Tool Room Mill

Operator's Addendum

96-0041 revB 2/02

Warranty Certificate

Covering Haas Automation, Inc., CNC Equipment
Effective June 1, 2000

LIMITED WARRANTY COVERAGE

All new Haas Tool Room Mills are warranted exclusively by Haas Automation's limited warranty as follows:

Each Haas Tool Room Mill CNC machine and its components (except those listed below under limits and exclusions) is warranted against defects in material and workmanship for a period of six (6) months from the date of installation.

Date of purchase is the date that a particular machine is installed at the end user.

An additional six (6) month warranty extension may be purchased from your authorized Haas distributor.

LIMITS and EXCLUSIONS

Components subject to wear during normal use and over time such as paint, window finish and condition, light bulbs, seals, etc., are excluded from this warranty.

Factory-specified maintenance procedures must be adhered to and recorded in order to maintain this warranty.

This warranty is void if the unit is subjected to misuse, neglect, accident, improper installation or application, or if the unit was improperly repaired by the customer or by an unauthorized service technician. Warranty service is available from your authorized Haas distributor.

Haas Automation is not responsible for any additional or incidental damage to parts, fixtures, machines, loss of time, or any other incidental damages that may be caused by a malfunction.

This warranty is transferrable from the original end-user to another party if the machine is sold via private sale before the end of the warranty period.

NOTE: Should you have a problem with your machine, please consult your owner's manual first. If this does not resolve the problem, call your authorized Haas distributor. As a final solution, call Haas directly at the number indicated below.

Haas Automation, Inc.
2800 Sturgis Road
Oxnard, CA 93030-8933
Phone: (805) 278-1800
FAX: (805) 278-8561

Haas Automation Inc.

WARRANTY REGISTRATION CERTIFICATE

The HAAS Tool Room Mill Machining Centers are warranted exclusively by Haas Automation's limited warranty as follows:

Each Haas Tool Room Mill CNC machine and its components (except those listed below under limits and exclusions) is warranted against defects in material and workmanship for a period of six (6) months from the date of installation.

Date of installation is the date that a particular machine is installed at the end user.

An additional six (6) month warranty extension may be purchased from your authorized Haas distributor.

LIMITS and EXCLUSIONS

Components subject to wear during normal use and over time such as paint, window finish and condition, light bulbs, seals, etc., are excluded from this warranty.

Factory-specified maintenance procedures must be adhered to and recorded in order to maintain this warranty.

This warranty is void if the unit is subjected to misuse, neglect, accident, improper installation or application, or if the unit was improperly repaired by the customer or by an unauthorized service technician. Warranty service is available from your authorized Haas distributor.

Haas Automation is not responsible for any additional or incidental damage to parts, fixtures, machines or loss of time that may be caused by a malfunction.

This warranty is transferrable from the original end-user to another party if the machine is sold via private sale before the end of the warranty period.

NOTE: Should you have a problem with your machine, please consult your owner's manual first. If this does not resolve the problem, call your authorized Haas distributor. As a final solution, call Haas directly at the number indicated below.

Haas Automation, Inc.
2800 Sturgis Road
Oxnard, California 93030-8933
Phone: (805) 278-1800
FAX: (805) 278-0861

In order to record the end user of this machine for updates and for product safety notices, we must have the machine registration returned immediately. Please fill out completely and mail to the above address to ATTENTION TOOL ROOM MILL REGISTRATIONS. **Please include a copy of your invoice** to validate your warranty date and to cover any additional options you may have purchased.

Company Name:________________________________ Contact Name: __________________________________

Address: ____

Dealer: ____

Date Installed:__________________________________

Model No. : ____________________________________ Serial Number: ________________________________

Telephone: (   )____________________________ FAX: ____

IMPORTANT NOTICE!!! PLEASE READ IMMEDIATELY!!!

This machine is equipped with an electronically-recorded serial number that cannot be altered. This is done to protect you in case of theft and to track machines when sold to other owners. After approximately 800 hours of use, the machine will automatically shut down if it has not been unlocked by Haas Automation. To unlock the machine, we must have the above registration with the serial number and the authorization from your dealer. You will receive a number from Haas that you will write in over the serial number on setting page (#26). The authorization from the dealer will come upon final acceptance of the machine. If, for any reason, the serial number of the machine is erased in memory, the machine will revert back to 200 hour limit for your protection.

1. INTRODUCTION

The Tool Room Mill includes features aimed at the machinist who is used to a manually positioned mill. These features implement the familiar quill stops and table stops, while giving full CNC capabilities. Manual handles on the Tool Room Mill give the machinist the feel of the cut, while the Index mode gives simple and powerful semi-automatic operation.

Position displays show the machinists where they are and where they are going. In addition, on-screen help is available as you scroll through the editable positions.

Quick-Start Guide

A Quick-Start Guide is included with the machine. Within this Quick Start Guide there are the 10 steps for a quick setup and machining operation. These 10 steps are:

1. Power On
2. Zeroing the Machine
3. Mounting the part
4. Operating modes
5. Cutting Tools
6. Setting the Work Coordinates
7. Tool Offsets
8. Programming
9. Trial Run
10. Machining a Part

Each of these steps is initially described in the Quick start guide. For more information on these steps refer to the Operator’s Manual. At the end of most steps there is reference to the specific section of the Operator’s manual that details the procedure.

The quick start guide gives specific keys that need to be pressed to accomplish the task. The Control Pad keys are listed in bold within brackets, for example <**RESET**>.

Safety

Read and Follow all safety warnings - Familiarize yourself with the safety section of the Operator’s Manual. Be aware of the other people around you in the shop; flying chips can seriously injure people, who may not be a safe distance away. Always wear safety glasses. Initial cuts/setups should be cut at a slower speed to reduce the possibility of tool or machine damage. As with any open frame mill, chip screens are highly recommended.

Specifications

Table

Travels: X - 30 inches,
Y - 12 inches,
Z - 16 inches.

Table: Length - 47.75 inches,
Width - 10.5 inches.

T-Slots: Width - 0.625 inches,
Center Distance - 4.00 inches (101 mm),
Quantity: 3 T-Slots.

Spindle Nose To Table: 4 inches to 20 inches.

Capacity: 300 lbs.

Spindle

Taper Size - #40 Taper,
Speed - 0-4000 RPM,
Transmission - Direct-Speed Belt Drive.

Spindle Motor Rating: 7.5HP peak

General

Feed Rates: Rapid on X-Axis - 200 ipm,
Rapid on Y-Axis - 200 ipm,
Rapid on Z-Axis - 200 ipm.

Maximum Cutting Feed Rate - 200 ipm.

Power Required: 3ph Power - 208V @ 25A 9KVA
1ph Power - 240V @ 40A 9KVA

Dimensions:

Operating Dimensions: Width - 90 inches,
Depth - 64 inches,
Height - 109 inches.
Weight - 3300 lb

Shipping Dimensions (Out-of-crate): Width - 69 inches,
Depth - 64 inches,
Height - 104 inches.

This section is a general overview of basic machining practices. It is intended to familiarize you with basic setup and operation techniques. Good machining practices extend tool life and in the end can save money.

Insert Selection

Although inserts are expendable it does not mean that an operator should be careless in his setup.

The following are the most common insert materials used. Each has a description of its characteristics and common usage.

High Speed Steel
- Allow for higher rake angles
- Resists chipping
- Resists softening due to high temperatures.

Carbide
- Good resistance to high temperatures
- Lower edge strength than high-speed steel
- Different composition of carbide can result in different finishes

Ceramic
- Yields good finish
- Requires negative rake angle due to low strength
- Requires very rigid setup
- Requires high horsepower

Diamond
- Four times harder than carbide
- Can retain their cutting edge for almost the length of the tool life
- Excellent stability for close tolerance work
- Excellent finish quality

NOTE: Remember to use the highest quality tooling designed for CNC machines to achive the best cutting condition possible.

Tool Wear

Tool life is dependant upon the following criteria:
- Cutting feedrate
- Tool and workpiece material
- How much material is being removed
- Proper workholding device
- Use of coolants
- Use of correct SFPM (RPM) for tool and material

Tools are subject to gradual wear from the following elements:
Abrasion (Friction and rubbing removes material from the cutter.) Caused by:
- Friction on the outside of the cutter as it passes through the material.

Adhesion High pressure and temperatures weld small chip particles to the cutter. Caused by:

- Low cutting speed
- High feed rate
- Negative cutting geometry
- 'Sticky' materials such as some stainless steels and pure aluminum
- Lack of coolants

Chipping The cutting edge is broken off instead of being worn away. Caused by:

- Excessive feed rate
- Interrupted cut
- Insert geometry too weak
- Chatter

Cratering Characterized by a smooth depression on the face of the insert. Caused by:

- Excessive cutting speed
- Ineffective use of coolant
- Friction
- Normal wear

Oxidation

- Occurs during very high cutting temperatures
- Weakens tool tips

Chemical wear A reaction between the cutter and the workpiece begins to corrode the insert (corrosion)

Coolant

There are a number of reasons why coolant is used in the machining process; it is used to dissipate heat generated during machining, reduces cutter friction, and promotes chip clearance. It also allows for high speed machining and increases tool life.

Coolant is not recommended when machining cast iron or steel, or when using carbide cutters. Carbide cutters can withstand high temperatures but cannot withstand the thermal shock of coolant.

Cutting fluids are best suited for soft materials such as aluminum alloys and brass.

A good flow of cutting fluid should be directed to both sides of the cutter whenever possible.

These are the most common types of cutting fluids:

Emulsion (water combined with mineral oils and additives)

- Used for light to moderate machining

Cutting oils (grease or solid additives)

- Limited to slow speed and low feed conditions due to flammability
- Expensive to use

Chemical or **semi-chemical fluids** (synthetic)

- Contain no petroleum oils
- Used for more difficult machining / grinding operations

Workpiece

The more you know about the workpiece, the better you can control the machining process. As a general rule ask these questions:

- What is the type of metal (alloy or steel)
- Has the part undergone any special process, i.e. case hardening, treated with additives or heat treated, etc?

Feed Rate

Feed rate is determined by the required surface finish and cutting force.
Expressed in:

- Inches or millimeters per minute
- Inches or millimeters per revolution
- Inches per tooth

Minimum chip thickness (chip load) is determined by the cutting force.
Maximum chip thickness (chip load) is determined upon the power of the machine and tool design.

Spindle Speed

RPM = speed at which the tooling is turning. The mill can be commanded in either clockwise (CW) or counterclockwise (CCW) direction. The type of application or style of tool will usually dictate the spindle direction.

Depth of Cut

The distance the cutter penetrates the workpiece, also referred to as chip load. This is determined by the following factors:

- Rigidity of the cutter and machine
- Machine capabilities
- Spindle horsepower

Machine Productivity

Use the load meter as an indicator of how the machine is cutting. Speeds and feeds should be adjusted after the initial cut. The initial cut will give an instant read-out of the performance of the machine. If adjustment are necessary, they should be made in 10% increments. Pay close attention to:

- Chip formation and color
- Chip load
- Monitor part and fixture during the cut
- Listen for any unusual noises

Application of Machining Conditions

Condition	Speed	Feed per tooth
Roughing	Decrease	Increase
Finishing	Increase	Decrease
End Milling	Increase	Decrease
Slotting	Increase	Decrease
Scale	Decrease	Increase
Tool Life	Decrease	Increase
Hard Material	Decrease	No Change
Soft Material	Increase	Increase
Heavy Depth of Cut	Decrease	Decrease

Surface Finish

A good finish depends on a number of variables. The following are a number of items to check to achieve a good finish:

- Good finish results in slower feeds and higher speeds
- Face milling produces the best finish
- Increasing the number of cutters (inserts) allow for a better finish
- Cuts should always be in the same direction
- Lighter depth of cuts will produce a better finish
- Coolants use can also affect part finish

Accuracy

Machine accuracy can be affected by a number of variables, such as:

Is the machine properly warmed up?
Holes should be center drilled first
Check the condition of the tooling

Cutting Tool Descriptions

Drill Used to create a cylindrical hole in a work piece. Drilled holes can be "through holes" or "blind holes". A "blind hole" is not cut entirely through a work piece.

Center drill A small drill with a pilot point. It is used to create a small hole with tapered walls. When a hole's location must be held to a close tolerance, use a center drill first and then use a regular drill to finish the hole. The tapered walls of the center-drilled hole will keep the regular drill straight when it begins to drill into the work piece.

Reamer Designed to remove a small amount of material from a drilled hole. The reamer can hold very close tolerance on the diameter of a hole, and give a superior surface finish. The hole **must be** drilled first, leaving .005 to .015 of an inch stock on the walls of the hole for the reamer to remove.

Tap Used to create screw threads inside of a drilled hole. NOTE: Care must be taken when using a milling machine to perform a tapping operation. For example, the spindle speed and feed must be synchronized.

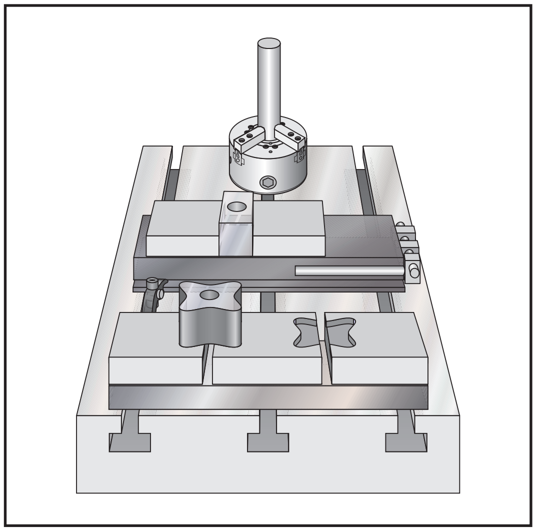
End Mill Shaped similar to a drill, but with a flat bottom (end). It is used primarily to cut with the side of the tool, to contour the shape of a work piece.

Bull End Mill A bull end mill is the same as a regular end mill except that there is a radius on the corner where the side meets the bottom. This radius can be up to ½ of the tool's diameter.

Ball End Mill A ball end mill is a bull end mill where the corner radius is exactly ½ the tool's diameter. This gives the tool a spherical shape at the end. It can be used to cut with side of the tool like an end mill.

Work Holding

Work holding is one of the most important elements of setting up any machine tool. Work holding is the method of clamping the work piece to the machine. The work piece must always be held securely before any cutting can take place. There are three basic types of work holding used in milling operations. They are as follows: a mill vise, clamps, and a chuck. The use of either type is dependant upon how large the cutting pressure on the workpiece is going to be. The maximum holding pressure of a manual clamp is determined by the strength of the operator. Large work holding forces require a pneumatic or hydraulic fixture.



Fixtures should be kept close to the center of the table in order to maintain a rigid setup. If placed at the ends of the table, harmonic vibrations could occur.

Before placing any type of work holding on your machine table, great care must be taken to be sure that the table is clean and free of chips and other debris. The work holding equipment also must be clean, free of debris, and have no burrs or dings that may cause instability or damage the table. If you plan to leave your work holding on the table for any length of time, a light coat of rust-preventative oil will help keep your table and work holding free of rust and corrosion.

The most common method of holding a work piece for machining is a mill vise. The vise is attached to the mill table using tee nuts and bolts. The tee nuts slide into the tee slots in the mill table and the bolts clamp the vise in position. Two bolts on either side of the vise hold it in place. For precision work, the vise must be set so that the clamping surfaces are parallel to the X or Y-axis of machine travel. This is done using an indicator.

To indicate a vise parallel to a machine axis, you will need an indicator and a magnetic base to hold it. Place the magnetic base anywhere on the bottom of the Z-axis head or the spindle housing. Jog the machine axis to bring the indicator tip to the clamping surface you want to indicate. Set the tip of the indicator so it begins to register on the indicator dial. Use the jog handle to move the axis you want the clamping surface to be parallel to and determine which direction the vise needs to be moved to become parallel. If the right side of the vise needs to be moved toward the back of the machine, tighten the bolt on the left side of the vise to be snug and leave the bolt on the right side of the vise loose. With Dead-blow Mallet, tap the vise until the clamping surface is parallel with the machine axis. Check the result by jogging the axis back and fourth. You may need to do this several times. When the vise is parallel, tighten all the bolts and check the set-up again. Adjust if necessary.

Another common type of work holding on a milling machine is clamps. If you have an odd shaped work piece or a large one that does not fit into a mill vise, you can clamp it directly to the mill table or fixture plate using clamps. Clamps are usually a bar type with an oval slot cut through the bar for a bolt and a tapped hole in the bar for a jackscrew. The jackscrew is set to be slightly longer than your work piece is tall. A small shim made of soft material .05” minimum thickness should be placed between the jackscrew and the machine table to prevent the screw from damaging the table when the clamp is tightened. Set the clamp on top of the work piece and the jackscrew and shim on the table. Place a bolt through the slot in the clamp and screw it into a tee nut in the table’s tee slot and tighten the bolt to increase the clamping pressure. A series of clamps around your part should hold it in place during machining. If you need to machine completely through the part, you will need to get the work piece off of the table. In this case, place blocks between your work piece and the table at the same locations where your clamps are. The blocks need to be directly under the clamps and all the blocks need to be the same height.

Another method of getting your work piece up off the table is to make a fixture plate. The fixture plate can be bolted to the machine table using tee nuts and bolts. Drill and tap holes where the clamps need to be. Clamp your part to the fixture plate as described above.

The third method of work holding is for round, cylindrical work pieces. A chuck with movable clamping jaws can be mounted to the machine table. The chuck works like the small chucks on a drill press or a drill motor. A chuck key is used to turn a screw in the side of the chuck, which moves all the clamping jaws simultaneously to clamp on a round work piece.

For information on other types of work holding or more information on the types discussed here, contact your local distributor of industrial supplies.

2. OPERATION

Haas has added new features to the control, targeted at manual and semi automatic operation of the Tool Room Mill. These include:

• Manual jogging on the X and Y-axis
• Jog Coordinate System
• Jog Travel Limits
• Jog index Distance
• Moving to coordinate system home

The tool room mill is equipped with a hand held safety switch. The button **must** be pressed any time automatic machining is taking place. Releasing the switch will cause the spindle and axes motion to stop. In order to resume automatic machining, the button and Cycle Start must be pressed (it is not necessary to hold the Cycle Start switch down).

Manual Jogging on the X and Y-axis

Shift –X:
Shift +X:
Shift –Y:
Shift +Y:

Toggle Manual jogging by pressing shift followed by the +X, -X or +Y, -Y key

This will release the servos for that axis. Both axes can be released at the same time. At any mode change, on any key jogging or change to the current handle jogging axis manual jogging will be turned off for both axes.

Jog Coordinate System

The Jog Coordinate System is displayed while in jog mode. This display displays the current machine position relative to the jog offsets. Simple methods for setting the origin of the jog coordinate system are provided.

The jog offset can be altered in the following ways while in jog mode and on the Jog position page (jog offsets are only used for X,Y,Z).

1) Press origin: Offsets will be set such that the current XYZ position is zero.

2) Press X, Y or Z followed by origin: This will zero out a single axis position.

3) Use the arrow keys to highlight X, Y or Z offset. Type in a number (be sure to use a decimal point for all numbers)
   a) Press WRITE and the number gets added to the current offset.
   b) Press F1 and the offset gets changed to the input number

Jog Travel Limits

Jog travel limits are available for X,Y, and Z. These limits are relative to the currently displayed in the "JOG POS" part of the coordinate system page. If you are on the page that displays all four positions it is relative to the Jog coordinate system

These limits are always active while in jog mode.

Set Max to the current offset +max travel (X=30, Y=12, Z=16) of the machine and Min to -max travel of the machine for them to be **inactive**.

To change the limits:

1) Use arrow keys to highlight X, Y or Z Max or min limit. Type in a number (be sure to use a decimal point for all numbers).
   a) Press WRITE and the number gets added to the current limit.
   b) Press F1 and the limit gets changed to the input number using the current jog offset.

2) Press DELETE and all limits are changed to +/- max travel

3) Press ALTER and the currently highlighted limit will be set to your current position.

Jog Index Distance

Jog indexing can be useful when cutting a pocket or a regular hole pattern. It allows one button press to move the selected axis a specific, repeatable distance. For example, when the X-axis jog index distance is set to 1.000, pressing the +X button will cause the mill to index the X-axis 1", each time the button is pressed.

To enter the jog indexing distance:

1) Highlight X, Y or Z index distance, Type in a number.
   a) Press WRITE and the number gets added to the current index distance.
   b) Press F1 and the index distance gets changed to the input number.

To Index an axis use the X, Y, and Z axes jog keys

1) Press shift followed by jog lock to place the machine in Index Jog Mode, then do any of the following:
   Press the +X key to index X in the + direction
   Press the -X key to index X in the - direction
   Press the +Y key to index Y in the + direction
   Press the -Y key to index Y in the - direction
   Press the +Z key to index Z in the + direction
   Press the -Z key to index Z in the - direction

NOTE: Jog indexing is limited by jog travel limits.

Feed Hold and Index Jogging

Pressing Feed Hold during an index jog will cause all axes motion of the Tool Room Mill to stop, however the spindle will continue to turn at the last programmed speed. To continue the cutting press the axis key for the same direction of the cut. Pressing the axis key for the opposite direction will cause the axis to move back to the previous starting location.

Pressing other index jog keys will display a message describing which axis is in feed hold.

Setting the Feed Rate

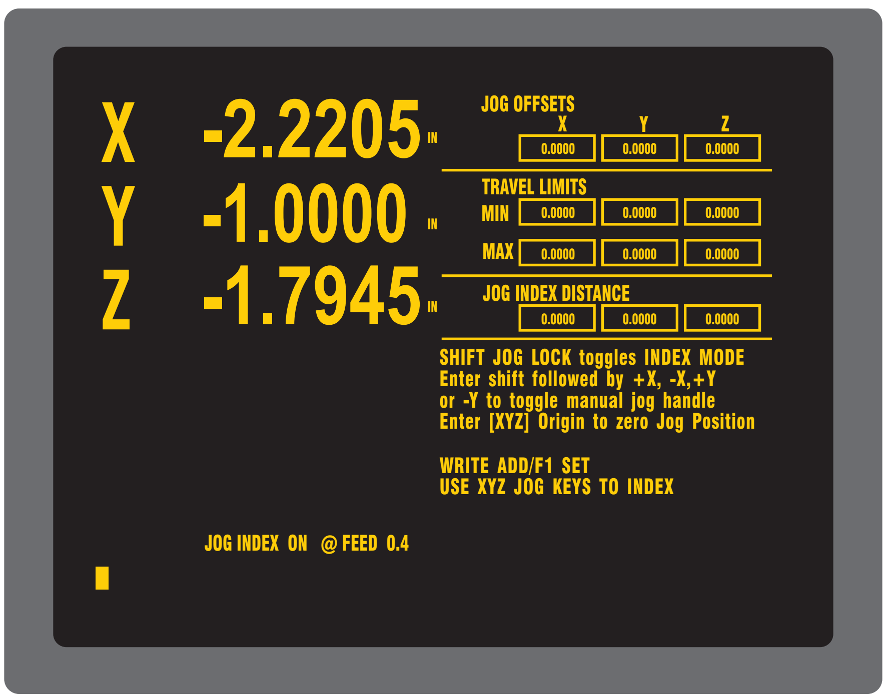
Any feed rate, up to the maximum, can be selected for Jog Lock and Jog Index by using the jog rate keys (.0001/.1, .001/1., .01/10, .1/100) in combination with the feed overrides.

Moving to Coordinate System Home

Pressing the HOME G28 will first move Z to Machine zero, then it will move X and Y to the current offset position.

Note: This feature is only possible when the machine is in jog mode and displaying a position page.

Set parameter 266,267 bit 7 DELAY AXIS 0 to 1 or else it will go to machine zero first.



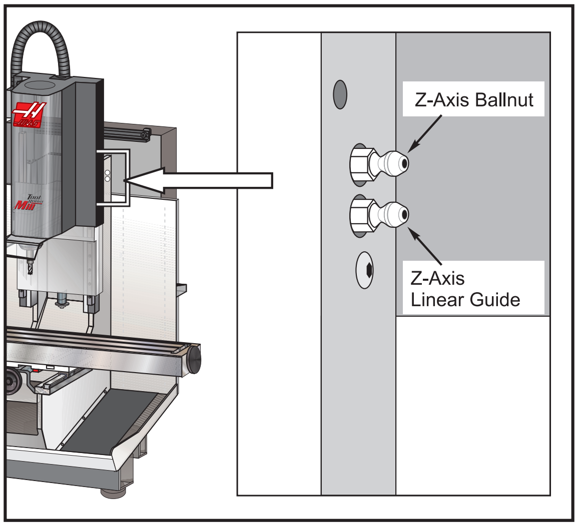
To set or change the travel limits:

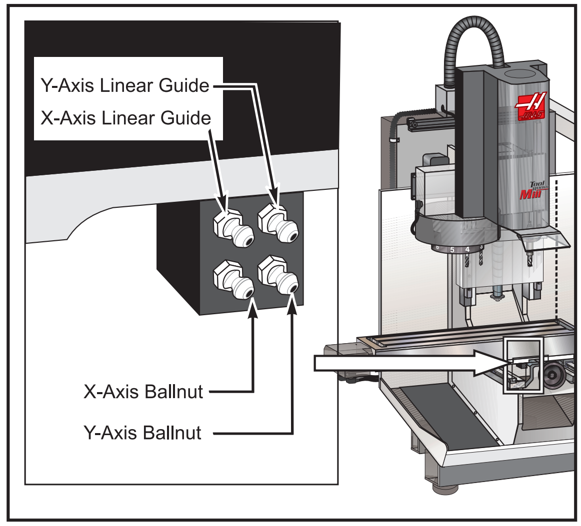
1) Highlight X, Y or Z Max. or Min. limit, and type in a number.
   - a) Press WRITE and the number gets added to the current limit.
   - b) Press F1 and the limit gets changed to the input number using the current jog offset.
2) Press DELETE and all limits are changed to +/- max travel.
3) Press ALTER and the currently highlighted limit will be set to your current position.

Maintenance

The Tool Room Mill must be lubricated manually. There are six lubrication fittings located on the Tool Room mill (see the following figures). To insure proper lubrication the X, Y and Z axes should be cycled daily and lubricated weekly, using a general-purpose lithium grease.

Lube until visible grease comes out of the ballnut and linear guide trucks.





Floppy Drive

Located on the side of the control pendant, the floppy drive allows easy uploading of machining programs using standard PC floppy disks. You can download to disk for backup storage or DNC directly off the disk.

BT Tooling

Your Toolroom Mill can be reconfigured to use your existing inventory of BT tooling, if you prefer.

4th-Axis

Boost productivity with a fully integrated Haas 4th-axis Rotary Table.

Coolant Pump

Supply coolant to your machine with an auxiliary coolant pump. This improves tool life and allows higher cutting speeds.

8 “M” Function

This option adds 8 additional outputs for each 8M option. The machine can be fitted with two 8M options for a total of 16 additional outputs. These outputs can be used to activate probes, auxiliary pumps or clamping devices etc. The 8M relay board contains 8 relay outputs (M21- M28) and 2 terminal strips P4 and P5. Each terminal strip has 12 positions which are Normally Open, Normally Closed and Common.

The Toolroom Mill comes with only two user inputs, M21 and M22.

Chip Guard

An optional chip guard shields the operator from errant chips and coolant. The sliding front doors open to give the operator plenty of space to load/unload parts and easy access to clean.