

2017

# Laguna Tools IQ PRO

User guide for IQ Machines equipped with  
the RichAuto B57E

Basic operations, , Quick Start, Mutli-Tool Programs,  
Maintenance and Troubleshooting



## TABLE OF CONTENTS

<b>1</b>	<b>Introduction</b> .....	<b>3</b>
<b>2</b>	<b>Document Key</b> .....	<b>3</b>
<b>3</b>	<b>Basic Operations</b> .....	<b>3</b>
	3.1 View of Machine .....	3
	3.2 Turning ON the machine .....	4
	3.3 Homing.....	4
	3.4 Basic Button Functions.....	5
	3.5 Changing Tools .....	6
<b>4</b>	<b>Quick Start Section</b> .....	<b>6</b>
<b>5</b>	<b>Multiple tool program</b> .....	<b>10</b>
	5.1 Setting tool offsets.....	10
	5.2 Setting XY work origin.....	11
	5.3 Setting Z work origin .....	11
	5.4 Run program.....	11
<b>6</b>	<b>Understanding Work Coordinate systems</b> .....	<b>12</b>
<b>7</b>	<b>Tool Cone Setup</b> .....	<b>12</b>
<b>8</b>	<b>Maintenance and Troubleshooting</b> .....	<b>13</b>
	8.1 Cleaning and Lubricating the Mechanics .....	13
	8.2 Resetting Tool locations.....	13
	8.2 Tool touch off - ToolSettingZ adjustment .....	14

## 1 INTRODUCTION

This document is intended to provide general information on how to operate the RichAuto B57E controller. Refer to the quick start section for accelerated start up.

## 2 DOCUMENT KEY

**Bold text - Hard Button click**

*Italicized text - List Selection*

Red text - Warns the user of potential hazards

Blue text - Additional information

## 3 BASIC OPERATIONS

### 3.1 VIEW OF MACHINE

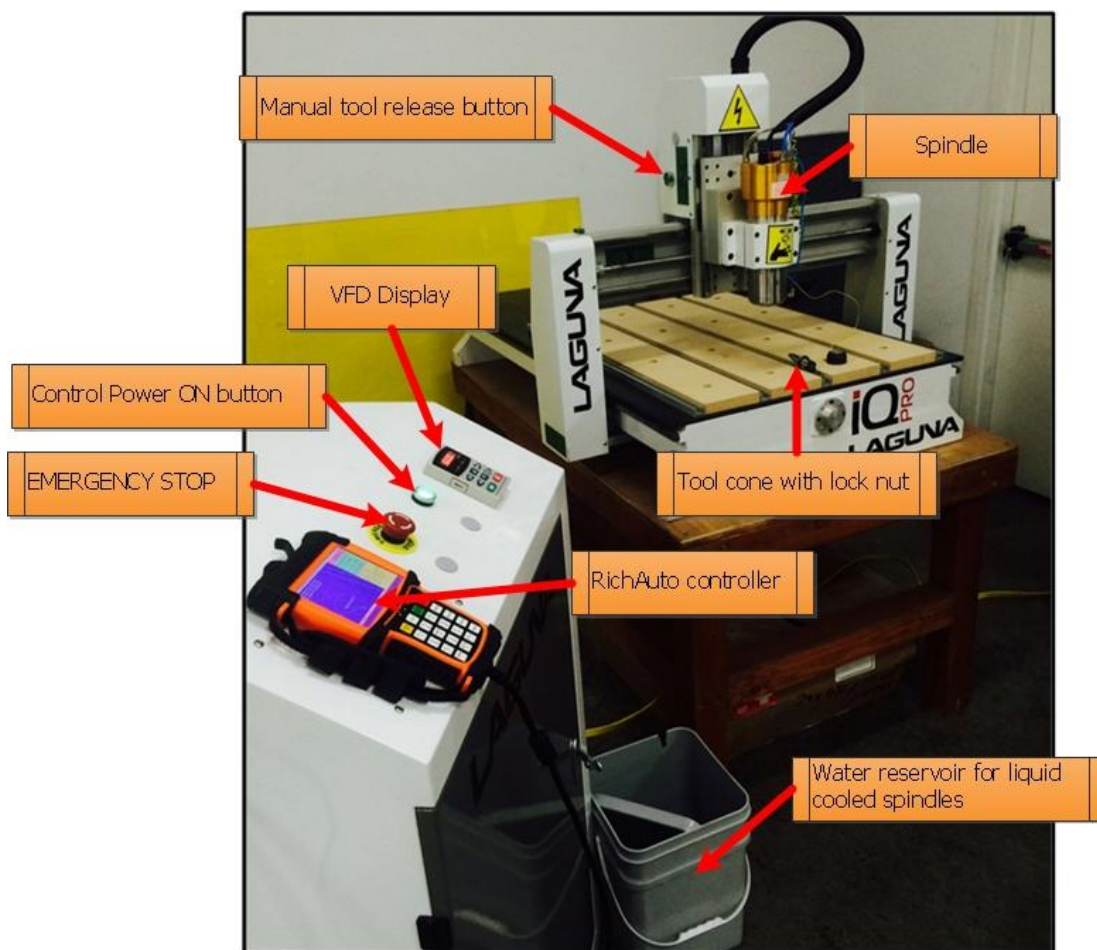


Figure 1. Front view of machine

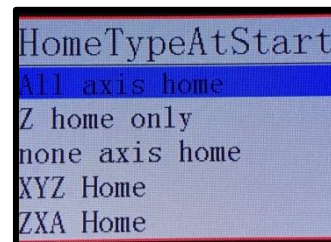
### 3.2 TURNING ON THE MACHINE

1. Turn the rotary switch located on the electrical cabinet to the ON position.
2. Press the Control Power ON/OFF button located on the control panel.



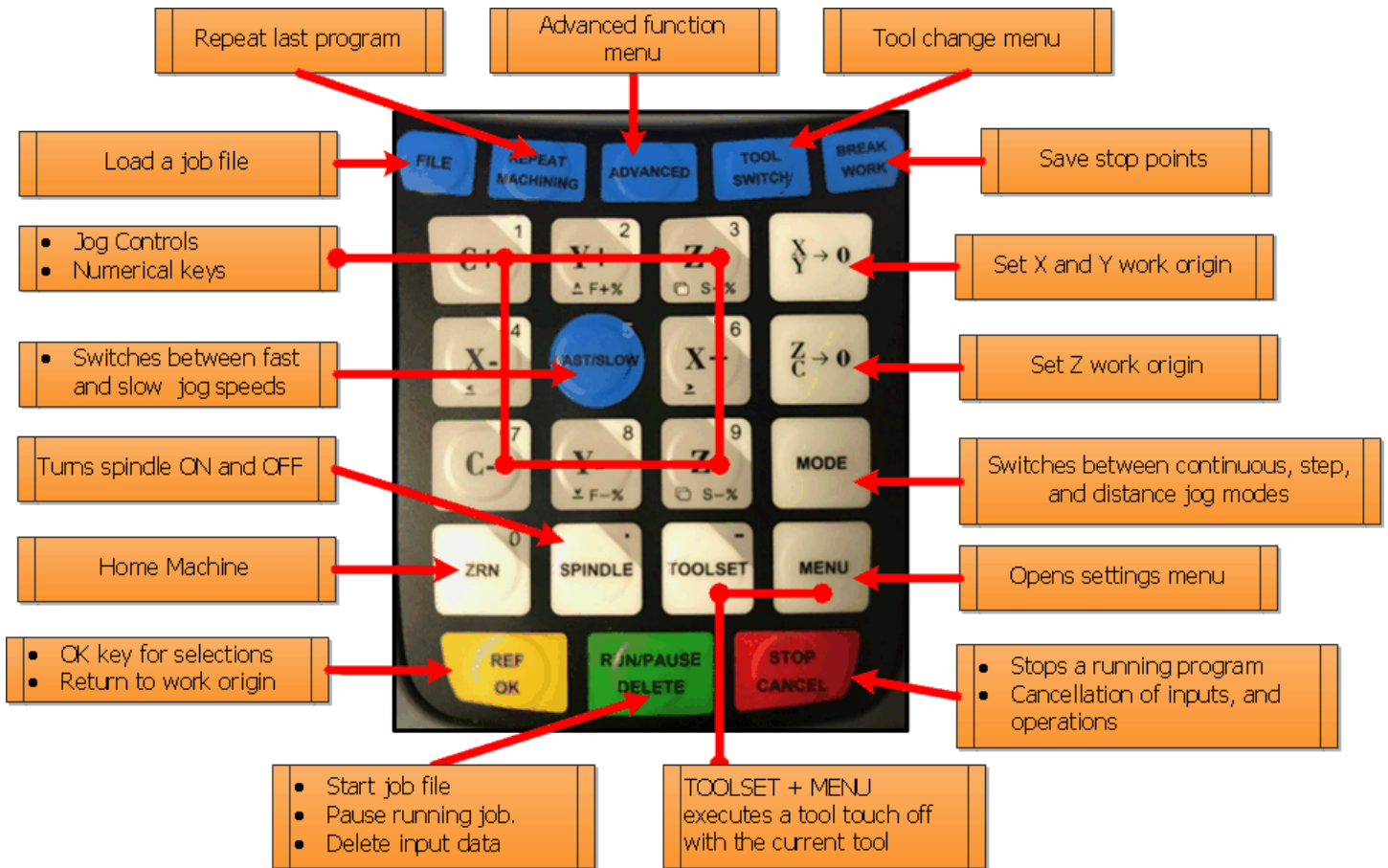
### 3.3 HOMING

- When the B57E controller is first powered on, the user will be prompted with the HomeTypeAtStart window.
- The machine needs to be homed each time the machine is powered.
  1. This resets your machine coordinates origin, relative to the home switches and flags.
  2. The tool locations are relative to the machine origin.
- The default selection is **All axis home**.
  1. Press **REF/OK** to begin homing all axes, or select another option.



### 3.4 BASIC BUTTON FUNCTIONS

- The RichAuto motion control system uses 1-button and 2-button combination functions.
- Only the most used button functions are covered in this section.



### 3.5 CHANGING TOOLS

1. A tool can be inserted or removed two ways.
  - Manually - Using the tool release button on the side of tool head (figure 1).
  - Automatically - Using the Tool Switch button.
    - Press **ToolSwitch** button
    - Use up and down Y buttons to highlight tool selection.
    - Press **REF/OK** button.



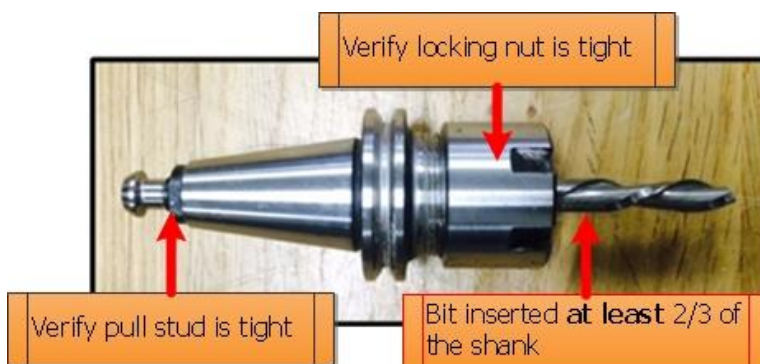
## 4 QUICK START SECTION

This section is a work flow from; turning the machine on, to starting your program.

**This walkthrough assumes only TOOL 1 is used.**

1. Power on machine (Section 3.1).
2. Home machine (Section 3.2).
3. Verify that the machine is connected to an air supply.

- a. The tool changer needs **6-7bar or 87psi-100psi**.
  - b. Use the pressure regulator on the back of the machine to adjust air pressure. As seen to the right.
4. Plug in the water pump and verify water is flowing through the spindle.
    - a. Only applicable to machines equipped with liquid cooled spindles.
  5. Setup a tool cone (figure 2) with the router bit you intend to use.
    - a. Use the manual tool release button located on the tool head to load the tool cone into the spindle.
    - b. It is pertinent that the spindle releases and engages the tool cone repeatedly. This quick check will prevent an error condition.



**Figure 2. Tool cone with locking nut, collet, and router bit**

6. Transfer your g-code program onto a USB and transfer into the controller's internal memory.

When running a program directly from a USB, memory transfer is less reliable. It is recommended to store the program in the controller's internal memory.

- a. **MENU** --> *Menu Function User Interface* --> **REF/OK**
  - b. *Machine Configuration* --> **REF/OK**
  - c. Scroll down to *Operate File* then Press **REF/OK**
  - d. Select *Copy File* press **REF/OK**
  - e. Select the *UDisk File* you want to copy press **REF/OK**
  - f. Press **STOP/CANCEL** a few times to return to the controller home screen.
7. Load Program into viewer
    - a. Press **FILE** --> *Internal File* --> **REF/OK** --> select your program --> **REF/OK**
  8. Mount work piece to the table top.
    - a. This can be done with the supplied table clamps (Figure 3).

- b. There exist a multitude of ways to secure your work piece. Find what method works best for your application before proceeding.
- c. Always consider clearance between the router bit and table clamps. When the machine changes tools it will need to travel to the back of the machine during program execution.

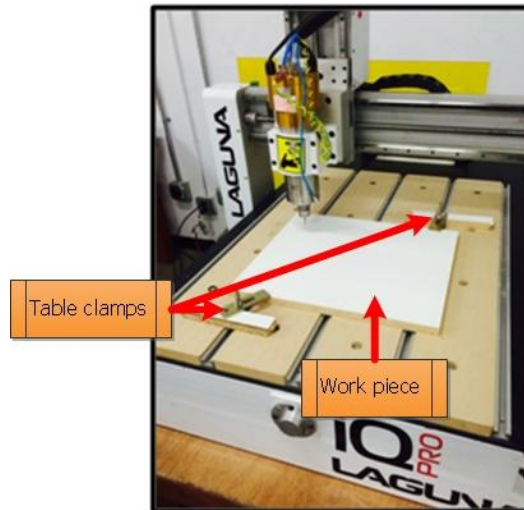


Figure 3. Simple work piece setup

9. Verify coordinate system (figure 4).
  - a. Workpiece 1 should be selected.
    - i. To change to workpiece 1, Hold **MENU** --> press **C+ /1** button
10. Verify which tool cone is currently in the spindle and compare to the **CurSpindl** parameter on the controller run screen (figure 4).
  - a. If current tool number is not tool 1, then switch tools using tool switch function (3.4).

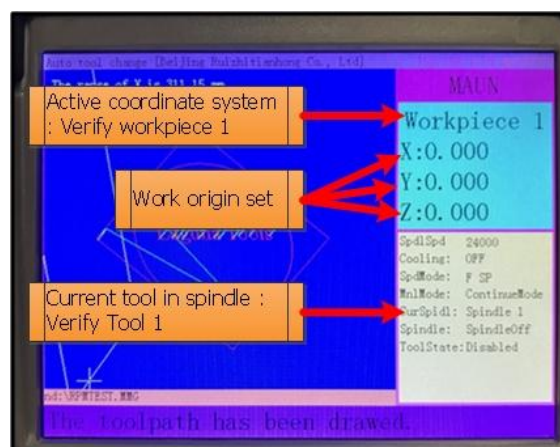


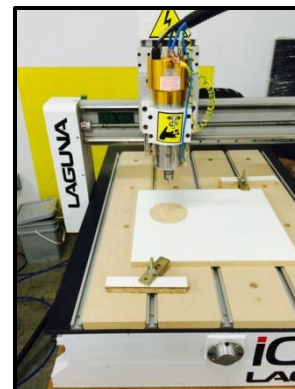
Figure 4. Control screen after loading program, and setting work origin.



11. Jog the spindle to the XY origin point. This is determined when the CNC program is created.
  - a. If you are using Vectric's V-Carve software, it is called the XY datum position.
  - b. Once the spindle is in position press the **XY-->0** button to set the XY origin point.
  - c. On the controller, you will see the X and Y axis coordinates go to zero (figure 4).

➤ **For multi-tool programs JUMP to section 5**

12. Set the Z origin point [Manually]
  - a. Set jog speed to low and carefully move the spindle down towards the top of your material until the desired Z zero position is obtained.
    - i. It may be helpful to use a sheet of paper by sliding it back and forth under the bit, while jogging the z-axis down. Once the paper is snagged by the bit you know your within a paper thickness (0.1mm) of your material.
  - b. Press the **ZC-->0** button.
  - c. On the controller, you will see the Z-axis coordinate go to zero (figure 4).
13. Press **REF/OK** button.
  - a. This will retract the spindle to a safe starting location.
14. Put on your **SAFETY GLASSES** and run the program!
  - a. In case of an emergency or machine failure you can hit the E-STOP button on the control cabinet. [In most cases the stop/cancel button on the hand held controller is a preferred stopping method.]
  - b. Press the **RUN/PAUSE/DELETE** button to start the program.
  - c. Accept the default runtime parameters by pressing **REF/OK**
15. While the program is running:
  - a. Feed rate can be adjusted by pressing the **Y+** or **Y-** buttons.
    - i. Y+ will increase the feed rate by 10%.
    - ii. Y- will decrease the speed by 10%
    - iii. The max speed is determined by the program file.
16. Program End.
  - a. At the end of a program the spindle will STOP and the Z-axis retract to a safe clearance height. As seen in the photo to the right.



## 5 MULTIPLE TOOL PROGRAM

There are preliminary steps that need to be done before running a multi-tool program.

- Prepare the machine(Section 4, steps 1-11)
- Prepare all tool cones for the job, and place into tool corresponding holders.
- Run procedure to calculate tool offsets(5.1)
- Set XY work origin(5.2).
- Set Z work origin(5.3).
- Run program(Section 4, steps 14-16)

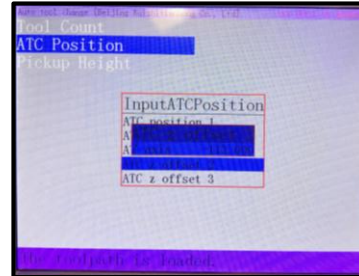
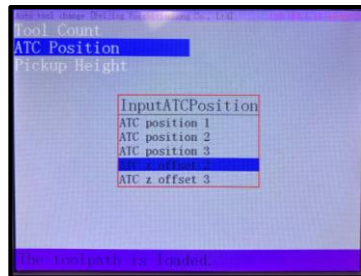
### 5.1 SETTING TOOL OFFSETS

- Place the designated tools into the corresponding tool holders on the machine, determined at the time of program creation.
- Change to workpiece 1 and tool 1(see section 3.5).
  - Verify **Workpiece 1**
  - Verify **CureSpindl: Spindle 1**, and tool 1 is in the spindle.
  - Manually jog the spindle into a safe location by the tool rack.
    - The machine will return to the starting position after changing tools.



- Press and hold the **MENU** button --> press **TOOLSET** button --> release both buttons.
  - The spindle will move over the tool touch off switch from its current position.
  - Rapid to a preset Z value(ToolSettingZ).
  - At a slower feed rate, it will lower until the switch is trigger, then retract.
- Once Tool 1 has touched off, Switch to Tool 2, using the Tool Switch button.

- Press and hold the **MENU** button --> press **TOOLSET** button -->release both buttons.
- Once Tool 2 has touched off, Switch to Tool 3, using the Tool Switch button.
- Press and hold the **MENU** button --> press **TOOLSET** button -->release both buttons.
- Switch back to Tool 1, using the Tool Switch button.
- Offsets are now calculated.



## 5.2 SETTING XY WORK ORIGIN

- Jog the spindle to the XY origin point. This is determined when the CNC program is created.
  - If you are using Vectric's V-Carve software, it is called the XY datum position.
  - Once the spindle is in position press the **XY-->0** button to set the XY origin point.
  - On the controller, you will see the X and Y axis coordinates go to zero (figure 4).

## 5.3 SETTING Z WORK ORIGIN

- Set the Z origin point [Manually]
  - a. Set jog speed to low and carefully move the spindle down towards the top of your material until the desired Z zero position is obtained.
    - i. It may be helpful to use a sheet of paper by sliding it back and forth under the router bit, while jogging the z-axis down. Once the paper is snagged by the bit you know your within a paper thickness(0.1mm).
  - b. Press the **ZC-->0** button.
  - c. On the controller, you will see the Z-axis coordinate go to zero (figure 4).

## 5.4 RUN PROGRAM

1. Transfer your g-code program onto a USB and transfer into the controller's internal memory.
 

When running a program directly from a USB, memory transfer is less reliable. It is recommended to store the program in the controller's internal memory.

  - a. **MENU** --> *Menu Function User Interface* --> **REF/OK**
  - b. *Machine Configuration* --> **REF/OK**
  - c. Scroll down to *Operate File* then Press **REF/OK**

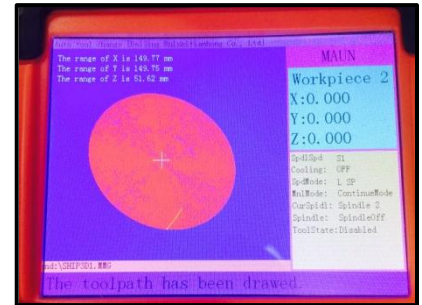
- d. Select *Copy File* press **REF/OK**
- e. Select the *UDisk File* you want to copy press **REF/OK**
- f. Press **STOP/CANCEL** a few times to return to the controller home screen.

## 2. Load Program into viewer

Press **FILE** --> *Internal File* --> **REF/OK** --> select your program --> **REF/OK**

## 6 UNDERSTANDING WORK COORDINATE SYSTEMS

- Machine Coordinates(Workpiece 0)
  - The origin is based off the home switches.
  - **Tool positions and TTO switch location are set relative to machine coordinates.**
  - Press **MENU + ZRN/0** to switch to workpiece 0.
- Hold down the **MENU + 1-9 buttons** to move between work coordinate systems.
- In the picture to the right the active work coordinate system is "Workpiece 2"
- Transitioning between Workpieces will in turn, transition between X, Y, and Z offsets made in each Workpiece.



## 7 TOOL CONE SETUP

1. Select a router bit and its corresponding collet.

**Note.** Collets & spindle nut must be cleaned regularly. Ensure that the slots in the collets are free of sawdust.

2. Press the collet into the spindle nut until it snaps into place.



3. Thread on the nut and collet by hand, onto the tool cone.

4. Insert the router bit into the collet.
5. Use the provided wrenches to tighten the spindle nut.

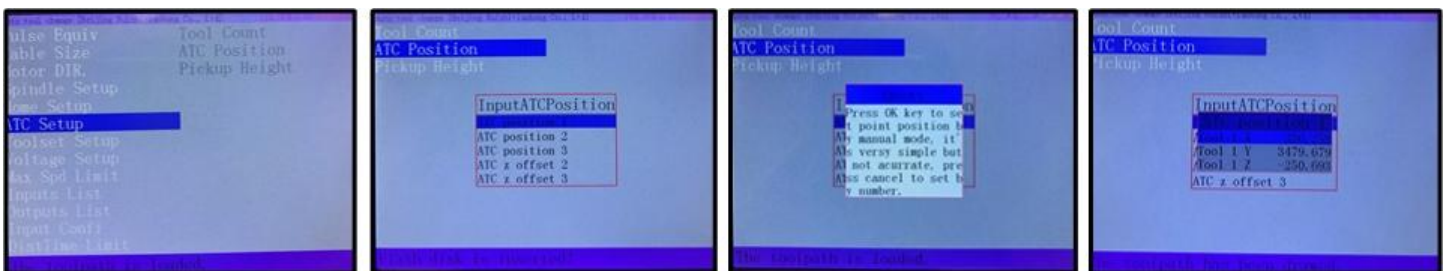
## 8 MAINTENANCE AND TROUBLESHOOTING

### 8.1 CLEANING AND LUBRICATING THE MECHANICS

- The IQ pro uses Linear guide rails, ball screws, and ball bearings. These need to be lubricated periodically.
  - Scheduled maintenance : Once a week, if the machine is used daily.
  - A visual inspection is always recommended. If the rails are dry to the touch, then get an oil can and a rag.
  - 30WT motor oil or white lithium grease is recommended.

### 8.2 RESETTING TOOL LOCATIONS

- Home the machine.
  - This will reset the controller's machine coordinates, relative to the position of the home switches and flags.
  - **Verify the controller is in machine coordinates (workpiece 0)**
    - **If not, press **MENU+ZRN/0** to change to machine coordinates.**
- Put an EMPTY tool cone into the spindle using the manual tool release button.
- Using the hand held controller's jog control, carefully guide the cone into the tool rack.
  - Step jog Mode can be used for smaller and precise increments.
- Record the X, Y, and Z position. Verify that these are in machine coordinates.
- Carefully jog the tool out of the tool rack.
- Navigate to the ATC stored locations, by following the below steps.
- **MENU --> Menu Function User Interface --> REF/OK**
- **Machine Setup --> REF/OK**
- Scroll down to **ATC setup--> REF/OK**
- Scroll up to **ATC position --> REF/OK**
- Press **STOP/CANCEL** to clear the warning.



- Select a ATC position.

- Select the X, Y, or Z position and press the **RUN/PAUSE/DELETE** button to edit.
- Use the numeric buttons to enter in the new tool position and press **ORIGIN/OK** to save.
- Repeat this process for each X, Y, and Z coordinate for tools 1, 2 and 3.
- Verify your work.
  - Test the tool change without a tool cone.
  - Test the tool change with a tool cone without a router bit.

## 8.2 TOOL TOUCH OFF - TOOLSETTINGZ ADJUSTMENT

- If the ToolSettingZ parameter needs to be adjusted follow this path
  - **MENU** --> *Menu function user interface* --> machine setup --> Toolset setup --> C.A.D. position --> select with **REF/OK** button.
  - Select *InPosition*, by pressing **RUN/PAUSE/DELETE** to change it.
  - Press **REF/OK**
  - Press **STOP/CANCEL** to set value manually.
- Change ToolSettingZ to desired value.

