Mini Arty

The Personal
CNC Turning & Milling Machine
4 Axis

Instruction Manual

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

Mini Arty CNC Machine

Content

Legacy Woodworking Machinery

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 it's 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.

Routers and Spindles are covered by their manufacturers warranties.

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
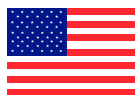You can contact the Legacy's technical support department at:
cncsupport@legacywoodworking.com
(801) 491-0010
(800) 279-4570

Legacy provides free technical support for the first 90 days to its customers worldwide. Any on-site training days purchased or received with the original purchase must be used with one year of purchase.

Proudly made in the USA

Mini Arty the Personal CNC Machine
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. 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.

Mini Arty the Personal CNC Machine
Operation Requirements & Machine Capacity

Mini Arty
Base Footprint

23.75"

22.75"

44"

40"

Headstock
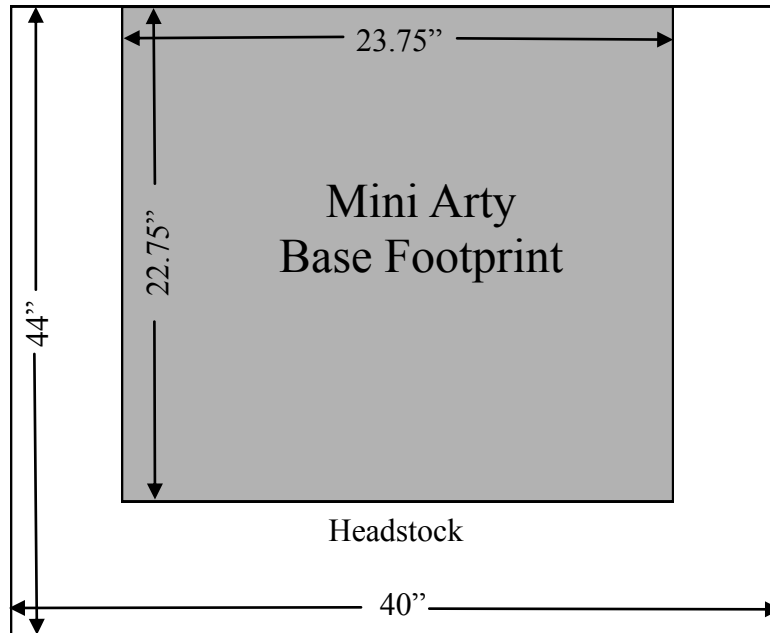
Mini Arty Full Machine Footprint

Machine Capacity:

Sheet Processing: 18" x 18"

Turning:
   5 inch Diameter x 18" length

Electrical Requirements

Mini Arty	110 VAC 15 service Amps
Side Mounted Control Tower	✓
Router	✓
Computer	✓

Setup

It is important to locate the Mini Arty on a level and stable surface.   You will need access to three 110V grounded outlets to run the Mini Arty. It is recommended that an extension cord not be used.

For ease of operation, we recommend putting the control computer to the left of the Mini Arty, allowing cables to stay tucked between the computer table and Mini.
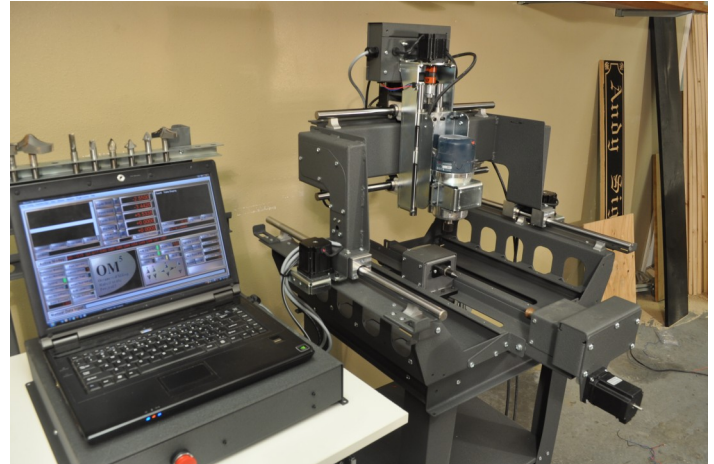
Note: The Control Tower is now mounted to the left side of the Floor Stand.

The Mini Arty comes with the gantry cables X, Y, Z, J-box and router power cord bundled. The A-axis cable runs from the A-axis gear box. Each cable is labeled to assure proper installation.

1. Connect the follow wires to the bottom of the control box labels back to front—X, Y, Z, A & J-box.
2. Plug in the Router and Control Tower Power cords into grounded 110V electrical outlets. The Router power cord exits the gantry with the Axis cables. The Control Tower power cord extends from the bottom of the bottom front panel of the Control Tower box.
3. Set up the Laptop; plug in the Laptop power cord to a grounded 110V electrical outlet. Plug in the Control Towers USB cable into the computer.

Control Tower Bottom View

X axis   Y axis   Z axis   A axis   J-Box

Control Tower Power cord & USB Cable

Control Tower Top View
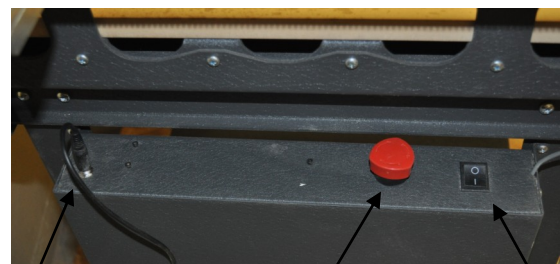
Smart Pad 1/4" jack receptacle   Emergency Stop Button   Power Switch

Start Up Operations

Start up sequence.

First: Power to the Mini Arty. The on/off button is found on the top of the Control Box.

Second: Power to the computer or laptop.

When powering on the Mini Arty for the first time, turn the router power button to the on position. With the power button in the on position, the control software will automatically turn the router on and off when running the G-code programs. Leave the router button in the on position.

The software operating system for the OM5 interface is Microsoft Windows XP, Vista or Windows 7. This is used with the Artisan2, Artisan, Arty, Mini Arty, Standard and Performance CNC machines The following icons were loaded to your desktop or laptop:
Recycle Bin, Mach3 Loader, Conversational CAM 3.1, Calculator, Notepad, and OM5 Screen Help.

Open the Mach3 loader to launch the OM5 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
> Arty
> **Mini Arty**
> Performance R&P
> Standard

 The Legacy CNC Control Software will open with the Reset button flashing—click "Reset

Click: "Reference Machine", which will drive the Z, Y and X axes to their individual homing switches and set up "Machine Home".

You are ready to begin machine operations.
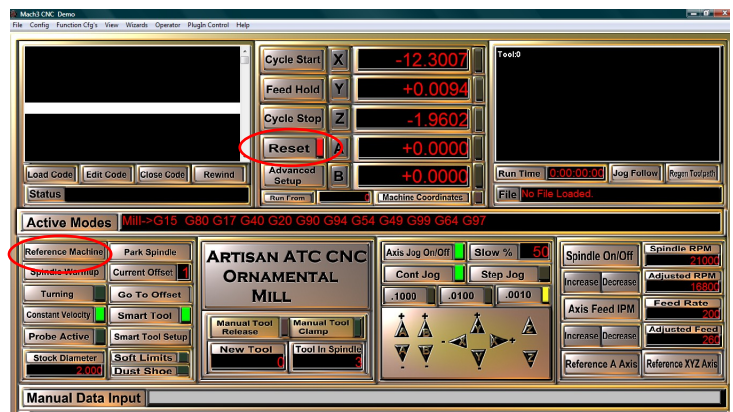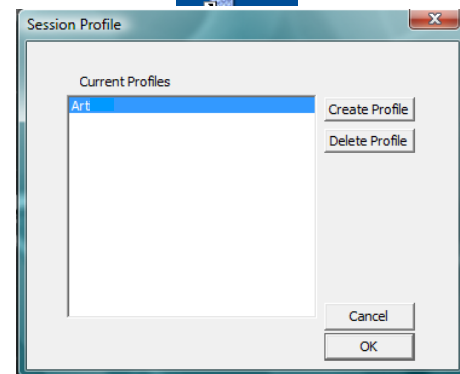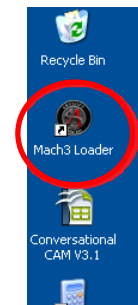
Shut Down Operations

It is recommended that you run click "Park Spindle" button to run the Park Spindle sequence prior to shutting down the Mini Arty and software.
Park Spindle is found on the Legacy CNC Control Software page.

To power down the Arty:

First: Shut down all programs running on the computer then shut down the computer. It is recommended to wait for the computer to shut down completely prior to shutting off the Mini.

Second: Shut off the power to the Control Box.
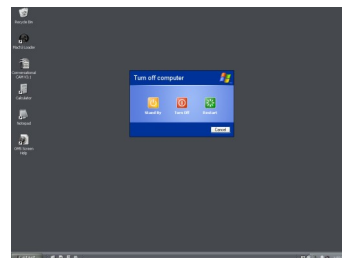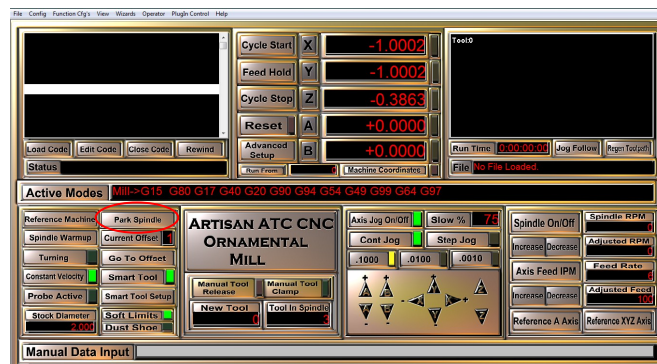
Emergency Stopping the Machine

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

1. The "**Esc**" (escape) button on your keyboard.

2. The "**Reset**" button found on the Legacy CNC Control screen. The Reset button will cut communication to the machine immediately stopping all machine movement. You will need to click Reset again to operate your machine.

3. The "**Cycle Stop**" button on the Legacy CNC control screen. Cycle Stop will instantly stop the program currently running. It will be necessary to rewind and restart the program once Cycle Stop is clicked. It is also advised to click the "Modal Reset" button after interrupting a running program. If the stop cause steps to be lost in one of the Axes, "Reference Machine" will also have to be run prior to beginning the program.

4. The **Emergency Stop Button** found on the top of the 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 Reset button. To reset the Emergency stop button, simply twist the button in the direction the arrows indicate.
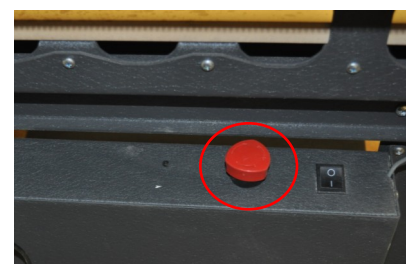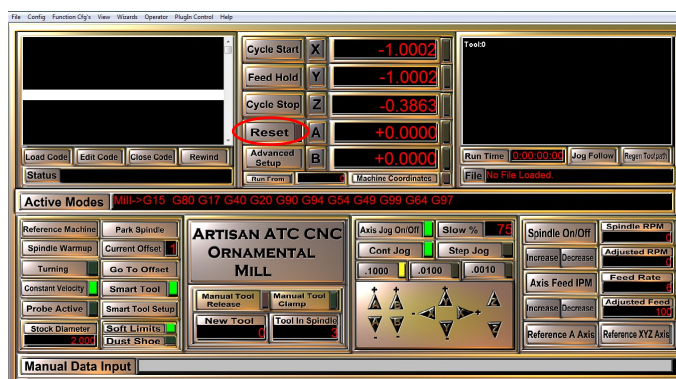
Mini Arty Home Switch locations:

There are three home switches on the Arty Personal CNC Machine. These switches are used during the "Reference Machine" sequence giving the Mini a stationary "home" location. The homing switches have a lever arm which is pressed by a trigger plate.

<u>*Care is needed when working close to the homing switches on the Mini. The lever arms can be bent or pulled off if clothing or tools catch them.*</u>

Z-axis homing switch

Switch

Lever arm
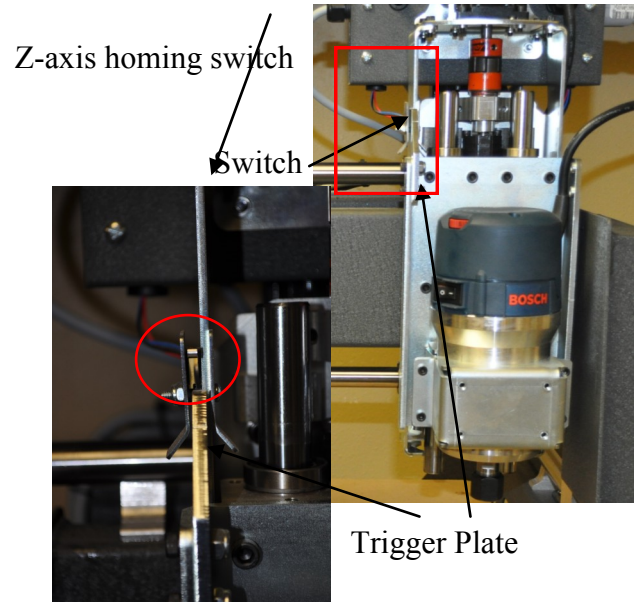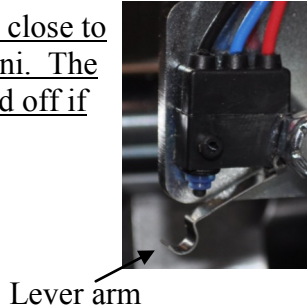
Trigger Plate

The Z-axis Home switch is located on the front left side of the Z-axis assembly.

The Y-axis Home switch is located on the back side of the gantry to the left of the Y-axis drive motor. The trigger plate is also mounted on the back of the gantry, left end of the rack drive.

Y-axis homing switch

Trigger Plate   Switch
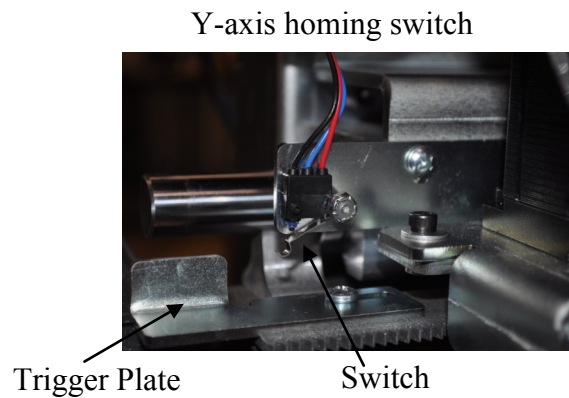
The X-axis homing switch is located on the left front end of the X-axis drive rack. The trigger plate is the gantry edge.

X-axis homing switch
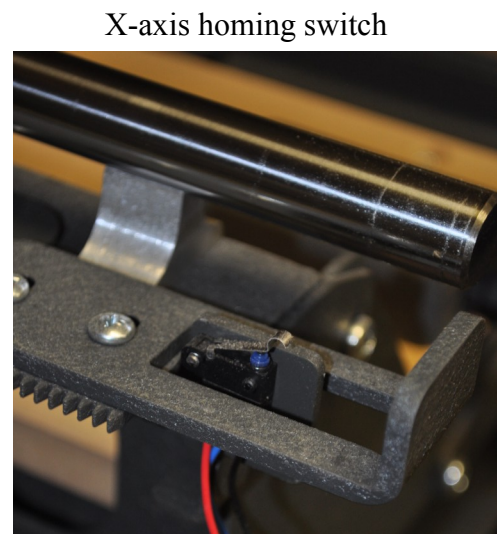
Smart Tool

Smart Tool was developed by Legacy Woodworking to aid in measuring tools used during the milling process. Smart Tool is a two part program within the Mach control software and activated through the Legacy CNC Control Interface screen.

"**Smart Tool**" is a program used for touching off the tool during a tool change (g-code M6), eliminating the need to touch off your tools or use tool tables.

"**Smart Tool Setup**" is a measuring sequence used to set the Z-0 height to the surface of a piece of flat stock.

The **Smart Tool** button with LED, turns Smart Tool on or off.
When on, at tool changes, the Smart Tool program will automatically measure the height of the tool by slowly touching off on the stationary pad, then continue running the program. (Mini Arty's use the Mobile Pad as the Stationary Pad).

When off, at tool changes, the control software will proceed directly into the program without measuring the tool height.

Machine Set-up

Smart Tool
Stationary Pad

Smart Tool
Mobile Pad

Smart Tool Mobile Pad
The Smart Tool Mobile Pad is used during the "Smart Tool Setup" sequence. You will use the "Smart Tool Setup" sequence to give Mach the location of the <u>surface of flat stock material</u>.   To do this:
1. Mount your flat stock and set the XY offset location.
2. Place the Mobile Pad on top of the flat stock material.
3. Center the spindle with tool, directly over the Mobile Pad.
4. Click the "Smart Tool Setup" button. The spindle will lower to the pad and "touch off".
5. Follow prompt to remove pad and press "Cycle Start". Spindle will move to the Stationary Pad and touch off. The Z axis will automatically be zeroed to the surface of the material.
This ends the "Smart Tool Setup" sequence.

Machine Set-up

Smart Tool key points:
1. Smart Tool on = measuring of tool after a M6 (tool change) g-code.
2. Smart Tool Setup = using the Mobile pad to set the Z axis "0" to the surface of flat stock.

Smart Tool Pad

The Mini Arty has been designed to use the Mobile Smart Tool pad as the Stationary Smart Tool pad. This design uses less of the work envelope for the Smart Tool sequences allowing machine operators the ability to work on larger work pieces. The Smart Tool touch off sequence is programmed into the Mach3/Legacy CNC Control software. For a more detailed description of the Smart Tool sequence,

Plug the Smart Tool pad into the 1/4" jack at the top/ black of the Control Box.

Smart Tool Pad

The Smart Pad location for the tool change touch off has been pre-programmed into the Legacy CNC Control software at X0, Y11 in Machine Coordinates. To find this location:

1. Turn off Smart Tool (click Smart Tool button to turn off LED)
2. Type into the Manual Data Input line: M6T1 and press enter—the machine will move the spindle to the Tool Change location—the pad location should be directly under the spindle so the Smart Tool sequence will touch off on the metal surface of the Smart Pad.
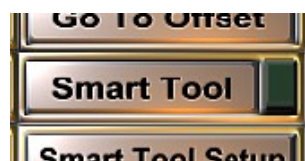
With Smart Tool on, when ever a M6 (tool change) is programmed and run, the Legacy CNC Control software will move to this location and pause for the tool change. Once the tool change is complete, press Cycle Start. The Legacy CNC Control software will then run the touch off sequence to measure the height of the tool and apply that height to the known Z0 and run the program.

When using the "Smart Tool Setup" button to find the Z-0 on flat stock you will use the Smart Tool Mobile pad for both touch off locations in the sequence.

1. Place the Smart Tool pad on the surface of the flat stock blank and position the spindle directly over the pad—click "Smart Tool Setup". A touch off sequence will be preformed and a pop-up screen will ask you to move the Smart Pad to the "Tool Change" position.
2. Move the Smart Pad to the tool change position and press ok. The Legacy CNC Control software will automatically move to the tool change location, perform the touch off sequence and set the Z-0 height for the part measured.

Setting Up Work Offsets for Flat Stock

A "Work Offset" is a location where "work" is performed. This location is "offset" from the home coordinates. When setting work offsets, you must refer to the XY origin selected in the design process. The XY origin is the point where X & Y converge, or the X zero and Y zero point. This XY origin must then be set up on the work piece in the same orientation. The XY origin location is manually set by the machine operator when beginning a new program using the default Offset 1. Multiple flat stock offsets can be set by typing in 2—5 in the "Current Offset" display screen.

Finding your XY origin

The XY origin location on the work offset should match the XY origin location on your program drawing.

Illustrated in the diagrams to the right—you can see that the operators orientation to the work piece changes. When working on a computer drawing, we sit with the X axis running left to right across the bottom of the screen. When operating the machine, the X axis is running away from us.

In Conversational CAM all Flat Stock programs are referenced to the operators left front corner ○ of the material OR right corner on machine orientation. Many CAM programs, including V-carve, Aspire, Bob-CAD and Art Cam allow you to choose the XY origin during material setup. The most common locations for the XY origin are the center point ● and the drawing left front = operators right front. ○
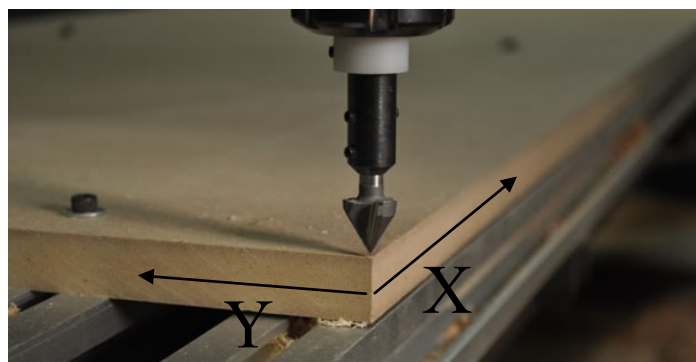
Center point

operator
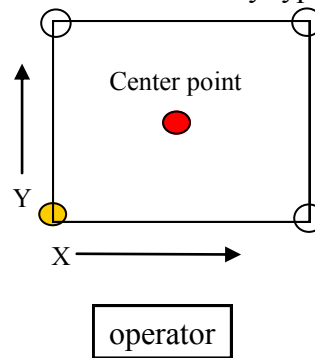
Computer Drawing Orientation

Headstock

operator

Center point
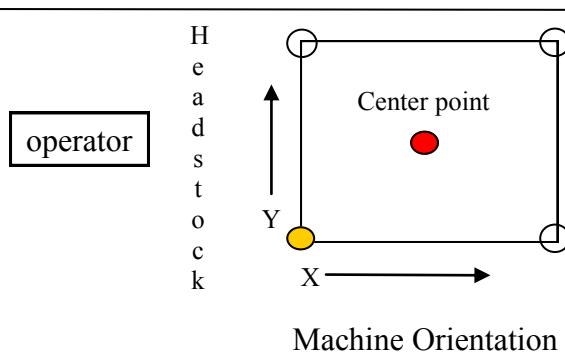
Machine Orientation

Once the XY origin location is determined, manually move the router to the XY origin location on the mounted material blank. Using a pointed router bit as a reference tool, jog to the XY origin location.

Verify that the "Current Offset" reads 1, then zero the X and Y axis DRO's by clicking the X and Y axis buttons.

The Z axis location will need to be set using the "Smart Tool Setup" process and Mobile Pad (page 16).

Reference Machine	Park Spindle
Spindle Warmup	Current Offset 1
Turning	Go To Offset
Constant Velocity	Smart Tool
Probe Active	Smart Tool Setup
Stock Diameter 2.000	Soft Limits
	Dust Shoe

X	+0.0000
Y	+0.0000
Z	-1.9602
A	+0.0000
B	+0.0000

Setting Up Work Offset for Turning

When setting up all work offsets, three key locations are required. The XY Origin (X-0 and Y-0) and the Z-0. The Turning Offset location has been preprogrammed into the Legacy CNC Control software for the X, Y and Z locations.

Y axis "0" is centered directly over the head-stock shaft.
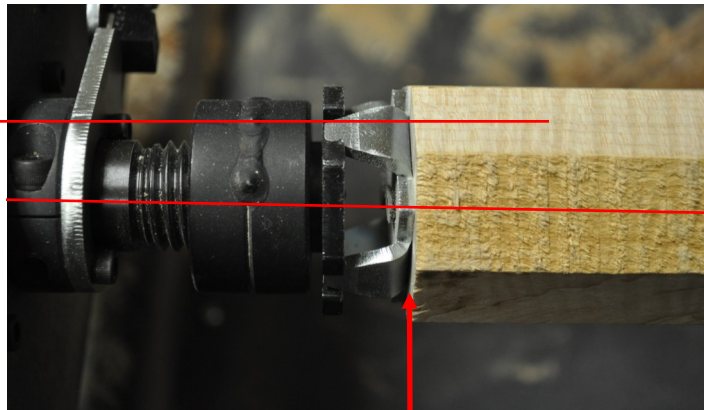
Z axis "0" location is the center point of the headstock shaft.

X axis "0" location is set at the end of the part when using a Index hub

Machine Set-up

To work in the turning offset, simply click "Turning" on the Machine Set-up area of the control screen. Check the "Current Offset" DRO to verify that 6 is displayed. Six is the offset number. A g-code value has also been assigned to the A-axis turning offset, G59.


If a different A axis offset position is needed, such as when using a four jaw chuck - it is recommended you set up a second turning offset using one of the other offset locations (5 for G58).

1. Go to the Turning Offset (6) and copy down the Work Offset Display DRO numbers for the Y and Z axis.
2. Go to a new offset number (2-5). Type in the numbers from offset 6 into the Y and Z Work Offset Display DRO's.
3. Move the spindle to the new X axis home location and zero the X axis Offset Display DRO

New X axis "0"home position
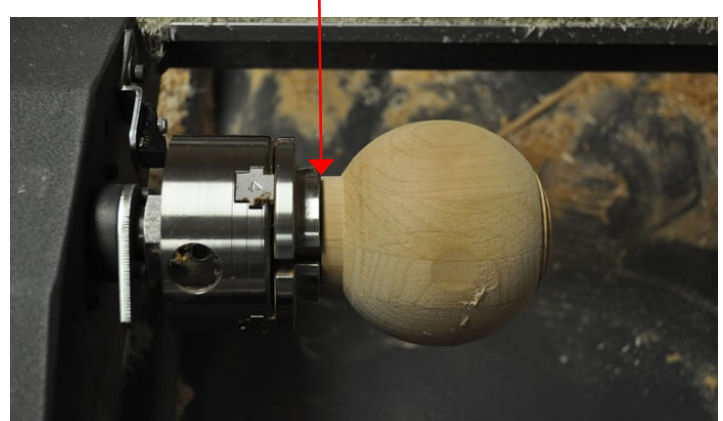
Work Offset Display

X	-1.0002
Y	-1.0002
Z	-0.3863
A	+0.0000
B	+0.0000
	Machine Coordinates

CNC Fast Start Reference Sheet
Turning

1. Power on to main CNC Tower.
2. Power on to computer.
3. Open Legacy CNC Control Interface by launching the Mach3 Loader found on your desktop, Start menu or toolbar. Choose your machine from the Session Profile screen.
4. Click the flashing red "System Reset" button.
5. Click "Set Machine Coordinates".
6. Click the "Turning" button and set the "Stock Diameter" display to the diameter of your work piece.
7. Turn on "Smart Tool".
8. Securely mount your work blank in the machine between centers (A axis).
9. Open Conversational CAM V3.11 from the desktop, Start menu or toolbar.
10. Enable Macros
11. From the Main Menu, choose one of the turning CAM process you wish to use.
12. Answer questions according to your work piece design. Choose "Smart Tool" for the tool touch off method.
13. Generate G-code
14. Click "Copy Code"
15. Open notepad and "paste" code to screen by right clicking the mouse and clicking "paste" OR going to Edit and "paste" on the notepad menu.
16. Name and save this notepad code to the G-code folder in My Documents. Use a name that is descriptive and will help you identify your part. Your program has now been saved to the hard drive and is ready to be accessed by the Legacy CNC Control Interface.
17. From the Legacy CNC Control screen click "load code". You should find the code in the G-code folder found in My Documents.
18. Load the code by double clicking the file name or choosing "open".
19. With your code loaded—Click "Cycle Start"
20. The program will begin and prompt a tool change—insert the correct tool.
21. After the tool change, click "Cycle Start" to continue the program through to the end following prompts for tool changes.
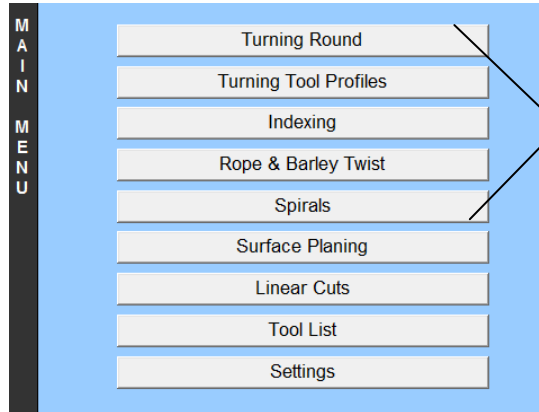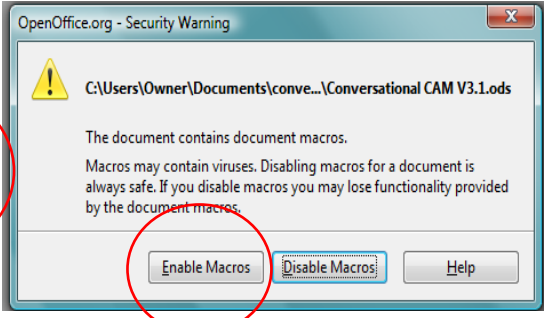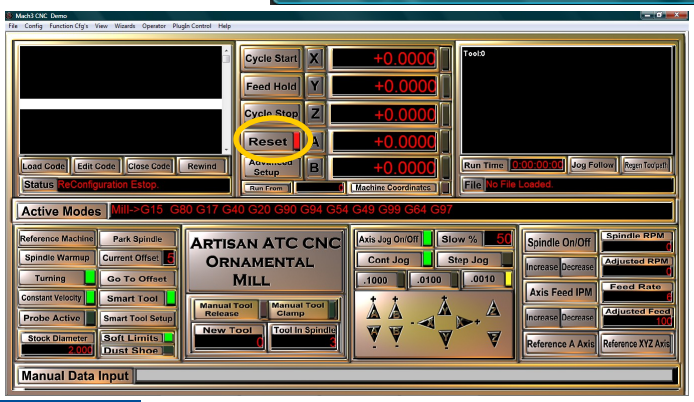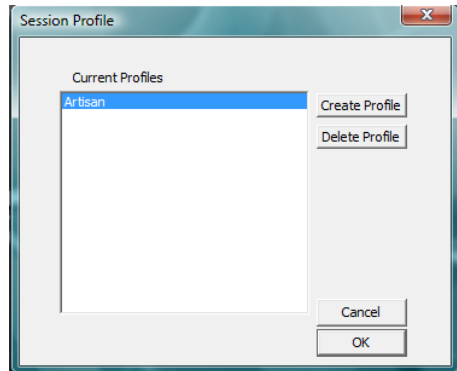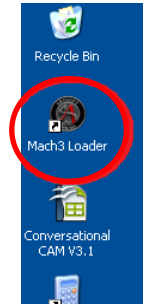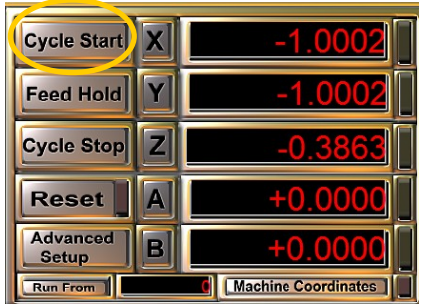
CNC Fast Start Reference Sheet
Flat Stock Milling

1. Power on to main CNC Tower.
2. Power on to computer.
3. Open Legacy CNC Control Interface by launching the Mach3 Loader found on your desktop, Start menu or toolbar. Choose your machine from the Session Profile screen.
4. Click the flashing red "System Reset" button.
5. Click "Set Machine Coordinates".
6. Make sure the Turning button is off, "Current Offset" should read 1.
7. Securely mount your work blank in the machine.
8. Place a center finding tool or pointed tool into the Spindle/Router.
9. Set your "Work Offset Position" for X and Y by manually moving the Spindle over your work piece to the XY origin (page 15)
10. Zero the "Work Offset Display" for X and Y.
11. Turn "Smart Tool" on.
12. Place the Mobile Smart Pad flat on your work piece, plug it into the Stationary Smart pad. Center the spindle above the pad.
13. Click "Smart Tool Setup". The machine will automatically run the "Smart Tool" sequence. Follow prompts to remove the Mobile Pad. Remove pad. Click OK to finish the routine.
14. Open Conversational CAM V3.11 from the desktop, Start menu or toolbar - Enable Macros.
15. From the Main Menu, choose the type of flat stock milling you wish to design.
16. Answer the questions according to your work piece design choosing "Smart Tool" for the tool touch off method.
17. Generate code.
18. Click "Copy Code" at the top of the screen.
19. Open notepad and "paste" code to notepad screen by right clicking the mouse and clicking "paste" OR going to Edit and "paste" on the notepad menu.
20. Name and save this notepad text file to the G-code folder in My Documents. Use a name that is descriptive and will help you identify your part. Your program has now been saved to the hard drive and is ready to be loaded by the Legacy CNC Control Interface.
21. From the Legacy CNC Control screen click "load code". You should find the code in the G-code folder found in My Documents.
22. Load the code by double clicking the file name or choosing "open".
23. With your code loaded—Click "Cycle Start"
24. The program will begin and prompt a tool change—insert the correct tool.
25. After the tool change, click "Cycle Start" to continue the program through to the end following prompts for tool changes.
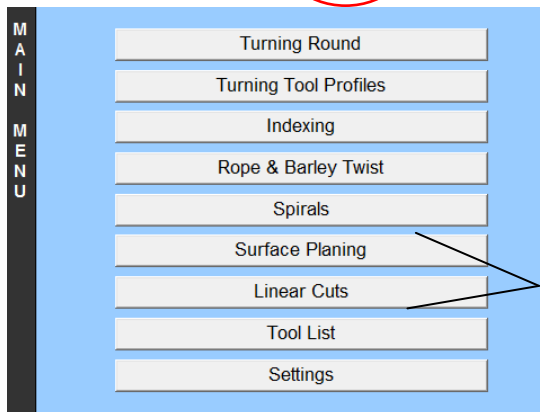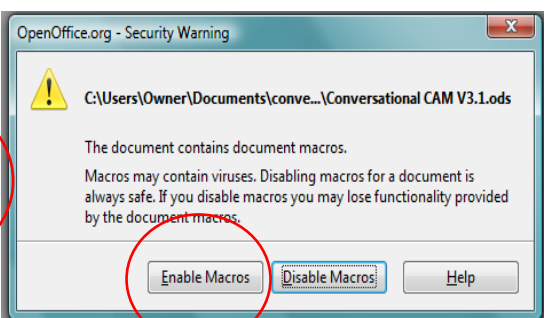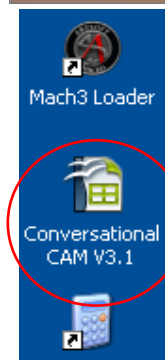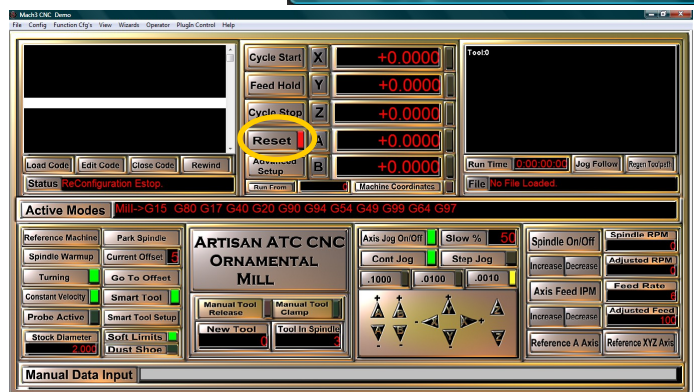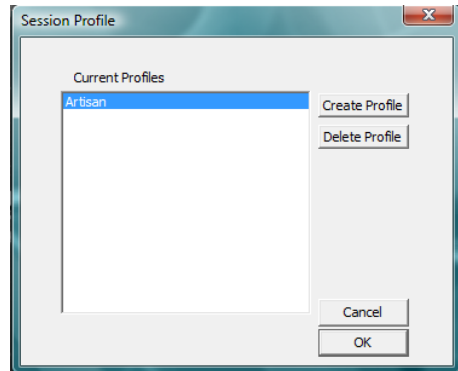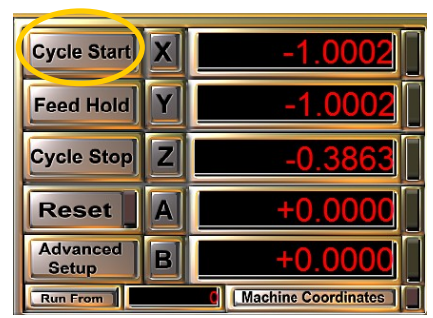
Flat Milling Processes

Trouble Shooting

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 "**C**omputer **A**ided **M**anufacturing" program which uses specific questions to de-sign 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 compo-nents without the use of a **CAD** (**C**omputer **A**ided **D**esign) software program to draw the parts. CAD pro-grams 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 40 - 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 ac-cording 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 acti-vated 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.
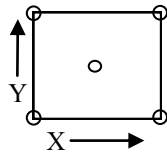
G-code reference sheet

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