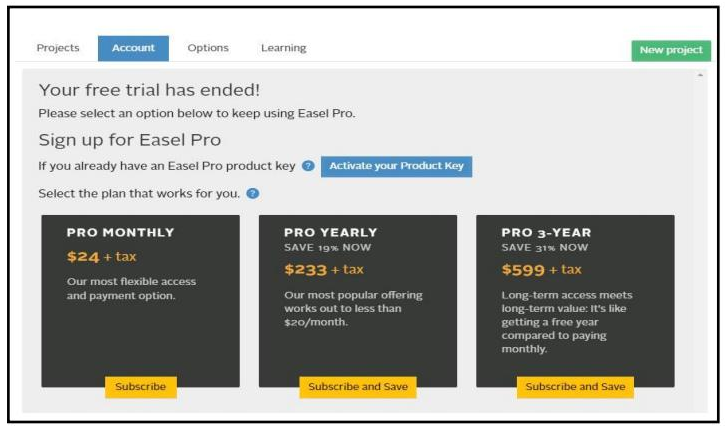


# Part 1 Software Installation

Projects:
The Projects section is where you manage your design files and creations within Easel.Here, you can create new projects, open existing ones, and organize your work.
You can save projects, make copies, and delete or rename them as needed.
This section allows you to easily access and work on your designs.

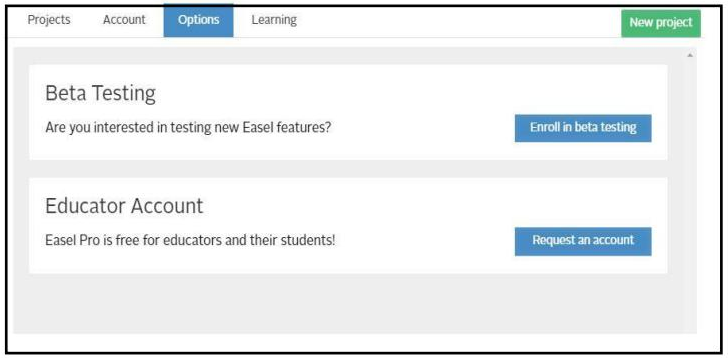



Account:
The Account section pertains to your user profile and settings.
Here, you can view and update your account information, such as name , email , and password.You may also manage subscription plans, billing details, and connected devices.
The Account section ensures you have control over your Easel user account and associated preferences.

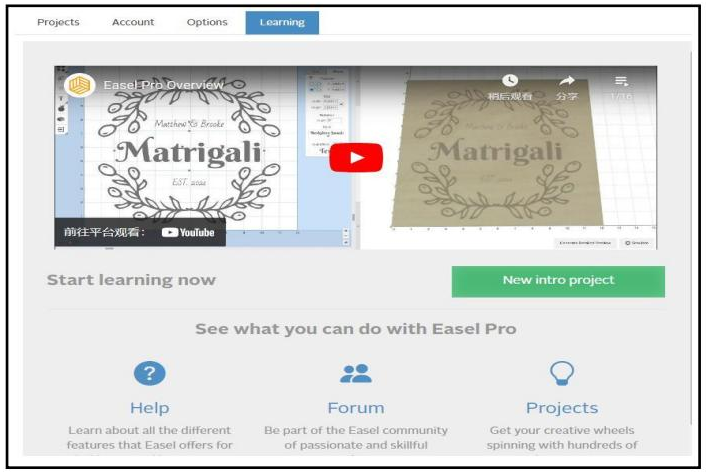


# Part 1 Software Installation

Options:
The Options section contains various settings and preferences for customizing your Easel experience.
Here, you can configure software settings related to workspace units, grid settings, snapoptions, and more.
You may also adjust project-specific settings such as material thickness and machine configurations.
Options provide flexibility and control over how Easel operates according to your needs and preferences.




Learning:
The Learning section is dedicated to resources for acquiring knowledge and improving skills in CNC and Easel usage.
Here, you can access tutorials, guides, and educational content to learn about CNC concepts, Easel features, and project techniques.
Learning materials may include videos, articles, and interactive tutorials to help you master CNC design and machining.
This section is valuable for beginners seeking to understand CNC fundamentals or experienced users looking to expand their capabilities.

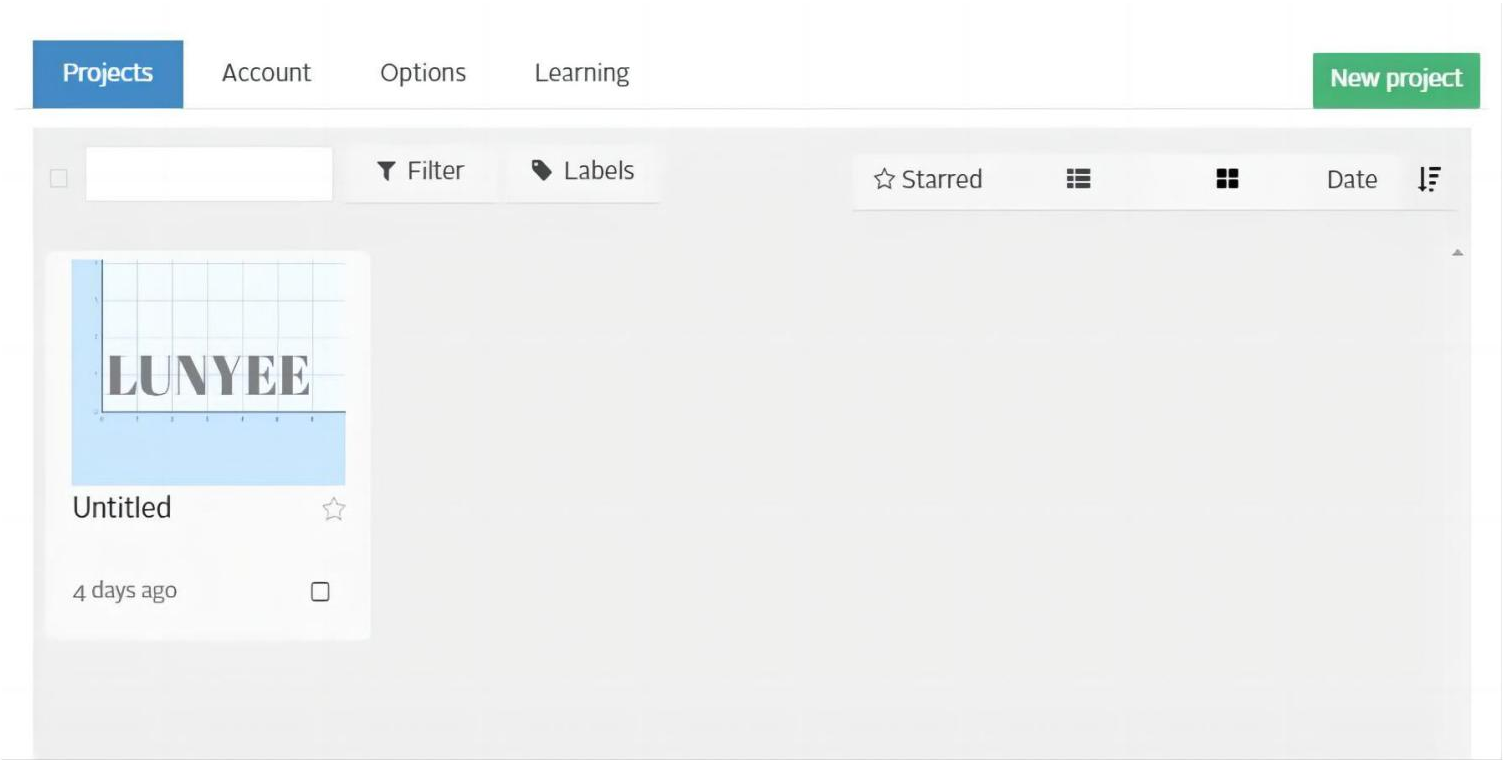


# Part 2 Software Settings

## Step 1 - Your first project in Easel !

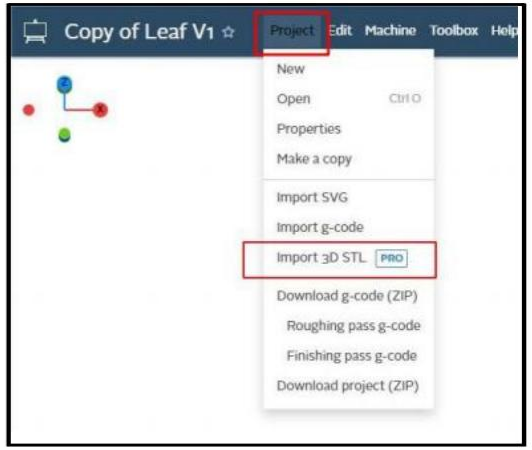
Create a new project by selecting one document in the Project stab (it can be“Intro to Easel”or any “ Untitled”)

Next we are about to enter the machine setting process.We need to match the varous parameters of the machine
with Easel so that the machine can successfully complete the connection.
(The next page will introduce the functions of each section
of Easel)



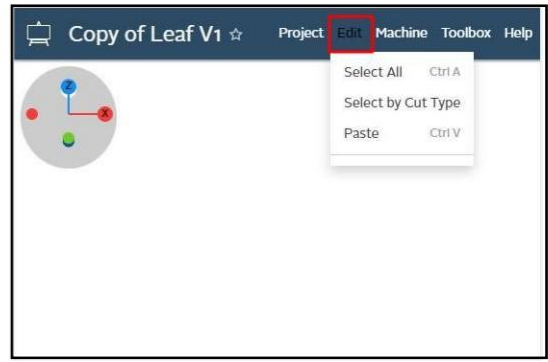
## Project:

The Project section is where you create, save, and manage your projects.
Here, you can start new projects or open existing ones.
You can name your projects, add descriptions, and save/load them at any time for easy management and editing.



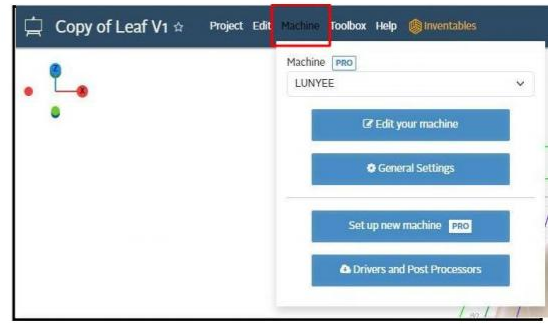

## Edit:

The Edit section contains various tools and functions for editing and adjusting your designs.
Here, you can select, move, resize, rotate, and delete elements within your project (such as shapes, text, images, etc.).
Use editing tools to precisely adjust the position, size, and appearance of elements to ensure your design meets your expectations.



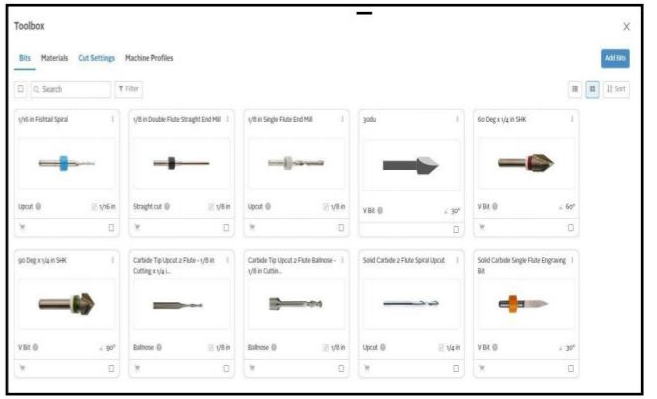

## Machine:

The Machine section is used to configure and manage settings related to your CNC machine.
Here, you can select your CNC device model and set parameters such as workspace size,tool diameter, cutting depth, etc.Ensure correct machine settings here so that Easel can generate cutting paths and commands suitable for your CNC device.



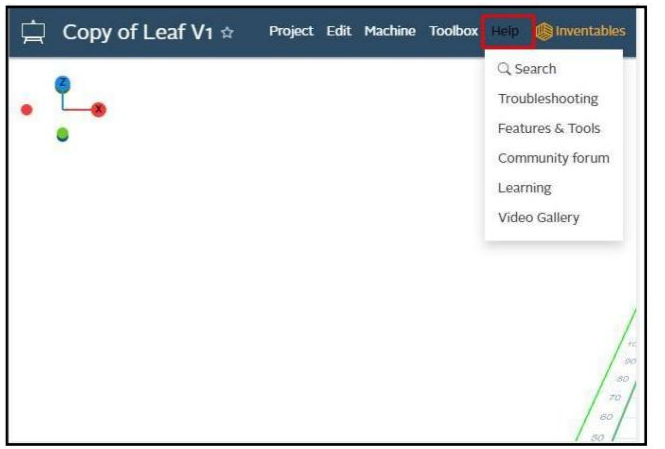
## Toolbook:

The Toolbook section is where you manage and select available tools. Here, you can browse and choose different types of tools, such as end mills of various diameters, engraving tools, etc.
Selecting the appropriate tool is crucial for successful CNC machining, so choosing the right tool here is important.




## Help:

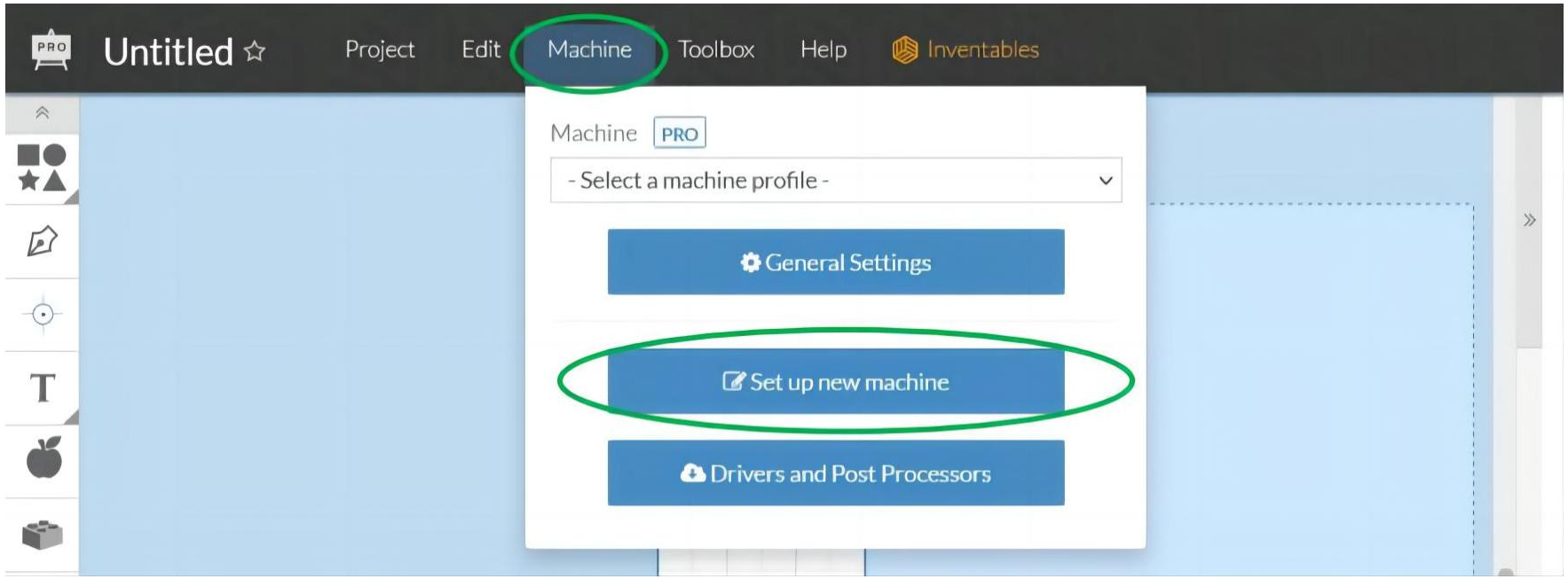
The Help section provides user guides, tutorials, and FAQs about Easel.
Here, you can find detailed explanations and guidance on specific features or issues.
If you encounter difficulties or have questions while using Easel, you can find answers or seek support in the Help section.



# Part 2 Software Settings

## Step 2 - Set up New Machine

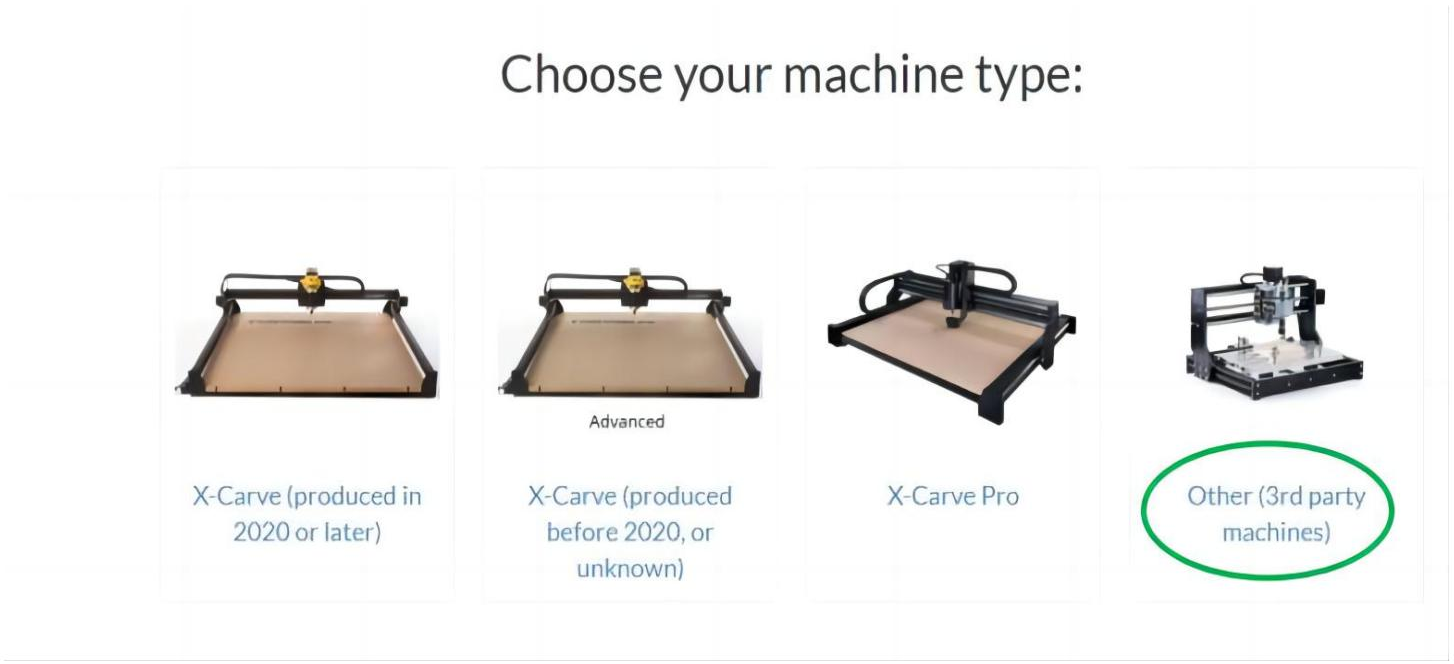
Click on the“Machine”menu,and select the blue“Set up new machine” button.



# Part 2 Software Settings

## Step 3 - Choose your machine

On the Choose your machine type screen,select the “Other(3rd Party Machines)”option.

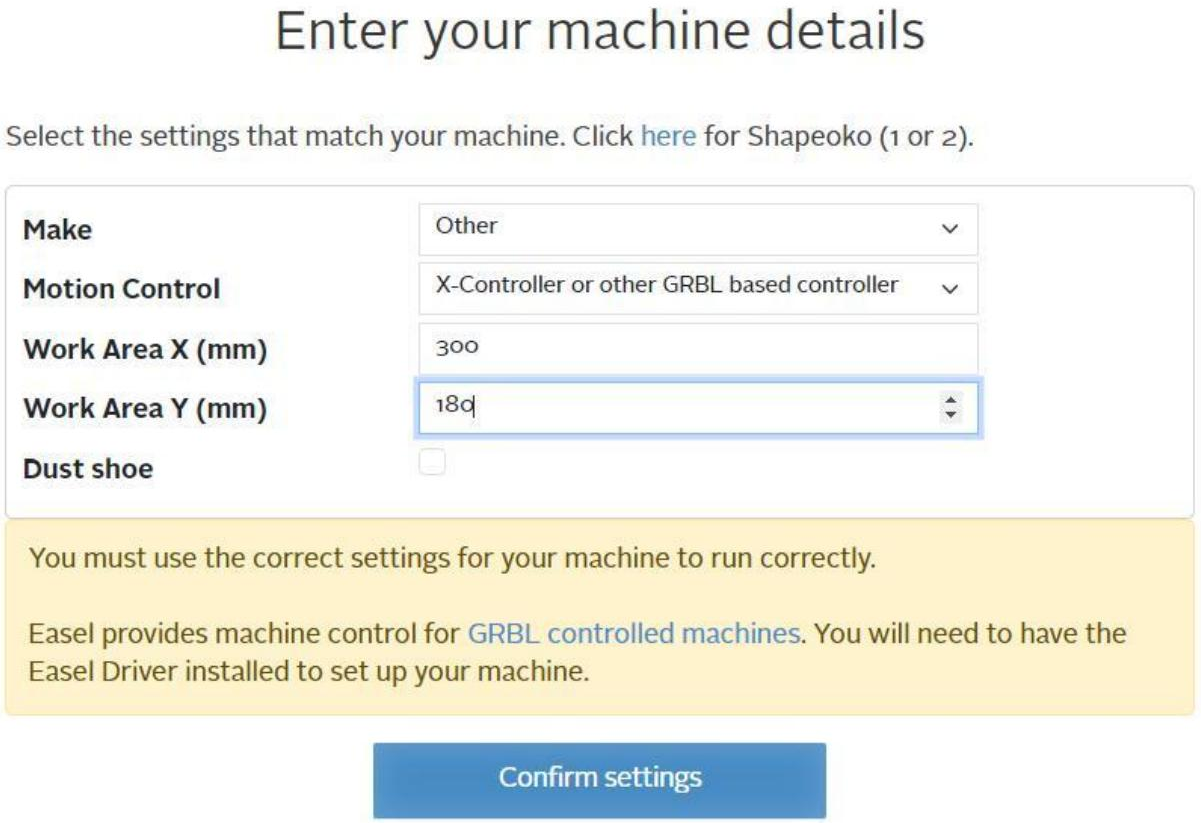


# Part 2 Software Settings

## Step 4 - Enter your machine details

Choose the options as shown in the picture.
We are working with Inventables to add the **LUNYEE**(Please configur and operate as shown on the right.)
If you have purchased the optional dust show with your machine,check the box.

Then click on the blue Confirm settings button.



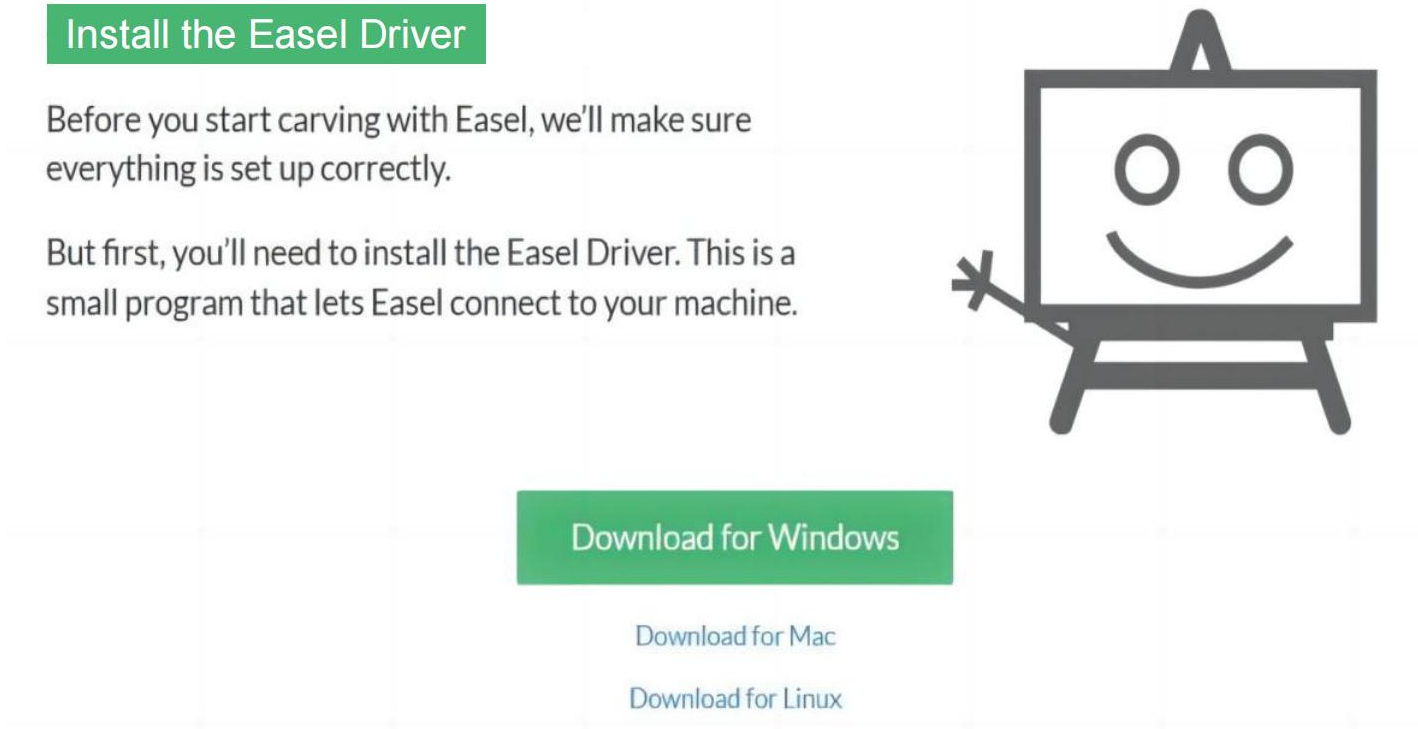

Since our product is not entered into Easel, we need to manually configure the parameters. For make, select other, and for motion control, select X-controller or other...,
work area X and Y respectively fill in the working area of the machine, X-300, Y-180

# Part 2 Software Settings

## Step 5 - Easel driver (Windows):

You will be automatically prompted to download and install Easel driver if it is not already installed; whether you're on a Mac or a Windows computer. This driver will allow Easel to talk to the USB port on your computer. lf the window below doesn't open, you can access the driverversion here:http://easel.inventables.com/sender_versions

W1.Click the green Download for Windows button,the Easel driver will automatically download to your computer. Most downloads are saved to the Downloads folder.



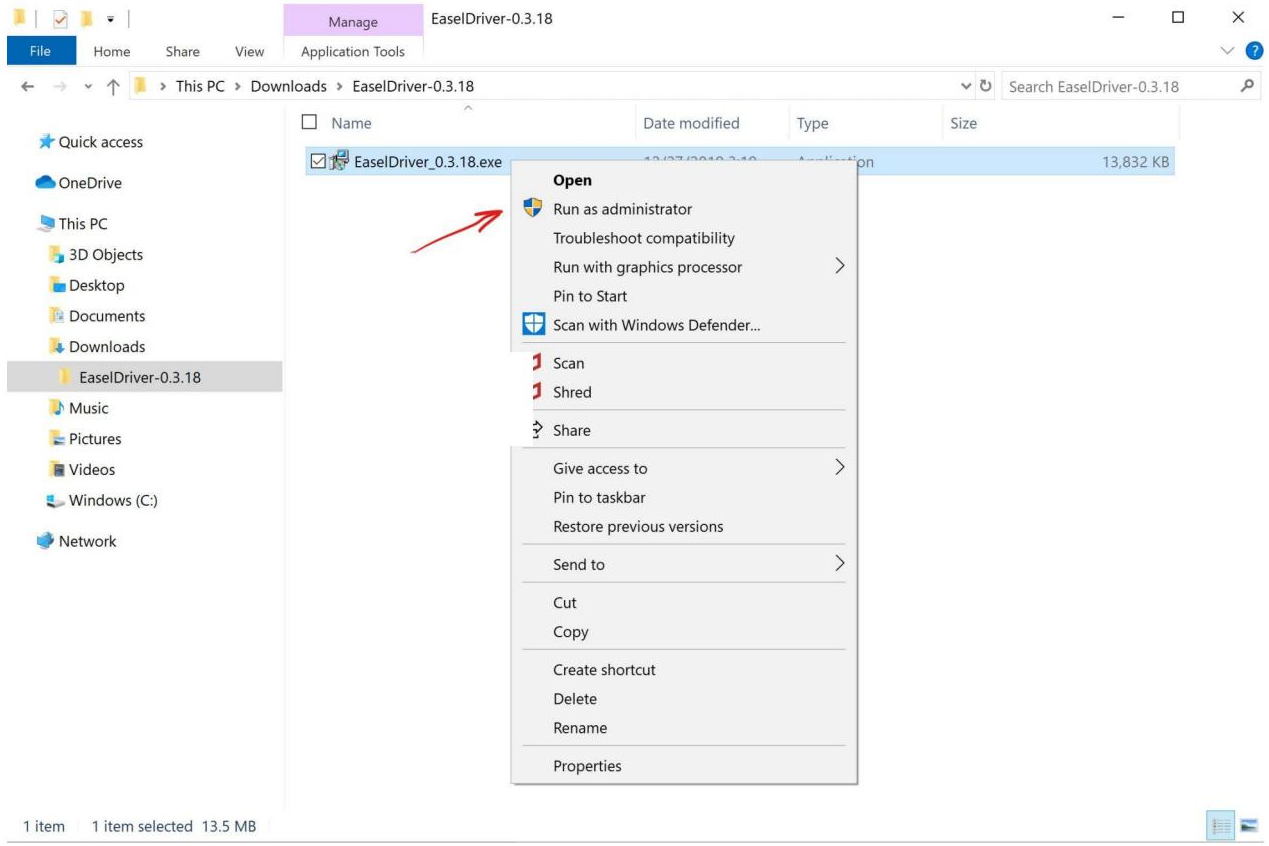
# Part 2 Software Settings

## Step 6 - Install Easel driver (Windows)

W2. Go to your Downloads folder, open the download and you should see the Easel Driver File (**EaselDriver-0.#.##.exe).**

W3. Then, right-click on it, select "Run as administrator" and follow the installation prompt. Follow the Installer instructions to complete the installation. You may need to refresh your browser and, in some cases, restart your computer.

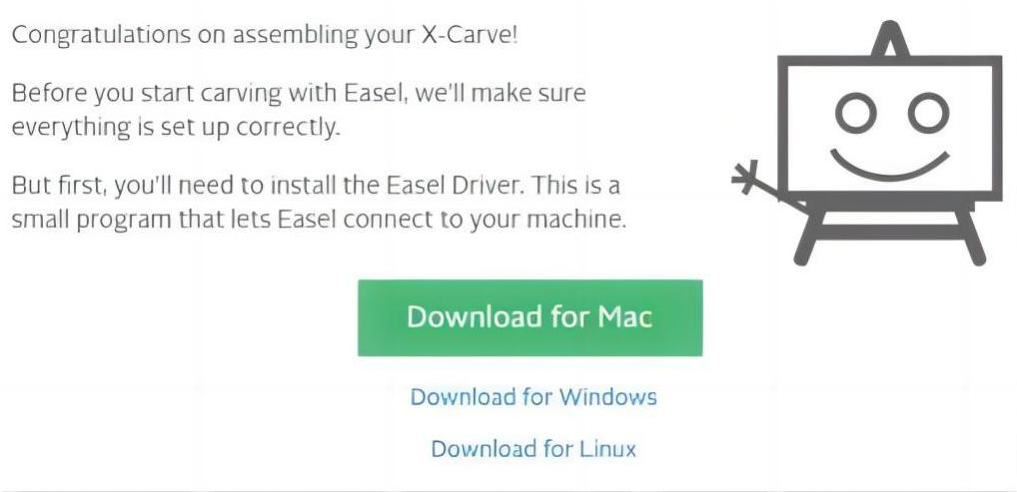
**Important:** Windows 10 security settings may pop up to confirm if you want to install the driver. You will need to click "More Info" and select "Run Anyway" to proceed with the installation.



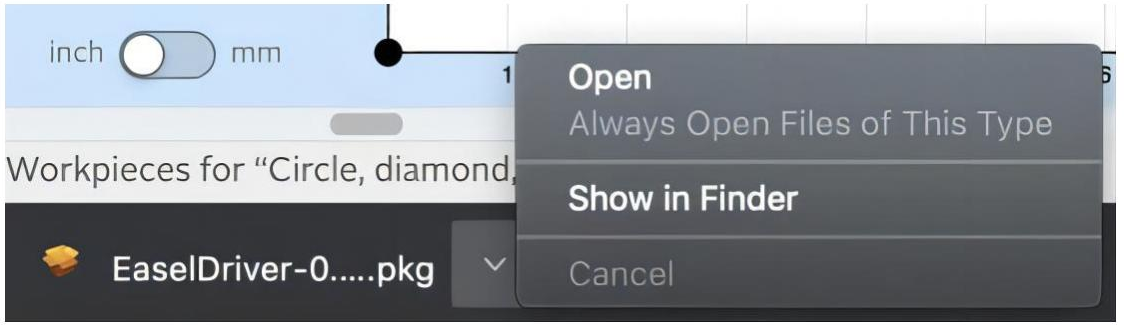
# Part 2 Software Settings

## Step 7 - Download and Install EaseDriver (MacOS)

M1.Click the green DownloadforMac button. EaselDriver will automatically download to your computer.You may also see the download at the bottom of your browser.



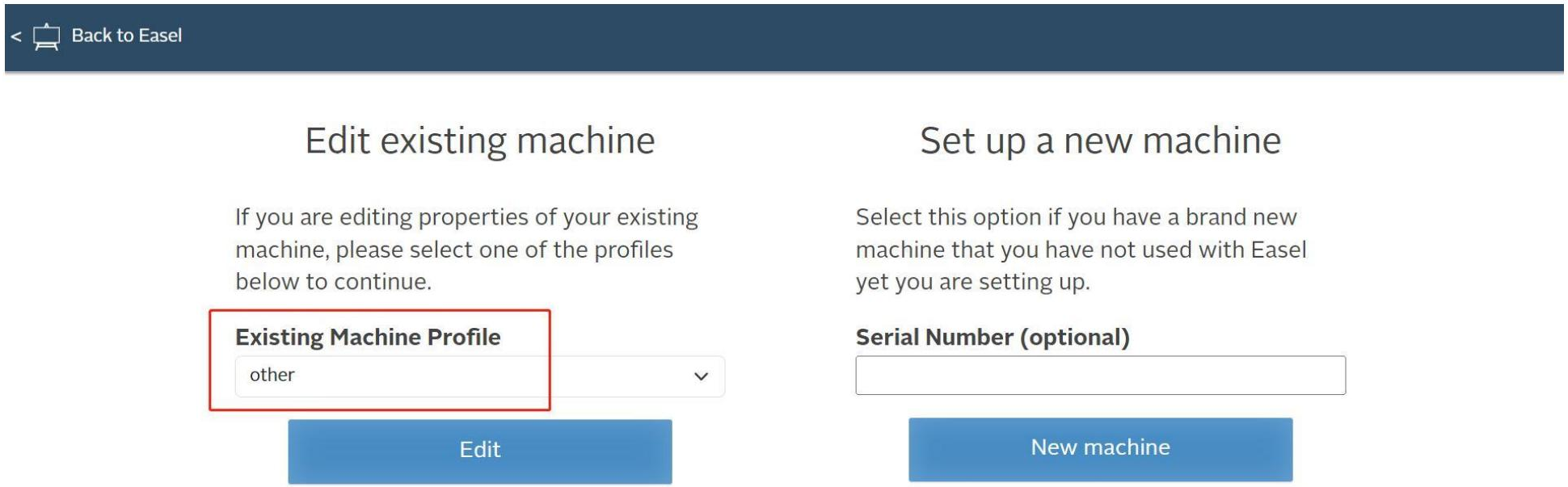

M2. Click the downloaded file (EaselDriver-0.#.##.pkg file and open it to run the installer. Follow the Installer instructions to complete the installation. You may need to refresh your browser and, in some cases, restart your computer.




If you do not jump to the next step after installing the driver, please restart the browser and log in to Easel again.

# Part 2 Software Settings

## Choose your machine



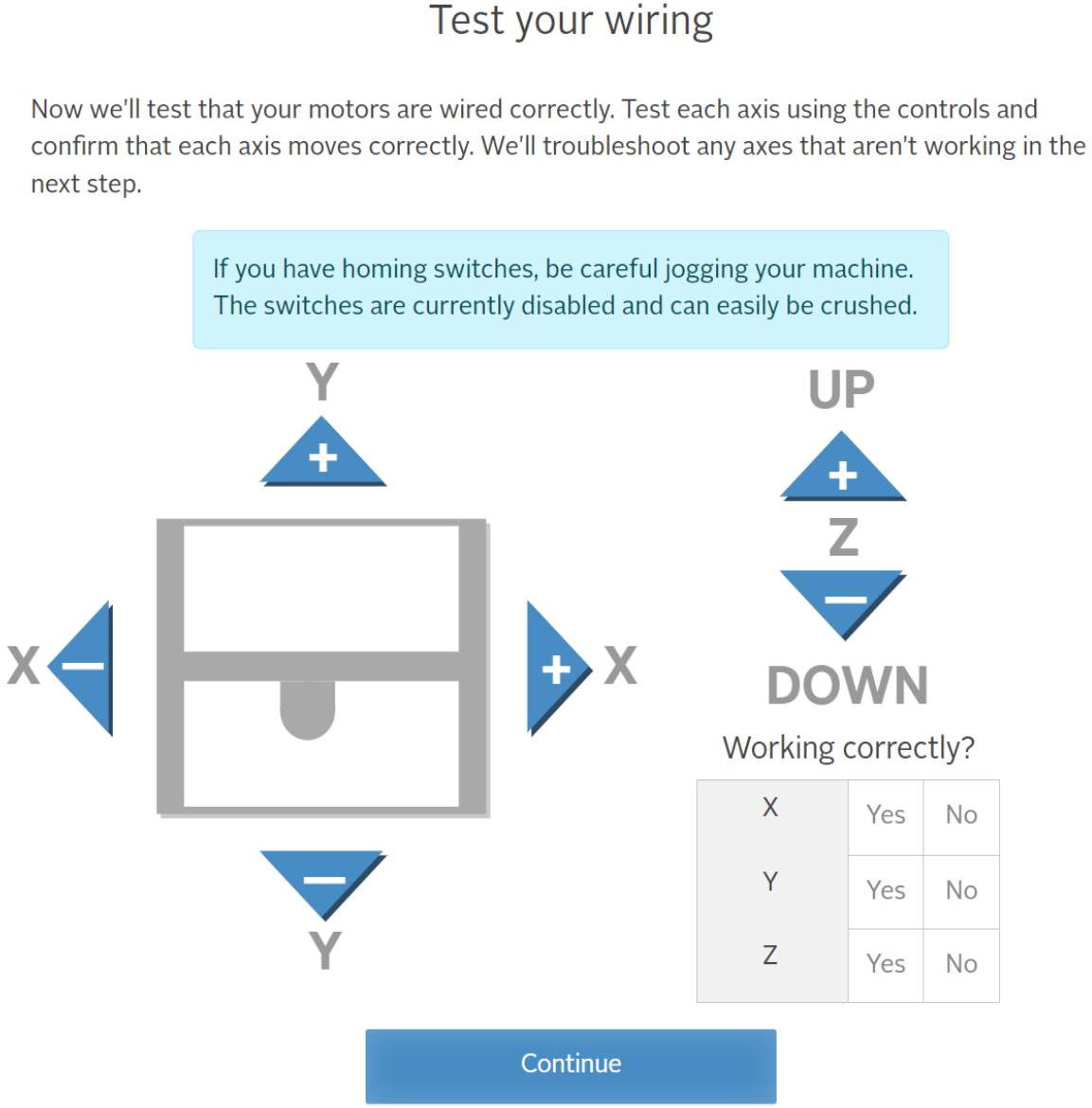
# Part 2 Software Settings

## Step 8 - Test your wiring

When the machine successfully connects, you'll be presented with a set of controls for jogging your machine.Tryjogging the machine using the directional arrows.

If everything is wired correctly you should get proper motion on each axis. If so, press on the Yes buttons for the X, Y and Z axis. Ifyou press No, it will swap the directions and fix the issue for you.

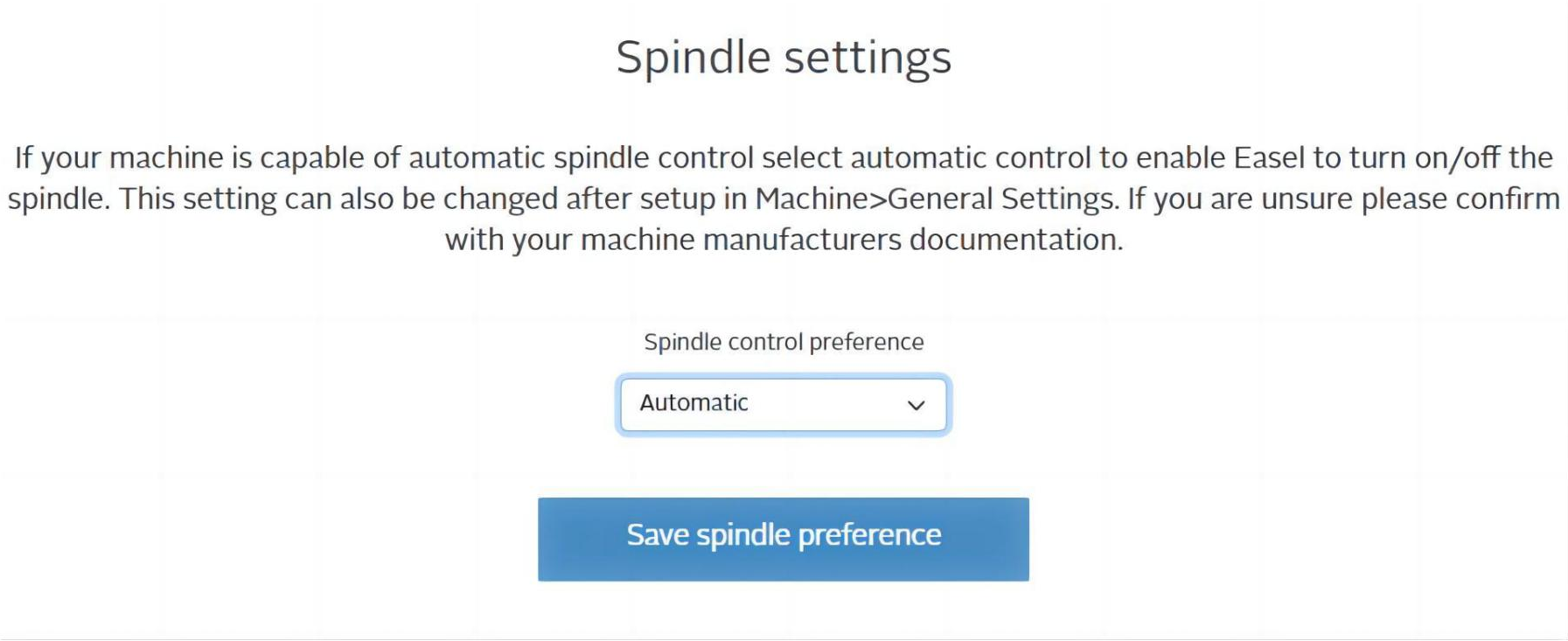
If the machine does not respond or stop suddenly when clicking the arrow it may be due to hitting the limit switch. Please first manually rotate the knob to return the axis to a safe positionthen find the machine's control panel and clickthe red reset button.
perform operation after reset.



# Part 2 Software Settings

## Step 9 - Spindle settings

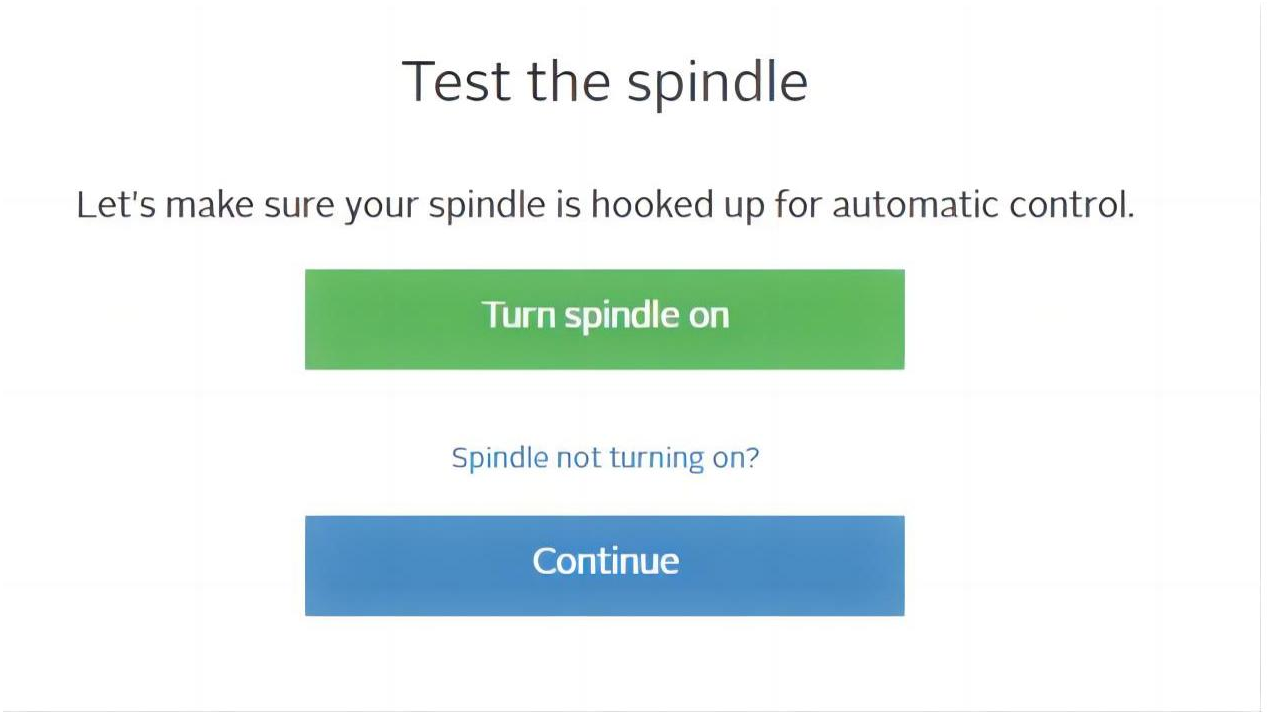
After entering the spindle setting interface,select "automatic".



# Part 2 Software Settings

## Step 10 - Test the spindle

Select "turn on the spindle"
If the spindle can rotate normall(usually it only rotates for about 5 seconds during testing),then please click "Continue"



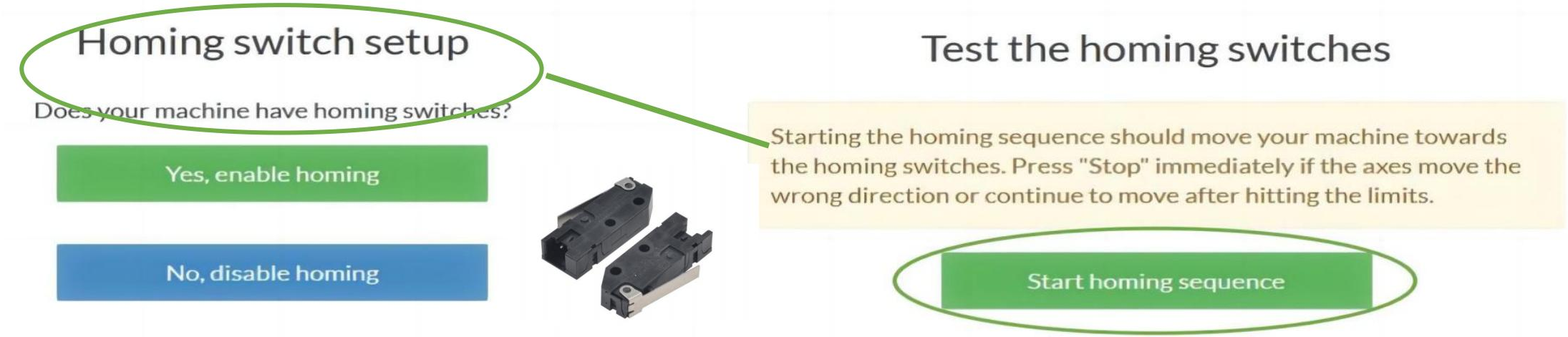
# Part 2 Software Settings

## Step 11- Limit switch setup

After you confirm your limit switches and Homing settings, you'll be prompted to enable or disable homing. Since the LUN YEE 3018 CNc is equipped with limit switches,select the “Yes, enable homing" option then click “Start homing sequence",the machine will go through a homing sequence to verify the limit switches.

***Ensure there is nothing in the way of any axes movement during this process.***

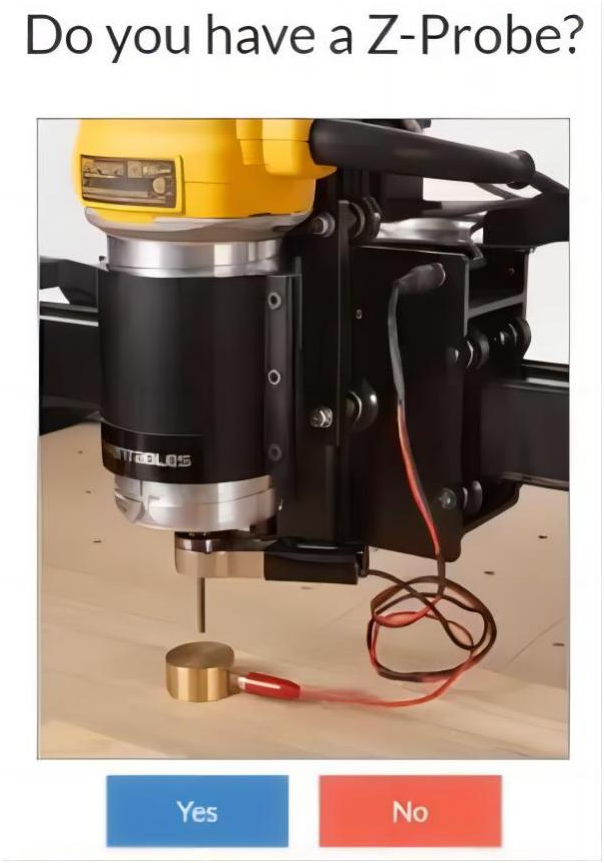
Limit switch function: CNC limit switches are used to detect the movement range of the machine tool and prevent it from exceeding the set working area to protect the safety of the equipment and workpiece.



# Part 2 Software Settings

## Step12-Z-Probe setup

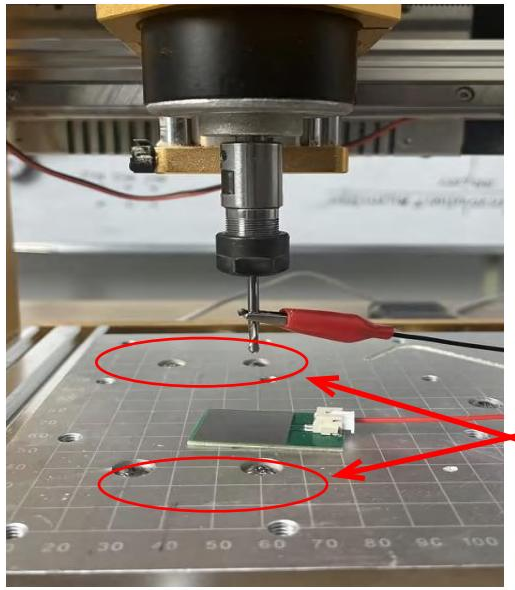
The LUNYEE 3018CNC Machine does have a Z-Probe.Click on the blue“Yes”button.

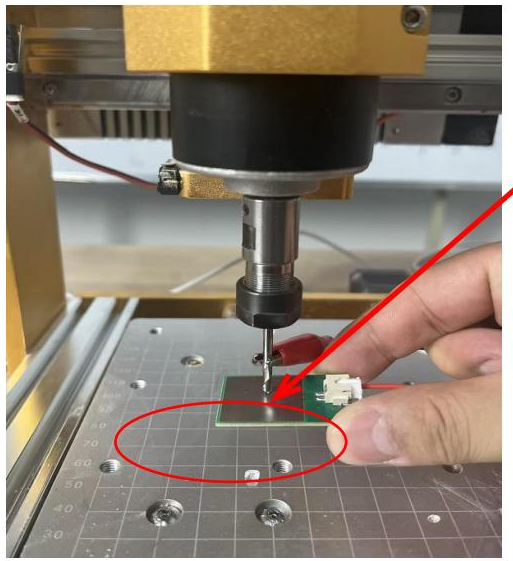


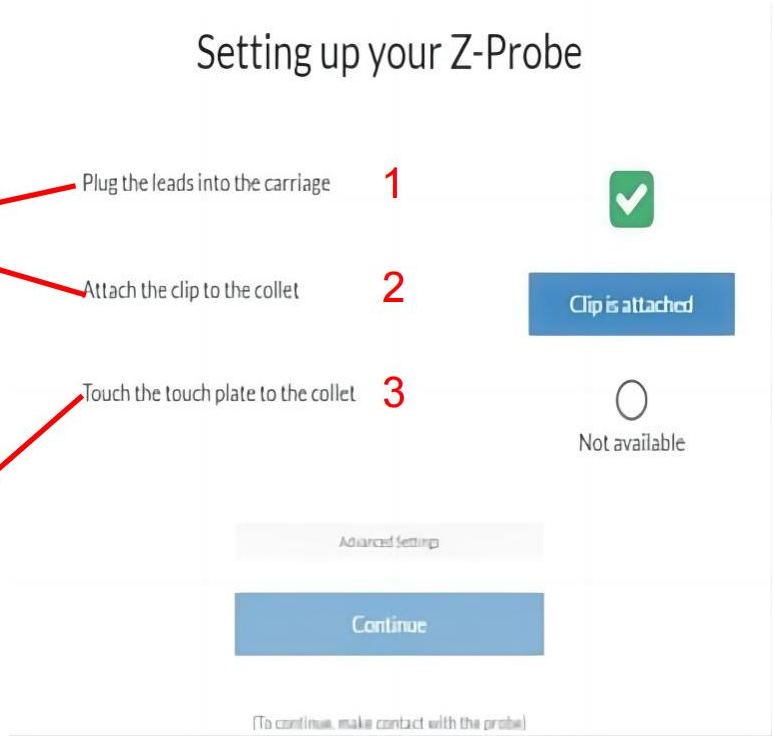
# Part 2 Software Settings

## Step 13 - 2-Probe configuration

Follow the prompts on the screen to confirm operation of the Z-Probe.
Connect the clip to the collet of the spindle, and touch the probe plate to the collet.Test whether connectivity is normal.
The indicator will turn green when contact is made.
Before continuing past this step, click on the “Advanced Settings" option to set the Z-Probe parameters.





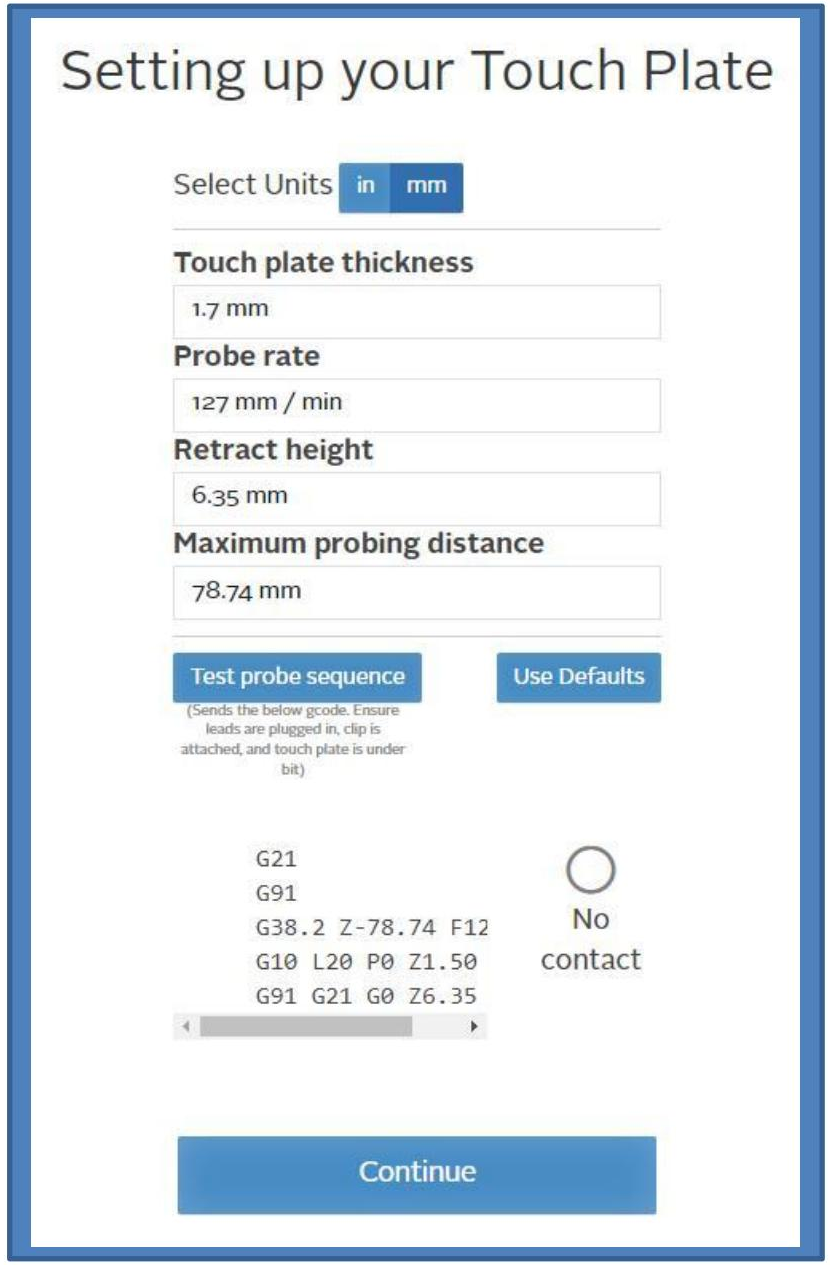


# Part 2 Software Settings

## Step 14 - Z-Probe Advanced Settings

Ensure the Touch plate thickness is set to 1.7mm. (You may need to select the “mm" option at the top of the screen.
Here we need to explain why 1.7mm is set, because the thickness of our tool presetter module is 1.7mm.
All other options can be left at default.

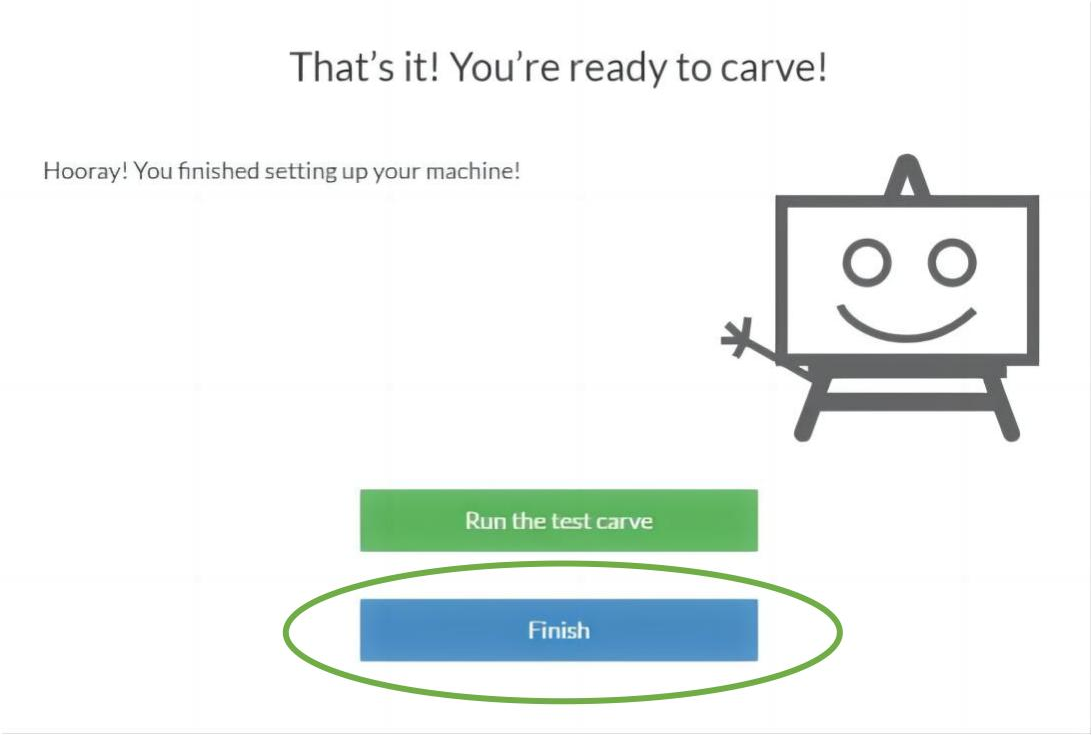
Click on the “continue” button.



# Part 2 Software Settings

## Step 15 - Finish Screen

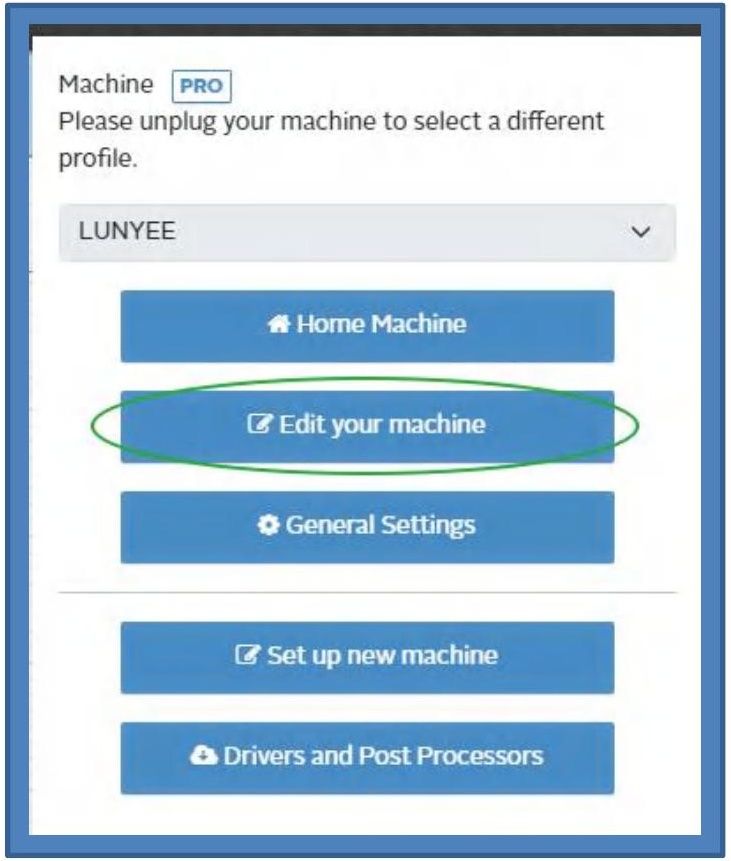
After setting up your machine, you will need to make a few more changes before you can run a test carve. Click on “Finish” and then you should be prompted to start a new project. Just click New Project and continue to the next Step.



# Part 2 Software Settings

## Step 16 - Verify Machine Settings 1/2

After you started a new project click on “Machine” at the top menu then click “Edit your machine”.



# Part 2 Software Settings

## Step 17 - Verify Machine Settings 2/2

Confirm the following settings:

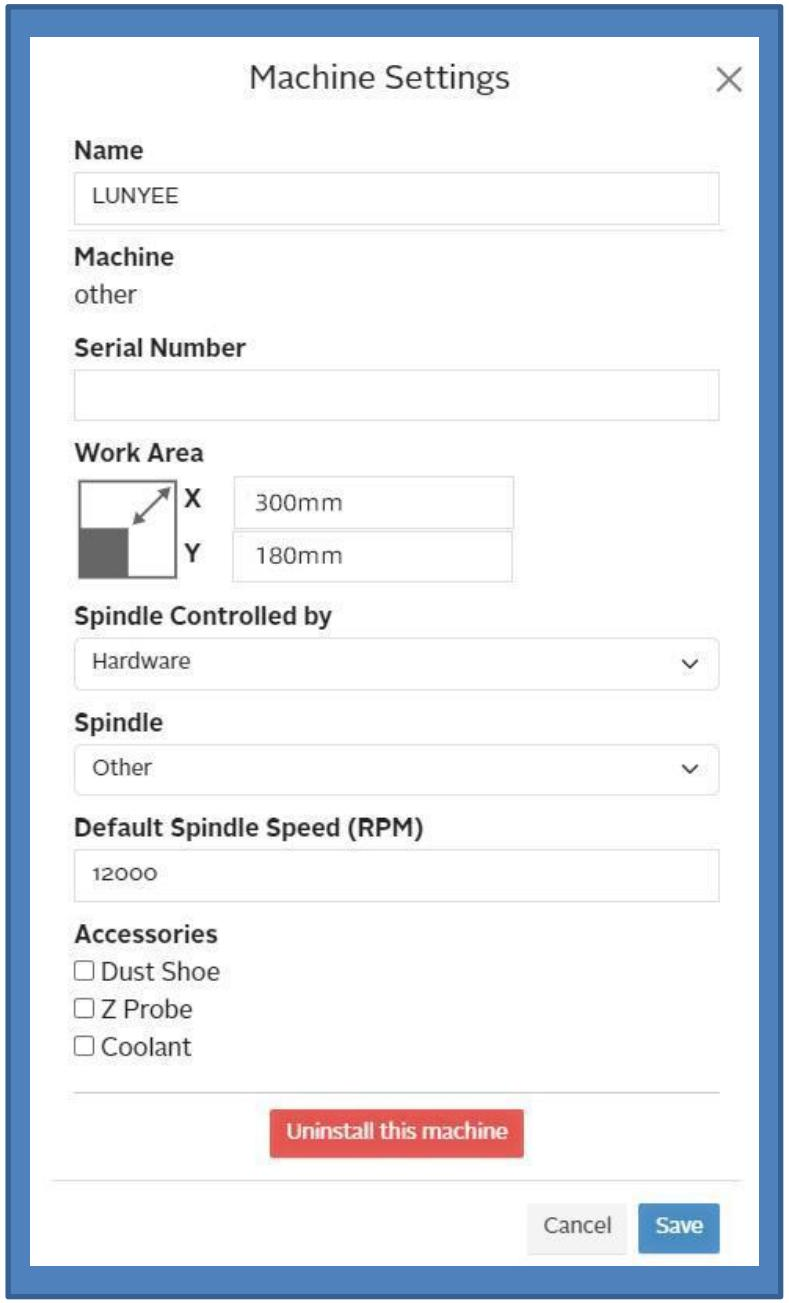
1. Work Area X: 300mm
2. Work Area Y: 180mm
3. Spindle Controlled By: Hardware
4. Spindle: Other
5. Default Spindle Speen: 12000
6. Check Z probe

If you have other accessories, you can also check them

make sure the box is checked.

Z Probe will be checked is it was previously setup.

Click Save and you are ready for your test carve.



# Part 3 Recommended Settings - Easel

## Metric system settings in Easel

We recommend that our customersuse the mm settings for better readings and precision, especially in the CNC world.



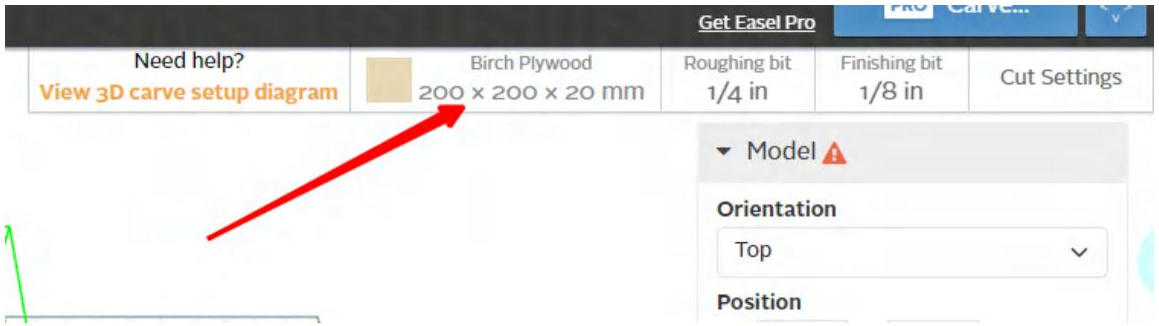
# Part 3 Recommended Settings - Easel

## Material type and Dimensions

1. Find the piece of wood or material you want to carve



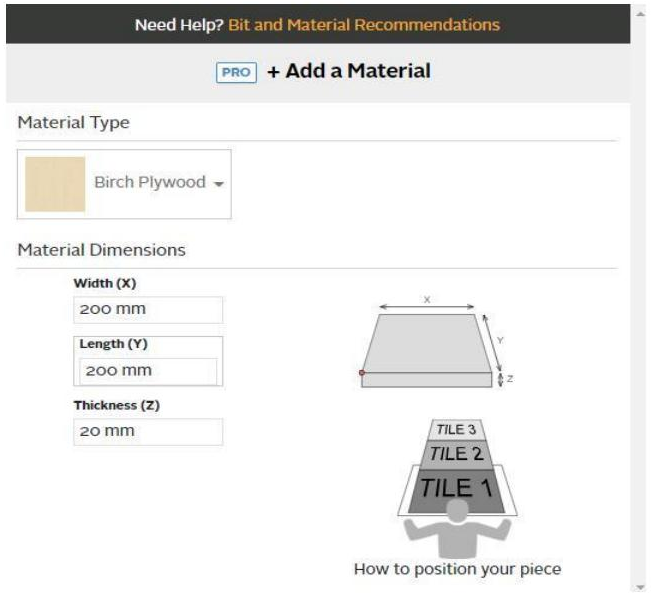
2. Select the material type in Easel




3. Take the measurements (Length, Width, Depth) of your material



4. Enter the Material Dimensions in Easel



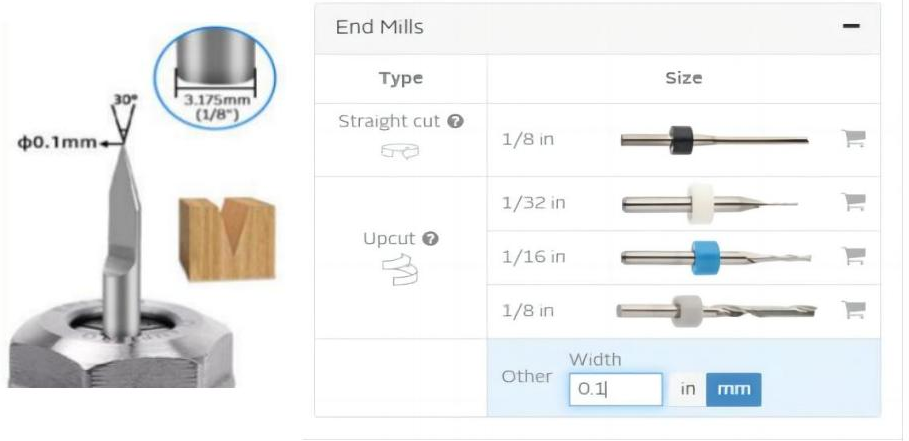
# Part 3 Recommended Settings - Easel

## Bit settings

5. Clamp your material down n your machine's table.

6. Insert your bit into your ER11. Specify the type of End  Mills you inserted using Other in Easel for any of the bits you have. (This will allow you not to have to purchase the Easel Pro Version)

3018 pro max： bits included in the 3018 pro max package are all 3.175*30°*0.1 end mill.




3018 pro ultra: bits included in 3018 pro ultra package:
2pcs 3.175*30°*0.1 sharp knives, end mill
2pcs triangular insert turning tools of 3.175*30°*0.1
2pcs single-flute spiral end mills of 3.175*2*12mm
2pcs single-flute spiral end mills of 3.175*12mm
2pcs 3.175*17mm double-flute spiral ball nose end mill



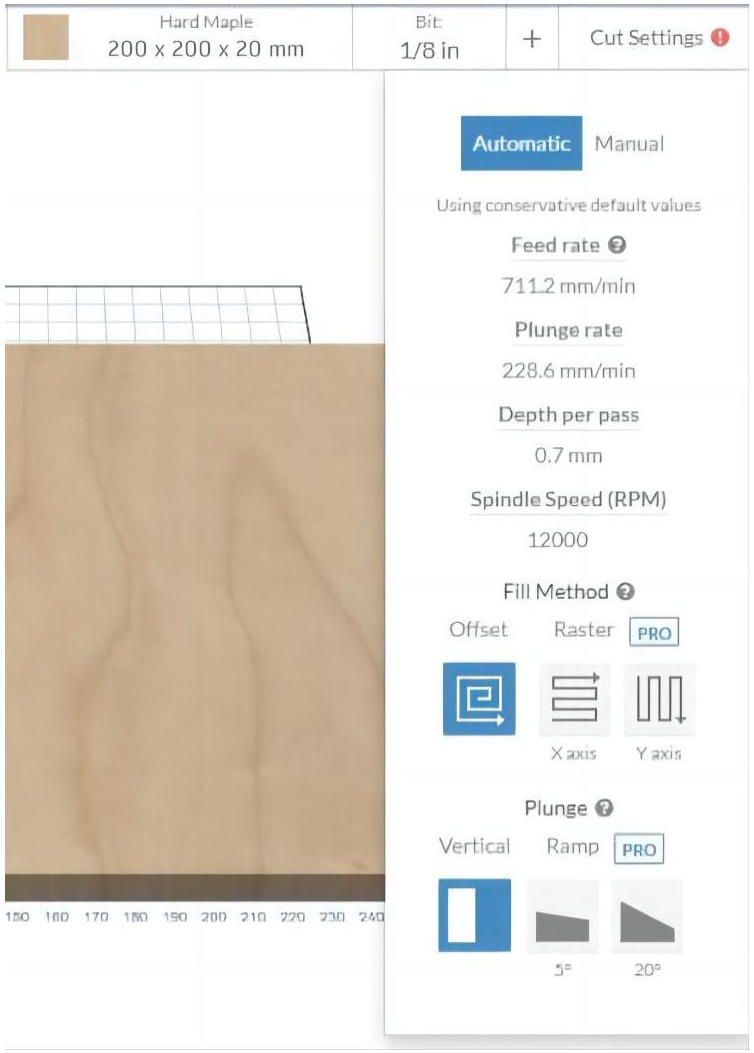
# Part 3 Recommended Settings - Easel

## Cut settings

Selecting the material type in Easel allows the software to calculate cut settings. Preliminary testing has shown that these Automatic settings are a good starting point for most users.

The user will need to adjust settings based on their personal experience and how the machine is operating. For smaller diameter bits, speeds WILL need to be drastically reduced to prevent bit breakage.

Note: If you are a CNC beginner, we recommend that you use automatic settings. The automatically set parameters can complete most engraving tasks.



# Part 4 FAQ

Q1：Why can't the computer connect to the machine?
A1:Open the device manager of the computer and check the serial port. If the serial port connection is successful, CH340 will be displayed. If not, please replace the USB interface or data cable.

Q2：Why can the machine only run for a while, then stop, then the software reports an error?

A2：Check whether you accidentally touched the E-stop button. If so, please rotate and pull up the E-stop button to reset it, and then unlock the E-stop through the software.

Q3：Why does the machine suddenly stop during operation, all the axIs cannot move and Spindle cannot rotate?A3：Please check whether the three axis hit the limit switch. If it hits the limit switch, the machine will be locked and cannot perform any operation. Please manually rotate the knob to move the axis that hits the limit switch away from the limit switch, and then unlock it in the software. Limit switch or long press the red reset button behind the control board.

Q4：Why does the axis make a buzzing sound when moving and is difficult to move?

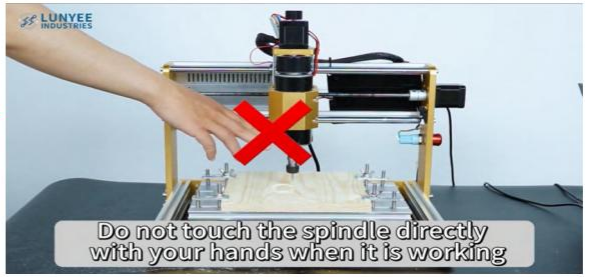
A4：This may be caused by the screw being squeezed and bent during transportation. Once you find sucha problem , please contact us immediately and we will send replacement parts for free.

Q5：Why can't the spindle be turned off?

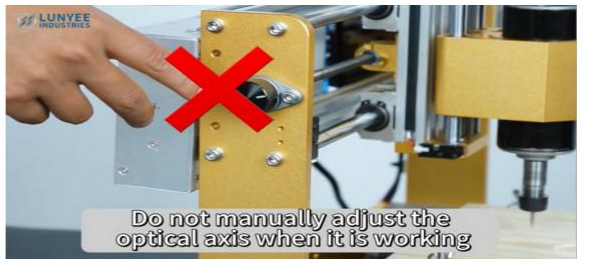
A5：The inability of the spindle to shut down may be caused by a power failure. Once you encounter such a problem, please contact us immediately for help.

# Part 4 Precautions

1.Do not manually adjust the optical  axis when it is working




2.Do not touch the spindle directly with your hands when it is working




3.Correct way to connect power




4.Adjust the supply voltage to match the household voltage and close the cover



# Part 5 Default GRBL Settings

$0=10 (Step pulse time)
$1=255 (Step idle delay)
$2=0 (Step pulse invert)
$3=1 (Step direction invert)
$4=0 (Invert step enable pin)
$5=0 (Invert limit pins)
$6=0 (Invert probe pin)
$10=3 (Status report options)
$11=0.010 (Junction deviation)
$12=0.002 (Arc tolerance)
$13=0 (Report in inches)
$20=0 (Soft limits enable)
$21=1 (Hard limits enable)
$22=1 (Homing cycle enable)
$23=3 (Homing direction invert)
$24=100.000 (Homing locate feed rate)
$25=1000.000 (Homing search seek rate)
$26=250 (Homing switch debounce delay)
$27=3.000 (Homing switch pull-off distance)
$30=1000 (Maximum spindle speed)
$31=0 (Minimum spindle speed)
$32=0 (Laser-mode enable)
$100=320.000 (X-axis travel resolution)
$101=320.000 (Y-axis travel resolution)
$102=400.000 (Z-axis travel resolution)
$110=5000.000 (X-axis maximum rate)
$111=5000.000 (Y-axis maximum rate)
$112=5000.000 (Z-axis maximum rate)
$120=300.000 (X-axis acceleration)
$121=300.000 (Y-axis acceleration)
$122=300.000 (Z-axis acceleration)
$130=600.000 (X-axis maximum travel)
$131=600.000 (Y-axis maximum travel)
$132=130.000 (Z-axis maximum travel)

# Part 5 Default GRBL Settings

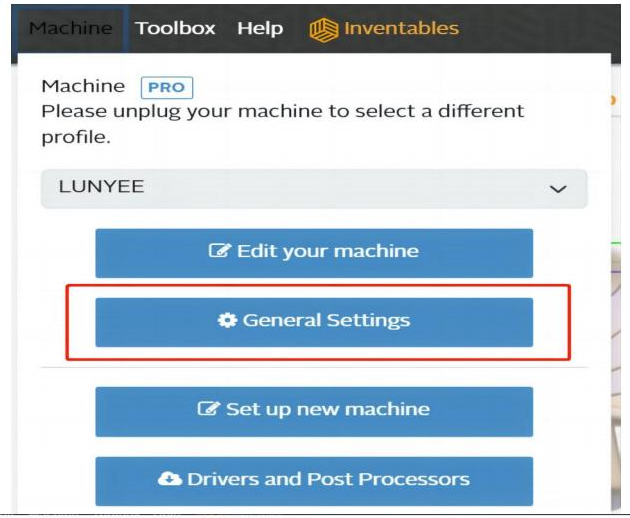
## Key GRBL Settings

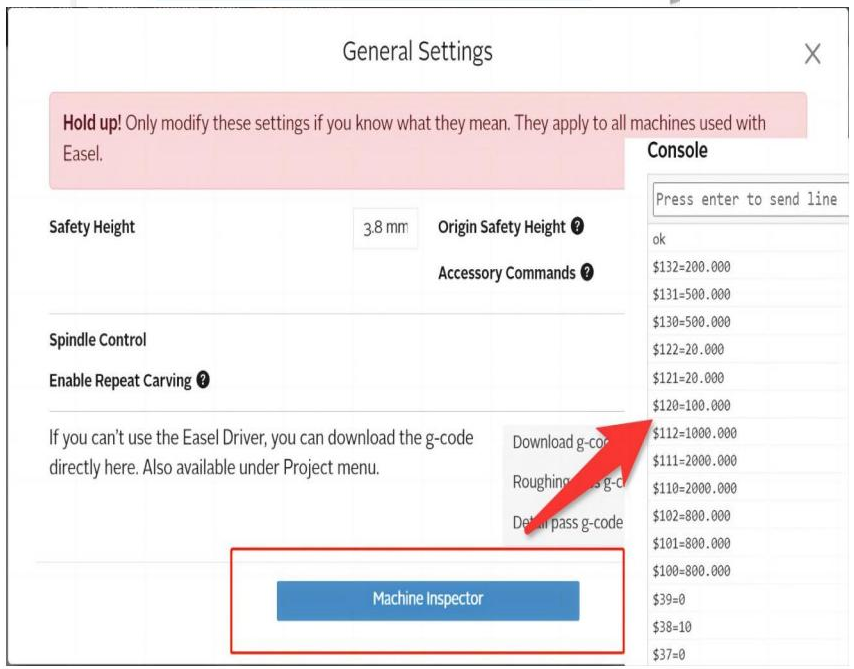
### Intro

The 3018 cnc motherboard has a series of settings on it that allow your machine to work correctly. We typically recommend that the end user not alter these settings, except as directed by Technical Support.

Go to Easel » Machine » General Settings» Machine Inspector.
This will display all the Grbl settings values from the router.
Copy these into a text editor and save the file somewhere as a baseline.
No matter how much you may mess up the settings,
you will always have a previous set to go back to.

You will see a list of $xx=yy lines where xx is the setting number and yy is the current value. To change a valuesend the command of $xx=zz in the console, where xx is the setting number and zz the new value. These will be permanently stored on the motherboard so they only need to be changed once.



# Part 5 Default GRBL Settings

## Key GRBL Settings

### $3 - Direction port invert mask 1/2

If an axis is moving in the wrong direction, you need to change one of the settings, namely $3,the direction port invert mask to be precise.

This is very easy to do. But before doing that:

- Check the connections from the stepper motors and make sure they are plugged into the correct X, Y and Z axis connections on the router board.
- Check that the axis is not binding by turning the screw by hand (disconnect the coupler first).
- Check that the grub screw on the coupler connecting the threaded rod to the stepper motor are tight and the coupler is not slipping.
- Swap the connection for the stepper motor into the router motherboard for that axis with another one. If it now moves (remember it is now a different axis) and the other one doesn't then it is likely to be a motherboard failure. Or possibly a faulty cable, so swap the cable with a working one just to be sure.

If your connections are fine along with your cables, stepper motors and board, then you need to change your $3 value stored in the motherboard.

# Part 5 Default GRBL Settings

## Key GRBL Settings

### $3 - Direction port invert mask 2/2

Use the following table: Select the row for your current $3 value, go across to the column for the axis to change and the value there is your new $3 value. This only allows you to change one axis at a time. If you need to change more than one, make multiple passes through the process.

Examples:

- Current $3 value is 2 and the X axis needs inverting: Select the 2 row and the X column, the new $3 value is 6.
- Current $3 value is 3, and the Y and Z axes need inverting: To invert the Y axis, select the 3 row and the Y column, the new $3 value is 1. To invert the Z axis, select the 1 row and the Z column, the new $3 value is 0.

| | New $3 Value | | |
|---|---|---|---|
| Current $3 Value | Invert X | Invert Y | Invert Z |
| 0 | 4 | 2 | 1 |
| 1 | 5 | 3 | 0 |
| 2 | 6 | 0 | 3 |
| 3 | 7 | 1 | 2 |
| 4 | 0 | 6 | 5 |
| 5 | 1 | 7 | 4 |
| 6 | 2 | 4 | 7 |
| 7 | 3 | 5 | 6 |

In Easel, type $3=x in the console command box where x is the revised number and hit enter to send it to the route motherboard. Everything should now be moving in the correct direction. If you changed any settings, make a note in your file containing your original settings just in case.

# Part 5 Default GRBL Settings

## Key GRBL Settings

### $100,$101,$102 - [X,Y,Z] steps/mm

These parameters controlled the distance travelled by each axis. These values are the number of microsteps required to move the spindle on each axis by 1mm. $100 sets the X axis, $101 the Y axis, and $102 the Z axis. These should all be set to the values listed in the Default Settings Table at the end of the manual.

### $30 and $31 – Max and Min Spindle Speed (RPM)

This sets the spindle speed range between 0.0% and 100%. Since the CrossRiver has an external speed control, these values should not require adjustment.

### $32 – Laser Mode

This is a Boolean value or on/off. 1 is on (laser connected) and 0 is off (spindle connected). If you don'thave a Laser module then just leave this set to 0. If you do wish to use the optional laser module, you will need to set Laser Mode on (1) when using the laser module.

# Part 5 Default GRBL Settings

## Key GRBL Settings

### $110, $111 and $112 – [X,Y,Z] Max rate, mm/min

Set in mm/min, these set the maximum rate each axis can move. Whenever Grbl plans a move, it checks whether or not the move causes any one of these individual axes to exceed their max rate. If so, it'll slow down the motion to ensure none of the axes exceed their max rate limits. This means that each axis has its own independent speed,which is extremely useful for limiting the typically slower Z-axis. Leave them at $110=5000, $111=5000 and $112=5000.

You can check all the defined GRBL settings and what they mean using the link below:
https://github.com/gnea/grbl/wiki/Grbl-v1.1-Configuration

Note – changing GRBL settings beyond those mentioned above is not recommended by LUNYEE, and is done at your own risk.

# Part 6 Best Practices For Assembly&Setup

## Quick Guide

**1.** Ensure everything is **wired and powered correctly** per the manufacturer guidelines.
**2.** Ensure your axes **move freely** and don't obviously bind. If you can't easily tell, try removing your steppers and check if they run under no load.
**3.** Ensure your stepper motors and axes linear mechanisms are all tight and secure. Small set screws on drivetrain components becoming loose is a very common problem. Re-tighten and try applying some non-permanent thread locker **(Loctite blue)** if it continually loosens.
**4.** For more difficult issues, try the process of elimination to quickly **isolate the problem.** Start by disconnecting everything from the Arduino (controller board). Test if Grbl is operating ok by itself. Then, add one thing at a time and test.
**5.** If your steppers are powered and making a grinding noise when trying to move, try **lowering the'$'acceleration and max rate settings**. This sound is a sign that your steppers are losing steps and notable to keep up due to too much torque load or going too fast.
**6.** Next, you need to make sure your machine is moving in the correct directions according to a Cartesian(XYZ) coordinate frame. Check all axes. If an axis is not moving correctly, **alter the $3 direction port mask setting to invert the direction**. Just keep in mind that motions are relative to the tool. So on a typical CNC gantry router, the tool will move rather than the fixed table. If the x-axis is aligned positive to the right, a positive motion command will move the tool to the right. Whereas, a moving table with a fixed tool will move the table to the left for the same command, because the tool is moving to the right relative to the table.
**7.** Finally, tune your settings to get close to your desired or max performance. Start by ensuring your **$100,$101, and $102 axes step/mm** settings are correct for your setup.

# Part 7 Terminology

**CNC (Computer Numerical Control):** A CNC machine is a motorized platform controlled by a computer according to specific input instructions.

**Units (mm or inches):** It is imperative that the units match. If you tell your software to work in inches and the router works in mm, you are going to get very bad or compressed results.

**G-code:** G-code stands for "Geometric Code". It is a programming language for CNC that instructs machines where and how to move. Most machines speak a different "dialect" of g-code, so the codes vary depending on type, make, and model.

**Controller or Motherboard:** This is the brain of your CNC working with an Arduino microprocessor. This is the bit of electronics that controls everything: stepper motors (axis movements),spindle motor (cutting speeds), laser (power settings), jogging, G-code commands received via USB or an offline controller, etc.

**Grbl :** This is the software which runs on the motherboard. It takes the G-code and translates it into the movements and speeds needed to cut a line for example.

**Closed-Loop Stepper motor:** A motor that is told to turn in small steps like turn clockwise 1.7˚ as opposed to a normal motor that will continue to rotate as long as the power is supplied. Each step of the motor will turn the threaded rod which will move the the axes connected to it. Closed-Loop motors use an encoder to ensure the motor is turning the desired amount, to help eliminate "lost steps"

**Spindle Motor:** This is the big electric motor which holds the cutting bit.

**Stock:** Whatever you are trying to engrave, cut or carve.

**Bed:** This is the flat MDF plywood or aluminium bed which holds the stock with slots in it to allow clamps to hold the stock down.

# Part 7 Terminology

**ER11:** This is the piece that fits onto the end of the spindle and holds the bit.

**Collet:** The piece inside the tool holder (ER11) which as it is tightened into a tapered hole contracts around the bit so holding it.

**Bit (or Tool):** The sharp blade which rotates and cuts bits out of the stock.

**Shank:** The part that goes into the collet of the tool holder.

**Flute:** A cutting edge at the end of the bit.

**Limit Switches or End Stops:** Switches which trigger at the limits of movement for each axis. During the operation, they will stop the router from trying to move past its physical limits by generating an alarm.

**Toolpath:** The path that the tool is moved along in order to do the job. Your software will provide a toolpath simulation so you can see on your computer what is going to happen before you send it to the router.

**Passes:** This is the number of times you tell the router to retrace the same path. If you want to cut a groove or pocket in a piece of wood 5mm deep you are not going to do it in one single, go down 5mm then cut! If the wood is very soft and the bit is very sharp, maybe, but you take the risk of overloading the router, wearing out or breaking the bit by trying to make it do too much work in one go. You can get around this by telling Easel to make multiple passes cutting out a little each time. As a general rule, the less material you take out the better the finish quality but the longer it takes.