AXIOM
PRECISION

i2R MANUAL

Rev. 3/2019

Axiom Tool Group, Inc. has provided this manual covering the safe operation and maintenance procedures for the i2R™ series machines. Contained in the manual are installation instructions, safety precautions, operational procedures and maintenance guidelines.

The instructions set forth in this document, will ensure that the i2R™ machine will provide consistent, long-term operation.

This manual was not intended to cover every facet of machine operation. The use of jigs, fixtures, after market CNC accessories, choice of materials and tooling may require additional research. Online forums, and CNC blogs are a good source of knowledge from experienced users.

Regardless of the methods in place, personal safety is always a priority.

If you have any questions or comments, please email the Axiom Precision Customer Service team at support@axiomtoolgroup.com.

Safety Information

The i2R™ is an electrical appliance and precision machine. Protect yourself and your investment. Read and understand the entire owner's manual before attempting assembly or operation. Read and understand the warnings posted on the machine and in this manual. Failure to comply with all the warnings may cause serious personal injury or costly damage to the machine.

This i2R™ CNC machine is designed and intended for use by properly trained and experienced personnel only. If you are not familiar with the proper and safe operation of a CNC machine, do not use the i2R™ until proper training and knowledge have been obtained.

Your i2R™ machine is intended for cutting wood, acrylics, wood-fiber composites, certain plastics and non-ferrous metals. Do not use this machine for other than its intended use. If used for other purposes, Axiom disclaims any real or implied warranty and holds itself harmless from any injury that may result from that use.

Shop Environment
1. Ensure that the floor can bear the weight of the machine and work pieces mounted on it.
2. Keep the floor around the machine clean and free of scrap material, oil and grease.
3. Do not lean lumber or other heavy materials against the gantry, guide rails or table.
4. Support the weight of the dust hose attached to the dust shoe accessory to prevent the weight of the hose from dislodging the dust shoe. Ensure that there is sufficient slack in the dust collection hose to allow the spindle to cover the entire work area.
5. Locate the machine away from overhead pipes and plumbing fixtures to prevent condensation from dripping on to the spoil boards and control system components.
6. Locate the machine away from sinks, faucets or other water supplies or storage to prevent splash-out that can damage the spoil boards and control system components.
7. Provide adequate room between the i2R™ and other machines in the shop to reduce the chance of accidental jarring when transporting lumber or other heavy materials through the shop and while materials are being worked on other machines.
8. Ensure adequate space between machines to allow for the possibility that the work piece will extend over the end of the i2R™ table.
9. Store cutting tools in a dry location and prevent contact to preserve the cutting edges.

Health and Safety

10. Always wear approved personal health and safety equipment as indicated for the materials and type of operations that will be performed. These should include a dust mask, hearing protection, safety clothing, and safety glasses/face shield. Do not rely on prescription or over-the-counter eyeglasses; they are *not* safety glasses. Wear ear protectors (plugs or muffs) even during short periods of operation.

11. Before operating this machine; remove any hand, wrist, and neck jewelry and roll sleeves up past the elbows. Be sure to not wear loose clothing which may become caught in the machine and confine long hair.
    a. Non-slip footwear or anti-skid floor strips are recommended.

12. Use a dust mask or other safeguards to avoid inhaling dust generated from wood products. Install dust collection equipment consistent with shop ventilation practices and budget. Remove dust and debris from the floor frequently to prevent slipping. Drilling, sawing, sanding or machining wood products generates wood dust and other substances known to the State of California to cause cancer. Wood products also emit chemicals known to the State of California to cause birth defects or other reproductive harm. (California Health and Safety Code Section 56).

13. In addition to other health hazards, dust from wood and other materials is flammable. Do not operate welding, wood burning, smelting, soldering or other high-heat tools on the i2R™ table or vicinity.

14. Do not operate this machine while tired or under the influence of drugs, alcohol or any medication.

Electrical

15. Make certain the switch is in the OFF position before connecting the machine to the power supply.

16. Make certain the machine is properly grounded and the circuit is protected with a fuse or circuit breaker in accordance with local codes. Install a separate circuit if necessary, to limit power loss when multiple machines in your shop are operating simultaneously. If necessary, place a cover on the outlet to prevent accidental disconnection.

17. Make all machine adjustments or maintenance with the machine unplugged from the power source.

18. Don't use in a damp or wet location.

19. Keep the electrical cord away from sharp edges, heat or moving parts, and do not store materials on top of it. Position the cord so it will not become a tripping hazard.

20. While the use of an extension cord is discouraged, it is recognized that the wiring layout of the shop may not allow the placement of the i2R™ directly next to an outlet. If it is necessary to use an extension cord, make sure the extension cord is in good condition, heavy enough to carry the current requirements, and installed to prevent a tripping hazard. An undersized cord will cause a reduction in voltage resulting in loss of power and overheating which may result in fire or electrical shock.

Other Safeguards

21. Remove and store adjusting keys and wrenches before turning on the power. If necessary, for visibility, apply safety markings to adjusting wrenches and keys.

22. Install safety guards consistent with general shop safety practices. Keep safety guards in place at all times when the machine is in use.

23. Check damaged parts immediately. Before further use of the machine, a guard or other part that is damaged should be carefully checked to determine if it will operate properly and perform its intended function.

24. Control liquids in your shop to limit the possibility of spillage which can damage the i2R™ causing or personal injury from electric shock or fire. Be careful with storage and use of cleaning fluids, finishes and solvents. Never use the table to apply or dry finishes.

Maintenance

25. Establish a weekly and monthly maintenance checklist and follow it diligently.
26. Routine maintenance should include periodic checks for alignment of moving parts, looseness or binding of moving parts, worn or bare wires, breakage of parts, skewed mounting and any other conditions that may affect its operation or cause injury. Analyze breakage or damage to determine the cause and take appropriate remedial action.
27. Do not operate the i2R™ if a component of the control system is damaged. It should be properly repaired or replaced before use.
28. Follow instructions for lubricating and changing accessories.
29. Store maintenance tools and supplies nearby, consistent with shop maintenance practices and resources.

Operational Practice

30. Never leave the machine running unattended. Always be in close reach of the emergency stop button.
31. Turn the power off and do not leave the machine until it comes to a complete stop.
32. Avoid pinch points and entanglement hazards. Keep hands and clothing away from the ball screws, thrust bearings, gantry, guide rails and rotating cutting tool while in operation.
33. Use the right tool at the correct speed and feed rate. Do not force a tool or attachment to do a job for which it was not designed. The right tool will do the job better and more safely.
34. Do not touch a cutting tool immediately after use. It will be hot and may cause skin burns. Exercise caution when handling the collet and spindle nut if the cutting tool is hot. Keep a heavy glove or oven mitt on hand for the purpose.
35. Do not lay a hot cutting tool on its side. Create a rack for cooling off hot cutting tools.
36. Use recommended accessories; improper accessories may be hazardous.
37. Do not use dull, gummy, or damaged cutting tools. Keep bits and other cutting tools clean and sharp for best and safest performance.
38. Turn off the machine before cleaning. Use a vacuum, brush or compressed air to remove chips or debris. Do not use hands.
39. Do not climb or stand on the machine. Serious personal injury and costly damage could occur if the machine tips over or the gantry is dislodged.
40. Remove loose items and unnecessary work pieces from the table before starting the machine.
41. Plan tool paths to make multiple passes rather than to take off a large amount of material at one time. This will reduce mechanical stress and heat on cutting tools.
42. Always secure a work piece to the spoil board using clamps, vacuum, or double-sided tape. If the work piece is mounted in a fixture, ensure that the fixture is securely held to the table. Never hold a work piece down by hand while operating.
43. Inspect the work piece material for any defects that may result in ejection of large pieces of scrap.
44. Make sure the work piece is free from nails, hardware, or other foreign objects.
45. After installing a cutting tool, make sure the collet is securely tightened. An unsecured cutting tool may fly loose from the collet and cause injury. Be sure that the adjusting wrenches have been removed and are secured before turning on the power.

Table of Contents

Glossary

4th Axis – An accessory that enables the CNC to rotate and index material, in a similar fashion to a lathe.

Axis – Direction of movement. On a three-axis machine: X (left-right), Y (front-back) & Z (up-down).

Array Work – Division of the machine work surface into square or rectangular work spaces so that the toolpath can be ran in each space to produce several identical profiles.

Ball End (Ball Nose) – A cutting tool that has a rounded cutting arc, where the arc diameter is equal to the cutting diameter.

Ball-screw – Drive system component. The ball-screw is rotated by the stepper motor and provided the means for moving the gantry and spindle along the axes.

Bed – The Bed of the CNC consists of a welded steel frame and an extruded aluminum table top, tongue and groove table top with integrated T-slots.

Breakpoint – The point in a toolpath file where a break is created. In a job file, a breakpoint will allow you to stop work and save a location to resume from later.

Chip Load – Chip load is the measure of the thickness of a chip a cutter will cut.

Climb Cut – Refers to the relationship of the cutting tool rotation to the direction of feed. A climb cut deflects the cutting away from the cut, with the direction of the feed.

Collet – A metal collar that holds the cutting tool within a spindle nut.

Conventional Cut – Refers to the relationship of the cutting tool rotation to the direction of feed. A conventional cut deflects the toward the cut, against the direction of the feed.

Compression Bit – A cutting tool with a combination of up and down shear cutting edges. Typically used for cutting laminate material to prevent tear-out on both sides of the sheet.

Deflection – Tool deflection occurs when the spindle speed and feed rate exert sufficient force to deflect the cutting tool. Deflection leads to excessive wear and chatter which can shorten tool life and will leave unwanted tooling marks on the material.

Down-Shear Bit – A cutting tool whose edges carve downward on the face of the toolpath. Reduces the potential for tear-out but requires a slower feed-rate.

Drag Knife – A cutting tool that holds a razor or knife blade used to cut and inscribe material. Used with spindle or router motors turned off, and freely spins as it follows the direction of the tool-path.

Dust Shoe – An accessory which aids in dust collection by limiting the spread of dust and debris by channeling the air flow through an attached dust collection system.

End-Mill – A cutting tool with a straight end, typically with a spiral flute(s). It creates a channel with a flat bottom perpendicular to the sides.

Feed and Speed – A combination of factors that must be set to control the work performed by the cutting tool.

Feed-rate – The speed at which the cutting tool moves along a work piece.

Flute Length – The length of the cutting portion on a router bit or cutting tool.

Flutes – The cutting edges or inserts of a router bit or cutting tool.

Finish Cut – A 3D tool-path the reduces or eliminates the irregular contours left by the rough cut.

Form Bit – A bit that carves a standard profile such as a round over, ogee or similar contours.

Gantry – The frame structure that straddles the bed and carries the spindle. It moves on guide rails along the length of the bed and is driven by the ball-screw system.

Gantry Bridge – The part of the gantry that lies between the upright towers. The upper bridge carries the spindle and cooling system. The lower bridge supports the weight of the gantry and affixes directly to the Y axis ball-screw.

Gantry Upright (Towers) – Parts of the gantry that extend upward, supporting the upper gantry bridge.

G-Code – A machine language that uses axis points and commands which the machine uses to move and perform functions.

Grid – The minimal movement units of the router. The router head automatically moves to the next grid position when the directional button is toggled in either continuous or step mode.

HHC – The handheld controller component of the control system.

Hold-down – A clamp or other such device used to firmly hold a workpiece or fixture to the table. Includes clamps, double-sided tape, vacuum pods and others.

Home Position – Mechanical point set using magnetic limits switches.

Origin – User designated zero point for the work piece. From which the router will reference the positioning of all cutting.

Plunge – The distance on the Z axis that the spindle and cutting tool moves toward, into, or along the material.

Plunge Rate – The speed of descent of the spindle on the Z axis.

Pocket Tool-path – A tool-path that creates a cavity in the horizontal surface of a work piece.

Post Processor – A software function that enables the CAD/CAM application to format G-Code enabling the control system of a CNC to follow the designated tool-paths.

Profile Tool-path – A tool paths that cuts along the profile of a set of vectors. Typically used to cut out the shape of a design.

Proximity Switch – A magnetic limit switch that is used to find the HOME position.

Restore Point – The point along a tool-path where the spindle will start to resume a tool-path following a break.

Rough Cut – A 3D toolpath where the initial cut is designed to remove unwanted material, leaving a rough contour.

Soft Limits – Limits on movement availability. Imposed by the work space boundaries and based on controller settings and the location of HOME. An "out of soft limits error" implies that based on the positioning of the workpiece, there is not enough room to move in a designated direction.

Speed Scale – Allows for the reduction of the feed-rate at the beginning of a tool-path to reduce the stress on a cutting tool as it first contacts the material.

Spindle Speed – Rotational speed of cutting tool (RPM).

Step Down – Distance in Z-axis that the cutting tool plunges into the material.

Step-over – The amount the cutting tool moves away from the previous cutting path as it routes the new path.

Stepper Motor – DC motor that moves in very precise steps upon the receipt of "pulses", which result in very accurate positioning and speed control.

Surfacing – The process of leveling the surface of the spoil boards so that they are perpendicular to the spindle.

Tool-path – User defined route which the cutter follows to machine a workpiece.

Touch-Off Puck – A device used to set the zero point (Origin) for the Z axis.

U-Disk – External data storage device that is inserted into a USB interface (USB Drive, Flash Disk etc).

Up-Shear Bit – A cutting tool whose edges carve upward along the face of the toolpath. Increases the potential for tear-out but allows for a slightly higher feed-rate.

VFD – Variable Frequency Drive which controls the speed (RPM) of the spindle. Enables the fine tuning of the spindle during the operation of a tool-path.

Working Envelope – The three-dimensional area that the spindle can travel within while cutting or milling.

Electrical Connections

WARNING All Electrical connections must be performed by a qualified electrical and follow any local codes and ordinances. Failure to comply may result in serious injury.

Electrical connections that are improperly installed or are outside operational specifications may cause machine damage and void any warranties that are in place.

The **i2R** model CNC machines are rated for 115V and are equipped with a standard plug that is designed for use on a grounded wall-outlet. It is recommended that these be connected to a dedicated 15-amp circuit.

If a grounded outlet is not available, an adapter may be used. The adapter should be used only temporarily until a properly grounded outlet can be installed.

Adapters will feature a green-colored lug or tab extending from the adapter which must be connected to a permanent ground, such as a properly grounded outlet box.

Canadian Electrical Code, C22.1 does not permit the use of a temporary adaptor.

Grounding Instructions:

Axiom CNC machines must be grounded. This grounding provides a path of least resistance for electrical current, which during a malfunction will reduce the risk of electrical shock.

All Axiom machines are equipped with an electrical cord with grounding conductor and plug. The plug must be used with a matching outlet that is properly installed and grounded in accordance with local codes and ordinances.

These plugs must not be modified, if a matching outlet is need, one must be installed by a qualified electrician.

Improper installation may result in electrical shock.

WARNING If grounding instructions are not completely understood or if in doubt as to whether the machine is properly grounded, a qualified electrician should be consulted.

Extension Cords:

While often discouraged, the use of an extension cord is often necessary for the location.

If an extension cord is necessary, it is recommended to place the machines as near to the power source as possible. Limit the length of cord to the required distance and make sure any cord used is in good conditions.

The use of worn or damaged cords should be discontinued with the cord replaced immediately.

When using an extension cord, be sure to use one that is heavy enough to carry the required current required and use only 3-wire extension cords that feature the correct 3-prong grounding plugs and 3-pole receptacles.

An undersized cord will cause a drop-in line voltage resulting in loss of power, overheating and runs the risk of fire.

See
Included Assembly
Instructions

Operations

Now that the machine is completely setup, it is recommended to become familiar with the handheld controller for operation of the machine, setting up a job and processing a file.

Controller:

Toolpaths, that are created in the design software are communicated to the CNC router through the handheld controller. In the case of the i2R machines, this is done through the RichAuto B11 series 3-axis controller.

Processing tool-paths or jobs can be performed either by using a USB Flash drive (USB2.0 of 8GB or smaller, FAT32) or from the internal memory.

Internal memory is recommended for jobs of 1 hour run time or longer, and any files that will be used frequently.

The RichAuto B11 Key pad which is seen here, 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).

   Combination functions, where two keys are pressed at the same time.

Operating Procedure:

Machine operations may be done several ways; either through the controller keypad or through the design files (tool-paths) which can be downloaded to the controller through the USB (Udisk) or ran from internal memory.

Before performing an operation:

1- Make sure machine stand leveling feet (if available) have been lowered to prevent the machine from moving.
2- Ensure that the workpiece is securely held to the table which can be done using any of the following:
    a. Clamps
    b. Fixtures
    c. Vacuum hold-downs (Optional Accessory)
    d. Double sided tape
3- Turn on the machine by pressing the Stainless (green backlit) power button. The controller display should light, displaying the loading screen.
    a. The emergency stop button may need released if it has been used (rotate clockwise to release).

Once done loading, a HOME operation must be performed before any other function.

HOME Position:

When turning on the machine, once the controller has finished loading, the following message will be displayed:

At this point, the controller can be used to scroll up and down the list using the **X+/X-** buttons. The top option of "All axis home" should be selected using the **OK/ORIGIN** button.
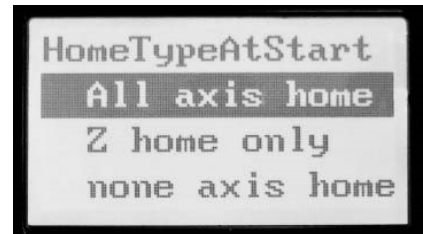
Home position is determined by a set of magnetic proximity sensors, one of which can be found on each axis. These sensors can be calibrated, however, their position should not change.

During a HOME operation the following will occur:

1- First the Z axis will raise to the top of the axis travel.
2- Then the X & Y axes will travel, simultaneously, to the front left corner of the machine.
3- When complete the controller will display the coordinate screen.

The coordinate screen will display the current position of the router in relation to the programmed **Origin** (0,0 point) as well as indicate which coordinate state is currently being used (coordinate states covered on pg. 16).

It is important that a **HOME** operation is performed at every start-up to ensure that the limit switches and internal software settings are properly working. All machine movements are software driven and based upon the **HOME** position.

At any time during a machine operation, the HOME operation can be selected to run from the controller using the HOME button. (Must be done from the main coordinate screen)

Selecting the **HOME** button will display the **HOME Type** menu where several options are available, such as:

1- Home all
2- Home X (X axis only)
3- Home Y (Y axis only)
4- Home Z (Z axis only)

Moving Router (Spindle) head:

Movement of the router or spindle head can be done manually several ways. Most of which can be toggled between using the **MODE** button. (Chosen mode will be displayed, temporarily on the lower portion of the controller screen)

Continuous (default mode) and **Step** mode, both use the internal grid settings (minimum movement distance) to aid in accuracy. The default grid has two factory settings depending on the movement speed mode selected (LSP or FSP). In either Continuous or Step mode, the machine will move to the next grid distance when the button is toggled.

The movement speed mode selected can be seen on the controllers coordinate screen display, on the far right, third row down. To switch between these two modes, use the **Fast/Slow** button.

LSP-Low speed- has a grid of 0.1mm for positional movements and a continuous speed that is set at 1500mm/min default. While in this mode, every tap of an axes directional button will move the axis 0.1mm (0.004”).

FSP-High speed-has a grid of 0.5mm for positional movements and a continuous speed that is set at 2500mm/min default. While in this mode, every tap of an axes directional button will move the axis 0.5mm (0.020”).

1- Continuous mode: Hold down any axis directional button, i.e. **X+/-, Y+/-, Z+/-** and the head will move until the button is released.
   a. The controller will display the location of the router in relation the current Origin as it moves.
   b. During the first 0.5 sec, the button is held down the machine will operate as if in step mode and move to the nearest grid at a lower speed.
2- Step mode: Tap (toggle) any of the directional buttons, i.e. **X+/-, Y+/-, Z+/-** to move the router head in grid-based increments. Useful for precise placement of coordinates and tooling.
   a. Movement speed is default at one grid distance per 0.5 sec.
   b. Low speed (**LSP**) default is 0.1mm grid
   c. High speed (**FSP**) default is 0.5mm grid
3- Distance mode: Allows the input of a set distance to which the router will move. To use the distance mode:
   a. Press **MODE**, to cycle through until it displays **Dist**.
      i. The numerical distance will be highlighted
   b. Enter the required distance and press **OK**.
      i. Display will now show **Dist** with the numerical distance no longer highlighted.
   c. Tap any of the directional buttons to move the router head the programmed distance in that direction.
4- GoTo Position: Allows a unique ability to move to a specified point. **(In relation to the programmed Origin.)** Press **Mode + Menu** simultaneously to bring up the **GoTo Pos** screen.
   a. The controller will display the **GoTo** coordinate screen.
   b. Use the up and down arrows found on the **X+/-** buttons to scroll up and down the list.
   c. With an axis highlighted, use the Delete button to remove the existing coordinate
   d. Enter the desired coordinates for the chosen axis and press **OK**.
   e. When all axes have been set, press **OK** a final time to approve the movement.
   Once the movement has been approved, all three axes will move simultaneously to the specified locations. Be mindful of fixtures and any hold-downs that may become obstructions during movement.

Setting Work Origin:

Creating an **Origin** will establish a zero point or reference point from which the machine will perform the cutting process set in the tool-path.

The **Origin** should match the zero point (<u>datum</u>) of the uploaded toolpath created in the design software.

An Origin for both the X & Y axes and the Z axis need to be set before beginning operation of a file. Failure to set both Origins may result in damage to the machine or the cutting tool.

To set the **Origin**:

1- Make sure the collet is properly inserted into the collet-nut before being threaded onto the machine. Then insert the tooling into the collet and tighten using the included wrenches.

   a. Use only ER-20 collets, of the correct size matching the shaft diameter of the selecting tooling.
2- Move the spindle or router to the desired location using the X and Y directional buttons.
   a. Remember that the controller can be switched between movement speeds using the **FSP & LSP** settings. **Step** mode or grid movements can be used to place the bit more accurately for the **Origin** placement.
   b. This location is typically one of the corners of the work material or the exact center point.
3- Press **XY-0** to set the **Origin** for the X and Y axes.
   a. The will instantly change the displayed coordinates of the X and Y axes to 0.00
4- There are a couple ways to set the **Z-0** or Z axis **Origin**, which may change depending on the material or router bit that is being used.
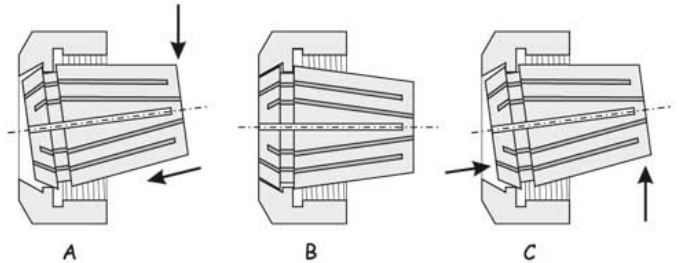
   a. Using the provided tool touch-off device. (Device plugs into female at the top of Z-axis)

   i. Place the device on top of the material, centered under the cutting tool. **It is recommended to use the top of the material, however be sure to select this in the design software when setting up the material parameters.**
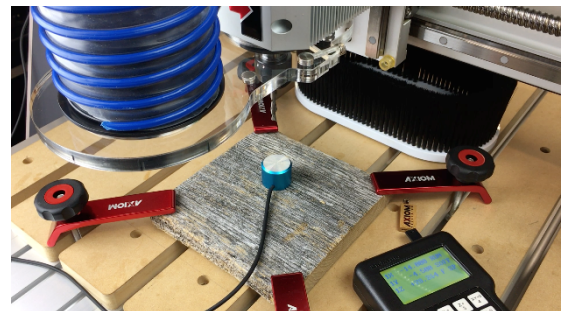
   ii. With material that is not uniform in thickness, the top of the spoil board (base of material) may be used for the tool touch-off device placement. **Again, be sure that the selection in the software matches the chosen location of the device.**

   b. Once the device is in place and centered under the bit, press the **Tool Set** to start the touch of procedure. If done correctly, the router head will slowly lower until the bit contacts the device and then return to a raised position. The **Z-axis Origin** is now programmed.
      i. It may be noted that a delay may occur during touch off. The bit may make contact, raise just slightly and slowly lower again to calibrate. This is a normal occurrence and may happen randomly.

c. <u>Fragile</u> bits may require that the Z-Axis Origin be set manually without the use of the touch-off device to prevent the tooling from breaking. **To set the Z-0 point manually:**

    i. Place the controller into **Step** mode, and slowly lower the router head while rotating the tooling by hand until resistance is felt as the bit contacts the workpiece.

    ii. For more precise placement, use a thin piece of paper. Slide the paper back and forth on the material until drag is felt as the tooling presses against the paper and material.

    iii. Once lowered to the desired location, use the **Z-0** button and select **Clear Z** to program the Z-axis origin.

⚠CAUTION   **It is recommended that the Z-axis Origin be set relative to the top of the work material to reduce risk of cutting into the work table.**

Once the Origin is completely set, the coordinate screen will reflect this information by showing the X & Y axes zeroed while the Z-Axis will have raised to the programmed safe height and will reflect that distance.

Remember that the Z-0 location is the surface of either the material or the spoil board, in which the controller should only read zero along the Z axis when the bit is touching that surface.

NOTE: When the machine is restarted, a HOME operation must be done to reset that reference point. However, the last programmed Origin (which is in reference to HOME) is still stored in memory. Simply press the Origin/OK button and the machine will return to this location.

Programming Additional Origins:

The RichAuto controller allows multiple offsets (**Origins**) to be programmed. By default, the coordinate screen will show something similar to the following:

```
1X      0.000 MAUN
1Y      0.000 SOFF
1Z      0.000 L SP
Continue
```
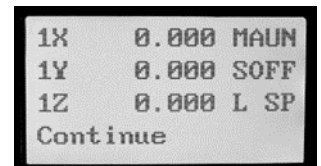
Where the #1 found before each axis identifies the coordinate state the controller is in. Each coordinate state can have a unique programmed **Origin**. In the example, the #1 coordinate state is being used.

To switch between coordinate states, press MENU & (any number 1-9). The coordinate screen will change and show that set of coordinates.

It may be easier to hold-down MENU; press the desired number; then let go of both buttons.

This feature is useful when creating fixtures or when separating the machine surface into quadrants for specific tasks.

All Origins are in reference to the same HOME and can be alternated as needed.

For example:

If using a 48" work area, to perform jobs that are less than 24" in size. The surface can be set up as two 24" areas. Using the same HOME.

In the Menu + 1 area (coordinate state #1), the **Origin** would be set for the first piece of material.

In the Menu +1 area (coordinate state #2), the **Origin** would be set for the second piece of material.

The materials can be loaded onto the machine, preferably using a fixture for ease of referencing. The first job can be performed from the #1 state. Then the controller can be switched to the #2 state, and the second piece of material can be machined.

Work Piece #2

Work Piece #1
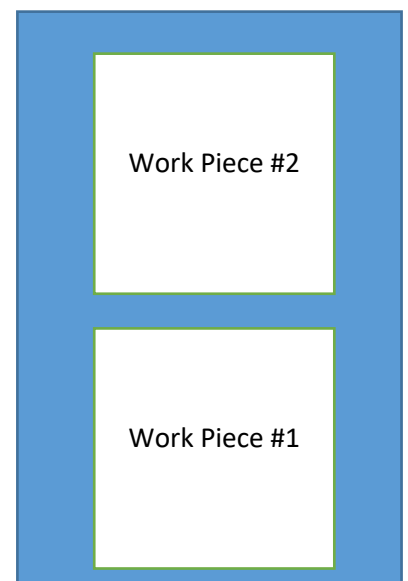
Setting Spindle RPM:

The i2R model machines are equipped with a VFD invertor for spindle RPM control. This can be found on the front of the white control box.

To adjust the RPM, simply turn the dial. Clockwise (right) to increase RPM and Counter-clockwise (left) to decrease RPM.

This can be adjusted at <u>any time</u>. Included during a machine operation. Allowing RPM to be dialed in dynamically as needed.

The display shows the RPM in 60Hz frequency. This means that while the RPM is 0-24,000, the display will show 0-400. Each 100-displayed is equal to 6000RPM.

Delta VFD - RPM Values	
Frequency	RPM
250	15,000
267	16,000
283	17,000
300	18,000
317	19,000
333	20,000
350	21,000
367	22,000
383	23,000
400	24,000
Frequency equals desired RPM/60	

Once the spindle becomes active, display will alternate from frequency to actual RPM.

The buttons on the VFD, are intended to be used for programming only and should not be used to set the spindle speeds.

Processing a File:

Once the **Origin** is placed (all axes) and the RPM is set, the file can be selected, and machining can begin.

To ease the use of a USB flash drive (U-disk), once files are saved to the U-disk always select to Eject the drive to safely remove it from the computer. This will help ensure that the controller will recognize the U-disk when it is inserted.

To begin (From the coordinate screen):

1- Press the **Run/Pause/Delete** button.
2- Select either the U-disk, Internal Memory a Recently used file (must be one current U-disk or in memory)
   a. Use the X+/- arrows to scroll up or down the list.
   b. Highlight the desired selection and press **OK** to select.
3- The files in that directory will be displayed.
   a. Display shows three files at a time.
   b. X+/- will scroll one item at a time.
   c. Y+/- will scroll one page at a time.
   d. Highlight the desired tool-path (**.mmg** filetype), press **OK** to select
4- The controller will display a set of working parameters for the file.
   a. <u>Work Speed</u>: Similar to feed-rate. This can be ignored as the controller is setup by default to take the feed-rate settings from the software. **Settings can be changed allowing feed-rate to be set here**.
   b. <u>Fast Speed</u>: Shuttle speed. This is the speed the machine will travel between cutting locations.
   c. <u>Speed Scale</u>: A percentage of the feed-rate. (**See Speed Scale Adjustment**)
5- Press **OK** to begin the tool-path operation.
   a. After a short count-down (4sec) during which the spindle will come up to speed, the file will begin.

Speed Scale Adjustment:

The speed scale (percentage of the feed-rate programmed within the software) is displayed before a tool-path file is set to begin. This can be changed both before the file begins (during the file parameters screen) and while the tool-path is in operation. To change this parameter setting:

1- Before the tool-path operation begins - While the file parameters screen is being displayed, highlight the Speed Scale setting.
    a. Use the **Delete** button to select and remove the original parameter. Press **OK** after entering the new parameter. **Must be 0.1 (10%) through 1.0 (100%) in 0.1 increments.**
2- During toolpath operation – While the file is in operation, the Y+/- buttons can be used to increase or decrease the Speed Scale in 0.1 (10%) increments.
    a. Cannot exceed 1.0 (100%) of the feed-rate programmed in the software.

Pausing During Tool-path Operation:

A file that is in operation, otherwise known as a job, can be paused. While **Paused**, some changes can be enacted. This function should only be used for very brief pauses to adjust settings, such as bit height or position.

To Pause a job:

1- Press the **Run/Pause** button during operation.
    a. The machine movement will immediately stop.
    b. Spindle remains active.
2- While the machine is **Paused**, the bit position can be adjusted in any direction.
    a. It is recommended this be done in **Step** mode, and in **Low** speed (**LSP**).
3- After making the adjustments, press **Run/Pause** to continue the file operation.
    a. The display will show a **Restore Position?** prompt.
        i. Press **OK** to resume using the modified settings.
        ii. Press **Cancel** to resume using the previous settings.

Be advised, changes that are made will only affect the tool-path in operation.

Setting a Breakpoint:

Like the **Pause** function, a Breakpoint or Stop can also be used. In this case, the Breakpoint operation can be used for extended periods including saving the location and returning to finish a job as much as several days later, after restarting the machine any number of times.

To create a Breakpoint:

1- Press the **Stop/Cancel** button during an operation.
    a. **Pressing a 2nd time will cancel the Save Break function and the position will be lost.**
    b. When the **Save Break** prompt appears, press **OK**.
2- The display will show the available locations (1-8).
    a. Previously stored Breakpoints can be overwritten.
    b. Use the **X+/-** buttons to scroll through the list
    c. Use the **OK** button to select he desired location.

Before resuming from a Breakpoint, a HOME operation needs to have been performed if the machine was just restarted.

3- When ready to resume working from the stored Breakpoint, press the **Run/Pause + (1-8)** together. **The saved location number (1-8) should be the number used.**
   a. Controller will display the same **Save Break** locations seen before.
   b. Use the **X+/-** buttons to highlight the desired file and press **OK** to select.
   c. The current g-code position for that break will be displayed. (**The g-code lines can be scrolled through at this point and the break resumed anywhere in the list**).
   d. If that g-code position is acceptable, press **OK**.
   e. The display will have moved on to the file parameters that appear before all file operations. Once these are approved the file will begin at the saved location.

Power Failures:

During an accidental power failure, a soft version of a **Breakpoint** is set where the current parameters and coordinates are automatically saved by the system. This information will become available to resume at the next immediate start of the machine.

The information is not saved permanently, thus if it is not resumed at the next start of the machine, the information will be lost.

When power is returned to the machine, perform a **HOME** operation. Afterwards the **Want to Restore** prompt will be displayed.

Press **OK** to restart the unfinished file operation.

⚠CAUTION | **If the operation is Canceled, ability to resume from the saved location will be permanently lost.**

Spoil Board Surfacing

Before beginning to use the machine, there are just a couple of key items that it is important to cover.

First, reading the manual. Typically, this is not stressed enough, with many questions being asked which would are often covered in the text.

Second, is the surfacing of the spoil boards which are included with the machine. Surfacing of these MDF strips ensures that the work plane is perfectly parallel to the path of the spindle while being perpendicular to the router bit itself.

Rest easy knowing that this simple surfacing operation is largely taken care by the Axiom team. A surfacing program has been pre-loaded into the handheld controller's internal memory.

To setup and run this stored job, please refer to the previous sections covering setup of ORIGN locations and file operations.

Then use the following:

1. Turn ON and HOME the CNC machine.
2. Set the X/Y Origin while in the HOME position.
3. Insert a 1.25" Mortising or Surfacing tool into the spindle. (Requires ¼" Collet)
   Bit is included in the Axiom 3-pc Start Bit Set
4. Jog the X/Y Axes just a couple inches toward the center of the table…and use the Touch-Off Device to set the Z Axis Origin at the top of the spoil boards.
5. Set the RPM to F300 (18,000 RPM)
6. Use the RichAuto Controller to select the surfacing program from the internal (SD) memory.

Each machine is loaded with the correct surfacing program for its table size…. with each file running both a Pocket and a Profile Tool-path.

i2R-4SFC (24" x 24")

i2R-6SFC (24" x 36")

i2R-8SFC (24" x 48")

The following is the Job Setup Sheet from which all the information for the surfacing job can be seen.
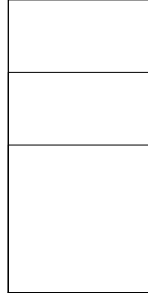
Job Setup Sheet
i2r

Vectric
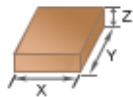PASSIONATE ABOUT CNC

Job Layout

Material Border

Material Setup

Material Block:
(X):600mm (Y):1210mm
(Z):25mm

Home / Start Position:
X:0mm Y:0mm
Z:20.32mm

Datum Position:
Z-Zero:Top of Material   XY: Bottom Left Corner   Clearance Z1: 5.08mm
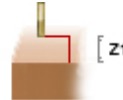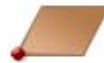
Toolpaths Summary
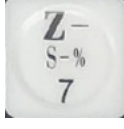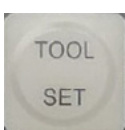
Toolpaths : 6	Tool Name:	Time Estimate : 02:14:28
i2R4PKT	[1] 1.25" Mortising Bit	00:28:05
i2R4PFL	[1] 1.25" Mortising Bit	00:02:23
i2R6PKT	[1] 1.25" Mortising Bit	00:41:40
i2R6PFL	[1] 1.25" Mortising Bit	00:02:59
i2R8PKT	[1] 1.25" Mortising Bit	00:55:47
i2R8PFL	[1] 1.25" Mortising Bit	00:03:34

RichAuto A11 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− ∨F-% 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.

Common Combination Keypad Shortcut Commands:

Key combination	Function
OK + MENU	Update Controller Software
MENU + 0	Machine Coordinate State (identified by prefix A)
MENU + (1-9)	Workpiece Coordinate States (store up to 9 different systems)
MENU + ON/OFF	Z-axis Tool Touch-off operations
RUN + (1-8)	Begin **Break** processing
RUN + FAST/SLOW	Open Advanced Process Menu
RUN + 9	Repeat Last Operation
MENU + MODE	Enter Coordinates for **Go To Position**
MODE + FAST/SLOW	Find Center (X or Y axis)
RUN + STOP/CANCEL	Last Work Info

Advanced Processing

Functions found under the Advanced Processing category serve to perform a specialized task. The Advanced Processing menu can be accessed using the **Advanced Function** key or the combination of **Run + FAST/SLOW**.

Array:

Allows multiple copies of a file to be processed along a set number of rows and columns.

Before running an **Array**, the parameters need to be set. Such parameters will include the row/column counts and the spacing between the rows/columns.

To set the **Array** parameters:

1- From the coordinate screen, press **Menu**
2- Select **Auto Pro Setup** and scroll down to find the **Work Array** option.
3- The Work Array menu will be display and has several options (press **Delete** to change these settings).
   a. Row & Column Counts-
   b. Row & Column Spacing-
   c. Interval-
4- After the parameters are setup, **Run/Pause + (1-8)** can be used to access the **Advanced Process** menu.
   a. Select **Array Work**
   b. Then Select the file to be used with the programmed settings.
   c. The normal file parameters will be displayed, and operation will begin once approved.
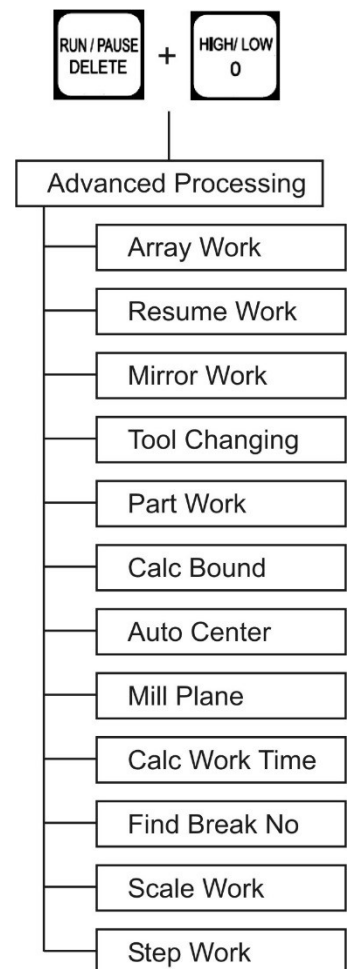
Resume Work:

Operates much the same as using **Run/Pause + (1-8)**, to display the **Save Break** locations. (For more information see **Setting a Breakpoint**).

Mirror Work:

Allows a job to be mirrored on either the X or Y axis. (**Its recommended to use an Origin point in the center of the material**).

Tool Changing:

While the controller does not allow automatic tool-changing as an option, there are a couple helpful selections available.

This option, when selected will bring the router head to the **HOME** position for tool-changing. The controller will display a prompt instructing for the user to change the tooling then to press the **OK** button when finished to return to the previous position.

⚠CAUTION This will not prompt for a Z-axis Tool Touch-Off. To avoid damage to the machine or job material, the Z-0 point must be set before continuing.

Stop State (Back Position):

In addition to the above option **(Advanced Processing Tool Changing)**, there is another option which can be more effective specifically when using fixtures.

The **Stop State** shares some similarities with the **Origin** point since it will allow a defined location that can be programmed anywhere within the machine limitations.

This point must be activated in the controller software before being programmed. Once complete, the machine will go to the programmed location at the end of each process. (**Location can be changed as needed**).

To program a Stop State:

1- From the coordinate screen, press **Menu**.
2- Select **Auto Pro Setup**, then scroll down to find the **Stop State**.
3- The Work Stop State menu will be displayed which has several options.
    a. **FinAct** (Finish Action)- press **Delete** to change the settings.
        i. Pickup Z- Will leave the X & Y axis wherever they are at the end of the file operation, raising the Z-axis to the set safe height. (**Default**)
        ii. Back to Work Origin- Will move the X & Y axes back to the **Origin**, while raising Z-axis to the safe height.
        iii. Back Home- Will move all axes back to the **HOME** position.
        iv. Back Position- Will move all axes to a programmed location after the file operation is complete.
    b. Programming the **Back Position** can be done similarly to the **Origin** point. Move all axes to the desired location and height.
        i. Press **Menu + Z-0** to set the location.
        Highly recommended that the Z-axis is raised to the highest elevation to help prevent damage as the machine travels.
    c. A specific set of coordinates can also be programmed by entering the desired location for each axis manually.

Part Work:

Using this processing option allows a specific line in the g-code from a file operation to be selected as the start and stop locations.

To setup a **Part Work** operation:

1- Select **Part Work**, then select the file.
   a. A message will be displayed prompting the input of the Start source.
      i. Press **OK**.
      ii. Scroll through the list of g-code lines to select the desired line. **(X+/- to move one line at a time. Y+/- to jump between the top/bottom of the list)**.
      iii. Press **OK**.
2- A message will be displayed prompting the input of the Stop source.
      i. Press **OK**.
      ii. Scroll through the list of g-code lines to select the desired line. **(X+/- to move one line at a time. Y+/- to jump between the top/bottom of the list)**.
      iii. Press **OK**.
3- The normal file parameters will be displayed, and the operation will begin once approved.

Calculate Boundaries:

This application will calculate the boundaries of the file. Allowing a user to verify that an operation will fit within the machine limitations.

Will show the operation boundaries for each axis in metric units.

Auto Center:

Allows a pair of individual locations to be selected along the X or Y axis, from which the exact center point can be located.

To use the **Auto Center** function:

1- Select **Auto Center**. Then choose to find the center along either the X-axis or the Y-axis.
2- The display will return to the coordinate screen.
   a. A **First of X/Y** prompt will be displayed at the bottom of the screen.
      i. Move the X-axis to the 1st location.
      ii. Press **OK**.
   b. A **Second of X/Y** prompt will be displayed at the bottom of the screen.
      i. Move the X-axis to the 2nd location.
      ii. Press **OK**.
   c. The X-axis will automatically adjust to the exact center of the two locations.

Mill Plane: (Surfacing Operation): *Not Recommended for Surfacing of Spoil Boards…Please refer to the Spoil Board Surfacing Section*

Performs a near automatic surfacing program where the user enters the metric X & Y limits. This does not require a tool-path or the entry of gcode but it does require a complete metric setup which is often complicated.

A surfacing file created in the CAD/CAM program of choice, consisting of a pocket toolpath raster style (preferably along the Y axis) and a profile toolpath centered on the line to clean up the outside edge.

There are two types of milling options; **Scan** & **Encircle.**

Regardless of the type chosen, this program does not use an origin. It accepts the current position of the bit to be the starting point (0,0) with all movements based from that location. A **"soft limits error"** may occur if the limits entered cannot be supported from current location.

It is suggested that a test run be performed with the Z axis elevated above the material

To use the **Mill Plane** function:

1- Select **Mill Plane**. Then choose either **Scan** or **Encircle**. (Setup requires metric units)
    a. **Scan** performs a surfacing program that follows a parallel path to either the X or the Y axes.
        i. **ScanType:** Choose either the X-axis or the Y-axis.
        ii. **Width/Height:** Enter the X and Y axis dimensions.
        iii. **Diameter:** Enter the diameter of the tooling.
        iv. **Depth:** Enter the total depth for the operation.
        v. **Z Step:** Enter the pass depth for the bit.
        vi. **TRatio:** Enter the step over percentage. (Default 0.8 or 80%)
    b. **Encircle** performs a surfacing program that moves in a circular pattern around the X and Y axes, slowly working its way inward. (Setup requires metric units)
        i. **ScanType:** Choose either **AC** or **C** for movement.
            1. **AC** = counterclockwise.
            2. **C** = clockwise.
        ii. **Width/Height:** Enter the X and Y axis dimensions.
        iii. **Diameter:** Enter the diameter of the tooling.
        iv. **Depth:** Enter the total depth for the operation.
        v. **Z Step:** Enter the pass depth for the bit.
        vi. **TRatio:** Enter the step over percentage. (Default 0.8 or 80%)
2- Operation should begin immediately. Be sure that the Z axis height and RPM are properly set before using this operation.

Calculate Work Time:

This will operate much the same as Time Estimation tooling found within the chosen CAD/CAM Design Software. However, this function will allow you to calculate the operating time of a specific file based upon the file parameters in the tool-path and the machine controller settings. (**Will not factor in Speed Scale adjustments**).

Find Break No:

Like both **Resume Work** (which is also found in the Advanced Process menu) and using **Run/Pause + (1-8)**, to display the **Save Break** locations.

While similar to the above, this function serves two purposes. It will both find the line number of a breakpoint for a specific file or gives the option to resume working from that point.

(Unlike the Save Break menu, this will not allow for the selection a different line to resume work from).

To use the Find Break No. option:

1- Select **Find Break No**.
    a. Select the file (either from Udisk or Internal memory).
    b. Prompt will give the following instructions. (Use X+/- buttons to scroll)
        i. Press **OK** to resume from existing break
        ii. Press any other key to display the break line number only.
    c. Approve speeds to begin machining if applicable

Scale Work:

This feature will allow for the increase or decrease in scale existing tool-paths. Origin and all other settings will remain the same.

Be sure that there is enough room within the machine limits to support the increase in scale.

To use the **Scale Work** option:

    1- Select **Scale Work**.
- a. Select the desired file (either from Udisk or Internal Memory).
- b. Set scale for the X, Y & Z Axes.
- c. Approve speeds to begin operation.

Step Work:

This function will allow the tool-path to be operated one step (one line of gcode) at a time.

To enter Step Work mode:

    1- Select **Step Mode**.
- a. Select the desired file (either from Udisk or Internal Memory).
- b. Prompt will give the following instructions. (Use X+/- buttons to scroll)
    - i. Press **RUN** to continue working (normal operation).
    - ii. Press **OK** to perform steps.
        1. Each press of **OK** will perform one step.

Data Restoration and Updates

Factory defined parameters and settings within the controller can be modified. It is generally recommended that any changes be done with caution unless being advised by Axiom Precision Technical Support.

These changes are easily done and may not always be intentional.

However, restoration of the factory settings is easily accomplished. New machines should arrive equipped with a USB drive containing a number of support files including restoration programming and a copy of the controller firmware for emergency restorations.

Restore/Backup Data:

Backup files can be created at the machine. This is how the above-mentioned restoration files were created.

If you have made changes to your controller settings, then it would be advisable to create a backup and store it via internal memory for use later should the parameters need restored.

To do this:

1- Press **MENU**.
   a. Scroll down and select **System Setup**.
   b. Scroll down and select **Backup Data**.
        i. Choose where to save the file (Flash Disk or Internal Memory)
   c. The Data.bak file is now created with all the current settings for use later.

If the controller needs to be restored and the necessary file is not available, contact the Axiom Technical Service department at support@axiomtoolgroup.com.

1- To use the Axiom supplied **Restore** files:
   a. First save the file (Data.bak) to a USB flash drive. (Or locate this file on the factory supplied USB)
        i. Insert the flash drive into the controller.
        ii. Press **MENU** & scroll down and select **System Setup**.
        iii. Scroll down and select **Restore Data**.
        iv. Choose the location where the file can be found. (Either the Internal Memory or the Flash Disk)

A prompt will be displayed indicating that the parameters have been restored to the factory settings.

When the supplied Axiom USB is inserted into the PC, users will notice that it contains two drives: one labeled Factory and another labeled User.

The Factory drive which is visible only at the PC contains the support files. In the event that a file on the Factory drive is needed at the machine, such as for an emergency restoration…. the required file can be copied/dragged to the User drive.

The User drive is visible to both the PC and the CNC Controller and can be used for restoration purposes or simply for normal CNC operation.

System Updates:

Occasionally as software evolves, updates may become available for the controller firmware.

(Current Firmware will be supplied with the machine on the Axiom USB).

If a firmware update is needed, one may be obtained from the Axiom Precision Technical Service department by emailing support@axiomtoolgroup.com.

Much like the Backup/Restore files, the current version of the firmware Update may be used to correct certain controller software settings.

To install a **System Update**: (Emergency State Option) For use when the Menu cannot be accessed.

1- Save the supplied file to a USB flash drive.
   a. With the machine powered **OFF**, insert the drive into the controller.
   b. Enter the Emergency State mode by holding down the **OK** button as the machine is turned **ON.**
      i. Release the **OK** button, once the loading screen appears.
   c. Select **Update System**.
   d. Select the location of the file (Udisk or Internal memory).
   e. Select the file.

The update will begin automatically. Once complete the controller will display a prompt indicating that the update has been successful, the controller will need to be restarted to complete.

After restarting the machine, normal operation may be resumed.

(Normal Menu Option).

1- Save the supplied file to a USB flash drive.
   a. With the machine powered **ON**, insert the drive into the controller.
   b. From the main coordinate screen, Press the **Menu** button.
   c. Scroll down and select **System Setup**.
      i. Select **Update System**.
      ii. Select the location of the file (Udisk or Internal memory).
      iii. Select the file.

Again, the update will begin automatically. Once complete the controller will display a prompt indicating that the update has been successful, the controller will need to be restarted to complete.

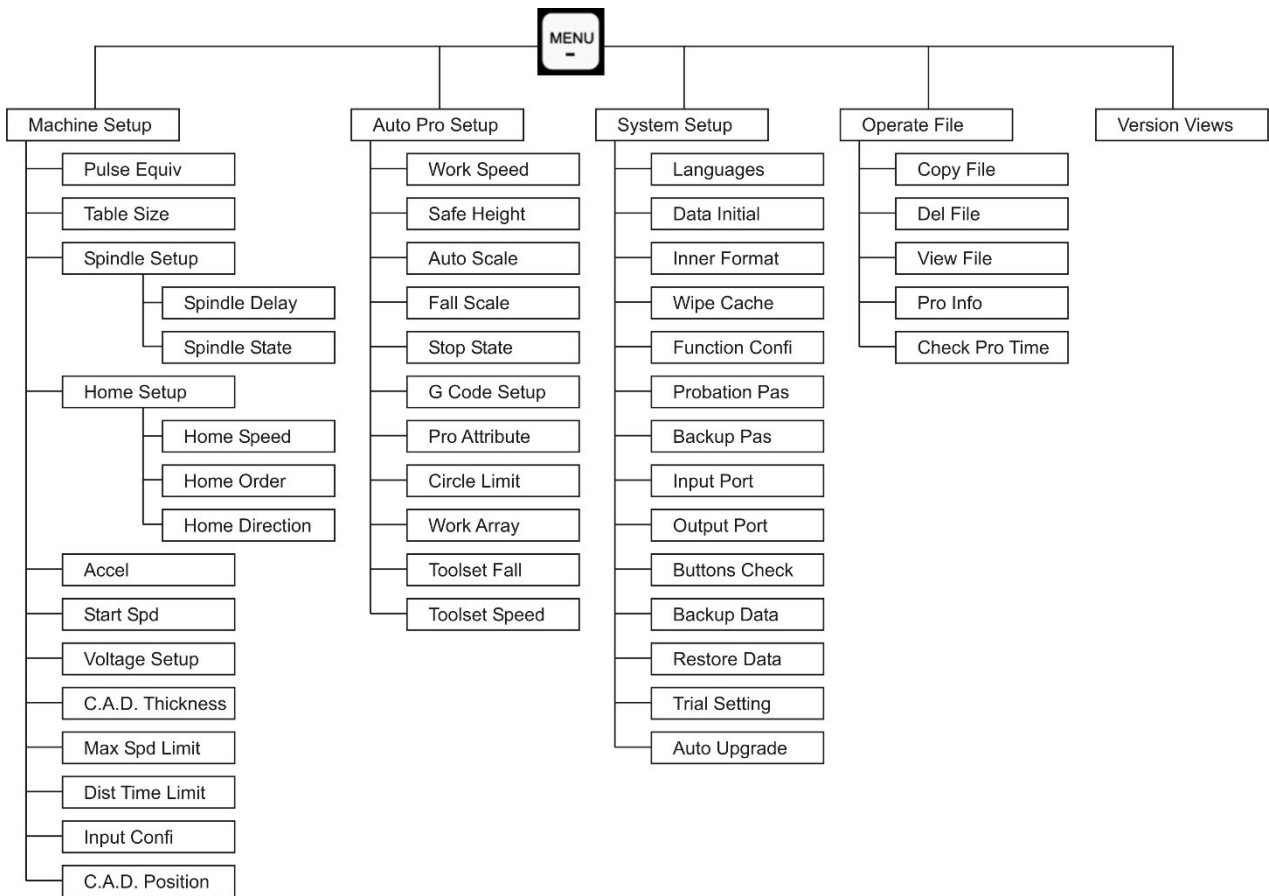After restarting the machine, normal operation may be resumed.

Controller MENU Layout

MENU

Machine Setup
- Pulse Equiv
- Table Size
- Spindle Setup
  - Spindle Delay
  - Spindle State
- Home Setup
  - Home Speed
  - Home Order
  - Home Direction
- Accel
- Start Spd
- Voltage Setup
- C.A.D. Thickness
- Max Spd Limit
- Dist Time Limit
- Input Confi
- C.A.D. Position

Auto Pro Setup
- Work Speed
- Safe Height
- Auto Scale
- Fall Scale
- Stop State
- G Code Setup
- Pro Attribute
- Circle Limit
- Work Array
- Toolset Fall
- Toolset Speed

System Setup
- Languages
- Data Initial
- Inner Format
- Wipe Cache
- Function Confi
- Probation Pas
- Backup Pas
- Input Port
- Output Port
- Buttons Check
- Backup Data
- Restore Data
- Trial Setting
- Auto Upgrade

Operate File
- Copy File
- Del File
- View File
- Pro Info
- Check Pro Time

Version Views

Machine Setup

These parameters are set by the manufacturer specifically for your CNC routing machine. It is usually unnecessary to change the Machine Setup defaults unless the parameters of the actual machine change.

Pulse Equivalent: Number of pulses sent by system to stepper motor for each 1mm of movement (unit: pulse/mm).

Preset Defaults: X Equivalent 2560.000

Y Equivalent 2560.000

Z Equivalent 5120.000

Table Size: Preset values; prevents machine over-travel. Verify table size from Specifications Chart.

Spindle Setup:

Spindle Delay: Sets the time at which the spindle starts after reading the file (unit: microseconds).

Spindle State: Sets the number of spindles in operation and the input /output shift points. Default set.

Home Setup:

Home Speed: Sets speed separately for each axis when homing; Default is X/Y: 2000 mm/min. and Z: 500 mm/min.

Home Order: Sets desired sequence of axes when homing; Default is Z, X & Y. Will move the Z-axis first, then move the X & Y-axes together.

Home Direction: Sets each axis for positive or negative direction, based on Home position.

Toolset Setup:

  C.A.D Position: Sets the option for placement of the Z-axis device, for use in setting the Z-0 point; Default is (In PLC) in place, referring to the current placement of the bit when the toolset command is triggered.

  C.A.D Pickup: Sets the distance for which the Z-raises after contacting the puck; Default is 60.00mm

  C.A.D Thickness: Sets the thickness for the Tool-Touch off device; Default is 25.00mm

  Toolset Speed: Speed setting for the Toolset operation; Default is 600.00 (units undefined)

Acceleration: Can be adjusted to improve line and curve motions; Default is 800 mm/s^2.

Start Speed: Sets skip speed and completion speed.

Voltage Setup: Sets input and output terminal status.

Max Speed Limit: Sets maximum speed of head movement; Defaults are X/Y: 3000; Z: 1500.

Distance Time Limit: Sets period in which machine can sit idle before reverting to continuous mode; Default is 5 seconds.

Input Configuration: Used to configure the pins used by the handheld controller driver board (DSP) for inputs and outputs.

Auto Pro Setup

Work Speed: Sets work speed (Default 3500 mm/min.) and fast speed (Default 2000 mm/min.).

Safe Height: Clearance height for Z axis (mm); Default set to 40.000mm

Auto Scale: Used to set Speed scale, referring to the allowed percentage of tool-path movement speed. Will always be set between 0.1 (10%) to 1.0 (100%) of max feed-rate which is set in the g-code. Always set in 0.1 (10% increments. Cannot exceed max 1.0 (100%) **(See Speed Scale Adjustment)**

Fall Scale: Sets fall scale (Default 0.200mm) and fall height (Default 5.000mm). Fall down scale takes effect when spindle descends to fall height.

Stop State: Establishes router stop position after machine is done; Default FinAct (Finish Action) is set to Pickup which will pick up the Z-axis to the safe height parameter.

G Code Setup: Sets special code read configuration in G code.

Pro Attribute: Allows for a number of controller parameters to be set for specific applications.

Circle Limit: Default is 1000.00

Work Array: Sets up array parameter, including column count, row count, column space, row space, and interval (microseconds).

Toolset Fall: Controls the distance that the Z-Axis will travel, before the controller will begin to search for the toolset signal; Default (0.00) Setting this too high may result in damaged bits when working with thick materials.

Toolset Speed: Speed of Z-Axis travel during the toolset operation.

System Setup

Languages: Default English

Data Initial: Restores factory system parameters.

Inner Format: Permanently deletes all files found on internal memory

Wipe Cache: Cleans up file fragmentation and system errors.

Function Configuration: Sets the operation parameters for a number of controller functions.

Probation Password: 20-digit password available from controller manufacturer.

Backup Password: Prevents customer parameters from overwriting original parameters. To cancel backup password, when display shows "Input New Password", do not enter a password, and press **OK**.

Input Port: Lists input pin/ports in use.

Output Port: Lists output pin/ports in use.

Buttons Check: Test function of keypad buttons. Screen display when each button is pressed; no display if button is not functioning. Press **OK** to quit.

Backup Data: Backs up menu parameters to either internal memory or flash disk, is not affected by system reformat.

Restore Data: Restore backup data to system.

Trial Setting: Set up trial passwords.

Auto Upgrade: Updates system firmware. Supports the (.pkg) file extension.

Operate File

Copy File – Copy files from U disk to internal memory.

Delete File – Delete files from internal memory.

View File – View files from either U disk or internal memory.

Processing Info – Shows number of files successfully processed.

Check Process Time – Shows time used for processing file(s).

Version View

Displays the current controller software information; preset by manufacturer.

Current Update Version: B11 Q13-1604

Maintenance

To ensure proper machine operation, it is recommended that both prismatic guides and rolled ball-screws on all axis are kept clean and regularly wiped down with a light machine oil lubricant (such as 3-IN-ONE oil).

Routine Checks: (Performed Daily with machine use)

1. Wipe the ball-screws and prismatic guides clean and dust free.
2. **Apply light machine oil to ball-screws and prismatic guides using a clean cloth.**
   a. Move the machine through travel limits to properly disperse lubricant.
3. Check cutting tool edges for chips and/or dullness.
4. Generally, inspect the machine for any damage.
5. Inspect for any loose components, bolts etc.
6. Verify that the Spindle and Collet Nut threads and clean of debris and undamaged.
7. Ensure that slots in collets are free of sawdust and debris.
8. Verify that the dust extractor is free of blockages.
9. Check that all electrical connectors are fitted correctly, are tight and damage free.
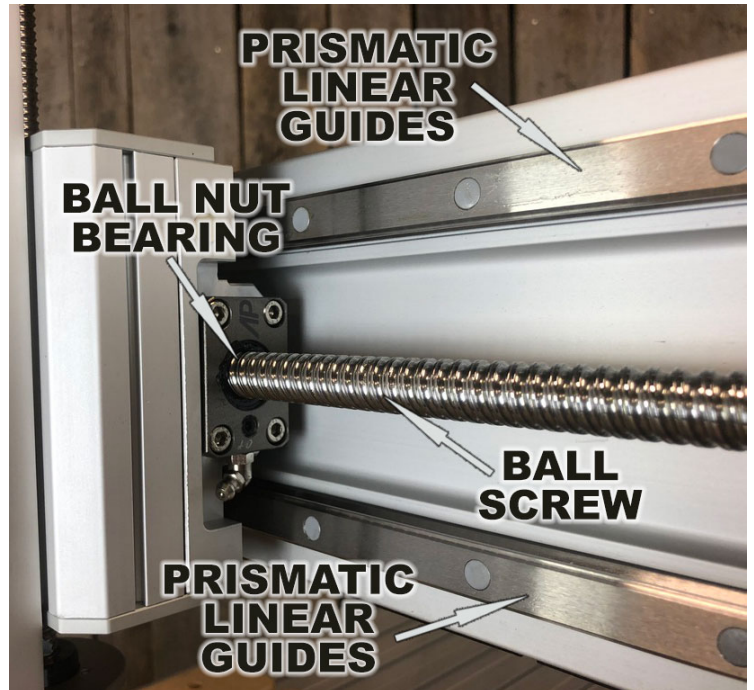
Ball Nut Bearing Service (500-Hour):

On all i2R series machines, it is recommended that the ball-nut bearing assembly be serviced every 500 machine hours.

The rolled ball screws which are found on each axis of any Axiom CNC machine feature a re-circulating bearing assembly (ball-nut bearing) which simply rides along the threaded shaft of the rolled ball screws…transferring the rotation of the stepper motors into linear movement along an axis.

Each of these bearing assemblies includes a grease fitting which can be used for lubrication during your service.

*Most any mineral-oil based grease can be used…**Lucas Oil Xtra Duty** is most often recommended*

To Lubricate this bearing assembly:

1. Position the Axis near the middle of its travel.

2. Loosen and remove the bearing assembly mounting screws (4 cap-screws per assembly)
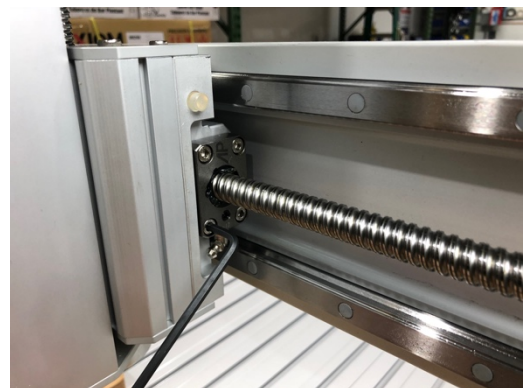
3. Slide the Axis away from the bearing assembly.

4. Attached grease gun to the grease fitting.

5. Use gentle pressure to insert grease into the assembly…being careful to not damage the seals.

It has been found to be advantageous to use the handheld controller to move the bearing assembly along the axis in both the +/- directions while the grease is being introduced. Allowing the grease to more effectively work into the assembly

6. Reposition the bearing assembly to the Axis.

7. Reinsert the mounting screws…leaving them slightly loose.

8. Use the controller to move this Axis to the far + direction…. placing the bearing assembly closest to the fixed end of the Axis

9. Continue to service the next axis or resume normal operation.

Troubleshooting:

Mechanical & Electrical Problems:

Symptom	Possible Cause	Correction *
Motor will not start: fuses blow or circuit breakers trip.	Short circuit in line cord or plug.	Inspect cord or plug for damaged insulation and shorted wires.
	Loose or separated connection.	Inspect all connections between motor and control box. *Some connections may be hidden.
	Blown Fuse	Test and Install correct fuses.
Motor attempts start but will not turn.	Jammed spindle.	Disconnect from power, try turning spindle by hand. Check reason for jamming.
	Motor faulty.	Replace spindle.
	Spindle ran without coolant.	Replace motor. Maintain coolant level.
	Incorrect voltage.	Check incoming voltage.
Motor overheats (shuts off).	Dull cutting tools.	Use sharp tools.
	No coolant in reservoir, or blockage in coolant path.	Fill the reservoir, check for obstacles in flow path.
Motor stalls, resulting in blown fuses or tripped circuit.	Motor overloaded.	Reduce load on motor.
	Short circuit in motor or loose connections.	Inspect connections on motor for loose or shorted terminals or worn insulation.
	Low voltage.	Correct low voltage conditions.
	Incorrect fuses or circuit breakers in power line.	Install correct fuses or circuit breakers.
Router cutting depth inconsistent.	Loose cutter.	Tighten cutter in spindle collet.
	Spoil boards not flat.	Check that spoil boards are flat and clean prior to fitting job.
	Excessive play in Z-axis ball screw.	Inspect and correct if needed.
Machine won't power on.	E-stop switch engaged.	Release red E-stop switch.
	No incoming power.	Check power plug connection. Check condition of power cable.
	Faulty start switches.	Inspect and replace.
	Internal breaker faulty.	Replace breaker.
Machine will not home.	Limit switches damaged, disconnected or misadjusted.	Inspect limit switches and correct problem.

Controller Function Problems:

* **WARNING:** Some corrections may require a qualified electrician.

Symptom	Possible Cause	Correction *
Controller display is blank or flickering.	Cable connections not tight.	Inspect and tighten connections between pendant and control box.
	Loose 24V connection to control board.	Check the 24V connection to the DSP control board in the control box.
	Interface damaged.	Replace damaged part.
	Controller damaged.	Replace controller.
Controller keeps restarting automatically.	Insufficient power supply.	Have qualified electrician check incoming power supply.
	Local power grid unstable.	Contact power company.
	Controller damaged.	Connect controller to computer via USB cable. If problem still occurs, replace controller.
Controller display reads "Out of Limit".	HOME operation has not been performed.	HOME all axes and retry.
	Design too large to fit within machine limits.	Check ORIGIN placement and design size.
	Incorrect table size.	Check table size parameters in Machine Setup.
Cannot set work origin on controller.	May be in Machine Coordinate State	Press MENU + 1 to revert to first working coordinate system.
	Buttons not functioning.	Enter **MENU**/System Setup/Buttons Check to verify function. If buttons not working, replace controller.
Z-axis fall is too fast during processing.	Working speed exceeds Z-axis maximum speed.	Set to safe speed in: **MENU**/Machine Setup/Max Speed Limit.
	Loose coupling, or transmission slipping.	Tighten connecting parts.
	Incorrect plunge speed in design software.	Check the plunge speeds and verify correct post processor is being used.
Z-axis depth not consistent each time same file is processed and after machine homes.	Spoil board or material not flat.	Re-mill (fly cut) spoil board/material
	Workpiece not consistent thickness	Check consistency
	Interference in Z-axis HOME process is creating a false origin.	Re-HOME Z-axis and check for proper operation.
Processed workpiece does not match file size.	Pulse equivalent incorrect.	Adjust pulse under: **MENU**/Machine Setup.
	Incorrect post processor	Verify that correct post processor was used.
	Wrong cutting tool used.	Use proper tool for process.

Symptom	Possible Cause	Correction *
Machine not stopping at HOME position	HOME detection plate beyond reach of proximity sensors.	Inspect and adjust.
	Proximity sensor wire is loose or damaged.	Check connections.
	Proximity sensor damaged.	Test and Replace.
	Poor connection at Interface board.	Inspect the appropriate connection.
	50-pin data cable is broken.	Replace cable.
Machine moves reverse direction when homing.	Incorrect HOME direction settings in Machine Setup.	Check settings.
	50-pin data cable is broken.	Replace data line.
Router head does not stop after contacting tool touch-off puck.	Poor connection of between the touch-off device and gantry connections.	Inspect connections inside touch-off device and gantry.
	Poor connection of spindle with "GND" terminal on interface board.	Restore proper connection.
Machine will not move after controller receives commands.	One axis not moving – may be poor connection.	Connect a different axis connection to this terminal to test. If it works, motor driver is okay. Check connections along corresponding drive and motor.
	All axes not moving.	First, check 50-pin cable connection to interface board. Then check power supply of motor drivers. Last check mechanical elements of axis system.
Machine moves to new position satisfactorily but does not return correctly to original position.	Mechanical elements, such as ball screws, are loose or misadjusted.	Inspect and correct as needed.
	The Stepper motors have slipped, and lost position.	Check speeds and proper lubrication on all ball screws and prismatic guides.
Abnormal operation when processing.	Program/drawing file is faulty.	Review program and reload to system. Make sure the file is completely saved before removing USB from PC.
After controller powers on, one or more axes move only one direction.	Improper connection between DSP board and motor drive.	Inspect connections.

Warranty:

Limited Warranty Coverage
Each Axiom Precision (the "Manufacturer") CNC machine ("Machine") and its components (except those listed below under limits and exclusions) is warranted against defects in material and workmanship for a period of 12 months from the date of sale. The foregoing is a limited warranty and it is the only warranty by manufacturer. Manufacturer disclaims all other warranties, express or implied, including but not limited to all warranties of merchantability and fitness for a particular purpose.

Repair or Replacement only
Manufacturer's liability under this agreement shall be limited to repairing or replacing, at the discretion of manufacturer, parts or components. Shipment for items replaced under warranty is free, but the shipment method is at the discretion of Axiom Tool Group, Inc. In general delivery for domestic customers will be by UPS ground service or USPS. If overnight or express delivery is requested, additional fees will apply.

Direct phone support is part of the equation that allows us to provide high value at low cost. You must be comfortable with general electrical and mechanical repair concepts, including the appropriate safety procedures, before working on your machine. If you do not have the required skills, you may need to find someone locally to assist you. We do not have factory technicians to send to your facility.

Limits and Exclusions of Warranty
Except as provided above, buyer agrees that all warranties express or implied, as to any matter whatsoever, including but not limited to warranties of merchantability and fitness for a particular purpose are excluded. Components subject to wear during normal use and over time such as paint, labels or decals, finish and condition, seals, bearings, spoil boards, flex cabling, etc., are excluded from this warranty. Axiom-specified maintenance procedures must be adhered to in order to maintain this warranty. This warranty is void if the machine is subjected to mishandling, misuse, neglect, accident, improper installation, improper maintenance, or improper operation or application, or if the machine was improperly repaired or serviced. Warranty of general machine tolerances is void if the machine is disassembled or altered by customer.

Without limiting the generality of any of the exclusions or limitations described in other paragraphs, manufacturer's warranty does not include any warranty that the machine or components will meet buyer's production specifications or other requirements or that operation of the machine and/or components will be uninterrupted or error-free. Manufacturer assumes no responsibility with respect to the use of the machine and components by buyer, and manufacturer shall not incur any liability to buyer for any failure in design, production, operation, performance or otherwise of the machine or components other than repair or replacement of same as set forth in the limited warranty above. Manufacturer is not responsible for any damage to parts, machines, business premises or other property of buyer, or for any other incidental or consequential damages that may be caused by a malfunction of the machine or components.

Contact Us:

For software related inquiries, training videos, software updates, free projects and more...visit Vectric Ltd directly at:

http://support.vectric.com

Visit us online at:

www.axiomprecision.com

Need Assistance? The Axiom Precision CNC Support Team can be reached at:

support@axiomtoolgroup.com

For machine and accessory orders, the Axiom Precision Sales Team can be reached at:

sales@axiomtoolgroup.com

Axiom Tool Group, Inc.

270 Broad St

Westerville, Ohio 43081

1-844-642-4902

BUILD BETTER STUFF.™