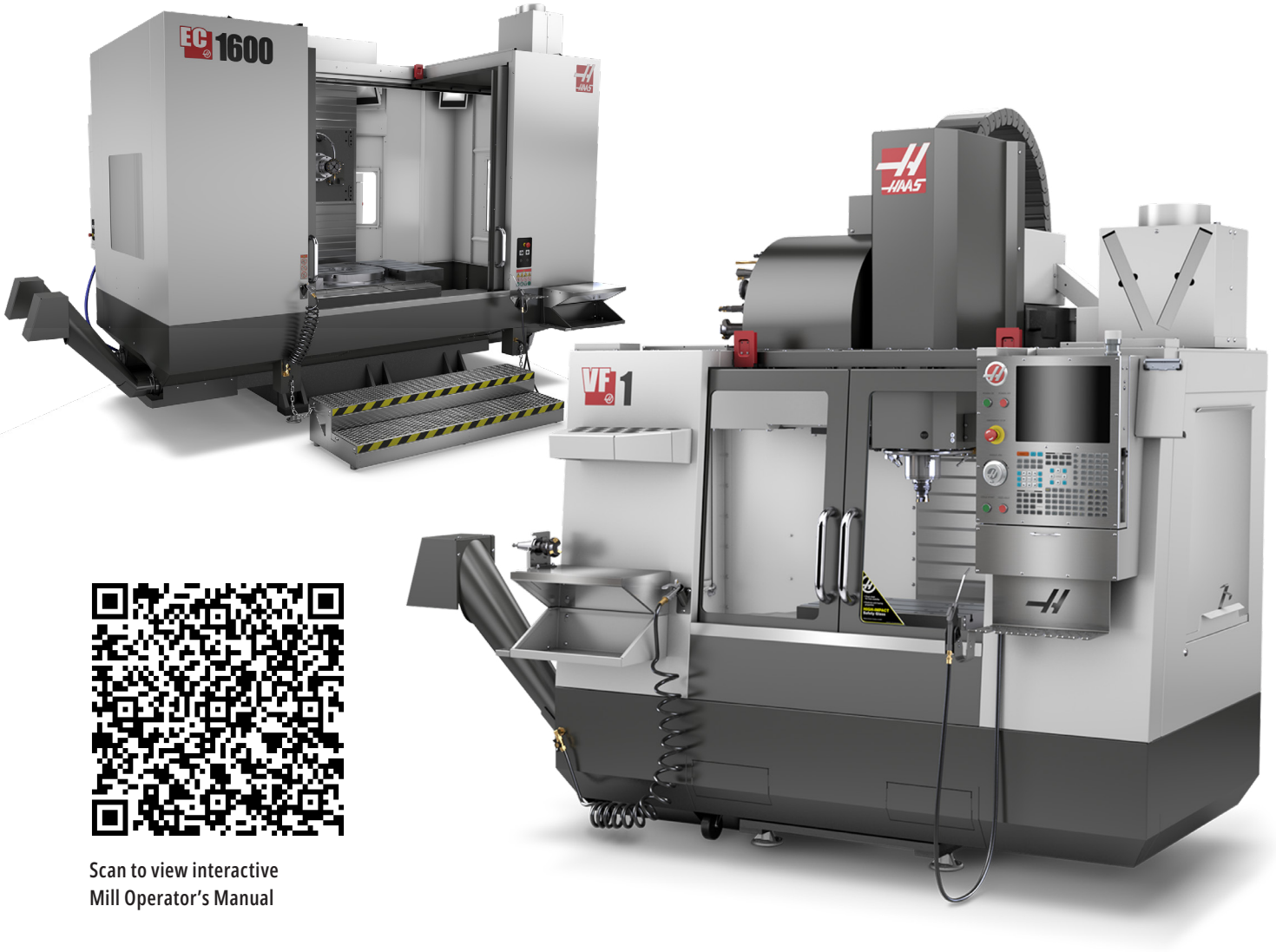
Original Instructions

2023 Mill Operator's Manual

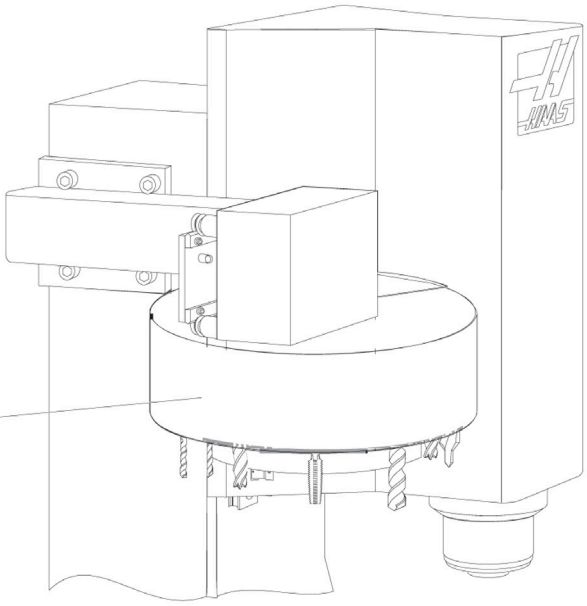
Features and functions of a Mill CNC machine.



Scan to view interactive
Mill Operator's Manual

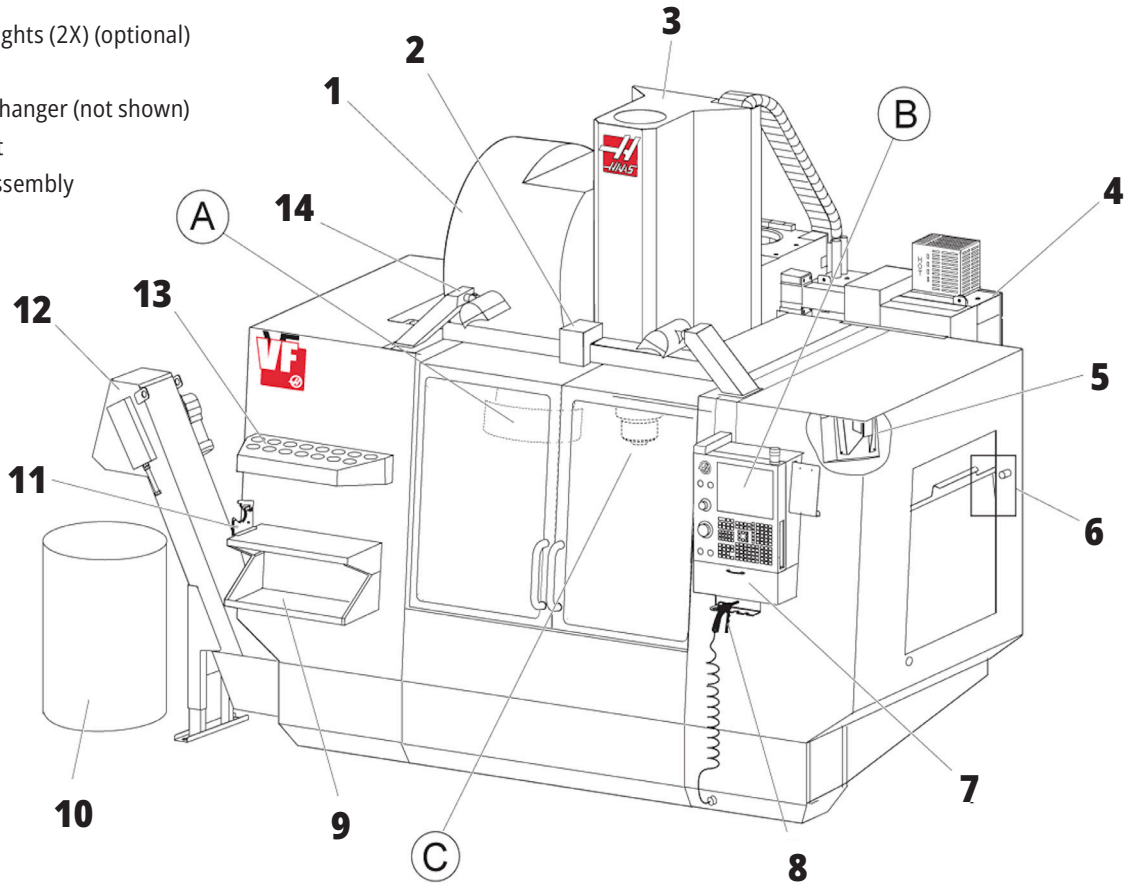
Vertical Mill - Features (Front View)

The following figures show some of the standard and optional features of your Haas Vertical Mill. Note that these figures are representative only; your machine's appearance may vary depending on the model and installed options.



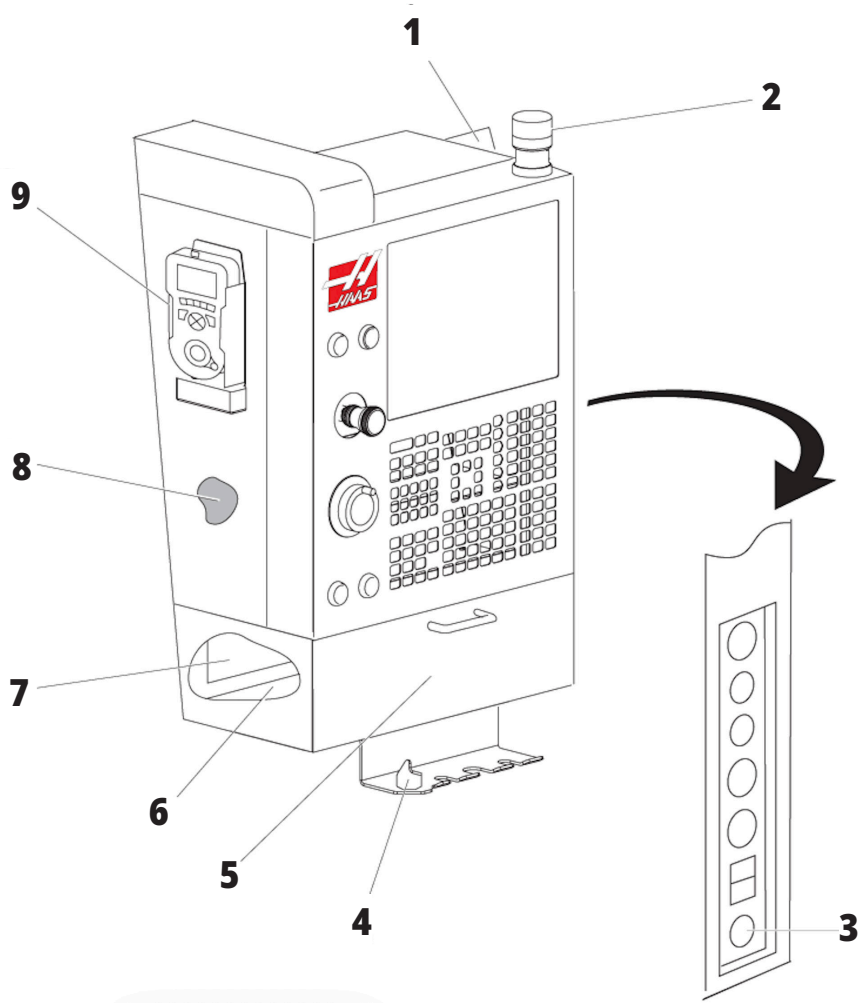
1. Side Mount Tool Changer (optional)
2. Auto Door (optional)
3. Spindle Assembly
4. Electrical Control Box
5. Work Light (2X)
6. Window Controls
7. Storage Tray
8. Air Gun
9. Front Work Table
10. Chip Container
11. Tool Holding Vise
12. Chip Conveyor (optional)
13. Tool Tray
14. High Intensity Lights (2X) (optional)

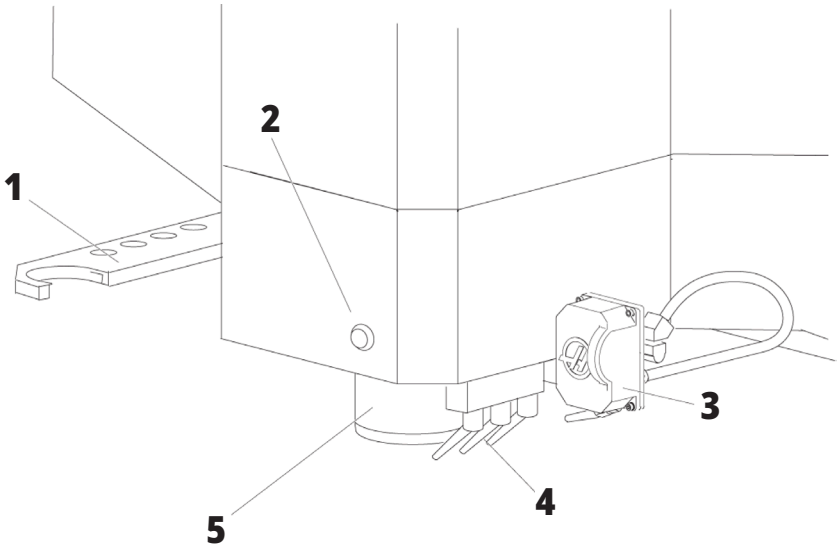
A. Umbrella Tool Changer (not shown)
B. Control Pendant
C. Spindle Head Assembly



Control Pendant

1. Clipboard
2. Work Beacon
3. Hold To Run (where equipped)
4. Vise Handle Holder
5. Storage Pull Down Access Door
6. Tool Tray
7. G- and M-code Reference List
8. Operator's Manual and Assembly Data (stored inside)
9. Remote Jog Handle





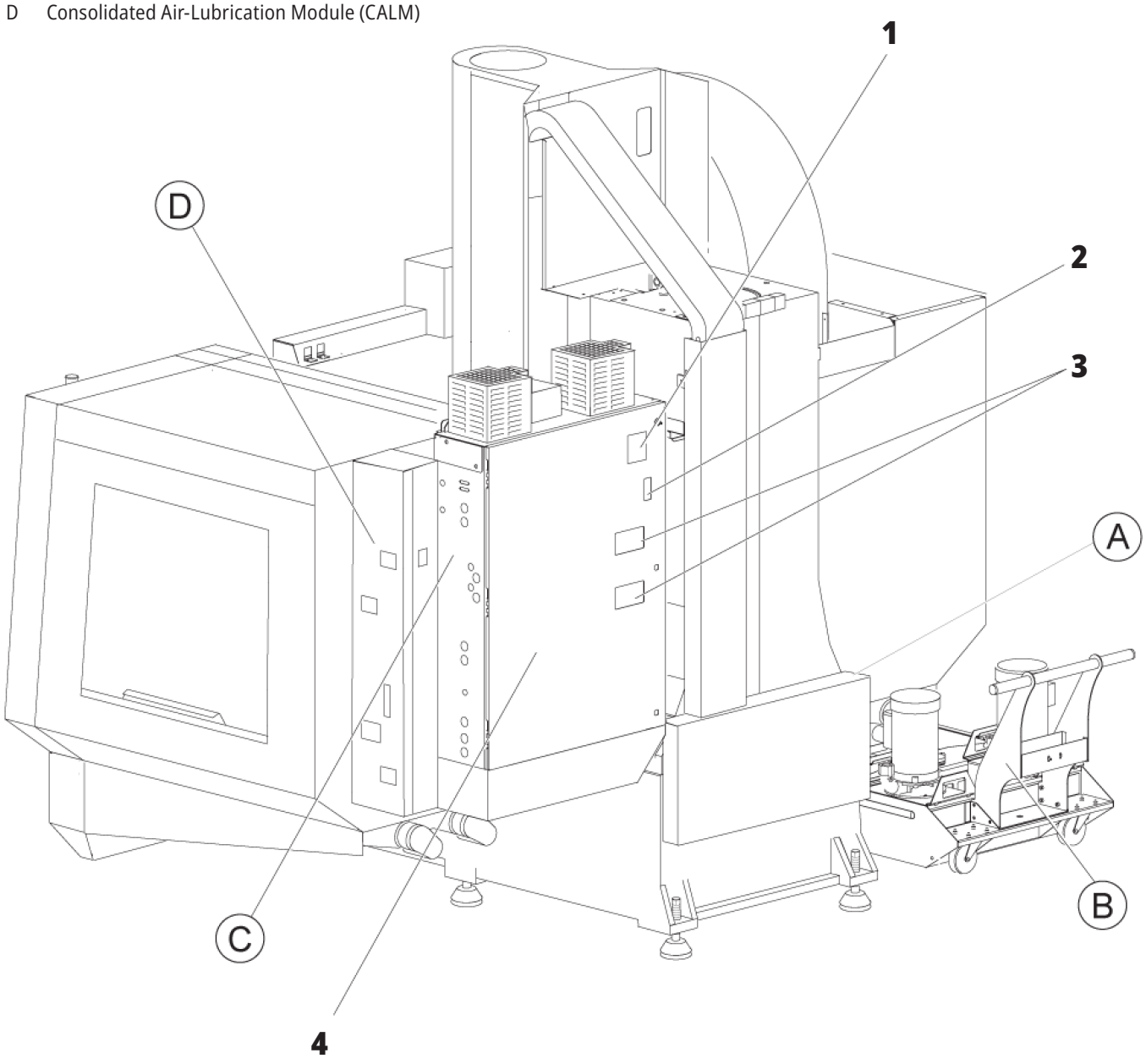
Spindle Head Assembly

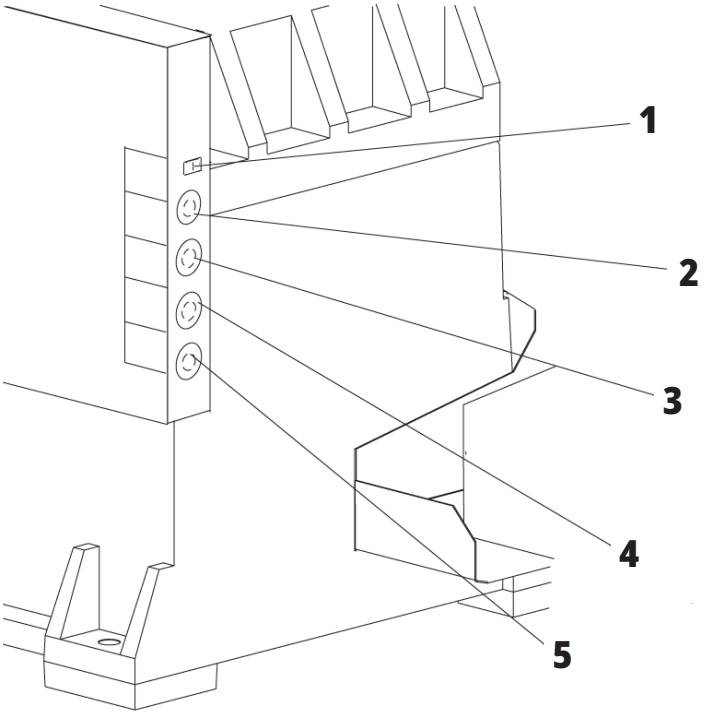
1. SMTC Double Arm (if equipped)
2. Tool Release Button
3. Programmable Coolant (optional)
4. Coolant Nozzles
5. Spindle

Vertical Mill Features (Rear View)

1. Data Plate
2. Main Circuit Breaker Switch
3. Vector Drive Fan (runs intermittently)
4. Control Cabinet

A Electrical Connectors
B Coolant Tank Assembly (movable)
C Electrical Control Cabinet Side Panel
D Consolidated Air-Lubrication Module (CALM)



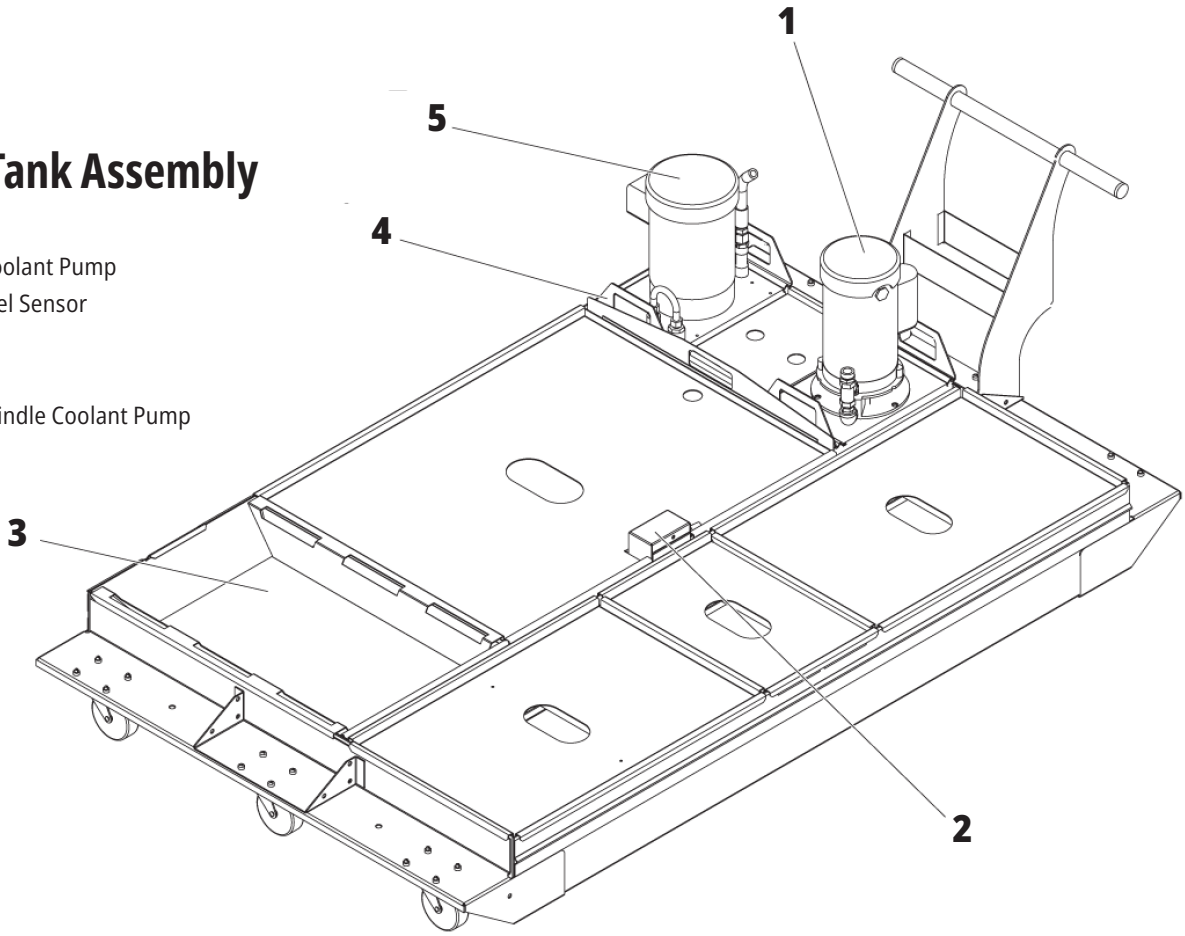


Electrical Connectors

1. Coolant Level Sensor
2. Coolant (Optional)
3. Auxiliary Coolant (Optional)
4. Washdown (Optional)
5. Conveyor (Optional)

Coolant Tank Assembly

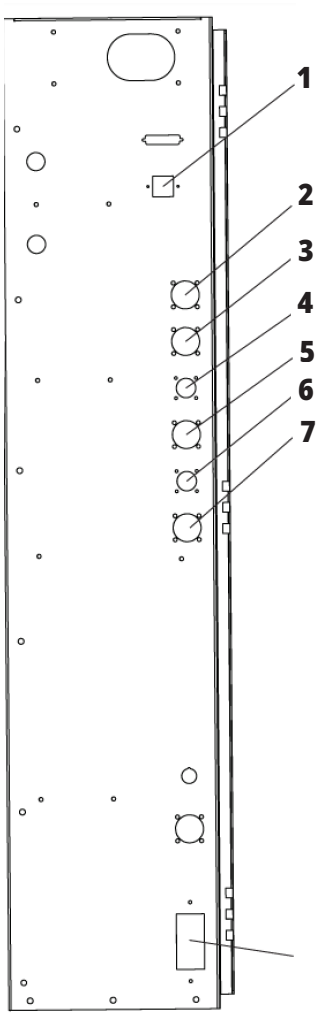
1. Standard Coolant Pump
2. Coolant Level Sensor
3. Chip Tray
4. Strainer
5. Through-Spindle Coolant Pump



1.2 | VERTICAL MILL OVERVIEW

Electrical Control Cabinet Side Panel

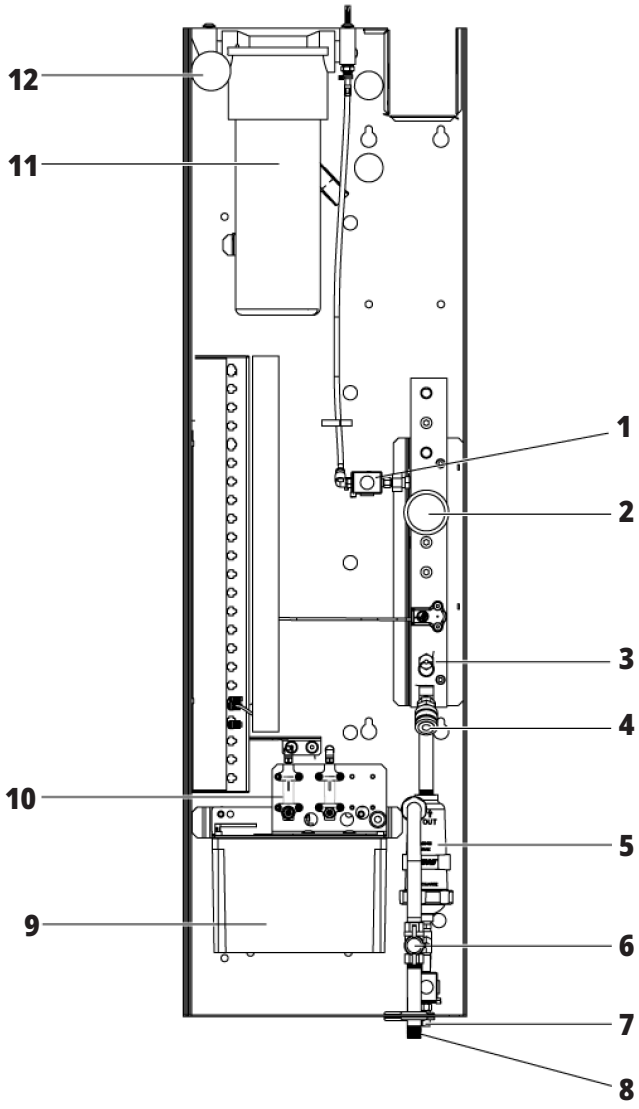
1. Ethernet (Optional)
2. A-Axis Scale (Optional)
3. B-Axis Scale (Optional)
4. A-Axis Power (Optional)
5. A-Axis Encoder (Optional)
6. B-Axis Power (Optional)
7. B-Axis Encoder (Optional)
8. 115 VAC @ 0.5A



Consolidated Air-Lubrication Module (CALM)

1. Min Lubrication Oil Solenoid
2. Air Pressure Gauge
3. Air Relief Valve
4. Rotary Table Air Supply
5. Air/Water Separator
6. Air Shut Off Valve
7. Purge Solenoid
8. Air Inlet Port
9. Spindle Lubrication Reservoir
10. Spindle Lubrication Sight Glass (2)
11. Axis Lubrication Oil Reservoir
12. Oil Pressure Gauge

NOTE: More details are shown on the decals inside of the access door.



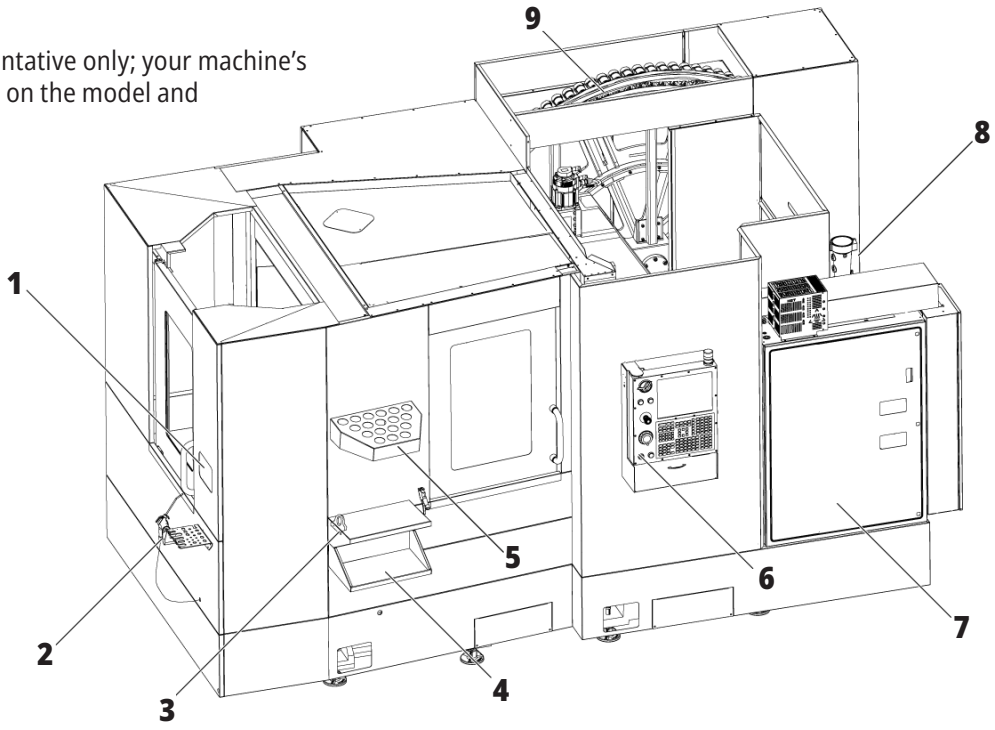
EC-400, EC500 Overview

The following figures show some of the standard and optional features of your EC-400, EC-500 horizontal mill. Some features are common with the vertical mill.

NOTE: These figures are representative only; your machine's appearance may vary depending on the model and installed options.

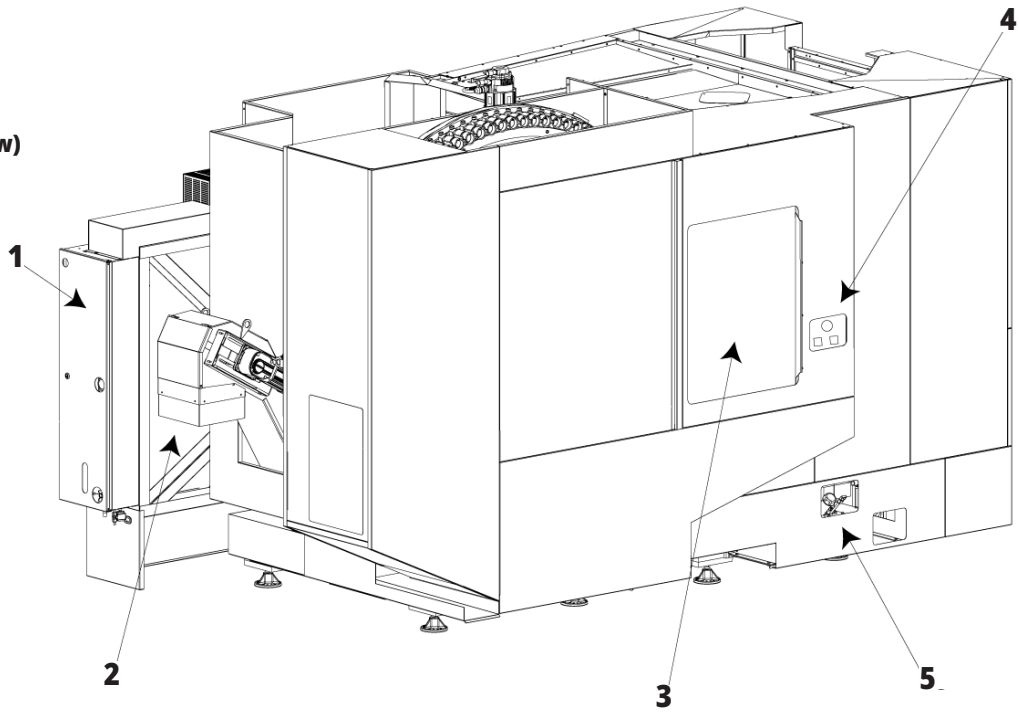
Horizontal Mill Features (EC-400/EC-500, Front view)

1. Load Station Emergency Stop
2. Air Gun
3. Tool Holding Vise
4. Front Table
5. Tool Crib
6. Control Pendant
7. Electrical Cabinet
8. Coolant Filters
9. Side Mount Tool Changer



Horizontal Mill Features (EC-400/EC-500, Rear Left view)

1. Lubrication Panel
2. Chip Conveyor
3. Tool Changer Access Door
4. Tool Changer Emergency Stop
5. Hydraulic Oil Refill

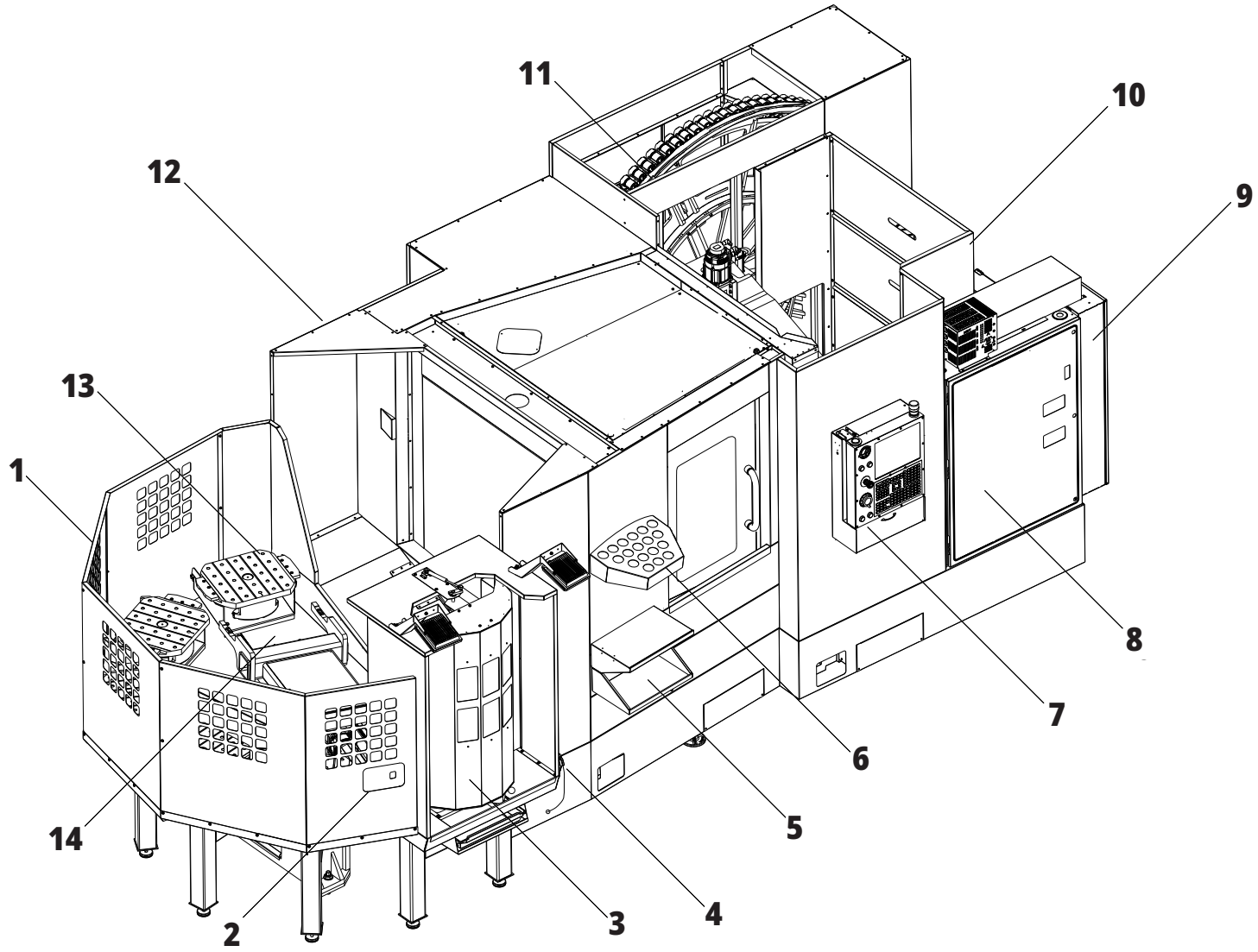


EC-400PP Overview

The following figures show some of the standard and optional features of your EC-400PP horizontal mill. Some features are common with the vertical mill.

NOTE: These figures are representative only; your machine's appearance may vary depending on the model and installed options. For more detail information on Pallet Pool machines refer to the Pallet Pool Operator's Manual.

1. Pallet Pool Assembly
2. Pallet Pool Emergency Stop
3. Pallet Pool Load Station
4. Air Gun
5. Front Table
6. Tool Crib
7. Control Pendant
8. Electrical Cabinet
9. Lubrication Panel
10. Coolant Filters
11. Side Mount Tool Changer
12. Tool Changer Emergency Stop
13. Hydraulic Oil Refill
14. Pallet Pool Slider Assembly



EC-1600 Overview

The following figures show some of the standard and optional features of your EC-1600 horizontal mill. Some features are common with the vertical mill.

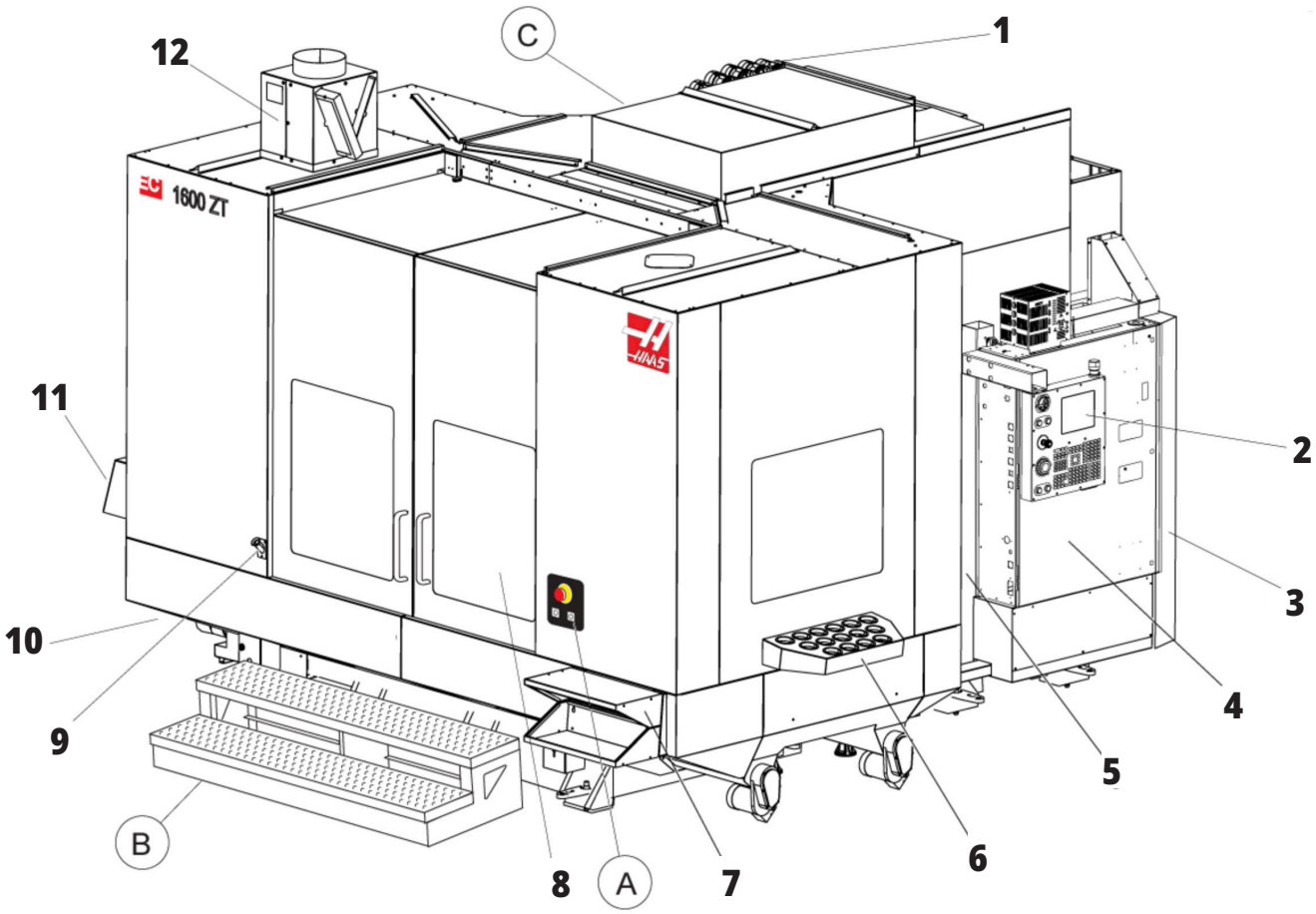
NOTE: These figures are representative only; your machine's appearance may vary depending on the model and installed options.

1. Side Mount Tool Changer SMTC
2. Control Pendant
3. Consolidated Air-Lubrication Module (CALM)
4. Electrical Control Box
5. Operator Spindle Access Door
6. Tool Tray
7. Front Work Table
8. Work Access Doors
9. Air Gun Holder
10. Coolant Tank Assembly (movable)
11. Dual Chip Conveyor
12. Enclosure Exhaust System (optional)

A Rotary Control

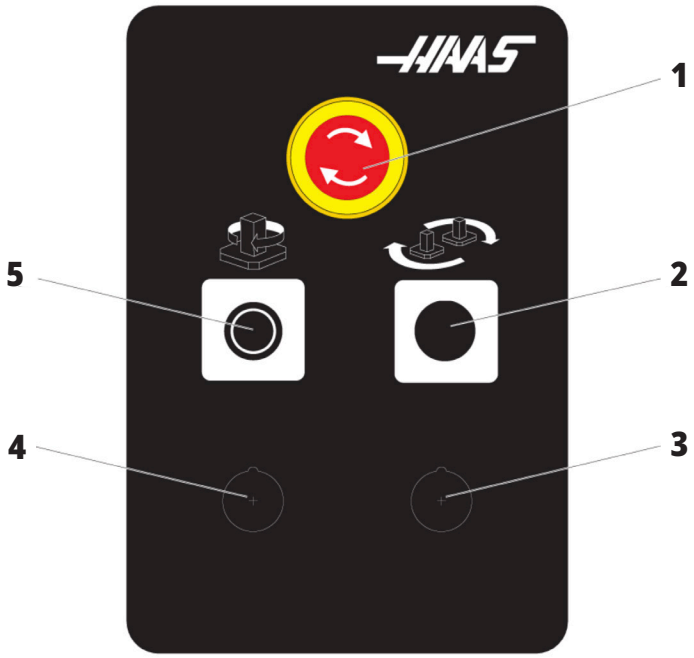
B Work Access Steps

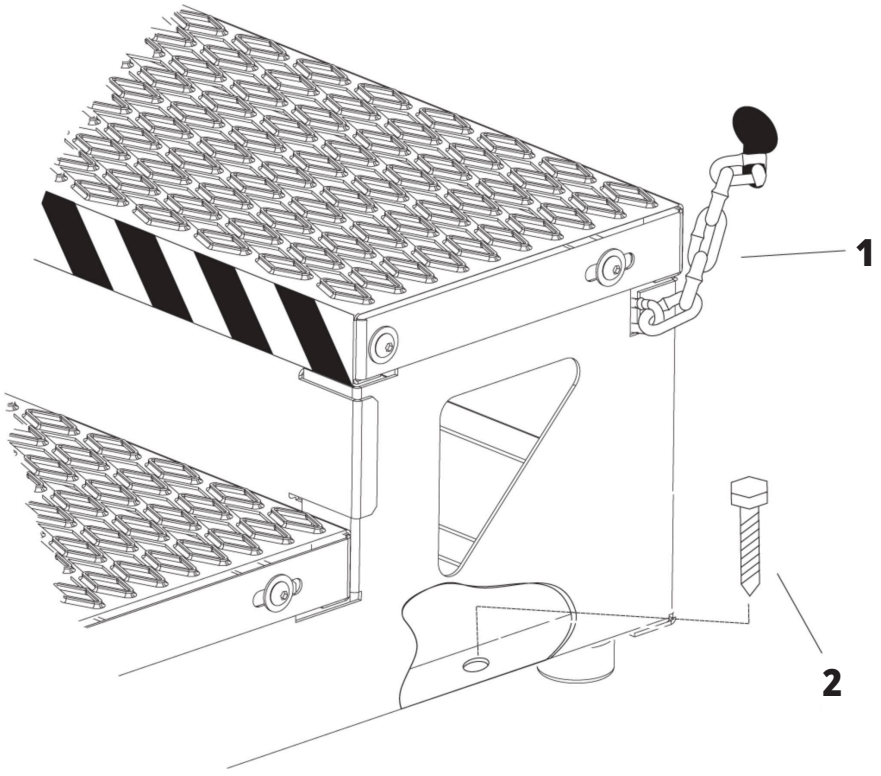
C Secondary ATC Controls



A) Rotary Control

1. Emergency Stop Button
2. (Optional)
3. (Optional)
4. (Optional)
5. Rotary Index Button



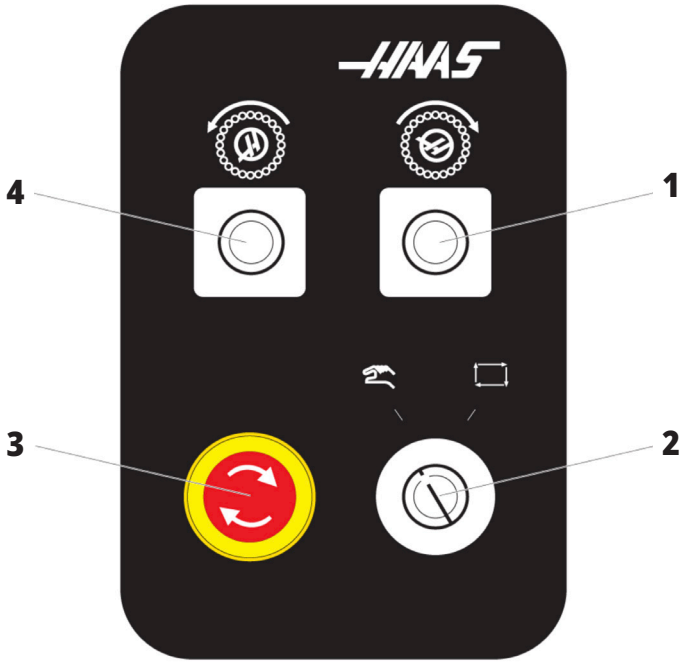


B) Work Access Steps

1. Chain to Enclosure
2. Floor Anchor Bolt
3. Secure the work platform with chains to the enclosure or bolts to the floor.

C) Secondary ATC Controls

1. Secondary ATC Forward Button
2. Manual/Automatic Tool Change Switch (enables [1] and [4] buttons)
3. Emergency Stop Button
4. Secondary ATC Reverse Button



1.4 | HORIZONTAL MILL SPECIFICATIONS

Horizontal Mill Specifications

For Horizontal Mill Series - Technical Specifications, scan the pre-installation QR codes.

EC-400/40T - PRE-INSTALLATION SPECIFICATION

EC-500/40T - PRE-INSTALLATION SPECIFICATION

EC-500/50T - PRE-INSTALLATION SPECIFICATION

EC-1600 - PRE-INSTALLATION SPECIFICATION

EC-1600ZT - PRE-INSTALLATION SPECIFICATION

EC-1600ZT-5AX - PRE-INSTALLATION SPECIFICATION

Copyright Information

Java Powered

This product uses Java Technology from Oracle Corporation and we request that you acknowledge that Oracle owns the Java Trademark and all Java related Trademarks and agree to comply with the trademark guidelines at

WWW.ORACLE.COM/US/LEGAL/THIRD-PARTY-TRADEMARKS/INDEX.HTML.

Any further distribution of the Java programs (beyond this appliance/machine) is subject to a legally binding End User License Agreement with Oracle. Any use of the commercial features for production purposes requires a separate license from Oracle.

Limited Warranty Certificate

Haas Automation, Inc.
Covering Haas Automation, Inc. CNC Equipment

Effective September 1, 2010

Haas Automation Inc. ("Haas" or "Manufacturer") provides a limited warranty on all new mills, turning centers, and rotary machines (collectively, "CNC Machines") and their components (except those listed below under Limits and Exclusions of Warranty) ("Components") that are manufactured by Haas and sold by Haas or its authorized distributors as set forth in this Certificate. The warranty set forth in this Certificate is a limited warranty, it is the only warranty by Manufacturer, and is subject to the terms and conditions of this Certificate.

Limited Warranty Coverage

Each CNC Machine and its Components (collectively, "Haas Products") are warranted by Manufacturer against defects in material and workmanship. This warranty is provided only to an end-user of the CNC Machine (a "Customer"). The period of this limited warranty is one (1) year. The warranty period commences on the date the CNC Machine is installed at the Customer's facility. Customer may purchase an extension of the warranty period from an authorized Haas distributor (a "Warranty Extension"), any time during the first year of ownership.

Repair or Replacement Only

Manufacturer's sole liability, and Customer's exclusive remedy under this warranty, with respect to any and all Haas products, shall be limited to repairing or replacing, at the discretion of the Manufacturer, the defective Haas product.

Disclaimer of Warranty

This warranty is Manufacturer's sole and exclusive warranty, and is in lieu of all other warranties of whatever kind or nature, express or implied, written or oral, including, but not limited to, any implied warranty of merchantability, implied warranty of fitness for a particular purpose, or other warranty of quality or performance or noninfringement. All such other warranties of whatever kind are hereby disclaimed by Manufacturer and waived by Customer.

Limits and Exclusions of Warranty

Components subject to wear during normal use and over time, including, but not limited to, paint, window finish and condition, light bulbs, seals, wipers, gaskets, chip removal system (e.g., augers, chip chutes), belts, filters, door rollers, tool changer fingers, etc., are excluded from this warranty. Manufacturer's specified maintenance procedures must be adhered to and recorded in order to maintain this warranty. This warranty is void if Manufacturer determines that (i) any Haas Product was subjected to mishandling, misuse, abuse, neglect, accident, improper installation, improper maintenance, improper storage, or improper operation or application, including the use of improper coolants or other fluids, (ii) any Haas Product was improperly repaired or serviced by Customer, an unauthorized service technician, or other unauthorized person, (iii) Customer or any person makes or attempts to make any modification to any Haas Product without the prior written authorization of Manufacturer, and/or (iv) any Haas Product was used for any non-commercial use (such as personal or household use). This warranty does not cover damage or defect due to an external influence or matters beyond the reasonable control of Manufacturer, including, but not limited to, theft, vandalism, fire, weather condition (such as rain, flood, wind, lightning, or earthquake), or acts of war or terrorism.

Without limiting the generality of any of the exclusions or limitations described in this Certificate, this warranty does not include any warranty that any Haas Product will meet any person's production specifications or other requirements, or that operation of any Haas Product will be uninterrupted or error-free. Manufacturer assumes no responsibility with respect to the use of any Haas Product by any person, and Manufacturer shall not incur any liability to any person for any failure in design, production, operation, performance, or otherwise of any Haas Product, other than repair or replacement of same as set forth in the warranty above.

Limited Warranty Certificate (Contin.)

Limitation of Liability and Damages

Manufacturer will not be liable to Customer or any other person for any compensatory, incidental, consequential, punitive, special, or other damage or claim, whether in an action in contract, tort, or other legal or equitable theory, arising out of or related to any Haas product, other products or services provided by Manufacturer or an authorized distributor, service technician, or other authorized representative of Manufacturer (collectively, "authorized representative"), or the failure of parts or products made by using any Haas Product, even if Manufacturer or any authorized representative has been advised of the possibility of such damages, which damage or claim includes, but is not limited to, loss of profits, lost data, lost products, loss of revenue, loss of use, cost of down time, business good will, any damage to equipment, premises, or other property of any person, and any damage that may be caused by a malfunction of any Haas product. All such damages and claims are disclaimed by Manufacturer and waived by Customer. Manufacturer's sole liability, and Customer's exclusive remedy, for damages and claims for any cause whatsoever shall be limited to repair or replacement, at the discretion of Manufacturer, of the defective Haas Product as provided in this warranty.

Customer has accepted the limitations and restrictions set forth in this Certificate, including, but not limited to, the restriction on its right to recover damages, as part of its bargain with Manufacturer or its Authorized Representative. Customer realizes and acknowledges that the price of the Haas Products would be higher if Manufacturer were required to be responsible for damages and claims beyond the scope of this warranty.

Entire Agreement

This Certificate supersedes any and all other agreements, promises, representations, or warranties, either oral or in writing, between the parties or by Manufacturer with respect to subject matter of this Certificate, and contains all of the covenants and agreements between the parties or by Manufacturer with respect to such subject matter. Manufacturer hereby expressly rejects any other agreements, promises, representations, or warranties, either oral or in writing, that are in addition to or inconsistent with any term or condition of this Certificate. No term or condition set forth in this Certificate may be modified or amended, unless by a written agreement signed by both Manufacturer and Customer. Notwithstanding the foregoing, Manufacturer will honor a Warranty Extension only to the extent that it extends the applicable warranty period.

Transferability

This warranty is transferable from the original Customer to another party if the CNC Machine is sold via private sale before the end of the warranty period, provided that written notice thereof is provided to Manufacturer and this warranty is not void at the time of transfer. The transferee of this warranty will be subject to all terms and conditions of this Certificate.

Miscellaneous

This warranty shall be governed by the laws of the State of California without application of rules on conflicts of laws. Any and all disputes arising from this warranty shall be resolved in a court of competent jurisdiction located in Ventura County, Los Angeles County, or Orange County, California. Any term or provision of this Certificate that is invalid or unenforceable in any situation in any jurisdiction shall not affect the validity or enforceability of the remaining terms and provisions hereof, or the validity or enforceability of the offending term or provision in any other situation or in any other jurisdiction.

Customer Satisfaction Policy

Dear Haas Customer,

Your complete satisfaction and goodwill are of the utmost importance to both Haas Automation, Inc. and the Haas distributor (HFO) where you purchased your equipment. Normally, your HFO will rapidly resolve any concerns you have about your sales transaction or the operation of your equipment.

However, if your concerns are not resolved to your complete satisfaction, and you have discussed your concerns with a member of the HFO's management, the General Manager, or the HFO's owner directly, please do the following:

Contact Haas Automation's Customer Service Advocate at 805-988-6980. So that we may resolve your concerns as quickly as possible, please have the following information available when you call:

- Your company name, address, and phone number
- The machine model and serial number
- The HFO name, and the name of your latest contact at the HFO
- The nature of your concern

If you wish to write Haas Automation, please use this address:

Haas Automation, Inc. U.S.A.
2800 Sturgis Road
Oxnard CA 93030
Att: Customer Satisfaction Manager
email: customerservice@HaasCNC.com

Once you contact the Haas Automation Customer Service Center, we will make every effort to work directly with you and your HFO to quickly resolve your concerns. At Haas Automation, we know that a good Customer-Distributor-Manufacturer relationship will help ensure continued success for all concerned.

INTERNATIONAL:

Haas Automation, Europe
Mercuriusstraat 28, B-1930
Zaventem, Belgium
email: customerservice@HaasCNC.com

Haas Automation, Asia
No. 96 Yi Wei Road 67,
Waigaoqiao FTZ
Shanghai 200131 P.R.C.
email: customerservice@HaasCNC.com

Customer Feedback

If you have concerns or questions regarding this Operator's Manual, please contact us on our website, www.HaasCNC.com. Use the "Contact Us" link and send your comments to the Customer Advocate.

Declaration of Conformity

Product: Mill (Vertical and Horizontal)*

*Including all options factory- or field-installed by a certified Haas Factory Outlet (HFO)

Manufactured By:

Haas Automation, Inc.
2800 Sturgis Road, Oxnard, CA 93030
805-278-1800

We declare, in sole responsibility, that the above-listed products, to which this declaration refers, comply with the regulations as outlined in the CE directive for Machining Centers:

Machinery Directive 2006/42/EC
Electromagnetic Compatibility Directive 2014/30/EU
Low Voltage Directive 2014/35/EC

Additional Standards:

EN 12417:2001+A2:2009
EN 60204-1:2018
EN ISO 13849-1:2015
ISO 10218:1-2:2011 (if robot included)
RoHS2: COMPLIANT (2011/65/EU) by Exemption per producer documentation.

Exempt by:

- Large scale stationary industrial tool.
- Lead as an alloying element in steel, aluminum, and copper.
- Cadmium and its compounds in electrical contacts.

Person authorized to compile technical file:

Kristine De Vriese
Phone: +32 (2) 4272151

Address:
Haas Automation Europe
Mercuriusstraat 28
B-1930 Zaventem
Belgium

CANADA: As the original equipment manufacturer, we declare that the listed products comply with regulations as outlined in the Pre-Start Health and Safety Reviews Section 7 of Regulation 851 of the Occupational Health and Safety Act Regulations for Industrial Establishments for machine guarding provisions and standards.

Further, this document satisfies the notice-in-writing provision for exemption from Pre-Start inspection for the listed machinery as outlined in the Ontario Health and Safety Guidelines, PSR Guidelines dated November 2016. The PSR Guidelines allow that notice in writing from the original equipment manufacturer declaring conformity to applicable standards is acceptable for the exemption from Pre-Start Health and Safety Review.

All Haas CNC machine tools carry the ETL Listed mark, certifying that they conform to the NFPA 79 Electrical Standard for Industrial Machinery and the Canadian equivalent, CAN/CSA C22.2 No. 73. The ETL Listed and cETL Listed marks are awarded to products that have successfully undergone testing by Intertek Testing Services (ITS), an alternative to Underwriters' Laboratories.

Haas Automation has been assessed for conformance with the provisions set forth by ISO 9001: 2015. Scope of Registration: Design and Manufacture of CNC Machines Tools and Accessories, Sheet Metal Fabrication. The conditions for maintaining this certificate of registration are set forth in ISA's Registration Policies 5.1. This registration is granted subject to the organization maintaining compliance to the noted standard. The validity of this certificate is dependent upon ongoing surveillance audits.

Safety Notes

CAUTION: Only authorized and trained personnel may operate this equipment. You must always act in accordance with the Operator's manual, safety decals, safety procedures, and instructions for safe machine operation. Untrained personnel present a hazard to themselves and the machine

IMPORTANT: Do not operate this machine until you have read all warnings, cautions, and instructions

CAUTION: The sample programs in this manual have been tested for accuracy, but they are for illustrative purposes only. The programs do not define tools, offsets, or materials. They do not describe workholding or other fixturing. If you choose to run a sample program on your machine, do so in Graphics mode. Always follow safe machining practices when you run an unfamiliar program.

All CNC machines present hazards from rotating cutting tools, belts and pulleys, high voltage electricity, noise, and compressed air. When you use CNC machines and their components, you must always follow basic safety precautions to reduce the risk of personal injury and mechanical damage.
The work area must be adequately illuminated to allow clear view and safe operation of the machine. This includes the operator work area and all areas of the machine that might be accessed during maintenance or cleaning. Adequate illumination is the responsibility of the user.

Cutting tools, workholding, workpiece and coolant are beyond the scope and control of Haas Automation, Inc. Each of these potential hazards associated with it (sharp edges, heavy lifting considerations, chemical composition, etc) and it is the responsibility of the user to take appropriate action (PPE, training, etc).

Cleaning of the machine is required during normal use and prior to maintenance or repair. Optional equipment is available to aid cleaning such as washdown hoses, chip conveyors and chip augers. Safe use of this equipment requires training and might require appropriate PPE and is the responsibility of the user.

This operator's manual is intended as a reference guide and is not to be the sole source of training. Complete operator training is available from the authorized Haas distributor.

Summary of Types of Operation for Haas Automation Machine Tools

Haas CNC Mills are intended for cutting and shaping of metals and other hard materials. They are general purpose in nature and a list of all of those materials and types of cutting would never be complete. Almost all cutting and shaping is performed by a rotating tool mounted in a spindle. Rotation of the mill is not required. Some cutting operations require liquid coolant. That coolant is also an option depending on the type of cutting.

Operations of Haas Mills are separated into three areas. They are: Operations, Maintenance, and Service. Operations and Maintenance are intended to be performed by a trained and qualified machine operator. This Operator's Manual contains some of the information necessary to operate the machine. All other machine operations are to be considered Service. Service is only to be performed by specially trained service personnel.

3.1 | MILL - SAFETY

Operation of this machine consists of the following:

1. **Machine Setup**
   Machine setup is done to initially set up the tools, offsets, and fixtures required to perform a repetitive function that later is called machine operation. Some machine setup functions can be done with the door open but are limited to “hold to run”.
2. **Machine operating in Automatic Mode**
   Automatic operation is initiated with Cycle-Start and can only be done with the doors closed.
3. **Operator loading and unloading of materials (parts)**
   Parts loading and unloading is what precedes and follows an automatic operation. This must be done with the doors open and all machine automatic motion is stopped when the door is open.
4. **Operator loading and unloading of cutting tools**
   Tool loading and unloading is done less often than setup. It is often required when a tool has become worn and must be replaced.

Maintenance only consists of the following:

1. **Adding and maintaining condition of coolant**
   Adding coolant and maintaining coolant concentration is required at regular intervals. This is a normal operator function and is either done from a safe location outside of the work enclosure or with the doors open and the machine stopped.
2. **Adding lubricants**
   Adding lubricants for spindle and axes is required at regular intervals. These are often months or years in length. This is a normal operator function and is always done from a safe location outside of the work enclosure.
3. **Cleaning chips out of the machine**
   Cleaning out of chips is required at intervals dictated by the type of machining performed. This is a normal operator function. It is performed with the doors open and all of the machine operation is stopped.

Service only consists of the following:

1. **Repairing of a machine that is not operating correctly**
   Any machine that is not operating correctly requires service by factory trained personnel. This is never an operator function. It is not considered maintenance. Installation and service instructions are provided separately from the Operator’s Manual.
2. **Machine moving, unpacking, and installation**
   Haas machines are shipped to a user’s location almost ready to operate. They still require a trained service person to complete the installation. Installation and service instructions are provided separately from the Operator’s Manual.
3. **Machine packing**
   Machine packing for shipment requires the same packing material supplied by Haas in the original shipment. Packing requires a trained service person to complete the installation. Shipping instructions are provided separately from the Operator’s Manual.
4. **Decommission, dismantle and disposal**
   Machine is not expected to be disassembled for shipment; it can be moved in its entirety in the same manner in which it was installed. Machine can be returned to the manufacturer’s distributor for disposal; manufacturer accepts any/all components for recycling per Directive 2002/96/EC.
5. **End-of-life disposal**
   End-of-life disposal must conform to the laws and regulations in the region the machine is located. This is a jointly the responsibility of the owner and seller of the machine. The risk analysis does not address this phase.

READ BEFORE OPERATING

DANGER: Do not enter the machining area any time the machine is in motion, or at any time that machine motion is possible. Severe injury or death may result. Motion is possible when the power is on and the machine is not in [EMERGENCY STOP].

Basic safety:

- This machine can cause severe bodily injury.
- This machine is automatically controlled and may start at any time.
- Consult your local safety codes and regulations before you operate the machine. Contact your dealer if you have questions about safety issues.
- It is the machine owner's responsibility to make sure that everyone who is involved in installing and operating the machine is fully acquainted with the operation and safety instructions provided with the machine, BEFORE they work with the machine. The ultimate responsibility for safety rests with the machine owner and the individuals who work with the machine.
- Use appropriate eye and ear protection when you operate the machine.
- Use appropriate gloves to remove processed material and to clean the machine.
- Replace windows immediately if they are damaged or severely scratched.
- Keep the side windows locked during operation (if available).

Electrical safety:

- The electrical power must meet the required specifications. Attempting to run the machine from any other source can cause severe damage and will void the warranty.
- The electrical panel should be closed and the key and latches on the control cabinet should be secured at all times, except during installation and service. At those times, only qualified electricians should have access to the panel. When the main circuit breaker is on, there is high voltage throughout the electrical panel (including the circuit boards and logic circuits) and some components operate at high temperatures; therefore, extreme caution is required. Once the machine is installed, the control cabinet must be locked, with the key available only to qualified service personnel.
- Do not reset a circuit breaker until the reason for the fault is investigated and understood. Only Haas-trained service personnel should troubleshoot and repair Haas equipment.
- Do not press [POWER UP] on the control pendant before the machine is fully installed.

Operation Safety:

DANGER: To avoid injury verify that the spindle has stopped turning before opening the doors. In the event of a loss of power the spindle will take much longer to coast to a stop.

- Do not operate the machine unless the doors are closed and the door interlocks are functioning correctly.
- Check for damaged parts and tools before you operate the machine. Any part or tool that is damaged should be properly repaired or replaced by authorized personnel. Do not operate the machine if any component does not appear to be functioning correctly.
- Rotating cutting tools can cause severe injury. When a program runs, the mill table and spindle head can move rapidly at any time.
- Improperly clamped parts machined at high speeds/feeds may be ejected and puncture the enclosure. It is not safe to machine oversized or marginally clamped parts.

CAUTION: Manual or Automatic closing of the enclosure doors is a potential pinch point. With Auto Door, the door may be programmed to close automatically, or by pressing the door open/close button on the operators pendant. Avoid putting hands or appendages in the door while closing either manually or automatically.

Release of person trapped in the machine:

- No person should ever be located inside the machine during operation.
- In the unlikely event that a person is trapped inside the machine the emergency stop button should be immediately be depressed and the person removed.
- If the person is pinched or entangled the machine should be powered off; then the machine axes can be moved by use of a large external force in the direction required to free the person.

3.2 | MILL - READ BEFORE OPERATING

Recover from a jam or blockage:

- Of the chip conveyor - Follow the cleaning instructions on the Haas service site (go to www.haascnc.com click on the Service tab). If necessary, close the doors and reverse the conveyor so the jammed part or material is accessible, and remove.
- Use lifting equipment or get assistance for lifting heavy and awkward parts.
- Of a tool and material/part - Close the doors, press [RESET] to clear and displayed alarms. Jog the axis so the tool and material are clear.
- Of the Automatic Tool Changer/tool and spindle - Press [RECOVER] and follow the on-screen instructions.
- If the alarms do not reset or you are unable to clear a blockage, contact your Haas Factory Outlet (HFO) for assistance.

Follow these guidelines when you work with the machine:

- Normal operation - Keep the door closed and guards in place (for non-enclosed machines) while the machine operates.
- Part loading and unloading – An operator opens the door, completes the task, closes the door, and then presses [CYCLE START] (starting automatic motion).
- Machining job set-up – When set-up is complete, turn the set-up key to lock out set-mode and remove the key.
- Maintenance / Machine Cleaner– Press [EMERGENCY STOP] or [POWER OFF] on the machine before you enter the enclosure.

Periodic inspection of machine safety features:

- Inspect door interlock mechanism for proper fit and function.
- Inspect safety windows and enclosure for damage or leaks.
- Verify all enclosure panels are in place.

Door Safety Interlock inspection:

- Inspect the door interlock, verify the door interlock key is not bent, misaligned, and that all fasteners are installed.
- Inspect the door interlock itself for any signs of obstruction or misalignment.
- Immediately replace an components of the Door Safety Interlock system that do not meet this criteria.

Door Safety Interlock verification:

- With the machine in run mode, close the machine door, run the spindle at 100 RPM, pull the door and verify the door does not open.

MACHINE ENCLOSURE AND SAFETY GLASS INSPECTION AND TESTING:

Routine Inspection:

- Visually inspect the enclosure and safety glass for any signs of distortion, breakage or other damage.

Replace the Lexan windows after 7 years or if they are damaged or severely scratched.

- Keep all safety glass and machine windows clean to allow proper viewing of the machine during operations.
- A daily visual inspection of the machine enclosure to verify all panels are in place should be performed.

Testing of machine enclosure:

- No testing of the machine enclosure is necessary.

Machine Environmental Limits

This table list the environmental limits for safe operation:

Environmental Limits (Indoor Use Only)

	MINIMUM	MAXIMUM
Operating Temperature	41 °F (5.0 °C)	122 °F (50.0 °C)
Storage Temperature	-4 °F (-20.0 °C)	158 °F (70.0 °C)
Ambient Humidity	20% relative, non-condensing	90% relative, non-condensing
Altitude	Sea Level	6,000 ft. (1,829 m)

CAUTION: Do not operate the machine in explosive atmospheres (explosive vapors and/ or particulate matter).

Machine with Haas Robot Package
Machine and robot environment is intended to be a machine shop or industrial installation. Shop lighting is the users responsibility.

Machine Noise Limits

CAUTION: Take precautions to prevent hearing damage from machine/machining noise. Wear ear protection, change your application (tooling, spindle speed, axis speed, fixturing, programmed path) to reduce noise, or restrict access to machine area during cutting.

Typical noise levels at the operator's position during normal operation are as follows:

- **A-Weighted** sound pressure level measurements will be 69.4dB or lower.
- **C-Weighted** instantaneous sound pressure levels will be 78.0dB or lower.
- **LwA** (sound power level A-weighted) will be 75.0dB or lower.

NOTE: Actual noise levels while cutting material are greatly affected by the user's choice of material, cutting tools, speeds and feeds, workholding and other factors. These factors are application specific and are controlled by the user, not Haas Automation Inc

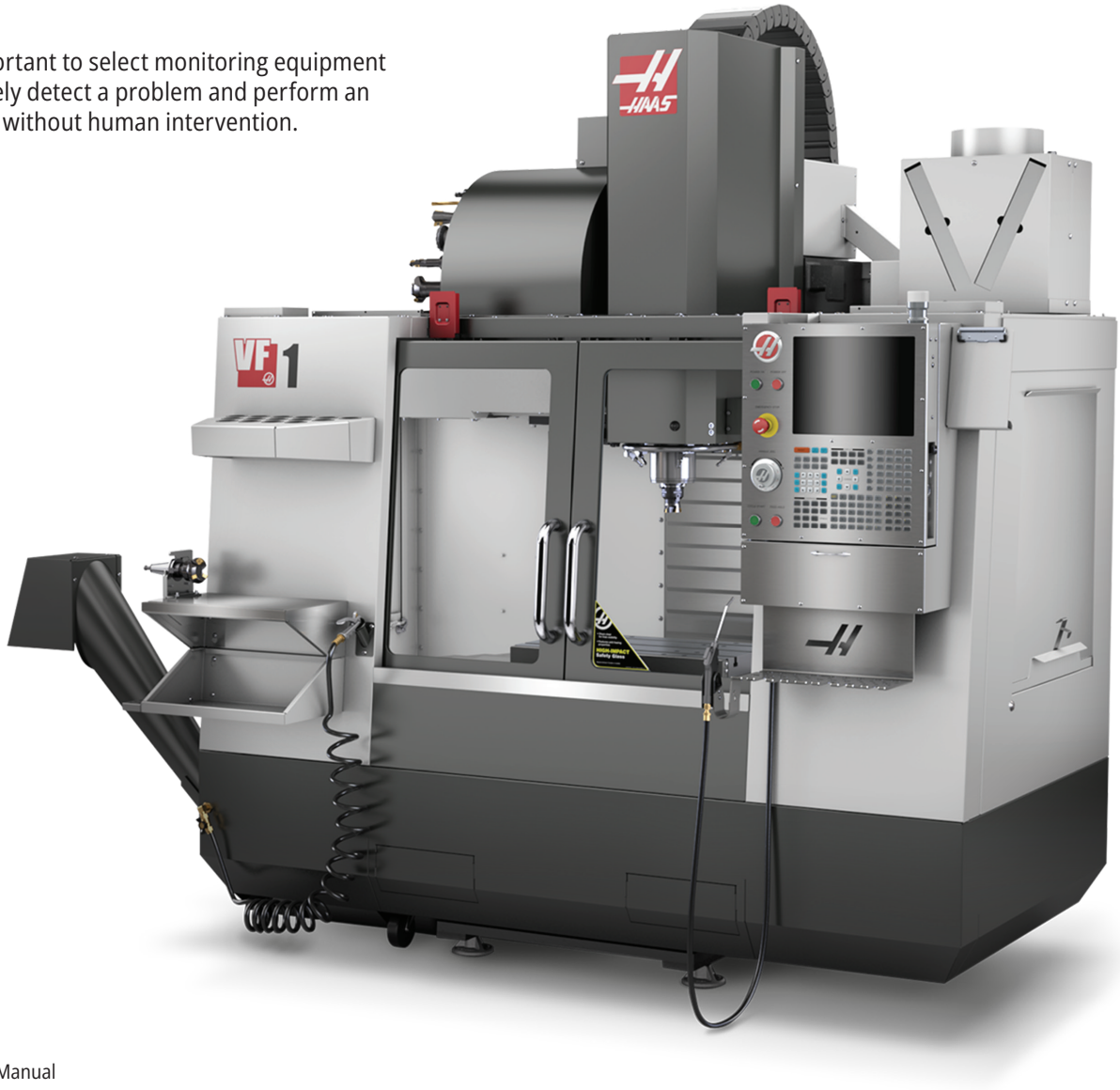
Unattended Operation

Fully enclosed Haas CNC machines are designed to operate unattended; however, your machining process may not be safe to operate unmonitored.

As it is the shop owner's responsibility to set up the machine safely and use best practice machining techniques, it is also the owner's responsibility to manage the progress of these methods. You must monitor your machining process to prevent damage, injury, or loss of life if a hazardous condition occurs.

For example, if there is the risk of fire due to the material machined, then you must install an appropriate fire suppression system to reduce the risk of harm to personnel, equipment, and the building. Contact a specialist to install monitoring tools before machines are allowed to run unattended.

It is especially important to select monitoring equipment that can immediately detect a problem and perform an appropriate action without human intervention.



Run/Setup Mode Restrictions

All Haas CNC machines are equipped with locks on the operator doors and a key switch on the side of the control pendant to lock and unlock setup mode. Generally, setup mode status (locked or unlocked) affects how the machine operates when the doors are opened.

Setup mode should be locked out (the keyswitch in the vertical, locked position) at most times. In Run and in Setup mode, the enclosure doors are locked closed during CNC program execution, spindle rotation or axis movement. The doors automatically unlock when the machine is not in cycle. Many machine functions are unavailable with the door open.

When unlocked, setup mode allows a skilled machinist more access to the machine to set up jobs. In this mode, machine behavior is dependent on whether the doors are opened or closed. The following charts summarize the modes and allowed functions.

Note: All these conditions follow assuming that the door is open and stays open before, during and the actions occur.

Danger: Do not attempt to override safety features. Doing so makes the machine unsafe and voids the warranty.

MACHINE FUNCTION	RUN MODE	SETUP MODE
Air Blast (AAG) On	Not allowed.	Not allowed.
Axis Jog using the Pendant Handle Jog	Not allowed.	Allowed.
Axis Jog using the RJH Handle Jog	Not allowed.	Not allowed.
Axis Jog using the RJH shuttle knob	Not allowed.	Not allowed.
Axis Rapid using Home G28 or Second Home	Not allowed.	Not allowed.
Axis Zero Return	Not allowed.	Not allowed.
Automatic Pallet Change	Not allowed.	Not allowed.
APC Operation Buttons	Not allowed.	Not allowed.
Chip Conveyor CHIP FWD, REV	Not allowed.	Not allowed.
COOLANT button on pendant	Not allowed.	Allowed.
COOLANT button on RJH.	Not allowed.	Allowed.
Move Programmable Coolant Spigot	Not allowed.	Allowed.
Orient Spindle	Not allowed.	Not allowed.
Run a program, CYCLE START button on the pendant	Not allowed.	Not allowed.
Run a program CYCLE START button on the RJH	Not allowed.	Not allowed.
Run a program (Pallet)	Not allowed.	Not allowed.
Spindle FWD / REV button on the pendant	Not allowed.	Not allowed.
Spindle FWD / REV on the RJH	Not allowed.	Not allowed.
Tool Change ATC FWD / ATC REV.	Not allowed.	Not allowed.
Tool Release from Spindle	Allowed.	Allowed.
Through Spindle Coolant (TSC) On	Not allowed.	Not allowed.
Tool Air Blast (TAB) On	Not allowed.	Not allowed.

ROBOT CELLS

A machine in a robot cell is allowed to run a program while the door is open, regardless of the position of the Run-Setup key. While the door is open, the spindle speed is limited to the lower of the factory RPM limit or Setting 292, Door Open Spindle Speed Limit. If the door is opened while the spindle RPM is above the limit, the spindle will decelerate to the limit RPM. Closing the door removes the limit and the programmed RPM is restored.

This open-door condition is allowed only while a robot communicates with the CNC machine. Typically, an interface between the robot and the CNC machine addresses the safety of both machines.

Mist Extraction / Enclosure Evacuation

Some models have a provision installed that will allow for a mist extractor to be attached to the machine.

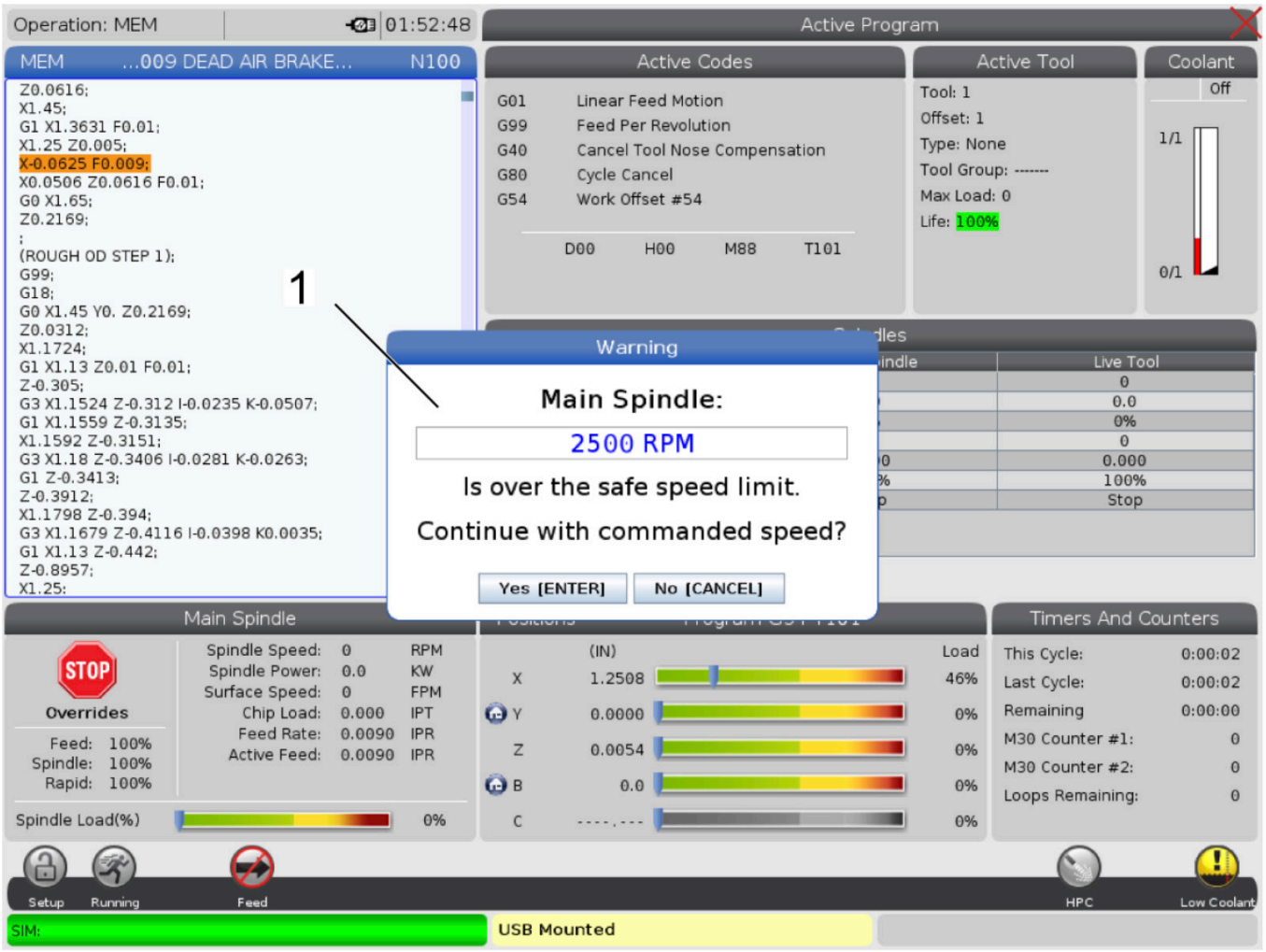
There is also an optional enclosure exhaust system available that helps keep the mist out of the machine enclosure.
It is entirely up to the owner/operator to determine if and what type of mist extractor is best suited for the application.

The owner/operator assumes all responsibility for the installation of the mist extraction system.

Spindle Safety Limit

Starting in software version 100.19.000.1100 a spindle safety limit has been added to the control.

This feature will display a warning message when the [FWD] or [REV] button is pressed and the previous commanded spindle speed is above the Spindle Maximum Manual Speed parameter. Press [ENTER] to go to the previous commanded spindle speed or press [CANCEL] to cancel the action.



MACHINE / SPINDLE OPTION	SPINDLE MAXIMUM MANUAL SPEED
Mills	5000

NOTE: These values can not be changed.

Modifications to the Machine

Haas Automation, Inc. is not responsible for damage caused by modifications you make to your Haas machine(s) with parts or kits not manufactured or sold by Haas Automation, Inc. The use of such parts or kits may void your warranty.

Some parts or kits manufactured or sold by Haas Automation, Inc. are considered user-installable. If you choose to install these parts or kits yourself, be sure to completely read the accompanying installation instructions. Make sure you understand the procedure, and how to do it safely, before you begin. If you have any doubts about your ability to complete the procedure, contact your Haas Factory Outlet (HFO) for assistance.

Improper Coolants

Coolant is an important part of many machining operations. When it is correctly used and maintained, coolant can improve part finish, lengthen tool life, and protect machine components from rust and other damage. Improper coolants, however, can cause significant damage to your machine.

Such damage can void your warranty, but it can also introduce hazardous conditions to your shop. For example, coolant leaks through damaged seals could create a slipping hazard.

Improper coolant use includes, but is not limited to, these points:

- Do not use plain water. This causes machine components to rust.
- Do not use flammable coolants.
- Do not use straight or "neat" mineral-oil products. These products cause damage to rubber seals and tubing throughout the machine. If you use a minimum-quantity lubrication system for near-dry machining, use only the recommended oils.

Machine coolant must be water-soluble, synthetic oil-based or synthetic-based coolant or lubricant.

NOTE: Be sure to maintain your coolant mixture to keep the coolant concentrate at acceptable levels. Improperly maintained coolant mixtures can allow machine components to rust. Rust damage is not covered by your warranty.
Ask your HFO or your coolant dealer if you have questions about the specific coolant that you plan to use.

Safety Decals

The Haas factory puts decals on your machine to quickly communicate possible hazards. If decals become damaged or worn, or if you need additional decals to emphasize a particular safety point, contact your Haas Factory Outlet (HFO).

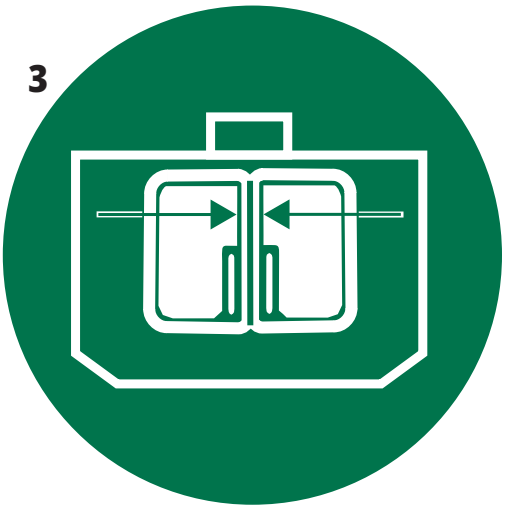
NOTE: Never alter or remove any safety decal or symbol.

Be sure to familiarize yourself with the symbols on the safety decals. The symbols are designed to quickly tell you the type of information they give:

- **Yellow Triangle -** Describes a hazard.
- **Red Circle with Slash-Through -** Describes a prohibited action.
- **Green Circle -** Describes a recommended action.
- **Black Circle -** Gives information about machine or accessory operation.

Example Safety Decal Symbols:
[1] Hazard Description, [2] Prohibited Action, [3] Recommended Action.

You may find other decals on your machine, depending on the model and options installed. Be sure to read and understand these decals.



3.11 | MILL - SAFETY DECALS

Hazard Symbols - Yellow Triangles

Moving parts can entangle, trap, crush, and cut.
Keep all parts of your body away from machine parts when they move, or whenever motion is possible. Motion is possible when the power is on and the machine is not in **[EMERGENCY STOP].**
Secure loose clothing, hair, etc.
Remember that automatically controlled devices can start at any time.

Do not touch rotating tools.
Keep all parts of your body away from machines parts when they move, or whenever motion is possible. Motion is possible when the power is on and the machine is not in **[EMERGENCY STOP].**
Sharp tools and chips can easily cut skin.

The Regen is used by the spindle drive to dissipate excess power and will get hot.
Always use care around the Regen.

There are high voltage components on the machine that can cause electrical shock.
Always use care around high voltage components.

There are high voltage components on the machine that can cause arc flash, and electrical shock.
Take care to avoid opening electrical enclosures unless components are de-energized or proper personal protective equipment is worn. Arc flash ratings are on the nameplate.

Long tools are dangerous, especially at spindle speeds higher than 5000 RPM. The tools can break and eject from the machine.
Remember that machine enclosures are intended to stop coolant and chips. Enclosures may not stop broken tools or thrown parts.
Always check your setup and tooling before you start machining.

Machining operations can create hazardous chips, dust, or mist. This is a function of the materials being cut, the metalworking fluid and cutting tools used, and the machining speeds/feeds.

It is up to the owner/operator of the machine to determine if personal protective equipment such as safety goggles or a respirator is required and also if a mist extraction system is needed.

Some models have a provision for connecting a mist extraction system. Always read and understand the Safety Data Sheets (SDS) for the workpiece material, the cutting tools, and the metalworking fluid.

Other Safety Information

IMPORTANT: You may find other decals on your machine, depending on the model and options installed. Be sure to read and understand these decals.

Prohibited Action Symbols - Red Circles with Slash-Through

Do not enter the machine enclosure when the machine is capable of automatic motion.
When you must enter the enclosure to complete tasks, press [EMERGENCY STOP] or power off the machine. Put a safety tag on the control pendant to alert other people that you are inside the machine, and that they must not turn on or operate the machine.

Do not machine ceramics.

Do not attempt to load tools with the spindle dogs misaligned with the cutouts in the toolholder V-Flange.

Do not machine flammable materials.
Do not use flammable coolants.
Flammable materials in particulate or vapor form can become explosive.
The machine enclosures is not designed to contain explosions or extinguish fire.

Do not use pure water as coolant. This will cause machine components to rust.
Always use a rust-inhibitive coolant concentrate with water.

Prohibited Action Symbols - Red Circles with Slash-Through

Keep the machine doors closed.

Always wear safety glasses or goggles when you are near a machine.
Airborne debris can cause eye damage.
Always wear hearing protection when you are near a machine.
Machine noise can exceed 70 dBA.

Make sure the spindle dogs are correctly aligned with the cutouts in the toolholder V-flange.

Note the location of the tool release button. Press this button only when you are holding the tool.
Some tools are very heavy. Handle these tools carefully; use both hands and have someone press the tool release button for you.

Informational Symbols – Black Circles

Maintain the recommended coolant concentration.
A "lean" coolant mixture (less concentrated than recommended) may not effectively prevent machine components from rusting.
A "rich" coolant mixture (more concentrated than recommended) wastes coolant concentrate without further benefit over the recommended concentration.

4.1 | MILL - CONTROL PENDANT

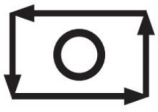
Control Pendant Overview

The control pendant is the main interface to your Haas machine. This is where you program and run your CNC machining projects. This control pendant orientation section describes the different pendant sections:

- **Pendant front panel**
- **Pendant right side, top, and bottom**
- **Keyboard**
- **Function/ Cursor Keys**
- **Display / Mode Keys**
- **Numeric / Alpha Keys**
- **Jog / Override Keys**

4.2 | PENDANT FRONT PANEL

Front Panel Controls

NAME	IMAGE	FUNCTION
POWER ON		Powers the machine on.
POWER OFF		Powers the machine off.
EMERGENCY STOP		Press to stop all axis motion, disable servos, stop the spindle and tool changer, and turn off the coolant pump.
HANDLE JOG		This is used to jog axes (select in HANDLE JOG Mode). Also used to scroll through program code or menu items while editing.
CYCLE START		Starts a program. This button is also used to start a program simulation in graphics mode.
FEED HOLD		Stops all axis motion during a program. The spindle continues to run. Press CYCLE START to cancel.

4.2 | MILL - PENDANT FRONT PANEL

Pendant Right Side, and Top Panels

NAME	IMAGE	FUNCTION
USB		Plug compatible USB devices into this port. It has a removable dust cap.
MEMORY LOCK		In the locked position, this keyswitch prevents alterations to programs, settings, parameters, and offsets.
SETUP MODE		In the locked position, this keyswitch enables all machine safety features. Unlock allows setup (refer to “Setup Mode” in the Safety section of this manual for details).
SECOND HOME		Press to rapid all axes to the coordinates specified in settings 268 - 270. (Refer to “Settings 268 - 270” in the Settings section of this manual for details).
AUTO DOOR OVERRIDE		Press this button to open or close the Auto Door (if equipped).
WORKLIGHT		These buttons toggle the internal worklight and High Intensity Lighting (if equipped).

Pendant Top Panel

BEACON LIGHT

Provides quick visual confirmation of the machine’s current status. There are five different beacon states:

Light Status	Meaning
Off	The machine is idle.
Solid Green	The machine is running.
Flashing Green	The machine is stopped but is in a ready state. Operator input is required to continue.
Flashing Red	A fault has occurred, or the machine is in Emergency Stop

Keyboard

Keyboard keys are grouped into these functional areas:

1. Function

2. Cursor

3. Display

4. Mode

5. Numeric

6. Alpha

7. Jog

8. Overrides

Special Symbol Input

Some special symbols are not on the keypad.

NAME	SYMBOL
_	underscore
^	caret
~	tilde
{	open curly brackets
}	closed curly brackets
\	backslash
|	pipe
<	less than
>	greater than

Do these steps to enter special symbols:

1. Press LIST PROGRAMS and select a storage device.
2. Press F3.
3. Select Special Symbols and press ENTER.
4. Enter a number to copy the associated symbol to the INPUT: bar.

For example, to change a directory's name to **MY_DIRECTORY:**

1. Highlight the directory with the name that you want to change.
2. Type MY.
3. Press F3.
4. Select SPECIAL SYMBOLS and press ENTER.
5. Press 1.
6. Type DIRECTORY.
7. Press F3.
8. Select RENAME and press ENTER.

4.4 | MILL - FUNCTION / CURSOR KEYS

Function Keys

List of Function Keys and How They Operate

NAME	KEY	FUNCTION
Reset	RESET	Clears alarms. Clears input text. Sets overrides to default values if Setting 88 is ON.
Power up	POWER UP	Zero returns all axes and initializes the machine control.
Recover	RECOVER	Enters tool changer recovery mode.
F1- F4	F1- F4	These buttons have different functions depending on the tab that is active.
Tool Offset Measure	TOOL OFFSET MEASURE	Records tool length offsets during part setup.
Next Tool	NEXT TOOL	Selects the next tool from the tool changer.
Tool Release	TOOL RELEASE	Releases the tool from the spindle when in MDI, ZERO RETURN, or HAND JOG mode.
Part Zero Set	PART ZERO SET	Records work coordinate offsets during part setup.

Cursor Keys

The cursor keys let you move between data fields, scroll through programs, and navigate through tabbed menus.

NAME	KEY	FUNCTION
Home	HOME	Moves the cursor to the top-most item on the screen; in editing, this is the top left block of the program.
Cursor Arrows	CURSOR ARROWS	Moves one item, block, or field in the associated direction. The keys depict arrows, but this manual refers to these keys by their spelled-out names.
Page Up, Page Down	PAGE UP, PAGE DOWN	Used to change displays or move up/down one page when viewing a program.
End	END	Moves the cursor to the bottom-most item on the screen. In editing, this is the last block of the program.

Display Keys

You use the Display keys to see the machine displays, operational information, and help pages.

NAME	KEY	FUNCTION
Program	PROGRAM	Selects the active program pane in most modes.
Position	POSITION	Selects the positions display.
Offsets	OFFSET	Displays the Tool Offset and Work Offset tabbed menu.
Current Commands	CURRENT COMMANDS	Displays menus for Devices, Timers, Macros, Active Codes, Calculators, Advanced Tool Management (ATM), Tool Table, and Media.
Alarms	ALARMS	Displays the Alarm viewer and Message screens.
Diagnostics	DIAGNOSTIC	Displays tabs for Features, Compensation, Diagnostics, and Maintenance.
Settings	SETTING	Displays and allows changing of user settings.
Help	HELP	Displays help information.

Mode Keys

Mode keys change the operational state of the machine. Each mode key is arrow shaped and points to the row of keys that perform functions related to that mode key. The current mode is always displayed in the top left of the screen, in Mode:Key display form.

NOTE: EDIT and LIST PROGRAM can also act as display keys, where you can access program editors and the device manager without changing the machine mode. For example, while the machine runs a program, you can use the device manager (LIST PROGRAM) or background editor (EDIT) without stopping the program.

NAME	KEY	FUNCTION
EDIT MODE KEYS		
Edit	EDIT	Lets you edit programs in the editor. You can access the Visual Programming System (VPS) from the EDIT tabbed menu.
Insert	INSERT	Enters text from the input line or the clipboard into the program at the cursor position.
Alter	ALTER	Replaces the highlighted command or text with text from the input line or the clipboard. **NOTE:** ALTER does not work for offsets.
Delete	DELETE	Deletes the item that the cursor is on, or deletes a selected program block.
Undo	UNDO	Undoes up to the last 40 edit changes, and deselects a highlighted block. **NOTE:** UNDO does not work for deleted highlighted blocks or to recover a deleted program.
MEMORY MODE KEYS		
Memory	MEMORY	Selects memory mode. You run programs in this mode, and the other keys in the MEM row control the ways in which the program is run. Shows OPERATION:MEM in upper left display.
Single Block	SINGLE BLOCK	Toggles single block on or off. When single block is on, the control runs only one program block each time you press CYCLE START.
Graphics	GRAPHICS	Opens Graphics mode.
Option Stop	OPTION STOP	Toggles optional stop on or off. When optional stop is on, the machine stops when it reaches M01 commands.
Block Delete	BLOCK DELETE	Toggles Block Delete On or Off. When Block Delete is On, the control ignores (does not execute) the code following a Forward Slash (/), on that same line.

4.5 | MILL - DISPLAY / MODE KEYS

NAME	KEY	FUNCTION
MDI MODE KEYS		
Manual Data Input	MDI	In MDI mode, you run unsaved programs or blocks of code entered from the control. Shows EDIT:MDI in upper left display.
Coolant	COOLANT	Turns the optional coolant on and off. Also, SHIFT + COOLANT turns on and off the optional Auto Air Gun / Minimum Quantity Lubrication functions
Handle Scroll	HANDLE SCROLL	Toggles Handle Scroll mode. This lets you use the jog handle to move the cursor in menus while the control is in jog mode.
Automatic Tool Changer Forward	ATC FWD	Rotates the tool carousel to the next tool.
Automatic Tool Changer Reverse	ATC REV	Rotates the tool carousel to the previous tool.
HANDLE JOG MODE KEYS		
Handle Jog	HANDLE JOG	Enters Jog mode.
.0001/.1 .001/1 .01/10 .1/100	.0001 /.1, .001 / 1., .01 / 10., .1 / 100.	Selects the increment for each click of the jog handle. When the mill is in MM mode the first number is multiplied by ten when jogging the axis (e.g., .0001 becomes 0.001 mm). The bottom number sets speed after you press JOG LOCK and an axis jog key or you press and hold an axis jog key. Shows SETUP:JOG in the upper left of the display.
ZERO RETURN MODE KEYS		
Zero Return	ZERO RETURN	Selects Zero Return mode, which displays axis location in four different categories: Operator, Work G54, Machine, and Dist (distance) To Go. Select the tab to switch between the categories. Shows SETUP:ZERO in the upper-left display.
All	ALL	Returns all axes to machine zero. This is similar to POWER UP, except a tool change does not occur.
Origin	ORIGIN	Sets selected values to zero.
Single	SINGLE	Returns one axis to machine zero. Press the desired axis letter on the Alpha keyboard and then press SINGLE
Home G28	HOME G28	Returns all axes to zero in rapid motion. HOME G28 will also home a single axis in the same manner as SINGLE. **CAUTION:** Make sure the axis motion paths are clear when you press this key. There is no warning or prompt before axis motion begins.

Mode Keys (Cont.)

NAME	KEY	FUNCTION
	LIST PROGRAM MODE KEYS	
List Programs	LIST PROGRAMS	Accesses a tabbed menu to load and save programs.
Select Programs	SELECT PROGRAMS	Makes the highlighted program the active program.
Back	BACK ARROW	Navigates to the screen you were on before the current one. This key operates like the BACK button on a web browser.
Forward	FORWARD ARROW	Navigates to the screen you went to after the current screen, if you have used the back arrow. This key operates like the FORWARD button on a web browser.
Erase Program	ERASE PROGRAM	Deletes the selected program in List Program mode. Deletes the entire program in MDI mode.

Numeric Keys

Use the numeric keys to type numbers, along with some special characters (printed in yellow on the main key). Press SHIFT to enter the special characters.

NAME	KEY	FUNCTION
Numbers	0-9	Types numbers.
Minus sign	-	Adds a minus (-) sign to the input line.
Decimal point	.	Adds a decimal point to the input line.
Cancel	CANCEL	Deletes the last character typed.
Space	SPACE	Adds a space to input.
Enter	ENTER	Answers prompts and writes input.
Special Characters	Press SHIFT, then a numeric key	Inserts the yellow character on the upper-left of the key. These characters are used for comments, macros, and certain special features.
+	SHIFT, then -	Inserts +
=	SHIFT, then 0	Inserts =
#	SHIFT, then .	Inserts #
*	SHIFT, then 1	Inserts *
‘	SHIFT, then 2	Inserts ‘
?	SHIFT, then 3	Inserts ?
%	SHIFT, then 4	Inserts %
$	SHIFT, then 5	Inserts $
!	SHIFT, then 6	Inserts !
&	SHIFT, then 7	Inserts &
@	SHIFT, then 8	Inserts @
:	SHIFT, then 9	Inserts :

Alpha Keys

Use the alpha keys to type the letters of the alphabet, along with some special characters (printed in yellow on the main key). Press SHIFT to enter the special characters.

NAME	KEY	FUNCTION
Alphabet	A-Z	Uppercase letters are the default. Press SHIFT and a letter key for lowercase.
End-of-block (EOB)	;	This is the end-of-block character, which signifies the end of a program line.
Parentheses	(,)	Separate CNC program commands from user comments. They must always be entered as a pair.
Shift	SHIFT	Accesses additional characters on the keyboard, or shifts to lower case alpha characters. The additional characters are seen in the upper left of some of the alpha and number keys.
Special Characters	Press SHIFT, then an alpha key	Inserts the yellow character on the upper-left of the key. These characters are used for comments, macros, and certain special features.
Forward Slash	SHIFT, then ;	Inserts /
Left Bracket	SHIFT, then (	Inserts [
Right Bracket	SHIFT, then)	Inserts]

4.7 | MILL - JOG / OVERRIDE KEYS

Jog Keys

NAME	KEY	FUNCTION
Chip Auger Forward	CHIP FWD	Starts the chip removal system in the forward direction (out of the machine).
Chip Auger Stop	CHIP STOP	Stops the chip removal system.
Chip Auger Reverse	CHIP REV	Starts the chip removal system in the "reverse" direction.
Axis Jog Keys	+X/-X, +Y/-Y, +Z/-Z, +A/C/-A/C and +B/-B (SHIFT +A/C/-A/C)	Jog axes manually. Press and hold the axis button, or press and release to select an axis and then use the jog handle.
Jog Lock	JOG LOCK	Works with the axis jog keys. Press JOG LOCK, then an axis button, and the axis moves until you press JOG LOCK again.
Coolant Up	CLNT UP	Moves the optional Programmable Coolant (P-Cool) nozzle up.
Coolant Down	CLNT DOWN	Moves the optional P-Cool nozzle down.
Auxiliary Coolant	AUX CLNT	Press this key in MDI mode to toggle the Through-Spindle Coolant (TSC) system operation, if equipped. Press SHIFT + AUX CLNT to toggle the Through Tool Air Blast (TAB) function, if equipped. Both functions also work in Run-Stop-Jog-Continue mode.

Override Keys

Overrides let you temporarily adjust the speeds and feeds in your program. For example, you can slow down rapids while you prove out a program, or adjust the feedrate to experiment with its effect on part finish, etc.

You can use Settings 19, 20, and 21 to disable the feedrate, spindle, and rapid overrides, respectively.

FEED HOLD acts as an override that stops rapid and feed moves when you press it. FEED HOLD also stops tool changes and part timers, but not tapping cycles or dwell timers.

Press CYCLE START to continue after a FEED HOLD. When the Setup Mode key is unlocked, the door switch on the enclosure also has a similar result but displays Door Hold when the door is opened. When the door is closed, the control is in Feed Hold and CYCLE START must be pressed to continue. Door Hold and FEED HOLD do not stop any auxiliary axes.

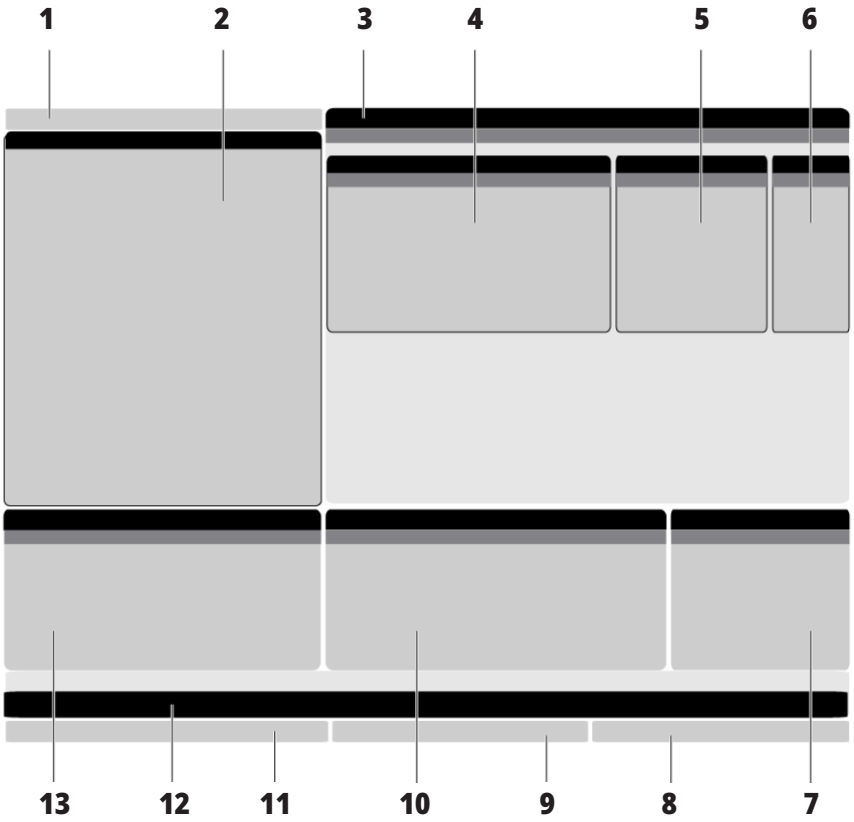
You can override the standard coolant setting by pressing COOLANT. The coolant pump remains either on or off until the next M-code or operator action (see Setting 32).

Use Settings 83, 87, and 88 to have M30 and M06 commands, or RESET, respectively, change overridden values back to their defaults.

NAME	KEY	FUNCTION
-10% Feedrate	-10% FEEDRATE	Decreases the current feedrate by 10%.
100% Feedrate	100% FEEDRATE	Sets an overridden feedrate back to the programmed feed rate.
+10% Feedrate	+10% FEEDRATE	Increases the current feedrate by 10%.
Handle Control Feed Rate	HANDLE FEED	Lets you use the jog handle to adjust the feedrate in 1% increments.
-10% Spindle	-10% SPINDLE	Decreases the current spindle speed by 10%
100% Spindle	100% SPINDLE	Sets the overridden spindle speed back to the programmed speed.
+10% Spindle	+10% SPINDLE	Increases the current spindle speed by 10%.
Handle Spindle	HANDLE SPINDLE	Lets you use the jog handle to adjust the spindle speed in 1% increments.
Forward	FWD	Starts the spindle in the clockwise direction.
Stop	STOP	Stops the spindle.
Reverse	REV	Starts the spindle in the counterclockwise direction.
Rapids	5% RAPID/ 25% RAPID/ 50% RAPID / 100% RAPID	Limits machine rapids to the value on the key.

Control Display

The control display is organized into panes that change with the different machine and display modes.



Basic Control Display Layout in Operation:Mem Mode (While a Program Runs)

1. Mode, Network, and Time Status Bar
2. Program Display
3. Main Display (size varies)/Program/ Offsets/Current Commands/ Settings/Graphics/Editor/VPS/Help
4. Active Codes
5. Active Tool
6. Coolant
7. Timers, Counters / Tool Management
8. Alarm Status
9. System Status Bar
10. Position Display / Axis Load
11. Input Bar
12. Icon Bar
13. Spindle Status

The active pane has a white background. You can work with data in a pane only when that pane is active, and only one pane is active at a time. For example, when you select the Tool Offsets tab, the offsets table background turns white. You can then make changes to the data. In most cases, you change the active pane with the display keys.

Tabbed Menu Basic Navigation

The Haas control uses tabbed menus for several modes and displays. Tabbed menus keep related data together in an easy-to-access format. To navigate these menus:

1. Press a display or mode key.

   The first time you access a tabbed menu, the first tab (or sub-tab) is active. The highlight cursor is at the first available option in the tab.

2. Use the cursor keys or the HANDLE JOG control to move the highlight cursor within an active tab.

3. To choose a different tab in the same tabbed menu, press the mode or display key again.

   NOTE: If the cursor is at the top of the menu screen, you can also press the UP cursor arrow key to select a different tab.

   The current tab becomes inactive.
4. Use the cursor keys to highlight a tab or a sub-tab, and press the DOWN cursor arrow key to use the tab.

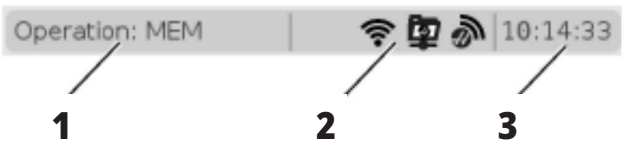
   NOTE: You cannot make the tabs active in the POSITIONS tabbed display.

5. Press a different display or mode key to work with a different tabbed menu.

Input Bar

The input bar is the data entry section located in the bottom-left corner of the screen. This is where your input appears as you type it.

Mode, Network, and Time Status Bar



This status bar in the top left of the screen is divided into three sections: mode, network, and time.

The Mode, Network, and Time Status bar shows [1] the current machine mode, [2] network status icons, and [3] the current time.

Mode, Key Access, and Mode Display

MODE [1]

The Haas control organizes machine functions into three modes: Setup, Edit, and Operation. Each mode shows on one screen all of the information you need to do tasks under that mode. For example, in Setup mode, you have access to the work offset table, the tool offset table, and position information. Edit mode gives you access to the program editor and optional systems like Visual Programming (VPS) (which contains Wireless Intuitive Probing (WIPS)). Operation mode includes Memory (MEM), the mode in which you run programs.

MODE	KEYS	DISPLAY [1]	FUNCTION
Setup	ZERO RETURN	SETUP: ZERO	Provides all control features for machine setup.
	HANDLE JOG	SETUP: JOG	
Edit	EDIT	ANY	Provides all program editing, management, and transfer functions.
	MDI	EDIT: MDI	
	LIST PROGRAM	ANY	
Operation	MEMORY	OPERATION: MEM	Provides all control features necessary to run a program.
	EDIT	OPERATION: MEM	Provides background editing of active programs.
	LIST PROGRAM	ANY	Provides background editing of programs.

Network

If you have networking installed on your Next Generation Control, icons in the center networking partition of the bar give you networking status. See the table for meanings of the networking icons.

	The machine is connected to a wired network with an Ethernet cable.
	The machine is connected to a wireless network with 70 - 100% signal strength.
	The machine is connected to a wireless network with 30 - 70% signal strength.
	The machine is connected to a wireless network with 1 - 30% signal strength.
	The machine is connected to a wireless network, but it is not receiving data packets.
	The machine is successfully registered with MyHaas and is communicating with the server.
	The machine had previously registered with MyHaas and has a problem connecting to the server.
	The machine is connected to a remote net share.

Settings Display

Press SETTING, then select the SETTINGS tab. Settings change the way the machine behaves; refer to the “Settings” section for a more detailed description.

Coolant Display

The coolant display appears in the upper-right of the screen in OPERATION:MEM mode.

The first line tells you if the coolant is ON or OFF.

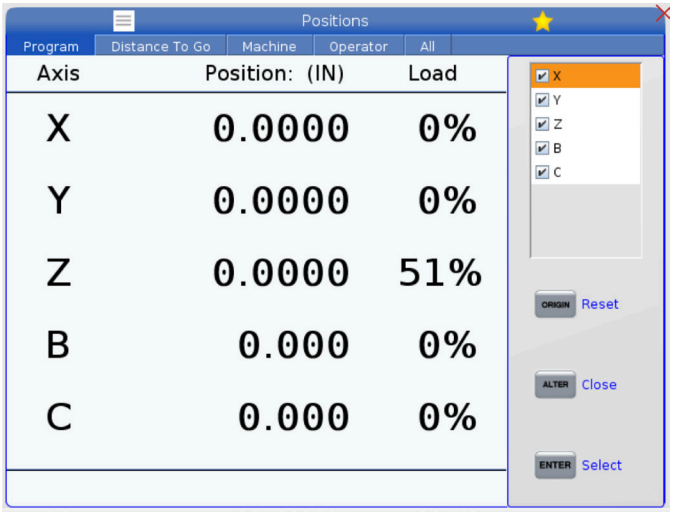
The next line shows the position number of the optional Programmable Coolant Spigot (P-COOL). The positions are from 1 to 34. If the option is not installed, no position number appears.

In the coolant gauge, a black arrow shows the coolant level. Full is 1/1 and empty is 0/1. To avoid coolant flow problems, keep the coolant level above the red range. You can also see this gauge in DIAGNOSTICS mode under the GAUGES tab.

Position Display

The Position display shows the current axis position relative to four reference points (Work, Distance-to-go, Machine, and Operator). In any mode, press POSITION and use cursor keys to access the different reference points displayed in tabs. The last tab display shows all the reference points on the same screen.

COORDINATE DISPLAY	FUNCTION
WORK (G54)	This tab displays the axis positions relative to part zero. On power-up, this position uses work offset G54 automatically. It displays the axis positions relative to the most recently-used work offset.
DIST TO GO	This tab displays the distance remaining before the axes reach their commanded position. When in SETUP:JOG mode, you can use this position display to show a distance moved. Change modes (MEM, MDI) and then switch back to SETUP:JOG mode to zero this value.
MACHINE	This tab displays the axis positions relative to machine zero.
OPERATOR	This tab shows the distance you have jogged the axes. This does not necessarily represent the actual distance the axis is from machine zero, except when the machine is first powered on.
ALL	This tab displays all reference points on the same screen.



Axis Display Selection

You can add or remove axes in the Positions displays. While a Positions display tab is active, press ALTER.

The axis display selection window comes in from the right side of the screen.

Use the cursor arrow keys to highlight an axis, and press ENTER to toggle it on and off for display. The positions display will show axes that have a check mark.

Press ALTER to close the axis display selector.

NOTE: You can display a maximum of (5) axes.

Offset Display

To access the offset tables, press OFFSET and select the TOOL tab or the WORK tab.

NAME	FUNCTION
TOOL	Display and work with tool numbers and tool length geometry.
WORK	Display and work with part zero locations.

Current Commands

This section describes the Current Commands pages and the types of data they show. The information from most of these pages also appears in other modes.

Press **CURRENT COMMANDS** to access the tabbed menu of available Current Commands displays.

Devices - The Mechanisms tab on this page shows hardware devices on the machine that you can command manually. For example, you can manually extend and retract the Parts Catcher or Probe Arm. You can also manually rotate the spindle clockwise or counterclockwise at a desired RPM.

Timers Display -This page shows:

- The current date and time.
- The total power on time.
- Total cycle start time.
- Total feed time.
- M30 counters. Each time the a program reaches an M30 command, both of these counters increment by one.
- Macro variable displays.

You also see these timers and counters in the lower-right section of the display in the OPERATION:MEM, SETUP:ZERO, and EDIT:MDI modes.

Macros Display - This page shows a list of the macro variables and their values. The control updates these variables as programs run. You can modify the variables in this display.

Active Codes - This page lists the active program codes. A smaller version of this display is included on the OPERATION:MEM and EDIT:MDI mode screens. Also when you press PROGRAM in any Operation mode, you see the active program codes.

Advanced Tool Management - This page contains information the control uses to predict tool life. Here is where you create and manage tool groups, and where you enter the maximum tool load percentage expected for each tool.

For more information, refer to the Advanced Tool Management section in the Operation chapter of this manual.

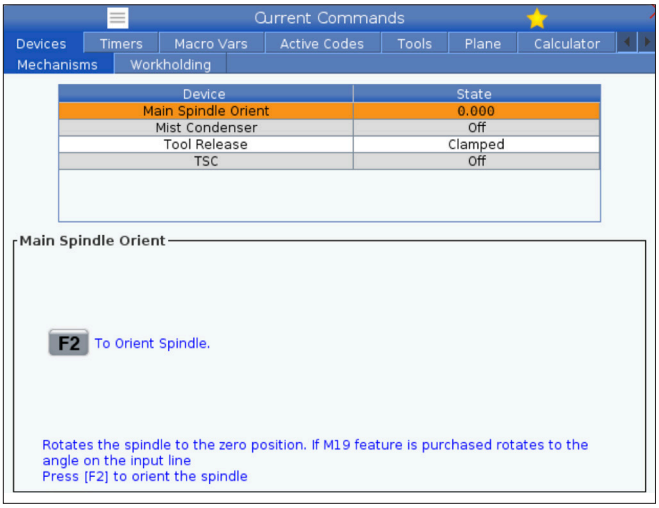
Calculator - This page contains the Standard, Milling/Turning, and Tapping calculators.

Media - This page contains the Media Player.

Devices - Mechanisms

The Mechanisms page displays possible machine components and options on your machine. Select the listed mechanism using the UP and DOWN arrows for more information on its operation and use. Pages give detailed instructions on the functions of the machine components, quick tips, as well as links to other pages to help you learn about and utilize your machine.

- Select the Devices tab in the Current Commands menu.
- Select the Mechanisms that you want to use.

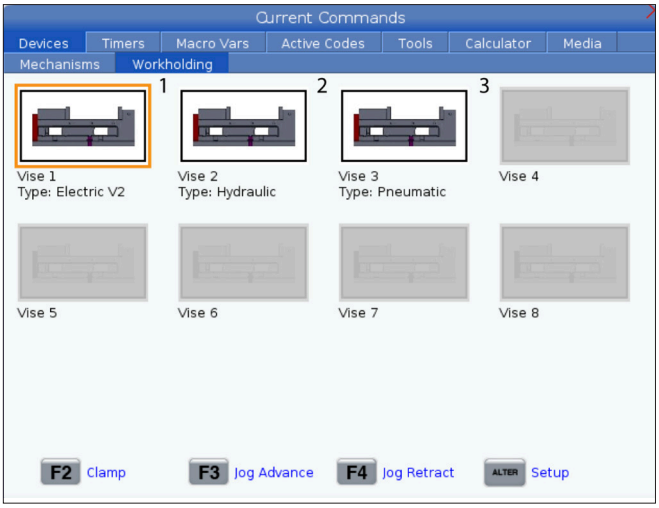


The Main Spindle option in Devices allows you to rotate the spindle clockwise or counterclockwise at a chosen RPM. The maximum RPM is limited by the machine's maximum RPM settings.

- Use the cursor arrow keys to move from field to field.
- Enter the RPM you want to rotate the spindle and press F2.
- Hold down F3 to rotate the spindle clockwise. Hold down F4 to rotate the spindle counterclockwise. The spindle comes to a stop when the button is released.

Devices - Workholding

Starting in software version **100.20.000.1110** a workholding tab was added to the control to support multiple workholding devices. The control will support Haas E-Vise [1], Hydraulic [2] and Pneumatic [3] vises.



The machine supports up to 3 foot pedals each would toggle Vise1, Vise2, and Vise3 respectively. If you have a single pedal you will need to enable Vise1 to the vise you want be actuated by the foot pedal.

NOTE: The E-vise is used on the Mill APL and Robot systems but can also be used as a stand-alone product.

You can actuate up to 8 workholding devices.

To access the Workholding page press Current Commands and navigate to **Devices > Workholding.**

From the Workholding display tab you will be able to:

- Setup the workholding devices
- Enable and Disable workholding devices
- Clamp and Unclamp
- Jog Advance / Retract (E-Vises only)

Time Adjustment

Follow this procedure to adjust the date or time.

1. Select the Timers page in Current Commands.
2. Use the cursor arrow keys to highlight the Date:, Time:, or Time Zone field.
3. Press **[EMERGENCY STOP].**
4. In the Date: field, type the new date in the format **MM-DD-YYYY**, including the hyphens.
5. In the Time: field, type the new time in the format **HH:MM**, including the colon. Press **[SHIFT]** and then 9 to type the colon.
6. In the Time Zone: field, press **[ENTER]** to select from the list of time zones. You can type search terms in the pop-up window to narrow the list. For example, type PST to find Pacific Standard Time. Highlight the time zone you want to use.
7. Press **[ENTER]**.

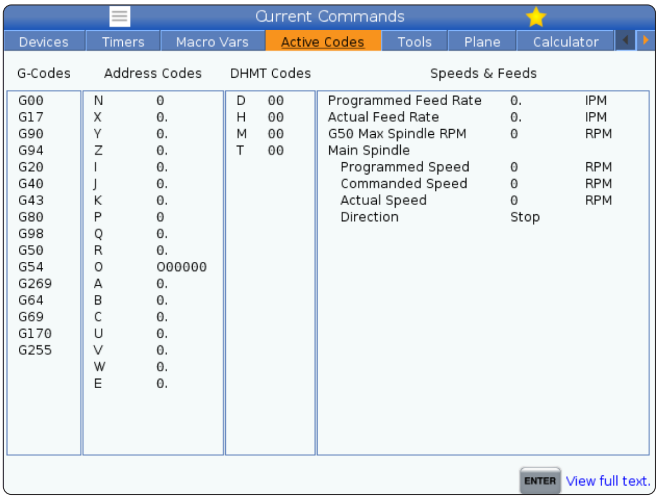
Timer and Counter Reset

You can reset the power-on, cycle-start, and feed cutting timers. You can also reset the M30 counters.

1. Select the Timers page in Current Commands.
2. Use the cursor arrow keys to highlight the name of the timer or counter that you want to reset.
3. Press ORIGIN to reset the timer or counter.

tip: You can reset the M30 counters independently to track finished parts in two different ways; for example, parts finished in a shift and total parts finished.

Current Commands - Active Codes



This display gives read-only, real-time information about the codes that are currently active in the program; specifically,

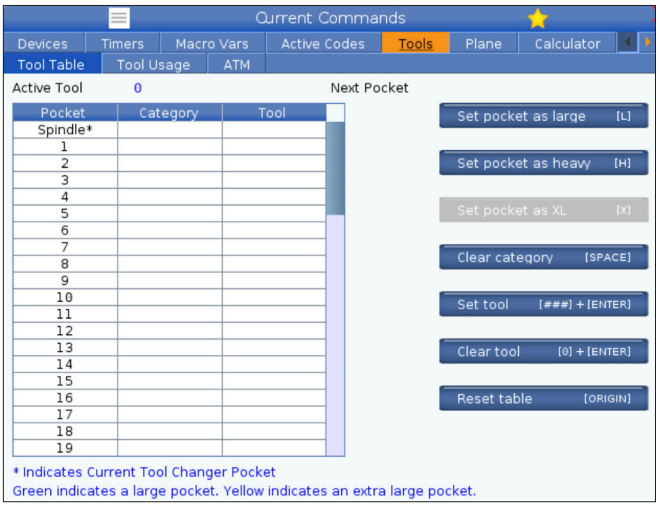
- the codes that define the current motion type (rapid vs linear feed vs circular feed)
- positioning system (absolute vs incremental)
- cutter compensation (left, right or off)
- active canned cycle, and work offset.

This display also gives the active Dnn, Hnn, Tnn, and most recent M-code. If an alarm is active, this shows a quick display of the active alarm instead of the active codes.

Tools - Tool Table

This section tells you how to use the tool table to give the control information about your tools.

To access the Tool Pocket Table, press CURRENT COMMANDS and choose the Tool Table tab.



Active Tool- Tells you the tool number that is installed in the spindle.

Active Pocket - this show you the next pocket number.

Set pocket as Large [L] - Use this flag when a large tool has a diameter of greater than 3" for 40-taper machines and greater than 4" for 50-taper machines. Scroll to the pocket of interest and press L to set the flag.

CAUTION: You cannot place a large tool in the tool changer if one or both of the surrounding pockets already contain tools. Doing so causes the tool changer to crash. Large tools must have the surrounding pockets empty. However, large tools can share adjoining empty pockets.

Set **pocket as heavy [H] -** Use this flag when a Heavy, Small Diameter 40-taper tool (4 lb or heavier) or a 50-taper tool (12 lb or heavier) is loaded into the spindle. Scroll to the pocket of interest and press H to set the flag.

Set **pocket as XL [X] -** Use this flag when two adjacent pockets at each side of the tool are needed. Scroll to the pocket of interest and press X to set the flag.

NOTE: This option only appears if your machine is a 50-taper.

Clear category [Space] - Highlight the desired tool and press SPACE to clear the flag.

Set tool [###] + [Enter]- Highlight the desired pocket and type in the tool number + Enter to set the desired tool number.

NOTE: You cannot assign a tool number to more than one pocket. If you enter a tool number that is already defined in the tool pocket table, you see an Invalid tool error.

Clear tool [0] + [Enter]- Highlight the desired pocket and press 0 + Enter to clear out the tool number.

Reset table [Origin] - Press ORIGIN with the cursor in the center column to use the ORIGIN menu. This menu lets you:

Sequence All Pockets - Makes all of the tool numbers sequential based on their pocket location, starting with 1.

Zero All Pockets - Removes all of the tool numbers from all of the pocket numbers.

Clear Category Flags - Removes the category designations from all of the tools.

***** Indicates current tool changer pocket.

Tools - Tool Usage

The **Tool Usage** tab contains information about the tools used in a program. This display will tell you information about each tool used in a program and statistics about each time it was used. It starts to collect information when user Main program start and clear information when meet codes M99, M299, M199.

To get to the Tool Usage display press CURRENT COMMANDS then go to Tools and then Tool Usage tab.

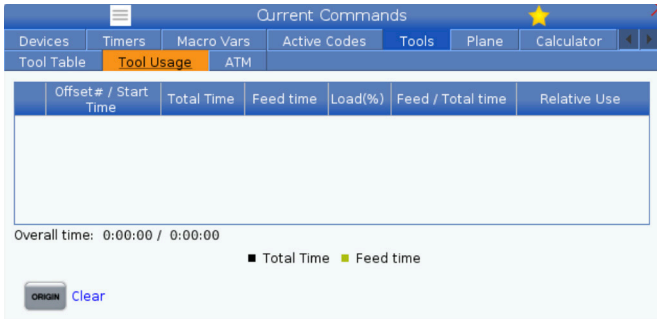
Start Time - When the tool was inserted to the spindle.

Total Time - The total time the tool has been in the spindle.

Feed time - Tool usage time.

Load% - The maximum load of the Spindle during a tool usage.

NOTE: This value is retrieved every second. The actual load compared to the recorded may vary.



Feed/Total time - A graphic representation of the feed time of the tool over the total time.

Engagement:

- Black Bar- The tool usage versus another tools.
- Gray Bar - This bar shows how long the tool was used in this usage related to other usages.

Macros Interface You can use these macros variables to set and collect the tool usage data.

MACRO VARIABLE	FUNCTION
#8608	Set the desired tool
#8609	Current tool number - if result more 0 (the tool was used)
#8610	Total time mentioned in #8609 tool number
#8611	Feed time of mentioned tool number
#8612	Total time
#8605	Next usage of a tool
#8614	Usage start time stamp
#8615	Usage Total time
#8616	Usage Feed time
#8617	Usage Max load

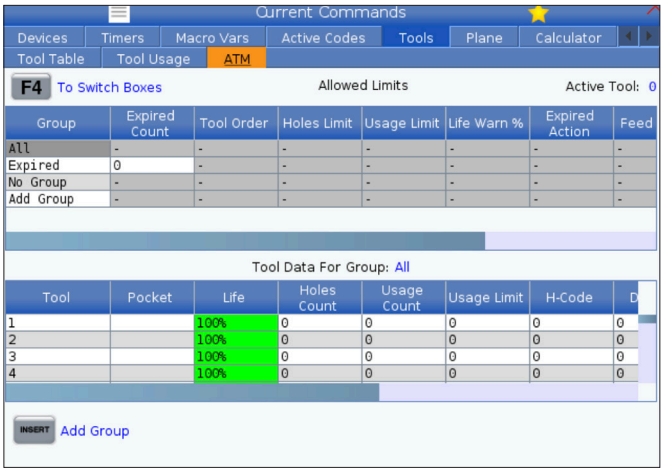
Tools - ATM

Advanced Tool Management (ATM) lets you set up groups of duplicate tools for the same or a series of jobs.

ATM classifies duplicate or backup tools into specific groups. In your program, you specify a group of tools instead of a single tool. ATM tracks the tool use in each tool group and compares it to your defined limits. When a tool reaches a limit, the control considers it "expired." The next time your program calls that tool group, the control chooses a non-expired tool from the group.

- When a tool expires:
- The beacon will flash.
- ATM puts the expired tool in the EXP group

Tool groups that contain the tool appear with a red background.



To use ATM, press CURRENT COMMANDS, and then select ATM in the tabbed menu. The ATM window has two sections: Allowed Limits and Tool Data.

ALLOWED LIMITS

This table gives data about all of the current tool groups, including default groups and user-specified groups. ALL is a default group that lists all of the tools in the system. EXP is a default group that lists all of the tools that are expired. The last row in the table shows all of the tools that are not assigned to tool groups. Use the cursor arrow keys or END to move the cursor to the row and see these tools.

For each tool group in the ALLOWED LIMITS table, you define limits that determine when a tool expires. The limits apply to all tools assigned to this group. These limits affect every tool in the group.

The columns in the ALLOWED LIMITS table are:

GROUP - Displays the tool group's ID number. This is the number you use to specify the tool group in a program.

EXP # - Tells you how many tools in the group are expired. If you highlight the ALL row, you see a list of all of the expired tools in all groups.

ORDER - Specifies the tool to use first. If you select ORDERED, ATM uses the tools in tool number order. You can also have ATM automatically use the NEWEST or OLDEST tool in the group.

USAGE - The maximum number of times the control can use a tool before it expires.

HOLES - The maximum number of holes a tool is allowed to drill before it expires.

WARN - The minimum value for tool life remaining in the group before the control gives a warning message.

LOAD - The allowed load limit for tools in the group before the control does the ACTION that the next column specifies.

ACTION - The automatic action when a tool reaches its maximum tool load percentage. Highlight the tool action box to change and press ENTER. Use the UP and DOWN cursor keys to select the automatic action from the pull down menu (ALARM, FEEDHOLD, BEEP, AUTOFEED, NEXT TOOL).

FEED - The total amount of time, in minutes, that the tool can be in a feed.

TOTAL TIME - The total amount of time, in minutes, that the control can use a tool.

TOOL DATA

This table gives information about each tool in a tool group. To look at a group, highlight it in the ALLOWED LIMITS table, and then press F4.

TOOL# - Shows the tool numbers used in the group.

LIFE - The percentage of life left in a tool. This is calculated by the CNC control, using actual tool data and the allowed limits the operator entered for the group.

USAGE - The total number of times that a program has called the tool (number of tool changes).

HOLES - The number of holes the tool has drilled/ tapped/ bored.

LOAD - The maximum load, in percent, exerted on the tool.

LIMIT - The maximum load allowed for the tool

FEED - The amount of time, in minutes, the tool has been in a feed.

TOTAL - The total amount of time, in minutes, the tool has been used.

H-CODE - The tool-length code to use for the tool. You can edit this only if Setting 15 is set to OFF.

D-CODE - The diameter code to use for the tool.

NOTE: By default, the H- and D-codes in Advanced Tool Management are set to equal the tool number that is added to the group.

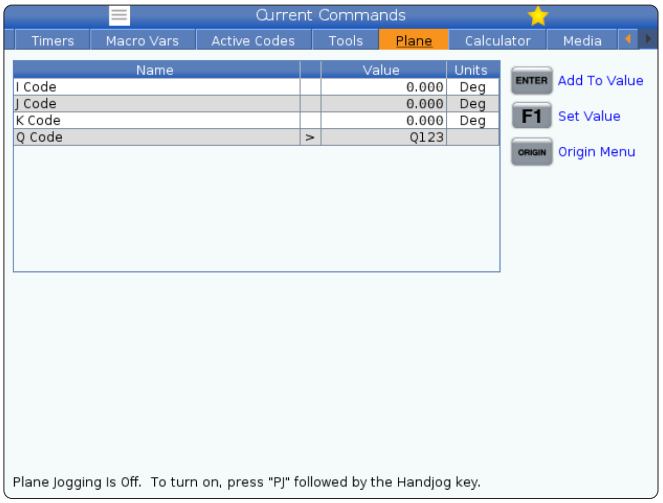
Plane

The Plane tab allows for a machine with a Gimbal Spindle define custom planes for jogging.

The plane tab can used in conjunction with G268 run in a program or from filling out the required fields.

Each one of the required fields has a help text at the bottom of the table.

To enter plane jogging mode type in "**PJ**" followed by **[HAND JOG]**.



Calculator

The calculator tab includes calculators for basic mathematical functions, milling, and tapping.

- Select the calculator tab in the Current Commands menu.
- Select the calculator tab that you want to use: Standard, Milling, or Tapping.

The standard calculator has functions like a simple desktop calculator; with available operations such as addition, subtraction, multiplication, and division, as well as square root and percentage. The calculator lets you easily transfer operations and results to the input line so that you can put them into programs. You can also transfer results into the Milling and Tapping calculators.

Use the number keys to type operands into the calculator.

Calculator (Contin.)

To insert an arithmetic operator, use the letter key that appears in brackets next to the operator you want to insert. These keys are:

KEY	FUNCTION	KEY	FUNCTION
D	Add	K	Square Root
J	Subtract	Q	Percentage
P	Multiply	S	Memory Store (MS)
V	Divide	R	Memory Recall (MR)
E	Toggle sign (+/-)	C	Memory Clear (MC)

After you have entered data into the calculator input field, you can do any of the following:

NOTE: These options are available for all calculators.

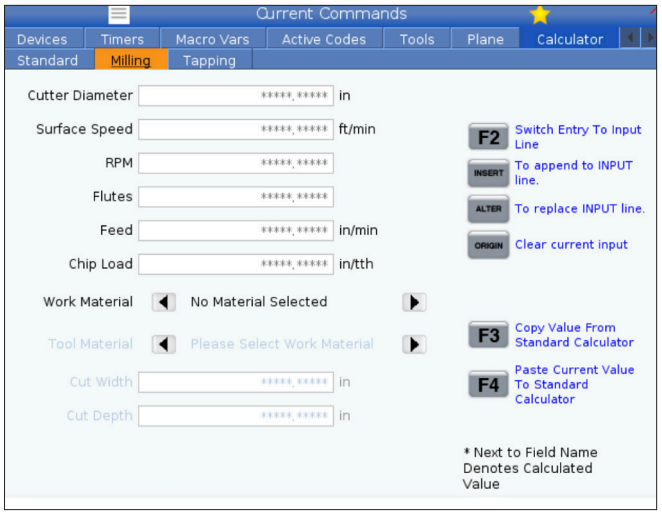
- Press ENTER to return the result of your calculation.
- Press INSERT to append the data or the result to the end of the input line.
- Press ALTER to move the data or the result to the input line. This overwrites the current contents of the input line.
- Press ORIGIN to reset the calculator.

Keep the data or the result in the calculator input field and select a different calculator tab. The data in the calculator input field remains available to transfer into the other calculators.

Milling /Turning Calculator

The milling/turning calculator lets you automatically calculate machining parameters based on given information. When you have entered enough information, the calculator automatically displays results in the relevant fields. These fields are marked with an asterisk (*).

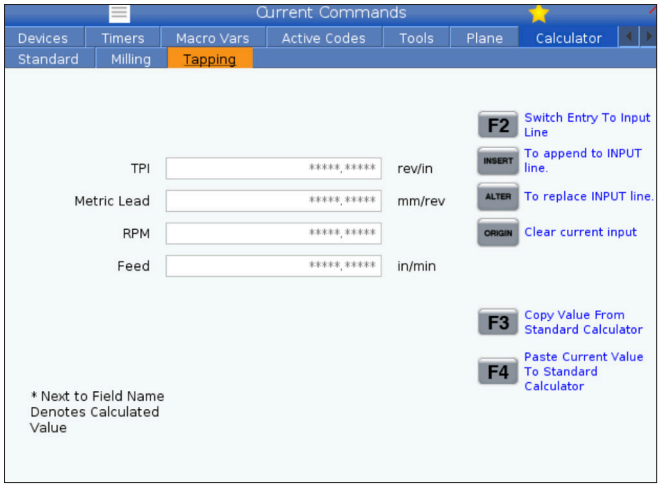
- Use the cursor arrow keys to move from field to field.
- Type known values in the appropriate fields. You can also press F3 to copy a value from the standard calculator.
- In the Work Material and Tool Material fields, use the LEFT and RIGHT cursor arrow keys to choose from the available options.
- wCalculated values appear highlighted in yellow when they are outside of the recommended range for the workpiece and tool material. Also, when all of the calculator fields contain data (calculated or entered), the milling calculator displays the recommended power for the operation.



Tapping Calculator

The tapping calculator lets you automatically calculate tapping parameters based on given information. When you have entered enough information, the calculator automatically displays results in the relevant fields. These fields are marked with an asterisk (*).

- Use the cursor arrow keys to move from field to field.
- Type known values in the appropriate fields. You can also press F3 to copy a value from the standard calculator.
- When the calculator has enough information, it puts calculated values in the appropriate fields.



Media Display

M130 Lets you display video with audio and still images during program execution. Some examples of how you can use this feature are:

Providing visual cues or work instructions during program operation

Providing images to aid part inspection at certain points in a program

Demonstrating procedures with video

The correct command format is M130(file.xxx), where file.xxx is the name of the file, plus the path, if necessary. You can also add a second comment in parentheses to appear as a comment in the media window.

Example: M130(Remove Lifing Bolts Before Starting Op 2) (User Data/My Media/loadOp2.png);

NOTE: M130 uses the subprogram search settings, Settings 251 and 252 in the same way that M98 does. You can also use the Insert Media File command in the editor to easily insert an M130 code that includes the file path. Refer to page 5 for more information.

$FILE Lets you display video with audio and still images outside of program execution.

The correct command format is ($FILE file.xxx), where file.xxx is the name of the file, plus the path, if necessary. You can also add a comment between the first parentheses and the dollar sign to appear as a comment in the media window.

To display the media file, highlight the block while in memory mode and press enter. $FILE media display block will be ignored as comments during program execution.

Example: (Remove Lifing Bolts Before Starting Op 2 $FILE User Data/My Media/loadOp2.png);

STANDARD	PROFILE	RESOLUTION	BITRATE
MPEG-2	Main-High	1080 i/p, 30 fps	50 Mbps
MPEG-4 / XviD	SP/ASP	1080 i/p, 30 fps	40 Mbps
H.263	P0/P3	16 CIF, 30fps	50 Mbps
DivX	3/4/5/6	1080 i/p, 30fps	40 Mbps
Baseline	8192 x 8192	120 Mpixel/sec	-
PNG	-	-	-
JPEG	-	-	-

NOTE: For the fastest loading times, use files with pixel dimensions divisible by 8 (most unedited digital images have these dimensions by default), and a maximum resolution of 1920 x 1080.

Your media appears in the Media tab under Current Commands. The media displays until the next M130 displays a different file, or M131 clears the media tab contents.

Example: (Remove Lifing Bolts Before Starting Op 2 $FILE User Data/My Media/loadOp2.png);

Alarms and Messages Display

Use this display to learn more about machine alarms when they occur, view your machine's entire alarm history, look up definitions of alarms that can occur, view created messages, and show the keystroke history.

Press ALARMS, then select a display tab:

The ACTIVE ALARM tab shows the alarms that currently affect machine operation. Use PAGE UP and PAGE DOWN to see the other active alarms.

The MESSAGES tab shows the messages page. The text you put on this page stays there when you power the machine off. You can use this to leave messages and information for the next machine operator, etc.

The ALARM HISTORY tab shows a list of the alarms that have recently affected machine operation. You can also search for an alarm number or alarm text. To do this type in the alarm number or the desired text and press F1.

The ALARM VIEWER tab shows a detailed description of all the alarms. You can also search for an alarm number or alarm text. To do this type in the alarm number or the desired text and press F1.

The KEY HISTORY tab shows up to the last 2000 keystrokes.

Example: (Remove Lifing Bolts Before Starting Op 2 $FILE User Data/My Media/loadOp2.png);

Add Messages

You can save a message in the MESSAGES tab. Your message stays there until you remove it or change it, even when you turn the machine off.

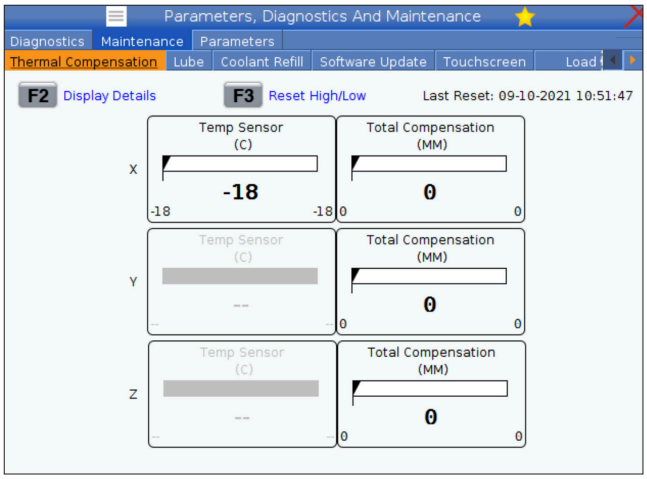
1. Press ALARMS, select the MESSAGES tab, and press the DOWN cursor arrow key.
2. Type your message.
   Press CANCEL to backspace and delete. Press DELETE to delete an entire line. Press ERASE PROGRAM to delete the entire message.

Maintenance

Thermal Compensation tab under Maintenance in Diagnostics that was released in software version **100.21.000.1130**.

This tab has two options to switch between, a simple gauge version and a more detailed view.

NOTE: For now, this tab is purely for informational purposes.



Device Manager (List Program)

You use the device manager (LIST PROGRAM) to access, save, and manage data on the CNC control, and on other devices attached to the control. You also use the device manager to load and transfer programs between devices, set your active program, and back up your machine data.

In the tabbed menu at the top of the display, the device manager (LIST PROGRAM) shows you only the available memory devices. For example, if you do not have a USB memory device connected to the control pendant, the tabbed menu does not show a USB tab. For more information on navigating tabbed menus, refer to Chapter 5.1.

The device manager (LIST PROGRAM) shows you the available data in a directory structure. At the root of the CNC control are the available memory devices in a tabbed menu. Each device can contain combinations of directories and files, many levels deep. This is similar to the file structure you find in common personal computer operating systems.

Device Manager Operation

Press LIST PROGRAM to access the device manager. The initial device manager display shows the available memory devices in a tabbed menu. These devices can include machine memory, the User Data directory, USB memory devices connected to the control, and files available on the connected network. Select a device tab to work with the files on that device.

Device Manager Initial Screen Example:

[1] Available Device Tabs,

[2] Search Box,

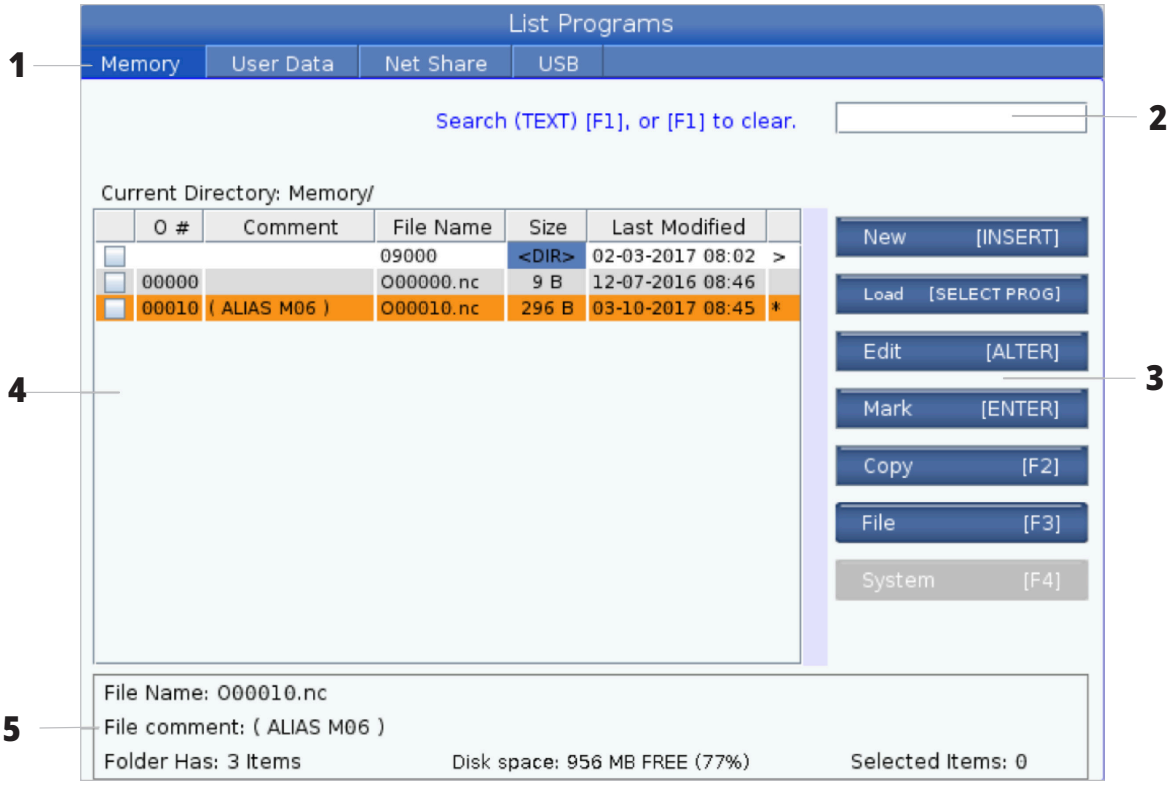
[3] Function Keys,

[4] File Display,

[5] File Comments (only available in Memory).

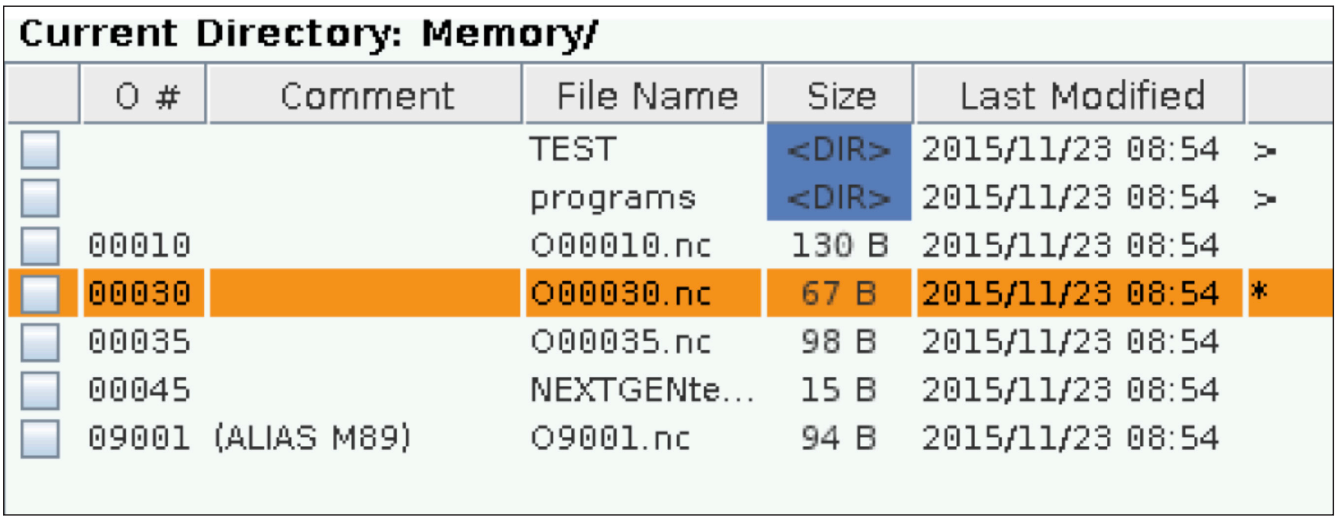
Use the cursor arrow keys to navigate the directory structure:

- Use the UP and DOWN cursor arrow keys to highlight and interact with a file or a directory in the current root or directory.
- Roots and directories have a right-hand arrow character (>) in the far-right column of the file display. Use the RIGHT cursor arrow key to open a highlighted root or directory. The display then shows the contents of that root or directory.
- Use the LEFT cursor arrow key to return to the previous root or directory. The display then shows the contents of that root or directory
- The CURRENT DIRECTORY message above the file display tells you where you are in the directory structure; for example: MEMORY/CUSTOMER 11/NEW PROGRAMS shows that you are in the subdirectory NEW_PROGRAMS inside the directory CUSTOMER 11, in the root of MEMORY.



File Display Columns

When you open a root or directory with the RIGHT cursor arrow key, the file display shows you a list of the files and directories in that directory. Each column in the file display has information about the files or directories in the list.



The columns are:

- **File selection check box (no label):** Press ENTER to toggle a check mark on and off in the box. A check mark in a box indicates that the file or directory selected for operations on multiple files (usually copy or delete).
- **Program O Number (O #):** This column lists the program numbers of the programs in the directory. The letter 'O' is omitted in the column data. Only available in the Memory tab.
- **File comment (Comment):** This column lists the optional program comment that appears in the first line of the program. Only available in the Memory tab.
- **File Name (File Name):** This is an optional name that the control uses when you copy the file to a memory device other than the control. For example, if you copy program O00045 to a USB memory device, the filename in the USB directory is NEXTGENtest.nc.
- **Size (Size):** This column shows the amount of storage space that the file takes up. Directories in the list have the designation <DIR> in this column.

  NOTE: This column is hidden by default, press the F3 button and select Show File Details to display this column.
- **Date Last Modified (Last Modified):** This column shows the last date and time when the file was changed. The format is YYYY/MM/DD HR:MIN.

  NOTE: This column is hidden by default, press the F3 button and select Show File Details to display this column.
- **Other Information (no label):** This column gives you some information about a file's status. The active program has an asterisk (*) in this column. A letter E in this column means that the program is in the program editor. A greater-than symbol (>) indicates a directory. A letter S indicates that a directory is part of Setting 252. Use the RIGHT or LEFT cursor arrow keys to enter or exit the directory.

Check Mark Selection

The check box column on the far left of the file display lets you select multiple files.

Press ENTER to place a check mark in a file's check box. Highlight another file and press ENTER again to put a check mark in that file's check box. Repeat this process until you have selected all of the files you want to select.

You can then do an operation (usually copy or delete) on all of those files at the same time. Each file that is part of your selection has a check mark in the check box. When you choose an operation, the control does that operation on all of the files with check marks.

For example, if you want to copy a set of files from the machine memory to a USB memory device, you would put a check mark on all of the files that you want to copy, then press F2 to start the copy operation.

To delete a set of files, put a check mark on all of files you want to delete, then press DELETE to start the delete operation.

NOTE: A check-mark selection only marks the file for further operation; it does not make the program active.

NOTE: If you have not selected multiple files with check marks, the control does operations only on the currently highlighted directory or file. If you have selected files, the control does operations only on the selected files and not on the highlighted file, unless it is also selected.

Select the Active Program

Highlight a program in the memory directory, then press SELECT PROGRAM to make the highlighted program active.

The active program has an asterisk (*) in the far-right column in the file display. It is the program that runs when you press CYCLE START in OPERATION:MEM mode. The program is also protected from deletion while it is active.

Create a New Program

Press INSERT to create a new file in the current directory. The CREATE NEW PROGRAM popup menu shows on the screen:

Create New Program Popup Menu Example: [1] Program O number field, [2] File Name field, [3] File comment field.

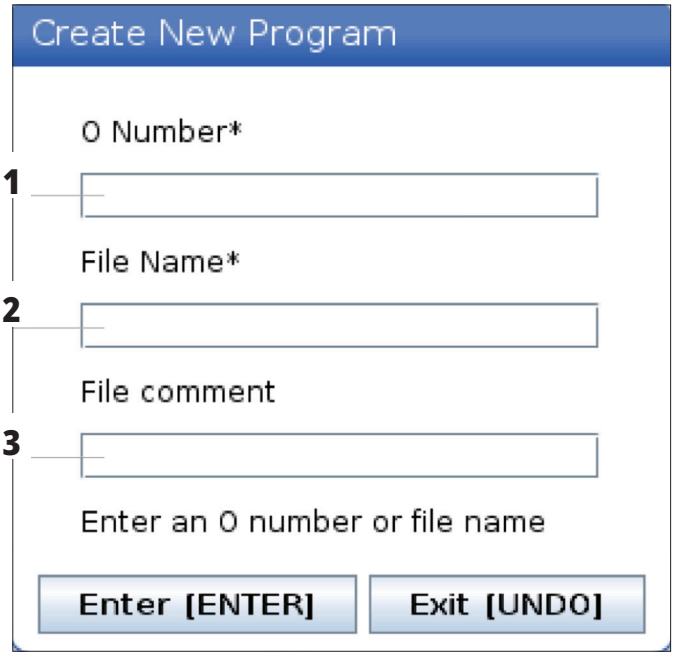
Enter the new program information in the fields. The Program O number field is required; the File Name and File comment are optional. Use the UP and DOWN cursors to move between the menu fields.

Press **UNDO** at any time to cancel program creation.

- Program O number (required for files created in Memory): Enter a program number up to (5) digits long. The control adds the letter O automatically. If you enter a number shorter than (5) digits, the control adds leading zeros to the program number to make it (5) digits long; for example, if you enter 1, the control adds zeros to make it 00001.

NOTE: Do not use O09XXX numbers when you create new programs. Macro programs often use numbers in this block, and overwriting them may cause machine functions to malfunction or stop working.

File Name (optional): Type a filename for the new program. This is the name the control uses when you copy the program to a storage device other than memory.



File comment (optional): Type a descriptive program title. This title goes into the program as a comment in the first line with the O number.

Press ENTER to save your new program. If you specified an O number that exists in the current directory, the control gives the message File with O Number nnnnn already exists. Do you want to replace it? Press ENTER to save the program and overwrite the existing program, press CANCEL to return to the program name popup, or press UNDO to cancel.

Edit a Program

Highlight a program, and then press **ALTER** to move the program into the program editor.

The program has the designation E in the far-right column of the file display list when it is in the editor, unless it is also the active program.

You can use this function to edit a program while the active program runs. You can edit the active program, but your changes do not take effect until you save the program and then select it again in the device manager menu.

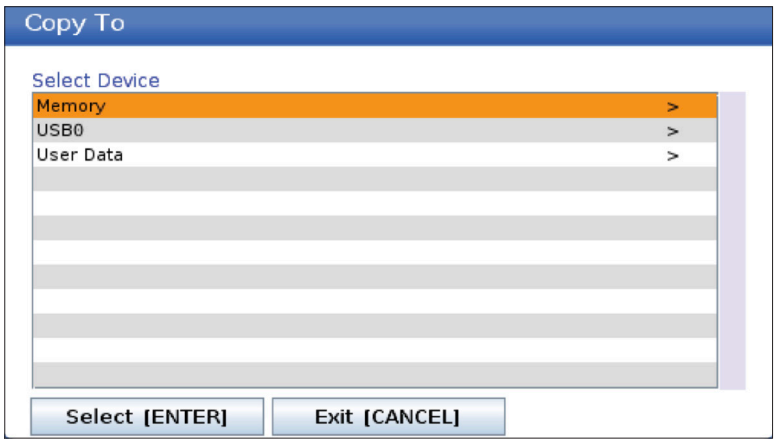
Copy Programs

This function lets you copy programs to a device or a different directory.

To copy a single program, highlight it in the device manager program list and press **ENTER** to assign a check mark. To copy multiple programs, check-mark all the programs you want to copy.

Press **F2** to start the copy operation.

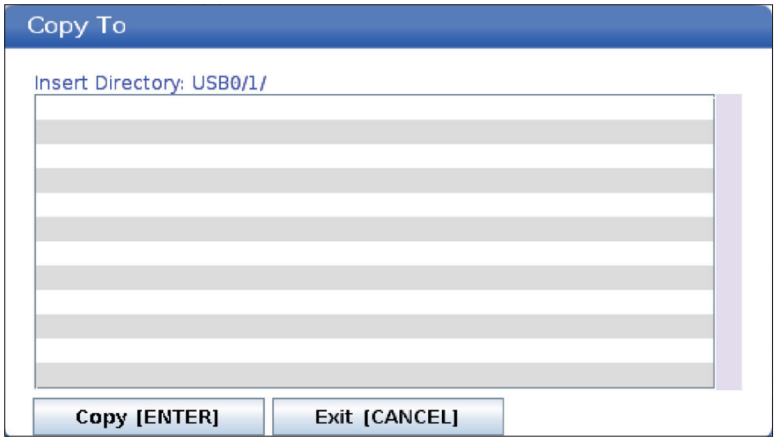
The Select Device popup appears.



Select Device

Use the cursor arrow keys to select the destination directory. RIGHT cursor to enter the chosen directory.

Press **ENTER** to complete the copy operation, or press **CANCEL** to return to the device manager.



Create / Select Programs for Editing

You use the Device Manager (LIST PROGRAM) to create and select programs for editing. Refer to the CREATE, EDIT, COPY A PROGRAM tab to create a new program.

Program Edit Modes

You use the Device Manager (LIST PROGRAM) to create and select programs for editing. Refer to the CREATE, EDIT, COPY A PROGRAM tab to create a new program.

The Haas control has (2) program edit modes: The program editor or manual data input (MDI). You use the program editor to make changes to numbered programs stored in an attached memory device (machine memory, USB, or net share). You use MDI mode to command the machine without a formal program.

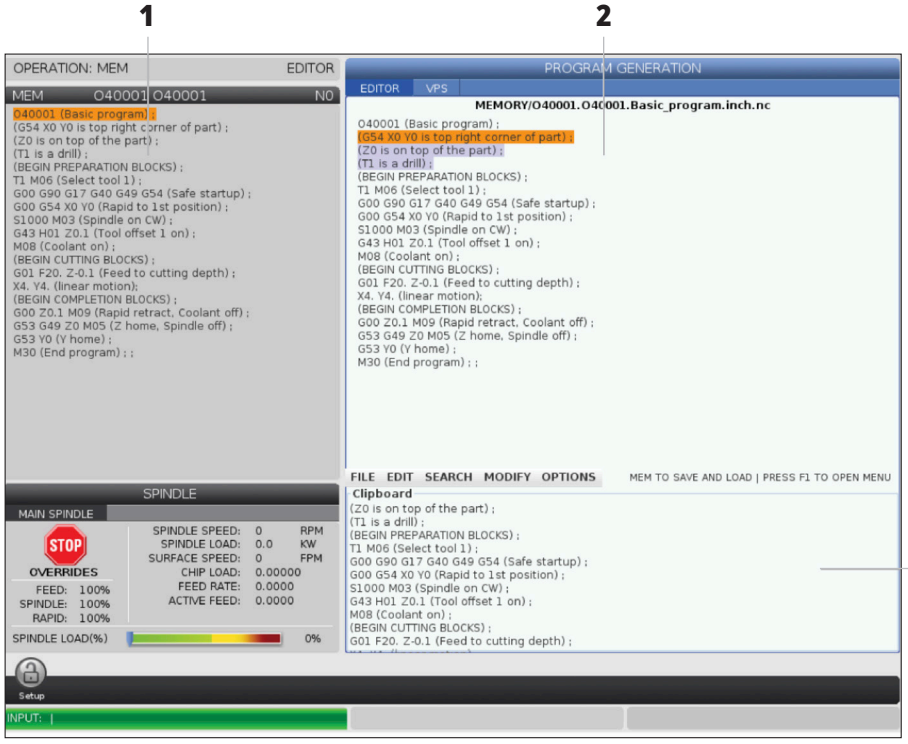
The Haas control screen has (2) program edit panes: The Active Program / MDI pane, and the Program Generation pane. The Active Program / MDI pane is on the left side of the screen in all display modes. The Program Generation pane appears only in EDIT mode.

Example Edit Panes.

[1] Active Program / MDI Pane,

[2] Program Edit Pane,

[3] Clipboard Pane



Basic Program Editing

This section describes the basic program editing functions. These functions are available when you edit a program.

1) To write a program, or make changes to a program:

- To edit a program in MDI, press MDI. This is EDIT:MDI mode. The program is displayed on the Active pane.
- To edit a numbered program, select it in the Device Manager (LIST PROGRAM), then press EDIT. This is EDIT:EDIT mode. The program is displayed in the Program Generation pane.

2) To highlight code:

- Use the cursor arrow keys or the jog handle to move the highlight cursor through the program.
- You can interact with single pieces of code or text (cursor highlighting), blocks of code, or multiple blocks of code (block selection). Refer to the Block Selection section for more information.

3) To add code to the program:

- Highlight the code block you want the new code to follow.
- Type the new code.
- Press INSERT. Your new code appears after the block you highlighted.

4) To replace code:

- Highlight the code you want to replace.
- Type the code you want to replace the highlighted code with.
- Press ALTER. Your new code takes the place of the code you highlighted.

5) To remove characters or commands:

- Highlight the text you want to delete.
- Press DELETE. The text that you highlighted is removed from the program.

6) Press UNDO to reverse up to the last (40) changes.

NOTE: You cannot use UNDO to reverse changes that you made if you exit EDIT:EDIT mode.

NOTE: In EDIT:EDIT mode, the control does not save the program as you edit. Press MEMORY to save the program and load it into the Active Program pane.

Block Selection

When you edit a program, you can select single or multiple blocks of code. You can then copy and paste, delete, or move those blocks in one step.

To select a block:

- Use the cursor arrow keys to move the highlight cursor to the first or last block in your selection.

NOTE: You can start a selection at the top block or the bottom block, and then move up or down as appropriate to complete your selection.

NOTE: You cannot include the program name block in your selection. The control gives the message GUARDED CODE.

- Press F2 to start your selection.
- Use the cursor arrow keys or the jog handle to expand the selection.
- Press F2 to complete the selection.

Actions with a Block Selection

After you make a text selection, you can copy and paste it, move it, or delete it.

NOTE: These instructions assume you have already made a block selection as described in the Block Selection section.

NOTE: These are actions available in MDI and the Program Editor. You cannot use UNDO to reverse these actions.

1) To copy and paste the selection:

- Move the cursor to the location where you want to put a copy of the text.
- Press ENTER.

The control puts a copy of the selection on the next line after the cursor location.

NOTE: The control does not copy the text to the clipboard when you use this function.

2) To move the selection:

- Move the cursor to the location where you want to move the text.
- Press ALTER.

The control removes the text from its current location and puts it in the line after the current line.

3) Press DELETE to delete the selection.

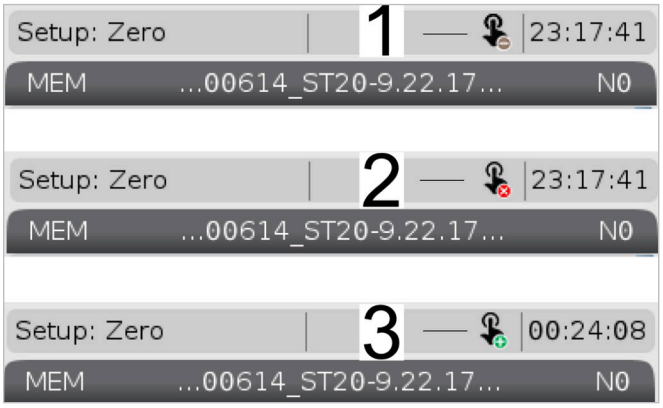
LCD Touchscreen Overview

The touchscreen feature allows you to navigate the control in a more intuitive way.

NOTE: If the touchscreen hardware is not detected at power on, a notification 20016 Touchscreen not detected will appear in the alarm history.

SETTINGS
381 - Enable / Disable Touchscreen
383- Table Row Size
396 - Virtual Keyboard Enabled
397 - Press and Hold Delay
398 - Header Height
399 - Tab Height
403 - Choice Popup Button Size

Touchscreen Status Icons



[1] Software does not support Touchscreen

[2] Touchscreen is Disabled

[3] Touchscreen is Enabled

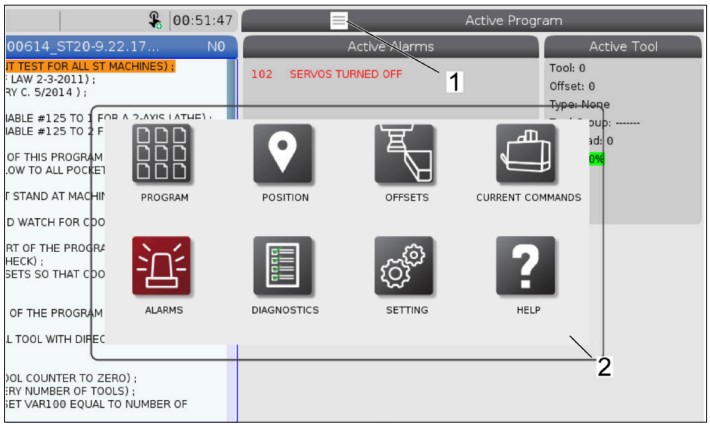
An icon appears on top left of the screen when the touchscreen is enabled or disabled.

Functions excluded from Touchscreen

FUNCTION	TOUCHSCREEN
RESET	Not available
Emergency Stop	Not available
Cycle Start	Not available
Feed Hold	Not available

LCD Touchscreen - Navigation Tiles

Press the Menu[1] icon on the screen to display the display icons [2].

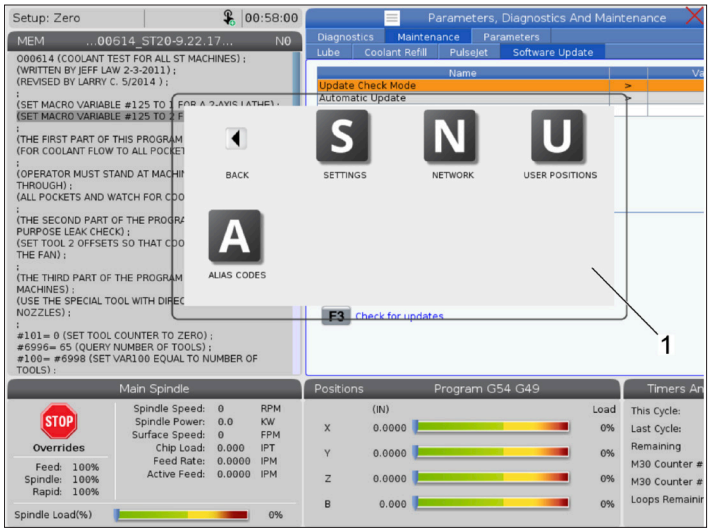


Settings options Icons [1].

Press and hold the display icon, to navigate to a specific tab. For example if you want to go to the Network page, press and hold the settings icon until the settings options [3] are shown.

Press the back icon to go back to the main menu.

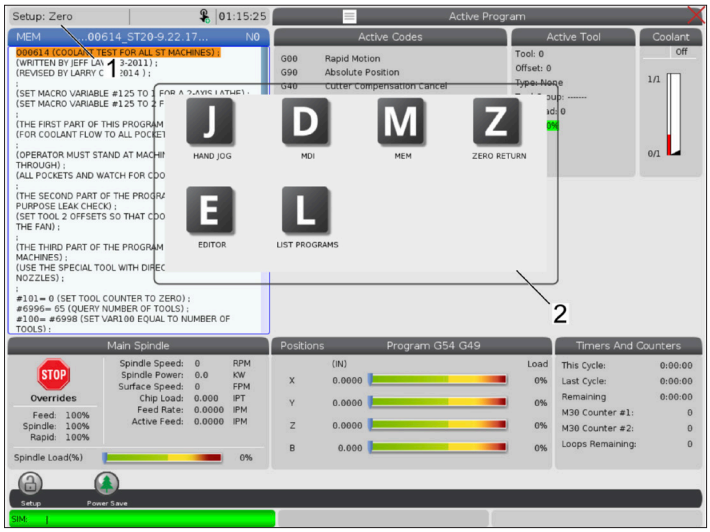
To close the popup box touch anywhere else outside the popup box.



Operation Mode Panel

Press the upper left corner [1] of the screen to have the operation mode panel popup box [2] to appear.

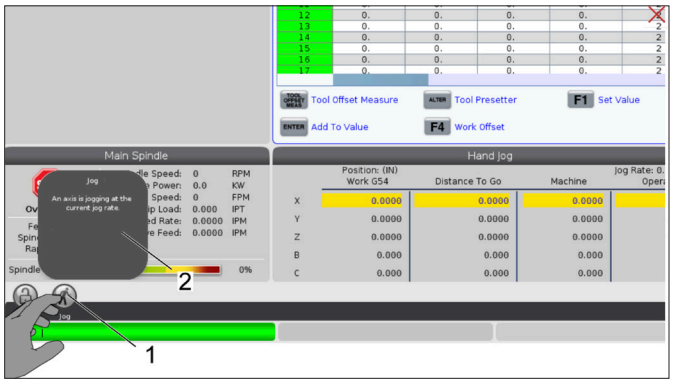
Press the mode icon to put the machine in that mode.



LCD Touchscreen - Selectable Boxes

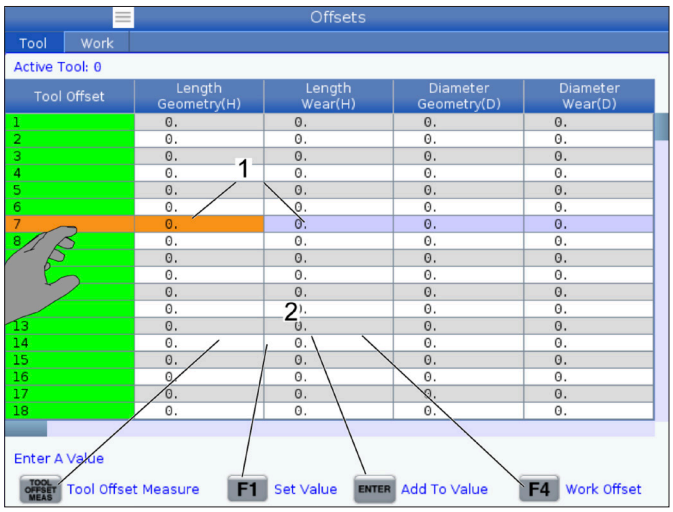
Icon Help

- Touch and hold the icons [1] on the bottom of the screen to see the meaning [2] of the icon.
- The help popup will disappear when you let go of the icon.



Selectable tables and function buttons.

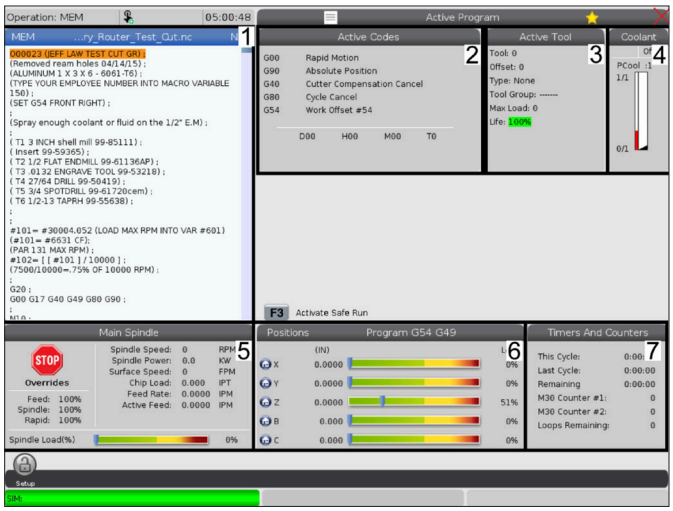
- The rows and columns fields [1] on tables are selectable. To increase the row size refer to setting 383 - Table Row Size.
- The function button icons [2] that appear on the boxes can also be pressed to use the function.



Selectable Display Boxes

- Display boxes [1 - 7] are selectable.

  For example if you want to go to the Maintenance tab, press the coolant display box [4].



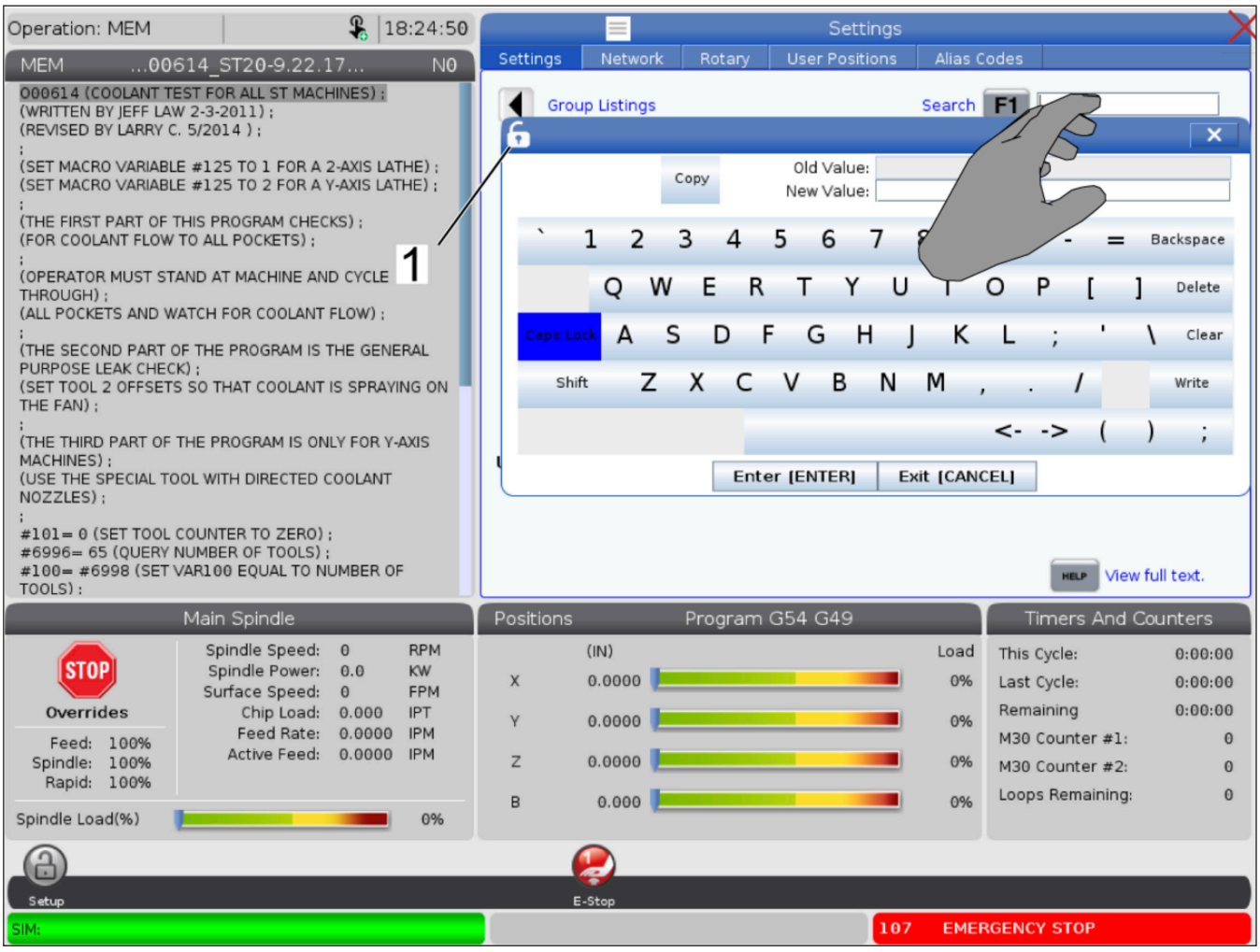
LCD Touchscreen - Virtual Keyboard

The virtual keyboard allows you to input text on the screen, without using the keypad.

To enable this function set setting 396 - Virtual Keyboard Enabled to On. Press and hold any input line for the virtual keyboard to appear.

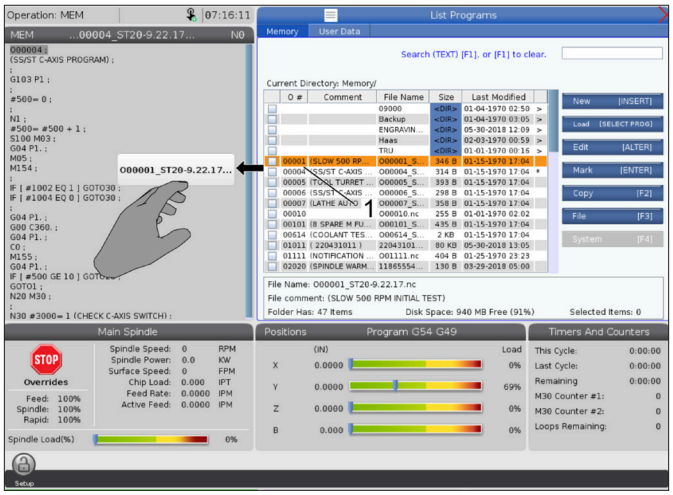
The keyboard can be moved holding your finger down on the blue top bar and dragging it to a new position.

The keyboard can also be locked in place by pressing the lock icon [1].



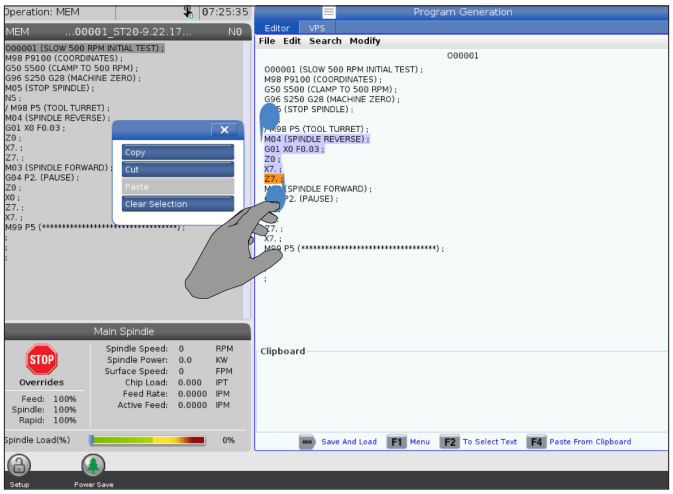
Drag and Drop from List Program

You can drag and drop programs from List Program to MEM by dragging the file [1] over to the MEM display.



Copy, Cut, and Paste Handle Bars

In edit mode you can drag your fingers across the code to use the handle bars to copy, cut and paste a section of the program.

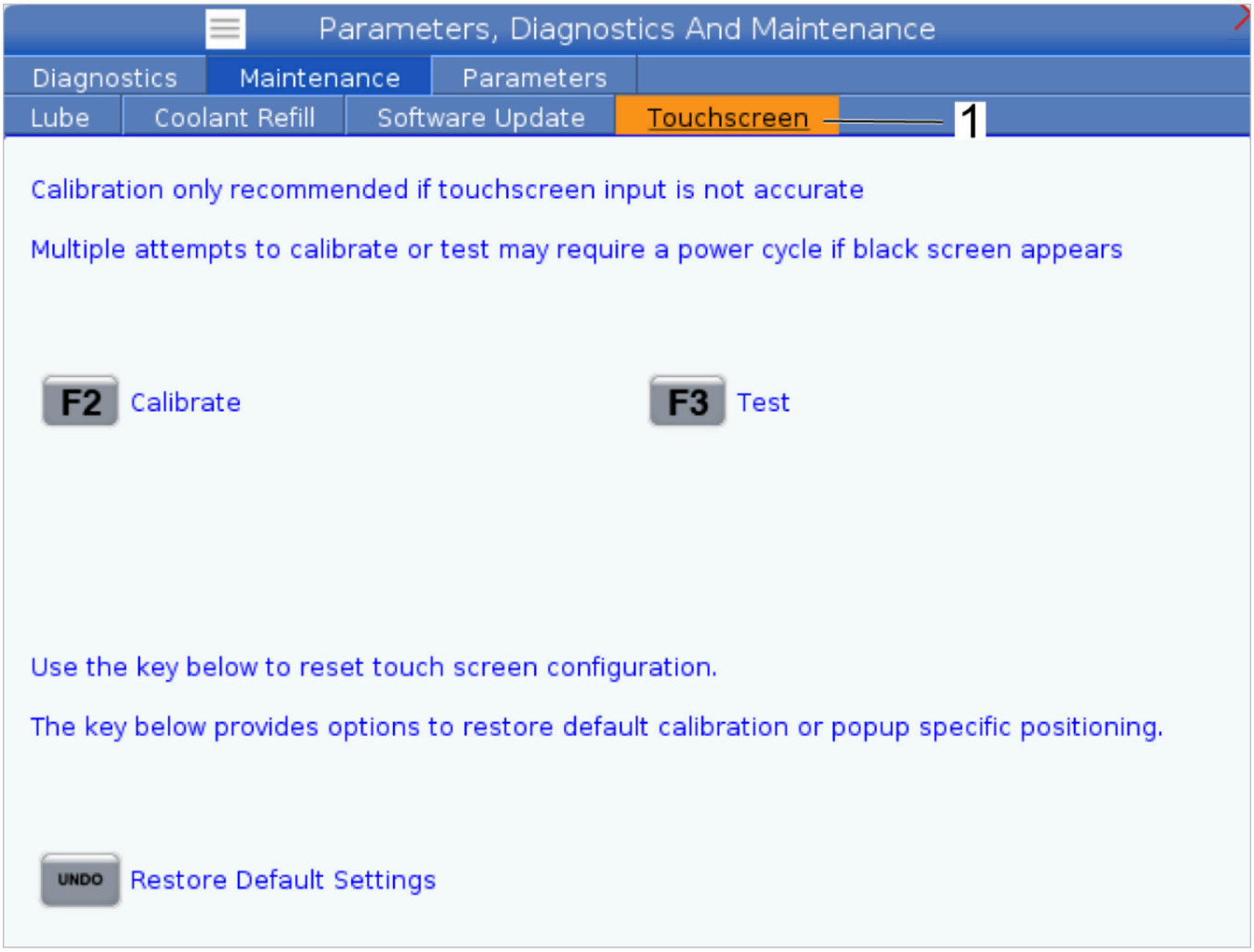


LCD Touchscreen - Maintenance

Touchscreen Configuration Tab

Use the touchscreen configuration page to calibrate, test, and restore default settings. The touchscreen configuration is located in the maintenance section.

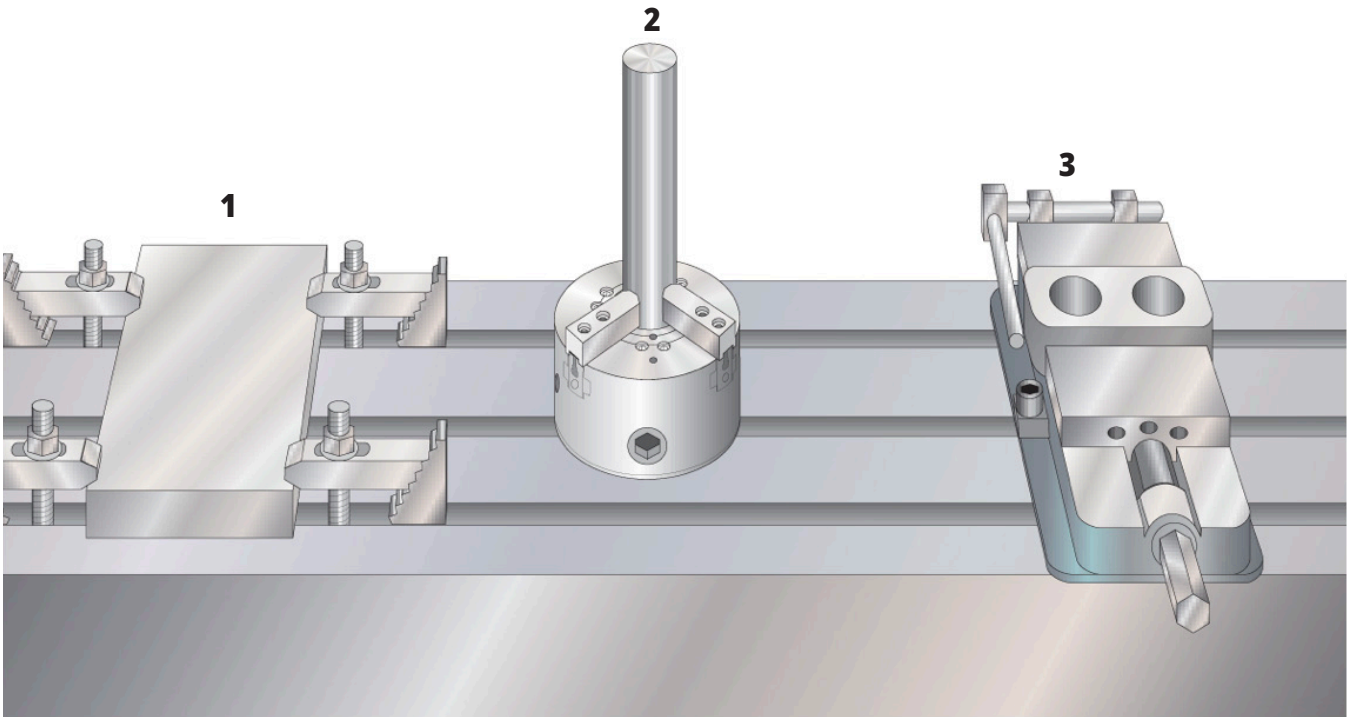
Press Diagnostic to go to the Maintenance and navigate to the Touchscreen tab.



Part Setup

Part Setup Examples:
[1] Toe clamp, [2] Chuck, [3] Vise.

Correct workholding is very important for safety, and to get the machining results that you want. There are many workholding options for different applications. Contact your HFO or workholding dealer for guidance.



Jog Mode

Jog mode lets you jog the machine axes to a desired location. Before you can jog an axis, the machine must establish its home position. The control does this at machine power-up.

To enter jog mode:

1. Press HANDLE JOG.
2. Press the desired axis (+X, -X, +Y,-Y, +Z, -Z, +A/C or -A/C, +B, or -B).
3. There are different increment speeds that can be used while in jog mode; they are .0001, .001, .01 and .1. Each click of the jog handle moves the axis the distance defined by the current jog rate. You can also use an optional Remote Jog Handle (RJH) to jog the axes.
4. Press and hold the handle jog buttons or use the jog handle control to move the axis.

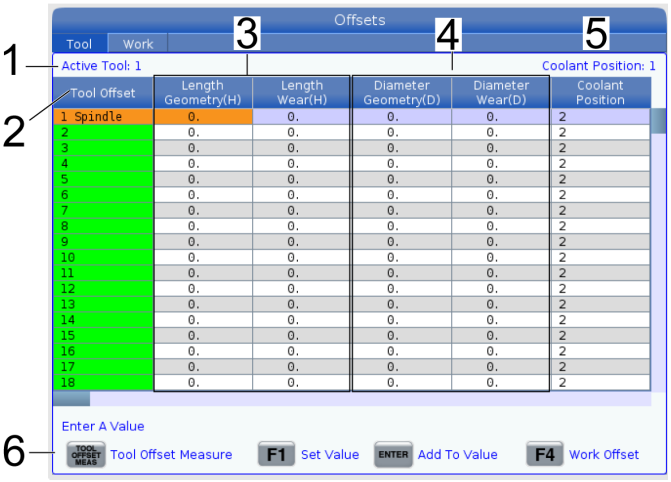
Tool Offsets

To machine a part accurately, the mill needs to know where the part is located on the table and the distance from the tip of the tools to the top of the part (tool offset from home position).

The tool offset behavior has been modified on Haas machines in the following ways:

- By default tool offsets will now always be applied, unless a G49/H00 (Mill) or Txx00 offset (Lathe) is explicitly specified.
- On mills, when a tool change occurs, the tool offset will automatically update to match the new tool. This behavior already existed on lathes.

Press the OFFSET button to view the tool offset values. The tool offsets can be entered manually or automatically with a probe. The list below will shows how each offset setting works.



1) Active Tool: - This tells you which tool is in the spindle.

2) Tool Offset (T) - This is the list of tool offsets. There is a maximum of 200 tool offsets available.

3) Length Geometry (H), Length Wear (H) - These two columns are tied to the G43 (H) values in the program. If you command a G43 H01; from within a program for tool #1, the program will use the values from these columns.

NOTE: The Length Geometry can be set manually or automatically by the probe.

4) Diameter Geometry (D), Diameter Wear (D) - These two columns are used for cutter compensation. If you command a G41 D01;

from within a program, the program will use the values from these columns.

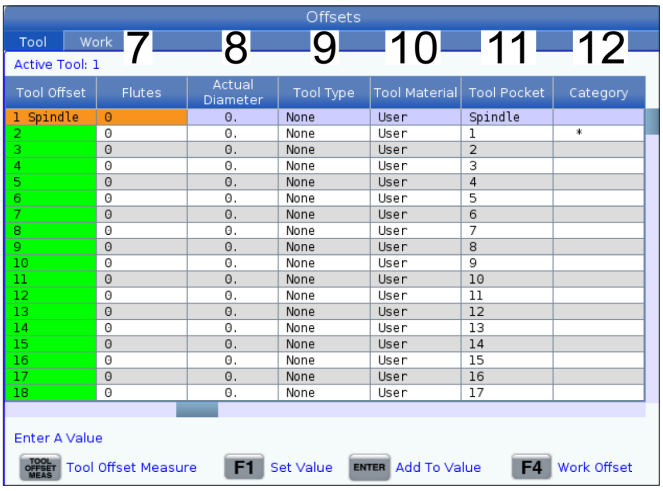
NOTE: The Diameter Geometry can be set manually or automatically by the probe.

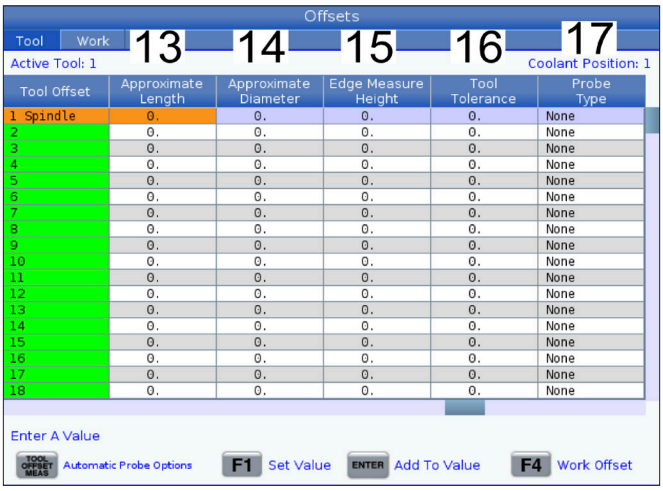
5) Coolant Position - Use this column for setting the coolant position for the tool in this row.

NOTE: This column will only show if you have the Programmable Coolant option.

6) These functions buttons allow you to set the offset values.

8.3 | MILL - TOOL OFFSETS





7) Flutes - When this column is set to the correct value, the control can calculate the correct Chip Load value displayed at the Main Spindle screen. The VPS feeds and speeds library will also use these values for calculations.

NOTE: The values set on the Flute column will not affect the operation of the probe.

8) Actual Diameter - This column is used by the control to calculate the correct Surface Speed value displayed at the Main Spindle screen.

9) Tool Type - This column is used by the control to decide which probe cycle to use to probe this tool. Press F1 to view the options: None, Drill, Tap, Shell Mill, End Mill, Spot Drill, Ball Nose and Probe. When this field is set to Drill, Tap, Spot Drill, Ball Nose and Probe, the probe will probe along the tool centerline for the length. When this field is set to Shell Mill or End Mill the probe will probe at the tools edge.

10) Tool Material - This column is used for calculations by the VPS feeds and speeds library. Press F1 to view the options: User, Carbide, Steel. Press Enter to set the material, or press Cancel to exit.

11) Tool Pocket - This column shows you what pocket the tool is currently in. This column is read only.

12) Tool Category - This column shows if the tool is set up as large, heavy or extra large. To make a change, highlight the column and press ENTER. The Tool Table will be displayed. Follow the instructions on the screen to make tool table changes.

13) Approximate Length - This column is used by the probe. The value in this field tells the probe the distance from the tip of the tool to the spindle gauge line.

NOTE: If you probe the length of a drill or a tap, or some tool that is not a shell mill or an end mill you can leave this field blank.

14) Approximate Diameter - This column is used by the probe. The value in this field tells the probe the diameter of the tool.

15) Edge Measure Height - This column is used by the probe. The value in this field is the distance below the tip of the tool that the tool needs to move, when the tool diameter is probed. Use this setting when you have a tool with a large radius or when you are probing a diameter on a chamfer tool.

16) Tool Tolerance - This column is used by the probe. The value in this field is used for checking tool breakage and wear detection. Leave this field blank if you are setting the length and diameter on the tool.

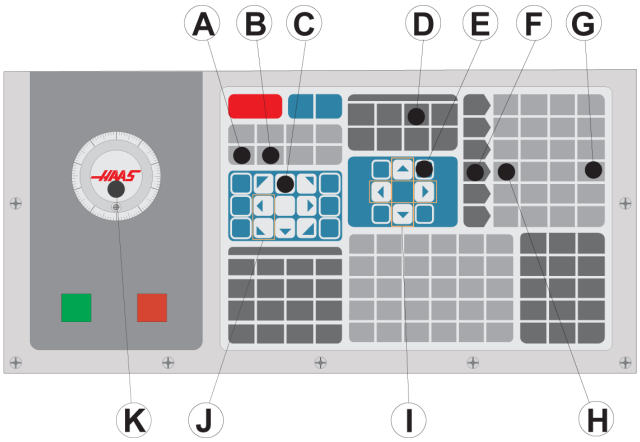
17) Probe Type - This column is used by the probe. You can select the probe routine you want to perform on this tool.

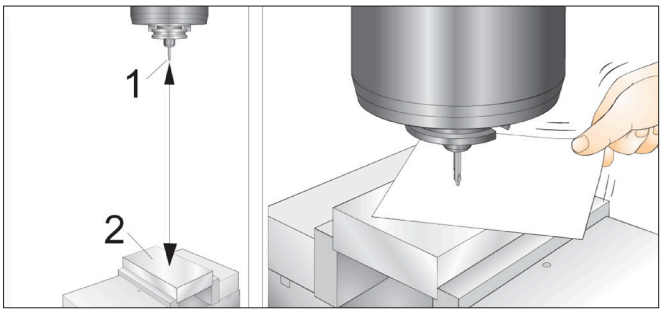
The choices are: 0 - No tool probing to be performed., 1- Length probing (Rotating)., 2 - Length probing (Non-Rotating)., 3 - Length and Diameter probing (Rotating). Press TOOL OFFSET MEASURE to set automatic probe options.

Set a Tool Offset

The next step is to touch off the tools. This defines the distance from the tip of the tool to the top of the part. Another name for this is Tool Length Offset, which is designated as H in a line of machine code. The distance for each tool is entered into the TOOL OFFSET table.

NOTE: When touching off tools or working on the stationary table, make sure the tilt axis is at 0 degrees (A0° or B0°).





1

Setting Tool Offset. With the Z Axis at its home position, Tool Length Offset is measured from the tip of the tool [1] to the top of the part [2].

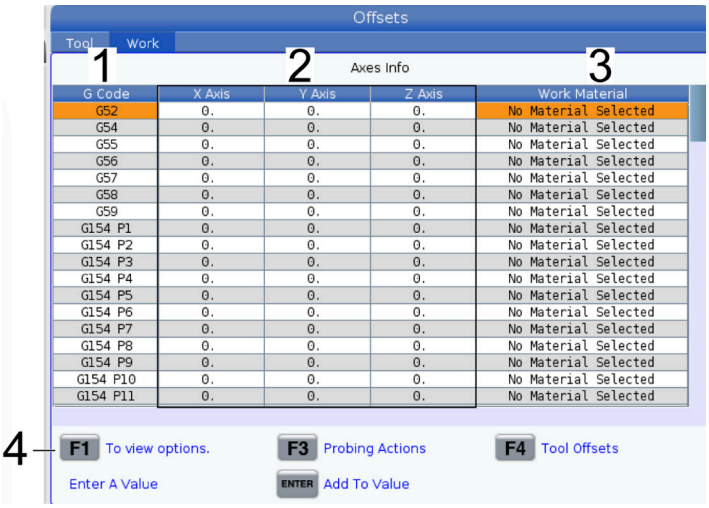
- Load the tool in the spindle [1].
- Press HANDLE JOG [F].
- Press .1/100. [G] (The mill moves at a fast rate when the handle is turned).
- Select between the X and Y axes [J], and with the use the jog handle [K] to move the tool near the center of the part.
- Press +Z [C].
- Jog the Z Axis approximately 1" above the part.
- Press .0001/.1 [H] (The mill moves at a slow rate when the handle is turned).

2

- Place a sheet of paper between the tool and the work piece. Carefully move the tool down to the top of the part, as close as possible, and still be able to move the paper.
- Press OFFSET [D] and select the TOOL tab.
- Highlight the H (length) Geometry value for position #1.
- Press TOOL OFFSET MEASURE [A]. A pop box may appear if the change is Greater Than Setting 142! Accept (Y/N). Press Y to accept.
- **CAUTION:** The next step causes the spindle to move rapidly in the Z Axis.
- Press NEXT TOOL [B].
- Repeat the offset process for each tool.

Work Offsets

Press the OFFSET, then the F4 to view the work offset values. The work offsets can be entered manually or automatically with a probe. The list below will shows how each work offset setting works.



1) G Code - This column displays all the available work offset G-codes. For more information on these work offsets, refer to the G52 Set Work Coordinate System (Group 00 or 12), G54 Work Offsets, G92 Set Work Coordinate Systems Shift Value (Group 00).

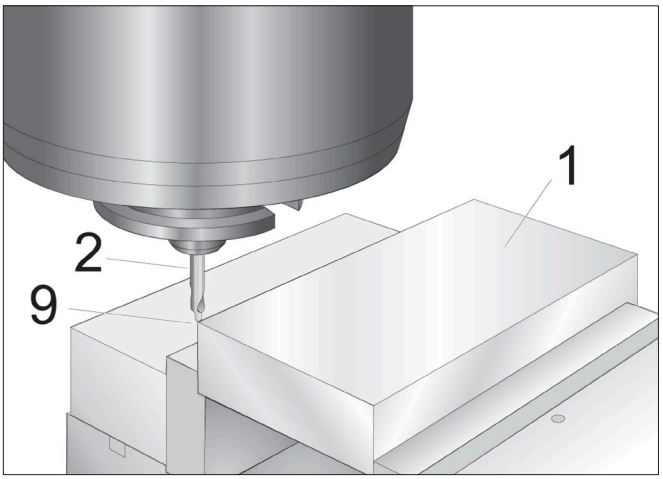
2) X, Y, Z, Axis - This column displays the work offset value for each axis. If rotary axis are enabled the offsets for these will be displayed on this page.

3) Work Material - This column is used by the VPS feeds and speeds library.

4) These functions buttons allow you to set the offset values. Type in the desired work offset value and press F1 to set the value. Press F3 to set a probing action. Press F4 to toggle from work to tool offset tab. Type in a value and press Enter to add to the current value.

Set a Work Offset

To machine a workpiece, the mill needs to know where the workpiece is located on the table. You can use an edge finder, an electronic probe, or many other tools and methods to establish part zero. To set the part zero offset with a mechanical pointer:



1

Place the material [1] in the vise and tighten.

Load a pointer tool [2] in the spindle.

Press HANDLE JOG [E].

Press .1/100. [F] (The mill moves at a fast speed when the handle is turned).

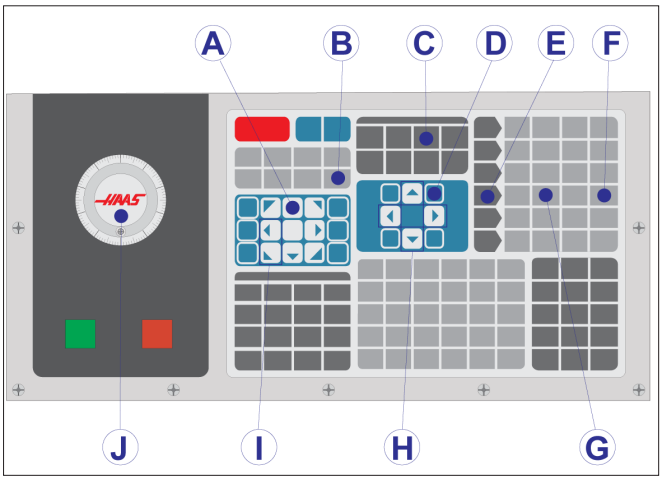
Press +Z [A].

Use the jog handle [J] to move the Z-Axis approximately 1" above the part.

Press .001/1. [G] (The mill moves at a slow speed when the handle is turned).

Jog the Z-Axis approximately. 0.2" above the part.

Select between the X and Y axes [I] and jog the tool to the upper left corner of the part (see illustration [9])



2

Navigate to the OFFSET>WORK [C] tab and press the DOWN cursor key [H] to activate the page. You can press F4 to toggle between Tool offsets and Work offsets.

Navigate to the G54 X Axis location.

CAUTION: In the next step, do not press PART ZERO SET a third time; this loads a value into the Z AXIS column. This causes a crash or a Z-Axis alarm when the program is run.

Press PART ZERO SET [B] to load the value into the X Axis column. The second press of PART ZERO SET [B] loads the value into the Y Axis column.

WIPS - Operators Manual

Please refer to the WIPS operators manual, operation section, for instructions on how to set tool offsets and work offsets using a probing cycle.

Scan QR code below to be redirected to website page.

WIPS Operation

Tool Changers

There are (2) types of mill tool changers: the umbrella style (UTC), and the side-mount tool changer (SMTC). You command both tool changers in the same way, but you set them up differently.

Make sure the machine is zero returned. If it is not, press POWER UP.

Use TOOL RELEASE, ATC FWD, and ATC REV to manually command the tool changer. There are (2) tool release buttons; one on the spindle head cover and another on the keyboard.

Loading the Tool Changer

CAUTION: Do not exceed the maximum tool changer specifications. Extremely heavy tool weights should be distributed evenly. This means heavy tools should be located across from one another, not next to each other. Ensure there is adequate clearance between tools in the tool changer; this distance is 3.6" for a 20-pocket and 3" for a 24+1 pocket. Check your tool changer specifications for the correct minimal clearance between tools.

NOTE: Low air pressure or insufficient volume reduces the pressure applied to the tool release piston and will slow down tool change time or will not release the tool.

WARNING: Stay away from the tool changer during power up, power down, and during tool changer operations.

Always load tools into the tool changer from the spindle. Never load a tool directly into the tool changer carousel. Some mills have remote tool changer controls to let you inspect and replace tools at the carousel. This station is not for initial loading and tool assignment.

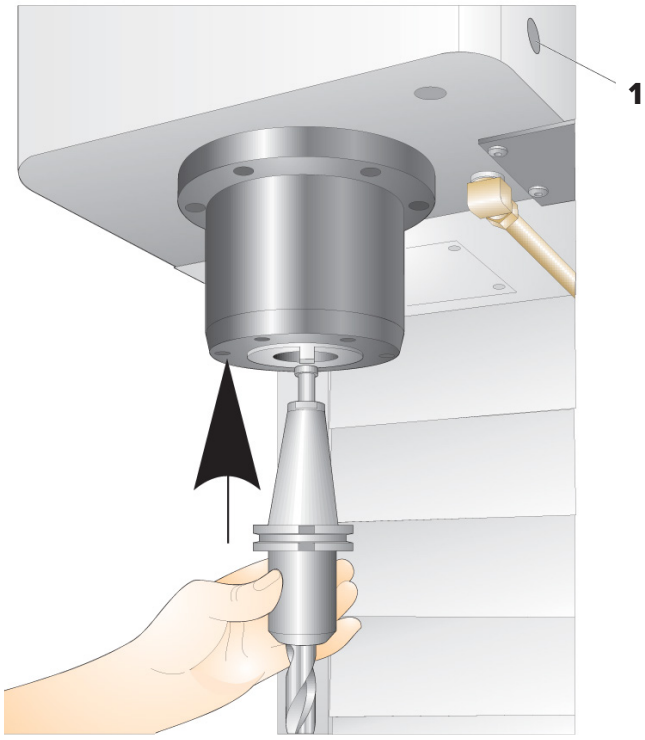
CAUTION: Tools that make a loud noise when released indicate a problem and should be checked before serious damage occurs to the tool changer or spindle.

Tool Loading for a Umbrella Tool Changer

1

This section tells you how to load tools into an empty tool changer for a new application. It assumes that the pocket tool table still contains information from the previous application.

Make sure your tool holders have the correct pull stud type for the mill.



2

Tools are loaded into the umbrella tool changer by first loading the tool into the spindle. To load a tool into the spindle, prepare the tool and then follow these steps:

Ensure the tools loaded have the correct pull stud type for the mill.

Press MDI/DNC for MDI mode.

Organize the tools to match to the CNC program.

Take tool in hand and insert the tool (pull stud first) into the spindle. Turn the tool so that the two cutouts in the tool holder line up with the tabs of the spindle. Push the tool upward while pressing the Tool Release button. When the tool is fitted into the spindle, release the Tool Release button.

Press ATC FWD.

Repeat Steps 4 and 5 with the remaining tools until all the tools are loaded.

Umbrella Tool Changer Recovery

If the tool changer jams, the control will automatically come to an alarm state. To correct this:

WARNING: Never put your hands near the tool changer unless there is an alarm displayed first.

1. Remove the cause of the jam.
2. Press RESET to clear the alarms.
3. Press RECOVER and follow the directions to reset the tool changer.

Tool Changers

There are (2) types of mill tool changers: the umbrella style (UTC), and the side-mount tool changer (SMTC). You command both tool changers in the same way, but you set them up differently.

Make sure the machine is zero returned. If it is not, press POWER UP.

Use TOOL RELEASE, ATC FWD, and ATC REV to manually command the tool changer. There are (2) tool release buttons; one on the spindle head cover and another on the keyboard.

Loading the Tool Changer

CAUTION: Do not exceed the maximum tool changer specifications. Extremely heavy tool weights should be distributed evenly. This means heavy tools should be located across from one another, not next to each other. Ensure there is adequate clearance between tools in the tool changer; this distance is 3.6" for a 20-pocket and 3" for a 24+1 pocket. Check your tool changer specifications for the correct minimal clearance between tools.

NOTE: Low air pressure or insufficient volume reduces the pressure applied to the tool release piston and will slow down tool change time or will not release the tool.

WARNING: Stay away from the tool changer during power up, power down, and during tool changer operations.

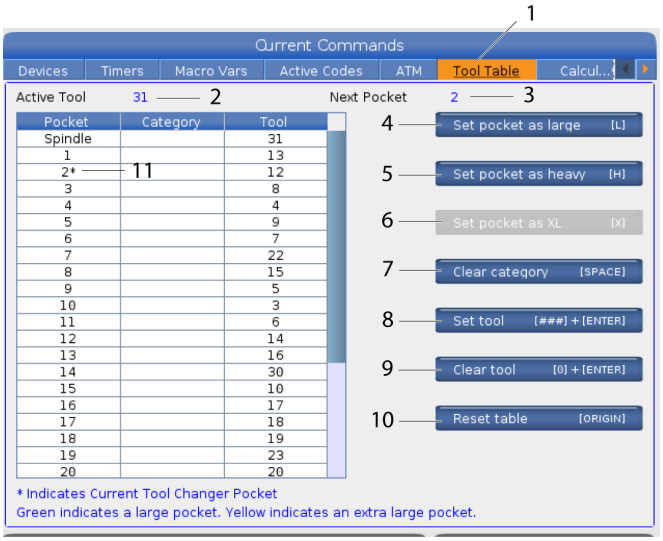
Always load tools into the tool changer from the spindle. Never load a tool directly into the tool changer carousel. Some mills have remote tool changer controls to let you inspect and replace tools at the carousel. This station is not for initial loading and tool assignment.

CAUTION: Tools that make a loud noise when released indicate a problem and should be checked before serious damage occurs to the tool changer or spindle.

Tool Table

This section tells you how to use the tool table to give the control information about your tools.

NOTE: If your machine has an umbrella-style tool changer, you will not use the tool pocket table.



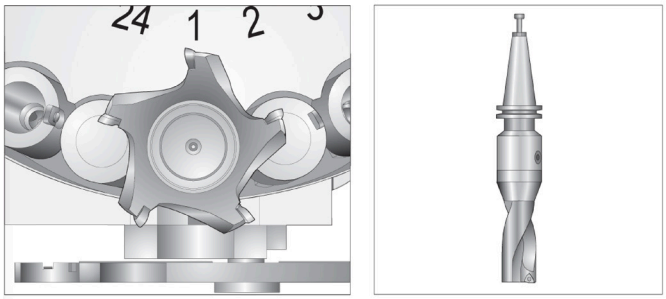
1) To access the Tool Pocket Table, press CURRENT COMMANDS and choose the Tool Table tab.

2) Active Tool - Tells you the tool number that is installed in the spindle.

3) Active Pocket - this show you the next pocket number.

4) Set pocket as Large [L] - Use this flag when a large tool has a diameter of greater than 3" for 40-taper machines and greater than 4" for 50-taper machines. Scroll to the pocket of interest and press L to set the flag.

CAUTION: You cannot place a large tool in the tool changer if one or both of the surrounding pockets already contain tools. Doing so causes the tool changer to crash. Large tools must have the surrounding pockets empty. However, large tools can share adjoining empty pockets.



A Large (left), and a Heavy (not Large) Tool (above)

5) Set pocket as heavy [H] - Use this flag when a Heavy, Small Diameter 40-taper tool (4 lb or heavier) or a 50-taper tool (12 lb or heavier) is loaded into the spindle. Scroll to the pocket of interest and press H to set the flag.

6) Set pocket as XL [X] - Use this flag when two adjacent pockets at each side of the tool are needed. Scroll to the pocket of interest and press X to set the flag.

NOTE: This option only appears if your machine is a 50-taper.

7) Clear category [Space] - Highlight the desired tool and press SPACE to clear the flag.

8) Set tool [###] + [Enter]- Highlight the desired pocket and type in the tool number + Enter to set the desired tool number.

NOTE: You cannot assign a tool number to more than one pocket. If you enter a tool number that is already defined in the tool pocket table, you see an Invalid tool error.

9) Clear tool [0] + [Enter]- Highlight the desired pocket and press 0 + Enter to clear out the tool number.

10) Reset table [Origin] - Press ORIGIN with the cursor in the center column to use the ORIGIN menu. This menu lets you:

Sequence All Pockets - Makes all of the tool numbers sequential based on their pocket location, starting with 1.

Zero All Pockets - Removes all of the tool numbers from all of the pocket numbers.

Clear Category Flags - Removes the category designations from all of the tools.

11) * Indicates current tool changer pocket.

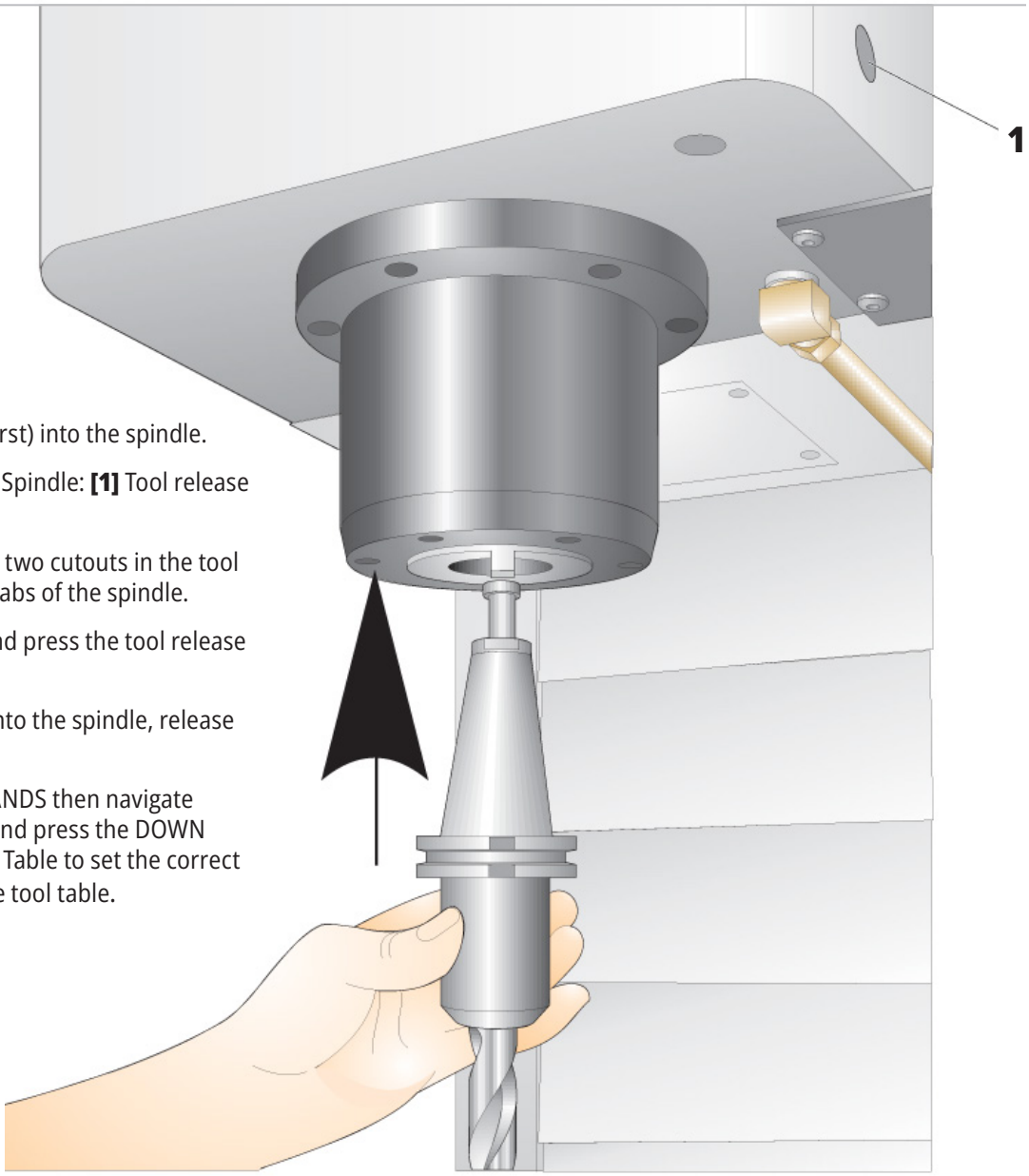
Tool Loading for a Side-Mount Tool Changer

1

This section tells you how to load tools into an empty tool changer for a new application. It assumes that the pocket tool table still contains information from the previous application.

Make sure your tool holders have the correct pull stud type for the mill.

Press CURRENT COMMANDS then navigate to the TOOL TABLE tab and press the DOWN cursor. Refer to the Tool Table to set the correct tool information into the tool table.



2

Insert tool 1 (pull stud first) into the spindle.

Inserting a Tool Into the Spindle: **[1]** Tool release button.

Turn the tool so that the two cutouts in the tool holder line up with the tabs of the spindle.

Push the tool upward and press the tool release button.

When the tool is fitted into the spindle, release the tool release button.

Press CURRENT COMMANDS then navigate to the TOOL TABLE tab and press the DOWN cursor. Refer to the Tool Table to set the correct tool information into the tool table.

High-Speed Side-Mount Tool Changer

The high-speed side-mount tool changer has an additional tool assignment, which is “Heavy”. Tools that weigh more than 4 pounds are considered heavy. You must designate heavy tools with H (Note: All large tools are considered heavy). During operation, an “h” in the tool table denotes a heavy tool in a large pocket.

As a safety precaution, the tool changer will run at a maximum of 25% of the normal speed when it changes a heavy tool. The pocket up/down speed is not slowed down. The control restores the speed to the current rapid when the tool change is complete. Contact your HFO for assistance if you have problems with unusual or extreme tooling.

H - Heavy, but not necessarily large (large tools require empty pockets on either side).

h - Heavy small diameter tool in a pocket designated for a large tool (must have empty pocket on both sides). The lower case “h” and “l” is placed by the control; never enter a lower case “h” or “l” into the tool table.

l - Small diameter tool in a pocket reserved for a large tool in the spindle.

Large tools are assumed to be heavy.

Heavy tools are not assumed to be large.

On non-high speed tool changers, “H” and “h” have no effect.

Using ‘0’ for a Tool Designation

In the tool table, enter 0 (zero) for the tool number to label a tool pocket “always empty”. The tool changer does not “see” this pocket, and it never tries to install or retrieve a tool from pockets with a ‘0’ designation.

You cannot use a zero to designate the tool in the spindle. The spindle must always have a tool number designation.

Moving Tools in the Carousel

If you need to move tools in the carousel, follow this procedure.

CAUTION: Plan the reorganization of the tools in the carousel ahead of time. To reduce the potential for tool changer crashes, keep tool movement to a minimum. If there are large or heavy tools currently in the tool changer, ensure that they are only moved between tool pockets designated as such.

Moving Tools in the Carousel

If you need to move tools in the carousel, follow this procedure.

CAUTION: Plan the reorganization of the tools in the carousel ahead of time. To reduce the potential for tool changer crashes, keep tool movement to a minimum. If there are large or heavy tools currently in the tool changer, ensure that they are only moved between tool pockets designated as such.

Moving Tools

The tool changer pictured has an assortment of normal-sized tools. For the purposes of this example, we need to move tool 12 to pocket 18 to make room for a large-sized tool in pocket 12.

Making Room for Large Tools: [1] Tool 12 to Pocket 18, [2] Large Tool in Pocket 12.

1) Select MDI mode. Press CURRENT COMMANDS and navigate to the TOOL TABLE display. Identify the tool number that is in pocket 12.

2) Type Tnn (where nn is the tool number from step 1). Press ATC FWD. This places the tool from pocket 12 into the spindle.

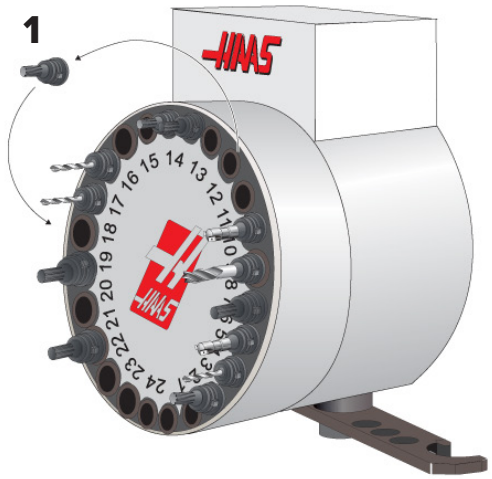
3) Type P18 then press ATC FWD to put the tool in the spindle into pocket 18.

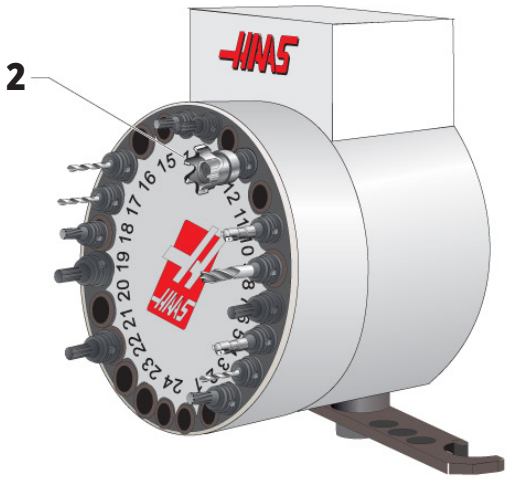
4) Scroll to pocket 12 in the TOOL TABLE and press L then ENTER to designate pocket 12 as large.

5) Enter the tool number into SPINDLE on the TOOL TABLE. Insert the tool into the spindle.

NOTE: Extra-large tools can be programmed as well. An "extra-large" tool is one that takes up three pockets; the diameter of the tool covers the tool pocket on either side of the pocket it is installed in. Contact your HFO to provide a special configuration if a tool this size is needed. The tool table must be updated since two empty pockets are needed between extra large tools.

6) Enter P12 into the control and press ATC FWD. The tool is placed into pocket 12.





SMTC Tool Pre-Call

Tool Pre-Call

To save time, the control looks ahead as far as 80 lines into your program to process and prepare machine motion and tool changes. When look-ahead finds a tool change, the control puts the next tool in your program into position. This is called “tool pre-call.”

Some program commands stop look-ahead. If your program has these commands before the next tool change, the control does not pre-call the next tool. This can cause your program to run slower, because the machine must wait for the next tool to move into position before it can change tools.

Program commands that stop look-ahead:

- Work offset selections (G54, G55, etc.)
- G103 Limit Block Buffering, when programmed without a P address or with a nonzero P address
- M01 Optional Stop
- M00 Stop Program
- Block Delete Slashes (/)
- A large number of program blocks executed at high speed

To make sure that the control pre-calls the next tool without look-ahead, you can command the carousel to the next tool position immediately after a tool change command, as in this code snippet:

T01 M06 (TOOL CHANGE) ;

T02 (PRE-CALL THE NEXT TOOL) ;

SMTC Door Switch Panel

Mills such as the MDC, EC-300 and EC-400 have a sub-panel to aid tool loading. The Manual/Automatic Tool Change switch must be set to “Automatic Operation” for automatic tool changer operation. If the switch is set to “Manual”, the two buttons, labeled with clockwise and counterclockwise symbols, are enabled and automatic tool changes are disabled. The door has a sensor switch which detects when the door is open.

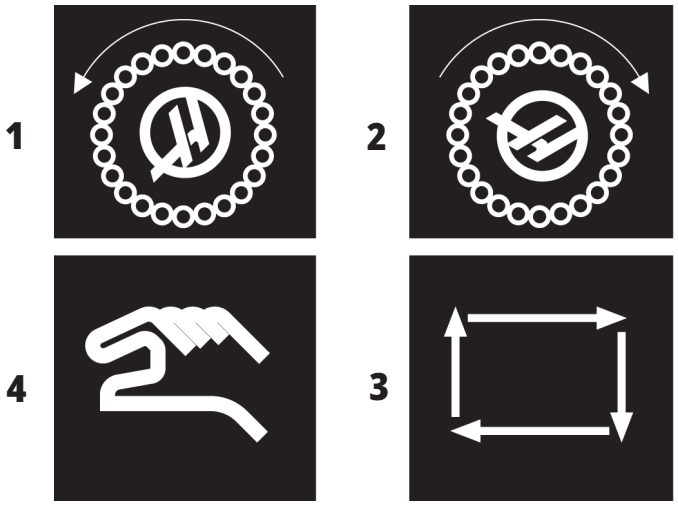
Tool Changer Door Switch Panel Symbols:

[1] Rotate Tool changer Carousel Counter-Clockwise,

[2] Rotate Tool Changer Carousel Clockwise,

[3] Tool Change Switch - Automatic Operation,

[4] Tool Change Switch - Manual Operation Selection.



SMTC Door Operation

If the cage door is opened while a tool change is in progress, the tool change stops and resumes when the cage door is closed. Any machining operations in progress remain uninterrupted.

If the switch is turned to manual while a tool carousel is in motion, the tool carousel stops and resumes when the switch is turned back to automatic. The next tool change will not execute until the switch is turned back. Any machining operations that are in progress remain uninterrupted.

The carousel rotates one position whenever a clockwise or counter-clockwise button is pressed once, while the switch is set to manual.

During tool changer recovery, if the cage door is open or the Tool Change switch is in the manual position and RECOVER is pressed, a message is displayed telling the operator the door is open or is in manual mode. The operator must close the door and set the switch to the automatic position in order to continue.

SMTC Recovery

If a problem occurred during a tool change, a tool changer recovery needs to be performed. Enter the tool changer recovery mode by:

Press RECOVER and navigate to the TOOL CHANGER RECOVERY tab.

Press ENTER. If there is no Alarm, the control first attempts an automatic recovery. If there is an alarm press RESET to clear the alarms and repeat from step 1.

At the VMSTC TOOL RECOVERY screen, press A to begin automatic recovery or E to exit.

If the automatic recovery fails, press M to continue for a manual recovery.

In manual mode, follow the instructions and answer the questions to perform a proper tool changer recovery.

The entire tool changer recovery process must be completed before exiting. Start the routine from the beginning if you exit the routine early.

Machine Power-On

This section tells you how to power-on a new machine for the first time.

- Press POWER ON until you see the Haas logo on the screen. After a self-test and boot sequence, the display shows the startup screen.

  The startup screen gives basic instructions to start the machine. Press CANCEL to dismiss the screen.
- Turn EMERGENCY STOP to the right to reset it.
- Press RESET to clear the startup alarms. If you cannot clear an alarm, the machine may need service. Contact your Haas Factory Outlet (HFO) for assistance.
- If your machine is enclosed, close the doors.
  WARNING: Before you do the next step, remember that automatic motion begins immediately when you press POWER UP. Make sure the motion path is clear. Stay away from the spindle, machine table, and tool changer.
- Press POWER UP.
  After the first POWER UP, the axes move toward their home positions. The axes then move slowly until the machine finds the home switch for each axis. This establishes the machine home position.

Press any of the following:

- CANCEL to dismiss the screen.
- CYCLE START to run current program.
- HANDLE JOG for manual operation.

Spindle Warm-Up

If your machine's spindle has been idle for more than (4) days, run the spindle warm-up program before you use the machine. This program brings the spindle up to speed slowly to distribute the lubrication and let the spindle reach a stable temperature.

Your machine includes a 20-minute warm-up program (O09220) in the program list. If you use the spindle at consistent high speeds, you should run this program every day.

Screen Capture

The control can capture and save an image of the current screen to an attached USB device or the User Data memory.

Enter a filename if desired. If no filename is entered, the system will use the default filename (see note).

Press SHIFT.

Press F1.

NOTE: The control uses the default filename snapshot#.png. The # starts from 0 and increments each time you capture a screen. This counter resets at power off. Screen captures that you take after a power cycle overwrite previous screen captures that have the same filename on the User Data memory.

Result:

The control saves the screen capture to your USB device or control memory. The message Snapshot saved to USB or Snapshot saved to User Data appears when the process finishes.

Error Report

The control can generate an error report that saves the state of the machine that is used for analysis. This is useful when helping the HFO troubleshoot an intermittent problem.

1. Press SHIFT.
2. Press F3.

NOTE: Be sure to always generate the error report with the alarm or the error is active.

Result:

The control saves the error report to your USB device or control memory. The error report is a zip file that includes a screen capture, the active program, and other information used for diagnostics. Generate this error report when an error or an alarm occurs. E-mail the error report to your local Haas Factory Outlet.

Basic Program Search

You can use this function to quickly find code in a program.

NOTE: This is a quick-search function that finds the first match in the search direction that you specify. You can use the Editor for a more full-featured search. Refer to Chapter 6.5 for more information on the Editor search function.

NOTE: This is a quick-search function that finds the first match in the search direction that you specify. You can use the Editor for a more full-featured search. Refer to The Search Menu for more information on the Editor search function.

Type the text you want to find in the active program.

Press the UP or DOWN cursor arrow key.

Result:

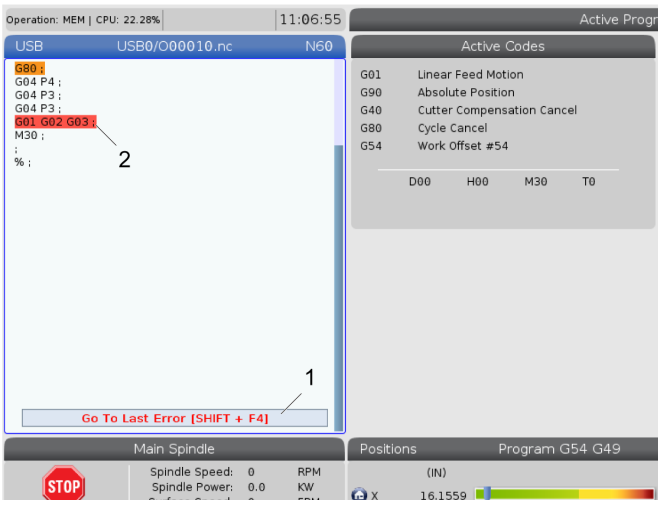
The UP cursor arrow key searches from the cursor position to the start of the program. The DOWN cursor arrow key searches to the end of the program. The control highlights the first match.

NOTE: Putting your search term within parentheses (), will search only within comment lines.

Locate the Last Program Error

Starting in software version **100.19.000.1100** the control can find the last error in a program.

Press **SHIFT + F4** to display the last line of G-code that generated the error.



Safe Run Mode

The purpose of Safe Run is to reduce damage to the machine in the event of a crash. It does not prevent crashes, but it raises an alarm sooner and backs off from the crash location.

NOTE: The Safe Run feature is available starting in software version 100.19.000.1300.

Safe Run Supported Machines

- VF-1 through VF-5
- VM-2/3
- UMC-500/750/1000
- All DM's
- All DT's
- All TM's
- ST-10 through ST-35

Common causes for crashes are:

Incorrect tool offsets.

Incorrect work offsets.

Wrong tool in the spindle.

NOTE: The Safe Run feature will only detect a crash in handle jog and rapid (G00), it will not detect a crash in a feed move.

Safe Run does the following:

- Slow down the speed of the motion.
- Increases the position error sensitivity.
- When a crash is detected, the control will immediately reverse the axis by a small amount. This will prevent the motor from continuing to drive into the object it has crashed into as well as relieve pressure from the crash itself. After Safe Run has detected a crash, you should be able to easily fit a piece off paper between the two surfaces that crashed.

NOTE: Safe Run is intended for running a program for the first time after writing or changing it. Is it not recommended to run a reliable program with Safe Run, as it increases cycle time significantly. The tool may break and the work piece may still be damaged in a crash.

11.4 | MILL OPERATION - SAFE RUN MODE

Safe Run is active during jogging as well. Safe Run can be used during job setup to protect against accidental crashes due to operator error.

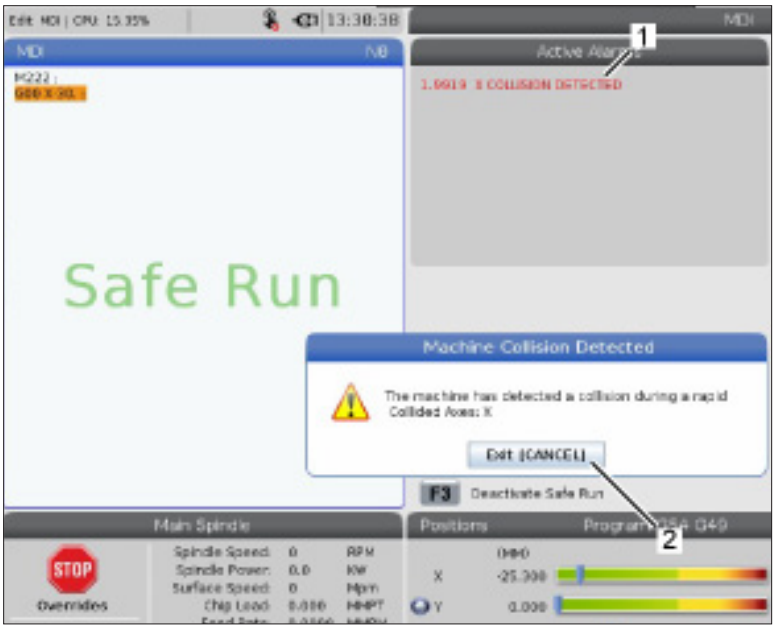
If your machine supports Safe Run, you will see a new icon in MDI with the text F3 Activate Safe Run [1]. Press F3 to turn Safe Run on/off. Safe Run Active state is noted by a water mark [2] in the program panel.

It is only active during rapid motions. Rapid motions include G00, Home G28, moving to tool changes, and the non-machining motions of canned cycles. Any machining motion such as a feed or tap will not have safe mode active.

Safe Run is not active during feeds due to the nature of crash detection. Cutting forces cannot be discerned from crashes.

When a crash is detected, all motion is brought to a stop, an alarm [1] is generated, and a popup [2] is generated letting the operator know that a crash was detected, and which axis it was detected on. This alarm can be cleared by reset.

In certain cases the pressure against the part may not have been relieved by the Safe Run back-off. In the worse case, an additional crash may be generated after you have reset the alarm. If this happens, turn Safe Run off and jog the axis away from the crash location.



Run-Stop-Jog-Continue

This feature lets you stop a running program, jog away from the part, and then start the program again.

1. Press FEED HOLD.
   Axis motion stops. The spindle continues to turn.
2. Press X, Y, Z, or an installed Rotary Axis (A for A Axis, B for B Axis, and C for C Axis), then press HANDLE JOG. The control stores the current X, Y, Z, and rotary axes positions.
3. The control gives the message Jog Away and displays the Jog Away icon. Use the jog handle or jog keys to move the tool away from the part. You can start or stop the spindle with FWD, REV, or STOP. You can command optional Through Spindle Coolant on and off with the AUX CLNT key (you must stop the spindle first). Command optional Through Tool Air Blast on and off with SHIFT + AUX CLNT keys. Command Coolant on and off with the COOLANT key. Command the Auto Air Gun / Minimum Quantity Lubrication options with SHIFT + COOLANT keys. You can also release the tool to change inserts.

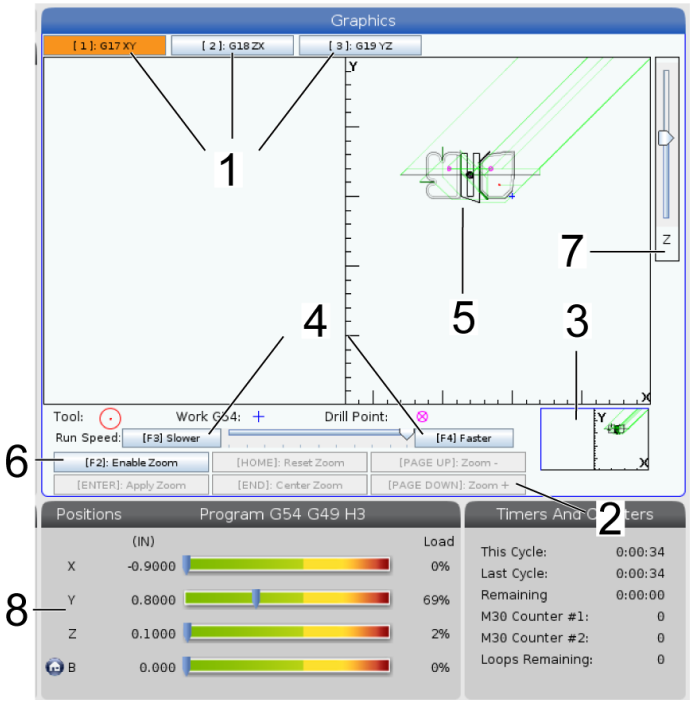
   CAUTION: When you start the program again, the control uses the previous offsets for the return position. Therefore, it is unsafe and not recommended to change tools and offsets when you interrupt a program.
4. Jog to a position as close as possible to the stored position, or to a position where there is an unobstructed rapid path back to the stored position.
5. Press MEMORY or MDI to return to run mode. The control gives the message Jog Return and displays the Jog Return icon. The control continues only if you return to the mode that was in effect when you stopped the program.
6. Press CYCLE START. The control rapids X, Y, and rotary axes at 5% to the position where you pressed FEED HOLD. It then returns the Z Axis. If you press FEED HOLD during this motion, axis motion pauses and the control gives the message Jog Return Hold. Press CYCLE START to resume the Jog Return motion. The control goes into a feed hold state again when the motion is finished.

   CAUTION: The control does not follow the same path that you used to jog away.
7. Press CYCLE START again and the program resumes operation.

   CAUTION: If Setting 36 is ON, the control scans the program to make sure the machine is in the correct state (tools, offsets, G- and M-codes, etc.) to safely continue the program. If Setting 36 is OFF, the control does not scan the program. This can save time, but it could cause a crash in an unproven program.

Graphics Mode

A safe way to troubleshoot a program is to press GRAPHICS to run it in graphics mode. No movement occurs on the machine, instead the movement is illustrated on the screen.



1) Axis Planes Press 1 to view the graphics in G17 plane, press 2 forG18 plane or press 3 to view in G19 plane.

2) Key Help Area The lower-left part of the graphics display pane is the function key help area. This area shows you the function keys that you can use, and a description of what they do.

3) Locator Window The lower-right part of the pane displays the simulated machine table area, and it shows where the simulated view is zoomed and focused.

4) Graphics Speed Press f3 or f4 to run the desired graphics speed.

5) Tool Path Window The large window in the center of the display gives a simulated view of the work area. It displays a cutting tool icon and simulated tool paths.

NOTE: Feed motion appears as a black line. Rapid moves appear as a green line. Drill cycle locations appear with an X.

NOTE: If Setting 253 is ON, the tool diameter is shown as a thin line. If it is OFF, the tool diameter specified in the Tool Offsets Diameter Geometry table is used.

6) Zoom Press F2 to display a rectangle (zoom window) that shows the area that the zoom operation will move to. Use PAGE DOWN to decrease the size of the zoom window (zoom in), and use PAGE UP to increase the size of the zoom window (zoom out). Use the cursor arrow keys to move the zoom window to the location you want to zoom, and press ENTER to complete the zoom. The control scales the tool path window to the zoom window. Run the program again to display the tool path. Press F2 and then HOME to expand the Tool Path window to cover the entire work area.

7) Z-Axis Part Zero Line The horizontal line on the Z-Axis bar at the top-right corner of the graphics screen gives the position of the current Z-axis work offset plus the length of the current tool. While a program simulation runs, the shaded portion of the bar indicates the depth of the simulated Z-Axis motion relative to the Z-Axis work zero position.

8) Position Pane The position pane displays axis locations just as it would during a live part run.

Basic Programming

A typical CNC program has (3) parts:

1) Preparation: This portion of the program selects the work and tool offsets, selects the cutting tool, turns on the coolant, sets spindle speed, and selects absolute or incremental positioning for axis motion.

2) Cutting: This portion of the program defines the tool path and feed rate for the cutting operation.

3) Completion: This portion of the program moves the spindle out of the way, turns off the spindle, turns off the coolant, and moves the table to a position from where the part can be unloaded and inspected.

This is a basic program that makes a 0.100" (2.54 mm) deep cut with Tool 1 in a piece of material along a straight line path from X = 0.0, Y = 0.0 to X = - 4.0, Y = - 4.0.

NOTE: A program block can contain more than one G-code, as long as those G-codes are from different groups. You cannot place two G-codes from the same group in a program block. Also note that only one M-code per block is allowed.

```
%
O40001 (Basic program) ;
(G54 X0 Y0 is top right corner of part) ;
(Z0 is on top of the part) ;
(T1 is a 1/2" end mill) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G17 G40 G49 G54 (Safe startup) ;
X0 Y0 (Rapid to 1st position) ;
S1000 M03 (Spindle on CW) ;
G43 H01 Z0.1 (Tool offset 1 on) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G01 F20. Z-0.1 (Feed to cutting depth) ;
X-4. Y-4. (linear motion) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%
```

Preparation

These are the preparation code blocks in the sample program O40001:

PREPARATION CODE BLOCK	DESCRIPTION
%	Denotes the beginning of a program written in a text editor.
O40001 (Basic program) ;	O40001 is the name of the program. Program naming convention follows the Onnnnn format: The letter “O”, or “o” is followed by a 5-digit number.
(G54 X0 Y0 is top right corner of part) ;	Comment
(Z0 is on top of the part) ;	Comment
(T1 is a 1/2" end mill) ;	Comment
(BEGIN PREPARATION BLOCKS) ;	Comment
T1 M06 (Select tool 1) ;	Selects tool T1 to be used. M06 commands the tool changer to load Tool 1 (T1) into the spindle.
G00 G90 G17 G40 G49 G54 (Safe startup) ;	This is referred to as a safe startup line. It is good machining practice to place this block of code after every tool change. G00 defines axis movement following it to be completed in Rapid Motion mode. G90 defines axis movements that will be completed in absolute mode (refer to page Absolute vs. Incremental Positioning (G90, G91) for more information). G17 defines the cutting plane as the XY plane. G40 cancels Cutter Compensation. G49 cancels tool length compensation. G54 defines the coordinate system to be centered on the Work Offset stored in G54 on the Offset display.

Preparation (Contin.)

PREPARATION CODE BLOCK	DESCRIPTION
X0 Y0 (Rapid to 1st position) ;	X0 Y0 commands the table to move to the position X = 0.0 and Y = 0.0 in the G54 coordinate system.
S1000 M03 (Spindle on CW) ;	M03 turns the spindle on in a clockwise direction. It takes the address code Snnnn, where nnnn is the desired spindle RPM. On machines with a gearbox, the control automatically selects high gear or low gear, based on the commanded spindle speed. You can use an M41 or M42 to override this. Refer to page M41 Low Gear Override / M42 High Gear Override for more information on these M-codes. On machines with a gearbox, the control automatically selects high gear or low gear, based on the commanded spindle speed. You can use an M41 or M42 to override this. Refer to M41 / M42 Low / High Gear Override for more information on these M-codes.
G43 H01 Z0.1 (Tool offset 1 on) ;	G43 H01 turns on Tool Length Compensation +. The H01 specifies to use the length stored for Tool 1 in the Tool Offset display. Z0.1 commands the Z Axis to Z=0.1.
M08 (Coolant on) ;	M08commands the coolant to turn on.

Cutting

These are the preparation code blocks in the sample program O40001:

CUTTING CODE BLOCK	DESCRIPTION
G01 F20. Z-0.1 (Feed to cutting depth) ;	G01 F20. defines axis movements after it to be completed in a straight line. G01 requires the address code Fnnn.nnnn. The address code F20. specifies that the feed rate for the motion is 20" (508 mm) / min. Z-0.1 commands the Z Axis to Z = - 0.1.
X-4. Y-4. (linear motion) ;	X-4. Y-4. commands the X Axis to move to X = - 4.0 and commands the Y Axis to move to Y = - 4.0.

Completion

COMPLETION CODE BLOCK	DESCRIPTION
G00 Z0.1 M09 (Rapid retract, Coolant off) ;	G00 commands the axis motion to be completed in rapid motion mode. Z0.1 Commands the Z Axis to Z = 0.1. M09 commands the coolant to turn off.
G53 G49 Z0 M05 (Z home, Spindle off) ;	G53 defines axis movements after it to be with respect to the machine coordinate system. G49 cancels tool length compensation. Z0 is a command to move to Z = 0.0. M05 turns the spindle off.
G53 Y0 (Y home) ;	G53 defines axis movements after it to be with respect to the machine coordinate system. Y0 is a command to move to Y = 0.0.
M30 (End program) ;	M30 ends the program and moves the cursor on the control to the top of the program.
%	Denotes the end of a program written in a text editor.

Absolute vs. Incremental Positioning (G90, G91)

Absolute (G90) and incremental positioning (G91) define how the control interprets axis motion commands.

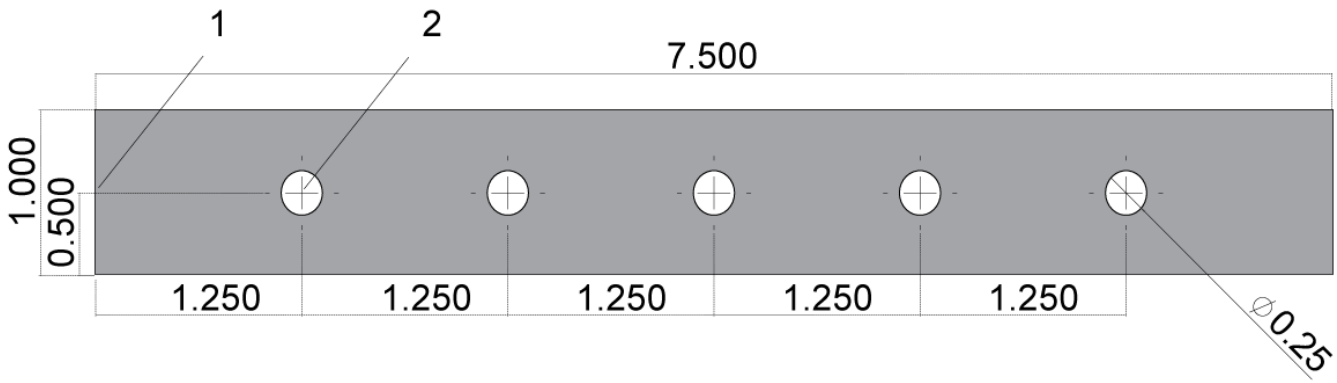
When you command axis motion after a G90 code, the axes move to that position relative to the origin of the coordinate system currently in use.

When you command axis motion after a G91, the axes move to that position relative to the current position.

Absolute programming is useful in most situations. Incremental programming is more efficient for repetitive, equally spaced cuts.

Figure 1 shows a part with 5 equally spaced Ø0.25" (13 mm) diameter holes. The hole depth is 1.00" (25.4 mm) and the spacing is 1.250" (31.75 mm) apart.

Figure 1 shows a part with 5 equally spaced Ø0.25" (13 mm) diameter holes. The hole depth is 1.00" (25.4 mm) and the spacing is 1.250" (31.75 mm) apart.

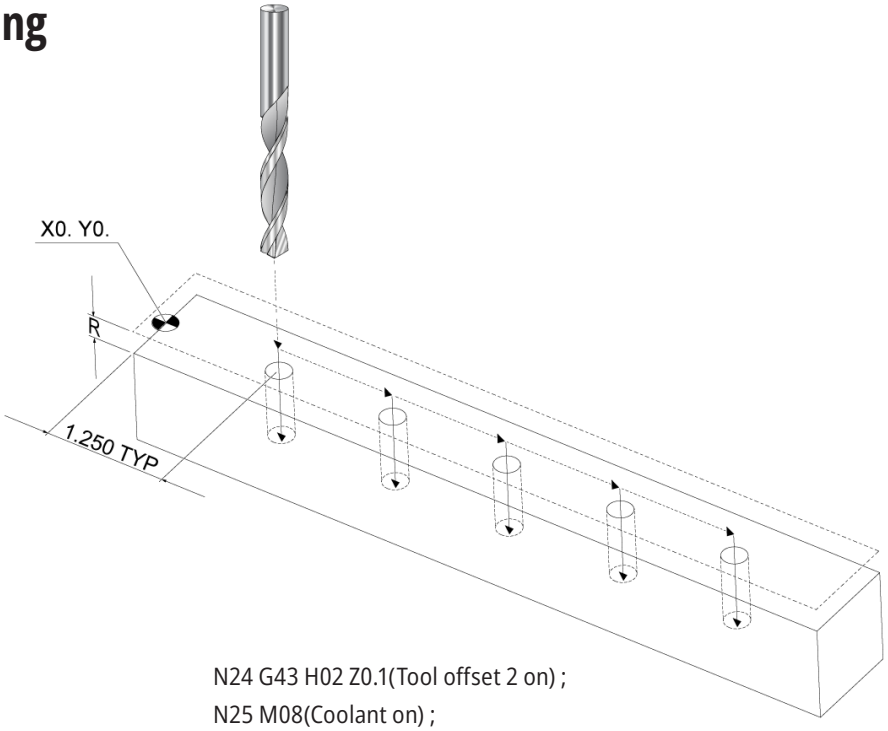


Absolute / Incremental Sample Program. G54 X0. Y0. for Incremental [1], G54 for Absolute [2]

Below and on the following page are two example programs that drill the holes as shown in the drawing, with a comparison between absolute and incremental positioning.

We start the holes with a center drill, and finish drilling the holes with a 0.250" (6.35 mm) drill bit. We use a 0.200" (5.08 mm) depth of cut for the center drill and 1.00" (25.4 mm) depth of cut for the 0.250" drill. G81, Drill Canned Cycle, is used to drill the holes.

Mill Incremental Positioning Example



%
O40002 (Incremental ex-prog) ;
N1 (G54 X0 Y0 is center left of part) ;
N2 (Z0 is on top of the part) ;
N3 (T1 is a center drill) ;
N4 (T2 is a drill) ;
N5 (T1 PREPARATION BLOCKS) ;
N6 T1 M06 (Select tool 1) ;
N7 G00 G90 G40 G49 G54 (Safe startup) ;
N8 X0 Y0 (Rapid to 1st position) ;
N9 S1000 M03 (Spindle on CW) ;
N10 G43 H01 Z0.1(Tool offset 1 on) ;
N11 M08(Coolant on) ;
N12 (T1 CUTTING BLOCKS) ;
N13 G99 G91 G81 F8.15 X1.25 Z-0.3 L5 ;
N14 (Begin G81, 5 times) ;
N15 G80 (Cancel G81) ;
N16 (T1 COMPLETION BLOCKS) ;
N17 G00 G90 G53 Z0. M09 (rapid retract, clnt off);
N18 M01 (Optional stop) ;
N19 (T2 PREPARATION BLOCKS) ;
N20 T2 M06 (Select tool 2) ;
N21 G00 G90 G40 G49 (Safe startup) ;
N22 G54 X0 Y0 (Rapid to 1st position) ;
N23 S1000 M03 (Spindle on CW) ;
N24 G43 H02 Z0.1(Tool offset 2 on) ;
N25 M08(Coolant on) ;
N26 (T2 CUTTING BLOCKS) ;
N27 G99 G91 G81 F21.4 X1.25 Z-1.1 L5 ;
N28 G80 (Cancel G81) ;
N29 (T2 COMPLETION BLOCKS) ;
N30 G00 Z0.1 M09 (Rapid retract, clnt off) ;
N31 G53 G90 G49 Z0 M05 (Z home, spindle off) ;
N32 G53 Y0 (Y home) ;
N33 M30 (End program) ;
%

Mill Absolute Positioning Example

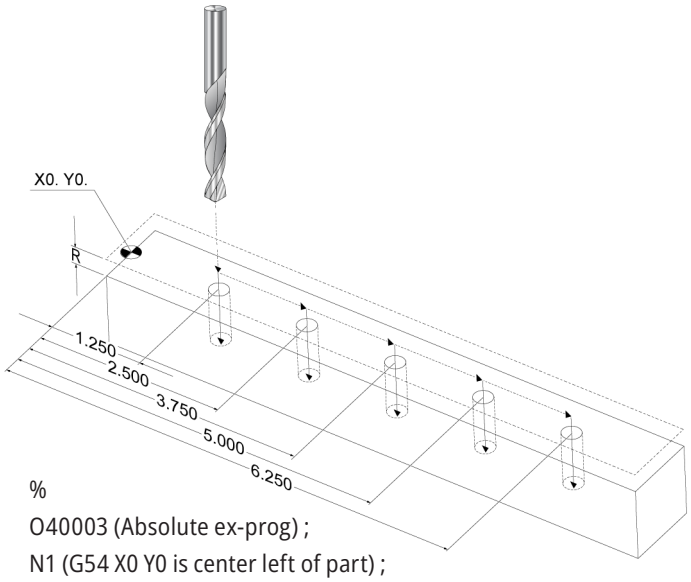
The absolute program method needs more lines of code than the incremental program. The programs have similar preparation and completion sections.

Look at line N13 in the incremental programming example, where the center drill operation begins. G81 uses the loop address code, Lnn, to specify the number of times to repeat the cycle. The address code L5 repeats this process (5) times. Each time the canned cycle repeats, it moves the distance that the optional X and Y values specify. In this program, the incremental program moves 1.25" in X from the current position with each loop, and then does the drill cycle.

For each drill operation, the program specifies a drill depth 0.1" deeper than the actual depth, because motion starts from 0.1" above the part.

In absolute positioning, G81 specifies the drill depth, but it does not use the loop address code. Instead, the program gives the position of each hole on a separate line. Until G80 cancels the canned cycle, the control does the drill cycle at each position.

The absolute positioning program specifies the exact hole depth, because the depth starts at the part surface (Z=0).




```
%
O40003 (Absolute ex-prog) ;
N1 (G54 X0 Y0 is center left of part) ;
N2 (Z0 is on top of the part) ;
N3 (T1 is a center drill) ;
N4 (T2 is a drill) ;
N5 (T1 PREPARATION BLOCKS) ;
N6 T1 M06 (Select tool 1) ;
N7 G00 G90 G40 G49 G54 (Safe startup) ;
N8 X1.25 Y0 (Rapid to 1st position) ;
N9 S1000 M03 (Spindle on CW) ;
N10 G43 H01 Z0.1 (Tool offset 1 on) ;
N11 M08 (Coolant on) ;
N12 (T1 CUTTING BLOCKS) ;
N13 G99 G81 F8.15 X1.25 Z-0.2 ;
N14 (Begin G81, 1st hole) ;
N15 X2.5 (2nd hole) ;
N16 X3.75 (3rd hole) ;
N17 X5. (4th hole) ;
N18 X6.25 (5th hole) ;
N19 G80 (Cancel G81) ;
N20 (T1 COMPLETION BLOCK) ;
N21 G00 G90 G53 Z0. M09 (Rapid retract, clnt off);
N22 M01 (Optional Stop) ;
N23 (T2 PREPARATION BLOCKS) ;
N24 T2 M06 (Select tool 2) ;
N25 G00 G90 G40 G49 (Safe startup) ;
N26 G54 X1.25 Y0 (Rapid to 1st position) ;
N27 S1000 M03 (Spindle on CW) ;
N28 G43 H02 Z0.1 (Tool offset 2 on) ;
N29 M08 (Coolant on) ;
N30 (T2 CUTTING BLOCKS) ;
N31 G99 G81 F21.4 X1.25 Z-1. (1st hole) ;
N32 X2.5 (2nd hole) ;
N33 X3.75 (3rd hole) ;
N34 X5. (4th hole) ;
N35 X6.25 (5th hole) ;
N36 G80 (Cancel G81) ;
N37 (T2 COMPLETION BLOCKS) ;
N38 G00 Z0.1 M09 (Rapid retract, Clnt off) ;
N39 G53 G49 Z0 M05 (Z home, Spindle off) ;
N40 G53 Y0 (Y home) ;
N41 M30 (End program) ;
%
```

G43 Tool Offset

The G43 Hnn Tool Length Compensation command should be used after every tool change. It adjusts the Z-Axis position to account for the length of the tool. The Hnn argument specifies which tool length to use. For more information see Setting Tool Offsets on the Operation section.

CAUTION: The tool length nn value should match the nn value from the M06 Tnn tool change command to avoid a possible collision.

Setting 15 - H & T Code Agreement controls whether the nn value needs to match in the Tnn and Hnn arguments. If Setting 15 is ON and the Tnn and Hnn do not match, Alarm 332 - H and T Not Matched is generated.

G54 Work Offsets

Work Offsets define where a work piece is located on the table.

Work Offsets available are G54-G59, G110-G129, and G154 P1-P99. G110-G129 and G154 P1-P20 refer to the same Work Offsets.

A useful feature is to set up multiple work pieces on the table and machining multiple parts in one machine cycle. This is accomplished by assigning each work piece to a different Work Offset.

For more information, reference the G-code section of this manual. Below is an example of machining multiple parts in one cycle. The program uses M97 Local Sub-Program Call in the cutting operation.

```
%
O40005 (Work offsets ex-prog) ;
(G54 X0 Y0 is center left of part) ;
(Z0 is on top of the part) ;
(T1 is a drill) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54(Safe startup) ;
X0 Y0 ;
(Move to first work coordinate position-G54) ;
S1000 M03 (Spindle on CW) ;
G43 H01 Z0.1 (Tool offset 1 on) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
M97 P1000 (Call local Subprogram) ;
G00 Z3. (Rapid retract) ;
G90 G110 G17 G40 G80 X0. Y0. ;
(Move to second work coordinate position-G110) ;
M97 P1000 (Call local Subprogram) ;
G00 Z3. (Rapid Retract) ;
G90 G154 P22 G17 G40 G80 X0. Y0. ;
(Move to third work coordinate position-G154 P22) ;
M97 P1000 (Call local Subprogram) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
N1000 (Local subprogram) ;
G81 F41.6 X1. Y2. Z-1.25 R0.1 (Begin G81) ;
(1st hole) ;
X2. Y2. (2nd hole) ;
G80 (Cancel G81) ;
M99 ;
%
```

Subprograms

Subprograms:

- Are usually a series of commands that are repeated several times in a program.
- Are written in a separate program, instead of repeating the commands many times in the main program.
- Are called in the main program with an M97 or M98 and a P code.
- Can include an L for repeat count. The subprogram call repeats L times before the main program continues with the next block.

When you use M97:

- The P code (nnnnn) is the same as the block number (Nnnnnn) of the local subprogram.
- The subprogram must be within the main program

When you use M98:

- The P code (nnnnn) is the same as the program number (Onnnnn) of the subprogram.
- If the subprogram is not in memory, the file name must be Onnnnn.nc. The file name must contain the O, leading zeros and .nc for the machine to find the subprogram.
- The subprogram must reside in the active directory, or in a location specified in Settings 251/252.
- Canned Cycles are the most common use of subprograms. For example, you might put the X and Y locations of a series of holes in a separate program. Then you can call that program as a subprogram with a canned cycle. Instead of writing the locations once for each tool, you write the locations only once for any number of tools.

Setting Up Search Locations

When program calls a subprogram, the control first looks for the subprogram in the active directory. If the control cannot find the subprogram, the control uses Settings 251 and 252 to determine where to look next. Refer to those settings for more information.

To build a list of search locations in Setting 252:

1. In the Device Manager (LIST PROGRAM), select the directory that you want to add to the list.
2. Press F3.
3. Highlight the SETTING 252 option in the menu, and then press ENTER.

The control adds the current directory to the list of search locations in Setting 252.

Result:

To see the list of search locations, look at the values of Setting 252 on the Settings page.

Local Subprogram (M97)

A local subprogram is a block of code in the main program that is referenced several times by the main program. Local subprograms are commanded (called) using an M97 and Pnnnnn, which refers to the N line number of the local subprogram.

The local subprogram format is to end the main program with an M30 then enter the local subprograms after the M30. Each subprogram must have an N line number at the start and a M99 at the end that will send the program back to the next line in the main program.

```
%
O40009 (Local subprogram ex-prog) ;
(G54 X0 Y0 is at the top left corner of part) ;
(Z0 is on top of the part) ;
(T1 is a spot drill) ;
(T2 is a drill) ;
(T3 is a tap) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54(Safe startup) ;
X1.5 Y-0.5 (Rapid to 1st position) ;
S1406 M03 (Spindle on CW) ;
G43 H01 Z1.(Tool offset 1 on) ;
M08(Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G81 G99 Z-0.26 R0.1 F7. (Begin G81) ;
M97 P1000 (Call local subprogram) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
M01 (Optional stop) ;
(BEGIN PREPARATION BLOCKS) ;
T2 M06 (Select tool 2) ;
G00 G90 G40 G49 (Safe startup) ;
G54 X1.5 Y-0.5 (Rapid back to 1st position) ;
S2082 M03 (Spindle on CW) ;
G43 H02 Z1. (Tool offset 2 on) ;
M08(Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G83 G99 Z-0.75 Q0.2 R0.1 F12.5 (Begin G83) ;
M97 P1000 (Call local subprogram) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
M01 (Optional stop) ;
(BEGIN PREPARATION BLOCKS) ;
T3 M06 (Select tool 3) ;
G00 G90 G40 G49 (Safe startup) ;
G54 X1.5 Y-0.5 ;
(Rapid back to 1st position) ;
S750 M03 (Spindle on CW) ;
G43 H03 Z1.(Tool offset 3 on) ;
M08(Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G84 G99 Z-0.6 R0.1 F37.5 (Begin G84) ;
M97 P1000 (Call local subprogram) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
(LOCAL subprogram) ;
N1000 (Begin local subprogram) ;
X0.5 Y-0.75 (2nd position) ;
Y-2.25 (3rd position) ;
G98 X1.5 Y-2.5 (4th position) ;
(Initial point return) ;
G99 X3.5 (5th position) ;
(R-plane return) ;
X4.5 Y-2.25 (6th position) ;
Y-0.75 (7th position) ;
X3.5 Y-0.5 (8th position) ;
M99 ;
%
```

External Subprogram (M98)

An external subprogram is a separate program that the main program references. Use M98 to command (call) an external subprogram, with Pnnnnn to refer to the program number you want to call.

When your program calls an M98 subprogram, the control looks for the subprogram in the main program's directory. If the control cannot find the subprogram in the main program's directory, it then looks in the location specified in Setting 251. An alarm occurs if the control cannot find the subprogram.

In this example, the subprogram (program O40008) specifies (8) positions. It also includes a G98 command at the move between positions 4 and 5. This causes the Z Axis to return to the initial starting point instead of the R plane, so the tool passes over the workholding.

The main program (Program O40007) specifies (3) different canned cycles:

1. G81 Spot drill at each position
2. G83 Peck drill at each position
3. G84 Tap at each position

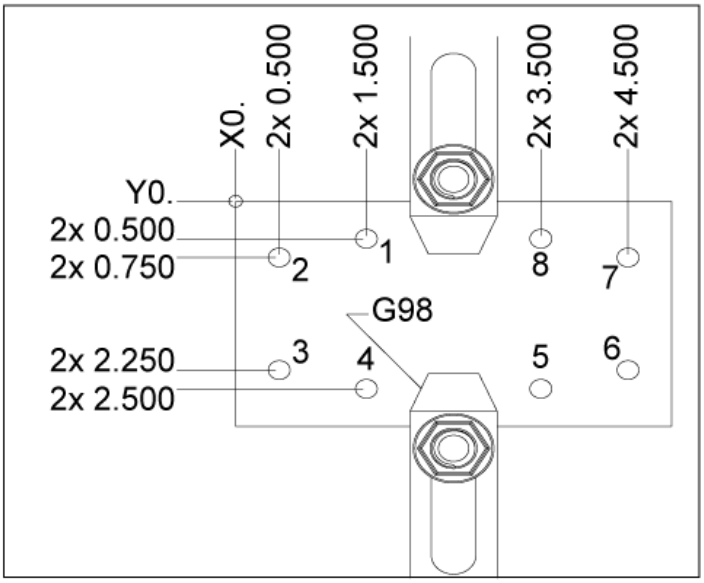
Each canned cycle calls the subprogram and does the operation at each position.

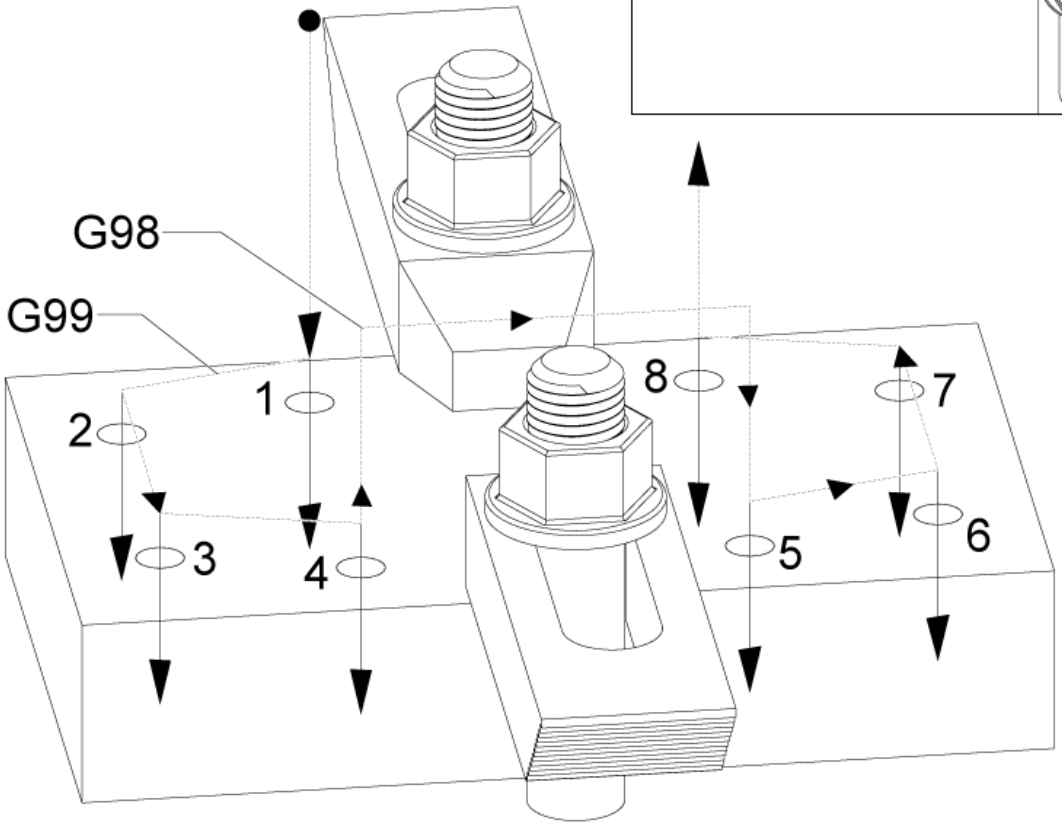
```
%
O40007 (External subprogram ex-prog) ;
(G54 X0 Y0 is center left of part) ;
(Z0 is on top of the part) ;
(T1 is a spot drill) ;
(T2 is a drill) ;
(T3 is a tap) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54 (Safe startup) ;
G00 G54 X1.5 Y-0.5 (Rapid to 1st position) ;
S1000 M03 (Spindle on CW) ;
G43 H01 Z1. (Tool offset 1 on) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G81 G99 Z-0.14 R0.1 F7. (Begin G81) ;
M98 P40008 (Call external subprogram) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z1. M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
M01 (Optional stop) ;
(BEGIN PREPARATION BLOCKS) ;
T2 M06 (Select tool 2) ;
G00 G90 G40 G49 G54 (Safe startup) ;
G00 G54 X1.5 Y-0.5 (Rapid to 1st position) ;
S2082 M03 (Spindle on CW) ;
G43 H02 Z1. (Tool offset 1 on) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G83 G99 Z-0.75 Q0.2 R0.1 F12.5 (Begin G83) ;
M98 P40008 (Call external subprogram) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z1. M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
M01 (Optional stop) ;
(BEGIN PREPARATION BLOCKS) ;
T3 M06 (Select tool 3) ;
G00 G90 G40 G49 G54 (Safe startup) ;
G00 G54 X1.5 Y-0.5 (Rapid to 1st position) ;
S750 M03 (Spindle on CW) ;
G43 H03 Z1. (Tool offset 3 on) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G84 G99 Z-0.6 R0.1 F37.5 (Begin G84) ;
M98 P40008 (Call external subprogram);
(BEGIN COMPLETION BLOCKS) ;
G00 Z1. M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%
```

External Subprogram (M98)

Subprogram

%
O40008 (Subprogram) ;
X0.5 Y-0.75 (2nd position) ;
Y-2.25 (3rd position) ;
G98 X1.5 Y-2.5 (4th position) ;
(Initial point return) ;
G99 X3.5 (5th position) ;
(R plane return) ;
X4.5 Y-2.25 (6th position);
Y-0.75 (7th position) ;
X3.5 Y-0.5 (8th position) ;
M99 (sub program return or loop) ;
%





Macros Introduction

NOTE: This control feature is optional; call your HFO for information on how to purchase it.

Macros add capabilities and flexibility to the control that are not possible with standard G-code. Some possible uses are: families of parts, custom canned cycles, complex motions, and driving optional devices. The possibilities are almost endless.

A Macro is any routine/subprogram that you can run multiple times. A macro statement can assign a value to a variable, read a value from a variable, evaluate an expression, conditionally or unconditionally branch to another point within a program, or conditionally repeat some section of a program.

Here are a few examples of the applications for Macros. The examples are outlines and not complete macro programs.

Tools For Immediate, On-Table Fixturing - You can semi-automate many setup procedures to assist the machinist. You can reserve tools for immediate situations that you did not anticipate in your application design. For instance, suppose a company uses a standard clamp with a standard bolt hole pattern. If you discovered, after setup, that a fixture needs an additional clamp, and suppose that you programmed macro subprogram 2000 to drill the bolt pattern of the clamp, then you only need this two-step procedure to add the clamp to the fixture:

a) Jog the machine to the X, Y, and Z coordinates and angle where you want to place the clamp. Read the position coordinates from the machine display.

b) Execute this command in MDI mode:

G65 P2000 Xnnn Ynnn Znnn Annn ;

where nnn are the coordinates determined in Step a). Here, macro 2000 (P2000) does the work since it was designed to drill the clamp bolt hole pattern at the specified angle of A. Essentially, this is a custom canned cycle.

Simple Patterns That Are Repeated - You can define and store repeated patterns with macros. For example:

a) Bolt hole patterns

b) Slotting

c) Angular patterns, any number of holes, at any angle, with any spacing

d) Specialty milling such as soft jaws

e) Matrix Patterns, (e.g. 12 across and 15 down)

f) Fly-cutting a surface, (e.g. 12 inches by 5 inches using a 3 inch fly cutter)

Automatic Offset Setting Based On The Program - With macros, coordinate offsets can be set in each program so that setup procedures become easier and less error-prone (macro variables #2001-2800).

Probing - Using a probe enhances the capabilities of the machine, some examples are:

a) Profiling of a part to determine unknown dimensions for machining.

b) Tool calibration for offset and wear values.

c) Inspection prior to machining to determine material allowance on castings.

d) Inspection after machining to determine parallelism and flatness values as well as location.

Useful G and M Codes

M00, M01, M30 - Stop Program

G04 - Dwell

G65 Pxx - Macro subprogram call. Allows passing of variables.

M29 - Set output relay with M-FIN.

M129 - Set output relay with M-FIN.

M59 - Set output relay.

M69 - Clear output relay.

M96 Pxx Qxx - Conditional Local Branch when Discrete Input Signal is 0

M97 Pxx - Local Sub Routine Call

M98 Pxx - Sub Program Call

M99 - Sub Program Return or Loop

G103 - Block Lookahead Limit. No cutter comp allowed.

M109 - Interactive User Input

Round Off

The control stores decimal numbers as binary values. As a result, numbers stored in variables can be off by 1 least significant digit. For example, the number 7 stored in macro variable #10000, may later be read as 7.000001, 7.000000, or 6.999999.

If your statement was

IF [#10000 EQ 7]... ; it may give a false reading. A safer way of programming this would be

IF [ROUND [#10000] EQ 7]... ;

This issue is usually a problem only when you store integers in macro variables where you do not expect to see a fractional part later.

Look-ahead

Look-ahead is a very important concept in macro programming. The control attempts to process as many lines as possible ahead of time in order to speed up processing. This includes the interpretation of macro variables. For example,

#12012 = 1 ;

G04 P1. ;

#12012 = 0 ;

This is intended to turn an output on, wait 1 second, and then turn it off. However, lookahead causes the output to turn on then immediately back off while the control processes the dwell. G103 P1 is used to limit lookahead to 1 block. To make this example work properly, modify it as follows:

G103 P1 (See the G-code section of the manual for a further explanation of G103) ;

;

#12012=1 ;

G04 P1. ;

;

;

;

#12012=0 ;

Block Look-Ahead and Block Delete

The Haas control uses block look-ahead to read and prepare for blocks of code that come after the current block of code. This lets the control transition smoothly from one motion to the next. G103 limits how far ahead the control looks at blocks of code. The Pnnaddress code in G103 specifies how far ahead the control is allowed to look. For additional information, refer to G103 Limit Block Look-Ahead (Group 00)

Block Delete mode lets you selectively skip blocks of code. Use a / character at the beginning of the program blocks that you want to skip. Press BLOCK DELETE to enter the Block Delete mode. While Block Delete mode is active, the control does not execute the blocks marked with a / character. For example:

Using a

/M99 (Sub-Program Return) ;

before a block with

M30 (Program End and Rewind) ;

makes the sub-program a main program when BLOCK DELETE is on. The program is used as a sub-program when Block Delete is off.

When a block delete token "/" is used, even if Block Delete mode is not active, the line will block look ahead. This is useful for debugging macro processing within NC programs.

Macro Variable Display Page

You save or load macro variables through the Net Share or USB port, much like settings, and offsets.

The local and global macro variables #1 - #33 and #10000 - #10999 are displayed and modified through the Current Commands display.

NOTE: Internal to the machine, 10000 is added to 3-digit macro variables. For example: Macro 100 is displayed as 10100.

1

Press CURRENT COMMANDS and use navigation keys to get to the Macro Vars page.

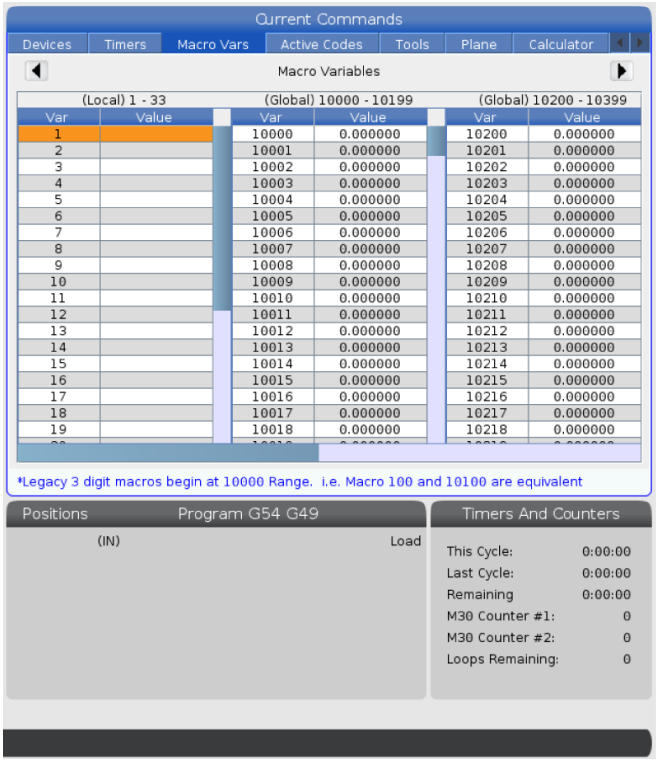
As the control interprets a program, the variable changes and results are displayed on the Macro Vars display page.

Enter a value (maximum is 999999.000000) and then press ENTER to set the macro variable. Press ORIGIN to clear macro variables, this displays the Origin clear entry popup. Press number 1 - 3 to make a selection or press CANCEL to exit.

2

To search for a variable, enter the macro variable number and press the up or down arrow.

The variables displayed represent values of the variables when the program runs. At times, this may be up to 15 blocks ahead of actual machine actions. Debugging programs is easier when a G103 P1 is inserted at the beginning of a program to limit block buffering. A G103 without the P value can be added after the macro variable blocks in the program. For a macro program to operate correctly it is recommended that the G103 P1 be left in the program during loading of variables. For more details about G103 see the G-code section of the manual.

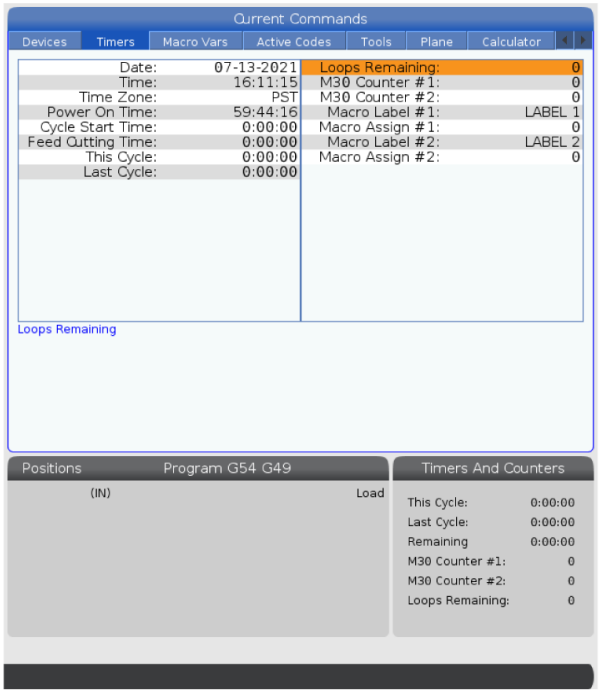


Display Macro Variables in the Timers And Counters Window

1

In the Timers And Counters window, you can display the values of any two macro variables and assign them a display name.

To set which two macro variables display in the Timers And Counters window:



2

Press CURRENT COMMANDS.

Use the navigation keys to select the TIMERS page.

Highlight the Macro Label #1 name or Macro Label #2 name.

Key in a new name, and press ENTER.

Use arrow keys to pick the Macro Assign #1 or Macro Assign #2 entry field (corresponding to your chosen Macro Label name).

Key in the macro variable number (without #) and press ENTER.

RESULTS:

On the Timers And Counters window, the field to the right of the entered Macro Label (#1 or #2) name displays the assigned variable value.

Macro Arguments

The arguments in a G65 statement are a means to send values to a macro subprogram and set the local variables of a macro subprogram.

The next (2) tables indicate the mapping of the alphabetic address variables to the numeric variables used in a macro subprogram.

Alphabetic Addressing

TABLE 1: Alphabetic Address Table

ADDRESS	VARIABLE		ADDRESS	VARIABLE
A	1		N	-
B	2		O	-
C	3		P	-
D	7		Q	17
E	8		R	18
F	9		S	19
G	-		T	20
H	11		U	21
I	4		V	22
J	5		W	23
K	6		X	24
L	-		Y	25
M	13		Z	26

13.3 | MILL MACRO - ARGUMENTS

TABLE 2: Alternate Alphabetic Addressing

ADDRESS	VARIABLE		ADDRESS	VARIABLE		ADDRESS	VARIABLE
A	1		K	12		J	23
B	2		I	13		K	24
C	3		J	14		I	25
I	4		K	15		J	26
J	5		I	16		K	27
K	6		J	17		I	28
I	7		K	18		J	29
J	8		I	19		K	30
K	9		J	20		I	31
I	10		K	21		J	32
J	11		I	22		K	33

13.3 | MILL MACRO - ARGUMENTS

Arguments accept any floating-point value to four decimal places. If the control is in metric, it will assume thousandths (.000). In example below, local variable #1 will receive .0001. If a decimal is not included in an argument value, such as:

G65 P9910 A1 B2 C3 ;

The values are passed to macro subprograms according to this table:

Integer Argument Passing (no decimal point)

ADDRESS	VARIABLE		ADDRESS	VARIABLE		ADDRESS	VARIABLE
A	.0001		J	.0001		S	1.
B	.0002		K	.0001		T	1.
C	.0003		L	1.		U	.0001
D	1.		M	1.		V	.0001
E	1.		N	-		W	.0001
F	1.		O	-		X	.0001
G	-		P	-		Y	.0001
H	1.		Q	.0001		Z	.0001
I	.0001		R	.0001		-	-

All 33 local macro variables can be assigned values with arguments by using the alternate addressing method. The following example shows how to send two sets of coordinate locations to a macro subprogram. Local variables #4 through #9 would be set to .0001 through .0006 respectively.

Example:

G65 P2000 I1 J2 K3 I4 J5 K6;

The following letters cannot be used to pass parameters to a macro subprogram: G, L, N, O or P.

Macro Variables

There are (3) categories of macro variables: local, global, and system.

Macro constants are floating-point values placed in a macro expression. They can be combined with addresses A-Z, or they can stand alone when used within an expression. Examples of constants are 0.0001, 5.3 or -10.

Local Variables

Local variables range between #1 and #33. A set of local variables is available at all times. When a call to a subprogram with a G65 command is executed, local variables are saved and a new set is available for use. This is called nesting of local variables. During a G65 call, all new local variables are cleared to undefined values and any local variables that have corresponding address variables in the G65 line are set to G65 line values. Below is a table of the local variables along with the address variable arguments that change them:

Variable:	1	2	3	4	5	6	7	8	9	10	11
Address:	A	B	C	I	J	K	D	E	F	-	H
Alternate:	-	-	-	-	-	-	I	J	K	I	J
Variable:	12	13	14	15	16	17	18	19	20	21	22
Address:	-	M	-	-	-	Q	R	S	T	U	V
Alternate:	K	I	J	K	I	J	K	I	J	K	I
Variable:	23	24	25	26	27	28	29	30	31	32	33
Address:	W	X	Y	Z	-	-	-	-	-	-	-
Alternate:	J	K	I	J	K	I	J	K	I	J	K

Variables 10, 12, 14- 16 and 27- 33 do not have corresponding address arguments. They can be set if a sufficient number of I, J and K arguments are used as indicated above in the section about arguments. Once in the macro subprogram, local variables can be read and modified by referencing variable numbers 1- 33.

When the L argument is used to do multiple repetitions of a macro subprogram, the arguments are set only on the first repetition. This means that if local variables 1- 33 are modified in the first repetition, then the next repetition will have access only to the modified values. Local values are retained from repetition to repetition when the L address is greater than 1.

Calling a subprogram via an M97 or M98 does not nest the local variables. Any local variables referenced in a subprogram called by an M98 are the same variables and values that existed prior to the M97 or M98 call.

Global Variables

Global variables are accessible at all times and remain in memory when power is turned off. There is only one copy of each global variable. Global variables are numbered #10000-#10999. Three legacy ranges: (#100-#199, #500-#699, and #800-#999) are included. The legacy 3 digit macro variables begin at the #10000 range; ie., macro variable #100 is displayed as #10100.

NOTE: Using variable #100 or #10100 in a program the control will access the same data. Using either variable number is acceptable.

Sometimes, factory-installed options use global variables, for example, probing and pallet changers, etc. Refer to the Macro Variables Table for global variables and their use.

CAUTION: When you use a global variable, make sure that no other programs on the machine use the same global variable.

System Variables

System variables let you interact with a variety of control conditions. System variable values can change the function of the control. When a program reads a system variable, it can modify its behavior based on the value in the variable. Some system variables have a Read Only status; this means that you cannot modify them. Refer to the Macro Variables Table for a list of system variables and their use.

Macro Variables

The macro variables table of local, global, and system variables and their usage follows. The new generation control variables list includes legacy variables.

NGC VARIABLE	LEGACY VARIABLE	USAGE
#0	#0	Not a number (read only)
#1- #33	#1- #33	Macro call arguments
#10000- #10149	#100- #149	General-purpose variables saved on power off
#10150- #10199	#150- #199	Probe values (if installed)
#10200- #10399	N/A	General-purpose variables saved on power off
#10400- #10499	N/A	General-purpose variables saved on power off
#10500- #10549	#500-#549	General-purpose variables saved on power off
#10550- #10599	#550-#599	Probe calibration data (if installed)
#10600- #10699	#600- #699	General-purpose variables saved on power off
#10700- #10799	N/A	General-purpose variables saved on power off
#700- #749	#700- #749	Hidden variables for internal use only
#709	#709	Used for the Fixture Clamp Input. Do not use for general purpose.
#10800- #10999	#800- #999	General-purpose variables saved on power off
#11000- #11063	N/A	64 discrete inputs (read only)
#1064- #1068	#1064- #1068	Maximum axis loads for X, Y, Z, A, and B Axes, respectively
#1080- #1087	#1080- #1087	Raw analog to digital inputs (read only)
#1090- #1098	#1090- #1098	Filtered analog to digital inputs (read only)
#1098	#1098	Spindle load with Haas vector drive (read only)
#1264- #1268	#1264- #1268	Maximum axis loads for C, U, V, W, and T-axes respectively
#1601- #1800	#1601- #1800	Number of flutes on tools #1 through 200
#1801- #2000	#1801- #2000	Maximum recorded vibrations of tools 1 through 200
#2001- #2200	#2001- #2200	Tool length offsets
#2201- #2400	#2201- #2400	Tool length wear

13.5 | MILL MACRO - VARIABLES TABLE

Macro Variables Table (continue)

NGC VARIABLE	LEGACY VARIABLE	USAGE
#2401- #2600	#2401- #2600	Tool diameter/radius offsets
#2601- #2800	#2601- #2800	Tool diameter/radius wear
#3000	#3000	Programmable alarm
#3001	#3001	Millisecond timer
#3002	#3002	Hour timer
#3003	#3003	Single block suppression
#3004	#3004	Override FEED HOLD control
#3006	#3006	Programmable stop with message
#3011	#3011	Year, month, day
#3012	#3012	Hour, minute, second
#3020	#3020	Power on timer (read only)
#3021	#3021	Cycle start timer
#3022	#3022	Feed timer
#3023	#3023	Present part timer (read only)
#3024	#3024	Last complete part timer (read only)
#3025	#3025	Previous part timer (read only)
#3026	#3026	Tool in spindle (read only)
#3027	#3027	Spindle RPM (read only)
#3028	#3028	Number of pallets loaded on receiver
#3030	#3030	Single block
#3032	#3032	Block delete
#3033	#3033	Opt stop
#3034	N/A	Safe Run (read only)

Macro Variables Table (continue)

NGC VARIABLE	LEGACY VARIABLE	USAGE
#3196	#3196	Cell safe timer
#3201- #3400	#3201- #3400	Actual diameter for tools 1 through 200
#3401- #3600	#3401- #3600	Programmable coolant positions for tools 1 through 200
#3901	#3901	M30 count 1
#3902	#3902	M30 count 2
#4001- #4021	#4001- #4021	Previous block G-code group codes
#4101- #4126	#4101- #4126	Previous block address codes.
#4101- #4126	#4101- #4126	Previous block address codes. NOTE: (1) Mapping of 4101 to 4126 is the same as the alphabetic addressing of Macro Arguments section; e.g., the statement X1.3 sets variable #4124 to 1.3.
#5001- #5006	#5001- #5006	Previous block end position
#5021- #5026	#5021- #5026	Present machine coordinate position
#5041- #5046	#5041- #5046	Present work coordinate position
#5061- #5069	#5061- #5069	Present skip signal position - X, Y, Z, A, B, C, U, V, W
#5081- #5086	#5081- #5086	Present tool offset
#5201- #5206	#5201- #5206	G52 work offsets
#5221- #5226	#5221- #5226	G54 work offsets
#5241- #5246	#5241- #5246	G55 work offsets
#5261- #5266	#5261- #5266	G56 work offsets
#5281- #5286	#5281- #5286	G57 work offsets
#5301- #5306	#5301- #5306	G58 work offsets
#5321- #5326	#5321- #5326	G59 work offsets
#5401- #5500	#5401- #5500	Tool feed timers (seconds)
#5501- #5600	#5501- #5600	Total tool timers (seconds)
#5601- #5699	#5601- #5699	Tool life monitor limit
#5701- #5800	#5701- #5800	Tool life monitor counter
#5801- #5900	#5801- #5900	Tool load monitor maximum load sensed so far

Macro Variables Table (continue)

NGC VARIABLE	LEGACY VARIABLE	USAGE
#5901- #6000	#5901- #6000	Tool load monitor limit
#6001- #6999	#6001- #6999	Reserved. Do not use.
#6198		NGC/CF flag
#7001- #7006	#7001- #7006	G110 (G154 P1) additional work offsets
#7021- #7026	#7021- #7026	G111 (G154 P2) additional work offsets
#7041- #7386	#7041- #7386	G112 - G129 (G154 P3 - P20)additional work offsets
#7501- #7506	#7501- #7506	Pallet priority
#7601- #7606	#7601- #7606	Pallet status
#7701- #7706	#7701- #7706	Part program numbers assigned to pallets
#7801- #7806	#7801- #7806	Pallet usage count
#8500	#8500	Advanced Tool Management (ATM) group ID
#8501	#8501	ATM percent of available tool life of all tools in the group
#8502	#8502	ATM total available tool usage count in the group
#8503	#8503	ATM total available tool hole count in the group
#8504	#8504	ATM total available tool feed time (in seconds) in the group
#8505	#8505	ATM total available tool total time (in seconds) in the group
#8510	#8510	ATM next tool number to be used
#8511	#8511	ATM percent of available tool life of the next tool
#8512	#8512	ATM available usage count of the next tool
#8513	#8513	ATM available hole count of the next tool
#8514	#8514	ATM available feed time of the next tool (in seconds)
#8515	#8515	ATM available total time of the next tool (in seconds)
#8550	#8550	Individual tool ID
#8551	#8551	Number of flutes of tools
#8552	#8552	Maximum recorded vibrations

Macro Variables Table (continue)

NGC VARIABLE	LEGACY VARIABLE	USAGE
#8553	#8553	Tool length offsets
#8554	#8554	Tool length wear
#8555	#8555	Tool diameter offsets
#8556	#8556	Tool diameter wear
#8557	#8557	Actual diameter
#8558	#8558	Programmable coolant position
#8559	#8559	Tool feed timer (seconds)
#8560	#8560	Total tool timers (seconds)
#8561	#8561	Tool life monitor limit
#8562	#8562	Tool life monitor counter
#8563	#8563	Tool load monitor maximum load sensed so far
#8564	#8564	Tool load monitor limit
#9000	#9000	Thermal comp accumulator
#9000- #9015	#9000- #9015	Reserved (duplicate of axis thermal accumulator)
#9016	#9016	Thermal spindle comp accumulator
#9016- #9031	#9016- #9031	Reserved (duplicate of axis thermal accumulator from spindle)
#10000- #10999	N/A	General purpose variables
#11000- #11255	N/A	Discrete inputs (read only)
#12000- #12255	N/A	Discrete outputs
#13000- #13063	N/A	Filtered analog to digital inputs (read only)
#13013	N/A	Coolant level
#14001- #14006	N/A	G110(G154 P1) additional work offsets
#14021- #14026	N/A	G110(G154 P2) additional work offsets
#14041- #14386	N/A	G110(G154 P3- G154 P20) additional work offsets
#14401- #14406	N/A	G110(G154 P21) additional work offsets

Macro Variables Table (continue)

NGC VARIABLE	LEGACY VARIABLE	USAGE
#14421- #15966	N/A	G110(G154 P22- G154 P99) additional work offsets
#20000- #29999	N/A	Settings
#30000- #39999	N/A	Parameters
#32014	N/A	Machine Serial Number
#50001- #50200	N/A	Tool Type
#50201- #50400	N/A	Tool material
#50401- #50600	N/A	Tool Offset Point
#50601- #50800	N/A	Estimated RPM
#50801- #51000	N/A	Estimated Feedrate
#51001- #51200	N/A	Offset Pitch
#51201- #51400	N/A	Actually VPS Estimated RPM
#51401- #51600	N/A	Work Material
#51601- #51800	N/A	VPS Feedrate
#51801- #52000	N/A	Approximate length
#52001- #52200	N/A	Approximate diameter
#52201- #52400	N/A	Edge Measure height
#52401- #52600	N/A	Tool Tolerance
#52601- #52800	N/A	Probe Type

System Variables In-Depth

System variables are associated with specific functions. A detailed description of these functions follows.

#550-#699 #10550- #10699 General and Probe Calibration Data

These general purpose variables are saved on power off. Some of these higher #5xx variables store probe calibration data. Example: #592 sets which side of the table the tool probe is positioned. If these variables are overwritten, you will need to calibrate the probe again.

NOTE: If the machine does not have a probe installed, you can use these variables as general-purpose variables saved on power off.

#1080-#1097 #11000-#11255 #13000-#13063 1-Bit Discrete Inputs

You can connect designated inputs from external devices with these macros:

NGC VARIABLE	LEGACY VARIABLE	USAGE
#11000-#11255	-	256 discrete inputs (read only)
#13000-#13063	#1080-#1087	Raw and Filtered analog to digital inputs (read only)

Specific input values can be read from within a program. The format is #11nnn where nnn is the Input Number. Press DIAGNOSTIC and select the I/O tab to see the Input and Output numbers for different devices.

Example:
#10000=#11018

This example records the state of #11018, which refers to Input 18 (M-Fin_Input), to variable #10000.

For available User Inputs on the I/O PCB, refer to the Robot Integration Aid reference document in the Haas service website.

#12000-#12255 1-Bit Discrete Outputs

The Haas control is capable of controlling up to 256 discrete outputs. However, a number of these outputs are reserved for the Haas control to use.

NGC VARIABLE	LEGACY VARIABLE	USAGE
#12000-#12255	-	256 discrete outputs

13.6 | MILL MACRO - VARIABLES

Specific output values can be read, or written to, from within a program. The format is #12nnn where nnn is the Output Number.

Example:

#10000=#12018 ;

This example records the state of #12018, which refers to Input 18 (Coolant Pump Motor), to variable #10000.

Maximum Axis Loads

These variables contain the maximum load an axis has achieved since the machine was last powered on, or since that Macro Variable was cleared. The Maximum Axis Load is the greatest load (100.0 = 100%) an axis has seen, not the Axis Load at the time that the control reads the variable.

#1064 = X Axis	#1264 = C axis
#1065 = Y Axis	#1265 = U axis
#1066 = Z Axis	#1266 = V axis
#1067 = A Axis	#1267 = W axis
#1068 = B Axis	#1268 = T axis

Tool Offsets

Each tool offset has a length (H) and diameter (D) along with associated wear values.

#2001-#2200	H geometry offsets (1-200) for length.
#2201-#2400	H geometry wear (1-200) for length.
#2401-#2600	D geometry offsets (1-200) for diameter.
#2601-#2800	D geometry wear (1-200) for diameter.

System Variables In-Depth (Cont.)

#3000 Programmable Alarm Messages

#3000 Alarms can be programmed. A programmable alarm will act like the built-in alarms. An alarm is generated by setting macro variable #3000 to a number between 1 and 999.

#3000= 15 (MESSAGE PLACED INTO ALARM LIST) ;

When this is done, Alarm flashes at the bottom of the display and the text in the next comment is placed into the alarm list.

The alarm number (in this example, 15) is added to 1000 and used as an alarm number. If an alarm is generated in this manner all motion stops and the program must be reset to continue. Programmable alarms are always numbered between 1000 and 1999.

#3001-#3002 Timers

Two timers can be set to a value by assigning a number to the respective variable. A program can then read the variable and determine the time passed since the timer was set. Timers can be used to imitate dwell cycles, determine part-to-part time or wherever time-dependent behavior is desired.

- #3001 Millisecond Timer - The millisecond timer represents the system time after power on in number of milliseconds. The whole number returned after accessing #3001 represents the number of milliseconds.
- #3002 Hour Timer - The hour timer is similar to the millisecond timer except that the number returned after accessing#3002 is in hours. The hour and millisecond timers are independent of each other and can be set separately.

System Overrides

Variable #3003 overrides the Single Block function in G-code.

When #3003 has a value of 1, the control executes each G-code command continuously even though the Single Block function is ON.

When #3003 has a value of zero, Single Block operates as normal. You must press CYCLE START to execute each line of code in single block mode.

```
#3003=1 ;
G54 G00 G90 X0 Y0 ;
S2000 M03 ;
G43 H01 Z.1 ;
G81 R.1 Z-0.1 F20. ;
#3003=0 ;
T02 M06 ;
G43 H02 Z.1 ;
S1800 M03 ;
G83 R.1 Z-1. Q.25 F10. ;
X0. Y0. ;
%
```

Variable #3004

Variable #3004 overrides specific control features during operation.

The first bit disables FEED HOLD. If variable #3004 is set to 1, FEED HOLD is disabled for the program blocks that follow. Set #3004 to 0 to enable FEED HOLD again. For example:

..

(Approach code - FEED HOLD allowed) ;

#3004=1 (Disables FEED HOLD) ;

(Non-stoppable code - FEED HOLD not allowed) ;

#3004=0 (Enables FEED HOLD) ;

(Depart code - FEED HOLD allowed) ;

...

Variable #3004 resets to 0 at M30.

This is a map of variable #3004 bits and the associated overrides.

E = Enabled D = Disabled

#3004	FEED HOLD	FEED RATE OVERRIDE	EXACT STOP CHECK
0	E	E	E
1	D	E	E
2	E	D	E
3	D	D	E
4	E	E	D
5	D	E	D
6	E	D	D
7	D	D	D

#3006 Programmable Stop

You can add stops to the program that act like an M00 - The control stops and waits until you press CYCLE START, then the program continues with the block after the #3006. In this example, the control displays the comment on the lower-center of the screen.

#3006=1 (comment here)

System Variables In-Depth (Cont.)

#3030 Single Block

In Next Generation Control when the system variable #3030 is set to a 1; the control will go into single block mode. There is no need to limit the lookahead using a G103 P1, the Next Generation Control will correctly process this code.

NOTE: For the Classic Haas Control to process system variable #3030=1 correctly, it is necessary to limit the lookahead to 1 block using a G103 P1 before the #3030=1 code.

#4001-#4021 Last Block (Modal) Group Codes

G-code groups let the machine control process the codes more efficiently. G-codes with similar functions are usually in the same group. For example, G90 and G91 are under group 3. Macro variables #4001 through #4021 store the last or default G code for any of 21 groups.

G-Codes Group number is listed next to it's description in the G-Code section.

Example:

G81 Drill Canned Cycle (Group 09)

When a macro program reads the group code, the program can change the behavior of the G-code. If #4003 contains 91, then a macro program could determine that all moves should be incremental rather than absolute. There is no associated variable for group zero; group zero G codes are Non-modal.

#4101-#4126 Last Block (Modal) Address Data

Address codes A-Z (excluding G) are maintained as modal values. The information represented by the last line of code interpreted by the lookahead process is contained in variables #4101 through #4126.

The numeric mapping of variable numbers to alphabetic addresses corresponds to the mapping under alphabetic addresses. For example, the value of the previously interpreted D address is found in #4107 and the last interpreted I value is #4104. When aliasing a macro to an M-code, you may not pass variables to the macro using variables #1 - #33. Instead, use the values from #4101 - #4126 in the macro.

#5001-#5006 Last Target Position

The final programmed point for the last motion block can be accessed through variables #5001 - #5006, X, Z, Y, A, B, and C respectively. Values are given in the current work coordinate system and can be used while the machine is in motion.

#5021-#5026 Current Machine Coordinate Position

To get the current machine axis positions, call macro variables #5021-#5026 corresponding to axis X, Y, Z, A, B, and C, respectively.

#5021 X Axis	#5022 Y Axis	#5023 Z Axis
#5024 A Axis	#5025 B Axis	#5026 C Axis

NOTE: Values CANNOT be read while the machine is in motion.

#5041-#5046 Current Work Coordinate Position

To get the current work coordinate positions, call macro variables #5041-#5046 corresponding to axis X, Y, Z, A, B, and C, respectively.

NOTE: The values CANNOT be read while the machine is in motion. The value of #504X has tool length compensation applied to it.

#5061-#5069 Current Skip Signal Position

Macro variables #5061-#5069 corresponding to X, Y, Z, A, B, C, U, V and W respectively, give the axis positions where the last skip signal occurred. Values are given in the current work coordinate system and can be used while the machine is in motion.

The value of #5063 (Z) has tool length compensation applied to it.

#5081-#5086 Tool Length Compensation

Macro variables #5081 - #5086 give the current total tool length compensation in axis X, Y, Z, A, B, or C, respectively. This includes tool length offset referenced by the current value set in H (#4008) plus the wear value.

#5201-#5326, #7001-#7386, #14001-#14386 Work Offsets

Macro expressions can read and set all work offsets. This lets you preset coordinates to exact locations, or set coordinates to values based upon the results of skip signal (probed) locations and calculations.

When any of the offsets are read, the interpretation look-ahead queue is stopped until that block is executed.

#6001-#6250 Settings Access with Macro Variables

Access settings through variables #20000 - #20999 or #6001 - #6250, starting from setting 1 respectively. Refer to Chapter 19 for the detailed descriptions of the settings that are available in the control.

NOTE: The #20000 - 20999 range numbers correspond directly to Setting numbers. You should use #6001 - #6250 for settings access only if you need your program to be compatible with older Haas machines

System Variables In-Depth (Cont.)

#6198 Next-Generation Control Identifier

The macro variable #6198 has a read-only value of 1000000.

You can test #6198 in a program to detect the control version, and then conditionally run program code for that control version. For example:

%

IF[#6198 EQ 1000000] GOTO5 ;

(Non-NGC code) ;

GOTO6 ;

N5 (NGC code) ;

N6 M30 ;

%

In this program, if the value stored in #6198 is equal to 1000000, go to Next Generation Control compatible code then end the program. If if the value stored in #6198 is not equal to 1000000, run the non-NGC program and then end the program.

#6996-#6999 Parameter Access With Macro Variables

These macro variables can access all parameters and any of the parameter bits, as follows:

- #6996: Parameter Number
- #6997: Bit Number (optional)
- #6998: Contains the value of the parameter number specified in variable #6996
- #6999: Contains the bit value (0 or 1) of the parameter bit specified in variable #6997.

NOTE: Variables #6998 and #6999 are read-only.

You can also use macro variables #30000 - #39999, starting from parameter 1, respectively. Contact your HFO for more details regarding parameter numbers.

USAGE:

To access the value of a parameter, copy the number of that parameter into variable #6996. The value of that parameter is available in macro variable #6998, as shown:

%

#6996=601 (Specify parameter 601) ;

#10000=#6998 (Copy the value of parameter 601 to variable #10000) ;

%

To access a specific parameter bit, copy the parameter number into variable 6996 and the bit number to macro variable 6997. The value of that parameter bit is available in macro variable 6999, as shown:

%

#6996=57 (Specify parameter 57) ;

#6997=0 (Specify bit zero) ;

#10000=#6999 (Copy parameter 57 bit 0 to variable #10000) ;

%

Pallet Changer Variables

The status of the pallets from the Automatic Pallet Changer is checked with these variables:

#7501-#7506	Pallet priority
#7601-#7606	Pallet status
#7701-#7706	Part program numbers assigned to pallets
#7801-#7806	Pallet usage count
#3028	Number of pallet loaded on receiver

#8500-#8515 Advanced Tool Management

These variables give information on Advanced Tool Management (ATM). Set variable #8500 to the tool group number, then access information for the selected tool group with the read-only macros #8501-#8515.

#8500	Advanced Tool Management (ATM). Group ID
#8501	ATM. Percent of available tool life of all tools in the group.
#8502	ATM. Total available tool usage count in the group.
#8503	ATM. Total available tool hole count in the group.
#8504	ATM. Total available tool feed time (in seconds) in the group.
#8505	ATM. Total available tool total time (in seconds) in the group.
#8510	ATM. Next tool number to be used.
#8511	ATM. Percent of available tool life of the next tool.
#8512	ATM. Available usage count of the next tool.
#8513	ATM. Available hole count of the next tool.
#8514	ATM. Available feed time of the next tool (in seconds).
#8515	ATM. Available total time of the next tool (in seconds).

System Variables In-Depth (Cont.)

#8550-#8567 Advanced Tool Management Tooling

These variables give information on tooling. Set variable #8550 to the tool offset number, then access information for the selected tool with the read-only macros #8551-#8567

NOTE: Macro variables #1601-#2800 give access to the same data for individual tools as #8550-#8567 give for Tool Group tools.

#50001 - #50200 Tool Type

Use macro variables #50001 - #50200, to read or write the tool type set in the tool offset page.

Available Tool Types for Mill

TOOL TYPE	TOOL TYPE#
Drill	1
Tap	2
Shell Mill	3
End Mill	4
Spot Drill	5
Ball Nose	6
Probe	7
Reserve for Future Use	8-20

G65 Macro Subprogram Call Option

G65 is the command that calls a subprogram with the ability to pass arguments to it. The format follows:

G65 Pnnnnn [Lnnnn] [arguments] ;

Arguments italicized in square brackets are optional. See the Programming section for more details on macro arguments.

The G65 command requires a P address corresponding to a program number currently located in the control's drive or path to a program. When the L address is used the macro call is repeated the specified number of times.

When a subprogram is called, the control looks for the subprogram on the active drive or the path to the program. If the subprogram cannot be located on the active drive, the control looks in the drive designated by Setting 251. Refer to the Setting Up Search Locations section for more information on subprogram searching. An alarm occurs if the control does not find the subprogram.

In Example 1, subprogram 1000 is called once without conditions passed to the subprogram. G65 calls are similar to, but not the same as, M98 calls. G65 calls can be nested up to 9 times, which means, program 1 can call program 2, program 2 can call program 3 and program 3 can call program 4.

Example 1:

G65 P1000 (Call subprogram O01000 as a macro) ;

M30 (Program stop) ;

O01000 (Macro Subprogram) ;

...

M99 (Return from Macro Subprogram) ;

In Example 2, the program LightHousing.nc is called using the path that it is in.

Example 2:

G65 P15 A1. B1.;

G65 (/Memory/LightHousing.nc) A1. B1.;

NOTE: Paths are case sensitive.

In Example 3, subprogram 9010 is designed to drill a sequence of holes along a line whose slope is determined by the X and Y arguments that are passed to it in the G65 command line. The Z drill depth is passed as Z, the feed rate is passed as F, and the number of holes to be drilled is passed as T. The line of holes is drilled starting from the current tool position when the macro subprogram is called.

Example 3:

NOTE: The subprogram program O09010 should reside on the active drive or on a drive designated by Setting 252.

G00 G90 X1.0 Y1.0 Z.05 S1000 M03 (Position tool) ;

G65 P9010 X.5 Y.25 Z.05 F10. T10 (Call O09010) ;

M30 ;

O09010 (Diagonal hole pattern) ;

F#9 (F=Feedrate) ;

WHILE [#20 GT 0] DO1 (Repeat T times) ;

G91 G81 Z#26 (Drill To Z depth) ;

#20=#20-1 (Decrement counter) ;

IF [#20 EQ 0] GOTO5 (All holes drilled) ;

G00 X#24 Y#25 (Move along slope) ;

N5 END1 ;

M99 (Return to caller) ;

Aliasing

Aliased codes are user defined G and M-codes that reference a macro program. There are 10 G alias codes and 10 M alias codes available to users. Program numbers 9010 through 9019 are reserved for G-code aliasing and 9000 through 9009 are reserved for M-code aliasing.

Aliasing is a means of assigning a G-code or M-code to a G65 P##### sequence. For instance, in the previous Example 2, it would be easier to write:

G06 X.5 Y.25 Z.05 F10. T10 ;

When aliasing, variables can be passed with a G-code; variables cannot be passed with an M-code.

Here, an unused G-code has been substituted, G06 for G65 P9010. In order for the previous block to work, the value associated with subprogram 9010 must be set to 06. Refer to the Setting Aliases section for how to setup aliases.

NOTE: G00, G65, G66, and G67 cannot be aliased. All other codes between 1 and 255 can be used for aliasing.

If a macro call subprogram is set to a G-code and the subprogram is not in memory, then an alarm is given. Refer to the G65 Macro Subprogram Call section on page 139 on how to locate the subprogram. An alarm occurs if the sub-program is not found.

If a macro call subprogram is set to a G-code and the subprogram is not in memory, then an alarm is given. Refer to the Macro Subprogram Call section on how to locate the subprogram. An alarm occurs if the sub-program is not found.

Icon Guide

Setup 	Setup mode is locked; the control is in Run mode. Most machine functions are disabled or limited while the machine doors are open.
Setup 	Setup mode is unlocked; the control is in Setup mode. Most machine functions are available, but may be limited while the machine doors are open.
Cycle Door 	The door must be cycled at least once to ensure that the door sensor is working. This icon appears after [POWER UP] If the user has not yet cycled the door.
Door Open 	Warning, door is open.
Pallet Load Door Open 	The pallet load station is open.
Light Curtain Breach 	This icon appears when the machine is idle and the light curtain is triggered. It also appears when a program is running and the light curtain is running. This icon disappears when the obstacle is removed from the light curtain line of sight.
Light Curtain Hold 	This icon appears when a program is running and the light curtain is triggered. This icon will clear the next time [CYCLE START] is pressed.
Running 	The machine is running a program.

Jog 	An axis is jogging at the current jog rate.
APL Mode 	This icon appears when the machine is in APL Mode.
Power Saving 	The power-saving servos-off feature is active. Setting 216, SERVO AND HYDRAULIC SHUTOFF, designates the time period allowed before this feature activates. Press a key to activate the servos.
Jog 	This icon appears while the control returns to the workpiece during a run-stop-jog-continue operation.
Jog 	You have pressed **[FEED HOLD]** during the return portion of a run-stop-jog-continue operation.
Jog 	This icon prompts you to jog away during a run-stop-jog-continue operation.
Feed Hold 	The machine is in feed hold. Axis motion has stopped, but the spindle continues to turn.
Feed 	The machine is executing a cutting move.

Icon Guide

Rapid 	The machine is executing a non-cutting axis move (G00) at the fastest possible rate. Overrides can affect the actual rate.
Dwell 	The machine is executing a dwell (G04) command.
Restart 	The control scans the program before a restart if Setting 36 is **ON**.
Singbk Stop 	**SINGLE BLOCK** mode is active, and the control needs a command to continue.
Door Hold 	Machine motion has stopped because of door rules.
Jog Lock 	Jog lock is active. If you press an axis key, that axis moves at the current jog rate until you press **[JOG LOCK]** again, or the axis reaches its limit.
Remote Jog 	The optional remote jog handle is active.
Vector Jog 	For five-axis machines, the tool will jog along the vector defined by the rotary positions.

Low Gearbox Oil Flow 	This icon appears when low gearbox oil flow persists for 1 minute.
Low Gearbox Oil 	The control detected a low gearbox oil level. **Note:** In software version 100.19.000.1100 and higher the control will monitor the gearbox oil level condition when spindle fan is turned **OFF**. After the spindle fan turns off, there is a delay before the gearbox oil level monitoring will begin. Press **[RESET]** to clear the low gearbox oil icon.
Rotary Lube 	Check and fill the rotary table lubrication oil reservoir.
Dirty TSC/HPFC Filter 	Clean the Through-Spindle Coolant or High-Pressure Flood Coolant filter.
Low Coolant Concentrate 	Fill the concentrate reservoir for the coolant refill system.
PulseJet Low Oil 	This icon appears when the system detects a low oil condition on the PulseJet oil reservoir.
Low Lube 	The spindle lubrication oil system detected a low oil condition, or the axis ball screw lubrication system detected a low grease or low pressure condition.

Icon Guide

Low Oil 	The rotary brake oil level is low.
Residual Pressure 	Before a lubrication cycle the system detected residual pressure from the grease pressure sensor. This can be caused by an obstruction in the axes grease lubrication system.
Mist Filter 	Clean the mist extractor filter.
Vise Clamp 	This icon appears when the vise is commanded to clamp.
Low Coolant (Warning) 	Coolant level is low.
Low PulseJet Oil 	The PulseJet oil level is low.
Mist Condenser 	This icon appears when the mist condenser is turned on.
Low Air Flow 	Inch Mode - Air flow is not sufficient for correct machine operation.

Low Air Flow 	Metric Mode - Air flow is not sufficient for correct machine operation.
Spindle 	When you press **[HANDLE SPINDLE]**, the jog handle varies the spindle override percentage.
Feed 	When you press **[HANDLE FEED]**, the jog handle varies the feed rate override percentage.
Handle Scroll 	When you press **[HANDLE SCROLL]**, the jog handle scrolls through the text.
Mirroring 	Mirroring mode is active. Either G101 is programmed or Setting 45, 46, 47, 48, 80, or 250 (mirror image of axis X, Y, Z, A, B, C) is set to **ON**
Brake 	A rotary axis brake, or a combination of rotary axis brakes, is unclamped.
Brake 	A rotary axis brake, or a combination of rotary axis brakes, is clamped

Icon Guide

HPU Oil Low 	The HPU oil level is low. Check the oil level and add the recommended oil for the machine.
HPU Oil Temperature (Warning) 	The oil temperature is too high to reliably operate the HPU.
Spindle Fan Failed 	This icon appears when the spindle fan stops operating.
Electronics Overheat (Warning) 	This icon appears when the control has detected cabinet temperatures are approaching levels that are potentially dangerous to the electronics. If the temperature reaches or exceeds this recommended level alarm **253 ELECTRONICS OVERHEAT** will be generated. Inspect the cabinet for clogged air filters and correctly operating fans.
Electronics Overheat (Alarm) 	This icon appears when the electronics remains in the overheat state for too long. The machine will not operate until the condition is corrected. Inspect the cabinet for clogged air filters and correctly operating fans.
Transformer Overheat (Warning) 	This icon appears when the transformer is detected to be overheated for more than 1 second.

Transformer Overheat (Alarm) 	This icon appears when the transformer remains in the overheat state for too long. The machine will not operate until the condition is corrected.
Low Voltage (Warning) 	The PFDM detects low incoming voltage. If the condition continues, the machine cannot continue to operate.
Low Voltage (Alarm) 	The Power Fault Detect Module (PDFM) detects incoming voltage that is too low to operate. The machine will not operate until the condition is corrected.
High Voltage (Warning) 	The PFDM detects incoming voltage above a set limit, but still within operating parameters. Correct the condition to prevent damage to machine components.
High Voltage (Alarm) 	The PFDM detects incoming voltage that is too high to operate and could cause damage to the machine. The machine will not operate until the condition is corrected.
Surge Protector Fault Detected 	Indicates a Surge Protector Fault has been detected. This icon is active until the fault has been cleared. **Warning:** If continuing to use the machine in this state. The electronics are capable of being damaged due to any electrical surge.
Robot Battery is Low 	Robot Battery is Low. Please replace the pulse coder batteries as soon as possible. Do **NOT** turn off the robot, otherwise it may require remastering. Reference 9156.062 ROBOT COMMAND FAILED SRVO-062 BZAL alarm in service documentation for more information.

Icon Guide

Low Air (Warning) 	The air pressure to the machine is too low to reliably operate pneumatic systems. Correct this condition to prevent damage to or incorrect operation of pneumatic systems.
Low Air (Alarm) 	The air pressure to the machine is too low to operate pneumatic systems. The machine will not operate until the condition is corrected. You may need a higher-capacity air compressor.
High Air (Warning) 	The air pressure to the machine is too high to reliably operate pneumatic systems. Correct this condition to prevent damage to or incorrect operation of pneumatic systems. You may need to install a regulator at the machine's air input.
High Air (Alarm) 	The air pressure to the machine is too high to operate pneumatic systems. The machine will not operate until the condition is corrected. You may need to install a regulator at the machine's air input.
Pendant E-Stop 	**[EMERGENCY STOP]** on the pendant has been pressed. This icon disappears when **[EMERGENCY STOP]** is released.
APC E-Stop 	**[EMERGENCY STOP]** on the pallet changer has been pressed. This icon disappears when **[EMERGENCY STOP]** is released.
Tool Changer E-Stop 	**[EMERGENCY STOP]** on the tool changer cage has been pressed. This icon disappears when **[EMERGENCY STOP]** is released.

Auxiliary E-Stop 	**[EMERGENCY STOP]** on an auxiliary device has been pressed. This icon disappears when **[EMERGENCY STOP]** is released.
Remote Jog Handle-XL (RJH-XL) E-Stop 	**[EMERGENCY STOP]** on the RJH-XL has been pressed. This icon disappears when **[EMERGENCY STOP]** is released.
Single Blk 	**SINGLE BLOCK** mode is active. The control executes programs (1) block at a time. Press **[CYCLE START]** to execute the next block.
Tool Life (Warning) 	The tool life remaining is below Setting 240, or the current tool is the last one in its tool group.
Tool Life (Alarm) 	The tool or tool group has expired, and no replacement tools are available.
Opt Stop 	**OPTIONAL STOP** is active. The control stops the program at each M01 command.
Blk Delete 	**BLOCK DELETE** is active. When Block Delete is On, the control ignores (does not execute) the code following a Forward Slash (/), on that same line.

Icon Guide

TC Door Open 	The side-mount tool changer door is open.
TC Manual Mode 	This icon appears when the tool carousel is in manual mode via the auto/manual switch. This switch is only on machines with tool cages.
TL CCW 	The side-mount tool changer carousel is rotating counter-clockwise.
TL CW 	The side-mount tool changer carousel is rotating clockwise.
Tool Change 	A tool change is in progress.
Tool Unclamped 	The tool in the spindle is unclamped.
Probe 	The probe system is active.
Conveyor Fwd 	The conveyor is active and moving forward.

Conveyor Rev 	The conveyor is active and moving in reverse.
TSC 	The Through-Spindle Coolant (TSC) system is active.
TAB 	The Tool Air Blast (TAB) system is active.
Air Blast 	The Auto Air Gun (AAG) is active.
HIL Light 	Indicates the optional High Intensity (HIL) are turned ON and the doors are open. Duration is determined by Setting 238.
Coolant 	The main coolant system is active.

RJH-Touch XL Overview

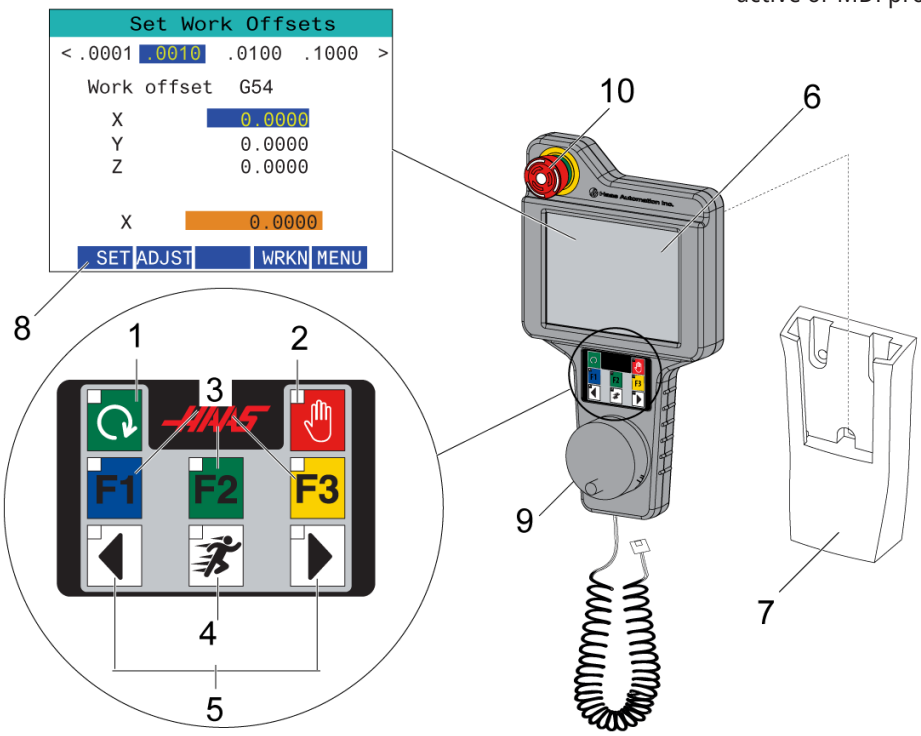
The Remote Jog Handle (RJH-Touch XL) is an optional accessory that gives you hand-held access to the control for faster and easier setups.

Your machine must have Next Generation Control software 100.21.000.1000 or higher to use all the RJH-Touch XL functions. The next sections explain how to operate the RJH-Touch.

The illustration shows these components:

1. Cycle Start. Has the same function as **[CYCLE START]** on the pendant.
2. Feed Hold. Has the same function as **[FEED HOLD]** on the control pendant.
3. Function Keys. These keys are for future use.
4. Rapid jog button. This key doubles the jogging speed when pressed simultaneously with one of the jog direction buttons.
5. Jog Direction Keys. These keys work the same as keypad jog arrow keys. You can press and hold to jog the axis.
6. LCD Touch Screen Display.
7. Holster. To activate the RJH-XL, lift it out of the holster. To deactivate the RJH-XL, put it back into the holster.
8. Function Tabs. These tabs have different functions in different modes. Press the function tab that corresponds to the function you want to use.
9. Handle Jog Wheel. This Handle Jog works like the jog handle on the control pendant. Each click of the handle jog moves the selected axis one unit of the selected jog rate.
10. E-Stop. Has the same function as **[EMERGENCY STOP]** on the pendant.

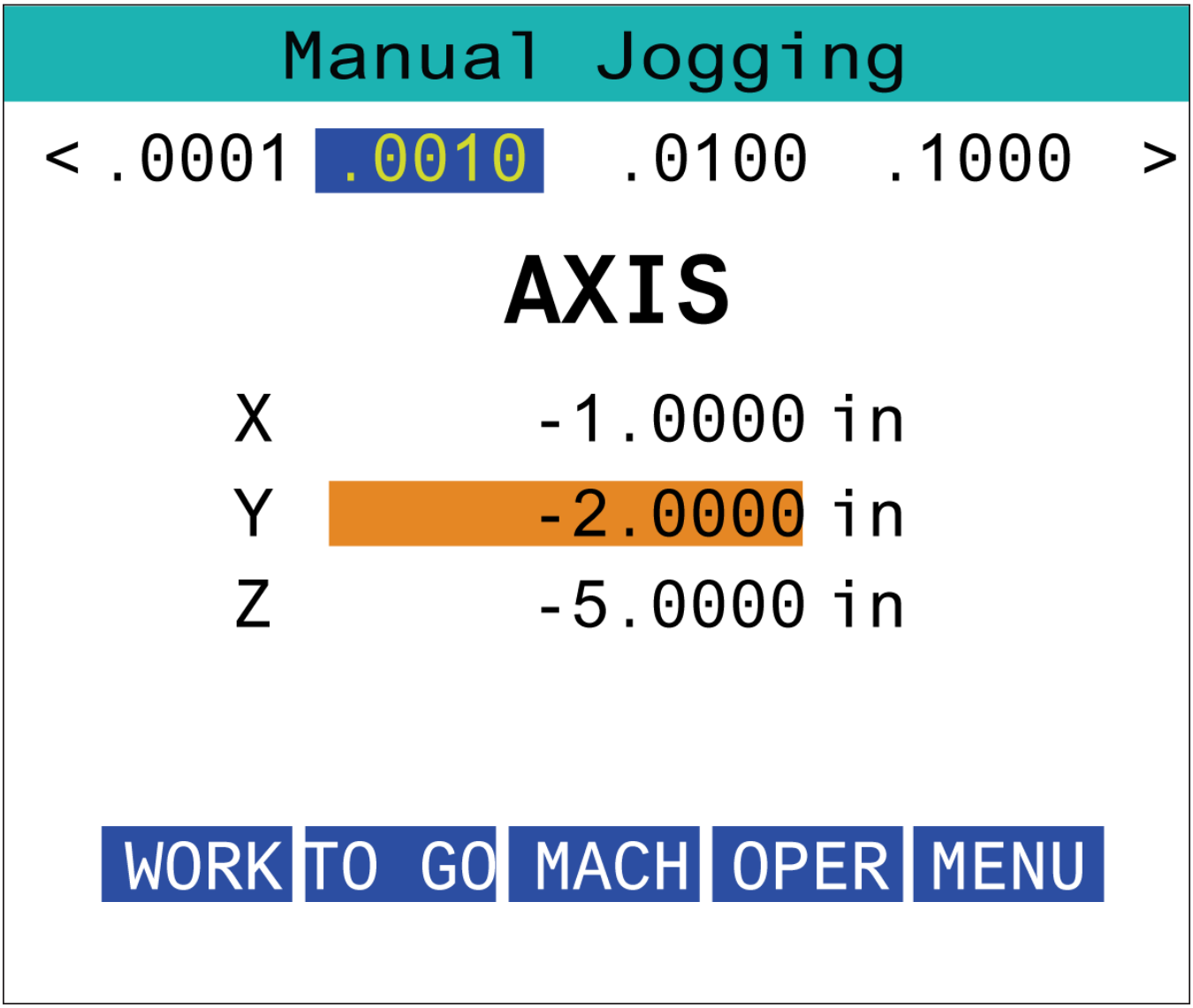
Most RJH functions are available in Handle Jog mode. In other modes, the RJH screen displays information about the active or MDI program.



NOTE: The RJH-XL cannot be removed when the machine is on.

RJH-Touch XL Manual Jogging

1. Press **[MENU]** on the screen.
2. Press **Manual Jogging** on the screen.
3. Press the **.0001, .0010, .0100, .1000** on the screen to change the jog rate.
4. Press the axis position on the screen or press **[F1]/[F3]** on the RJH-XL to change the axis.



RJH-XL - Tool Offsets

To access this function on the RJH-XL, press **[OFFSET]** on the control pendant and select the Tool Offsets page, or select **TOOL OFFSETS** from the RJH-XL operation mode menu.

Press the **.001, .0010, .0100, or .1000** on the screen to change the jog rate.

Press the axis position on the screen or press **[F1]/[F3]** on the RJH-XL to change the axis.

Press the **[NEXT]** on the screen to change to the next tool.

To change the tool offset, highlight the **TOOL OFFSET** field and use the handle to change the value.

Use the jog handle to jog the tool to the desired position. Press the **[SETL]** function key to record the tool length.

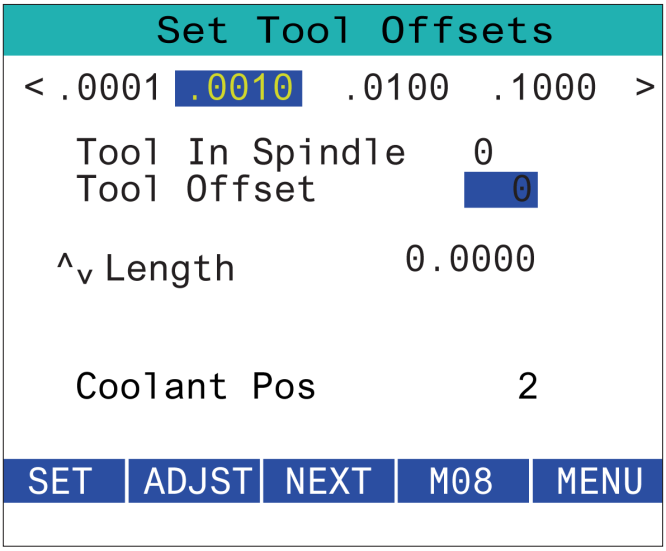
To adjust the tool length, for example, if you want to subtract from the tool length the thickness of the paper you used to touch off the tool:

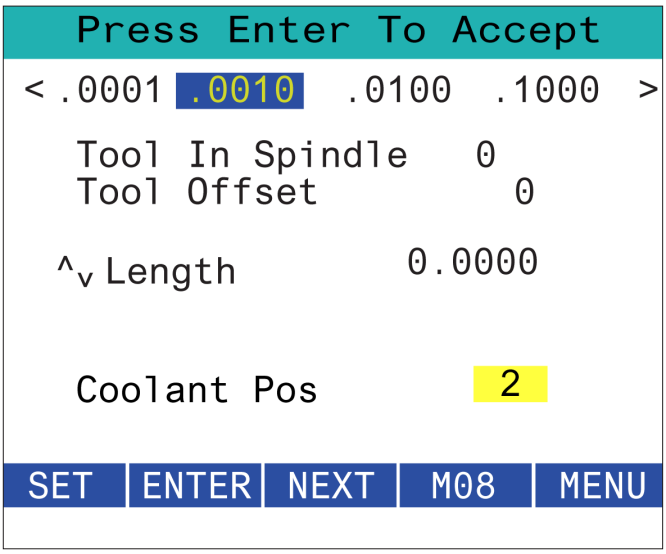
1. Press the **[ADJUST]** button on the screen.
2. Use the handle jog to change the value (positive or negative) to add to the tool length.
3. Press the **[ENTER]** button on the screen.

NOTE: If your machine has the programmable Coolant option, you can adjust the spigot position for the tool by:

1. Highlight the **COOLANT POS** field.
2. Press the **[ADJST]** button on the screen and use the handle jog to change the value.
3. Press the **[ENTER]** button on the screen to accept the coolant position change.

Press the **[M08]** button on the screen to turn on the coolant and test the spigot position. Press the button on the screen again to turn off the coolant.





RJH-XL - Work Offsets

To access this function on the RJH-XL, press **[OFFSET]** on the control pendant and select the Work Offsets page, or select WORK OFFSETS from the RJH-XL operation mode menu.

Press the **.0001, .0010, .0100, or .1000** on the screen to change the jog rate.

Press the axis position on the screen or press **[F1]/[F3]** on the RJH-XL to change the axis.

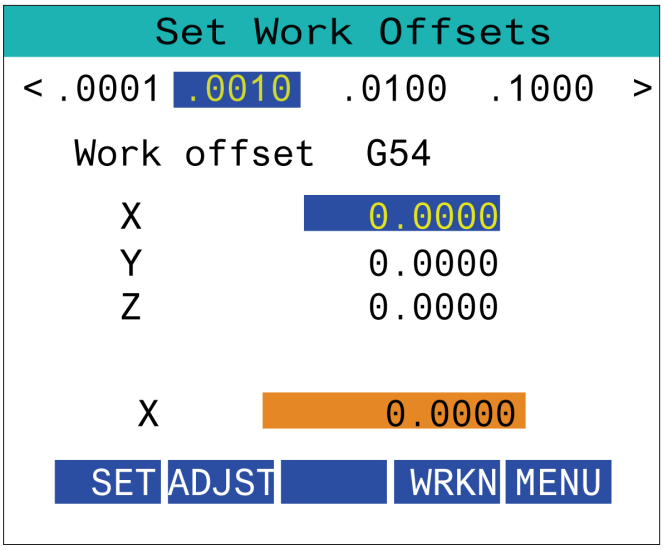
To change the work offset number, press the **[WORKN]** button on the screen and use the handle jog knob to select a new offset number.

Press the **[ENTER]** button on the screen to set the new offset. To move the axes use the handle jog wheel.

When you reach the offset position in an axis, press the **[SET]** button on the screen to record the offset position.

To adjust an offset value:

1. Press the **[ADJUST]** function key.
2. Use the pulse knob to change the value (positive or negative) to add to the offset.
3. Press the **[ENTER]** function key.



RJH-XL - Work Offsets

To access this function on the RJH-XL, press **[OFFSET]** on the control pendant and select the Work Offsets page, or select WORK OFFSETS from the RJH-XL operation mode menu.

Press the **.0001, .0010, .0100, or .1000** on the screen to change the jog rate.

Press the axis position on the screen or press **[F1]/[F3]** on the RJH-XL to change the axis.

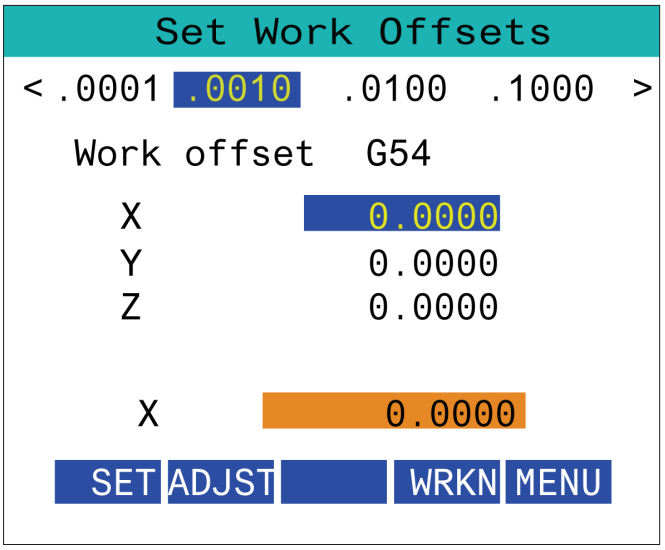
To change the work offset number, press the **[WORKN]** button on the screen and use the handle jog knob to select a new offset number.

Press the **[ENTER]** button on the screen to set the new offset. To move the axes use the handle jog wheel.

When you reach the offset position in an axis, press the **[SET]** button on the screen to record the offset position.

To adjust an offset value:

1. Press the **[ADJUST]** function key.
2. Use the pulse knob to change the value (positive or negative) to add to the offset.
3. Press the **[ENTER]** function key.

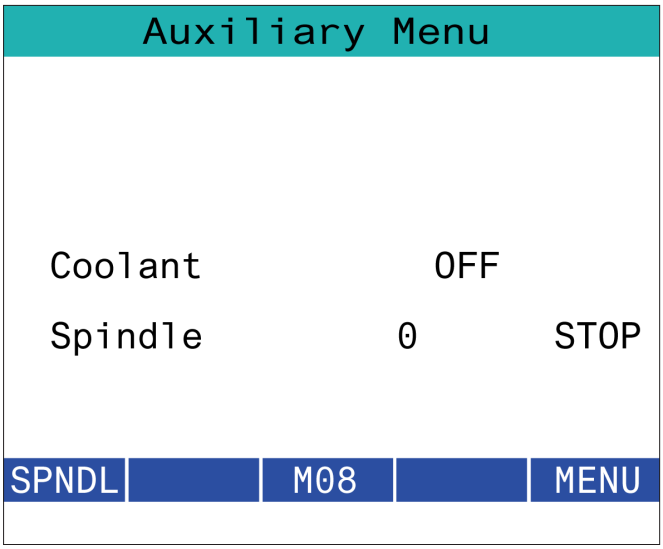


RJH-XL - Auxiliary Menu

To access this function on the RJH-XL, select **AUXILIARY** MENU from the RJH-XL operation mode menu.

The **[SPNDL]** button on the screen will turn the spindle clockwise and counter-clockwise.

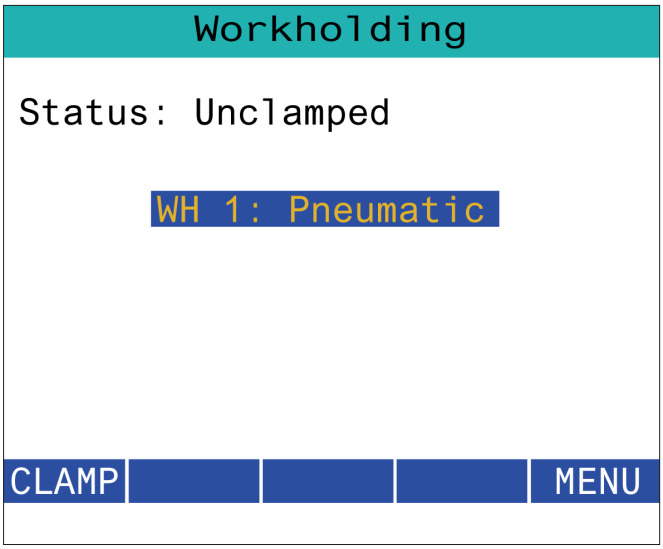
The **[M08]** button on the screen can control the coolant.



RJH-XL - Workholding

To access this function on the RJH-XL, press the **[CURRENT COMMANDS]** button on the control pendant then under the devices tab navigate to the working tab, or select **WORKHOLDING** from the RJH-XL operation mode menu.

Press the **[CLAMP]/[UNCLA]** buttons on the screen to clamp/unclamp the selected vise.



RJH-XL - Program Mode

NOTE: The images show MDI, but the following instructions apply to both MDI and MEM.

When the MDI or MEM is pressed on the pendant there are 4 main tabs [1] on the RJH: **WORK**, **TO GO**, **MACH**, and **OPER**.

When **[WORK]** is highlighted, the screen shows the axis positions in relation to part zero.

When **[TO GO]** is highlighted, the screen shows the distance remaining before the axes reach their commanded position.

When **[MACH]** is highlighted, the screen displays the axes positions in relation to machine zero.

When **[OPER]** is highlighted, the screen displays the distance the axes have been jogged.

At the bottom of the screen there are 5 buttons [2]: **SINGL**, OPSTP, **BLK D**, **M08**, **MENU**.

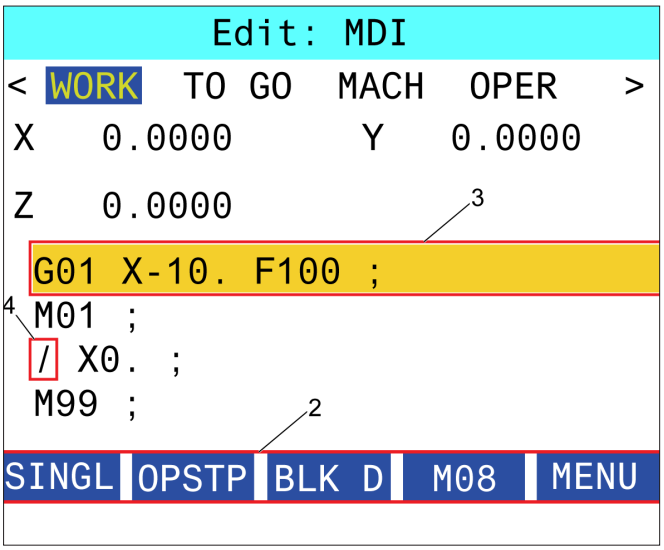
When **SINGL** is pressed it will execute the highlighted line [3] and stop and when **[CYCLE START]** is pressed it'll execute the next line and stop, and so on.

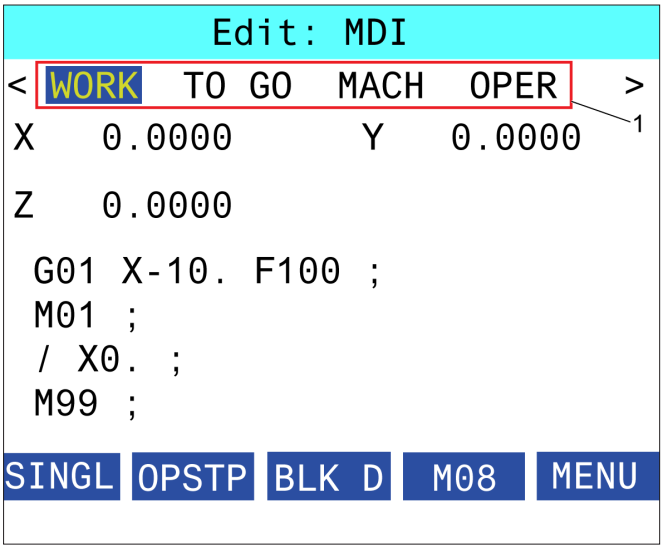
OPSTP is optional stop, when this is pressed the program will stop at every M01 encountered.

NOTE: On machines with an autodoor, **OPSTP** will stop at every M01 and open the door(s).

BLK D is block delete, when this is pressed any line beginning with a forward slash [4] will be skipped when the program is ran.

When **M08** is pressed, the coolant turns on and the button will then read **M09** which will turn the coolant off when pressed.





Introduction

In addition to the standard functions included with your machine, you may also have optional equipment with special programming considerations. This section tells you how to program these options.

You can contact your HFO to purchase most of these options, if your machine did not come equipped with them.

Feature List

The Feature List contains both standard and purchasable options.

To access the list:

Press **[DIAGNOSTIC].**

Navigate to the Parameters and then the Features tab. (Purchased options are marked in green and their status is set as PURCHASED.)

Purchased Options Enable/Disable

To enable or disable a purchased option:

Highlight the option on the **FEATURES** tab.

Press **[ENTER]** to turn ON/OFF the option.

NOTE: If the featured option is turned OFF, the option is not available.

Option Tryout

Some options have a 200-hour tryout available. The FEATURES tab Status column shows the options available for tryout.

NOTE: If an option does not have a tryout, the Status column shows FEATURE DISABLED, and you must purchase the option to use it.

To start tryout:

Highlight the feature.

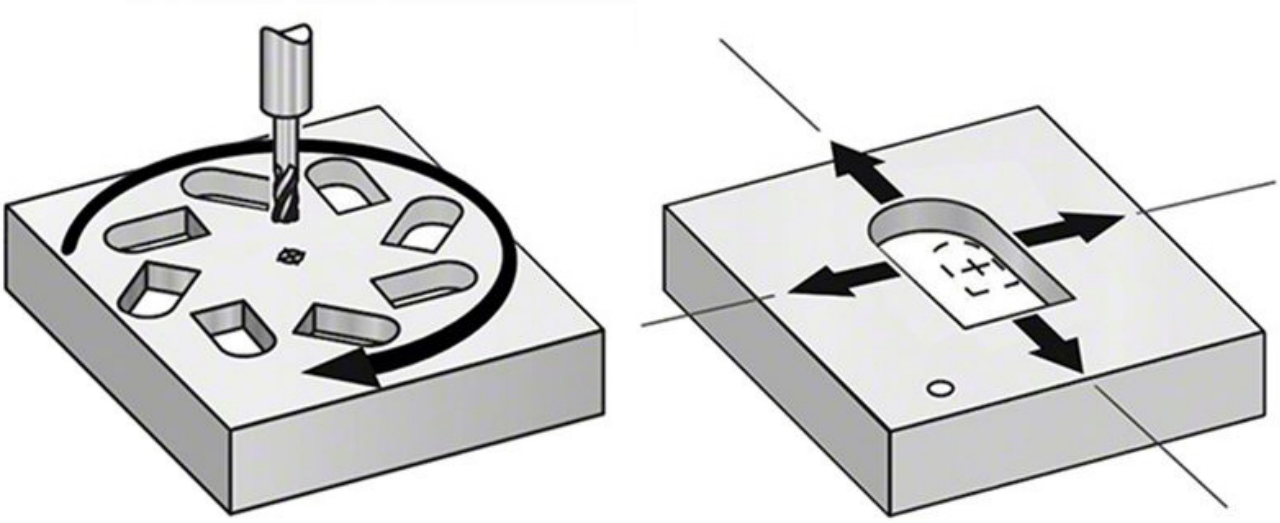
Press **[ENTER]**. Press **[ENTER]** again to disable the option and stop the timer.

The status of the feature changes to TRYOUT ENABLED, and the date column shows the remaining hours left in the tryout period. When the tryout period expires, the status changes to EXPIRED. You cannot extend the tryout time for expired options. You must purchase them to use them.

NOTE: Tryout time is only updated while the option is enabled.

Rotation and Scaling

Rotation lets you rotate a pattern to another location or around a circumference. Scaling reduces or enlarges a tool path or pattern.



Visual Programming System (VPS)

VPS lets you quickly build programs from program templates. To access VPS, press EDIT and then select the VPS tab.

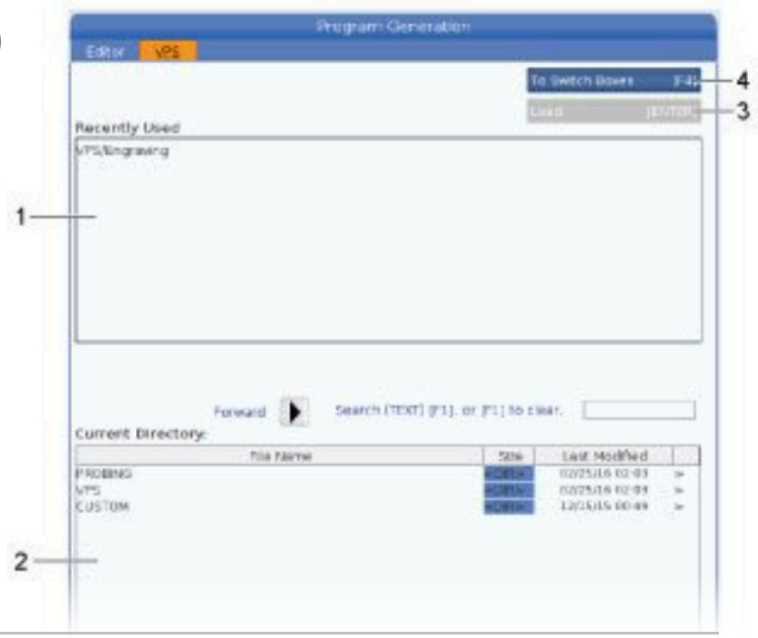
VPS Starting Screen.

[1] Recently Used Templates,

[2] Template Directory Window,

[3] ENTER to Load a Template,

[4] F4 to Switch Between Recently Used and Template Directory.



VPS Engraving Program Generation Window Example.

[1] Variable Illustration,

[2] Variables Table,

[3] Variable Description Text,

[4] Template Illustration,

[5] Generate G-code F4,

[6] Run in MDI CYCLE START,

[7] Clear ORIGIN,

[8] Default value was changed indicator.

VPS Example

When you use VPS, you choose a template for the function you want to program, and then enter variables to create a program. The default templates include probing and part features. You can also create custom templates. Contact the Applications department at your HFO for help with custom templates.

1. Press EDIT, and then select the VPS tab.
2. Use the cursor arrow keys to highlight the VPS menu option. Press the RIGHT cursor arrow key to select the option.
3. Highlight and select the Engraving option from the next menu.
4. In the Program Generation window, use the UP and DOWN cursor arrow keys to highlight the variable rows.
5. Type a value for the highlighted variable and press ENTER. The control will display an asterisk (*) next to variable if the default value is changed. To set the variable back to default press the ORIGIN button. Press the DOWN cursor arrow key to move to the next variable.

16.3 | MILL - VISUAL PROGRAMMING SYSTEM (VPS)

To generate the example engraving cycle, we use these variable values. Note that all of the position values are given in work coordinates.

NAME	DESCRIPTION	VALUE
WORK_OFFSETS	Work Offset Number	54
T	Tool Number	1
S	Spindle Speed	1000
F	Feedrate	15
M8	Coolant (1 - YES / 0 - NO)	1
X	Starting X Position	2
Y	Starting Y Position	2
R	R-Plane Height	0.05
Z	Z Depth	-0.005
P	Text or Serial Number Switch (0 - Text, 1 - Serial Number)	0
J	Text Height	0.5
I	Text Angle (Degrees from Horizontal)	45
TEXT	Text to Engrave	TEXT TO ENGRAVE

6. With all of the variables entered, you can press **[CYCLE START]** to immediately run the program in MDI, or F4 to output the code to either the clipboard or MDI without running the program.

```
%
O11111 ;
(Engraving) ;
( TOOL 1 ) ;
( SPINDLE 1000 RPM / FEED 15. ) ;
( DEPTH -0.005 ) ;
T1 M06 ;
G00 G90 G54 X2. Y2. S1000 M03 ;
G43 Z0.05 H1 ;
M08 ;
G00 G90 G54 X2. Y2. ;
( TEXT ENGRAVING : TEXT TO
ENGRAVE ) ;
G47 E7.5000 F15. I45. J.5 P0 R0.05
Z-0.005 (TEXT TO ENGRAVE) ;
G0 Z0.05 M09 ;
M05 ;
G91 G28 Z0. ;
G91 G28 Y0. ;
M01 ( END ENGRAVING ) ;
%
```

16.4 | MILL - RIGID TAPPING

Rigid Tapping

This option synchronizes the spindle's RPM with the feedrate during a tapping operation.

16.5 | MILL - M19 SPINDLE ORIENTATION

M19 Spindle Orientation

The Spindle Orientation lets you position the spindle to a programmed angle. This option provides inexpensive, accurate positioning.

16.6 | MILL - HIGH-SPEED MACHINING

High-Speed Machining

The Haas high-speed machining option allows faster feedrates and more complex tool paths. HSM uses a motion algorithm called Acceleration Before Interpolation combined with full look-ahead to provide contouring feeds up to 1200 ipm (30.5 m/min) without risk of distortion to the programmed path. This reduces cycle times, improves accuracy, and smooths motion.

16.7 | MILL - ADDITIONAL MEMORY OPTION

Additional Memory Option

The Spindle Orientation lets you position the spindle to a programmed angle. This option provides inexpensive, accurate positioning.

Probing

You can use an optional probe system to set offsets, check work, measure tools, and check tools. This section describes basic probe use and troubleshooting.

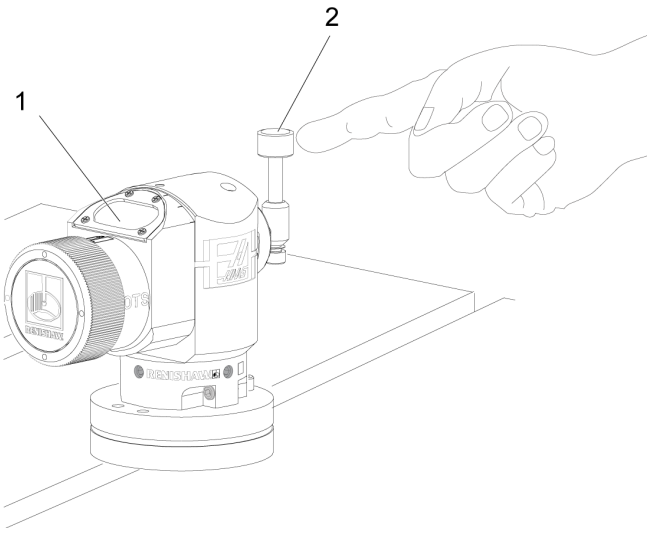
Check Tool Probe

Do these steps to make sure the tool probe operates correctly:

1. In MDI mode, run:
M59 P2 ;
G04 P1.0 ;
M59 P3 ;
This turns on tool probe communication, delays one second, and turns on the tool probe. The LED [1] on the tool probe flashes green.

2. Touch the stylus [2].
The machine makes a “beep” sound and the LED becomes red [1]. This tells you that the tool probe is started.

3. Press RESET to deactivate the probe.
The probe LED [1] turns off.



Check Work Probe

Do these steps to make sure the work probe operates correctly:

1 Select the work probe with a tool change, or manually insert the work probe into the spindle.

2 In MDI mode, run M69 P2 ;
This starts communication with the work probe.

3 in MDI mode: run M59 P3 ;

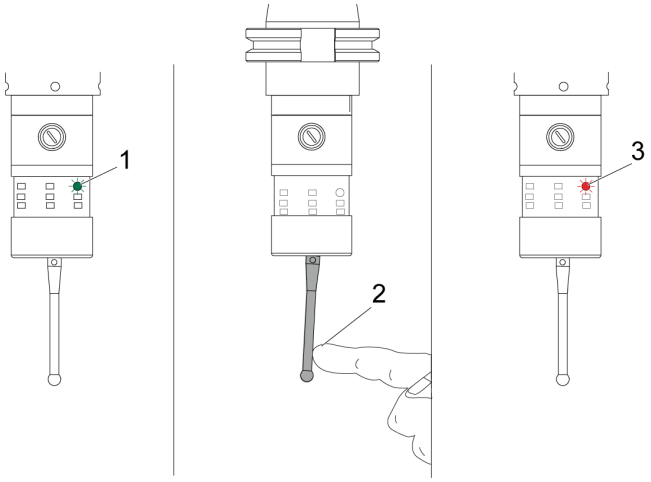
The probe LED flashes green [1].

4 Touch the stylus [2].

The machine makes a “beep” sound and the LED becomes red [3]. This tells you that the work probe is started.

5 Press RESET to deactivate the probe.

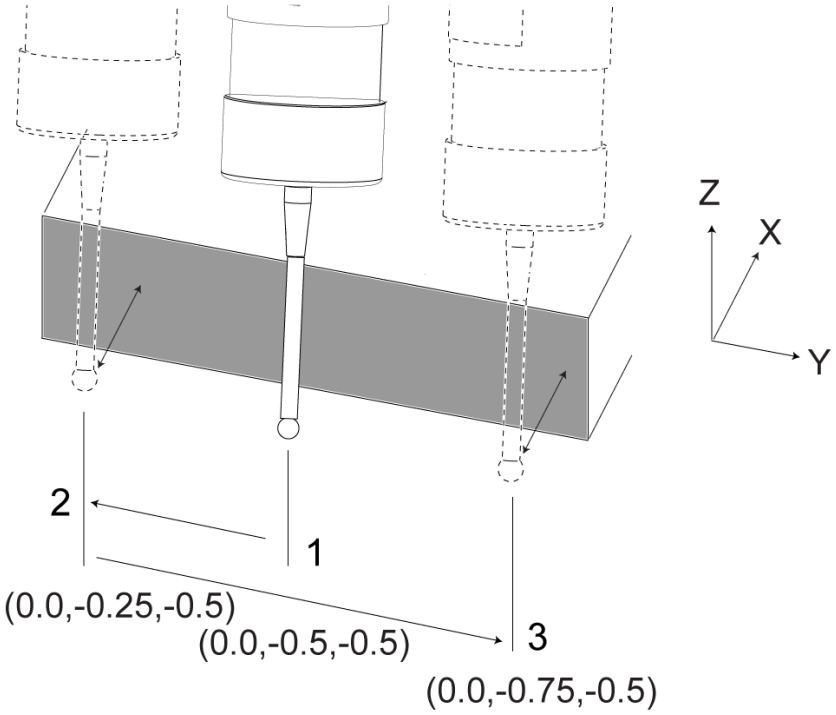
The work probe LED turns off [1].



Probe Example

You can use a probe to check your part for correct dimensions during the machining process. For example, this program uses the work probe to check for squareness. The program uses G65 to call 9XXXXX macro programs created specifically for probing. You can find more information on these programs in the Renishaw manuals on the web at www.haascnc.com then click on the Service tab.

The program does the following:



1 After a tool change, home, and adding tool length compensation, the system turns on the work probe and moves to a safe start location.

2 The probe stylus moves adjacent to the surface at the required Z-Axis point to provide a central start position [1].

3 The cycle makes two measurements, symmetrically about the start position, to establish the surface angle [2], [3].

4 Finally, the probe stylus moves to its safe out position, tuns off the probe, and returns home.

Example:

```
%
O00010 (CHECK FOR SQUARE) ;
T20 M06 (PROBE) ;
G00 G90 G54 X0. Y0. ;
G43 H20 Z6. ;
G65 P9832 (WORK PROBE ON) ;
G65 P9810 Z-0.5 F100. (SAFE MOVE) ;
G65 P9843 Y-0.5 D0.5 A15. (ANGLE MEAS.) ;
G65 P9810 Z6. F100. (SAFE OUT) ;
G65 P9833 (WORK PROBE OFF) ;
G00 G90 G53 Z0. ;
M01 ;
;
;
( PART PROGRAM ) ;
G00 G90 G54 X0. Y0. ;
T2 M06 (1/2" END MILL) ;
G00 G90 G43 H02 Z1.5 ;
G68 R#189 ;
G01 X-2. F50. ;
M30 ;

%
```

16.8 | MILL - PROBING

Probe Use with Macros

Macro statements select and turn on and off the probe the same as M-codes.

M-CODE	SYSTEM VARIABLE	MACRO VALUE	PROBE
M59 P2 ;	#12002	1	Tool Probe Selected
M69 P2 ;	w#12002	0	Work Probe Selected
M59 P3 ;	#12003	1	Probe Enable
M69 P3 ;	#12003	0	Probe Disable

If you assign the system variable to a viewable global variable, you can see the macro value change in the Macro Vars tab under CURRENT COMMANDS.

For example,

M59 P3 ;

#10003=#12003 ;

The global variable #10003 shows the output from M59 P3 ; as 1.000000. This means that either the tool probe or work probe is on.

16.9 | MILL - MAXIMUM SPINDLE SPEED

Maximum Spindle Speed

This option increases the maximum speed at which you can run the machine spindle.

16.10 | MILL - COMPENSATION TABLES

Compensation Tables

With this option, the control stores a compensation table to correct for small errors in the rotary worm gear, as well as small errors in X, Y, and Z.

Mill G-Codes Introduction

This page gives detailed descriptions of the G-codes that you use to program your Mill machine.

CAUTION: The sample programs in this manual have been tested for accuracy, but they are for illustrative purposes only. The programs do not define tools, offsets, or materials. They do not describe workholding or other fixturing. If you choose to run a sample program on your machine, do so in Graphics mode. Always follow safe machining practices when you run an unfamiliar program.

NOTE: The sample programs in this manual represent a very conservative programming style. The samples are intended to demonstrate safe and reliable programs, and they are not necessarily the fastest or most efficient way to operate a machine. The sample programs use G-codes that you might choose not to use in more efficient programs.

SCAN QR CODE FOR WHAT ARE G-CODES?

CODE	DESCRIPTION	GROUP
G00	Rapid Motion Positioning	01
G01	Linear Interpolation Motion	01
G02	Circular Interpolation Motion CW	01
G03	Circular Interpolation Motion CCW	01
G04	Dwell	00
G09	Exact Stop	00
G10	Set Offsets	00
G12	Circular Pocket Milling CW	00
G13	Circular Pocket Milling CCW	00
G17	XY Plane Selection	02
G18	XZ Plane Selection	02
G19	YZ Plane Selection	02
G20	Select Inches	06
G21	Select Metric	06
G28	Return To Machine Zero Point	00

CODE	DESCRIPTION	GROUP
G29	Return From Reference Point	00
G31	Feed Until Skip	00
G35	Automatic Tool Diameter Measurement	00
G36	Automatic Work Offset Measurement	00
G37	Automatic Tool Offset Measurement	00
G40	Cutter Compensation Cancel	07
G41	2D Cutter Compensation Left	07
G42	2D Cutter Compensation Right	07
G43	Tool Length Compensation + (Add)	08
G44	Tool Length Compensation - (Subtract)	08
G47	Text Engraving	00
G49	G43/G44/G143 Cancel	08
G50	Cancel Scaling	11
G51	Scaling	11
G52	Set Work Coordinate System	00 or 12

17.1 | MILL G-CODES INTRODUCTION

CODE	DESCRIPTION	GROUP
G53	Non-Modal Machine Coordinate Selection	00
G54	Select Work Coordinate System #1	12
G55	Select Work Coordinate System #2	12
G56	Select Work Coordinate System #3	12
G57	Select Work Coordinate System #4	12
G58	Select Work Coordinate System #5	12
G59	Select Work Coordinate System #6	12
G60	Uni-Directional Positioning	00
G61	Exact Stop Mode	15
G64	G61 Cancel	15
G65	Macro Subprogram Call Option	00
G68	Rotation	16
G69	Cancel G68 Rotation	16
G70	Bolt Hole Circle	00
G71	Bolt Hole Arc	00
G72	Bolt Holes Along an Angle	00
G73	High-Speed Peck Drilling Canned Cycle	09
G74	Reverse Tap Canned Cycle	09
G76	Fine Boring Canned Cycle	09
G77	Back Bore Canned Cycle	09
G80	Canned Cycle Cancel	09
G81	Drill Canned Cycle	09

CODE	DESCRIPTION	GROUP
G82	Spot Drill Canned Cycle	09
G83	Normal Peck Drilling Canned Cycle	09
G84	Tapping Canned Cycle	09
G85	Boring Canned Cycle	09
G86	Bore and Stop Canned Cycle	09
G89	Bore In, Dwell, Bore Out Canned Cycle	09
G90	Absolute Position Command	03
G91	Incremental Position Command	03
G92	Set Work Coordinate Systems Shift Value	00
G93	Inverse Time Feed Mode	05
G94	Feed Per Minute Mode	05
G95	Feed per Revolution	05
G98	Canned Cycle Initial Point Return	10
G99	Canned Cycle R Plane Return	10
G100	Cancel Mirror Image	00
G101	Enable Mirror Image	00
G103	Limit Block Buffering	00
G107	Cylindrical Mapping	00
G110	#7 Coordinate System	12
G111	#8 Coordinate System	12
G112	#9 Coordinate System	12
G113	#10 Coordinate System	12

17.1 | MILL G-CODES INTRODUCTION

CODE	DESCRIPTION	GROUP
G114	#11 Coordinate System	12
G115	#12 Coordinate System	12
G116	#13 Coordinate System	12
G117	#14 Coordinate System	12
G118	#15 Coordinate System	12
G119	#16 Coordinate System	12
G120	#17 Coordinate System	12
G121	#18 Coordinate System	12
G122	#19 Coordinate System	12
G123	#20 Coordinate System	12
G124	#21 Coordinate System	12
G125	#22 Coordinate System	12
G126	#23 Coordinate System	12
G127	#24 Coordinate System	12
G128	#25 Coordinate System	12
G129	#26 Coordinate System	12
G136	Automatic Work Offset Center Measurement	00
G141	3D+ Cutter Compensation	07
G143	5-Axis Tool Length Compensation +	08
G150	General Purpose Pocket Milling	00
G154	Select Work Coordinates P1-P99	12
G156	Broaching Canned Cycle	09

CODE	DESCRIPTION	GROUP
G167	Modify Setting	00
G174	CCW Non-Vertical Rigid Tap	00
G184	CW Non-Vertical Rigid Tap	00
G187	Setting the Smoothness Level	00
G234	Tool Center Point Control (TCPC)	08
G253	G253 Orient Spindle Normal To Feature Coordinate System	00
G254	Dynamic Work Offset (DWO)	23
G255	Cancel Dynamic Work Offset (DWO)	23
G266	Visible Axes Linear Rapid % Motion	00
G268	Enable Feature Coordinate System	02
G269	Disable Feature Coordinate System	02

Mill G-Codes Introduction

The main cutting G-codes are categorized into interpolation motion and canned cycles. Interpolation motion cutting codes are broken down into:

G01 - Linear Interpolation Motion

G02 - Clockwise Circular Interpolation Motion

G03 - Counter-Clockwise Circular Interpolation Motion

G12 - Clockwise Circular Pocket Milling

G13 - Counter-Clockwise Circular Pocket Milling

Circular Interpolation Motion

G02 and G03 are the G-codes for circular cutting motions. Circular Interpolation Motion has several optional address codes to define the arc or circle. The arc or circle begins cutting from the current cutter position [1] to the geometry specified within the G02/ G03 command.

Arcs can be defined using two different methods. The preferred method is to define the center of the arc or circle with I, J and/or K and to define the end point [3] of the arc with an X, Y and/or Z. The I J K values define the relative X Y Z distances from the starting point [2] to the center of the circle. The X Y Z values define the absolute X Y Z distances from the starting point to the end point of the arc within the current coordinate system. This is also the only method to cut a circle. Defining only the I J K values and not defining the end point X Y Z values will cut a circle.

The other method to cut an arc is to define the X Y Z values for the end point and to define the radius of the circle with an R value.

Below are examples of using the two different methods to cut a 2" (or 2 mm) radius, 180 degree, counter-clockwise arc. The tool starts at X0 Y0 [1], moves to the starting point of the arc [2], and cuts the arc to the end point [3]:

Linear Interpolation Motion

G01 Linear Interpolation Motion is used to cut straight lines. It requires a feedrate, specified with the Fnnn.nnnn address code. Xnn.nnnn, Ynn.nnnn, Znn.nnnn, and Annn.nnn are optional address codes to specify cut. Subsequent axis motion commands will use the feed rate specified by G01 until another axis motion, G00, G02, G03, G12, or G13 is commanded.

Corners can be chamfered using the optional argument Cnn.nnnn to define the chamfer. Corners can be rounded using the optional address code Rnn.nnnn to define the radius of the arc. Refer to G01 Linear Interpolation Motion (Group 01) for more information.

1

Method 1:

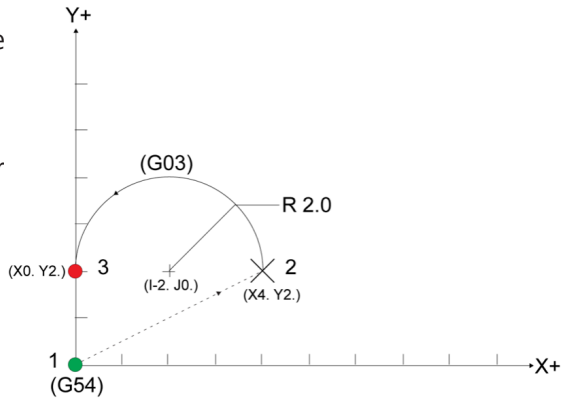
```
%
T01 M06 ;
...
G00 X4. Y2. ;
G01 F20.0 Z-0.1 ;
G03 F20.0 I-2.0 J0. X0.
Y2. ;
...
M30 ;
%
```

Method 2:

```
%
T01 M06 ;
...
G00 X4. Y2. ;
G01 F20.0 Z-0.1 ;
G03 F20.0 X0. Y2. R2. ;
...M30 ;
%
```

Below is an example of how to cut a 2" (or 2 mm) radius circle:

```
%
T01 M06 ;
...
G00 X4. Y2. ;
G01 F20.0 Z-0.1 ;
G02 F20.0 I2.0 J0. ;
...
M30 ;
%
```



Cutter Compensation

Cutter compensation is a method of shifting the tool path so that the actual centerline of the tool moves to either the left or right of the programmed path.

Normally, cutter compensation is programmed to shift the tool in order to control feature size. The offset display is used to enter the amount that the tool is to be shifted.

The offset can be entered as either a diameter or radius value, depending on Setting 40, for both the geometry and wear values. If diameter is specified, the shift amount is half of the value entered.

The effective offset values are the sum of the geometry and wear values. Cutter compensation is only available in the X Axis and the Y Axis for 2D machining (G17). For 3D machining, cutter compensation is available in the X Axis, Y Axis, and Z Axis (G141).

General Description of Cutter Compensation

G41 selects cutter compensation left. This means that the control moves the tool to the left of the programmed path (with respect to the direction of travel) to compensate for the tool radius or diameter defined in the tool offsets table (Refer to Setting 40). G42 selects cutter compensation right, which moves the tool to the right of the programmed path, with respect to the direction of travel.

A G41 or G42 command must have a Dnnn value to select the correct offset number from the radius / diameter offset column. The number to use with D is in the far-left column of the tool offsets table. The value that the control uses for cutter compensation is in the GEOMETRY column under D (if Setting 40 is DIAMETER) or R (if Setting 40 is RADIUS).

If the offset value is negative, cutter compensation operates as though the program specifies the opposite G code. For example, a negative value entered for a G41 will behave as if a positive value was entered for G42. Also, when cutter compensation is active (G41 or G42), you may use only the X-Y plane (G17) for circular motions. Cutter compensation is limited to compensation in only the X-Y plane.

G40 cancels cutter compensation and is the default condition when you power on your machine. When cutter compensation is not active, the programmed path is the same as the center of the cutter path. You may not end a program (M30, M00, M01, or M02) with cutter compensation active.

The control operates on one motion block at a time. However, it will look ahead at the next (2) blocks that have X or Y motions. The control checks these (3) blocks of information for interference. Setting 58 controls how this part of cutter compensation works. Available Setting 58 values are Fanuc or Yasnac.

If Setting 58 is set to Yasnac, the control must be able to position the side of the tool along all of the edges of the programmed contour without overcutting the next two motions. A circular motion joins all of the outside angles.

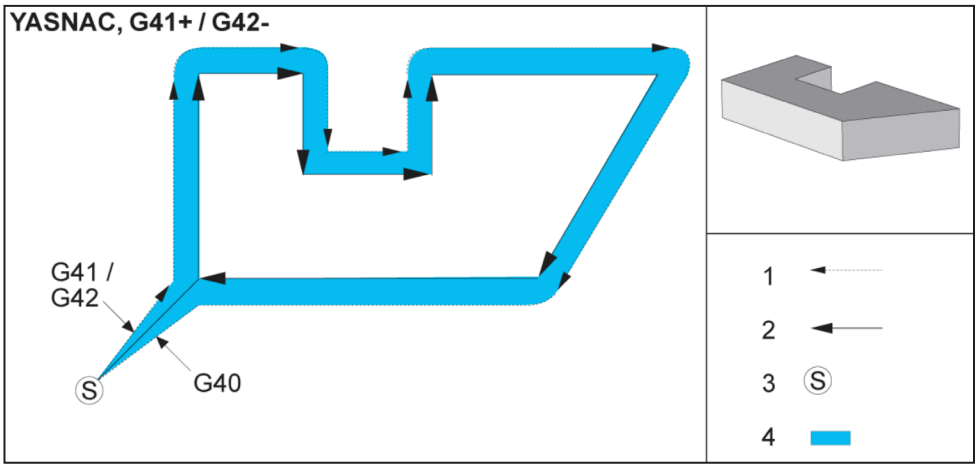
If Setting 58 is set to Fanuc, the control does not require that the tool cutting edge be placed along all edges of the programmed contour, preventing overcutting. However the control will generate an alarm if the cutter's path is programmed so that it will overcut. The control joins outside angles less than or equal to 270 degrees with a sharp corner. It joins outside angles of more than 270 degrees with an extra linear motion.

Cutter Compensation

These diagrams show how cutter compensation works for the possible values of Setting 58. Note that a small cut of less than the tool radius and at a right angle to the previous motion will work only with the Fanuc setting.

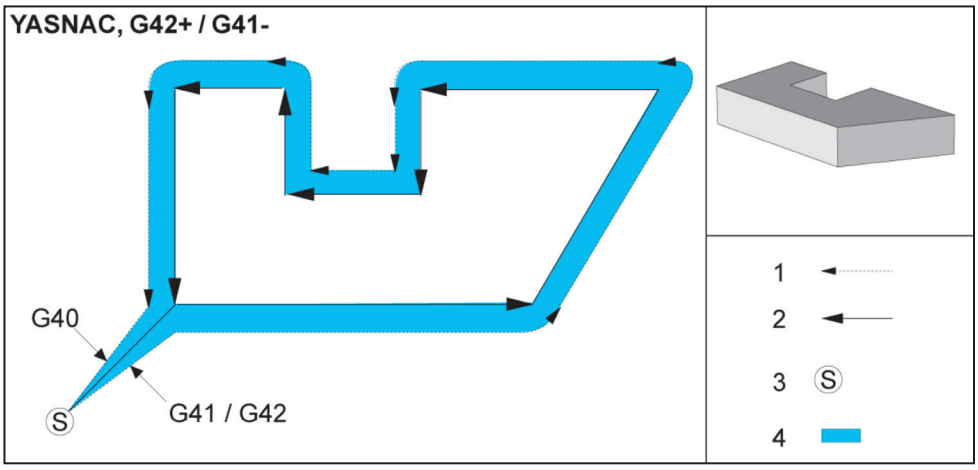
Cutter Compensation, YASNAC Style,G41 with a Positive Tool Diameter or G42 with a Negative Tool Diameter:

[1] Tool Path Actual Center,
[2] Programmed Tool Path,
[3] Start Point,
[4] Cutter Compensation. G41 / G42 and G40 are commanded at the start and end of the tool path.



Cutter Compensation, YASNAC Style, G42 with a Positive Tool Diameter or G41 with a Negative Tool Diameter:

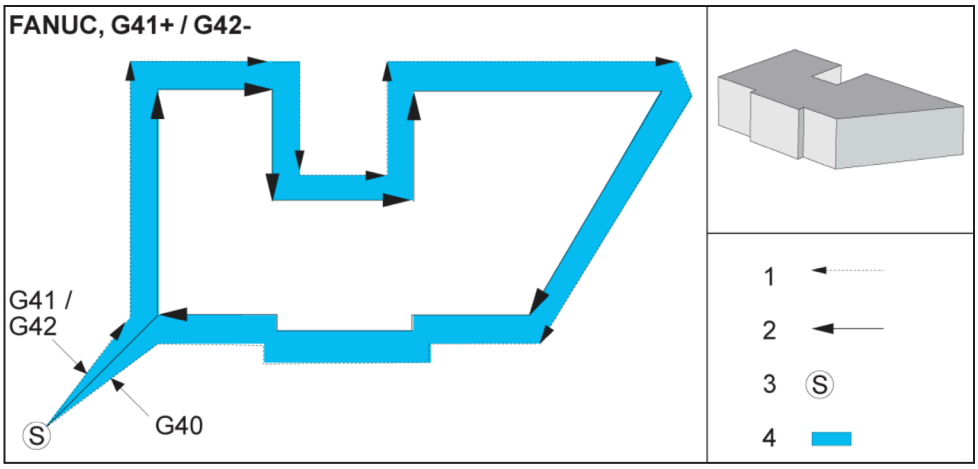
[1] Tool Path Actual Center,
[2] Programmed Tool Path,
[3] Start Point,
[4] Cutter Compensation. G41 / G42 and G40 are commanded at the start and end of the tool path.



Cutter Compensation (Contin.)

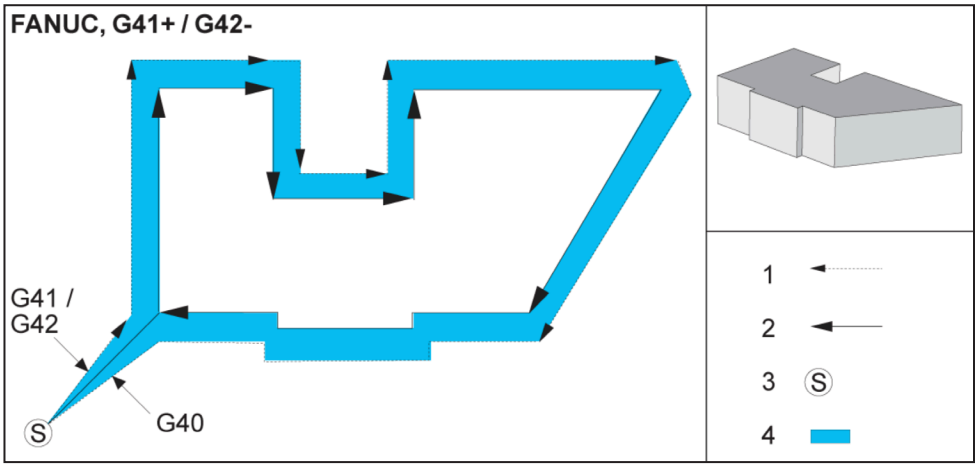
Cutter Compensation, FANUC Style, G41 with a Positive Tool Diameter or G42 with a Negative Tool Diameter:

[1] Tool Path Actual Center,
[2] Programmed Tool Path,
[3] Start Point,
[4] Cutter Compensation. G41 / G42 and G40 are commanded at the start and end of the tool path.



Cutter Compensation, FANUC Style, G42 with a Positive Tool Diameter or G41 with a Negative Tool Diameter:

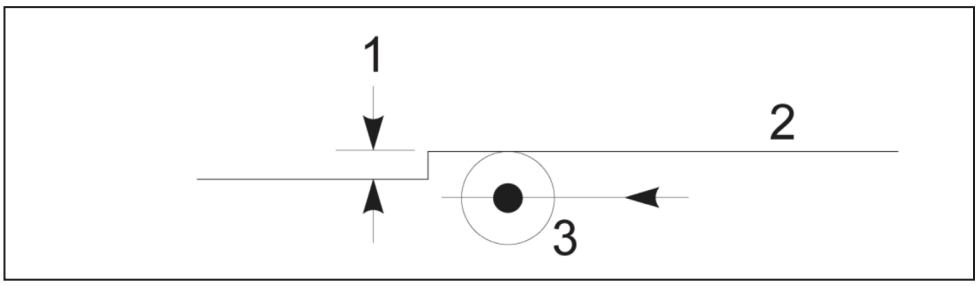
[1] Tool Path Actual Center,
[2] Programmed Tool Path,
[3] Start Point,
[4] Cutter Compensation. G41 / G42 and G40 are commanded at the start and end of the tool path.



Improper Cutter Compensation:

[1] Move is less than cutting comp radius,
[2] Workpiece,
[3] Tool.

NOTE: A small cut of less than tool radius and at a right angle to the previous motion will only work with the Fanuc setting. A cutter compensation alarm will be generated if the machine is set to the Yasnac setting.



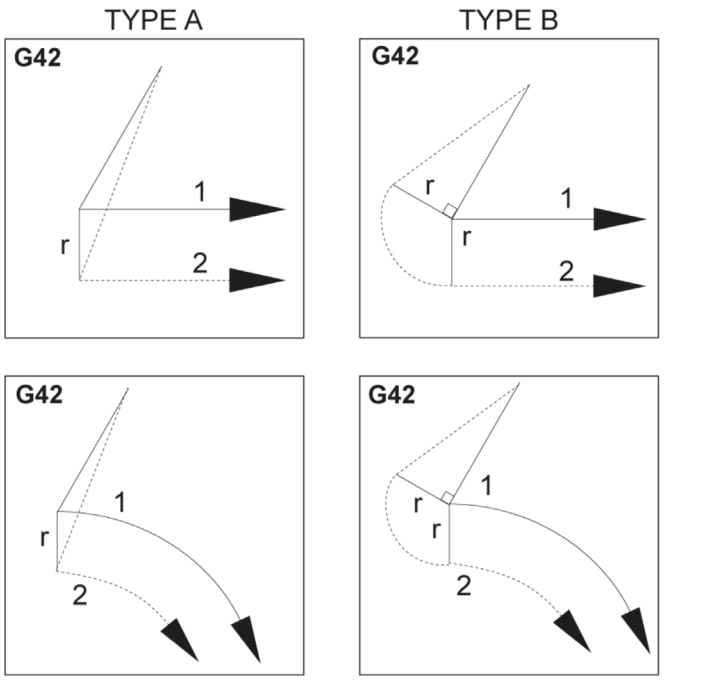
Feed Adjustments in Cutter Compensation

When using cutter compensation in circular moves, there is the possibility of speed adjustments to what has been programmed. If the intended finish cut is on the inside of a circular motion, the tool should be slowed down to ensure that the surface feed does not exceed what was intended by the programmer. There are problems, however, when the speed is slowed by too much. For this reason, Setting 44 is used to limit the amount by which the feed is adjusted in this case. It can be set between 1% and 100%. If set to 100%, there will be no speed changes. If set to 1%, the speed can be slowed to 1% of the programmed feed.

When the cut is on the outside of a circular motion, there is no speed-up adjustment made to the feed rate.

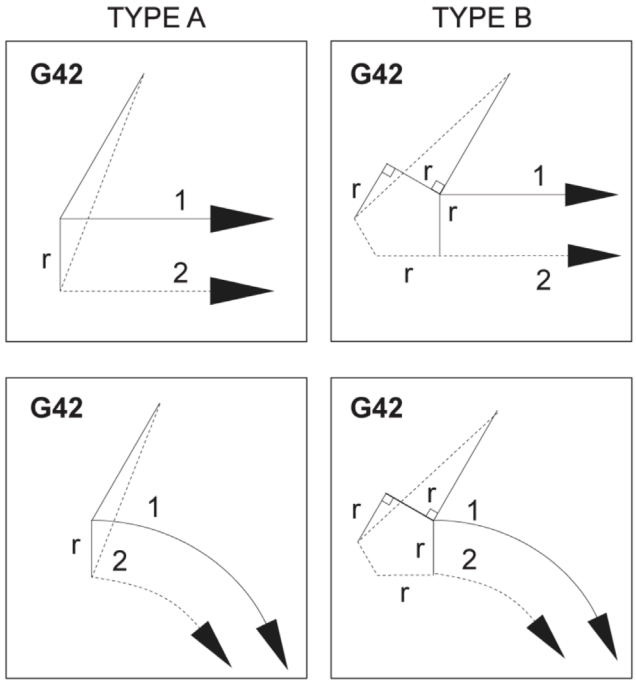
Cutter Compensation Entry (Yasnac)
Type A and B:

[1] Programmed Path,
[2] Tool Center Path,
[r] Tool Radius



Cutter Compensation Entry (Fanuc style)
Type A and B:

[1] Programmed Path,
[2] Tool Center Path,
[r] Tool Radius



Circular Interpolation and Cutter Compensation

In this section, the usage of G02 (Circular Interpolation Clockwise), G03 (Circular Interpolation Counterclockwise) and Cutter Compensation (G41: Cutter Compensation Left, G42: Cutter Compensation Right) is described.

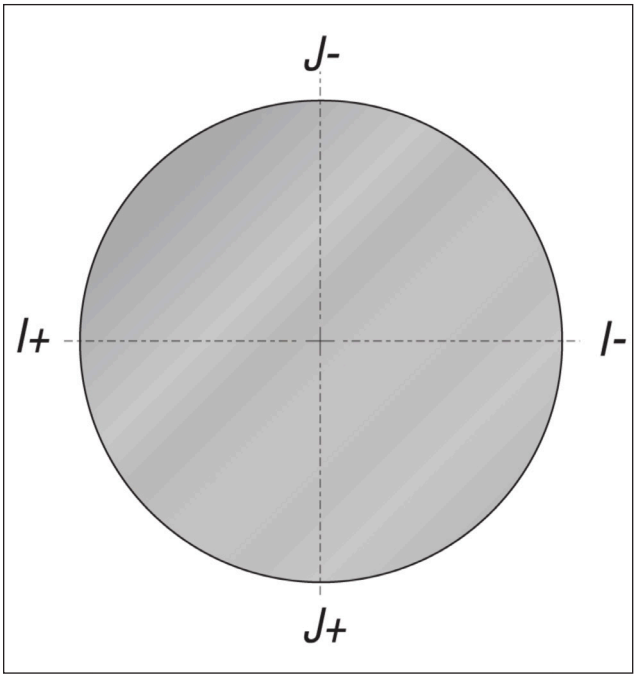
Using G02 and G03, we can program the machine to cut circular moves and radii. Generally, when programming a profile or a contour, the easiest way to describe a radius between two points is with an R and a value. For complete circular moves (360 degrees), an I or a J with a value must be specified. The circle section illustration will describe the different sections of a circle.

By using cutter compensation in this section, the programmer will be able to shift the cutter by an exact amount and be able to machine a profile or a contour to the exact print dimensions. By using cutter compensation, programming time and the likelihood of a programming calculation error is reduced due to the fact that real dimensions can be programmed, and part size and geometry can be easily controlled.

Here are a few rules about cutter compensation that you must follow closely for successful machining operations. Always refer to these rules when you write your programs.

This illustration shows how the tool path is calculated for the cutter compensation.

The detail section shows the tool in the starting position and then in the offset position as the cutter reaches the workpiece.



Circular Interpolation G02 and G03:

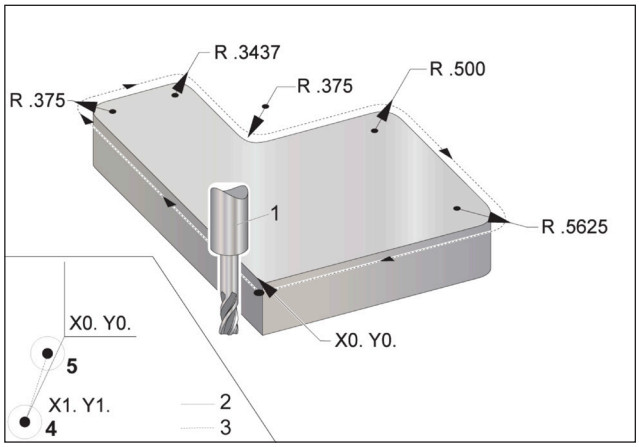
[1] 0.250” diameter endmill,

[2] Programmed path,

[3] Center of Tool,

[4] Start Position,

[5] Offset Tool Path.



Circular Interpolation and Cutter Compensation (Contin.)

Programming exercise showing tool path.

This program uses cutter compensation. The tool path is programmed to the centerline of the cutter. This is also the way the control calculates for cutter compensation.

```
%
O40006 (Cutter comp ex-prog) ;
(G54 X0 Y0 is at the lower left of part corner) ;
(Z0 is on top of the part) ;
(T1 is a .250 dia endmill) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54 (Safe startup) ;
X-1. Y-1. (Rapid to 1st position) ;
S1000 M03 (Spindle on CW) ;
G43 H01 Z0.1(Tool offset 1 on) ;
M08(Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G01 Z-1. F50. (Feed to cutting depth) ;
G41 G01 X0 Y0 D01 F50. (2D Cutter Comp left on) ;
Y4.125 (Linear motion) ;
G02 X0.25 Y4.375 R0.375 (Corner rounding) ;
G01 X1.6562 (Linear motion) ;
G02 X2. Y4.0313 R0.3437 (Corner rounding) ;
G01 Y3.125 (Linear motion) ;
G03 X2.375 Y2.75 R0.375 (Corner rounding) ;
G01 X3.5 (Linear motion) ;
G02 X4. Y2.25 R0.5 (Corner rounding) ;
G01 Y0.4375 (Linear motion) ;
G02 X3.4375 Y-0.125 R0.5625 (Corner rounding) ;
G01 X-0.125 (Linear motion) ;
G40 X-1. Y-1. (Last position, cutter comp off) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%
```

Canned Cycles

Canned cycles are G-codes that do repetitive operations such as drilling, tapping, and boring. You define a canned cycle with alphabetic address codes. While the canned cycle is active, the machine does the defined operation every time you command a new position, unless you specify not to.

Canned cycles simplify part programming. Most common Z-axis repetitive operations, such as drilling, tapping, and boring, have canned cycles. When active, a canned cycle executes at every new axis position. Canned cycles execute axis motions as rapid commands (G00) and the canned cycle operation is performed after the axis motion. This applies to G17, G19 cycles, and Y-Axis movements on Y-Axis lathes.

Drilling Canned Cycles

All four drill canned cycles can be looped in G91, Incremental Programming mode.

- The G81 Drill Canned Cycle is the basic drilling cycle. It is used for drilling shallow holes or for drilling with Through Spindle Coolant (TSC).
- The G82 Spot Drill Canned Cycle is the same as the G81 Drill Canned Cycle except that it can dwell at the bottom of the hole. The optional argument Pn.nnn specifies the duration of the dwell.
- The G83 Normal Peck Drilling Canned Cycle is typically used for drilling deep holes. Peck depth can be variable or constant and always incremental. Qnn.nnn. Do not use a Q value when programming with I, J, and K.
- The G73 High-Speed Peck Drilling Canned Cycle is the same as the G83 Normal Peck Drilling Canned Cycle except that tool peck retraction is specified with Setting 22 - Can Cycle Delta Z. Peck drilling cycles are advised for hole depths greater than 3 times the diameter of the drill bit. The initial peck depth, defined by I, should generally be a depth of 1 tool diameter.

Tapping Canned Cycles

There are two tapping canned cycles. All tapping canned cycles can be looped in G91, Incremental Programming mode.

The G84 Tapping Canned Cycle is the normal tapping cycle. It is used for tapping right-hand threads.

G74 Reverse Tap Canned Cycle is the reverse thread tapping cycle. It is used for tapping left-hand threads.

Boring and Reaming Cycles

There are (5) boring canned cycles. All boring canned cycles can be looped in G91, Incremental Programming mode.

- The G85 Boring Canned Cycle is the basic boring cycle. It will bore down to the desired height and return to the specified height.
- The G86 Bore and Stop Canned Cycle is the same as the G85 Boring Canned Cycle except that the spindle will stop at the bottom of the hole before returning to the specified height.
- The G89 Bore In, Dwell, Bore Out Canned Cycle is the same as G85 except that there is a dwell at the bottom of the hole, and the hole continues to be bored at the specified feed rate as the tool returns to the specified position. This differs from other boring canned cycles where the tool either moves in Rapid Motion or hand jog to return to the return position.
- The G76 Fine Boring Canned Cycle bores the hole to the specified depth and after boring the hole, moves to clear the tool from hole before retracting.
- The G77 Back Bore Canned Cycle works similar to G76 except that before beginning to bore the hole, it moves the tool to clear the hole, moves down into the hole, and bores to the specified depth.

R Planes

R Planes, or return planes, are G-code commands that specify the Z-Axis return height during canned cycles.

The R Plane G-codes remain active for the duration of the canned cycle it is used with. G98 Canned Cycle Initial Point Return moves the Z axis to the height of the Z axis prior to the canned cycle.

G99 Canned Cycle R Plane Return moves the Z axis to the height specified by the Rnn.nnnn argument specified with the canned cycle.

Special G-Codes

Special G-codes are used for complex milling. These include:

- Engraving (G47)
- Pocket Milling (G12, G13, and G150)
- Rotation and Scaling (G68, G69, G50, G51)
- Mirror Image (G101 and G100)

Engraving

The G47 Text Engraving G-code lets you engrave text (including some ASCII characters) or sequential serial numbers with a single block of code.

Refer to G47 Text Engraving (Group 00) for more information on engraving.

Pocket Milling

There are two types of pocket milling G-codes on the Haas control:

Circular Pocket Milling is performed with the G12 Clockwise Circular Pocket Milling Command and the G13 Counter-Clockwise Circular Pocket Milling Command G-codes.

The G150 General Purpose Pocket Milling uses a subprogram to machine user-defined pocket geometries.

Make sure that the subprogram geometry is a fully closed shape. Make sure that the X-Y starting point in the G150 command is within the boundary of the fully closed shape. Failure to do so may cause Alarm 370 - Pocket Definition Error.

Refer to G12 Circular Pocket Milling CW / G13 Circular Pocket Milling CCW (Group 00) for more information on the pocket milling G-codes.

Rotation and Scaling

NOTE: You must purchase the rotation and scaling option to use these features. A 200-hour option tryout is also available.

G68 Rotation is used to rotate the coordinate system in the desired plane. You can use this feature together with G91 Incremental Programming mode to machine symmetrical patterns. G69 cancels rotation.

G51 applies a scaling factor to the positioning values in blocks after the G51 command. G50 cancels scaling. You can use scaling together with rotation, but be sure to command scaling first.

Refer to G68 Rotation (Group 16) for more information on the rotation and scaling G-codes.

Mirror Image

G101 Enable Mirror Image will mirror axis motion about the specified axis. Settings 45-48, 80 and 250 enable mirror imaging about the X, Y, Z, A, B, and C axes.

The mirror pivot point along an axis is defined by the Xnn. nn argument. This can be specified for a Y Axis that is enabled on the machine and in the settings by using the axis to mirror as the argument. G100 cancels G101.

Refer to G100/G101 Disable/Enable Mirror Image (Group 00) for more information on the mirror image G-codes.

Mill M-Codes Introduction

This page gives detailed descriptions of the M-codes that you use to program your machine.

CAUTION: The sample programs in this manual have been tested for accuracy, but they are for illustrative purposes only. The programs do not define tools, offsets, or materials. They do not describe workholding or other fixturing. If you choose to run a sample program on your machine, do so in Graphics mode. Always follow safe machining practices when you run an unfamiliar program.

NOTE: The sample programs in this manual represent a very conservative programming style. The samples are intended to demonstrate safe and reliable programs, and they are not necessarily the fastest or most efficient way to operate a machine. The sample programs use G-codes that you might choose not to use in more efficient programs.

M-codes are miscellaneous machine commands that do not command axis motion. The format for an M-code is the letter M followed by two to three digits; for example M03. Only one M-code is allowed per line of code. All M-codes take effect at the end of the block.

M-CODE	DESCRIPTION
M00	Stop Program
M01	Optional Program Stop
M02	Program End
M03	Spindle Forward Command
M04	Spindle Reverse Command
M05	Spindle Stop Command
M06	Tool Change
M07	Shower Coolant On
M08 / M09	Coolant On / Off
M10 / M11	Engage / Release 4th Axis Brake
M12 / M13	Engage / Release 5th Axis Brake
M16	Tool Change
M19	Orient Spindle

M-CODE	DESCRIPTION
M21-M25	Optional User M Function with M-Fin
M29	Set Output Relay with M-Fin
M30	Program End and Reset
M31	Chip Conveyor Forward
M33	Chip Conveyor Stop
M34	Coolant Increment
M35	Coolant Decrement
M36	Pallet Part Ready
M39	Rotate Tool Turret
M41 / M42	Low / High Gear Override
M46	Qn Pmm Jump to Line
M48	Validate That The Current Program is Appropriate for Loaded Pallet
M50	Pallet Change Sequence

18.1 | MILL M-CODES INTRODUCTION

M-CODE	DESCRIPTION
M51-M55	Set Optional User M-codes
M59	Set Output Relay
M61-M65	Clear Optional User M-codes
M69	Clear Output Relay
M70/M71	Workholding Clamp / Unclamp
M73 / M74	Tool Air Blast (TAB) On / Off
M75	Set G35 or G136 Reference Point
M78	Alarm if Skip Signal Found
M79	Alarm if Skip Signal Not Found
M80 / M81	Auto Door Open / Close
M82	Tool Unclamp
M83 / M84	Auto Air Gun On / Off
M86	Tool Clamp
M88 / M89	Through-Spindle Coolant On / Off
M90 / M91	Fixture Clamp Input On / Off
M95	Sleep Mode
M96	Jump If No Input
M97	Local Sub-Program Call
M98	Sub-Program Call
M99	Sub-Program Return or Loop
M104 / M105	Probe Arm Extend/ Retract
M109	Interactive User Input

M-CODE	DESCRIPTION
M116 / M117	Vise Air Chips Blast On/Off
M130 / M131	Display Media / Cancel Display Media
M138 / M139	Spindle Speed Variation On/Off
M158 / M159	Mist Condenser On/Off
M160	Cancel Active PulseJet
M161	PulseJet Continuous Mode
M162	PulseJet Single Event Mode
M163	PulseJet Modal Mode
M199	Pallet / Part Load or Program End
M300	M300 - APL/Robot Custom Sequence

Mill Settings Introduction

This page gives detailed descriptions of the settings that control the way that your machine works.

List of Settings

Inside the **SETTINGS** tab, the settings are organized into groups. Use the **[UP]** and **[DOWN]** cursor arrow keys to highlight a setting group. Press the **[RIGHT]** cursor arrow key to see the settings in a group, . Press the **[LEFT]** cursor arrow key to return to the setting group list.

To quickly access a single setting, make sure the **SETTINGS** tab is active, type the setting number, and then press **[F1]** or, if a setting is highlighted, press the **[DOWN]** cursor.

Some settings have numerical values that fit in a given range. To change the value of these settings, type the new value and press **[ENTER]**. Other settings have specific available values that you select from a list. For these settings, use the **[RIGHT]** cursor to display the choices. Press [UP] and **[DOWN]** to scroll through the options. Press **[ENTER]** to select the option.

SETTING NUMBER	DESCRIPTION
1	Auto Power Off Timer
2	Power Off at M30
4	Graphics Rapid Path
5	Graphics Drill Point
6	Front Panel Lock
8	Prog Memory Lock
9	Dimensioning
10	Limit Rapid at 50%
15	H and T Code Agreement
17	Opt Stop Lock Out
18	Block Delete Lock Out
19	Feedrate Override Lock
20	Spindle Override Lock

SETTING NUMBER	DESCRIPTION
21	Rapid Override Lock
22	Can Cycle Delta Z
23	9xxx Progs Edit Lock
27	G76 / G77 Shift Dir.
28	Can Cycle Act w/o X/Y
29	G91 Non-modal
31	Reset Program Pointer
32	Coolant Override
33	Coordinate System
34	4th Axis Diameter
35	G60 Offset
36	Program Restart
39	Beep @ M00, M01, M02, M30

19.1 | MILL SETTINGS - INTRODUCTION

SETTING NUMBER	DESCRIPTION
40	Tool Offset Measure
42	M00 After Tool Change
43	Cutter Comp Type
44	Min F Radius CC%
45	Mirror Image X Axis
46	Mirror Image Y Axis
47	Mirror Image Z Axis
48	Mirror Image A Axis
52	G83 Retract Above R
53	Jog w/o Zero Return
56	M30 Restore Default G
57	Exact Stop Canned X-Y
58	Cutter Compensation
59	Probe Offset X+
60	Probe Offset X-
61	Probe Offset Y+
62	Probe Offset Y-
63	Tool Probe Width
64	Tool Offset Measure Uses Work
71	Default G51 Scaling
72	Default G68 Rotation
73	G68 Incremental Angle

SETTING NUMBER	DESCRIPTION
74	9xxx Progs Trace
75	9xxx Progs Single BLK
76	Tool Release Lockout
77	Scale Integer F
79	5th-Axis Diameter
80	Mirror Image B Axis
81	Tool At Power Up
82	Language
83	M30/Resets Overrides
84	Tool Overload Action
85	Maximum Corner Rounding
86	M39 Lockout
87	Tool Change Resets Override
88	Reset Rests Override
90	Max Tools To Display
101	Feed Override -> Rapid
103	Cyc Start/Fh Same Key
104	Jog Handle to SNGL BLK
108	Quick Rotary G28
109	Warm-Up Time in Min.
110	Warmup X Distance
111	Warmup Y Distance

19.1 | MILL SETTINGS - INTRODUCTION

SETTING NUMBER	DESCRIPTION
112	Warmup Z Distance
113	Tool Change Method
114	Conveyor Cycle Time (minutes)
115	Conveyor On-Time (minutes)
117	G143 Global Offset
118	M99 Bumps M30 Cntrs
119	Offset Lock
120	Macro Var Lock
130	Tap Retract Speed
131	Auto Door
133	Repeat Rigid Tap
142	Offset Chng Tolerance
143	Machine Data Collection Port
144	Feed Override -> Spindle
155	Load Pocket Tables
156	Save Offsets with Program
158	X Screw Thermal Comp%
159	Y Screw Thermal Comp%
160	Z Screw Thermal Comp%
162	Default To Float
163	Disable .1 Jog Rate
164	Rotary Increment

SETTING NUMBER	DESCRIPTION
165	Ssv Variation (RPM)
166	Ssv Cycle
188	G51 X Scale
189	G51 Y Scale
190	G51 Z Scale
191	Default Smoothness
196	Conveyor Shutoff
197	Coolant Shutoff
199	Backlight Timer
216	Servo and Hydraulic Shutoff
238	High Intensity Light Timer (minutes)
239	Worklight Off Timer (minutes)
240	Tool Life Warning
242	Air Water Purge Interval
243	Air Water Purge On-Time
245	Hazardous Vibration Sensitivity
247	Simultaneous XYZ Motion in Tool Change
249	Enable Haas Startup Screen
250	Mirror Image C Axis
251	Subprogram Search Location
252	Custom Subprogram Search Location
253	Default Graphics Tool Width

19.1 | MILL SETTINGS - INTRODUCTION

SETTING NUMBER	DESCRIPTION
254	5-Axis Rotary Center Distance
255	MRZP X Offset
256	MRZP Y Offset
257	MRZP Z Offset
261	DPRNT Store Location
262	DPRNT Destination File Path
263	DPRNT Port
264	Autofeed Step Up
265	Autofeed Step Down
266	Autofeed Minimum Override
267	Exit Jog Mode After Idle Time
268	Second Home Position X
269	Second Home Position Y
270	Second Home Position Z
271	Second Home Position A
272	Second Home Position B
273	Second Home Position C
276	Workholding Input Monitor
277	Lubrication Cycle Interval
291	Main Spindle Speed Limit
292	Door Open Spindle Speed Limit
293	Tool Change Mid Position X

SETTING NUMBER	DESCRIPTION
294	Tool Change Mid Position Y
295	Tool Change Mid Position Z
296	Tool Change Mid Position A
297	Tool Change Mid Position B
298	Tool Change Mid Position C
300	MRZP X Offset Master
301	MRZP Y Offset Master
302	MRZP Z Offset Master
303	MRZP X Offset Slave
304	MRZP Y Offset Slave
305	MRZP Z Offset Slave
306	Minimum Chip Clear Time
310	Min User Travel Limit A
311	Min User Travel Limit B
312	Min User Travel Limit C
313	Max User Travel Limit X
314	Max User Travel Limit Y
315	Max User Travel Limit Z
316	Max User Travel Limit A
317	Max User Travel Limit B
318	Max User Travel Limit C
323	Disable Notch Filter

19.1 | MILL SETTINGS - INTRODUCTION

SETTING NUMBER	DESCRIPTION
325	Manual Mode Enabled
330	MultiBoot Selection Time out
335	Linear Rapid Mode
356	Beeper Volume
357	Warm Up Cycle Start Idle Time
369	PulseJet Injection Cycle Time
370	PulseJet Single Squirt Count
372	Parts Loader Type
375	APL Gripper Type
376	Light Curtain Enable
377	Negative Work Offsets
378	Safe Zone Calibrated Geometry Reference Point X
379	Safe Zone Calibrated Geometry Reference Point Y
380	Safe Zone Calibrated Geometry Reference Point Z
381	Enable Touchscreen
382	Disable Pallet Changer
383	Table Row Size
389	Vise Unclamped Safety Check
396	Enable / Disable Virtual Keyboard
397	Press and Hold Delay
398	Header Height
399	Header Tab

SETTING NUMBER	DESCRIPTION
400	Pallet Ready Beep Type
403	Change Popup Button Size
408	Exclude Tool From Safe Zone
409	Default Coolant Pressure
416	Media Destination
420	ATC Button Behavior
421	General Orient Angle
422	Lock Graphics Plane
423	Help Text Icon Size
424	Mist Extractor Condenser Time Out

Network Tab

Scan QR codes below to see the help information for Wire / WIFI Connection Setup, Haas Drop, Haas Connect.

NOTE: The Haas Drop and HaasConnect feature can be access through the MyHaas Application.

NETWORKING

MYHAAS

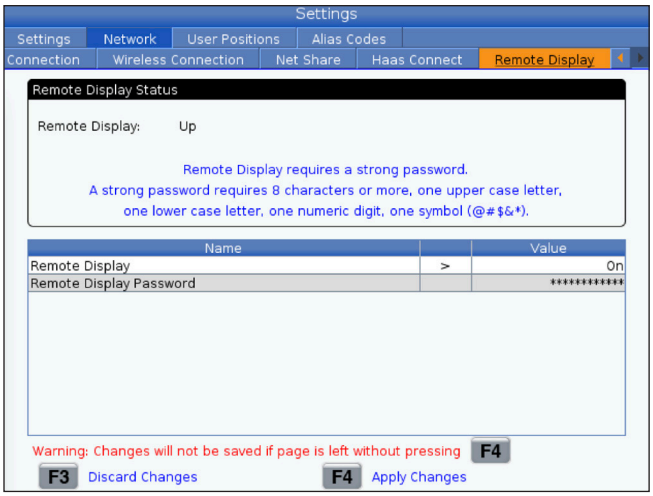
Remote Display View

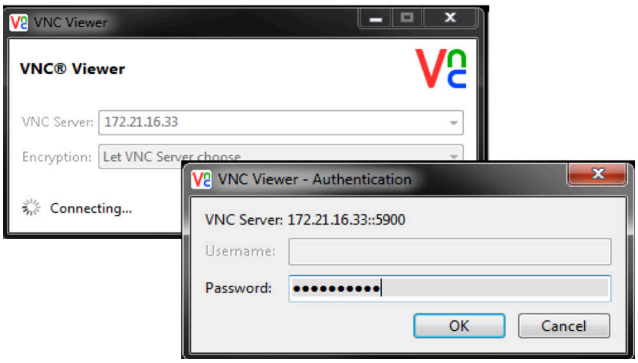
This procedure tells you how to view the machine display on a computer. The machine must be connected to a network with an Ethernet cable or with a wireless connection.

NOTE: The Remote Display tab is available in software version **100.18.000.1020 or higher**.

NOTE: You must download the VNC Viewer to your computer. Go to www.realvnc.com to download the free VNC Viewer.

Refer to Network Connection section for information on how to connect your machine to a network.





1 Push the SETTING button.

Navigate to the Wired Connection or Wireless Connection tab in the Network tab

Write down the IP Address for your machine.

Navigate to the Remote Display tab in the Network tab.

Turn ON the Remote Display.

Set the Remote Display Password.

NOTE: The Remote Display feature requires a strong password, follow the guide lines on the screen.

Press F4 to apply settings.

2 Open the VNC Viewer application on your computer.

Enter your IP Address in VNC Server. Select Connect.

At the login box enter the password you entered at the Haas control.

Select OK.

The machine display shows on your computer screen

Enable Rotary Axes

Scan QR code below to go to the Rotary Installation Procedure.

ROTARY ENABLE PROCEDURE

Overview

This tab collects settings that control user-defined positions such as second home, tool change mid-positions, spindle center line, tailstock and travel limits.

Refer to the Settings section of this manual for more information about these position settings.

CAUTION: Incorrectly set user positions can cause machine crashes. Set user positions with caution, especially after you have changed your application in some way (new program, different tools, etc.). Verify and change each axis position separately.

To set a user position, jog the axis into the position you want to use, and then press F2 to set the position. If the axis position is valid, a crash warning appears (except for user travel limits). After you verify that you want to make the change to the position, the control sets the position and makes the setting active.

If the position is not valid, the message bar at the bottom of the screen gives a message to explain why the position is not valid.

To inactivate and reset user position settings, press ORIGIN while the user positions tab is active, then choose from the menu that appears.

- Press 1 to remove the value of the currently selected position setting and make it inactive.
- Press 2 to remove the values of all second home position settings and make them inactive.
- Press 3 to remove the values of all Tool Change Mid-Position settings and make them inactive.
- Press 4 to remove the values of all Max User Travel Limit settings and make them inactive.
- Press CANCEL to exit the menu without making changes.

Interactive Manuals

Scan QR code to view these interactive manuals

PRODUCT	MILL OPERATOR'S MANUAL SUPPLEMENTS	SERVICE MANUAL
Desktop Mill	Desktop Mill - Interactive Operators Manual Supplement	N/A
Compact Mill	Compact Mill - Interactive Operator's Manual Supplement	N/A
Gantry - Series	Gantry-Series - Interactive Operator's Manual Supplement	N/A
Mill APL	Mill - APL - Interactive Operator's Manual Supplement	Haas Automatic Parts Loader - Interactive Service Manual
Pallet Pool	Pallet Pool - Interactive Operator's Manual Supplement	Pallet Pool - Interactive Service Manual
VF Pallet Pool	VF-Pallet Pool - Interactive Operator's Manual	
Rotary	Rotary - Interactive Operator's Manual Supplement	Rotary - Interactive Service Manual
UMC-Series	UMC-Series - Interactive Operator's Manual Supplement	UMC-Series - Interactive Service Manual
VR-Series	VR-Series - Interactive Operator's Manual Supplement	N/A

OTHER EQUIPMENT	OPERATOR'S MANUAL	SERVICE MANUAL
Autodoor	N/A	Autodoor - Interactive Service Manual
Haas Robot Package	Haas Robot Package - Interactive Operator's Manual	Haas Robot Package - Interactive Service Manual
HSF-325	HSF-325 Interactive Operator's/Service Manual	
HTS400	HTS400 - Interactive Operator's/Service Manual	
Haas Tooling and Workholding		Haas Tooling and Workholding - Interactive Service Manual
Lubrication Systems	N/A	Lubrication Systems - Interactive Service Manual
Chip Removal and Coolant	N/A	Chip Removal and Coolant - Interactive Service Manual
WIPS and WIPS-L	WIPS - Interactive Operator's Manual Supplement	N/A
CAN Bus Systems	N/A	CAN Bus Systems - Interactive Service Manual