

2018

# Laguna Tools Smartshop M

User guide for Smartshop machines  
equipped with the RichAuto B57E

Basic operations, Quick Start, Mutli-Tool Programs, Creating a  
G-code file with V-carve, Maintenance and Troubleshooting,  
Installation Outline, Default Settings



## 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 .....	5
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.....	12
<b>6</b>	<b>Understanding Work Coordinate systems .....</b>	<b>13</b>
<b>7</b>	<b>Tool Cone Setup.....</b>	<b>13</b>
<b>8</b>	<b>Maintenance and Troubleshooting .....</b>	<b>14</b>
8.1	Cleaning and Lubricating the Mechanics .....	14
8.2	Resetting Tool locations.....	14
8.2	Tool touch off - ToolSettingZ adjustment .....	15
<b>9</b>	<b>Creating a G-Code file.....</b>	<b>16</b>
<b>10</b>	<b>Appendix.....</b>	<b>21</b>
10.1	Hiteco Spindle Pneumatic Specifications .....	21
10.2	Installation Outline .....	22

## 1 INTRODUCTION

This document is intended to provide general information on how to operate the Smartshop M machine, equipped with a RichAuto B57E controller. For accelerated start up, refer to quick start section.

## 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

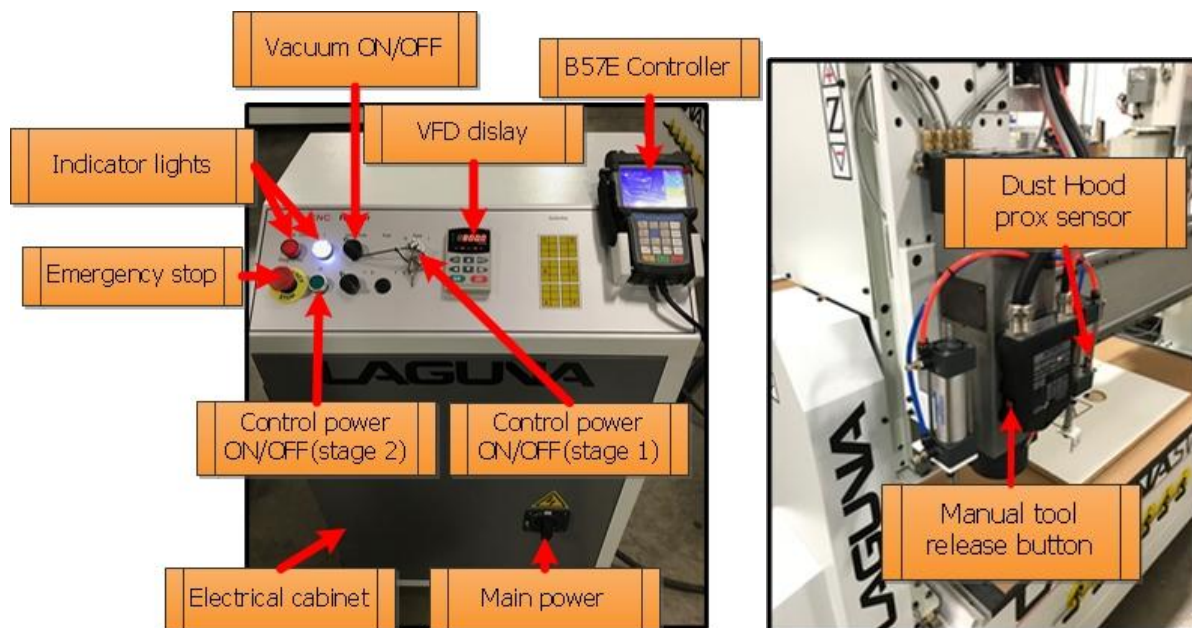
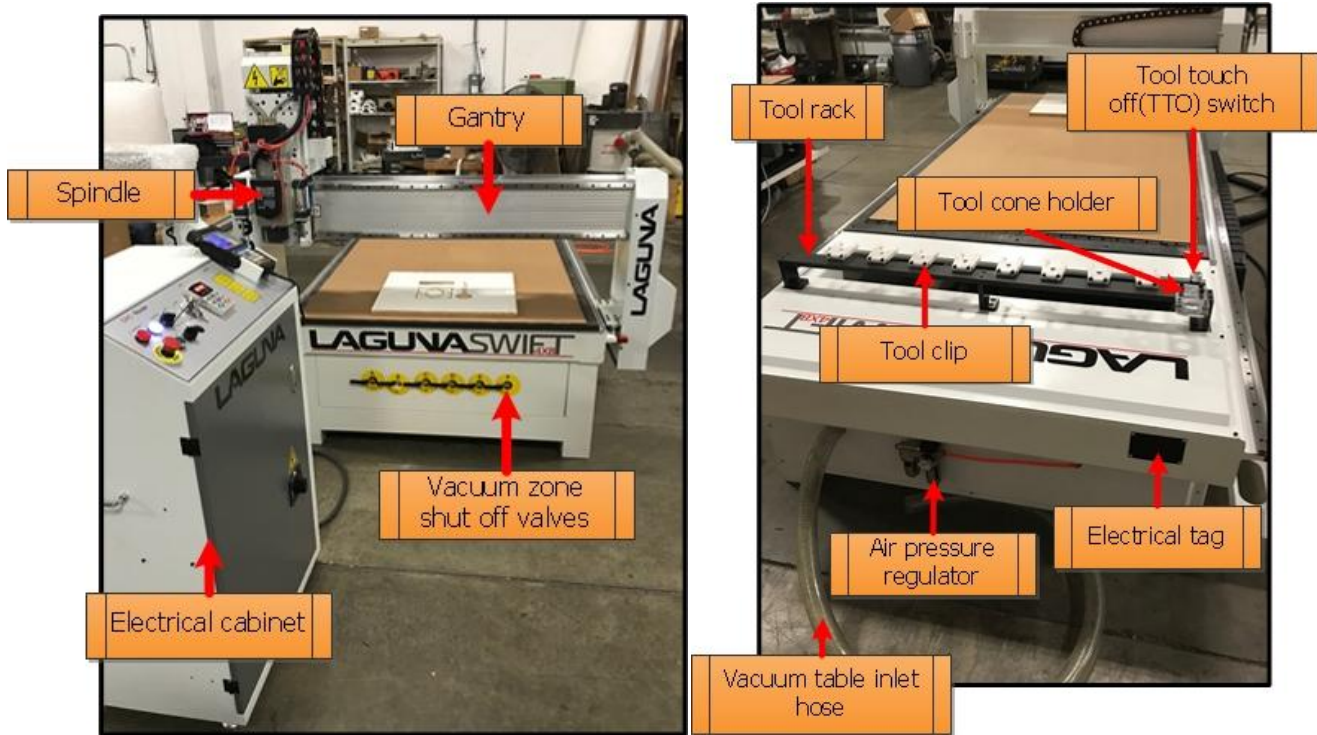


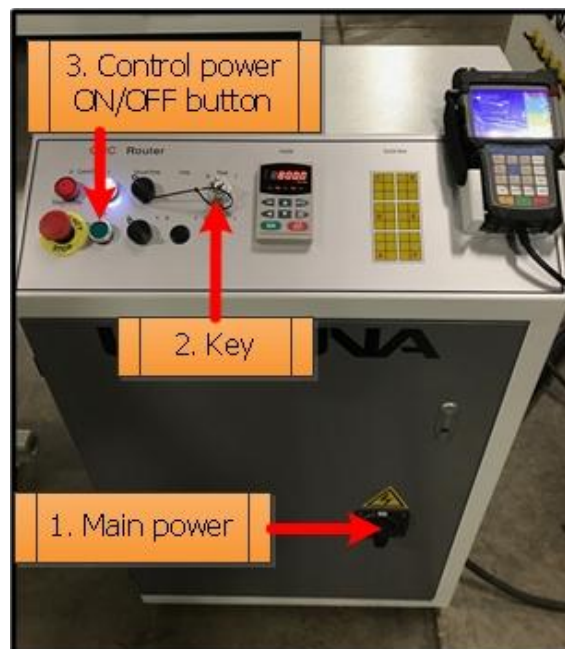
Figure 1. Front of electrical cabinet, and view of spindle



**Figure 2. Front and rear view of machine**

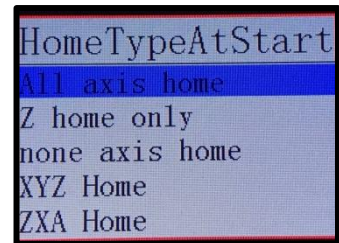
### 3.2 TURNING ON THE MACHINE

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



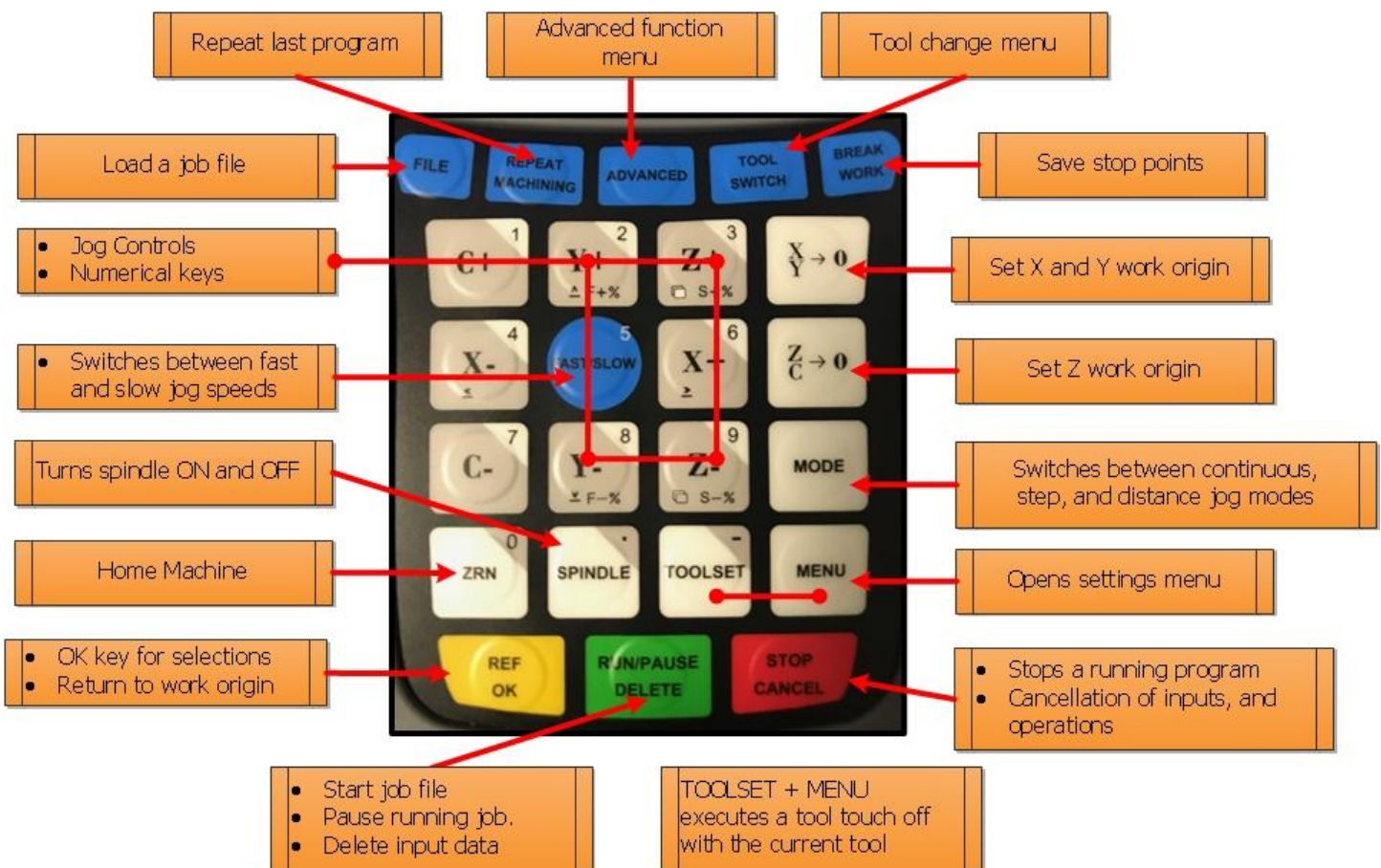
### 3.3 HOMING

- 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.
  - This resets your machine coordinates origin, relative to the home switches and flags.
  - The tool locations are relative to the machine origin.
- The default selection is **All axis home**.
  - 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 the spindle (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



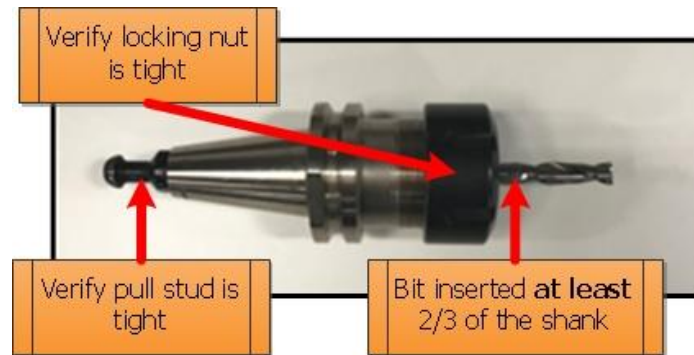
## 4 QUICK START SECTION

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 **6-6.5bar or 87psi-95psi**.
  - b. Use the pressure regulator (figure 2) on the back of the machine to adjust air pressure.
4. Set up a tool cone with the router bit you intend to use (Section 7).
    - a. Use the manual tool release button located on the tool head to load the tool cone into the spindle.
    - b. It is important that the spindle releases and engages the tool cone repeatedly. This quick check will prevent an error condition.



**Figure 3. Tool cone with locking nut, collet, and router bit**

5. 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.
6. Load Program into viewer
    - a. Press **FILE** --> *Internal File* --> **REF/OK** --> select your program --> **REF/OK**
  7. Mount workpiece 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 workpiece.  
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.

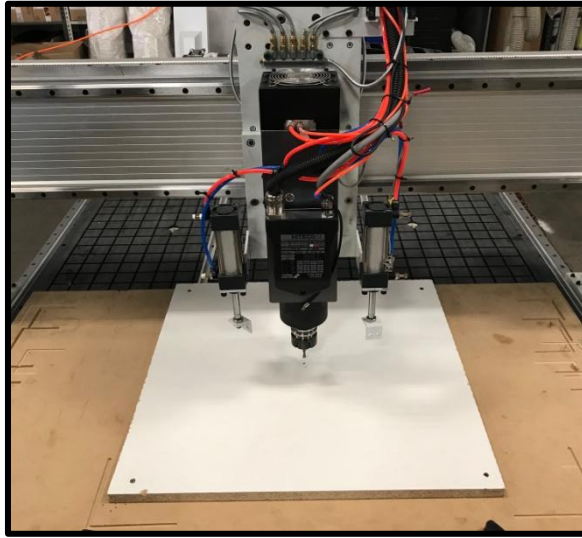


Figure 4. Simple workpiece setup

8. Verify coordinate system (figure 4).
  - a. Workpiece 1 should be selected.
    - i. To change to workpiece 1, Hold **MENU** --> press **C+/1 button**, then release both buttons
9. 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).

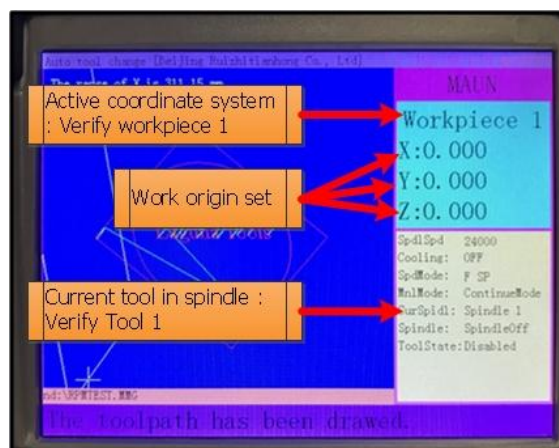


Figure 5. Control screen after loading program, and setting work origin.

10. 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 mutli-tool programs JUMP to section 5**

11. 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 slide a sheet of paper back and forth under the bit, while jogging the z-axis down. Once the paper is snagged by the bit you know you are 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).
12. Press **REF/OK** button.
  - a. This will retract the spindle to a safe starting location.
13. 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 handheld 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**
14. 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.
15. Program End.
  - a. At the end of a program the spindle will STOP and the Z-axis retract to a safe clearance height.

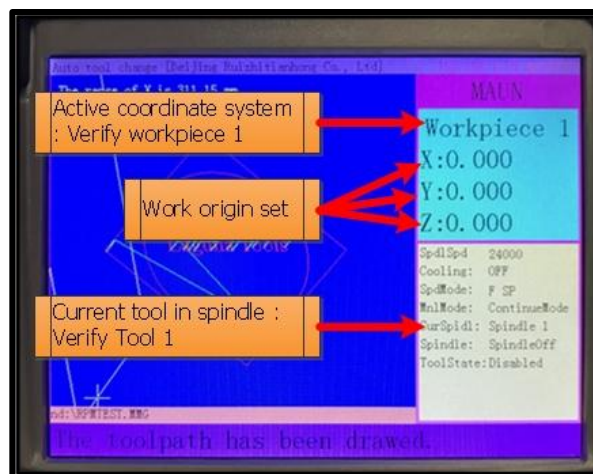
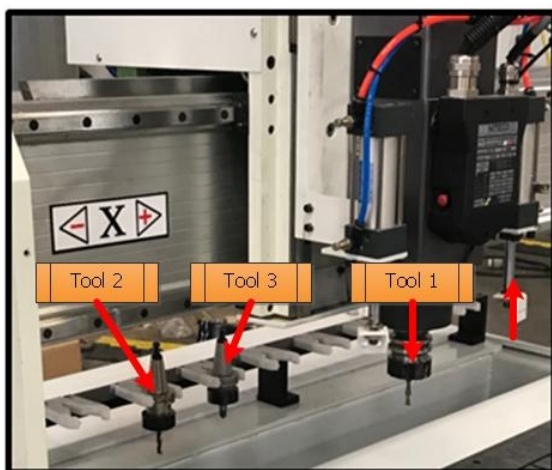
## 5 MULTIPLE TOOL PROGRAM

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

- Prepare the machine (Section 4, steps 1-10).
- 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 with tool 1 (5.3).
- Run program (Section 4, steps 13-15).

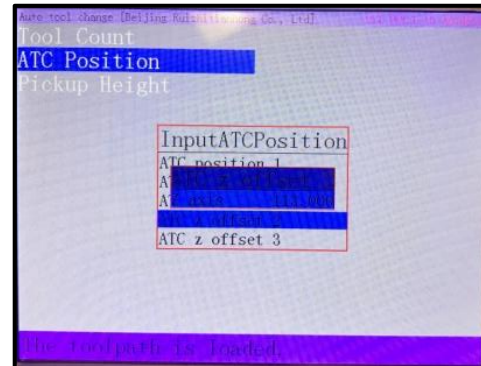
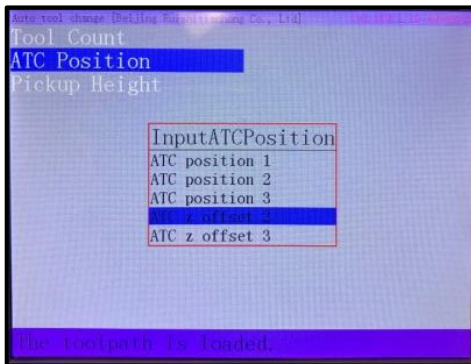
### 5.1 SETTING TOOL OFFSETS

- 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 **CureSpindle: 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.



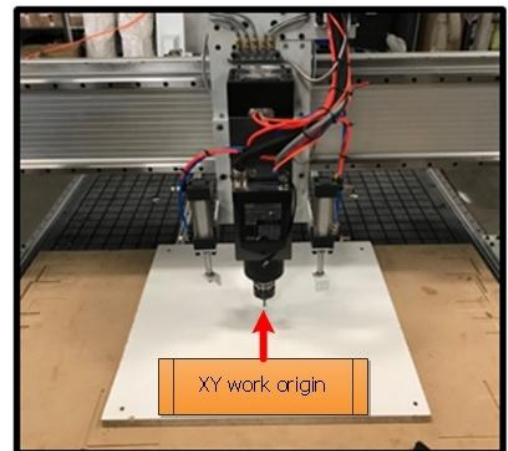
- Press and hold the **MENU** button --> press **TOOLSET** button --> release both buttons.
  - The machine will move the spindle over the tool touch off switch.
  - The Z-axis will rapid to a preset Z value (ToolSettingZ).
  - At a slower feed rate, it will lower until the switch is triggered, then 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.



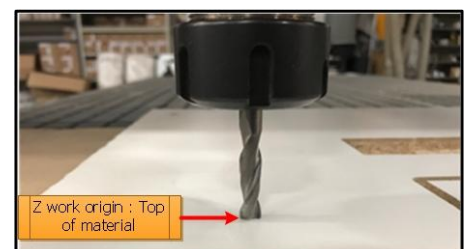
## 5.2 SETTING XY WORK ORIGIN

- 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)
  - 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.
    - It may be helpful to slide a piece of paper back and forth under the router bit, while jogging the z-axis down. Once the paper is snagged by the bit you know you are 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.

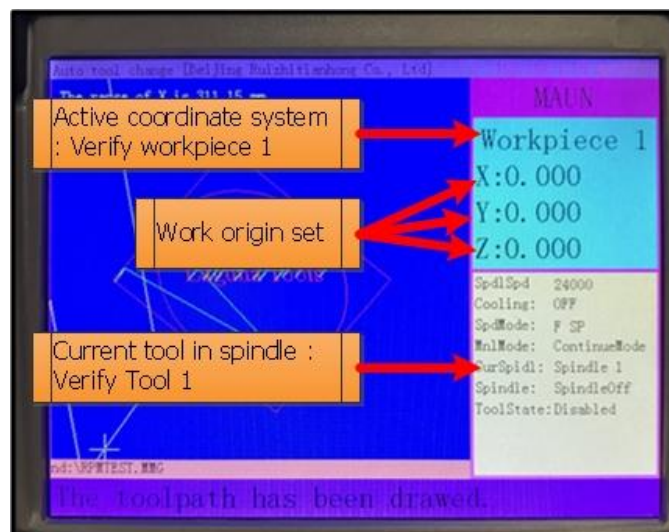
## 5.4 RUN PROGRAM

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.

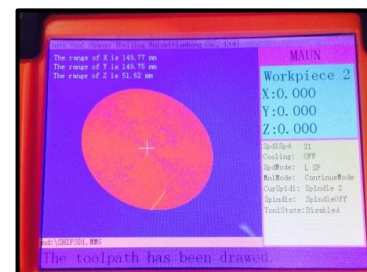
2. Load Program into viewer

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



## 6 UNDERSTANDING WORK COORDINATE SYSTEMS

- Machine Coordinates(Workpiece 0)
  - Machine origin is established using 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

- Select a router bit and its corresponding collet.
 

**Note.** The collets & spindle nut must be cleaned regularly. Ensure that the slots in the collets are free of sawdust.
- Press the collet into the spindle nut until it snaps into place.



- Thread on the nut and collet by hand, onto the tool cone.
- Insert the router bit into the collet.
- Tighten the spindle nut. Use the provided wrench and tool cone holder on the back of the machine frame (figure 2).



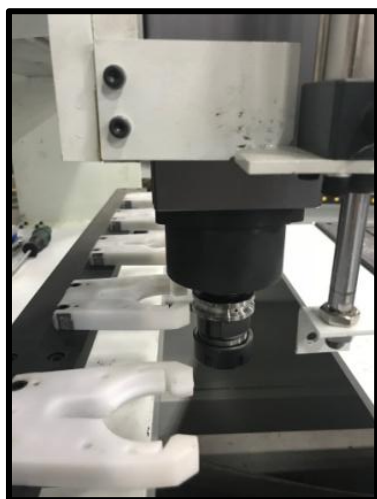
## 8 MAINTENANCE AND TROUBLESHOOTING

### 8.1 CLEANING AND LUBRICATING THE MECHANICS

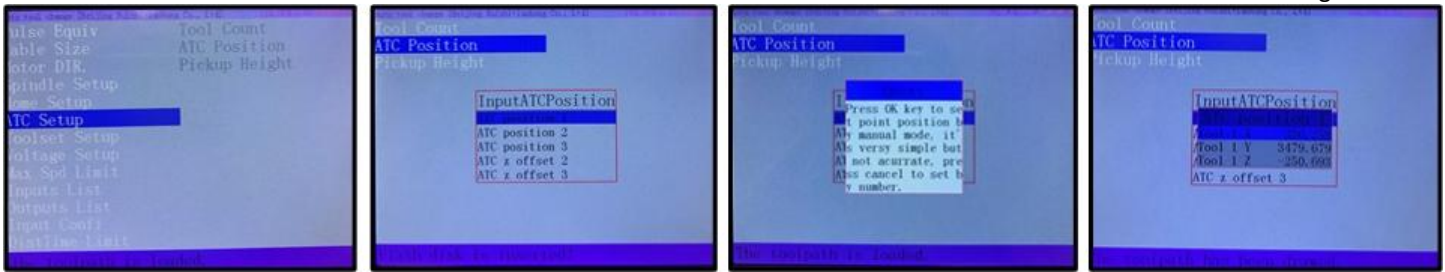
- The SmartshopM uses linear guide rails, ball screws, and ball bearings. These need to be lubricated periodically.
  - Scheduled maintenance: Once a week, if the machine is used daily.
  - A visual inspection is always recommended. If the rails are dry to the touch, then use an oil can and a rag to lubricate them.
  - 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 handheld controller's jog control, carefully guide the cone into the tool rack.



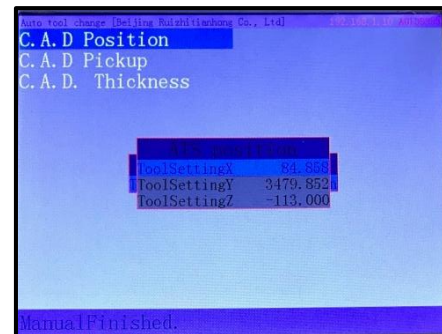
- Record the X, Y, and Z position. Verify that these are in machine coordinates.
- Carefully jog the tool out of the tool rack.
- Navigate to the ATC stored locations, by following the below steps.
- **MENU** --> *Menu Function User Interface* --> **REF/OK**
- *Machine Setup* --> **REF/OK**
- Scroll down to *ATC setup*--> **REF/OK**
- Scroll up to *ATC position* --> **REF/OK**
- Press **STOP/CANCEL** to clear the warning.



- Select an 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.

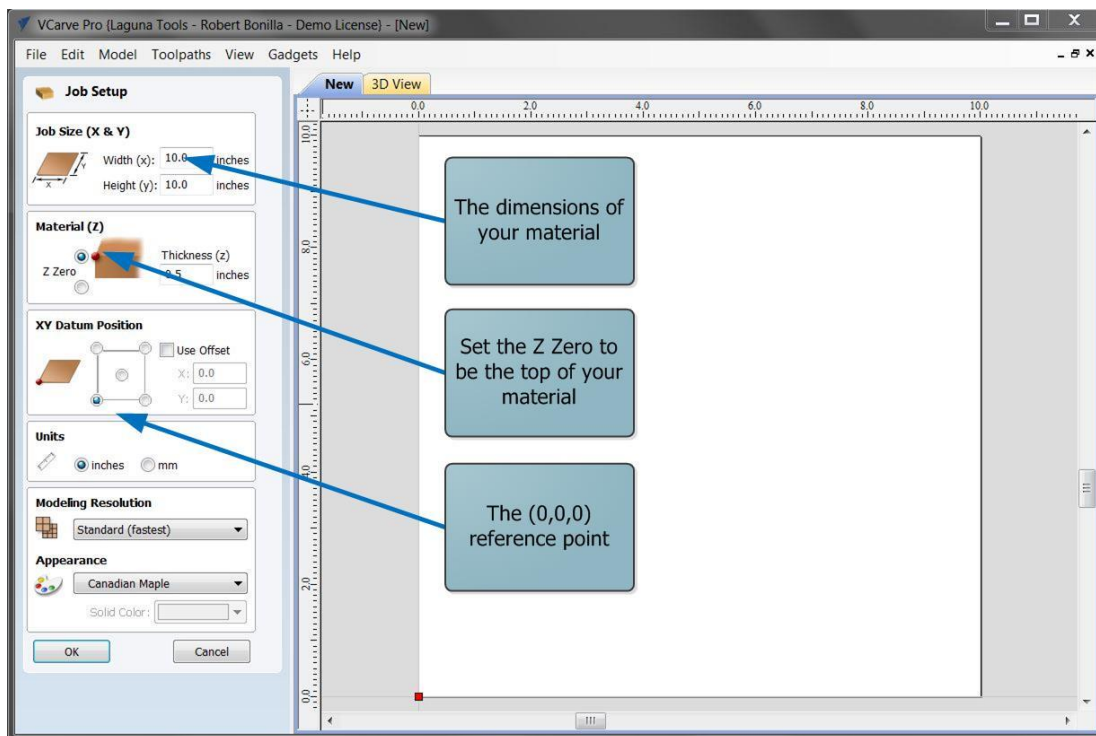
## 8.2 TOOL TOUCH OFF - TOOLSETTINGZ ADJUSTMENT

- 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.

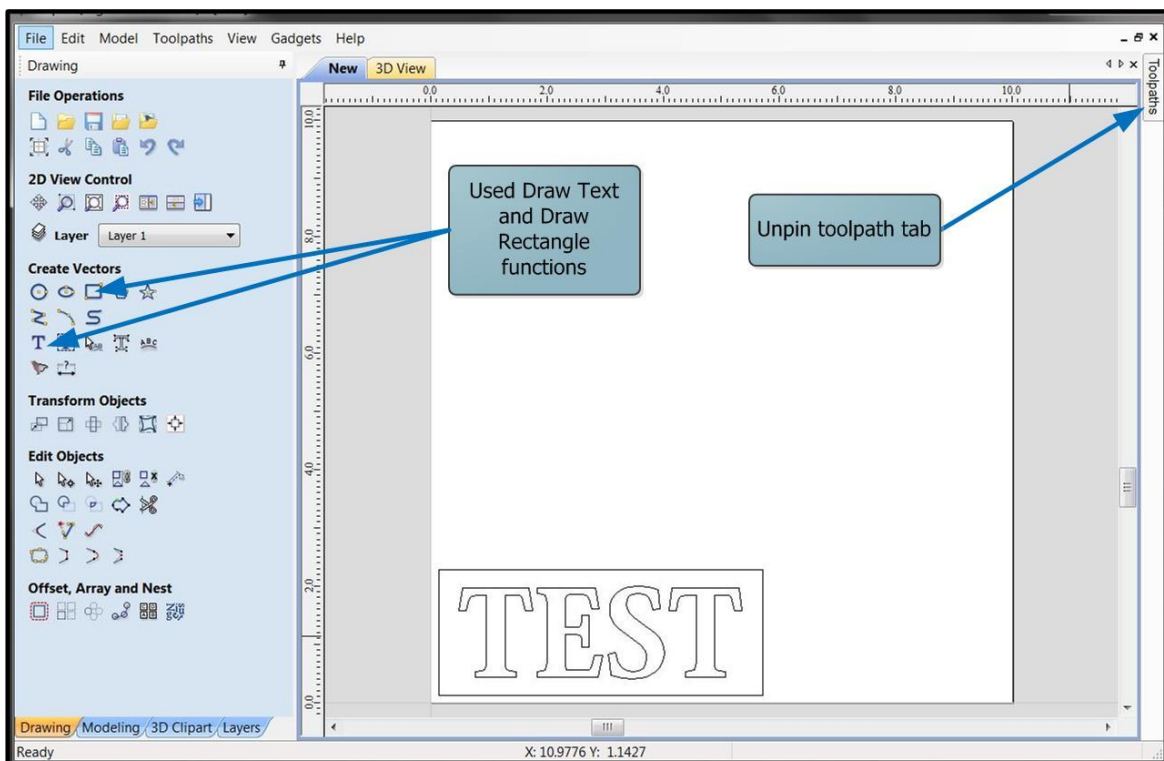


## 9 CREATING A G-CODE FILE

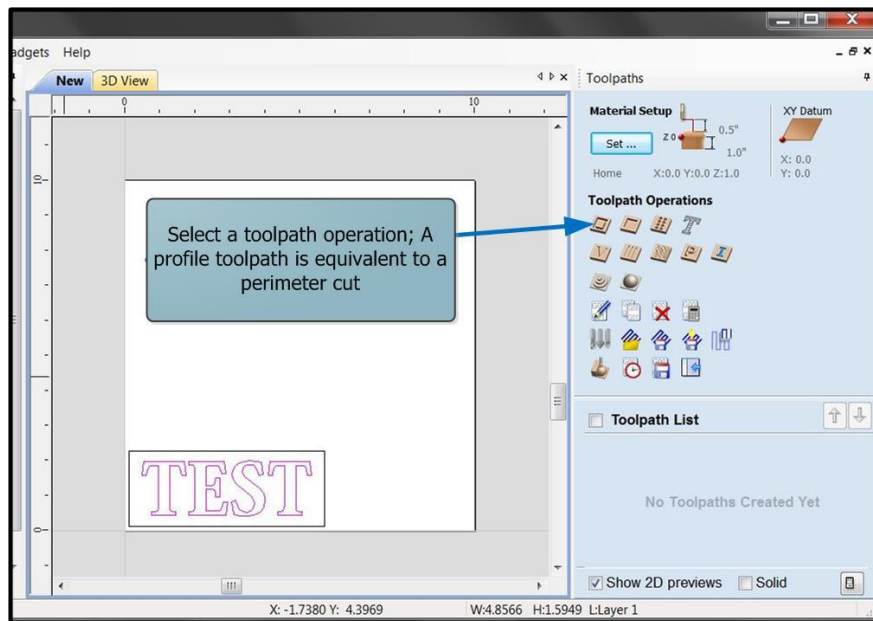
- Create a new project



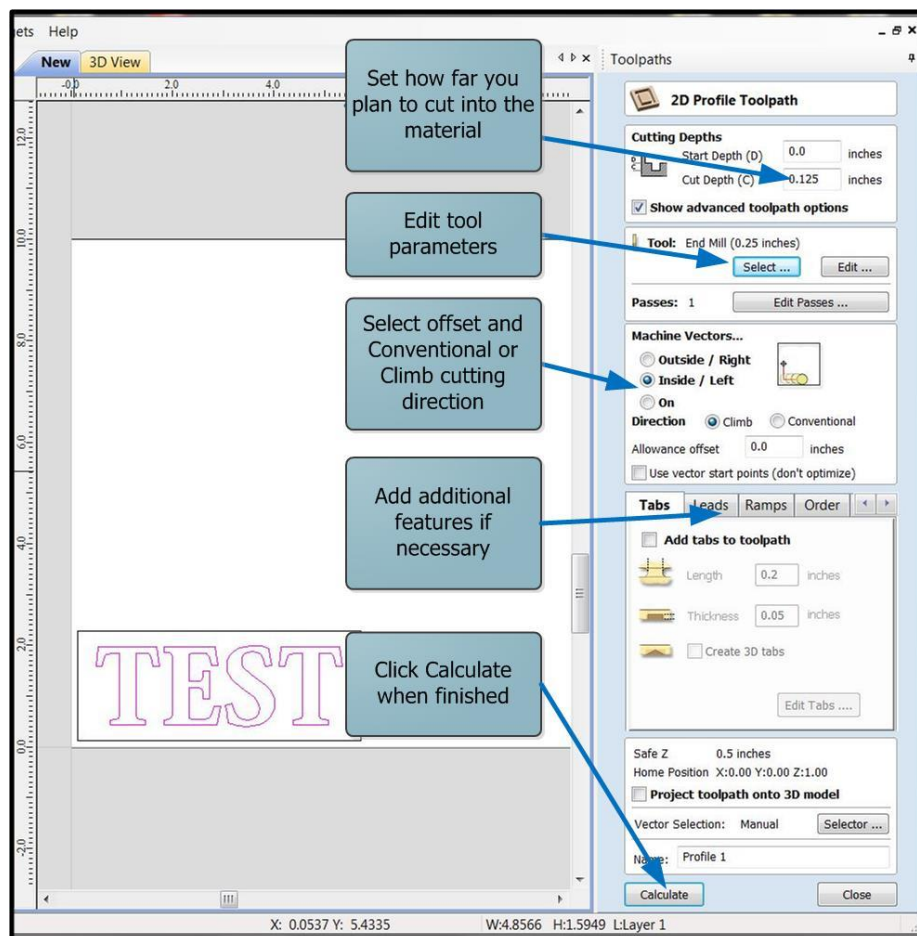
- Sketch your part



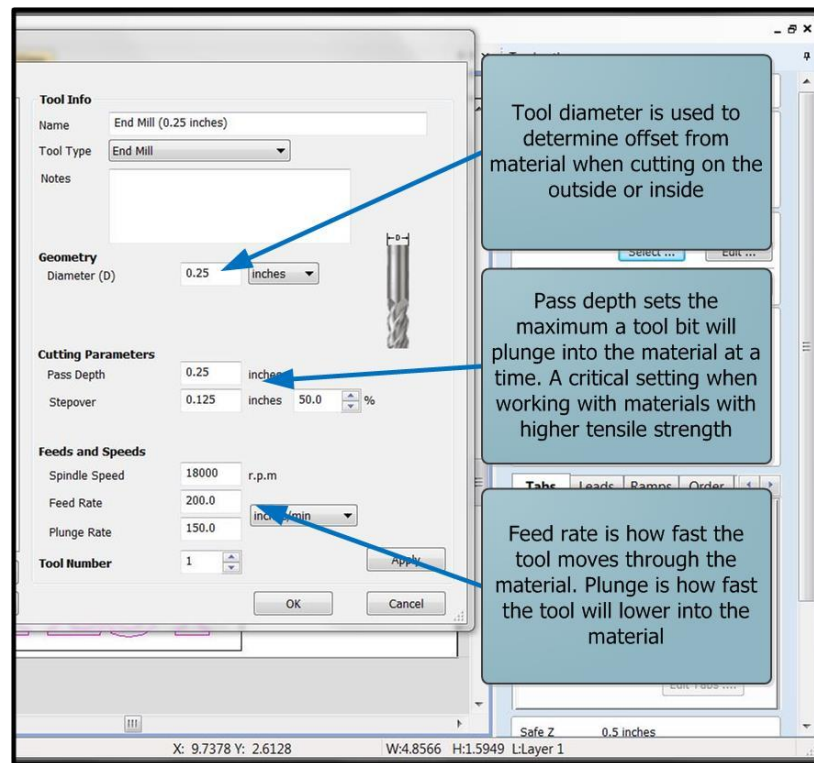
- **Open Toolpath tab and select a toolpath function**



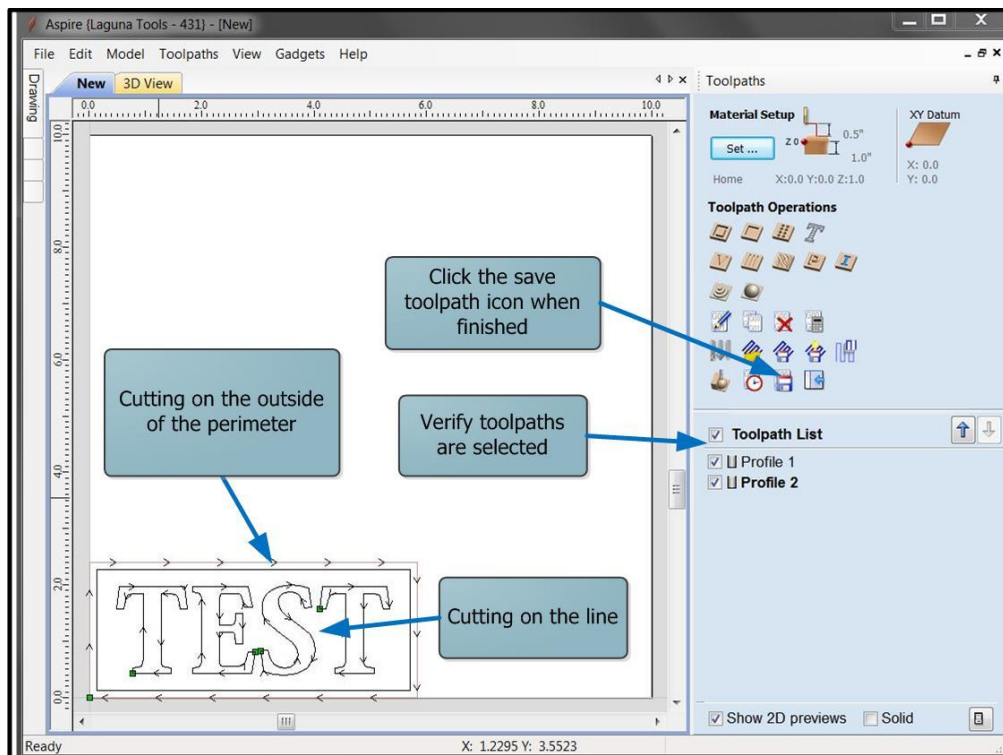
- **Edit toolpath parameters**



- **Edit tool parameters**



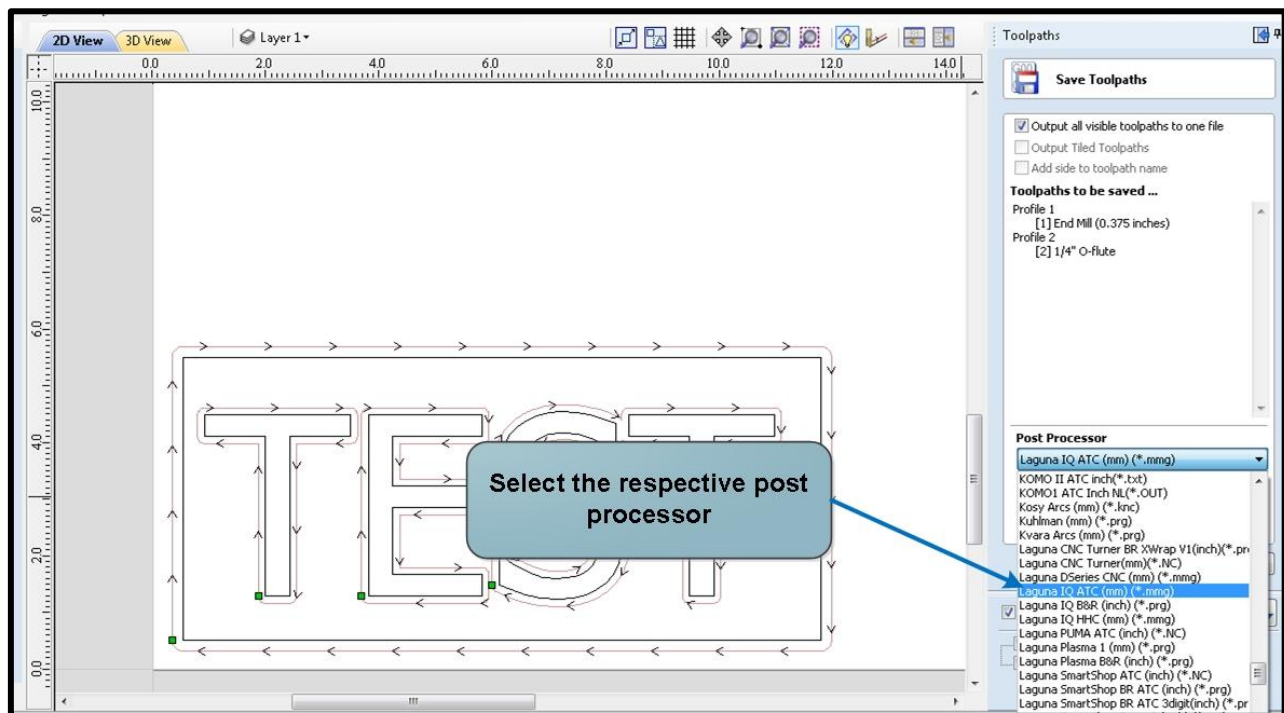
- **Visually verify that your toolpath is correct**





- **Save Toolpath**

- As shown below, the Smartshop M is compatible with the **Laguna IQ ATC(mm) post processor**.



- Inspect the g-code file

```

TEST_SmartshopM - Notepad
File Edit Format View Help
N0 (Filename: TEST_SmartshopM)
N10 (Machine: Laguna IQ ATC)
( Material size)
( X= 635.000, Y= 635.000, Z= 6.350)
N40 G54
N50 T1 M06
N60 M03 S24000
N70 G00 X8.8210 Y13.0333 Z38.1000
N80 G00 Z25.4000
N90 G01 Z-6.3500 F7620.0
N100 G01 Y140.0333 F20320.0
N110 G02 X13.5835 Y144.7958 R4.7625
N120 G01 X299.3335
N130 G02 X304.0960 Y140.0333 R4.7625
N140 G01 Y13.0333
N150 G02 X299.3335 Y8.2708 R4.7625
N160 G01 X13.5835
N170 G02 X8.8210 Y13.0333 R4.7625
N180 G00 Z38.1000
N190 M05
N200 T2 M06
N210 S18000 M3
N220 G00 X47.3844 Y32.9470 Z38.1000
N230 G00 Z25.4000
N240 G01 Z-3.1750 F762.0
N250 G01 Y101.3043 F2540.0
N260 G01 X23.3194
N270 G02 X20.1444 Y104.4793 R3.1750
N280 G01 Y114.0678
N290 G02 X23.3194 Y117.2428 R3.1750
N300 G01 X88.5864
N310 G02 X91.7615 Y114.0678 R3.1750
N320 G01 Y104.4793
N330 G02 X88.5864 Y101.3043 R3.1750
N340 G01 X64.5214
N350 G01 Y32.9470
N360 G02 X61.3464 Y29.7720 R3.1750
N370 G01 X50.5594
N380 G02 X47.3844 Y32.9470 R3.1750
N390 G01 Z-6.3500 F762.0
N400 G01 Y101.3043 F2540.0
N410 G01 X23.3194
N420 G02 X20.1444 Y104.4793 R3.1750
N430 G01 Y114.0678
N440 G02 X23.3194 Y117.2428 R3.1750
N450 G01 X88.5864
N460 G02 X91.7615 Y114.0678 R3.1750
N470 G01 Y104.4793
N480 G02 X88.5864 Y101.3043 R3.1750
N490 G01 X64.5214
N500 G01 Y32.9470
N510 G02 X61.3464 Y29.7720 R3.1750

```

- It is always a good practice to first run a program in the air without any material, until a work flow is established.

## 10 APPENDIX

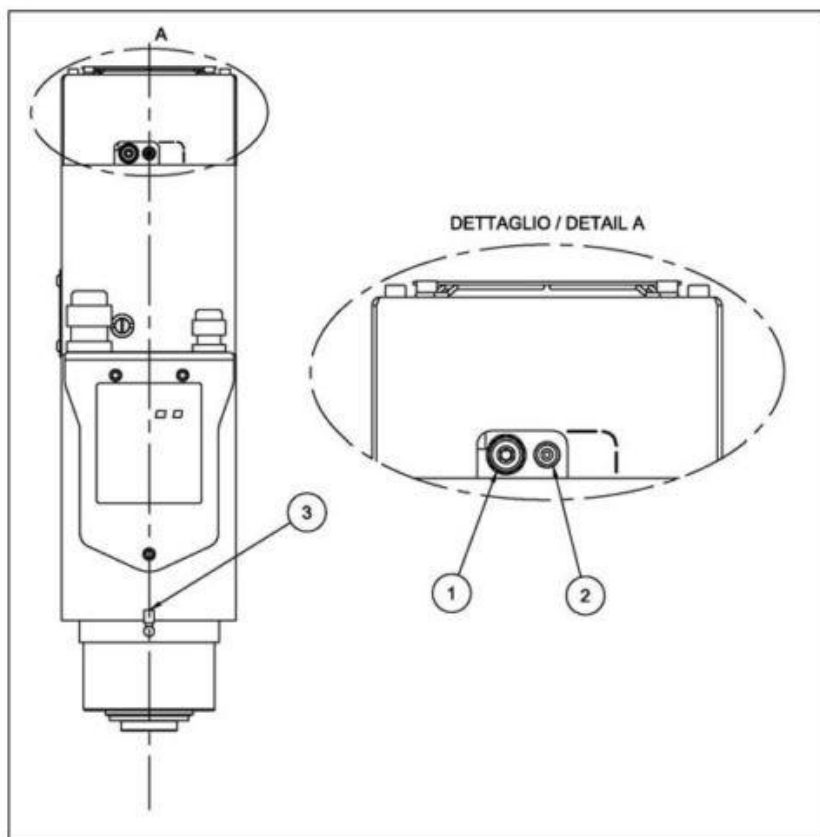
### 10.1 HITECO SPINDLE PNEUMATIC SPECIFICATIONS

Model: QD-1F 4/12 24 I30 NC CB BT

#### 4.4.2 Pneumatic connection points to the electrospindle

The pneumatic connections are indicated in figure.

(4.4



Ref.	Name	Function	Pressure	Ø Tube
1	<b>RELEASE TOOL</b>	Tool release	6 ÷ 6.5 bar	Ø6x4
2	<b>LOCK TOOL</b>	Tool lock	6 ÷ 6.5 bar	Ø4x2.5
3	<b>PRESSURIZATION</b>	Pressurization dried air	0.5 bar	Ø6x4

Pressure values lower than the indicated ones in the previous table may cause wrong operations like the tool non-loosening from the electrospindle.

## 10.2 INSTALLATION OUTLINE

### → Power

- Refer to the electrical tag on the back of the machine OR on the electrical cabinet (figure 1).
- Input power goes through the cabinet to the top of the rotary switch (figure 2).

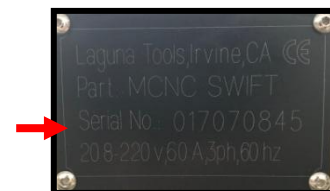


Figure 1

### → Air

- The machine uses compressed air for the dust hood and tool changer.
- Connect an air supply line to the air pressure regulator located on the back of the machine (figure 3).
- The stock fitting is for an 8mm diameter hose.
  - Replace fitting if needed.
- 90-100 psi



Figure 3



Figure 2

### → Vacuum Pumps

Vacuum pumps will vary between machines.

- Refer to the vacuum pump electrical tag for correct wiring.
- Run the appropriate gauge wire from the vacuum pump to the electrical cabinet.
  - There will be a contactor(s) located inside of the electrical cabinet where the vacuum power is connected (figure 4).
- Connect the table's vacuum hose to the inlet of the vacuum pump (figure 5).



Figure 4

### → Dust Collection

- Set up a dust collector with a 4" hose connected to the dust hood (figure 6).
- For a simple solution, insert the provided tubing into the holder on the right side of the gantry.
- Secure the dust hose to the tubing, and verify travel is not restricted.

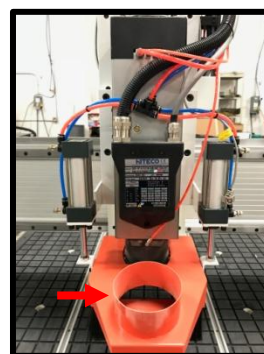


Figure 6

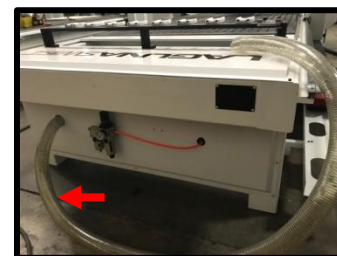


Figure 5

## 10.2 DEFAULT MACHINE SETTINGS

Revision: December 04, 2017

## MACHINE PARAMETERS

MENU &gt;&gt; Machine Parameter Setting &gt;&gt;OK

X equal	100.000 pls per mm	Size of X	1305.000 mm
Y equal	100.000 pls per mm	Size of Y	2915.098 mm
Z equal	400.000 pls per mm	Size of Z	260.000 mm
SpdOn Delay	4000 ms	SpdOffDelay	0 ms
XSpeedOfHome	3000.000 mm/min	YSpeedOfHome	3000.000 mm/min
ZSpeedOfHome	750.000 mm/min	Home actions	100.000
Feeler Block	84.717 mm	Max X-Speed	18000.000 mm/min
Max X+ Speed	18000.000 mm/min	Max Y-Speed	18000.000 mm/min
Max Y+ Speed	18000.000 mm/min	Max Z-Speed	15000.000 mm/min
Max Z+ Speed	5000.000 mm/min		

To change a value: Highlight field &gt;&gt; RUN&gt;&gt; Enter/Select Value&gt;&gt;OK

## WORK PARAMETERS

MENU &gt;&gt; Work Parameter Setting &gt;&gt;OK

WorkSpeed	3000.000 mm/min	Linear accel	800.000	mm/s <sup>2</sup>
Curve accel	1000.000 mm/s <sup>2</sup>	FastSpeed	8000.000	mm/min
Safe height	100.000 mm	XSpeedOfSlow	1000.000	mm/min
XSpeedOfFast	10000.000 mm/min	YSpeedOfSlow	1000.000	mm/min
YSpeedOfFast	10000.000 mm/min	ZSpeedOfSlow	500.000	mm/min
ZSpeedOfFast	1500.00 m/min	SpeedScale	1.000	
FallDown Scale	1.000	Act after	Pickup Z	
FallDownHeight	5.000 mm	G abs center	FALSE	
G read F code	Read F	G spindle	On by code	
G read T code	Read ATC		100.000	
G read S code	Read S	Array interval	0	ms
CircleLimit	1000.000	ArrayColCount	1	
ArrayRowCount	1	ArrayColSpace	0.000	mm
ArrayRowSpace	0.000 mm			

To change a value: Highlight field &gt;&gt; RUN&gt;&gt; Enter/Select Value&gt;&gt;OK

## OTHER SETTINGS

MENU&gt;&gt;MENU FUNCTION&gt;&gt;SYSTEM SETUP&gt;&gt;FUNCTION CONFI&gt;&gt;SpdOutpt&gt;&gt;

SglAng

MENU&gt;&gt;MACHINE FUNCTION USER INTERFACE/MACHINE SETUP&gt;&gt;

&gt;&gt;TOOLSET SETUP/TOOLSET SPEED/TS SPD

400.00 mm/min

&gt;&gt;CAD POSITION/TOOLSET IN POS/OK/OK/CANCEL/TOOL SETTING Z

-140.00 mm

To change setting: Highlight field &gt;&gt; RUN&gt;&gt; Enter/Select Value&gt;&gt;OK



# **LAGUNA**

2072 Alton Parkway. Irvine, CA 92606  
Ph: 800.234.1976 | [www.lagunatools.com](http://www.lagunatools.com)

Laguna Tools is not responsible for errors or omissions.  
Specifications subject to change. Machines may be shown with optional accessories.

© 2018, Laguna Tools, Inc. LAGUNA® and the LAGUNA Logo® are the  
registered trademarks of Laguna Tools, Inc. All rights reserved.