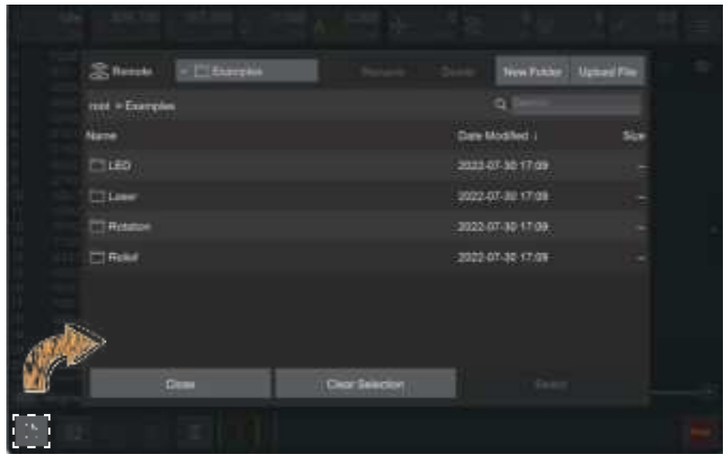
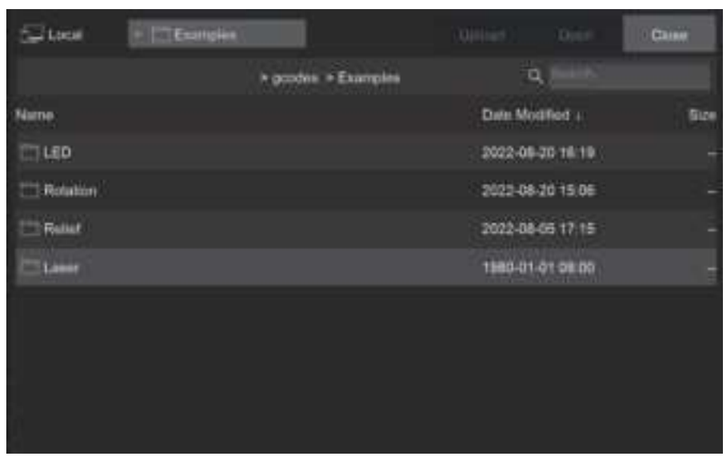
Task Toolbar

1、File management and selection
Carvera Air's G-Code is executed in the controller to ensure the task's efficiency and stability and avoid task failures caused by WiFi or USB connection instability. Therefore, the G-Code file needs to be uploaded to the machine before running the program. We have created examples folder on the machine and have already uploaded sample G-Code files.

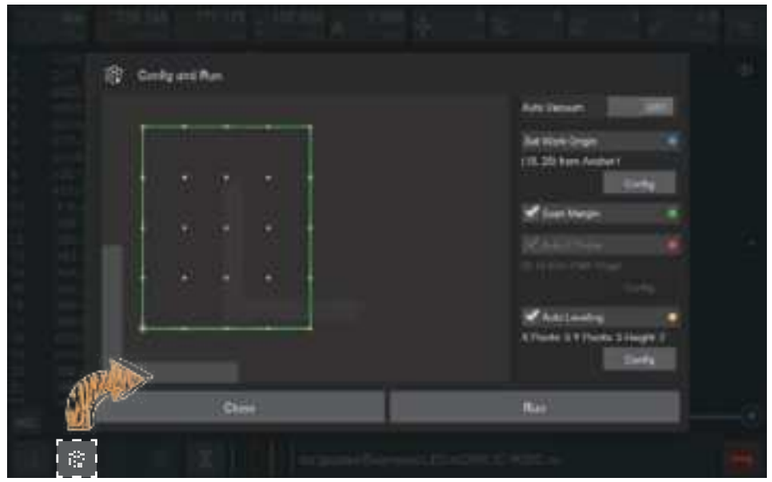


1.1. Remote file management:
1.1.1. Rename: Rename files or folders in the machine.
1.1.2. Delete: Delete files or folders in the machine.
1.1.3. New Folder: Create a new folder under the current file path.
1.1.4. Upload File: Switch to local file interface for uploading.
1.1.5. Close: Close the file management interface.
1.1.6. Clear Selection: Clear the currently selected G code file.
1.1.7. Select files: Select the G-Code file to run.
1.1.8. Recent Places: Short cut for recently used directories.
1.2. Local file browsing:



It opens the gcodes subdirectory under the local program installation directory by default. Therefore, we recommend putting your G-Code files here for easy management.
Upload: Select and upload local files.
Open: Open the file and preview it without uploading it (you can also do this without connecting the Carvera).
Close: Back to the remote file management interface.
Recent Places: Short cut for recently used directories.

2. Task configuration and execution
If you have used CNC before, you definitely know that a CNC machining task requires a lot of preparation work, including setting the work coordinates (XY axis tool setting), Z-axis tool setting, workpiece levelling (PCB processing) and so on.
Because Carvera Air has automatic detection and leveling functions, we have integrated these settings and task execution into one interface.
We provide an innovative method that using “anchor points" for work coordinate setting, allowing you to locate XY-axis positions accurately through easy configuration.



2.1. Auto Vacuum: Allows the machine to automatically toggle the dust collection signal during task execution enabling automatic control of external dust collection equipment. We will provide detailed instructions in the 'Tool Kit' chapter.
2.2. Machining area preview: Display the preview of the machining area according to the current work coordinates and G-Code file.
L-shaped gray symbol: Anchor point position. You can choose to install the L-shaped bracket to be located either to point 1 or point 2.
Blue Circle: Zero position of work coordinate.
Green line area: G-Code file machining range.
Green Bold line area: Indicates that Scan Margin is activated to automatically scan the area before machining.
Red Circle: Z probe position when Auto Z Probe is selected.
Yellow Circle matrix: Levelling matrix when Auto Levelling is selected.

2.3. Set Work Origin: Set the work coordinates zero point relative to anchor points - the X/Y axis distance relative to anchor point 1 or 2. (Only X distance is needed when performing 4-Axis machining). Please note that the work coordinate settings will take effect immediately.
2.4. Scan Margin: Scan the rectangle path area before machining. When scanning, machine will switch to the wireless probe and turn on the red laser for observation. We recommend new users turn on this while running job.
2.5. Auto Z Probe: Z-axis tool setting is required after changing the workpiece or the zero points of the work coordinate. When doing Z probe, machine will automatically switch to the wireless probe and do probe at the set position.

Work Origin: Perform Z axis probe at a certain distance related to the X/Y axis of the working coordinate zero point.

Path Origin: Perform Z axis probe at a certain distance related to the actual X/Y machining starting point (lower left corner). This method is selected by default.

2.6. Auto Leveling: If requiring uniform machining depth, please select the automatic leveling option such as PCB engraving. You can set the leveling matrix size and the lifting height when moving horizontally during the leveling process. The less the flatness, the higher the lifting height needs to be. The higher the requirement for machining consistency, the denser the matrix. For PCB engraving, it is better to have matrix dots spaced about 1 cm apart.
2.7. Run: Click to start machining process. If you set the scanning area, Z-axis tool setting or automatic leveling, the G-Code file will be executed after the automatic detection is completed.

Note: The wireless probe is a precise device that integrates mechanics and electronics, which is easy to damage. Please be careful when using it. Make sure no obstacles can block its way. We suggest new users turn on the scan margin function to preview the route. It is recommended to leave sufficient margin between L bracket position and path start point, greater or equal to

3、Task control

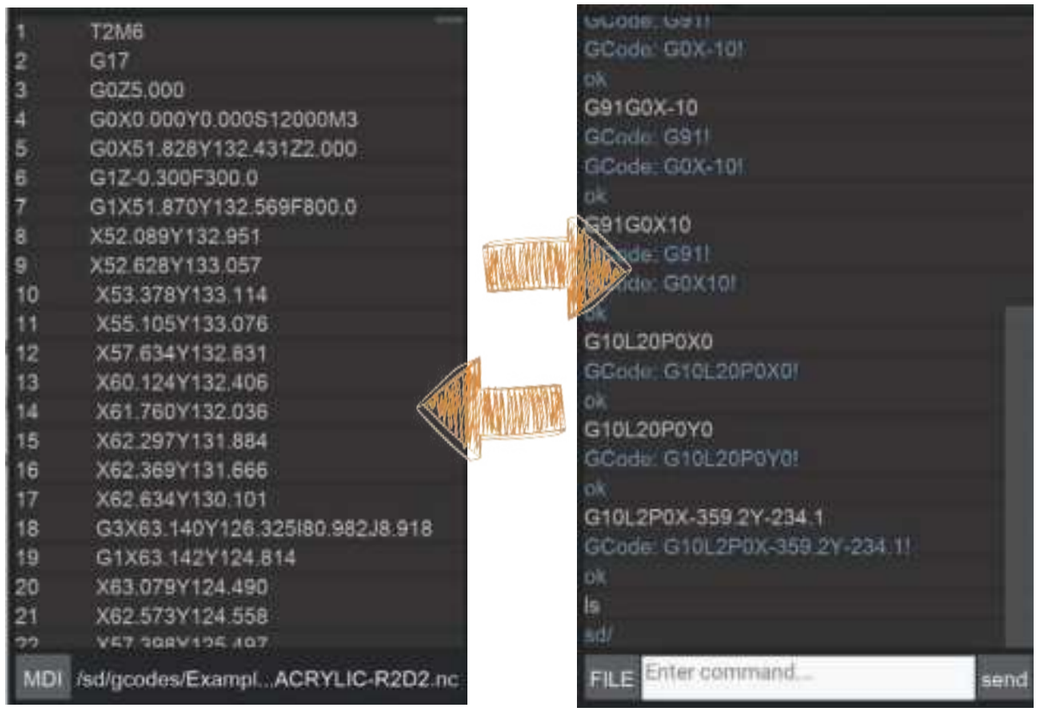
3.1. Task pause: Pause the running G-Code task and you can manually control the machine afterwards. But if you want to resume the task, please remember to turn on the spindle, or it may damage the tool if it resumes without rotating the spindle.
3.2. Task stop: Terminate the current G-Code task.
3.3. Task hold: Similar to task pause, but the pause speed is faster and cannot control the machine manually during holding.
3.4. Task track: Display the current G-Code task name, running time, percentage and other information.
3.5. Emergency stop: Immediately stops the current task, turn off the spindle, the same function as the physical button in front the machine.

Note: The current task cannot be stopped immediately when the task is paused or held. It has to wait for the buffer zone finish. If emergency occurs that needs to stop the machine immediately, please click the emergency stop button in the software or press the physical emergency stop button on the front panel of the machine. Carvera is built by closed-loop servo motors, and it saves the coordinate and status at each step. Therefore, no needs to reset the tool after reboot/unlock.

G-Code/MDI

1. G-Code interface: Display the currently opened remote or local G- Code file. When the task is running, it will track and highlight the running line in real-time.

2. MDI: Display detailed send/receive commands, similar to the log information. In specific cases, you can manually enter the g-code for operation and diagnosis. Enter “clear” to clear the current command area.

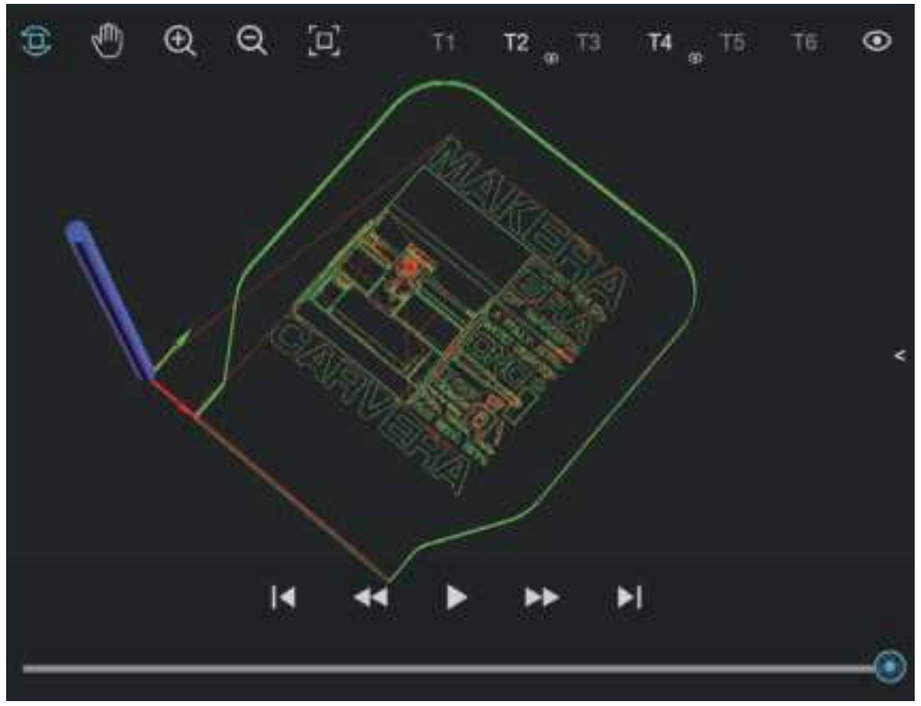


G-Code Preview

1. Graphical preview: Open the G code file to preview the G code graphics in the tool path preview area. Green lines are G1/G2/G3 code, and red lines are fast moving G0 code.

2. Display control toolbar: You can pan (right mouse button), rotate (left mouse button), zoom in (scroll wheel up), zoom out (scroll wheel down), and restore the preview (double-click) . You can also select to show/hide the G-Codes for for different tools.

3. Playback toolbar: When the task is not running, you can play, fast forward, or backward the preview. When the task is running, the preview graph shows the real-time machining trace.



Manual Operation

1. Jogging control: Manually control the movement of the X/Y/Z axis and rotate the A axis at G0 fast speed (3000mm/min by default). You can set movement distance.

2. Status control: You can unlock, reboot or reset the device.

3. Automatic detection: Automatically scan the machining area; perform Z-axis probe or automatic leveling. The difference between here and the task configuration is that you can just apply one-time detection, without executing the G code file.

4. Move to the specified location: Provide a shortcut to quickly move to the specific location; including anchor points 1, 2, working zero points, G-code starting point, and clearance point (the upper right corner next to the machine zero points by default. You can quickly move the machine to there before cleaning the working surface after machining process end).

5. Set Orign: Manually set the work origin coordinates or use the manual probe to perform X, Y, and Z axis probing. See detailed introductions in the next chapter.

Errors

The device triggers an alarm when encountering an abnormal. Some alarms can be closed by unlocking. Some require rebooting the machine.

Alarm Type	Reboot	Causes
Halt Manually	No	Press emergency button on the machine or software
Home Fail	No	The return zero limit switch did not trigger
Probe Fail	No	Exceeded the maximum detection distance but has no signal
Calibrate Fail	No	Tool calibration probe malfunction
Spindle Overheated	No	Spindle overheating
Cover opened	No	The protective cover is opened during machining (cover detection enabled)
Wired Probe Error	No	No response from the Wired probe
Emergency Stop	No	Emergency stop is pressed
Hard Limit Triggered	Yes	Motion out of range
X/Y/Z Motor Error	Yes	X/Y/Z Servo motor block
Spindle Error	Yes	Spindle stall or other errors
SD card Error	Yes	SD card reading error
Machine Is Sleeping	Yes	Machine is sleeping

General Workflow

Different from the cumbersome operation process of general CNC, Carvera Air greatly simplifies the machining process. The general operation steps are as follows:

1. Turn on the Carvera Air device and wait for the homing to end.
2. Fix the workpiece to the anchor point.
3. Open the control software and connect to the device.
4. Upload and open the G-Code file.
5. Open the task setting box, and set the working zero point and automatic detection rules.
6. Start the job, and change tools when a tool change command is triggered and the machine prompts.
7. Wait until the machining process ends.

Tool Kit
Instruction

Wired Probe

The wired probe is a key component supporting Z-axis probing, machining boundary scanning, and automatic leveling.

1. Installation: It is stored on the right side of the spindle by default, and the cable has been connected and is ready for immediate use. The installation and removal method for the wired probe is the same as that for a standard milling bit.

2. Probing: When the wired probe is triggered, the green indicator light turns on.

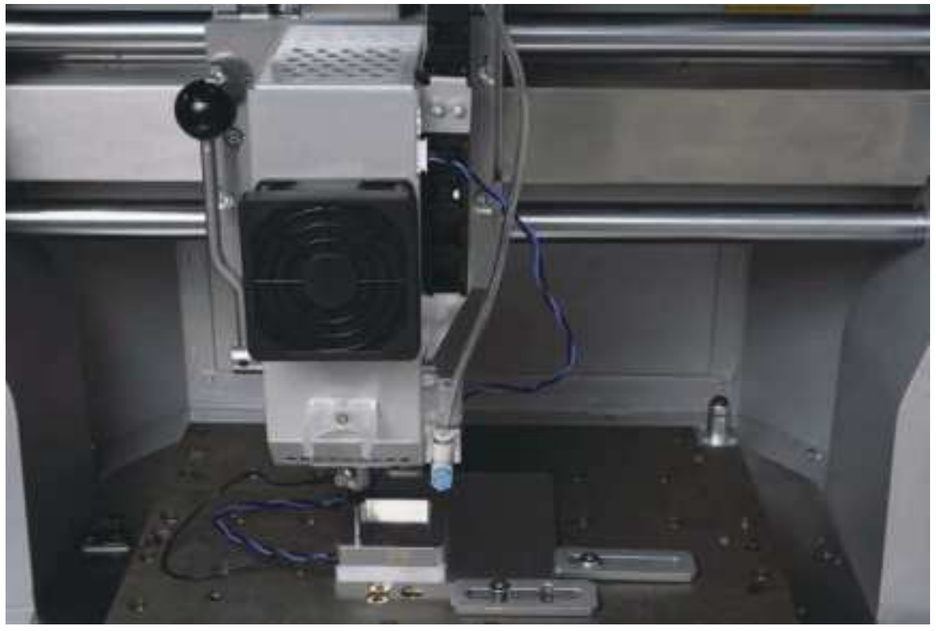
3. Laser indicator: Press the wired probe twice to turn on the laser indicator (used for manual tool setting of the XY axis). The laser indicator will also be turned on automatically when probing, scanning boundaries, or auto-leveling.

4. Test: Open the drop-down list in the status toolbar and select the diagnostic function, the diagnostic status dialog box will pop up. Press the wired probe and you can see the Probe signal is triggered,
indicating the wired probe is working.

Note: Unlike the crash of milling bits, the crash of the wired probe may cause serious damage to the probe. Please use the wired probe with caution. Always observe its traveling path, and stop the machine when finding any obstacles to protect the wired

Manual Probe

Use case: Normally, using the wired probe and the anchor-based positioning system is quite enough for most jobs. But when you need to place the workpiece not at the anchor point and need to accurately find the origin, you can use the manual probe.



Usage:

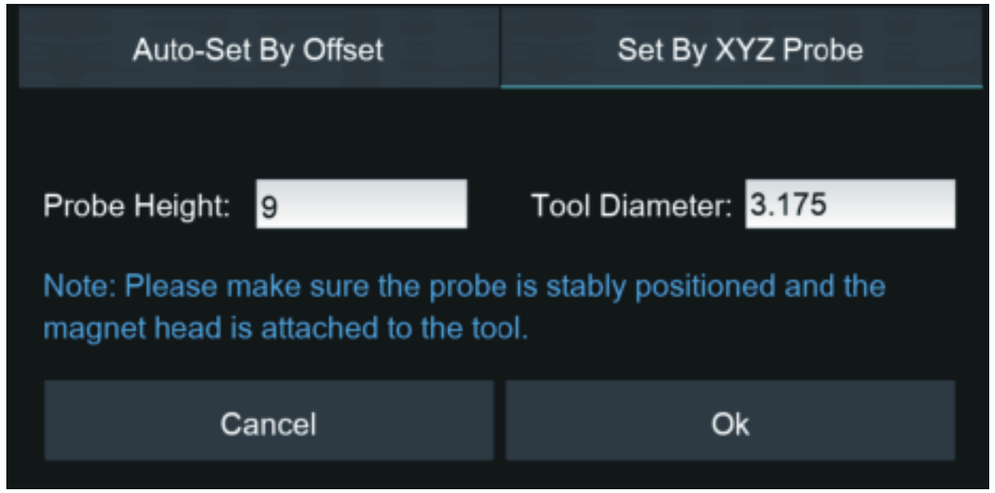
1. Unplug the wired probe and plug the manual probe into the 4-pin socket on the top right of the spindle shell.

2. Place the manual probe (white plastic side) against the lower left corner of the workpiece firmly.

3. Move the machine and let the milling bit be positioned in the square area of the manual probe.

4. Attach the magnetic end of the manual probe to the spindle shaft as shown.

5. Click the 'Set Origin->Set By XYZ Probe" function, and set the height offset and the diameter of the milling bit. (It is recommended to use the 3.175mm diameter test rod we provided for probing so that the default parameters can be applied directly)



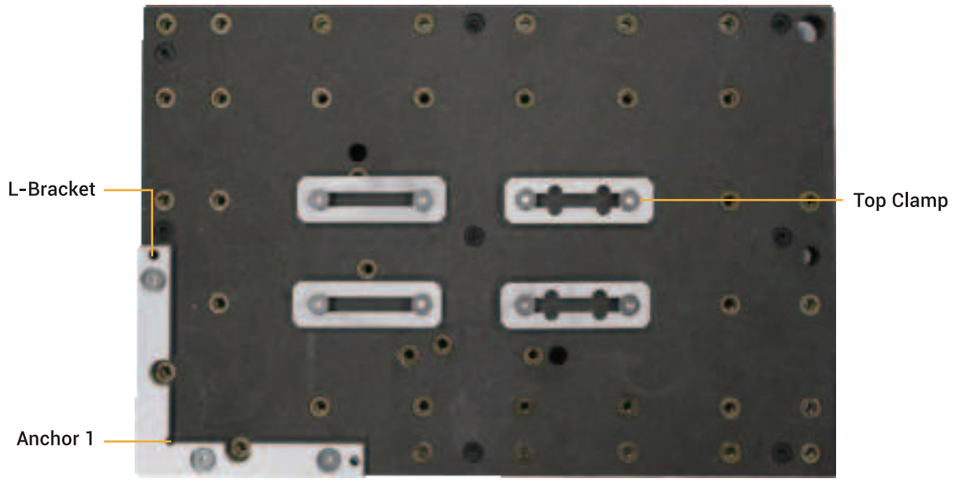
Note: Tip: The manual probe process will automatically set the origin of X, Y, and Z axes. There is no need to set them again.

Emergency Stop Button

Just like the main button in front of the Carvera Air machine, when any unexpected situation occurs, you can quickly press the emergency stop button to stop the machine, and the machine will stop immediately and enter the alarm state. You need to go to the control software to unlock the machine before continuing to use it. The emergency stop button has a self-locking function, just turn the emergency stop button clockwise to unlock it.

Workholding Tools

Workholding is one of the most important steps when using a CNC machine. Carvera Air provides two different workholding methods and corresponding tools to adapt to different types, shapes, and sizes of workpieces. While holding the workpiece, you can also locate the workpiece to the pre-defined position accurately by using Carvera Air's anchor-based system.



1. L-Bracket: Carvera Air provides a L-shaped bracket, which can be fixed at one of the two anchor points with two 4mm dowel pins and three M5 screws (the thick bracket uses long screws). The lower left corner is Anchor 1, and the middle position is Anchor 2. There are two semi-circular openings of the thin positioner, you can put two M5 screws to fix the lower-left corner of the workpiece there.

2. Top Clamps: The top clamp usually fixes the workpiece with a thickness of less than 2 cm together with the L-Bracket.The purpose of the top clamp with a cross groove is to facilitate the use of
long sides to fix the workpiece. We recommend using shims at the end of top clamps to fix

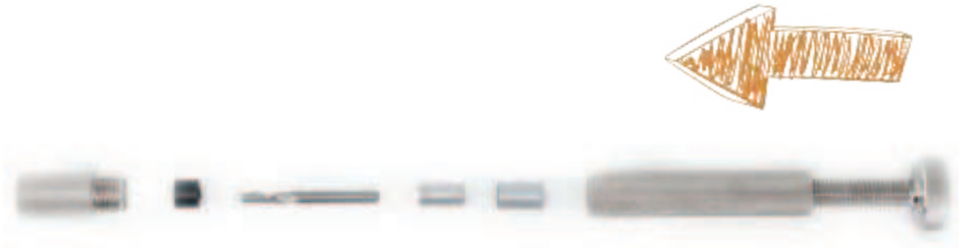
Note: The top clamp is not very thick, so please do not screw too tight.

Note: If you need to cut through a workpiece, we highly recommend placing a 1-2mm thick waste board (as the complimentary one) under the workpiece. This can avoid damage to the workbench.

Note: Please select corresponding length screws to fix different workpieces to avoid scratching the plate under workbench.

Bit Collar Installer

To better cooperate with Carvera Air's quick tool-changing mechanism (Limit the installation depth), the collar installer can be used to install collars when replacing new milling bits. The collar installer can do both installation and removal. (the collar is reusable).



Collar installation: As shown in the figure above, unscrew the front part of the installer. Insert the collar and the tool. Put in the installation metal ring (support 3.175/4/6/6.35mm). Loosen the tail pressure screw, screw back the front part, and tighten the tail pressure screw to complete the installation.After the installation is complete, the collar will be embedded with the tool, leaving a length of about 12mm at the tail for clamping.

Collar removal: As shown in the figure above, unscrew the front part of the installer, put the tool with the collar. Put in the removal thimble, loosen the tail pressure screw, screw back the front part, tighten the tail pressure screw, and the removal is complete.

Note: The milling bits are sharp, be careful when install and uninstall collars.

Dust Collection Module

Chip evacuation is an important part of CNC but is usually ignored by other desktop-level CNC machines. Carvera Air provides a convenient way for you to attach an external vacuum.

1. Use cases: The primary factor in deciding whether to use the dust collection system is potential interference. If obstacles in the machining path block the dust shoe, do not use the dust collection system. Dust collection is ideal for machining thin and flat workpieces, such as plates. Avoid using dust collection when machining thick or irregular workpieces, or when utilizing the 4th axis.

2. Installation/Removal: Two fixed magnets are positioned above the dust shoe, allowing for easy magnetic installation and removal. The notch on the dust shoe aligns with the spindle shape for precise positioning. When not in use, the dust pipe can be secured into the upper latch.

3. Dust Extraction: Connect an external vacuum cleaner to the dust port on the rear of the machine (22mm inner diameter; an adapter may be required). You can manually start and stop the dust collection system.

Alternatively, you can automatically control the external dust collection equipment using the machine's external control port. For detailed instructions, please refer to the official knowledge base: https://wiki.makera.com/

Note: Installing the dust shoe may interfere with rapid tool changes. You can choose to remove the dust shoe during tool changes based on your usage preferences.

Note: Use caution when installing or removing the dust shoe, particularly if a milling bit is present in the spindle.

Spindle Collet Installer

Carvera Air comes with a 1/8 inch(3.175mm) spindle collet. This is the commonly used size for desktop-level CNC machines, which can meet most machining requirements. For special sizes such as 4mm/6mm/6.35mm, you can change the spindle collet and the tail shaft of the wired probe.

1. Change spindle collet: Use the handle to drop the current tool, insert the spindle collet installer into the current collet, and rotate counterclockwise to remove the current collet. Use the same way to install the new collet. (Do not over tighten)

2. Change the tail shaft of the wired probe: Rotate the wired probe's tail shaft counterclockwise to uninstall. Use the same method to change the new tail shaft. (Do not over tighten)

Air Assist Module

Use case: There are two main use cases for the air assist module. One is for chip removal and cooling during CNC machining, especially when machining metal materials. The second is to prevent the material from burning during laser engraving to improve the engraving quality.

Installation: An normal small air pump is more than enough for the Carvera air assist module, insert the 8 mm pipe into the plug at the back of the machine, and ensure that the air pipe is firmly fixed.

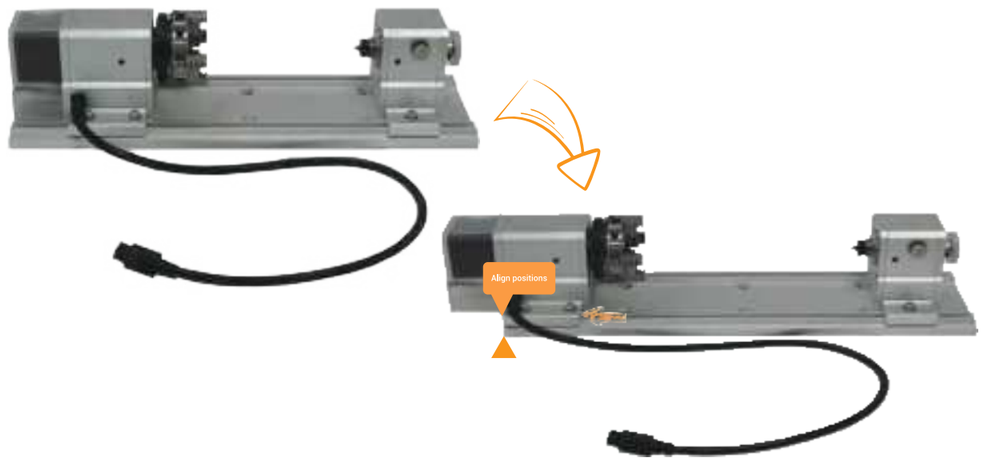
Air control:
1. Adjust the blue knob at the end of the air assist module to control the airflow, pull the knob to adjust, and press the blue knob to lock. Turn the knob clockwise to decrease the flow and counterclockwise to increase the flow.

3. The angle of the air nozzle can be adjusted to match different tool lengths and laser focus position.

Note: The air assist module and dust collection module cannot operate simultaneously. Please remove the dust shoe before using the air assist module. When the air assist module is not used for an extended period, turn off the air pump.

Rotary Module

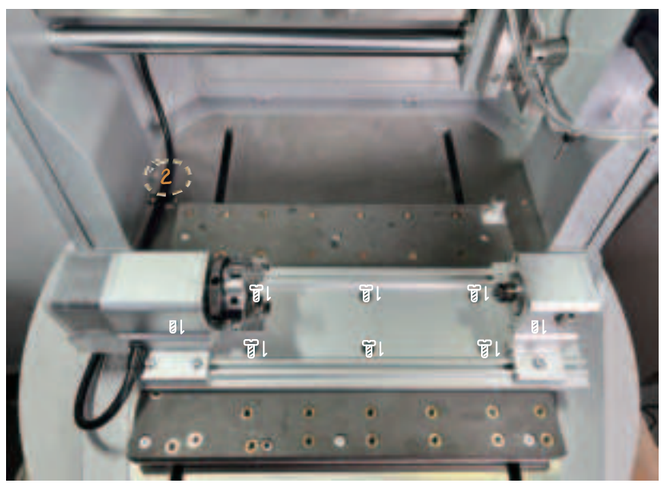
1. Preparation: To facilitate transportation, the spindle box for the 4th axis is fixed in the right position. Refer to the illustration to loosen the four screws securing the spindle box. Carefully align the leftmost side of the 4th axis spindle box with the left side of the 4th axis base plate.



2. Installation:

2.1 Using two 4mm alignment pins, position the 4th axis at the center of the worktable as shown in the illustration. Fasten the 4th axis to the worktable with six M5×20 screws.

2.2 Ensure the machine is powered off before proceeding.Bend the 4th axis connection cable towards the rear inside the machine. Insert the cable into the 4th axis interface at position 1 in the illustration (ensure the dust cover at the interface is open first). Using the hex key and M4×8 screws provided in the 4th axis packaging, secure the cable clamp to position 2 shown in the illustration (remove the existing screws from the original position first).



3. Workpiece holding:

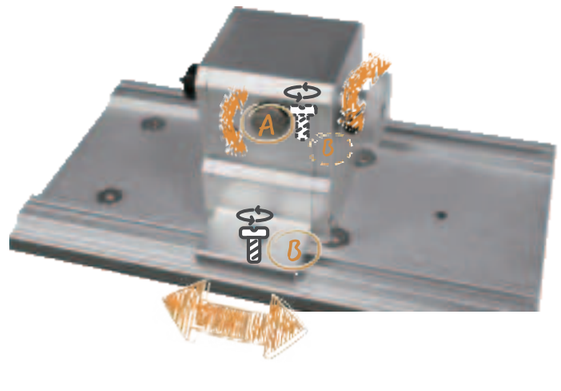
3.1. Loosen the locking screw at position A on the tail stock as shown in the figure, adjust the tail stock tip to the right side by turning the knob counterclockwise.

3.2. Loosen the 2 fixing screws at position B.

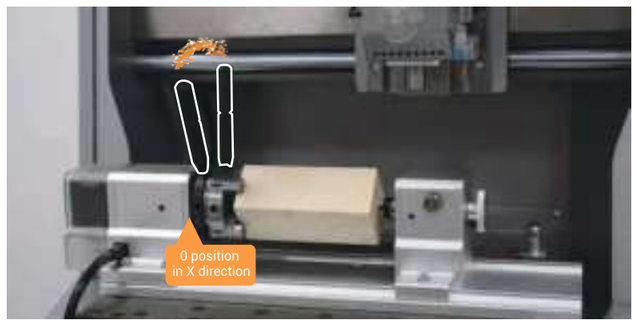
3.3. Use two wrenches to adjust the opening size of the chuck and place the workpiece in.

3.4. Align the tail stock tip to the end of the workpiece, and tighten the two fixing screws at position B.（We highly recommend drilling a small hole at the end of the block tail for better holding strength, especially for hard materials.)

3.5. Use two wrenches to tighten the chuck, push the tail stock tip close to the workpiece by turning the knob clockwise, and lock the locking screw at position A. (Don't push too hard to the workpiece, it's ok when there is no gap and backlash, better drill a small hole first at the center of the workpiece tail for fixing.)



4. Software Settings: The right edge of the headstock is the reference point for setting the working coordinates of the rotary module. When performing rotary machining, you only need to set the distance between the X axis and the reference point and set Y to 0.



Note: Usually, you don't need to move the head stock, because the reference point on the head stock should be fixed for precise positioning. Ensure that the holding position/size/G-Code/work coordinate of the workpiece match with each other, otherwise it may cause damage to the module or tool bit.

Laser Module

The Carvera Air machine has an optional 5W diode laser module, which can engrave wood, plastic, and other materials, an excellent complement to the Milling function.

Installation/Removal: Insert the laser module as you would a standard milling bit, ensuring its orientation matches the direction indicated in the image. Confirm that the laser module does not rotate post-installation. Connect the cable to the laser module and to the 3-pin socket located at the top right of the spindle casing.

Optional - you can use the included silicone tube to connect the air inlet of the laser module to the air nozzle for Air Assist during laser processing, achieving better laser engraving results.

Note: The focal length of the laser module is fixed at approximately 5mm below the laser module (including the protective plate). Please ensure the material is flat, and make sure there are no obstacles higher than 5mm around the workpiece.

Note: Always wear laser protection goggles when using laser function.

CARVERA AIR

Feed & Speed

The following recommended parameters are based on current tests. The machining speed with small diameter tools/hard materials should be slow and fast in the opposite. We will conduct more tests and provide more detailed parameter recommendations in the future on our website.

Material	Tools	Milling depth (mm)	Feed Speed (mm/min)	Plunge speed (mm/min)	Spindle speed (RPM)
PCB	V-bit	0.1	200-500	200	12000
Wood	Single flute spiral bit	0.5-2	500-1000	300	10000
Plastic	Single flute spiral bit	0.5-2	500-1000	300	10000
Carbon/ Glass fiber	Corn bit	0.3-0.5	500-1000	300	10000
Aluminum/ copper	Single flute spiral bit for metal	0.1-0.2	300-500	200	12000

Note: Please start the test from the lower limit of the parameter, and adjust them based on the test results.

CAM

1. Summary

Carvera Air adapt standard G-Codes and is compatible with the open-source GRBL rules. Therefore, when using CAM software, select GRBL or standard G-Code should be all right. For laser engraving, you need to add sentences in front of the G-Code to switch to laser mode (M321), and return to the laser Z focus point (G0Z0), and add (M322) to exist laser mode after finished.

2. Software Recommendation

For Beginners: We recommend MakeraCAM as your CAM learning software. You can visit www.makera.com to learn more and download it.

For Experienced Users: If you are proficient with CNC, you can confidently use third-party CAM software such as Fusion 360, VCarve, etc.

Future Enhancements: We will continue to enhance the functionality of MakeraCAM and provide support for additional third-party CAM software.

Instruction Manual | www.makera.com | support@makera.com