

ARTISAN II

5 Axis CNC Turning/Milling Machine

Instruction Manual



Legacy Woodworking Machinery
435 W. 1000 N.
Springville, UT 84663

Content

Warranty and Repair information 3
Safety Guidelines and Warnings 4
Machine footprint 5
Electrical and Air requirements 6

Artisan Set Up

Artisan Side Tower Connections 7
Proximity Switch locations 8
Start-up operation 9
Shut-down operation 10
Stopping Machine during operation 10

Reference Sheets

Artisan Fast Start - Turning 11
Artisan Fast Start - Flat Stock 11
Trouble Shooting 12
Glossary 13 - 14
G-code Reference Sheet 15

ARTISAN

5 Axis CNC Turning/Milling Machine

Product of:

Legacy Woodworking Machinery

435 W. 1000 N.

Springville, UT 84663

WARRANTY

Legacy Woodworking Machinery warrants to the original purchaser that its products are free from defects in material and workmanship.

This warranty covers electronic control devices for one year from the date of delivery.

This warranty covers the frame and mechanical devices for three years from the date of delivery.

This warranty does not cover any damage to Legacy products that result from improper installation, accident abuse, misuse, natural disaster, insufficient or excessive electrical supply, abnormal mechanical or environmental conditions, or any unauthorized disassembly, repair or modification.

Repairs

If product repair or replacement is necessary, the Customer is responsible for all shipping charges, freight, insurance and proper packaging to prevent breakage or damage in transit.

Technical Support

You can contact the Legacy's technical support department at:

cncsupport@legacywoodworking.com

(801) 491-0010

(800) 279-4570

Legacy provides free technical support for the first 90 days to its customers worldwide.

Proudly made in the USA

SAFETY GUIDELINES

Read and Observe All Guidelines Before Operating This Machine

1. Work area safety

- a) Keep work area clean and well lit.
- b) Do not operate machine in explosive atmospheres, such as in the presence of flammable liquids, gases or vapors.
- c) Do not operate machine in wet conditions.

2. Electrical safety

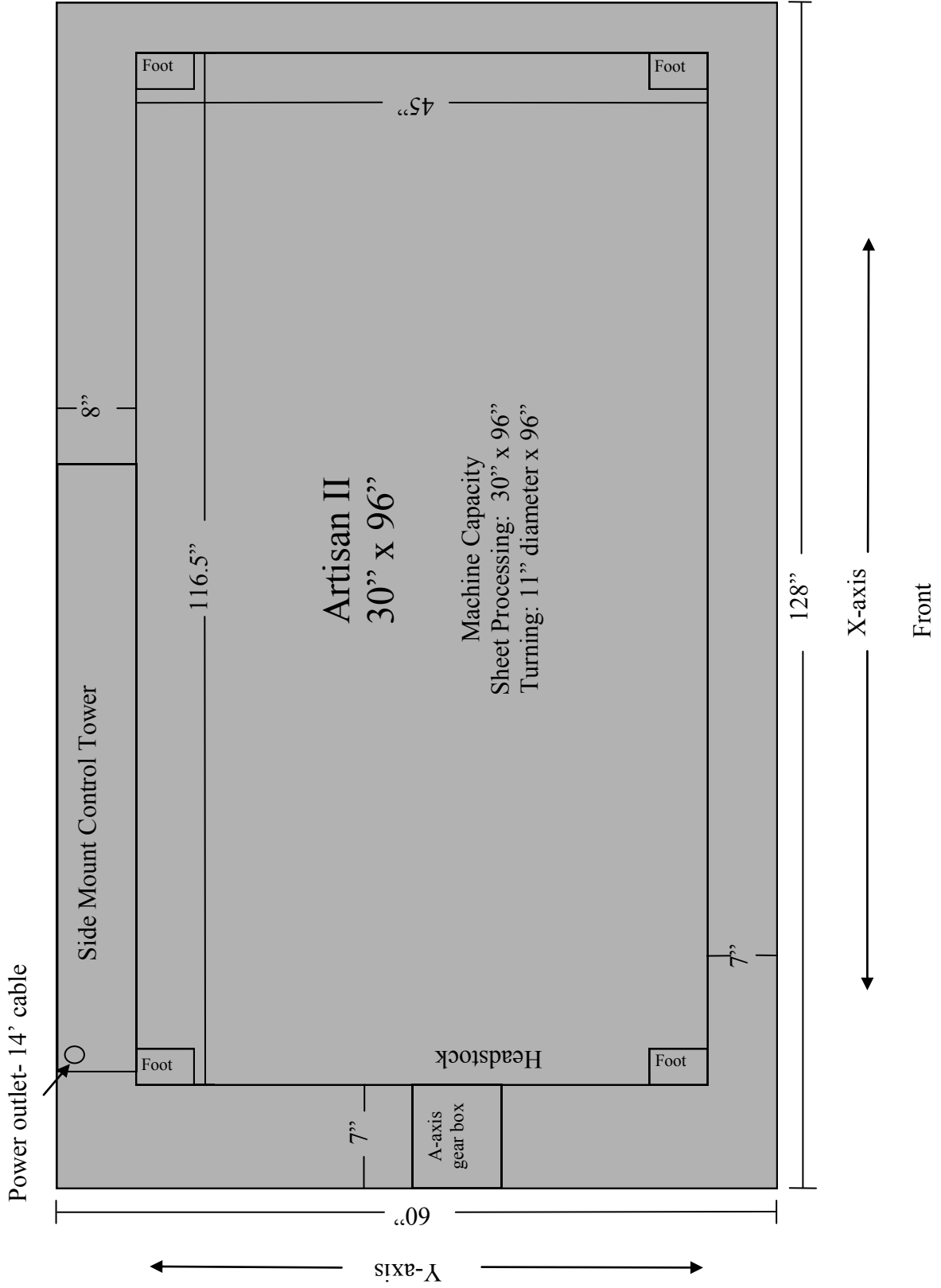
- a) Never modify the power plug in any way. Power plugs must match the outlet and do not use adapter plugs.
- b) Do not abuse the power cords. Keep cords away from heat, sharp edges and moving parts.
- c) All electrical panels should be closed during operation.
- d) All electrical connections and outlets must meet local and state code.
- e) Do not use extension cords.

3. Personal Safety

- a) This machine is automatically controlled and may start at any time.
- b) Only trained personnel should operate this machine. Untrained personnel present a hazard to themselves and the machine, and improper operation will void the warranty.
- c) Do not operate this machine while you are under the influence of drugs, alcohol or medication.
- d) Always wear eye protection. Use safety equipment such as ear protection and dust masks.
- e) Never reach into the machine while the machine is in operation. Do not rest or lean against the machine while the machine is in operation.
- f) Dress properly. Do not wear loose clothing or jewelry. Keep hair, clothing and gloves away from moving parts.
- g) Improperly clamped parts may be ejected. Machining oversized and marginally clamped parts is not safe.

*** It is the shop owner's responsibility to make sure that everyone who is involved in installing and operating the machine is thoroughly acquainted with the operation and safety instructions provided with the machine BEFORE they perform any actual work. The ultimate responsibility for safety rests with the shop owner and the individuals who work with the machine.**

Artisan II Machine Footprint



Artisan II

Electrical and Air Requirements

Artisan II	220 VAC grounded with 20 service Amps	480VAC grounded with 30 service Amps	Compressed Air 120 PSI
	Single Phase (110AC w/ 15 amp additional line required)	Three Phase	Three Phase
Columbo 5 hp Air Cooled	Standard Configuration	Available upon Request - Additional charges may apply	Available upon request Additional charges may apply
DLS 90 Dyna-Loc HSK Air Cooled with ATC	Standard Configuration	Available upon Request - Additional charges may apply	Available upon request Additional charges may apply 115 PSI—4 CFM (clean and dry required)

Manufacturer recommendations for the DLS 90 Dyna-Loc HSK spindle.

Complete manufacturer manual available at www.pdscolumbo.com

The air quality in a spindle is fundamental for correct use of the device and long life. Particularly, as in all devices that use compressed air, is the likelihood that condensation could form due to variations in pressure and temperature. This could be hazardous for the precision bearings in the spindle and the reliability of the clamping system.

To avoid this problem, hereunder you can find some general specifications about the air quality according to ISO 8573.1 Class 1 normative:

Accepted pollution dimensions	0.1	mm
Accepted pollution concentration	0.1	mg/m ³
Accepted oil particles dimensions	0.01	mg/m ³

Moisture Specifications:

Accepted pollution dimensions	0.1	mm
Accepted pollution concentration	0.1	mg/m ³
Accepted oil particles dimensions	0.01	mg/m ³

Where an air dryer is not available in the feeding line, a filtration-drying system is recommended. Particularly, a membrane type can guarantee efficiency and low cost.

Typical Pressure Requirements for the electric spindle:

Input	Typical Pressure
M6 Tubing	7 to 8 Bar (102 to 116 psi)

The air supply must never exceed 11 Bar (160 psi)!

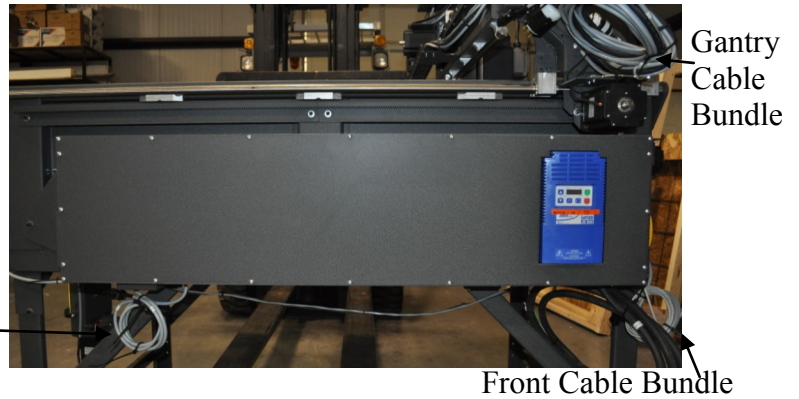
It takes more than point-of-use filters to remove dirt, oil, and water from compressed air. While particulate filters are the right choice for combating dirt and liquid water and oil, they do not lower dew points or remove oil aerosols. (Dew point, a measure of relative humidity, is the temperature at which water begins to condense as compressed air cools. As long as the compressed air temperature does not drop below the established dew point the system will remain dry.)

What is not commonly known is that liquid oil aerosols must be removed, along with the water, for a system to be truly clean and dry. These small submicron aerosols can cause just as much damage as wet air. The reason is that the oil is burnt, acidic, and will condense inside pneumatic components causing sticking and downtime. Dryers and filtration systems must match the air compression system in volume to be effective.

Uncrate and position the Artisan,

Do not apply power to the Control Tower at this stage!

The Control Tower has been mounted on the side of the Artisan. The cable connections are located on the underside of the Control Tower.



← Underside of Control Tower
Front of machine

Fan or Pump Power Plug
(Air & H2O cooled only)



Spindle Power Plug Lower J box Upper J box B axis A axis Z axis Y axis X axis R X axis L

CAUTION

You can easily damage the pins or sockets in the connectors by forcing them into place when pins are misaligned.

Align the plug and connector bodies with the mating “keyways and alignment slots”. Rotate the plug until the keyway and alignment slot match, checking the male pins alignment to the female sockets.

Tighten the locking ring in place.

Each cable is labeled. Carefully unwind the bundled cables. Attach each cable end to the matching receptacle on the underside of the tower.

The Gantry Bundle are all plugged into the control tower:

- X axis-Left (X - L)
- X axis-Right (X - R)
- Y axis
- Z axis
- Upper J (Upper Junction Box)
- Spindle Power Cable

Front Cable Bundle:

- Machine Main Power Cable (110V or 220V) to the main grounded power source.
- A-axis to the control tower.
- Stationary Pad cable (which will need to be plugged into the stationary pad)
- USB Cable (goes to the computer)

Back Cable Bundle both plug into the control tower.

- B axis
- Lower J (Lower Junction Box)



Proximity Switches:

The Artisan control software acquires the home position for each axis by reading signals sent from each proximity switch during the “Set Machine Coordinates” sequence.

A-axis proximity switch is located on the headstock directly below the shaft. The trigger plate is mounted to the headstock shaft.

Once “Reference Machine” is complete, the control software has the needed information to correctly run the machine within the pre-programmed limits.

Y-axis proximity switch is located on the top gantry toward the front of the machine. When looking at the gantry from the headstock end of the machine, the proximity switch will be to on the top right end of the gantry.

A Legacy CNC mill can not be driven into the physical axis barrier unless the Soft Limits has been turned off. If a axis is driven past the proximity switch and into the physical limit of the machine, you will need to run “Reference Machine”.

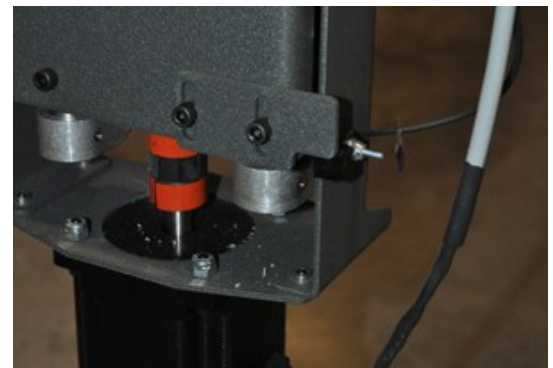
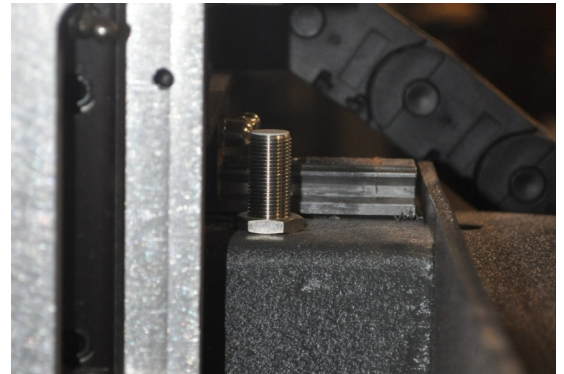
X-axis proximity switch is located on the back inside panel of the gantry. The trigger plate is attached to the top of the back X-axis frame rail close to the headstock end of the machine.

Beginning a program with one of the proximity switches activated will cause that axis to run backwards—make sure all axes are moved off the limit switches prior to running a program.

Z-axis proximity switch is located on the top gantry toward the front of the machine. When looking at the gantry from the headstock end of the machine, the proximity switch will be to on the top right end of the gantry.

The proximity switches are triggered when metal enters the switches magnetic field, direct contact is not recommended.

B-axis proximity switch is located on the lower back edge of the B-axis assembly housing. The adjustable trigger plate is mounted to the moving back plate of the B-axis assembly.



Start Up:

First: Power to the Control Tower. The on/off button is found on the front end of the side mounted control box.

Second: Power to the computer. The computer power button is located on the top of the control tower. This button is either large and red or flat and silver.



Control Box
Power Switch

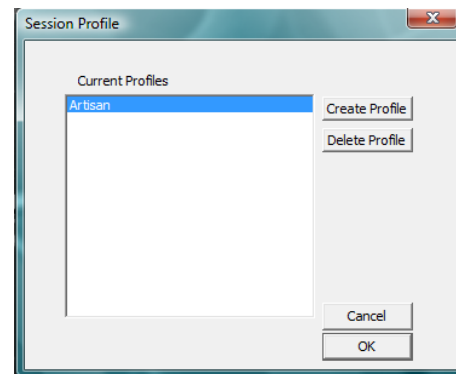
The software operating system for the Legacy CNC interface (Mach3) is Microsoft Windows XP, Vista or Windows 7. This is used with the Artisan, OM5, Standard and Performance CNC machines. The following icons will load to your desktop: Recycle Bin, Mach3 Loader, Conversational CAM 3.1, Calculator, Notepad, and OM5 Screen Help.



Open the Mach3 loader to launch the Legacy CNC Control Software.

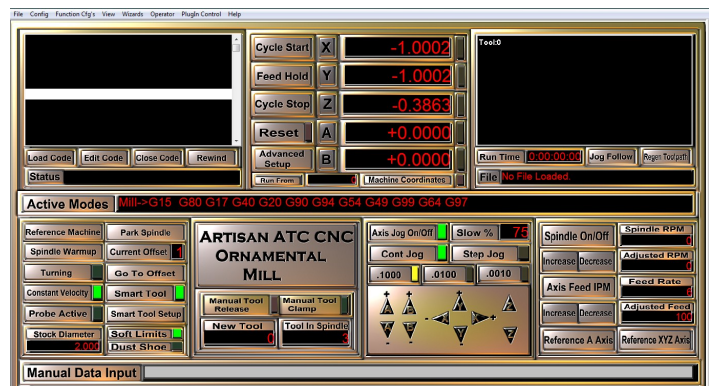
A pop-up menu will appear—select the machine listed and click OK. This screen is preset during manufacturing to reference the type of CNC machine you purchased:

- Artisan II**
- Artisan
- Arty
- Mini Arty
- Performance
- Standard



The Legacy CNC Control Software will open to the main control screen. The “Reset” button will be flashing red. Open communication between the control software and machine by clicking the “Reset” button off.

Your machine is now ready for use.



Shut Down Operations

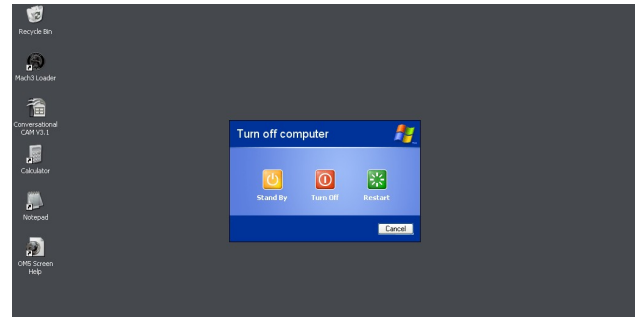
It is recommended that you run the “Park Spindle” program, located within the OM5 Control Software on the “Machine Setup” screen, before shutting down your machine. This will move the X, Y and Z axes to the home position.

To power down the Arty:

First: Shut down all programs running on the computer then shut down the computer.

Wait for the computer to shut down completely!

Second: Shut off the power to the Control Panel.



Stopping the Machine

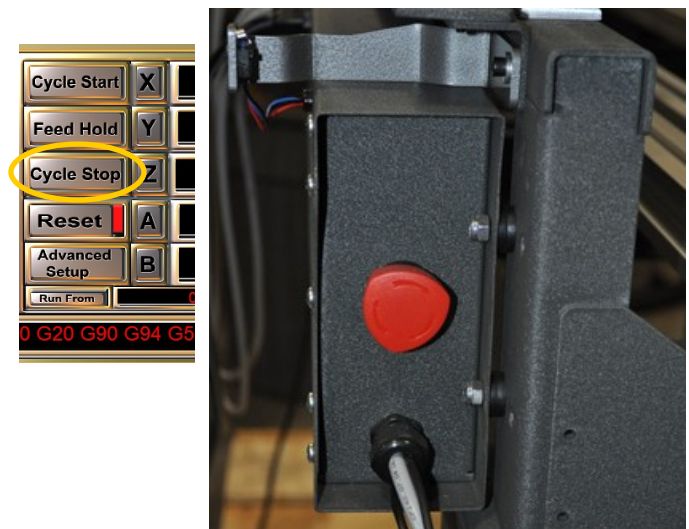
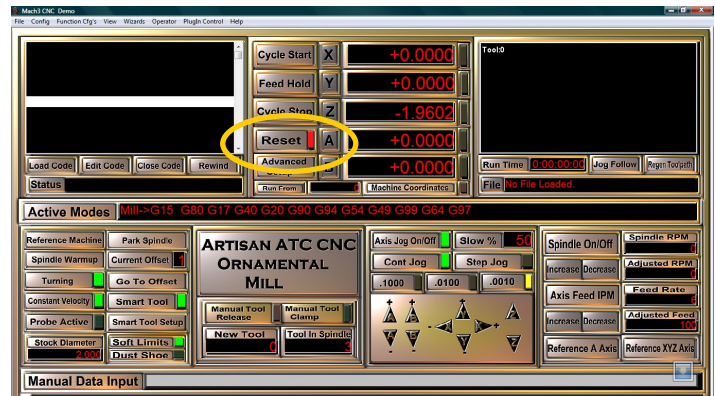
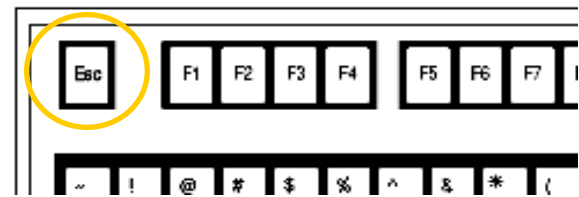
There are four recommended ways to stop the machine during operation.

1. The “Esc” button on your keyboard.

2. The “System Reset”. You will need to reset the System Reset button to continue further milling.

3. The **Emergency Stop Button** found on the end of the computer control box. When the Emergency Stop Button is pressed, power will be cut to the motors. It will be necessary to reset the Emergency Stop button and click the System Reset button. To reset the Emergency stop button, simply twist the button in the direction the arrows indicate.

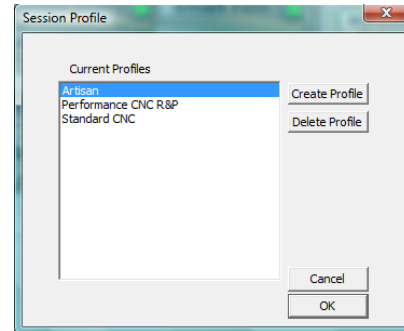
4. The “Cycle Stop” button found on the Run Machine screen. Cycle Stop will instantly stop the program currently running. Because of the abrupt stop, step may be lost—run the “Reference Machine” sequence and reset “Active Modes”.



Machine Start Up & Set Up



1. Turn machine power and computer power on.
2. Open Mach3 loader
3. Choose your machine from “Session Profile” screen.
4. With Mach open to the Legacy CNC Control Interface, click the flashing “Reset” button.
5. Reference Machine by clicking the Reference Machine button.
6. Spindle Warm up—if you have an Air or H2O spindle.
7. Choose Turning on or off.

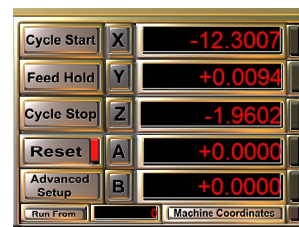
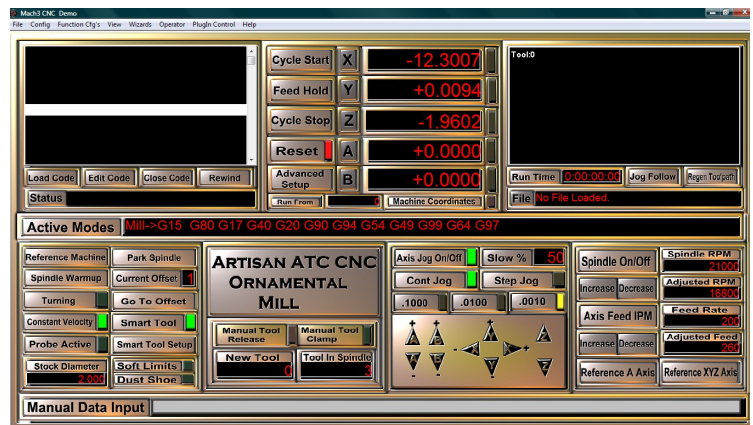


Turning

8. Enter the diameter of the stock in Stock Diameter.
9. Current Offset should display 6
10. Turn Smart Tool on or off
11. Place blank between centers.
12. ATC machines—do tool locations match programmed tool changes?
13. Load Code
14. Cycle Start to begin running program
15. Follow program prompts in Status line for tool changes.
16. Program will end and rewind when finished.

Flat Stock Milling

8. Current Offset should read 1.
9. Mount flat blank into machine.
10. Set up Offset location by driving X & Y axes to the design XY origin.
11. Zero X and Y axis buttons.
12. Place spindle over part.
13. Place Mobile Smart Pad on part and directly under spindle.
14. Run Smart Tool Setup (this will set your Z axis)
15. ATC machines—do tool locations match programmed tool changes?
16. Load Code
17. Cycle Start to begin running program.
18. Follow program prompts in Status line for tool changes.
19. Program will end and rewind when finished.



Occurrence	Possible Cause	Corrective Action
Smooth Stepper Error code:	This code sometime appears when the “Esc” key is pressed during a running sequence, i.e. touching off the smart tool	Call Legacy for the most recent download plug-in for the Smooth Stepper.
Clicking noise from motors .	One of the motors was driven into the physical limits.	Bring the motor back within the proximity switch limits. Run “Set Machine Coordinates”.
When running “Set Machine Coordinates” the Z axis moves down, away from the proximity switch.	One of the proximity switches were enabled when “Set Machine Coordinates” was activated.	Move all axes off the proximity switches and run “Set Machine Coordinates” again. You can check if any proximity switches are engaged by checking to see if any of the M1 thru M5 Home DROs are lit on the Diagnostic screen .
If you have burn marks on the wood.	1—Bit is dull 2—Router or Spindle RPM is to high 3—Your feedrate is to slow	1—Sharpen bit 2—Slow Router or Spindle RPMs down 3—Increase your feedrate
System Reset will not move	1—Emergency Stop Button is engaged. 2—No communication with the Smooth Stepper	Release the Emergency Stop Button Power all the way down and restart
The machine’s movement is jerky when running a program, especially while carving.	Machine is in the G61 exact stop mode. Hardware is lose allowing excessive play in movement of machine.	Replace the G61 in the “Active Modes” code line with G64. You could also add a G64 to the preamble of the code you are running. Check and tighten all hardware and fittings. Contact Legacy if this does not fix problem.
Error: Can not run G53 incremental	Machine modal settings do not match program modal settings. This can occur when using the emergency stop button, the escape button, or system reset while a program is running.	Power off the computer and control tower and restart. To avoid this error, click the modal reset button prior to restarting the program after using a emergency stop button.

Bull Nose: The Bull Nose center is a 1.4”diam. x 1.25” Delrin, self lubricating, center designed to be placed on the tailstock shaft. The Bull Nose has been manufactured to an exact thickness to guarantee accurate Z-axis “0” settings when choosing to use the Bull Nose as a touch off method.

and

Bull Nose is a tool touch off method used with the Standard CNC OM5 Control Software specifically designed to accurately set the Z-axis home position for A-axis turning when a homing location is not available. When choosing “Bull Nose” as a touch off method, a Bull Nose is placed on the tailstock shaft and the cutter is brought down to the Bull Nose surface, the Z axis is zeroed, then .07 is subtracted for Z to establish Z “0” for the center of the A work offset.

Conversational CAM: Is a “Computer Aided Manufacturing” program which uses specific questions to design turnings and flat milling, then produces the G-code needed to run the Legacy CNC milling machine. Conversational CAM was designed by Legacy to streamline the design process of commonly milled components without the use of a **CAD** (Computer Aided Design) software program to draw the parts. CAD programs are design programs only and do not produce g-code.

Feed rate: Is the speed the cutter moves over the part, measured in IPM “inches per minute”. When using the A-axis, the feed rate is the speed the part moves past the cutter. Guidelines: larger diameter of cutter = slower feed rate. Deeper the cut = slower feed rate. If the bit chatters, slow down the feed rate speed up the RPMs. Recommended feed rates for the Standard CNC machines are 80 - 120 IPM.

G-code: functions in the Numerical control programming language. G-codes are the codes that position the tool and do the actual work. M-codes manage the machine; T-codes relate to tool numbers. S-codes relate to tool speed. F-codes represent feed rate; H-codes represent tool height offsets. The programming language of Numerical Control (NC) is sometimes informally called G-code. But in actuality, G-codes are only a part of the NC-programming language that controls NC and CNC machine tools.

Legacy CNC Control Interface: The Legacy CNC Control Interface software has been custom designed by Legacy as an interface to Mach3 and is the control software used to run all Legacy CNC milling machines. The Legacy CNC Control Interface software was designed as a user friendly screen interface to help utilize the unique milling attributes of the Legacy CNC mills.

Machine Coordinates: Machine coordinates represent the “home” location set on each axis when running the “Reference Machine” setup sequence. When choosing to display the “Machine Coordinates”, the DRO’s will show the distance each axis has moved in relationship to the “home” location.

Part: Refers to the work stock placed in the Legacy Mill.

and

Is a tool touch off method used in Conversational CAM where the router bit is brought down to the surface of the part to establish the cutter length in reference to the machine Z axis “0” position.

Reference Machine: This machine set up button initiates a preprogrammed sequence which drives each axis to the “homing switch” installed on that axis, triggers the switch, moves off the switch and sets the Machine Coordinates to that axes location.

RPM—Rotations Per Minute: Recommended Maximum Spindle/Router Bit RPM. Adjust listed speeds according to your Spindles/Router max RPM capabilities.

Bit Diameter	RPM
0.00” - 1.00”	21,000
1.00” - 2.00”	19,000
2.00” - 2.50”	16,000
2.50” - 3.0”	13,000

Smart Tool: This Legacy designed programming process is placed within the Mach control software and activated through the Legacy CNC Control Interface. When activated, this program will automatically set the Z axis height for each cutter. Eliminating the need to touch off your tools or use tool tables.

Smart Tool Mobile Pad: A mobile touch off pad, designed by Legacy to aid in the Smart Tool Touch Off process. Each machine comes standard with one mobile pad.

Smart Tool Touch Off: This Legacy designed process allows you to accurately set the Z axis “0” location to the surface of your flat stock work piece by using the Smart Tool Mobile Pad.

Stepper Motor: is a brushless, synchronous electric motor that can divide a full rotation into a large number of steps. The motor's position can be controlled precisely without any feedback mechanism (open loop controller).

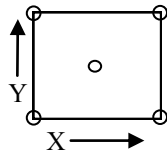
Surface Planing: To smooth or level the uneven surface of a work piece by using a flat bottom bit to accurately remove material. This process can be programmed in Conversational CAM.

Tool Offset: Is a tool touch off method option in Conversational CAM. You must first set up a tool library in the Legacy CNC Control Interface “Advanced Setup” screen. Once a tool library has been set up, you can choose to access the tool heights by the tool numbers listed in the library. This will allow the operator to assign a number and saved tool data on individual tools.

Tool Touch Off Methods: These are the methods available for measuring the tool height in relationship to the Z “0”. The four tool touch off methods used are Smart Tool, Tool Offset, Part and Bull nose.

Work Offset: The Work Offset represents a location “offset” from the Home Coordinates where work will be performed. The Work Offset for flat stock milling is manually set by moving the spindle head to the work location along the X and Y axes, positioning the spindle head at the XY origin set up in the design program, and then zeroing the X and Y axes using the Axis buttons in the Work Offset display area. The default flat stock Work Offset number is 1 (G54 is the g-code location). Additional flat stock offset locations can be set using any number between 1-5. The Work Offset for turning has been pre-set for each machine. The turning Work Offset number is 6 (G59) and is automatically located at the A-axis turning center.

XY Origin: The XY origin is the name for the X zero and Y zero locations selected for a program during the designing process on flat stock material. This location, once selected, must remain constant throughout the designing process and setting of the machine work offset. Five locations are available center, top left, top right, bottom left, bottom right.



G-codes are the codes that position the tool and do the actual work.

M-codes manage the machine

G0: Rapid positioning
 G1: Linear interpolation
 G2: Clockwise circular/helical interpolation
 G3: Counterclockwise circular/helical interpolation
 G4: Dwell
 G10: Coordinate system origin setting
 G12: Clockwise circular pocket
 G13: Counterclockwise circular pocket
 G15/G16: Polar Coordinate moves in G0 and G1
 G17: XY plane select
 G18: XZ plane select
 G20/G21: Inch/Millimeter unit
 G28: Return home
 G30: Return home
 G31: Straight probe
 G40: Cancel cutter radius compensation
 G41/G42: Start cutter radius compensation left/right
 G43: Apply tool length offset (plus)
 G49: Cancel tool length offset
 G50: Reset all scale factors to 1.0
 G51: Set axis data input scale factors
 G52: Temporary coordinate system offsets
 G53: Move in absolute machine coordinate
 G54: Use fixture offset 1
 G55: Use fixture offset 2
 G56: Use fixture offset 3
 G57: Use fixture offset 4
 G58: Use fixture offset 5
 G59: Use fixture offset 6
 G61/G64: Exact stop/Constant Velocity mode
 G68/G69: Rotate program coordinate
 G70/G71: Inch/Millimeter unit
 G80: Cancel motion mode
 G90: Absolute distance mode
 G91: Incremental distance mode
 G92: Offset coordinates and set parameters
 G93: Inverse time feed mode
 G94: Unit per minute
 G98: Rapid Height by Z height
 G99: Rapid Height by R height

M0: Program stop
 M1: Optional program stop
 M2: Program end
 M3/M4: Rotate spindle clockwise/counterclockwise
 M5: Stop spindle rotation
 M6: Tool change
 M30: Program end and rewind
 M47: Repeat program from first line
 M98: Call subroutine
 M99: Return from subroutine/repeat

S codes are related to the tool Speed

F codes are related to the tool Feed

T codes are tool related and represent the tool number.

H codes are tool height related and represent the height offset listed in the tool library for the tool listed. Example: H6 = height offset for Tool 6