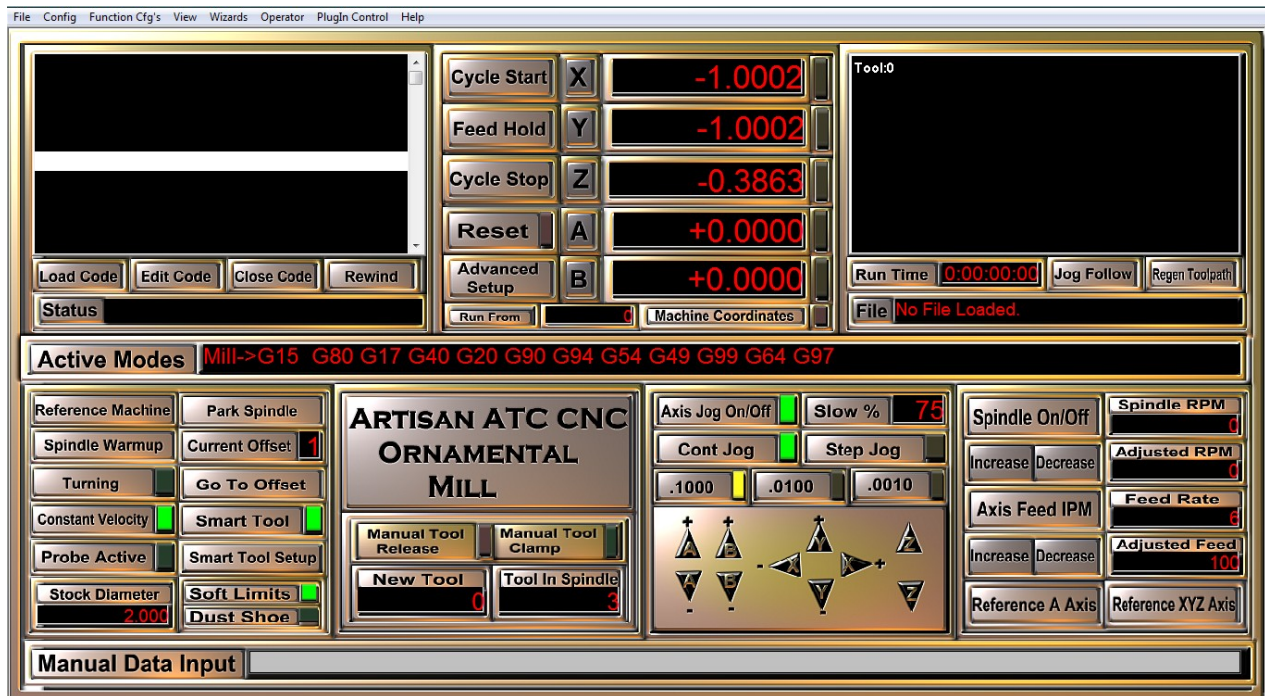# Legacy Woodworking Machinery
5 Axis CNC Turning/Milling Machines

Legacy CNC Control Interface
for Mach 3
Instruction Manual

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

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

Introduction

Legacy CNC Control Interface for Mach 3

The Legacy CNC Control Interface for Mach3 combines the unique features of Legacy's 5-axis milling machines with Mach3's powerful CNC Control software. The Interface has been designed to make operating a Legacy 5 axis CNC easy to understand and allows you to take full advantage of Legacy's unique CNC design and features.
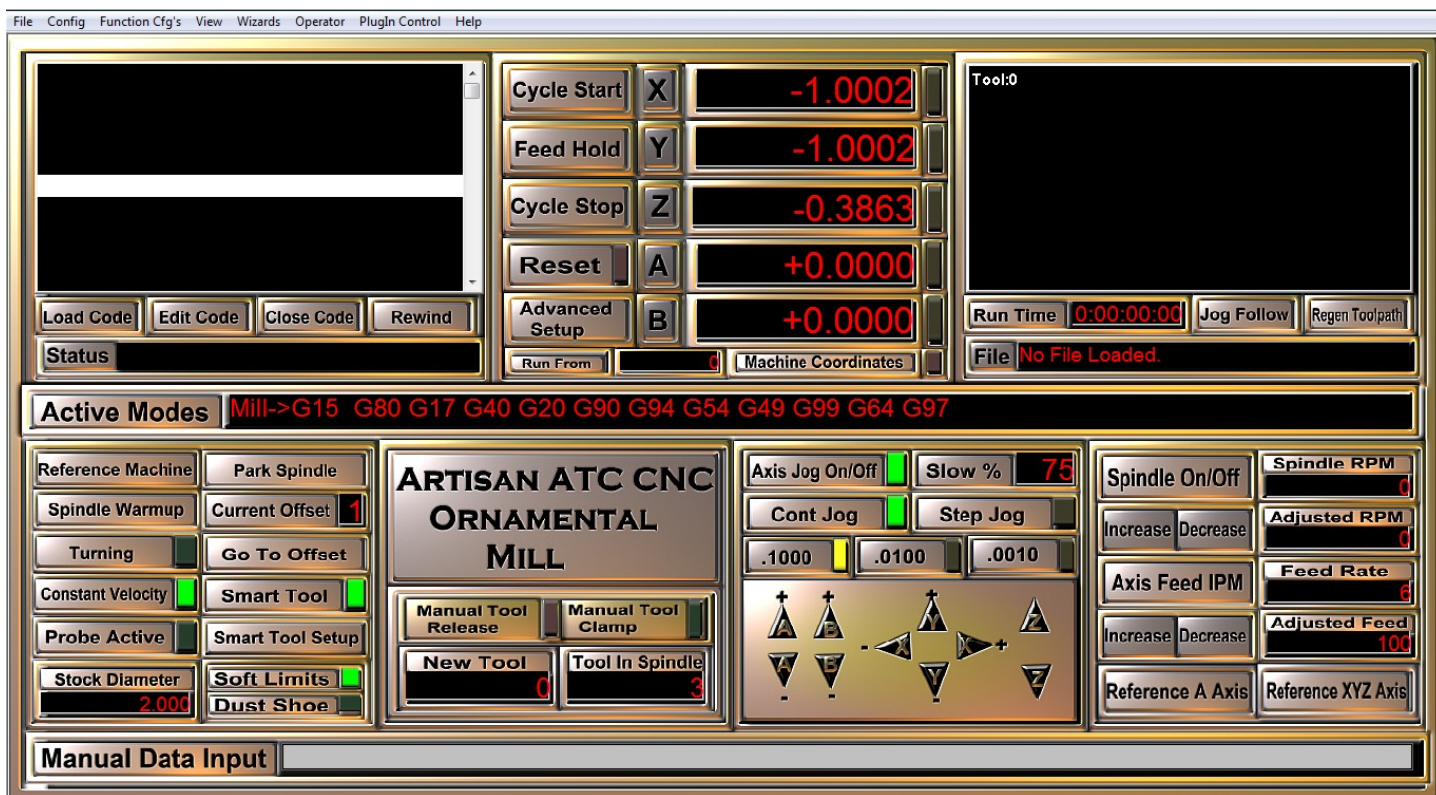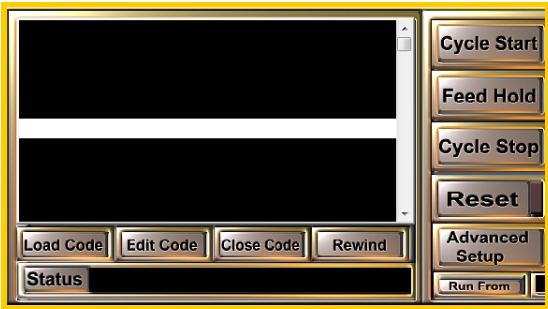
Overview

Advanced function drop-down menu bar

File Config Function Cfg's View Wizards Operator PlugIn Control Help

The Advanced drop-down menu bar accesses the main program operations for Mach3 and the Legacy CNC Control Interface.

G-code Display & Controls

|	
	Cycle Start
	Feed Hold
	Cycle Stop
	Reset
Load Code Edit Code Close Code Rewind	Advanced Setup
Status	Run From

The G-code display screen will show the g-code currently loaded to run your machine. The G-code controls load, start, pause, stop, rewind, edit and close your g-code.

Work Offset Display

X	-1.0002
Y	-1.0002
Z	-0.3863
A	+0.0000
B	+0.0000
	Machine Coordinates
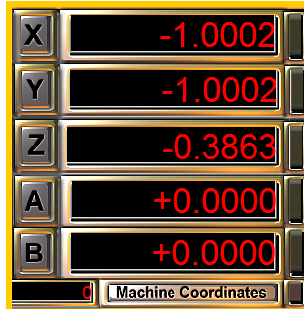
The work offset display shows the location of the spindle in reference to the Work Offset or Machine Coordinates.

Toolpath Display

Tool:0

Run Time 0:00:00:00 Jog Follow Regen Toolpath
File No File Loaded.
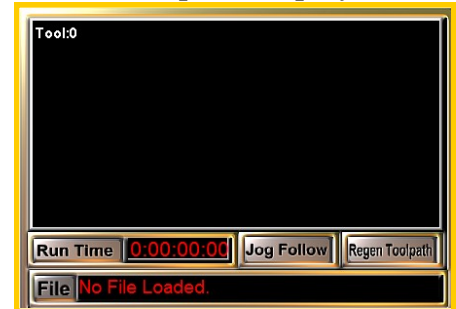
The Toolpath display shows a digital representation of the tool paths generated by the current g-code program loaded into the Legacy CNC Control Interface.

Active Mode Display

Active Modes Mill->G15 G80 G17 G40 G20 G90 G94 G54 G49 G99 G64 G97

The Active Modes display shows the active default g-code settings

Machine Set-up

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

The buttons found in the Machine Set-up area are used to set up the machine prior to running a g-code program

Auto Tool Change Controls

ARTISAN ATC CNC ORNAMENTAL MILL

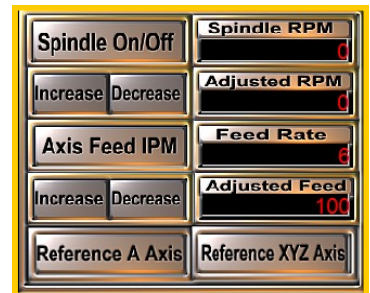| Manual Tool Release	Manual Tool Clamp
New Tool 0	Tool In Spindle 3

The Auto tool change controls and displays are only used when operating a Legacy CNC with the optional ATC spindle.

Jog controls (manual movement)

Axis Jog On/Off	Slow % 75
Cont Jog	Step Jog
.1000	.0100 .0010

The Jog Controls allow for manual control of each axis.

Spindle Speed & Feed Rate Controls

| Spindle On/Off	Spindle RPM 0
Increase Decrease	Adjusted RPM 0
Axis Feed IPM	Feed Rate 6
Increase Decrease	Adjusted Feed 100
Reference A Axis	Reference XYZ Axis

The Spindle Speed and Feed Rate controls allow for manual adjustments "on the fly" while the machine is in operation.

### Manual Data Input Line

Manual Data Input

The Manual Data Input line allows you to manually input g-code to move the machine as needed.
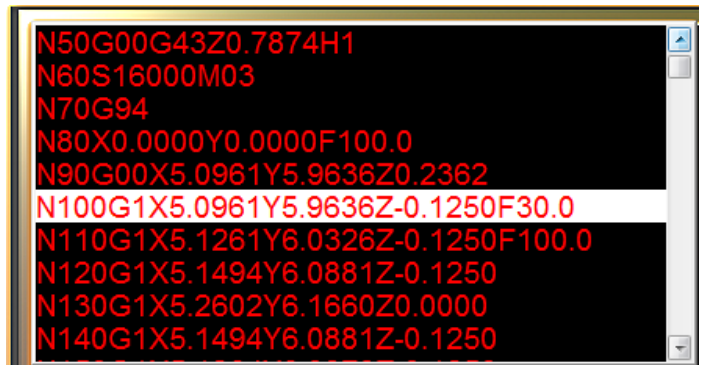
Advanced function drop-down menu bar
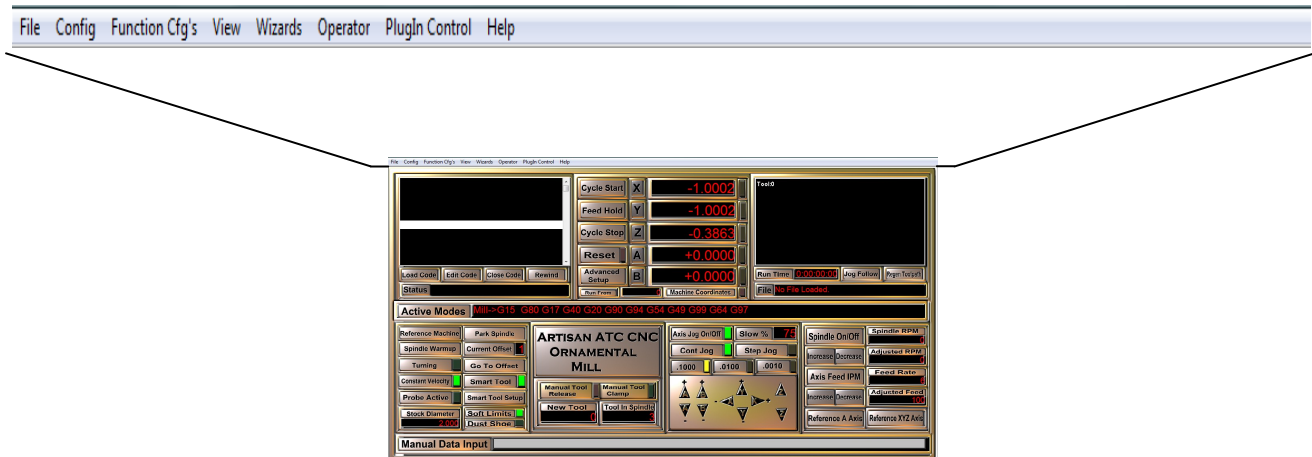
The top drop-down menu items are all advanced function items related to the Mach3 programming processes. These screens are used to set up the original settings and calibrations for each machine. Motor steps and frequencies, soft limits, homing switch locations and other machine set-up information can be found in these screens. **Changes should not be made to these values unless under the direction from a member of Legacy's customer support team.**

G-code Display & Controls
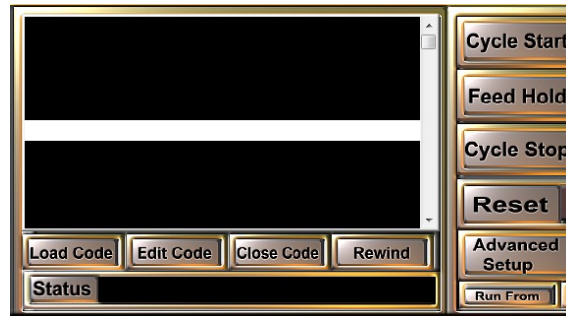
G-code Display Screen

The g-code display screen shows the currently loaded g-code program. The highlighted line (seen here in white) represents the line of code being read by the machine. You can use the up and down arrow keys on the right side of the screen to scroll through your program.


```
N50G00G43Z0.7874H1
N60S16000M03
N70G94
N80X0.0000Y0.0000F100.0
N90G00X5.0961Y5.9636Z0.2362
N100G1X5.0961Y5.9636Z-0.1250F30.0
N110G1X5.1261Y6.0326Z-0.1250F100.0
N120G1X5.1494Y6.0881Z-0.1250
N130G1X5.2602Y6.1660Z0.0000
N140G1X5.1494Y6.0881Z-0.1250
```
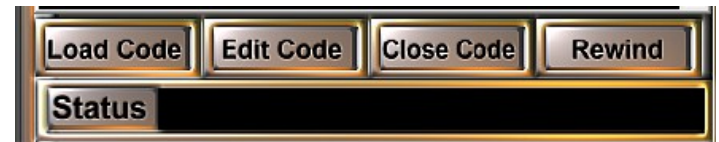
G-code control buttons

Load Code—loads a g-code file into the Legacy Control screen and checks for syntax errors. If g-code files are transferred to the control computer, these files should be copied to the C: drive prior to loading into the Legacy CNC Control Interface. Do not run a G-code file from a flash drive.

Edit Code—opens the loaded g-code into Notepad to allow editing. Closing Notepad will prompt you to save any changes made to the code file.

Close Code—closes the g-code file currently open in the display screen.

Rewind—Used to rewind the g-code file back to the beginning when the program has been interrupted.

Status - The Status line will display prompts, errors, code names and other information in regards to what the machine is doing or needs you to do.

Control Buttons

Cycle Start—Once a g-code program is loaded, you click on Cycle Start to begin the program or continue the program after a Feed Hold or Tool Change.

Feed Hold—This will pause a running program. There will be a slight delay before the program pauses. If a program is paused, the spindle/router will need to be shut off manually. You continue running a paused program by turning on the spindle and pressing Cycle Start.

Cycle Stop—This will stop a running program immediately. It is recommend that you click "Feed Hold" prior to clicking Cycle Stop—otherwise the machine will stop abruptly and may require you to run a "Reference Machine" sequence if steps are lost.

The best emergency stop button is the "Esc" key on your keyboard!

Reset—When the light is flashing, there is not information passing from the computer to the machine. Reset will be flashing each time Mach3 loader is opened. This will be the first button you click when starting Mach.

Advanced Setup—button takes you to the Advanced Setup page (information found on page 8)

Advanced Setup Screen

Many of the screen areas on the "Advanced Setup Screen' function the same as the on the main control screen. The Tool Path Display, most of the G-code Display and Control area, Reference Machine , Manual Tool release, Manual Tool Clamp, Manual Data Input, and Active Modes all function the same and will not be covered here.

To return to the main control screen, click the "Run Machine button.

This bank of DRO's are layout for diagnostic processes. The Probe and Home indicators are also listed on the main control screen.

Axis Calibration button takes you to the axis calibration screen. Each axis was calibrated as part of the machines original setup and testing process. If one the axes on your machine requires calibration, click the "Axis Calibration" button and follow the directions.

1. Choose the axis to calibrate.
2. Enter how far the axis should travel.
3. Measure then enter how far the axis traveled
4. The Legacy CNC Control software will automatically calculate the difference and recalibrate the axis. Accept new step calibration.
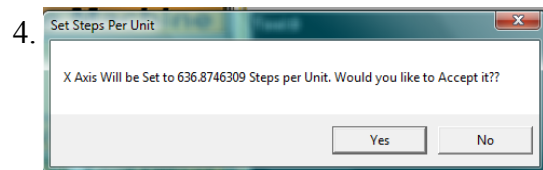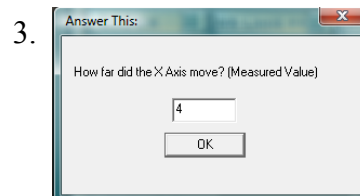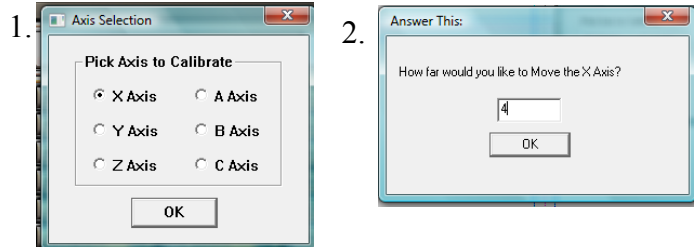
The Axis calibration has been set.

The "**A Axis Offset**" allows you to adjust the A-axis cutting diameter when turning diameters are cutting over or under size. The A Axis Offset uses the Smart Toll Stationary pad height in reference to the A-axis center location to make needed adjustments "on the fly" without having to calibrate the axis.

- If the diameter is to large: Increase the "A Axis Offset" by half of the oversize amount.
- If the diameter is to small: Decrease the "A Axis Offset" by half of the undersized amount.

Once this offset is fine tuned, you should not have to make additional adjustments.

These buttons have not been programmed. Please refer to the G-code section of this manual for G-code explanations. For Screen Help videos, please go to www.legacycncwoodworking.com.
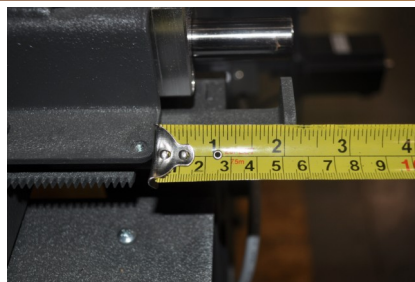
Axis Calibration:

Go to the Advanced Setup screen from the Mach3 Legacy CNC control software main page.
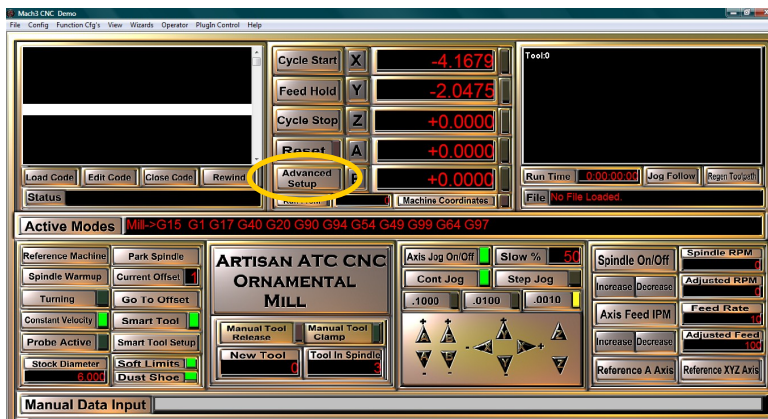
Using the jog controls, manually move axis being calibrated close to the home position. Measure the starting location based on a non moving set point.

For the X, we measure off of the stop bracket at the front end of the X-axis rail.
The pictured measurement reads 2 inches.

Click on the "Axis Calibration" button, a pop menu will appear.

Choose the axis you wish to calibrate and click OK.
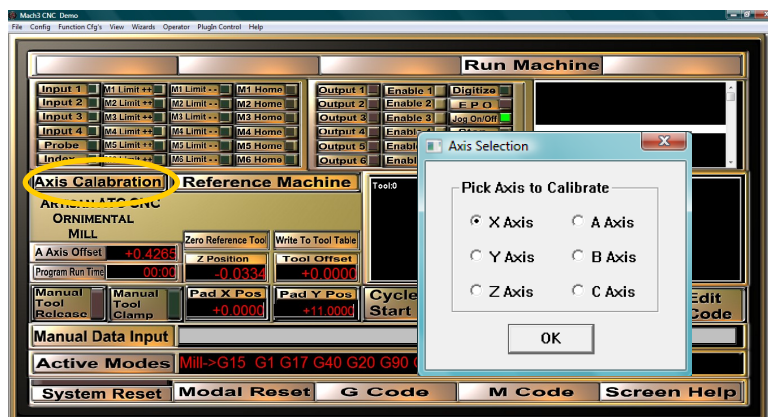
Answer the pop up questions—

"How far would you like to Move the axis?

Start with a short distance such as 2 inches. Do not exceed the total length of the axis.

Click OK.

Measure the distance the axis moved, using the same reference start point.

Enter the exact value in the next pop up screen. "How far did the Axis move? (Measure Value)
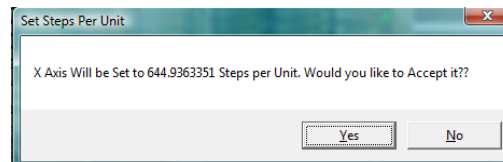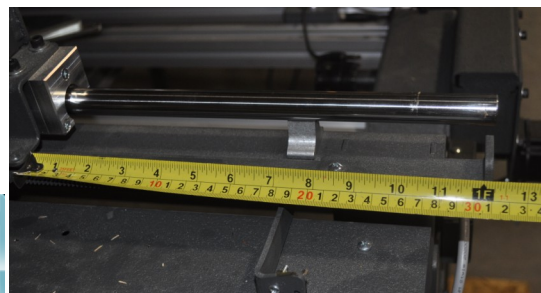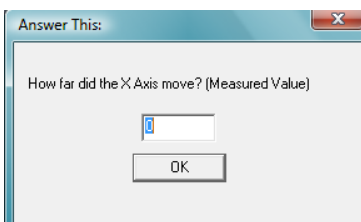
Click OK
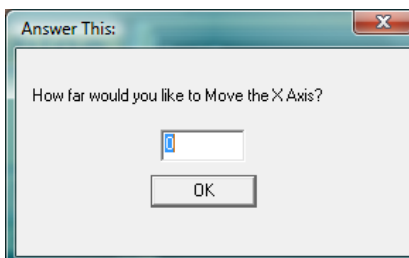
Steps per unit (inch) will automatically be set

Accept the reset.

Repeat process to verify steps, using longer distances until full axis length is traveled accurately.

Work Offset Display

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

Axis location Digital readout

Axis Buttons

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

Homing Switch indicator lights

Axis Buttons
Each Axis button acts as a label for the Axis location digital read-out line <u>and</u> when clicked acts as a zeroing button for that Axis.

When setting up a "Work Offset" it is important to zero the X and Y axes, creating a custom XY origin in reference to the current location of the spindle/router.

Axis location Digital Readout
Each Axis DRO (digital read-out) displays the distance the axis has moved from the set "Work Offset" location.
When the "Machine Coordinates" button has been clicked, it read the distance the axis is from the machine home coordinates.

Homing Switch indicator Lights
The homing switch indicator lights will come on when the axis homing switch is triggered. You will see each axis indicator light come on during the "Reference Machine" sequence. Do not initiate the "Reference Machine" sequence if one of these indicator lights is on—move the spindle/router off the switch.

Machine Coordinates—this button changes the Axis location DRO display from "Work Offset" to "Machine Coordinates".

Machine Coordinates

It is important to never run a program or initiate a "Smart Tool" command while the Machine Coordinates DRO's are displayed. Turn off the Machine Coordinates DRO by clicking the Machine Coordinates button.

Toolpath Display & Active Modes

Tool Path Display Screen
When a g-code program is loaded, Mach reads through the code and generates a tool path display onto the tool path display screen. The blue lines represent the cutting tool paths, the red line represent the spindle/router movement above the surface of the part. You can move, zoom and rotate this display using your mouse controls. Right mouse buttons to drag. Left mouse button to rotate. Mouse wheel to zoom.
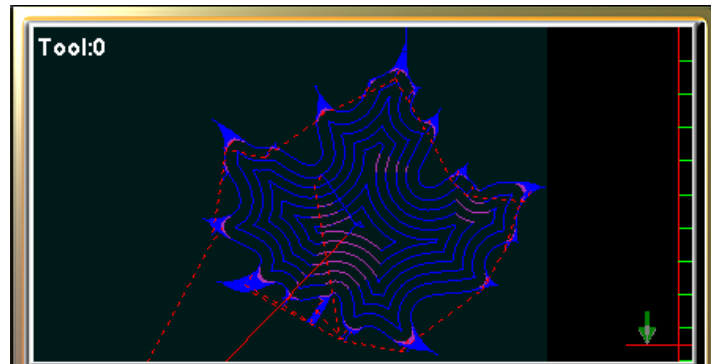
Run Time: displays the actual run time for the program running in Mach.

Jog Follow: Changes the view is the Tool Path display screen from an overall view to a "follow the router" view.

Regen Toolpath: Regenerating Toolpath prompts Mach to read through the loaded G-code and regenerate the tool path display.

File: Lists the loaded codes directory file name.
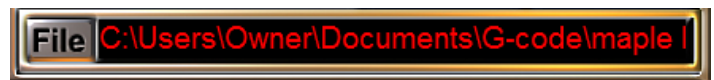
Active Modes: This display line will show the g-codes the machine is currently operating in. These pre-set operating g-codes will automatically be set each time the machine is turned on, guaranteeing the machine is set to operate at the optimal performance level. Loaded g-code programs will override this pre-set programming. The Active Modes button will reset the operating g-code back to the pre-set codes.

Machine Set-up

Reference Machine: when clicked, this 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 axis location creating the Machine "Home" Coordinates.

Spindle Warmup: When a air cooled or H2O spindle has been installed, this button has been preprogrammed to run the factory specified warm up process, which takes 7—11 minutes. It is recommend you run the Spindle Warm once the spindle has sat idle for more than 24 hours.

Turning: Clicking this button will change the work offset location between the turning offset location #6 and the flat stock offset location #1. The DRO will be on when the turning location is in operation.

Constant Velocity: Changes the Active Modes G-code line from the default G64 to G61. Read about Constant Velocity in the G-code definitions library.

Probe Active: Is a function DRO. When the axis homing switch is activated, the DRO will light. You can also check the circuit on the Smart Tool Mobil pad by touching it to the cutter head which should activate the Probe Active DRO.

Stock Diameter: The field allow for manual entry of the stock diameter being milled in the A axis offset location. It is important to give Mach an average cutting diameter for the blank being placed into the machine. This allows Mach to create a safe A-axis rotation speed based on diameter of part and programmed feed rate.

Park Spindle: When clicked, will drive the X, Y & Z axes to within 1/2" of the homing switches. **"Parking Spindle" at the end of each day**.

Current Offset: This DRO displays the current work offset.

Go To Offset: When clicked, this button will initiate a rapid move to the current offset location. Caution is needed when using the "Go To Offset" button. It is recommended you "0" the Z-axis before clicking.

Smart Tool: this button turns the Smart Tool on and off. The DRO is lit when Smart Tool is on—for more details on Smart Tool refer to page 16

Smart Tool Setup: When clicked, this button will initiate the Smart Tool Setup sequence, used to set the Z axis to "0" on flat stock.

Soft Limits: This button turns the Soft Limits on and off. The DRO is lit when the Soft Limits are on.

Dust Shoe: This button turns the Dust Shoe on and off. The DRO is lit when the Dust Shoe is on. When lit, Mach has been programmed to pause during the Tool Change process for the removal and replacement of the optional dust shoe. Dust Shoe available at www.kentCNC.net

Auto Tool Change Controls

Manual Tool Release: This button will manually release the tool from the spindle—be prepared to catch the tool holder, it will drop out.

Manual Tool Clamp: This button will manually clamp the tool holder into the spindle. When placing a tool manually into the machine, make sure the Tool in Spindle display reads the empty tool port location.

New Tool: This DRO displays the next tool programmed for pick-up listed in the g-code program being run.

Tool in Spindle: This DRO displays the tool location number for the tool currently in the spindle. You can also type in a tool number by double clicking on the display screen.
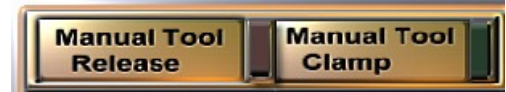
Jog controls
(manual movement)
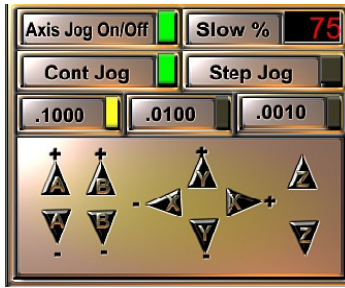
Axis Jog On/Off: turns the jog controls on or off. The DRO it lit when on.

Slow%: You can choose to manually jog your machine. A G0 command code will still drive at full speed.

Cont Jog: Continues Jog—when lit, the machine will run at max speed along the axis when the corresponding arrow is pressed, continuing to move until the arrow button is released.

Step Jog: when lit, changes the manual movement of the machine to individual steps. Step sizes are .1000, .0100 and .0001 of an inch. You will move one step to one click of the axis arrow

The DRO will be lit to the right of the chosen step size button.

The "Step Jog" keyboard short cut is the Control (Ctrl) key. Hold down the control key while using any of the axes arrow button to change from Continues Jog to Step Jog.

You can use your mouse to click the on screen Jog arrows. Or choose to use the corresponding keyboard controls.

Keyboard controls

The keyboard manual jog controls are layout as follows:
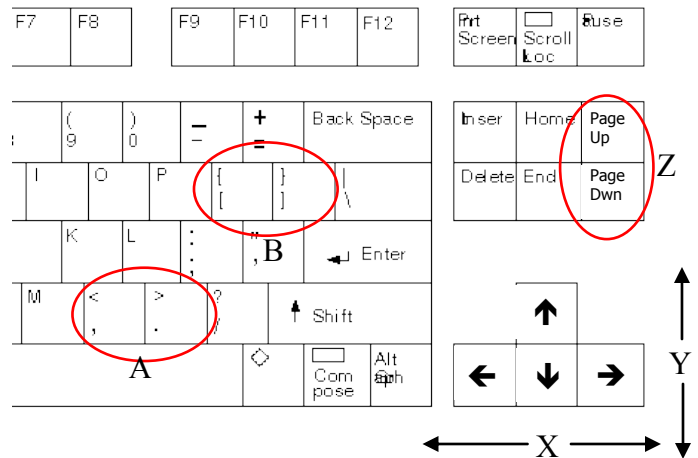X-axis - Arrows right and left
Y-axis - Arrows up and down
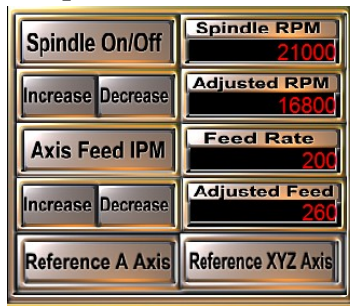Z-axis - Page Up is up; Page Down is down
A-axis - less than (<) is clockwise;
     greater than (>) is counter clockwise.
B-axis - Left bracket is down, Right bracket is up

Spindle Speed & Feed Rate Controls

Spindle On/Off: This button allows you to manually turn your spindle on or off.

Increase/Decrease: These buttons allow you to manually increase or decrease the spindle speed.   This can be done while a program is running.

Axis Feed IPM: Axis Feed **I**nches **P**er **M**inute is often referred to as the feedrate. This being, how fast is the cutter moving across the work piece. This is a label only.

Increase/Decrease: These buttons allow you to manually increase or decrease the Axis Feed IPM while a program is running in 10% increments.

Spindle RPM: This DRO screen shows the programmed spindle speed found in the g-code program currently being run. You can also manually type in a spindle speed and click spindle on to manually start the spindle.

Adjusted RPM: This DRO screen displays the actual Spindle speed when increases or decreases have been applied to the programmed spindle RPMs.

Feed Rate: The DRO screen displays the programmed Feed Rate found in the g-code program currently being run.

Adjusted Feed: This DRO screen displays the percentage of the original feed rate.

Reference XYZ Axis: allows you to reference just the X, Y and Z axes. This is useful when a table has been mounted on the bed rails making a B reference impossible.


Manual Data Input Line

As you become more comfortable with G-code, you will be able to manually move the machine by typing the g-code commands into the Manual Data Input line.

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.
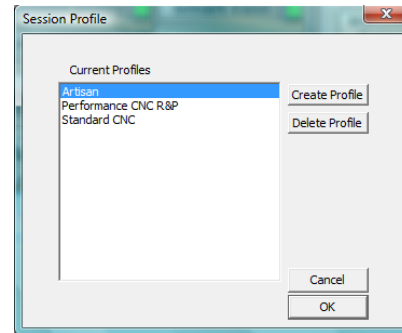
Turning

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.

Setting Up Work Offsets for Flat Stock
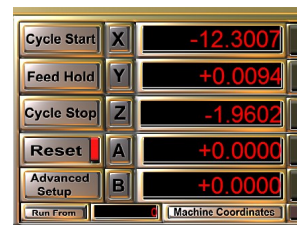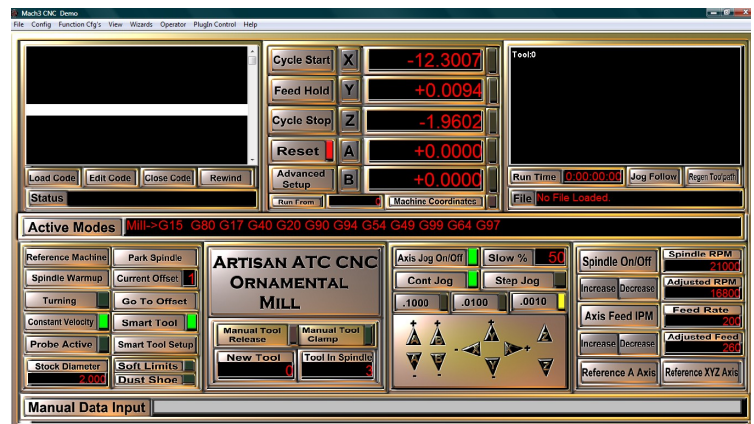
A "Work Offset" is a location where "work" is performed. This location is "offset" from the home coordinates. When setting work offsets, you must refer to the XY origin selected in the design process. The XY origin is the point where X & Y converge, or the X zero and Y zero point. This XY origin must 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
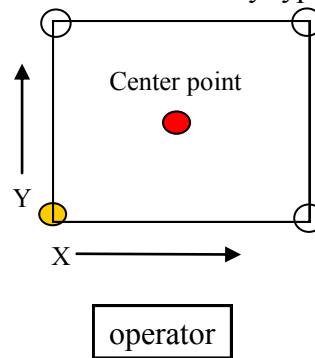
Computer Drawing Orientation

Headstock

operator

Center point
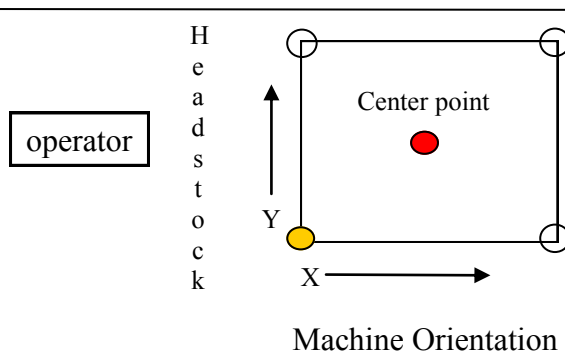
Machine Orientation

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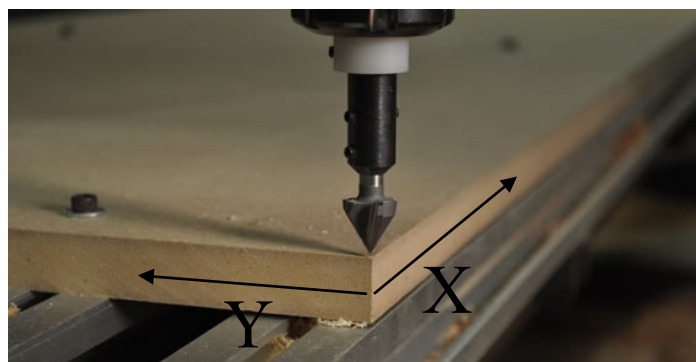
Verify that the "Current Offset" reads 1, then zero the X and Y axis DRO's by clicking the X and Y axis buttons.

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

Smart Tool

This feature automatically sets the Z axis when changing cutters, elming process is placed within the Mach control software and activated through the Legacy CNC Control Interface. When "ON", this program will automatically set the Z axis height for each tool, eliminating the need to touch off your tools or use tool tables. The two physical components of the Smart Tool process are the Smart Tool Stationary Pad and the Smart Tool Mobile Pad, both standard on the Legacy CNC mills. The Mini Arty comes standard with the Mobile Pad only, which acts as both Stationary and Mobile Pad.

Smart Tool Stationary Pad

The Stationary Pad has a fixed location. The location is set in the Advanced Setup screen. When "Smart Tool" is on, the control software will run the preprogrammed Smart Tool touch-off sequence as part of the M6 (tool change) G-code. The touch off sequence measures the tool by slowly lowering the tool to the stationary pad surface. When the metallic surface of the pad and the metallic surface of the tool come into contact, the needed circuit is completed and a measurement is taken at that precise location. Once a measurement is taken, the software automatically calculates the height of the bit, refers that height to the Z "0" set for the work location and runs the G-code program.

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 surface of flat stock material. 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.

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.

Machine Set-up

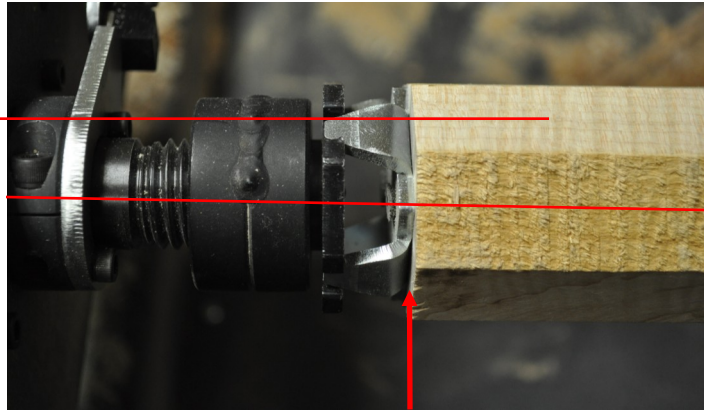Smart Tool
Stationary Pad

Smart Tool
Mobile Pad

Machine Set-up

Setting Up Work Offset for Turning

The Turning Offset location has been preprogrammed into the Legacy CNC Control software for the location diagramed below.

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 Index hub are used

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 Off-set 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
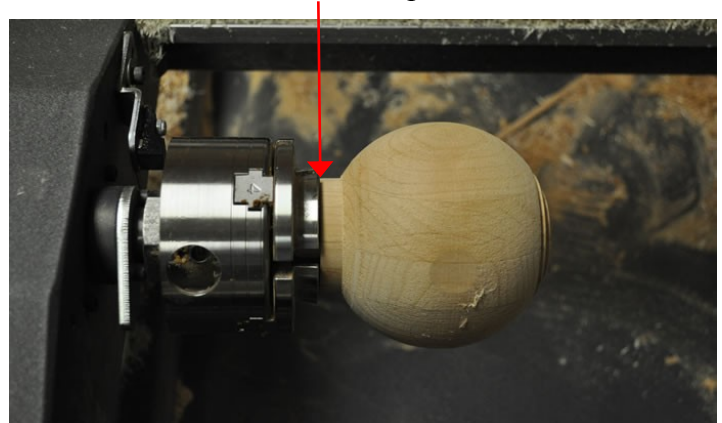
Work Offset Display

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

Tool Touch Off Method – Part

Touching off on the part allows the operator to manually bring the tool to the part surface to set the Z-axis "0" to the surface of the part while simultaneously measuring the height of the tool .

Benefits in using the Part touch off method:

   Allows you to mill exact plunge depths without entering the diameter of the part.

   Allows you to run the same program on different diameter or thicknesses of parts.

   When Surface Planing - Part touch off allows you to resurface the same part without having to rewrite the program.

The disadvantage is you have to touch off the part every time the program is run.

Conversational CAM will write a pause for touching off the part within the generated code when "Part" is chosen as the tool touch off method. After the tool change, the spindle will move to the XY origin of the part for the touch off sequence. After bringing the tool to the surface of the part, click "cycle start" to set the Z-axis to "0" at that location and begin the milling process.

Other CAM programs will not write a pause for touching off the part during the program. You will need to touch off the part and manually set the Z-axis to "0" prior to running the program.

Note: Remember to click "Smart Tool Off" when setting up a milling process using "Part" as the tool touch off method.

You can touch off on both flat stock and turned parts. To increase your touch off accuracy, flat plane or turn round your work stock as the first step in your milling process.

To touch off "Part"

Using the manual jog controls, move the cutter to the surface of the part. It is important to use the "Step Jog" feature to allow small controlled movements. Remember the "control key" is the Step Jog shortcut key on your keyboard - while pressed, you will move in the set Step Jog mode.

Visual accuracy -

   1. Bring the tool down to within .125" of the surface of the part.

   2. Using the Jog mode, lower the tool to the surface of the part.

   3. Click "Cycle Start" if using a Conversational CAM during a program or "0" your Z-axis if using other design programs for set up before starting a program.

If a closer tolerance is required, we recommend using a piece of paper to achieve tolerances of .003 - .004".

   1. Bring the tool down to within .125" of the surface of the part.

   2. Place a piece of paper between the cutter tip and wood and continue sliding the paper gently back and forth while slowly lowering the cutter.

   3. You will begin to feel the paper drag then catch against the cutter. It is important to use the Jog Controls to lower the spindle. Stop when the paper can no longer move back and forth.

   4. Click "Cycle Start" if using a Conversational CAM program or "0" your Z-axis if using other design programs.
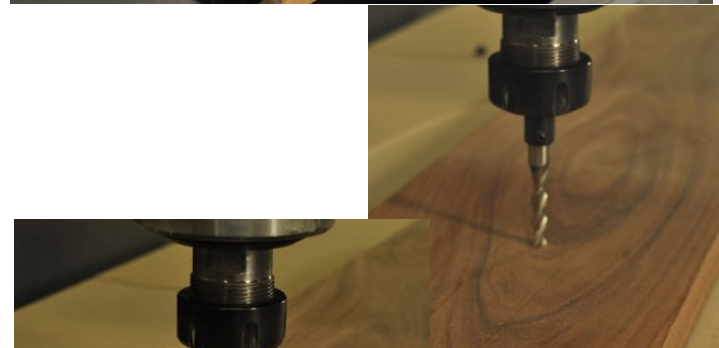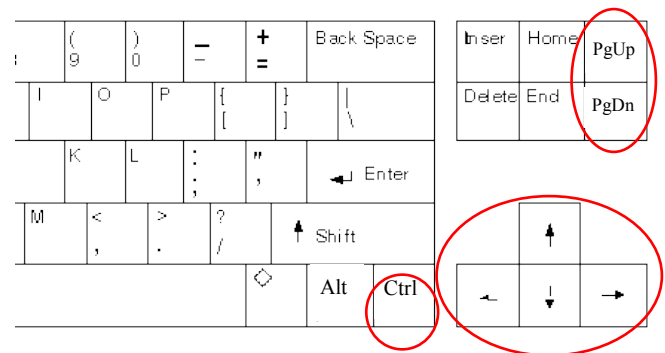
Tool Touch Off Method – Bull Nose

The Bull Nose touch off method was designed for the CNC system that does not have Smart Tool installed and is used to set the Z"0" when turning between centers. The Bull Nose center is a 1.4"diam. x 1.25" Delrin center designed to be placed over the point of the tailstock shaft.

When using the Bull Nose in the tool touch off method the Bull Nose is placed on the tailstock shaft and the cutter is brought down to the Bull Nose surface to establish the Z "0" for the A-axis.

Using Bull Nose with Conversational CAM programs

When using "Bull Nose" as the touch off method in Conversational CAM, code will automatically generate to run the "Bull Nose" touch off sequence.

1 - Once the code is loaded into the OM5 Control Software, and "Cycle Start" is clicked, a prompt will appear in the message screen saying "insert and touch off tool 1" (the tool number will reflect the tool you specified while programming this part).

2 - Insert the correct tool and bring the cutter tip to the surface of the Bull Nose. Remember to us the jog control mode to move the spindle. Once the cutter tip is lightly touching the Bull Nose, click "Cycle Start".

3 - The Z position will automatically be written to .7", which is the radius of the Bull Nose. The program will continue and finish as programmed.

Bull Nose with other CAM programs

All other CAM programs will not generate code to automatically touch off the Bull Nose when the program is running. The touch off sequence will have to be performed manually during the tool change sequence of the program.

1 - At the tool change prompt, insert the correct tool.

2 - Bring the tool down to the Bull Nose surface using the Jog mode controls.

3 - With the cutter tip lightly touching the Bull Nose, reset the "Z-axis Position" to .7".

4 - Raise the spindle and turn the router on - click "Cycle Start". The program will continue and finish as programmed.

Bull Nose Center

Trouble Shooting

Occurrence	Possible Cause	Corrective Action
Clicking noise from motors or running into a physical obstacle .	One of the motors was driven into the physical limits.	Bring the motor back within the proximity switch limits. Run "Reference Machine".
When running "Reference Machine" one of the axes runs the wrong direction away from the proximity switch.	One of the proximity switches was enabled when "Reference Machine" was activated.	Move all axes off the proximity switches and run "Reference Machine" again. You can check if any of the proximity switches are engaged by checking to see if the M1 thru M5 Home DROs are lit on the Advanced Setup 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 reset machine, no movement	1—Emergency Stop Button is engaged. 2—No communication with the Smooth Stepper	Release the Emergency Stop Button Power off the computer and control tower and restart.
Error: Can not run G53 incremental	You are in the G91 mode	Press Reset to clear the message, type G90 in the Manual Data Input line, press enter.

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 <u>and</u> setting of the machine work offset. Five locations are available center, top left, top right, bottom left, bottom right.
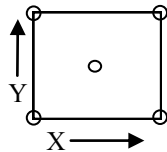
G-code reference sheet

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

M-codes manage the machine

M0: Program stop

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: Unite per minute
G98: Rapid Height by Z height
G99: Rapid Height by R height

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