SIMPLY TECHNOLOGIES

SIMPLY SERIES
OWNER'S MANUAL

Simply Technologies Group, Inc.
www.simplytechnologies.xyz
1-800-288-2961

TABLE OF CONTENTS

WELCOME TO THE SIMPLY SERIES

Congratulations on your purchase of a **Simply Technologies SIMPLY Series** CNC machine. You've chosen a versatile, cost-effective platform designed to bring precision and reliability to schools, makerspaces, and small businesses alike.

The new SIMPLY Series includes three sizes, each with prismatic guides, ball screws, and a powerful, yet easy to use DSP pendant controller. We also provide a full complement of accessories ensuring that you'll receive a package that's specifically tailored to your CNC needs

At Simply Technologies, we believe CNC technology should be approachable, not intimidating. This manual is your guide to safe installation, confident operation, and long-term maintenance of your machine, so you can focus on the work that matters.

If you have any questions, visit **www.simplytechnologies.xyz/support** or contact us directly at **steve@simplytechnologies.xyz**.

SECTION 1: SAFETY INFORMATION

Safety is everyone's responsibility. Please read and understand this section before operating your machine.

1.1 General Safety

- Read this manual in full before using the machine.
- Only trained and qualified personnel should operate this machine.
- Do not bypass any safety systems or operate the machine with guards removed.
- Always wear appropriate Personal Protective Equipment (PPE) including:
    - Safety glasses or face shield
    - Hearing protection
    - Dust mask or respirator when cutting materials that generate fine dust
- Tie back long hair, remove jewelry, and avoid loose clothing.
- Keep children, visitors, and untrained personnel away from the work area.

1.2 Electrical Safety

- The SIMPLY Series requires 110V, Single Phase 15A (or optional 220V, Single Phase, 10A) power with proper grounding.
- Do not use adapters or modify the supplied plug.
- Only qualified electricians should perform wiring changes.
- Inspect all cords regularly for damage and replace if necessary.
- Avoid using extension cords. If unavoidable, use a heavy-duty, grounded cord no longer than 10 feet.

1.3 Workspace Safety

- Ensure the floor can support the machine's weight and workpieces.
- Keep the area clean, dry, and free of obstacles.
- Avoid placing materials on or leaning against the gantry or rails.
- Properly secure dust collection hoses to avoid drag on the gantry.
- Ensure adequate clearance around the machine for safe material handling.

1.4 Operational Safety

- Never leave the machine unattended while running a program.
- Always be ready to activate the Emergency Stop button.
- Ensure all workpieces are properly clamped before starting.
- Avoid touching moving parts, especially the spindle and cutting tools.
- Let tools cool before handling after use.
- Regularly inspect bits and collets for wear or damage.

SECTION 2: E-STOP & LOCKOUT PROCEDURE

2.1 Emergency Stop (E-Stop) Function

Your SIMPLY Series is equipped with an Emergency Stop Button located on the Control Box. Use this button to immediately stop all machine motion and spindle operation in the event of unsafe conditions or unexpected behaviour.

How to Activate the Emergency Stop

1. Press the Red Emergency Stop Button firmly.
2. Assess the situation to ensure no one is at risk and that the machine is safe to restart.

How to Reset After Emergency Stop

1. Twist the Emergency Stop Button clockwise to release it.
2. Reinitialize the machine by performing a Home operation.
3. Verify all workpiece clamps and tool settings before restarting any job.

2.2 Lockout Procedure

To prevent unauthorized or accidental operation—especially in schools or shared workspaces—the Emergency Stop Button can be locked out with a standard padlock.

How to Lock Out the Machine

1. Press the Emergency Stop Button to engage it.
2. Insert a padlock through the designated hole in the button.
3. Secure the lock and remove the key, storing it in a safe location.

Note: Only authorized personnel should have access to the lockout key. Locking out the machine ensures that it cannot be restarted until the lock is removed, adding an extra layer of safety in environments with multiple users.

SECTION 3: MACHINE SPECIFICATIONS

The DISCOVERY Series is engineered to provide reliable CNC capabilities with a strong focus on educational, prototyping, and light production environments.

Below are the general specifications across DISCOVERY models. Specific details may vary slightly depending on model number and spindle configuration.

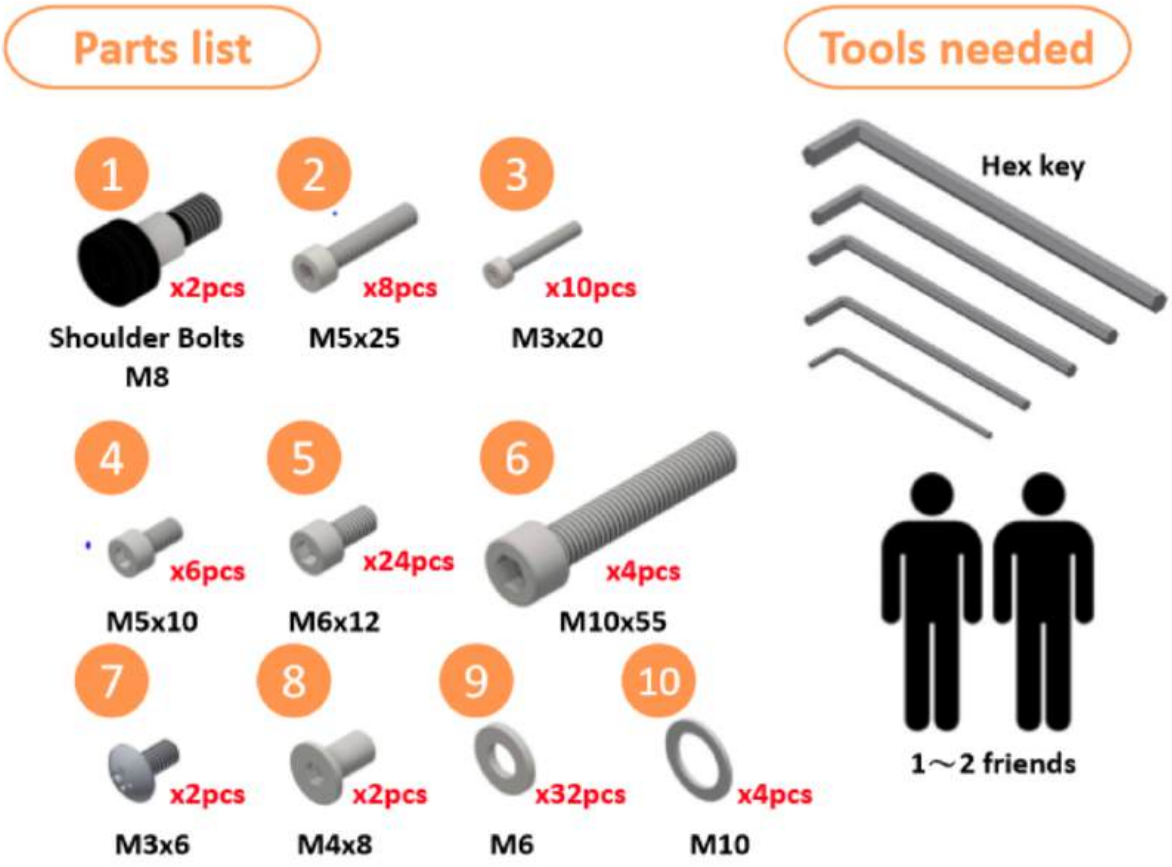
	SIMPLY 4	SIMPLY 6	SIMPLY 8
SKU	SIMPLY4	SIMPLY6	SIMPLY8
X-Axis Travel	23.81″ (605mm)	23.81″ (605mm)	23.81″ (605mm)
Y-Axis Travel	24.01″ (610mm)	35.82″ (910mm)	47.63″ (1210mm)
Z-Axis Travel	3.937″ (100mm)	3.937″ (100mm)	3.937″ (100mm)
Spindle Type	1HP Air Cooled	1HP Air Cooled	1HP Air Cooled
Spindle Configuration	ER11 (1/8” & 1/4”)	ER11 (1/8” & 1/4”)	ER11 (1/8” & 1/4”)
Controller	RichAuto B11	RichAuto B11	RichAuto B11
Rapid Feed Rate	150ipm (3.81m/min)	150ipm (3.81m/min)	150ipm (3.81m/min)
Resolution	± 0.00393" (0.10mm)	± 0.00393" (0.10mm)	± 0.00393" (0.10mm)
Power Requirements	AC 110V, 50-60Hz, 15A, 1-Phase (Optional 220V, 50-60Hz, 10A, 1-Phase)	AC 110V, 50-60Hz, 15A, 1-Phase (Optional 220V, 50-60Hz, 10A, 1-Phase)	AC 110V, 50-60Hz, 15A, 1-Phase (Optional 220V, 50-60Hz, 10A, 1-Phase)
Machine Base	High Rigidity Aluminum Extrusion	High Rigidity Aluminum Extrusion	High Rigidity Aluminum Extrusion
Working Table	High Rigidity Aluminum Extrusion	High Rigidity Aluminum Extrusion	High Rigidity Aluminum Extrusion
Gantry Bridge	High Rigidity Aluminum Extrusion	High Rigidity Aluminum Extrusion	High Rigidity Aluminum Extrusion
Gantry Supports	Steel Plate	Steel Plate	Steel Plate
Machine Dimensions	38″L x 35″W x 28″H	50″L x 35″W x 28″H	62″L x 35″W x 28″H

SECTION 4: GLOSSARY OF TERMS

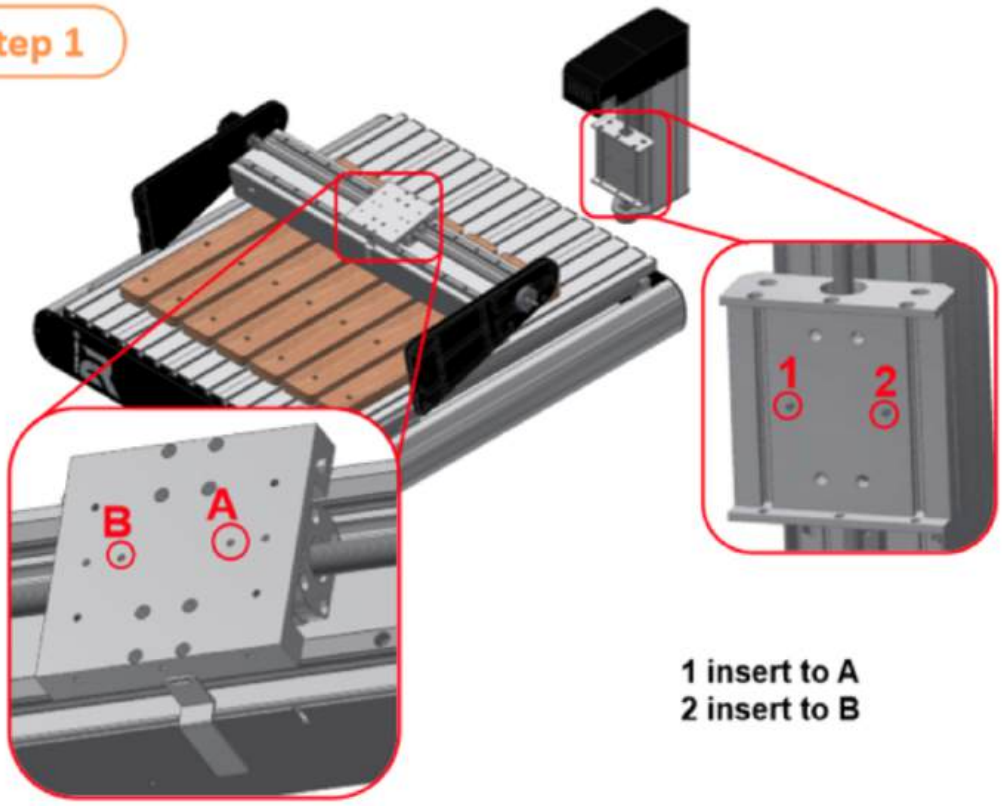
Term	Definition
CNC	Computer Numerical Control – the automation of machine tools via a computer executing pre-programmed sequences of commands.
G-Code	The programming language used to control CNC machines, defining toolpaths, movements, and operations.
Toolpath	The route or path a tool follows to cut or engrave a part.
Spindle	The rotating component that holds and spins the cutting tool.
Workpiece	The material being machined.
Fixture	A device used to securely hold the workpiece in place during machining.
Z-Axis	The vertical axis in a 3D coordinate system, typically representing up and down movement.
X-Axis	The horizontal axis (left to right movement on most machines).
Y-Axis	The depth axis (front to back movement on most machines).
Router Bit	A cutting tool used in CNC routing, typically for woodworking.
Feed Rate	The speed at which the cutting tool moves through the material, typically in inches or mm per minute.
Plunge Rate	The speed at which the tool lowers into the material.
RPM	Revolutions Per Minute – how fast the spindle or cutting tool rotates.
Step-Over	The horizontal distance the tool moves over between passes.
Pass Depth	The maximum depth the tool will cut in a single pass.
Home Position	The machine's reference point, often set at the start of a job (0,0,0).
Origin	The starting coordinate for the toolpath, often set by the operator on the workpiece.
Zeroing	The process of setting the machine's tool to the origin point.
Post Processor	A software component that translates CAM toolpaths into G-code specific to a CNC machine or controller.
CAM	Computer-Aided Manufacturing – software used to create toolpaths from CAD designs.
CAD	Computer-Aided Design – software used to create precise drawings and models for manufacturing.
Stepper Motor	A type of motor commonly used in CNC machines that moves in fixed steps for precise positioning.
Controller	The hardware and software interface that interprets G-code and drives machine movement.

SECTION 5: SETUP & ASSEMBLY

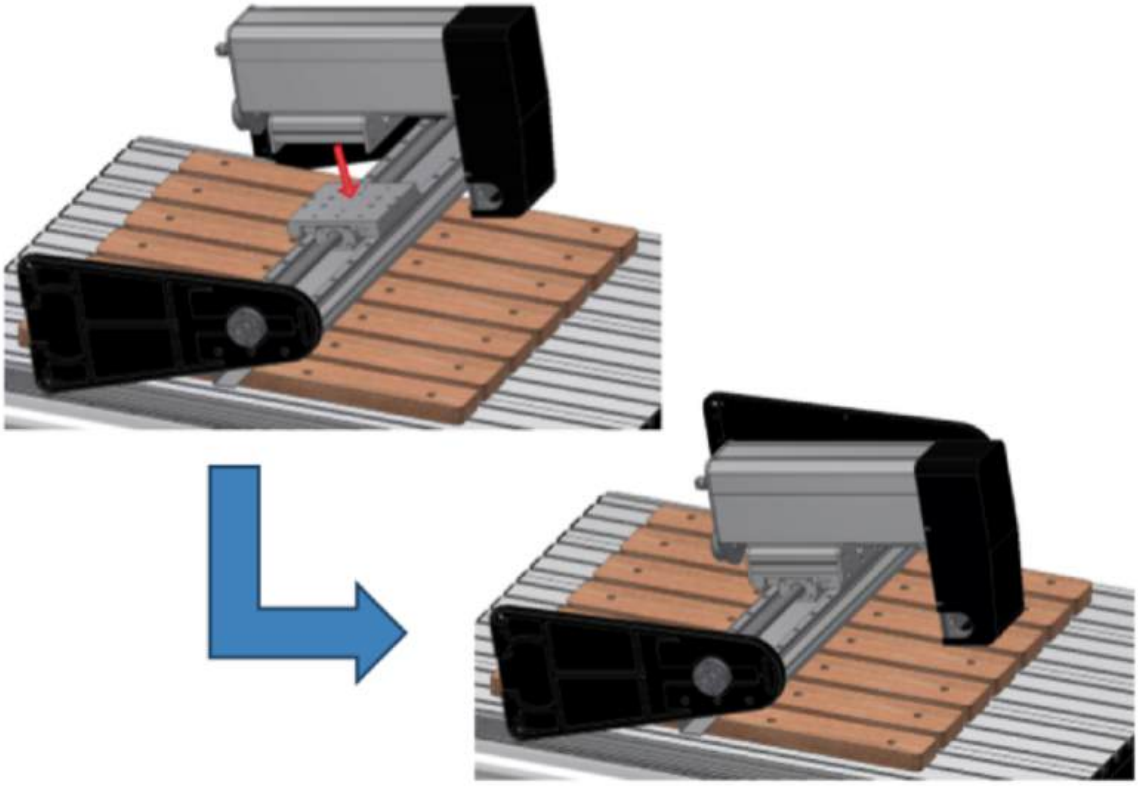
5.1 Instructions For Machine Assembly



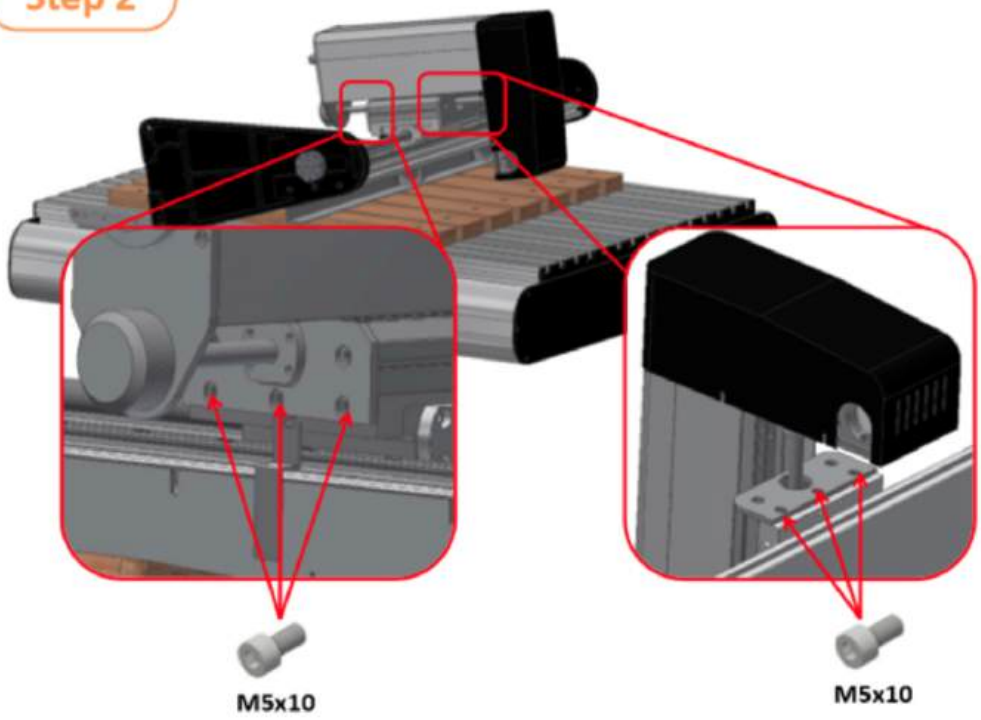
Step 1



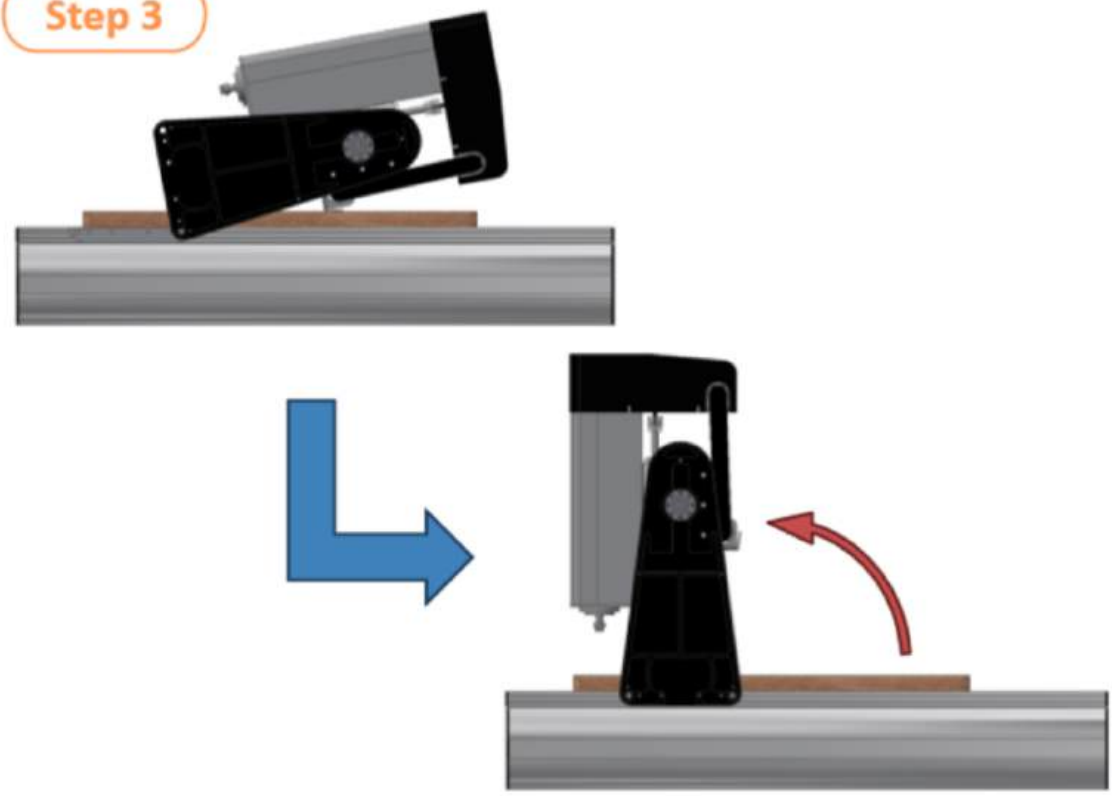
1 insert to A
2 insert to B



Step 2

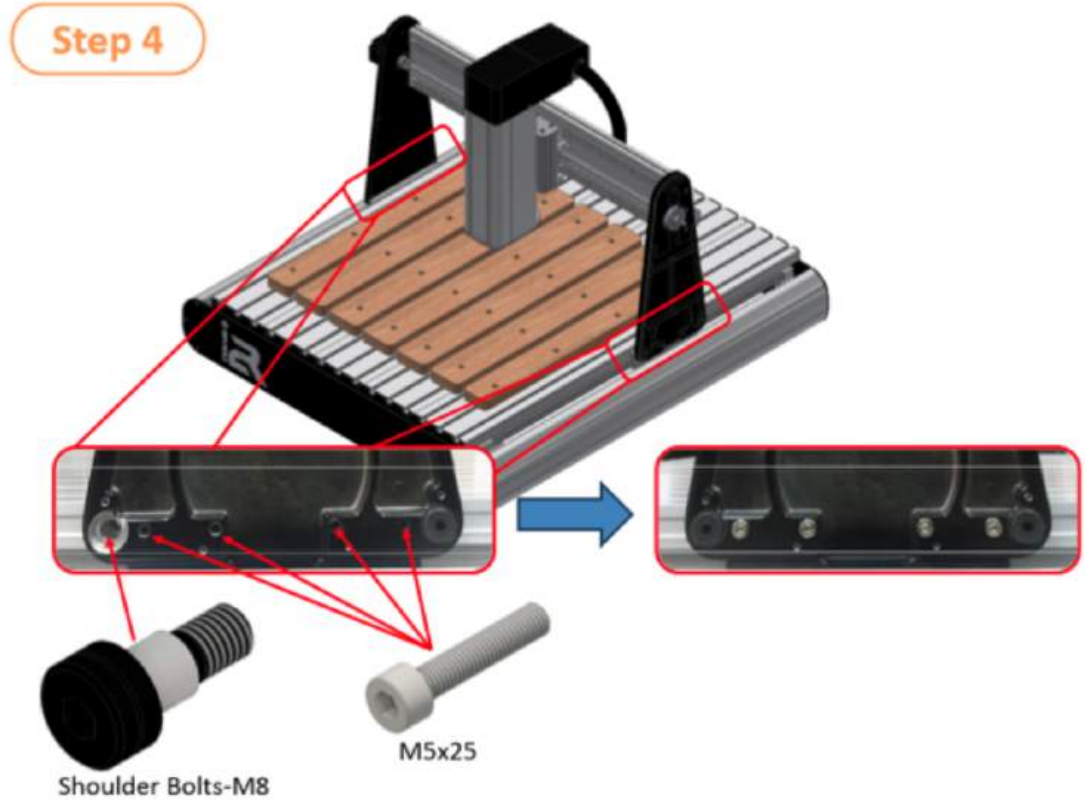


Step 3

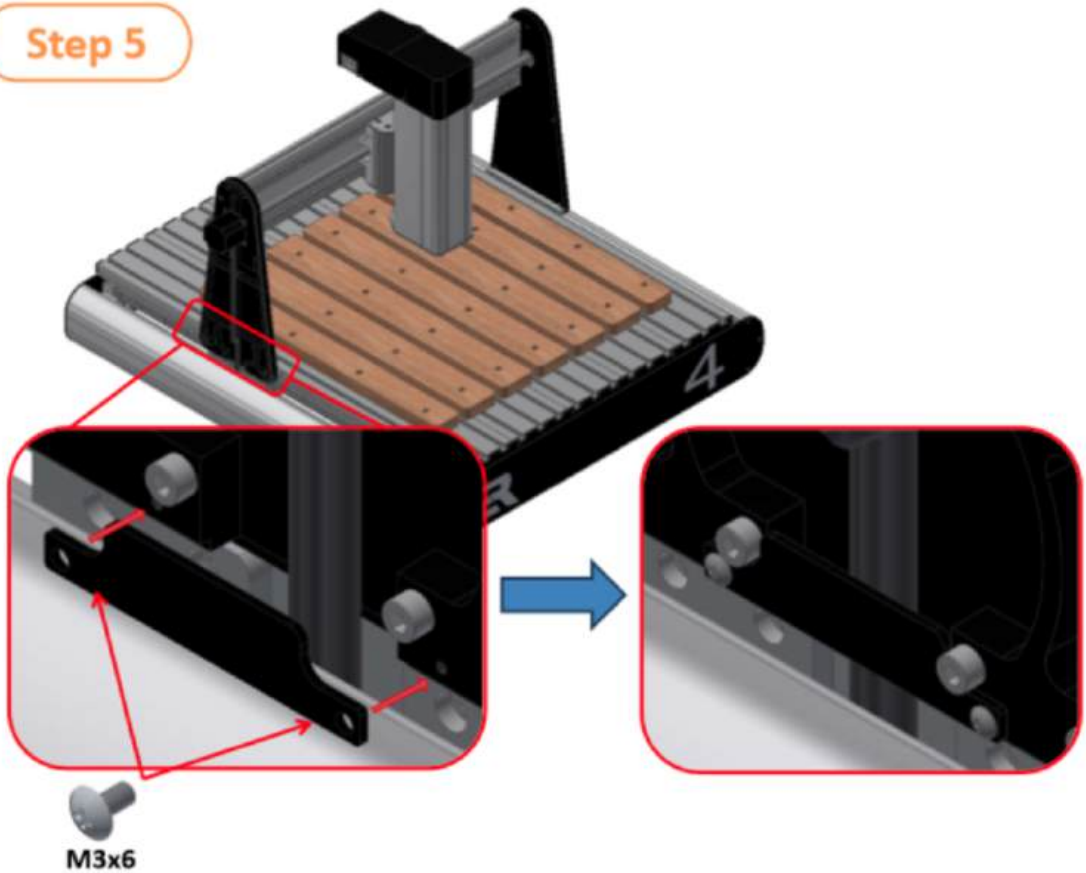


Installing the machine assembly

Step 4

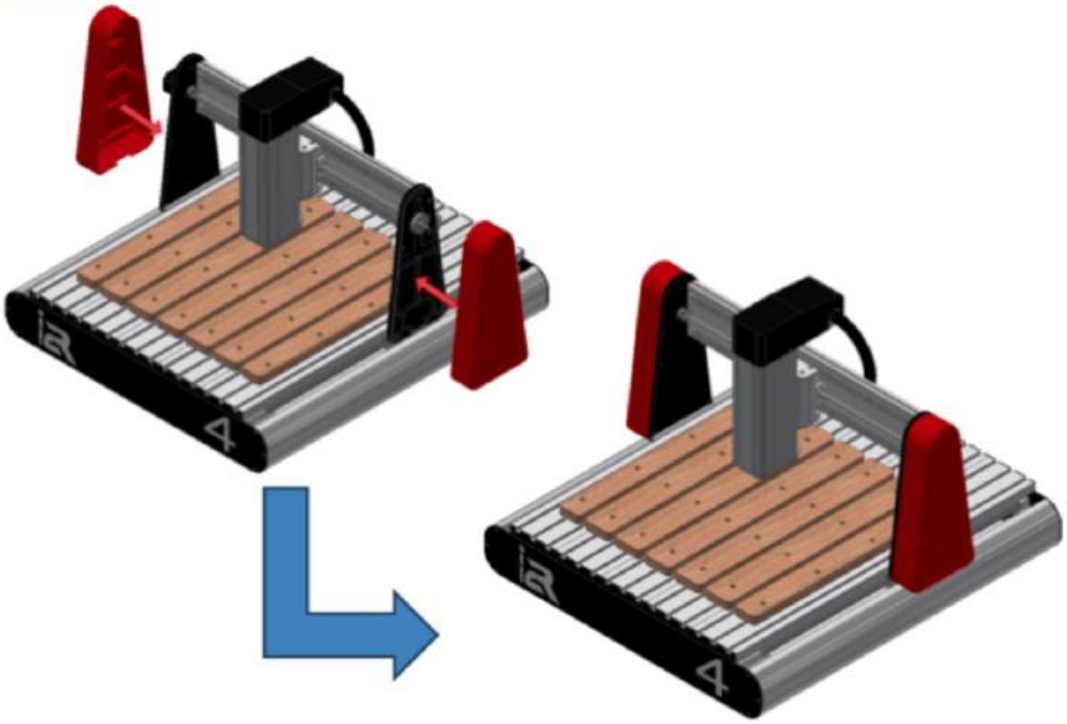


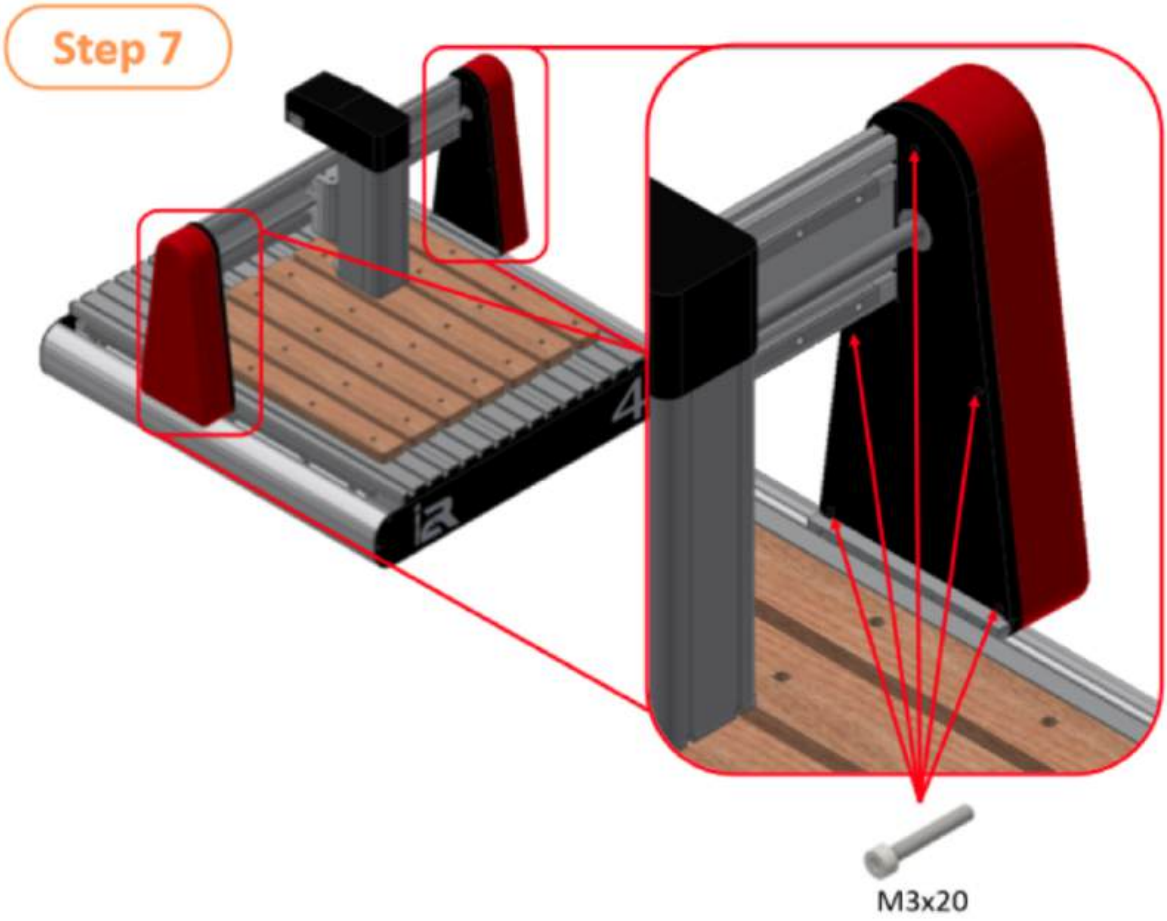
Step 5



M3x6

Step 6




Step 7
M3x20

SECTION 6: ELECTRICAL CONNECTIONS

⚠️ **WARNING:**
All electrical connections must comply with local electrical codes. Installation should be performed by a qualified electrician if you are unsure of your power source or wiring requirements. Improper installation can result in equipment damage, fire, or serious injury.

6.1 Power Requirements

- **Voltage:** 110V AC / 220V
- **Amperage:** 15A / 10A
- **Phase:** Single Phase
- **Plug Type:** NEMA 6-20 (Pre-installed)

Important: This machine requires a dedicated 110V / 220V circuit. Do not use power bars, adapters, or modify the supplied plug.

6.2 Grounding Instructions

Proper grounding is essential for safe operation.

- This machine is equipped with a grounded power cord and NEMA 6-20 plug.
- Ensure the outlet is properly installed and grounded in compliance with national and local codes.
- Never remove the ground pin or use ungrounded adapters.

If you are unsure about grounding or circuit requirements, contact a licensed electrician.

6.3 Extension Cords

⚠️ **Warning:** Extension cords are not recommended.If absolutely necessary:

- Use only a 10-gauge or heavier, grounded, 3-conductor cord.
- Limit length to 10 feet (3 meters) or less.
- Inspect regularly for damage.
- Avoid creating a trip hazard.

6.4 Cable Management

- Route all cables safely to prevent damage or tripping.
- Do not place heavy objects on power cords.
- Ensure cables are not stretched, pinched, or exposed to heat.

SECTION 7: CONTROLLER OVERVIEW

The SIMPLY Series uses the **RichAuto B11 Controller**, a streamlined interface designed for straightforward machine control. This section introduces the key screens, functions, and workflows to help you confidently navigate the controller.

Toolpaths that are created in the design software are communicated to the CNC router through the handheld controller. In the case of the SIMPLY machines, this is done through the RichAuto B11 series 3-axis controller. Processing toolpaths or jobs can be performed either by using a USB Flash drive (USB2.0 of 8GB or smaller, FAT32) or from the internal memory. (512MB).

The RichAuto B11 Keypad uses either **Single touch functions** where buttons may perform one of many single tasks or if held down, will perform a task continuously (such as machine movement), or **Combination functions**, where two keys are pressed at the same time.

7.1 Saving a File to Internal Memory

After completing the **Homing procedure**, follow these steps to copy a toolpath file from your USB (U-disk) to the controller's internal memory:

Steps:

1. From the main coordinate screen, press the MENU button.

2. Use the **X+ / X–** buttons to scroll and highlight **OPERATE FILE**, then press **ORIGIN/OK**.

3. The screen will display options such as **Copy File** and **Delete File**.

4. Highlight **COPY FILE**, then press **ORIGIN/OK**.

5. When prompted to select the file source, choose **UDISK** (your USB drive).

6. A list of available files on the USB drive will be displayed.
   Use the arrow keys to highlight the desired file and press **ORIGIN/OK**.
7. A brief loading screen will appear while the file is copied.

8. Once complete, the file is successfully saved to the controller's **internal memory**, ready for selection and operation.

Tip: Always ensure your file is correctly formatted for your CNC controller (e.g., .mmg, .nc, or .tap, depending on setup).

7.2 Operating Procedure

CNC machine operations can be initiated in multiple ways:

- Via the controller keypad (manual commands)
- **Using design files (toolpaths)** transferred to the controller via USB (U-disk) or accessed from internal memory

Before running any job, complete the following steps to ensure safe and accurate operation:

Pre-Operation Checklist:

Stabilize the Machine

- If your machine is mounted on a stand with leveling feet, ensure the feet are fully **lowered and locked** to prevent movement during operation.

Secure the Workpiece

- Confirm the material is **firmly fixed to the machine bed** using one or more of the following:
    - Clamps
    - Fixtures
    - Vacuum hold-down system (optional accessory)
    - Double-sided tape (for lightweight or thin materials)

Power On the Machine

- Press the **stainless steel power button** (green backlit) to power on the system.
- The controller screen will illuminate and display the **loading screen**.
- If the **emergency stop button** is engaged, release it by turning it **clockwise**.

Perform a HOME Operation

- Once the controller has finished loading, a **HOME operation** must be completed **before any other function can be performed**.
- Refer to the “Homing the Machine” section for detailed instructions.

Following these steps before every operation ensures safe startup, precise tool alignment, and proper machine function.

7.3 HOME Position

Each time the machine is powered on, the controller will go through a brief loading sequence. Once complete, the screen will display the **HOME menu**.

1. Selecting a HOME Operation
   - Use the **X+ / X–** buttons to scroll through the available options.
   - Highlight the top option: **All Axis Home**.
   - Press the **OK / ORIGIN** button to begin the homing procedure.

How Homing Works

The HOME position is established using **magnetic proximity sensors**, one on each axis. These sensors are calibrated during manufacturing and typically do not require adjustment.

During the All Axis Home operation:

1. The **Z-axis** raises to the top of its travel first.

2. The **X and Y axes** then move simultaneously to the **front-left corner** of the machine.

3. Once complete, the controller displays the **Coordinate Screen**, showing:
   - The current position relative to the active **Origin** (0,0 point)
   - The currently selected **coordinate state.**

Why Homing Is Important

Running a HOME operation on startup is **essential**. It ensures:

- The machine's internal positioning system is properly aligned
- The limit switches and software-driven movement logic are functioning correctly
- All subsequent movements are referenced accurately from the HOME position

Note: All CNC motion is software-based and requires an accurate HOME reference to operate safely.

Re-Homing During Operation

You can perform a HOME operation at any time by pressing the **HOME** button from the **main coordinate screen**.

The following options will be available:

1. **Home All** – Homes all axes (X, Y, and Z)
2. **Home X** – Homes only the X-axis
3. **Home Y** – Homes only the Y-axis
4. **Home Z** – Homes only the Z-axis

Use the **X+/X–** buttons to scroll, and press **OK** to confirm your selection.

7.4 Moving Spindle Head

The movement of the router or spindle head can be done manually in several modes, which are toggled using the **MODE** button. The current movement mode is temporarily displayed at the bottom of the controller screen when selected.
The primary movement modes include:

- Continuous Mode (default)
- Step Mode
- Distance Mode
- GoTo Position Mode

Each mode offers unique advantages depending on the operation required.

Movement Speed Settings (LSP / FSP)

Movement speeds are controlled by the **Fast/Slow** button and are shown on the controller's coordinate screen (far right, third row down):

- LSP (Low Speed Mode)
  - Grid size: 0.1 mm (0.004") per step
  - Continuous speed: 1500 mm/min (default)
- FSP (Fast Speed Mode)
  - Grid size: 0.5 mm (0.020") per step
  - Continuous speed: 2500 mm/min (default)

Continuous Mode

In Continuous Mode, holding down any directional button (X+/–, Y+/–, Z+/–) will move the spindle until the button is released.

- The controller will display the live position of the router head in relation to the current Origin.
- During the first 0.5 seconds of holding the button, the movement behaves like Step Mode and moves slowly to the next grid point before accelerating.

Step Mode

Step Mode is used for precise positioning. Tapping a directional button will move the spindle **one grid unit per press**.

- **LSP**: 0.1 mm per tap
- **FSP**: 0.5 mm per tap
- Each movement occurs over approximately 0.5 seconds for smoother control

This mode is ideal for fine-tuning bit placement or setting accurate Origins.

Distance Mode

Distance Mode allows you to program an exact movement distance, then move in that distance with a single tap.
To use Distance Mode:

1. Press **MODE** until the screen displays `Dist`.

2. The distance field will appear highlighted. Enter the desired distance using the number keys.

3. Press **OK** to confirm. The distance field will remain, but the highlight will disappear.

4. Tap any directional button (X+/–, Y+/–, Z+/–) and the spindle will move the programmed distance in that direction.

This is useful for consistent repositioning or repeating precise movements.

GoTo Position

GoTo Position allows the router to move to a specific coordinate in reference to the current Origin. To use GoTo Position:

1. Press **MODE + MENU** simultaneously to open the GoTo Position screen.

2. Use the **X+/–** buttons to scroll through the coordinate fields (X, Y, Z).

3. To edit a coordinate:
   - Highlight the axis
   - Press **Delete** to clear the existing value
   - Enter the new coordinate value
   - Press **OK**

4. After all desired axes are entered, press **OK** once more to confirm and execute the move.

Caution: All three axes will move simultaneously. Be sure to check for obstructions, hold-downs, or fixtures that could interfere with safe movement.

7.5 Setting Work Origin

Setting the **Work Origin** establishes the reference point (also called the zero point or datum) from which the machine will execute the toolpath. This point must match the zero location defined in your design software for accurate and safe operation.

Important: You must set the Origin for **X, Y, and Z** before starting any job. Failure to do so may result in damage to the material, tooling, or machine.

Step-by-Step Instructions

1. Insert and Secure the Tooling

- Ensure the **collet** is correctly inserted into the collet nut before threading it onto the spindle.
- Insert the selected tool into the collet and tighten it using the provided wrenches.
  - Use only ER-20 collets.
  - Match the collet size to the shank diameter of your tool.

2. Set the X and Y Origin

- Use the directional buttons to move the spindle to your desired X and Y location.
  - Adjust movement speed using **FSP** (Fast Speed) and **LSP** (Low Speed).
  - Use **Step Mode** or **Grid Mode** for more precise positioning.
- The Origin is typically set at a corner of the material or at its center, depending on your toolpath setup.
- Once the spindle is positioned, press the **XY-0** button.
  - The X and Y coordinates on the controller will reset to **0.00**.

3. Set the Z (Height) Origin

There are two methods for setting the Z-axis Origin:

A. Using the Tool Touch-Off Device (Recommended)

1. Plug the device into the female port at the top of the Z-axis.
2. Place the touch-off puck on top of the material, centered under the cutting tool.
   - Be sure the material surface you are referencing matches the **Z-zero location selected in your design software**.
   - If the material is not flat or consistent, you may reference the **spoilboard** instead.
3. With the puck in place, press **Tool Set** on the controller.
   - The spindle will lower until the bit contacts the puck, then rise to a safe height.
   - A brief delay or a two-step contact process may occur; this is normal.
4. The Z-axis Origin is now programmed.

Note: Always confirm that the Z-zero setting in your software matches your actual reference surface (material top or spoilboard).

B. Manual Z-Origin Setting (Use for fragile tools or high precision)

1. Switch the controller to **Step Mode**.
2. Slowly lower the tool while gently rotating it by hand.
3. Stop when the tool just makes contact with the surface (material or spoilboard).
   - For more precision, place a sheet of paper between the tool and material.
   - Lower until slight resistance is felt when sliding the paper.
4. Once positioned, press **Z-0**, then select **Clear Z** to set the Z-axis Origin.

Tip: It is generally safer to reference the **top of the material** to avoid cutting into the worktable.

After Setting the Origin

- The controller will now display **X: 0.00, Y: 0.00**, and the Z-axis will rise to its safe height.
- The Z-axis will only read **0.00** when the bit is physically touching the surface used to set the Z-zero.

Reusing the Last Origin

- If the machine is powered off or restarted, you must perform a **HOME** operation to re-establish machine coordinates.
- Your last saved Origin (relative to HOME) is still stored in memory.
- Press the **Origin/OK** button to return the spindle to the last programmed Origin location.

7.6 Programming Additional Origins (Coordinate States)

The RichAuto controller supports multiple **coordinate states**, each with its own programmed **Origin**. By default, the controller displays something like this on the coordinate screen:

#1 X:000.000 Y:000.000 Z:000.000

The number before each axis (e.g., **#1**) indicates the current coordinate state. The controller supports up to **nine** different coordinate states (1–9), each with its own saved Origin.

To switch between coordinate states:

- Press and hold **MENU**, then press a number key (1–9), and release both buttons.
- The coordinate screen will update to reflect the selected state.

Tip: It is often easiest to hold down **MENU**, tap the desired number key, then release both.

When to Use Multiple Origins

This feature is useful when:

- Running multiple jobs on different parts of the table.
- Using jigs or fixtures for repeatable setups.
- Dividing the machine bed into zones for small batch production.

Example: Dividing a 48” Table

If your machine has a 48” work area and you are running two jobs under 24” in width, you can divide the surface into two zones:

1. **Coordinate State #1** (`MENU + 1`):
   Set the Origin for the **left** 24” zone and position your first material.

2. **Coordinate State #2** (`MENU + 2`):
   Set the Origin for the **right** 24” zone and position your second material.

Both Origins reference the same **HOME** point, so you can quickly switch between them using the **MENU + number** shortcut. This allows you to machine multiple parts in sequence without re-zeroing the machine.

7.7 Spindle RPM Control (VFD Inverter)

SIMPLY model machines are equipped with a **Variable Frequency Drive (VFD)** for spindle RPM control. The VFD is located on the **front of the white control box**.

To adjust spindle RPM:

- Turn the **RPM dial**:
  - Clockwise (right) to increase RPM
  - Counterclockwise (left) to decrease RPM

- This can be adjusted **at any time**, even while a job is running, allowing for dynamic RPM control during machining.

RPM Display Format:

- The display shows RPM in **frequency (Hz)**.

- 0–400 Hz = 0–24,000 RPM
- Each 100 Hz = 6,000 RPM

- When the spindle is active, the display will alternate between frequency and real RPM.

Note: The **buttons on the VFD** are used for programming only. Do **not** use them to set spindle speed.

Processing a File

Once the **Origin is set (X, Y, and Z)** and the **RPM is adjusted**, you can begin running a toolpath file.

Important: Always properly eject your USB flash drive ("U-disk") from the computer before inserting it into the controller. This ensures the controller can read the files without issue.

To Start a Job:

1. From the main coordinate screen, press the Run/Pause/Delete button.

• Choose the file location:

- U-Disk, Internal Memory, or Recently Used
- Use **X+ / X–** to scroll through the list
- Highlight your selection and press **OK**

3. Select the file:
   - The file list displays **three items at a time**
     1. **X+ / X–** scrolls one file at a time
     2. **Y+ / Y–** scrolls one page at a time
   - Highlight the desired **toolpath file (.mmg)** and press **OK**

4. Review the working parameters:
   - **Work Speed:** Equivalent to feed rate (can be ignored if feed rate is controlled from the design software — which is the default).
   - **Fast Speed:** The travel speed between cut points (also called shuttle speed).
   - **Speed Scale:** A percentage modifier for the programmed feed rate. (See “Speed Scale Adjustment” if changes are needed.)

5. Press **OK** to begin the toolpath.
   - A short countdown (approx. 4 seconds) will begin.
   - The spindle will ramp up to speed, and then the file will start.

Speed Scale Adjustment

The **Speed Scale** setting allows you to adjust the feed rate (as a percentage of the value programmed in your design software). This can be useful for fine-tuning cutting performance based on material behaviour, tooling, or machine response.

You can adjust the Speed Scale in **two ways**:

- **Before** the toolpath begins
- **During** the operation

1. Adjusting Speed Scale *Before* the Operation

When the toolpath file is selected and the **file parameter screen** is displayed:

1. Use the arrow keys to highlight **Speed Scale**.
2. Press the **Delete** button to clear the existing value.
3. Enter the new Speed Scale value using the number keys.
4. Press **OK** to confirm.

Valid range: 0.1 (10%) to 1.0 (100%), in 0.1 increments

2. Adjusting Speed Scale *During* Operation

While the file is running:

- Press **Y+** to increase the Speed Scale
- Press **Y–** to decrease the Speed Scale
- Each press changes the speed by **0.1 (10%)**

Note: The Speed Scale cannot exceed **1.0 (100%)** of the feed rate programmed in the toolpath software.

This feature is especially useful for slowing down during detailed cuts or increasing speed during long clearing passes — without stopping the job or modifying the file.

7.8 Pausing During Toolpath Operation

A toolpath file (also referred to as a "job") can be **paused mid-operation** to make minor adjustments. This function is intended for **brief pauses only**, such as fine-tuning bit height or position.

Important: Pausing is not intended for long interruptions. Always monitor the spindle and movement carefully when using this function.

To Pause a Job:

1. Press the **Run/Pause** button during active operation.
   - The machine movement will stop immediately.
   - The spindle will remain active and continue spinning.

2. While paused, you may adjust the **bit position** in any direction.
   - It is strongly recommended to use **Step Mode** and **Low Speed (LSP)** for greater control and safety.

3. After adjustments are complete, press **Run/Pause** again to resume the job.

Restore Position Prompt:

After resuming, the controller will display:

“Restore Position?”

- Press **OK** to resume the operation **from the new bit position** (modified settings).
- Press Cancel to resume the operation from the original saved position (before the pause).

Note: Any changes made during a pause will only affect the current toolpath file and **will not be saved** for future jobs.

7.9 Setting a Breakpoint (Save & Resume Job Later)

The **Breakpoint** function allows you to stop a job mid-operation and **save the exact location** so that it can be resumed later—even days afterward and across multiple machine restarts. This is different from a simple pause, which is intended for brief adjustments during a job.

To Set a Breakpoint:

1. While the machine is running a toolpath, press the **Stop/Cancel** button.
   - A prompt will appear: **Save Break?**

2. Press **OK** to confirm the Save Break operation.
   - Pressing **Stop/Cancel** a second time will cancel the save and discard the current position.

3. The screen will display Breakpoint storage slots (1–8):
   - Use **X+ / X–** to scroll through the list.

- Existing breakpoints can be **overwritten**.
- Press **OK** to save the Breakpoint to the selected slot.

To Resume from a Breakpoint:

If the machine has been powered off since the Breakpoint was saved, perform a **HOME operation** before proceeding.

1. Press **Run/Pause + [1–8]** simultaneously to select the Breakpoint slot where the job was saved.

2. The controller will display the **list of saved Breakpoints**.

3. Use **X+ / X–** to highlight the desired file and press **OK**.

4. The controller will now display the **exact g-code line** where the break was recorded. You can scroll through the g-code and resume from any point if needed.

5. Once the desired resume point is selected, press **OK**.

6. The **file parameters screen** (Speed, Shuttle Speed, Speed Scale) will appear as usual. Review and confirm these settings.

7. Press **OK** once more to resume the job from the saved Breakpoint.

Tip: Use Breakpoints for long jobs, unexpected interruptions, or when tool or material changes are required over multiple sessions.

7.10 Power Failures and Job Recovery

In the event of an unexpected **power failure**, the system will automatically create a **temporary Breakpoint**, saving the current job parameters and machine coordinates. This allows you to **resume the operation once power is restored**, but only if resumed during the next startup.
Important: This temporary save is **not permanent**. If it is not resumed during the next power-up cycle, the saved data will be lost.

To Resume After a Power Failure:

1. When power is restored, **turn on the machine** and allow the controller to load.

2. Perform a full **HOME operation** to re-establish the machine's reference point.

3. After homing, the controller will display a prompt:

- “Want to Restore?”

4. Press **OK** to resume the unfinished job from the last saved location.
   - If **Cancel** is selected instead, the system will discard the saved state and the job **cannot be resumed**.

Tip: For critical or long-running jobs, consider manually setting a Breakpoint periodically in addition to relying on automatic recovery.

7.11 RichAuto B11 Functions

Key	Function	Key	Function
X+ ▲ 1	X-Axis positive movement (Right). Scroll up. (One line) Number 1 input.	Y+ ^F+% 2	Y-Axis positive movement (Backwards). Increase feed rate (10% increments). Scroll up. (Whole page). Number 2 input
X− ▼ 5	X-Axis negative movement (Left). Scroll down. (One line) Number 5 input.	Y− VF-% 6	Y-axis negative movement (Forwards). Decrease feed rate (10% increments). Scroll Down. (Whole page). Number 6 input
Z+ S+% 3	Z-Axis positive movement (Upward). Number 3 input.	XY→0	Set **Origin** for X & Y axes.
Z− S-% 7	Z-axis negative movement (Downward). Number 7 input.	Z→0	Set **Origin** for Z-axis.
HOME 9	Opens **HOME** menu. (Front Left Corner). Number 9 Input.	FAST/SLOW 0	Manual movement (FSP or LSP). Number 0 input.
SPINDLE .	Spindle On/Off. Decimal input.	MENU –	Opens Menu. Negative symbol input.
ORIGIN OK	X & Y axes move to **Origin** and Z-axis raise to minimum safe height. Confirm commands.	MODE	Movement selection (continuous, step, or distance).
RUN/PAUSE DELETE	Run Tool-path Menu/Pause operation. Delete inputs.	STOP CANCEL	Stop Tool-path operation. Cancel commands. Move back one Menu screen.
BREAK WORK	Opens Break-Work Menu.	TOOL SET	Set **Origin** for Z-Axis using Tool Touch-Off device.
ADVANCED FUNCTION	Opens Advanced Processing Menu.	REPEAT MACHINING	Repeat Last Operation.

Section 8: Spoilboard Surfacing

Why Surfacing Matters

Your machine comes with a pre-installed **MDF spoilboard** mounted to the aluminum T-slot table.

While the MDF board is machined flat at the factory, slight variations can occur during shipping, installation, or environmental changes (humidity, temperature).

Surfacing your spoilboard ensures:

- A **flat reference plane** for consistent cutting depths.
- Improved accuracy when cutting through materials.
- Extended spoilboard lifespan by resetting the surface.

Recommended Surfacing Tool

Use a **large diameter surfacing bit (e.g., 1.5" or 2")** for best results.
Ensure the tool is sharp and properly secured in the collet.

Creating a Surfacing Toolpath

You can create a surfacing toolpath using software like **VCarve or Aspire**.

Toolpath Parameters Example:

- **Tool**: 1.5" Spoilboard Surfacing Bit
- **Cut Depth**: 0.5mm (0.02") per pass
  (Use multiple passes if needed)
- **Stepover**: 80% of tool diameter
- **Feed Rate**: 150 IPM
- **Spindle Speed**: 18,000 RPM (or as recommended by bit manufacturer)

Note: Ensure the toolpath is slightly **larger than the spoilboard area** to cover the full surface.

Running the Surfacing Toolpath

1. Secure the spoilboard to the machine table if it has been removed.
2. Load the toolpath file onto your USB drive.
3. Set XY zero to the front-left corner of the spoilboard.
4. Set Z zero to the surface of the spoilboard using the touch-off puck.
5. Run the toolpath, monitoring the process at all times.

Spoilboard Maintenance Tip

After surfacing, check your **hold-down methods (clamps, screws, etc.)** and re-adjust them if necessary.

Repeat surfacing periodically to maintain a flat working surface as needed.

Section 9: Maintenance

9.1 Maintenance Schedule

Proper maintenance ensures optimal performance and extends the life of your CNC machine.
Use the following **checklists** to maintain your machine.

Daily Maintenance (Before or After Use)

- Visually inspect **cables, hoses, and connections** for damage.
- Remove dust and debris from **rails, ball screws, and spindle area**.
- Ensure **spindle collet and nut** are clean and free of debris.
- Check for **loose bolts or fasteners** on the machine frame and gantry.

Weekly Maintenance

- Apply **light machine oil** to linear guide rails and ball screws.
- Inspect **spindle cooling system** (check coolant level if applicable).

- Confirm **gantry moves smoothly** without hesitation or grinding.
- Check **tool touch-off puck** for functionality.

Monthly Maintenance

- Perform a **full machine cleaning**, removing all dust from electrical enclosures.
- Inspect **belt tension** (if applicable) and adjust if necessary.
- Verify **controller buttons and display** are functioning properly.
- Test **emergency stop and safety features** for reliability.
- Inspect **spindle cooling system hoses and fittings** for leaks.

Quarterly Maintenance

- Check **spindle bearings** for noise or rough operation.
- Verify **controller firmware** is up to date (contact support if unsure).
- Review **machine level** and re-level if needed.

9.2 Bearing and Rail Lubrication

Use **light machine oil** or **rail-specific lubricant** on:

- **Linear Guide Rails**
- **Ball Screws**

Do not over-lubricate, as excess oil attracts dust. Wipe off any buildup after application.

Section 10: Fuse Locations and Electrical Safeguards

10.1 Fuse Protection Overview

Your Performance Series CNC machine is equipped with **fuse protection** to safeguard the control system and spindle from electrical faults.

If the machine becomes **non-responsive** or **fails to power on**, inspect the fuses before seeking service.

10.2 Fuse Locations

Main Controller Box Fuses:

- Located inside the **main control box** (lower access panel).
- Protects:
    - **Handheld Controller**
    - **Power Supply Circuitry**
    - **Spindle VFD (Variable Frequency Drive)**

Spindle VFD Fuses:

- Located inside the **VFD enclosure**.
- Protects:
    - **Spindle motor and VFD circuits**

10.3 Replacing Fuses

⚠️ **Warning:**
Always **disconnect power** before opening the control box or VFD enclosure.

1. Turn off the machine and **unplug from power**.
2. Open the **control box panel** using appropriate tools.
3. Locate the **fuse holders**.
4. Carefully **remove and inspect** each fuse.
5. Replace **only with the same type and rating** as specified on the fuse label or in the electrical diagram.
6. Reassemble the panel and restore power.

Note:
Repeated fuse failures indicate an **underlying electrical issue**. Contact Simply Technologies Support before proceeding with further operation.

10.4 Electrical Safeguards Summary

- Always use a **dedicated 110V, 15A / 220V, 10A, single-phase circuit** with proper grounding.
- Do **not** bypass fuses or safety devices.
- Avoid exposing the controller to **moisture, excessive dust, or heat**.
- Regularly inspect **cables, plugs, and connectors** for wear or damage.
- Keep the **control box closed** during operation to prevent contamination.

Section 11: Machine Troubleshooting Guide

Even with proper use and maintenance, issues can occasionally arise. Use this guide to identify and resolve common problems.

11.1 Mechanical Issues

Problem	Possible Cause	Solution
Gantry moves unevenly or jerks	Dirty or dry rails/ball screws	Clean and lubricate rails and ball screws
Machine loses position or skips steps	Loose couplers or motor connections	Inspect and tighten all mechanical couplings
Inconsistent cut depth across material	Uneven spoilboard or improperly set Z origin	Resurface spoilboard and reset Z zero
Unusual noises during movement	Dry bearings or misaligned rails	Lubricate and inspect alignment

11.2 Electrical / Controller Issues

Problem	Possible Cause	Solution
Controller does not power on	Blown fuse, disconnected power	Check and replace fuse, verify power connection
No display or frozen screen	Controller cable loose or damaged	Reseat or replace controller cable
Machine won't Home or loses connection	Controller communication fault	Power cycle the machine, check cables

Spindle stops mid-job without error	Power interruption	Check eCNC for possible error messages, power cycle the machine, ensure G-Code is correct
USB files not recognized	Unsupported file format or corrupt USB	Ensure file is proper G-Code, try a different USB
Breakpoint not resuming correctly	Incorrect origin reset after power loss	Re-home machine and verify origin before resuming
Emergency Stop won't reset	Button not released or damaged	Twist to release, inspect for mechanical failure

Tip:
If you encounter a problem not listed here, visit **www.simplytechnologies.xyz/support** or contact **steve@simplytechnologies.xyz** for further assistance.

Section 12: Warranty Information

Limited Warranty Coverage

Simply Technologies warrants your **SIMPLY Series CNC Machine** to be **free from defects in materials and workmanship** for a period of **12 months** from the date of delivery.

What's Covered

- Frame and Structural Components
- RichAuto B11Controller (hardware only)
- Motors and Drives
- Spindle Motor
- Electrical Components
- Included Accessories (Touch-Off Puck, Wrenches, etc.)

What's Not Covered

- Consumable Parts (Spoilboard, Bits, Collets)
- Cosmetic Damage or Normal Wear and Tear

- Damage Caused by:
  - Improper installation
  - Unauthorized modifications
  - Power surges or improper electrical setup
  - Abuse, misuse, or negligence
  - Operation outside specified guidelines
- Software or File Errors
- Third-party accessories not sold or approved by Simply Technologies

Warranty Claims Process

1. Contact **support@simplytechnologies.xyz** with a detailed description of the issue, including:
   - Machine serial number
   - Purchase date
   - Photos or videos of the issue (if applicable)
2. Our support team will assess the claim and provide instructions.
3. Replacement parts or repair services will be provided at Simply Technologies' discretion.

Note: Customer is responsible for shipping costs on non-warranty repairs.

Section 13: Contact & Support Information

Simply Technologies Support Team

📧 **steve@simplytechnologies.xyz**

🌐 **www.simplytechnologies.xyz/support**

Thank you for trusting **Simply Technologies** with your CNC production needs.
We are committed to helping you succeed for years to come.