



HAAS SERVICE AND OPERATOR MANUAL ARCHIVE

Mill Operators Manual 96-8000 Rev AL English July 2011

- This content is for illustrative purposes.
- Historic machine Service Manuals are posted here to provide information for Haas machine owners.
- Publications are intended for use only with machines built at the time of original publication.
- As machine designs change the content of these publications can become obsolete.
- You should not do mechanical or electrical machine repairs or service procedures unless you are qualified and knowledgeable about the processes.
- Only authorized personnel with the proper training and certification should do many repair procedures.

**WARNING: Some mechanical and electrical service procedures can be extremely dangerous or life-threatening.
Know your skill level and abilities.**

All information herein is provided as a courtesy for Haas machine owners for reference and illustrative purposes only. Haas Automation cannot be held responsible for repairs you perform. Only those services and repairs that are provided by authorized Haas Factory Outlet distributors are guaranteed.

Only an authorized Haas Factory Outlet distributor should service or repair a Haas machine that is protected by the original factory warranty. Servicing by any other party automatically voids the factory warranty.



Mill Operator's Manual

96-8000 Rev AL July 2011



Haas Automation, Inc., 2800 Sturgis Road, Oxnard, CA 93030 | HaasCNC.com



HAAS AUTOMATION, INC.
LIMITED WARRANTY CERTIFICATE
Covering Haas Automation, Inc. CNC Equipment
Effective January 1, 2009

Haas Automation Inc. ("Haas" or "Manufacturer") provides a limited warranty to all new mills, turning centers and rotary machines (collectively, "CNC Machines") and its 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 and 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 the final purchaser and 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 delivered to the Customer's facility. Customer may purchase an extension of the warranty period from Haas or an authorized Haas distributor (a "Warranty Extension").

Repair or Replacement Only

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

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, chip removal system, 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, (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 this warranty above.

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.

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.



Warranty Registration

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

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

In order to record the end-user customer of this machine for updates and for product safety notices, we must have the machine registration returned immediately. Please fill out completely and mail to the above address to ATTENTION (VF-1, GR-510, VF-6, etc. — whichever is applicable) REGISTRATIONS. Please include a copy of your invoice to validate your warranty date and to cover any additional options you may have purchased.

Company Name: _____ **Contact Name:** _____

Address: _____

Dealer: _____ **Date Installed:** _____ / _____ / _____

Model No. : _____ **Serial Number:** _____

Telephone: (_____) _____ **FAX:** (_____) _____

The equipment contains a pre-set automatic shut off feature that causes the equipment to automatically cease operation after 800 hours of use. This feature protects the buyer from theft. Unauthorized use of machine is kept to a minimum as the machine will stop running programs once the allotted time runs out. Operation may be resumed by use of the access code; contact your dealer for codes.



Customer Satisfaction Procedure

Dear Haas customer,

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

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

Contact Haas Automation's Customer Service Center by calling 800-331-6746 and ask for the Customer Service Department. So that we may resolve your concerns as quickly as possible, please have the following information available when you call:

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

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

Haas Automation, Inc.

2800 Sturgis Road

Oxnard, CA 93030

Att: Customer Satisfaction Manager

e-mail: Service@HaasCNC.com

Once you contact the Haas Automation Customer Service Center, we will make every effort to work directly with you and your distributor 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.



Customer Feedback

If you have any concerns or questions in regard to the Haas Operator's manual, please contact us via our e-mail, pubs@haascnc.com. We look forward to any suggestions you may have.

Certification



All Haas CNC machine tools carry the ETL Listed mark, certifying that the 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.



The ISO 9001:2008 certification from ISA, Inc. (an ISO registrar) serves as an impartial appraisal of Haas Automation's quality management system. This achievement affirms Haas Automation's conformance with the standards set forth by the International Organization for Standardization, and acknowledges the Haas commitment to meeting the needs and requirements of its customers in the global marketplace.

The Information contained in this manual is constantly being updated. The latest updates and other helpful information is available online as a free download in .pdf format (go to www.HaasCNC.com and click on "Manual Updates under the "Customer Service" drop-down menu in the navigation bar).

Original instructions



CONTENTS

Safety 1

- Machine Safety
- Best Practices
- Warning Decals

Safety

Introduction 15

- Machine Overview
- Control Description
- Option descriptions

Introduction

Operation 43

- General Operation
- Programming Introduction
- Control Features

Operation

Programming 103

- Edit Menu
- Search Menu
- Macros
- Quick Code

Programming

Subroutines 145

- External
- Local

Subroutines

4th and 5th Axis Programming.... 149

4th & 5th Axis

G Codes, M Codes, Settings 155

- Canned Cycles
- Machine Commands

G & M Codes
Settings

Maintenance 263

- Air / Electrical Requirements
- Recommended Fluids
- Maintenance Intervals

Maintenance

Index 285

Index



Declaration of Conformity

Product:

CNC Milling Centers (Vertical and Horizontal)

*Including all options factory installed 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 2004/108/EC
- Low Voltage Directive 2006/95/EC
- Additional Standards:
 - EN 60204-1:2006/A1:2009
 - EN 614-1:2006+A1:2009
 - EN 894-1:1997+A1:2008
 - EN 954-1 Safety of machinery - Safety - related parts of control systems part 1: General principles for design: (1997)
 - EN 14121-1:2007

RoHS: COMPLIANT by Exemption per producer documentation. Exempt by:

- a) Large scale stationary industrial tool
- b) Monitoring and control systems
- c) Lead as an alloying element in steel

Person authorized to compile technical file:

Patrick Goris

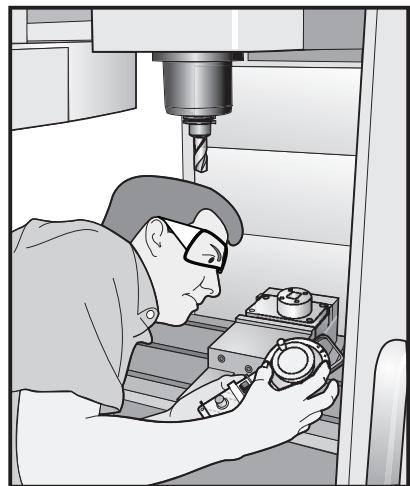
Address: Haas Automation Europe
Mercuriusstraat 28
B-1930 Zaventem
Belgium

Original instructions



SAFETY

Safety



THINK SAFETY! DON'T GET CAUGHT UP IN YOUR WORK

All milling machines contain hazards from rotating parts, belts and pulleys, high voltage electricity, noise, and compressed air. When using CNC machines and their components, basic safety precautions must always be followed to reduce the risk of personal injury and mechanical damage.

Important — This machine to be operated only by trained personnel in accordance with the Operator's manual, safety decals, safety procedures and instructions for safe machine operation.

Safety Contents

Machine Safety Notes / Best Practices	2
Setup Mode	4
Uses and Guidelines for Proper Machine Operation	5
Decal Examples.....	8
Declaration of Warning, Caution, and Notes	12
FCC Compliance	13

General Product Use Specifications and Limits

Environmental (Indoor use only)*		
	Minimum	Maximum
Operating Temperature	41° F (5° C)	122° F (50° C)
Storage Temperature	-4° F (-20° C)	158° F (70° C)
Ambient Humidity	20% relative, non-condensing	90% relative, non-condensing
Altitude	Sea Level	6000 ft. (1829 m)

Noise		
	Minimum	Maximum**
Emitted from all areas of machine during use at a typical operator's position	70 dB	Greater than 85dB

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

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



READ BEFORE OPERATING THIS MACHINE:

- ◆ Only authorized personnel should work on this machine. Untrained personnel present a hazard to themselves and the machine, and improper operation will void the warranty.
 - ◆ Check for damaged parts and tools before operating 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. Contact your shop supervisor.
 - ◆ Use appropriate eye and ear protection while operating the machine. ANSI-approved impact safety goggles and OSHA-approved ear protection are recommended to reduce the risks of sight damage and hearing loss.
 - ◆ Do not operate the machine unless the doors are closed and the door interlocks are functioning properly. Rotating cutting tools can cause severe injury. When a program is running, the mill table and spindle head can move rapidly at any time in any direction.
 - ◆ The Emergency Stop button is the large, circular red switch located on the Control Panel. Pressing the Emergency Stop button will instantly stop all motion of the machine, the servo motors, the tool changer, and the coolant pump. Use the Emergency Stop button only in emergencies to avoid crashing the machine.
 - ◆ 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 and the key available only to qualified service personnel.
 - ◆ DO NOT modify or alter this equipment in any way. If modifications are necessary, all such requests must be handled by Haas Automation, Inc. Any modification or alteration of any Haas Milling or Turning Center could lead to personal injury and/or mechanical damage and will void your warranty.
 - ◆ Consult your local safety codes and regulations before operating the machine. Contact your dealer anytime safety issues need to be addressed.
 - ◆ It is the shop owner's responsibility to make sure that everyone who is involved in installing and operating the machine is thoroughly acquainted with the installation, operation, and safety instructions provided with the machine BEFORE they perform any actual work. The ultimate responsibility for safety rests with the shop owner and the individuals who work with the machine.
- ◆ **This machine is automatically controlled and may start at any time.**



- ◆ This machine can cause severe bodily injury.
- ◆ Do not operate with the doors open.
- ◆ Avoid entering the machine enclosure.
- ◆ Do not operate without proper training.
- ◆ Always wear safety goggles.
- ◆ Never place your hand on the tool in the spindle and press ATC FWD, ATC REV, NEXT TOOL, or cause a tool change cycle. The tool changer will move in and crush your hand.
- ◆ To avoid tool changer damage, ensure that tools are properly aligned with the spindle drive lugs when loading tools.
- ◆ The electrical power must meet the specifications in this manual. Attempting to run the machine from any other source can cause severe damage and will void the warranty.
- ◆ Do not press POWER UP/RESTART on the control panel until after the installation is complete.
- ◆ Do not attempt to operate the machine before all of the installation instructions have been completed.
- ◆ Never service the machine with the power connected.
- ◆ Improperly clamped parts machined at high speeds/feeds may be ejected and puncture the safety door. Machining oversized or marginally clamped parts is not safe.
- ◆ Windows must be replaced if damaged or severely scratched - Replace damaged windows immediately.
- ◆ Do not process toxic or flammable material. Deadly fumes can be present. Consult material manufacturer for safe handling of material by-products before processing.
- ◆ The spindle head can drop without notice. Personnel must avoid the area directly under the spindle head.
- ◆ Do not reset a circuit breaker until the reason for the fault is investigated. Only Haas-trained service personnel should troubleshoot and repair the equipment.
- ◆ Follow these guidelines while performing jobs on the machine:
Normal operation - Keep the door closed and guards in place while machine is operating.

Part loading and unloading – An operator opens the door or guard, completes task, closes door or guard before pressing cycle start (starting automatic motion).



Tool loading or unloading – A machinist enters the machining area to load or unload tools. Exit the area completely before automatic movement is commanded (for example, next tool, ATC/Turret FWD/REV).

Machining job set-up – Press emergency stop before adding or removing machine fixtures.

Maintenance / Machine Cleaner – Press emergency stop or power off the machine before entering enclosure.

Do not enter the machining area anytime the machine is in motion; severe injury or death may result.

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 machines safely and use best practice machining techniques, it is also their responsibility to manage the progress of these methods. The machining process must be monitored to prevent damage if a hazardous condition occurs.

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

It is especially important to select monitoring equipment that can immediately perform an appropriate action without human intervention to prevent an accident, should a problem be detected.

SETUP MODE

All Haas mills 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 lock/unlock status affects how the machine operates when the doors are opened.

This feature replaces the following settings and parameters within the control:

- Setting 51, Door Hold Override
- Parameter 57 bit 7, Safety Circuit
- Parameter 57 bit 31, Door Stop Sp
- Parameter 586, Max Door Open Sp RPM.

Setup mode should be locked out (the keyswitch in the vertical, locked position) at most times. In locked mode, the enclosure doors are locked closed during the execution of a CNC program, spindle rotation or axis movement. The doors automatically unlock when the machine is not in cycle. Most 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. With the door closed in setup mode, opening the doors will stop motion and reduce spindle speed. The machine will allow several functions in setup mode with the doors opened, usually at reduced speed. The following charts summarize the modes and allowed functions.

DO NOT ATTEMPT TO OVERRIDE SAFETY FEATURES. DOING SO WILL MAKE THE MACHINE UNSAFE AND VOID THE WARRANTY.

Robot Cells

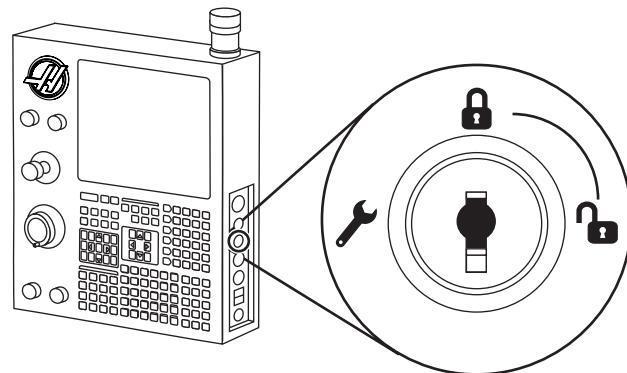
A machine in a robot cell is allowed to run, unrestricted, with the door open while in Lock/Run mode.

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

A robot-cell integrator will be able to test the open-door conditions of the CNC machine and assure the safety of the robot cell

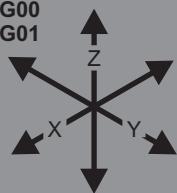
Machine Behavior with Door Open

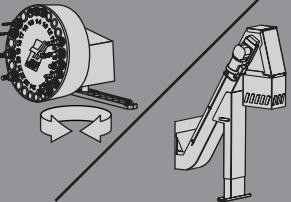
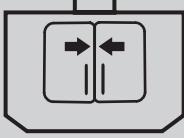
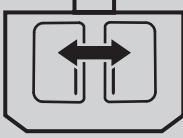
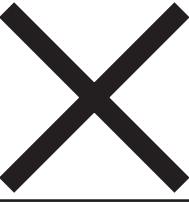
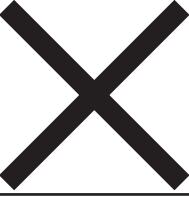
MACHINE FUNCTION	LOCKED	UNLOCKED
Maximum Rapid	Not allowed.	25%
Cycle Start	Not allowed, no machine motion or program execution.	With the Cycle Start button held down, machine motion is initiated provided the commanded spindle speed does not exceed 750 RPM mills, 50 RPM lathes.
Spindle CW/CCW	Yes, but user must press and hold CW/CCW button and maximum 750 RPM mills, 50 RPM lathes.	Yes, but maximum 750 RPM mills, 50 RPM lathes.
Tool Change	Not allowed.	Not allowed.
Next Tool feature	Not allowed.	Allowed with the Next Tool button pressed and held.
Opening doors while program is running.	Not allowed. Door is locked.	Yes, but axis motion will stop and the spindle will slow to a maximum 750 RPM in mills, 50 RPM lathes.
Conveyor motion	Not allowed.	Yes, but user must press and hold conveyor button.



	100%	Press and Hold 750 RPM
	100%	Press and Hold 750 RPM



		
	100%	0%
	25%	25%

		
	100%	
	100%	



USES AND GUIDELINES FOR PROPER MACHINE OPERATION

All milling machines contain hazards from rotating cutting tools, belts and pulleys, high voltage electricity, noise, and compressed air. When using milling machines and their components, basic safety precautions should always be followed to reduce the risk of personal injury and mechanical damage. **READ ALL APPROPRIATE WARNINGS, CAUTIONS, AND INSTRUCTIONS BEFORE OPERATING THIS MACHINE.**

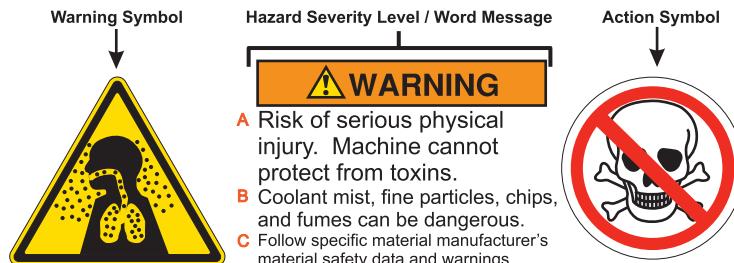
MODIFICATIONS TO THE MACHINE

DO NOT modify or alter this equipment in any way. If modifications are necessary, all such requests must be handled by Haas Automation, Inc. Any modification or alteration of any Haas machining center could lead to personal injury and/or mechanical damage and will void your warranty.

SAFETY DECALS

To help ensure that CNC tool dangers are quickly communicated and understood, hazard symbol decals are placed on Haas Machines in locations where hazards exist. If decals become damaged or worn, or if additional decals are needed to emphasize a particular safety point, contact your dealer or the Haas factory. **Never alter or remove any safety decal or symbol.**

Each hazard is defined and explained on the general safety decal, located at the front of the machine. Particular locations of hazards are marked with warning symbols. Review and understand the four parts of each safety warning, explained below, and familiarize yourself with the symbols on the following pages.



Warning Symbol - Identifies the potential hazard and reinforces the word message.

Word Message - Clarifies or reinforces the intent of the warning symbol.

A: Hazard.

B: Consequence if warning is ignored.

C: Action to prevent injury. Also refer to Action Symbol.

Hazard Severity Level - Color-coded to indicate risk in ignoring a hazard.

Red + "DANGER" = Hazard WILL cause death or serious injury if ignored.

Orange + "WARNING" = Hazard COULD cause death or serious injury if ignored.

Yellow + "CAUTION" = Hazard MAY cause minor to moderate injury if ignored.

Blue + "NOTICE" = Indicates an action to prevent damage to the machine.

Green + "INFORMATION" = Details about machine components.

Action Symbol: Indicates actions to prevent injury. Blue circles indicate mandatory actions to avoid harm, red circles with diagonal slashes indicate prohibited actions to avoid harm.



MILL WARNING DECALS

Safety

DANGER



Electrocution hazard.
Death by electric shock can occur.
Turn off and lock out system power before servicing.



Automatic Machine may start at any time.
Injury or death could be caused by untrained operator.
Read and understand operator's manual and safety signs before using this machine.



Risk of serious physical injury. Machine cannot protect from toxins.
Coolant mist, fine particles, chips, and fumes can be dangerous.
Follow specific material manufacturer's material safety data and warnings.



Risk of serious bodily injury.
The enclosure may not stop every type of projectile.
Double-check job set up before beginning any machining operations.
Always follow safe machining practices. Do not operate with doors or windows open or guards removed.



Risk of fire and explosion.
Machine is not designed to resist or contain blasts or fire.
Do not machine explosive or flammable materials or coolants.
Refer to specific material manufacturer's material safety data and warnings.



Risk of bodily injury.
Serious cuts, abrasions, and physical injury may result from slips and falls.
Avoid using the machine in wet, damp, or poorly lit areas.



Severe injury can occur.
Moving parts can entangle, trap, and cut. Sharp tools or chips can cut skin easily.
Ensure the machine is not in automatic operation before reaching inside.



Risk of eye and ear injury.
Flying debris into unprotected eyes can cause loss of sight.
Noise levels can exceed 70 dBA.
Must wear safety glasses and hearing protection when operating or in the area of machine.

Safety windows may become brittle and lose effectiveness when exposed to machine coolants and oils over time. If signs of discoloration, crazing, or cracking are found, replace immediately. Safety windows should be replaced every two years.

WARNING



Severe injury can occur.
Moving parts can entangle and trap.
Always secure loose clothing and long hair.



Risk of serious bodily injury.
Follow safe clamping practices. Inadequately clamped parts can be thrown with deadly force.
Securely clamp workpieces and fixtures.



Impact hazard.
Machine components can crush and cut.
Do not handle any part of the machine during automatic operation.
Always keep clear of moving parts.

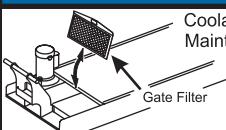


Moving parts can crush.
The tool changer will move in and crush your hand.
Never place your hand on the spindle and press ATC FWD, ATC REV, NEXT TOOL, or cause a tool change cycle.

- Do not allow untrained personnel to operate this machine.
- Do not alter or modify machine in any way.
- Do not operate this machine with worn or damaged components.
- No user serviceable parts inside. Machine must be repaired or serviced by authorized service technicians only.

©2009 Haas Automation, Inc
29-0769 Rev E

NOTICE



Coolant Tank Maintenance

Clean the filter screen weekly.

Remove the coolant tank cover and clean out any sediment inside the tank weekly.

Do not use plain water, permanent corrosion damage will result. Rust inhibiting coolant is required.

Do not use toxic or flammable liquids as a coolant.



LATHE WARNING DECALS

DANGER



Electrocution hazard.
Death by electric shock can occur.
Turn off and lock out system power before servicing.



Risk of serious physical injury. Machine cannot protect from toxins.
Coolant mist, fine particles, chips, and fumes can be dangerous.
Follow specific material manufacturer's material safety data and warnings.



Automatic Machine may start at any time.
Injury or death could be caused by untrained operator.
Read and understand operator's manual and safety signs before using this machine.



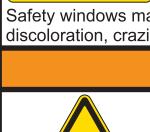
Risk of fire and explosion.
Machine is not designed to resist or contain blasts or fire.
Do not machine explosive or flammable materials or coolants.
Refer to specific material manufacturer's material safety data and warnings.



Severe injury can occur.
Moving parts can entangle, trap, and cut. Sharp tools or chips can cut skin easily.
Ensure the machine is not in automatic operation before reaching inside.



Risk of serious bodily injury.
The enclosure may not stop every type of projectile.
Double-check job set up before beginning any machining operations.
Always follow safe machining practices.
Do not operate with doors or windows open or guards removed.



Risk of eye and ear injury.
Flying debris into unprotected eyes can cause loss of sight.
Noise levels can exceed 70 dBA.
Must wear safety glasses and hearing protection when operating or in the area of machine.



Safety windows may become brittle and lose effectiveness when exposed to machine coolants and oils over time. If signs of discoloration, crazing, or cracking are found, replace immediately. Safety windows should be replaced every two years.

WARNING



Severe injury can occur.
Moving parts can entangle and trap.
Always secure loose clothing and long hair.



Risk of serious bodily injury and impact hazard.



Risk of serious bodily injury.
Inadequately clamped parts can be thrown with deadly force.
High RPM reduces chuck clamping force.
Do not machine using an unsafe setup or exceed rated chuck RPM.



Unsupported bar can whip with deadly results.
Do not extend barstock past end of drawtube without adequate support.
Do not apply excessive machining forces, doing so can dislodge the bar from support.



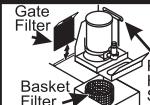
Moving parts can cut.
Sharp tools can cut skin easily.
Do not handle any part of the machine during automatic operation.
Do not touch rotating work pieces.



- Do not allow untrained personnel to operate this machine.
- Restrict access to open frame lathes.
- Use steady rest or tailstock to support long bars and always follow safe machining practices.
- Do not alter or modify machine in any way.
- Do not operate this machine with worn or damaged components.
- Machine must be repaired or serviced by authorized technicians only.



NOTICE



Clean the filter screen weekly.

Remove the coolant tank cover and clean out any sediment inside the tank weekly.

Do not use plain water, permanent corrosion damage will result. Rust inhibiting coolant is required.

Basket Filter

Pump Hanging Slot

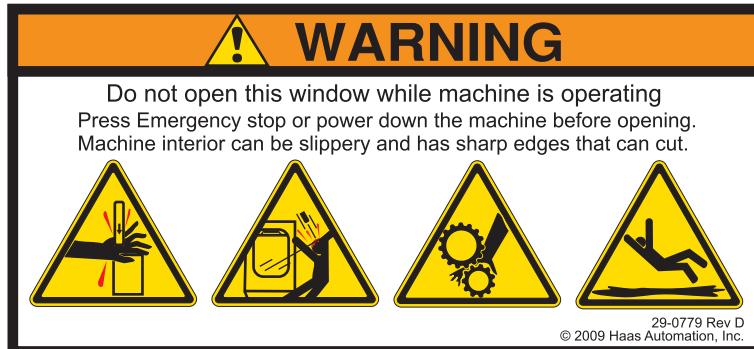
29-0765 Rev F
© 2009 Haas Automation, Inc.



OTHER SAFETY DECALS

Other decals may be found on your machine, depending on the model and options installed:

Safety



Refer to the APC Section for further explanation.



DECLARATION OF WARNINGS, CAUTIONS, AND NOTES

Throughout this manual, important and critical information is prefaced with the word "Warning", "Caution" and "Note"

Warnings are used when there is an extreme danger to the operator and/or to the machine. Take all steps necessary to heed the warning given. Do not continue if you cannot follow the warning instructions. An example warning is:

WARNING! Never put hands between tool changer and spindle head.

Cautions are used when there is the potential for minor personal injury or mechanical damage, for example:

CAUTION! Power down the machine before performing any maintenance tasks.

Notes give additional information to the operator about a particular step or procedure. This information should be taken into consideration by the operator as the step is performed to ensure there is no confusion, for example:

NOTE: If machine is equipped with the optional extended Z-clearance table, follow these guidelines:



FCC COMPLIANCE

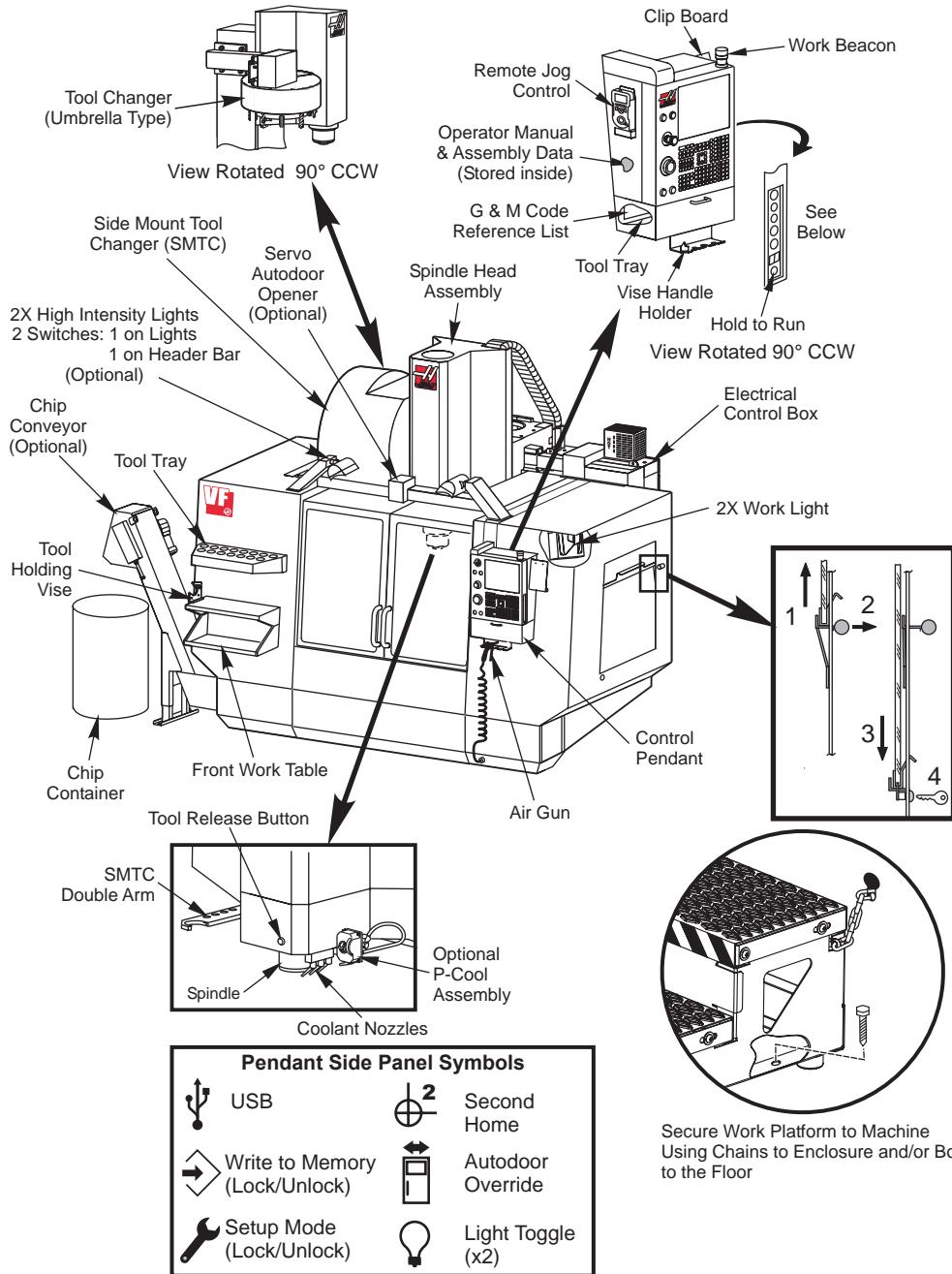
This equipment has been tested and found to comply with the limits for a Class A digital device, pursuant to Part 15 of the FCC Rules. These limits are designed to provide reasonable protection against harmful interference when the equipment is operated in a commercial environment. This equipment generates, uses, and can radiate radio frequency energy and, if not installed and used in accordance with the instruction manual, may cause harmful interference to radio communications. Operation of this equipment in a residential area is likely to cause harmful interference in which case the user will be required to correct the interference at his own expense.





INTRODUCTION

The following is a visual introduction to a HAAS mill. Some of the features shown will be highlighted in their appropriate sections.





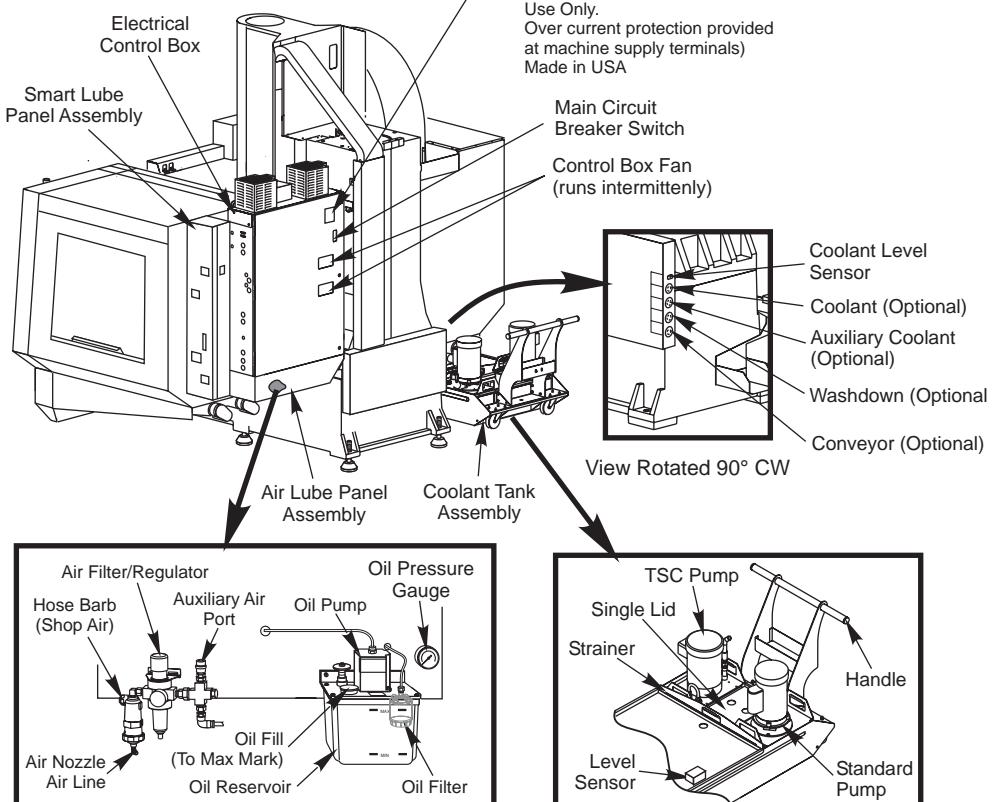
DATA PLATE

Model
Serial Number
Date of Manufacture
Voltage
Phase
Hertz
Full Load
Largest Load
Short Circuit Interrupting Capacity
Wiring Diagram
Short Circuit Current
Arc Flash Rating
NEMA Type 1 Enclosure for Indoor Use Only.
Over current protection provided at machine supply terminals
Made in USA

Main Circuit Breaker Switch
Control Box Fan (runs intermittently)

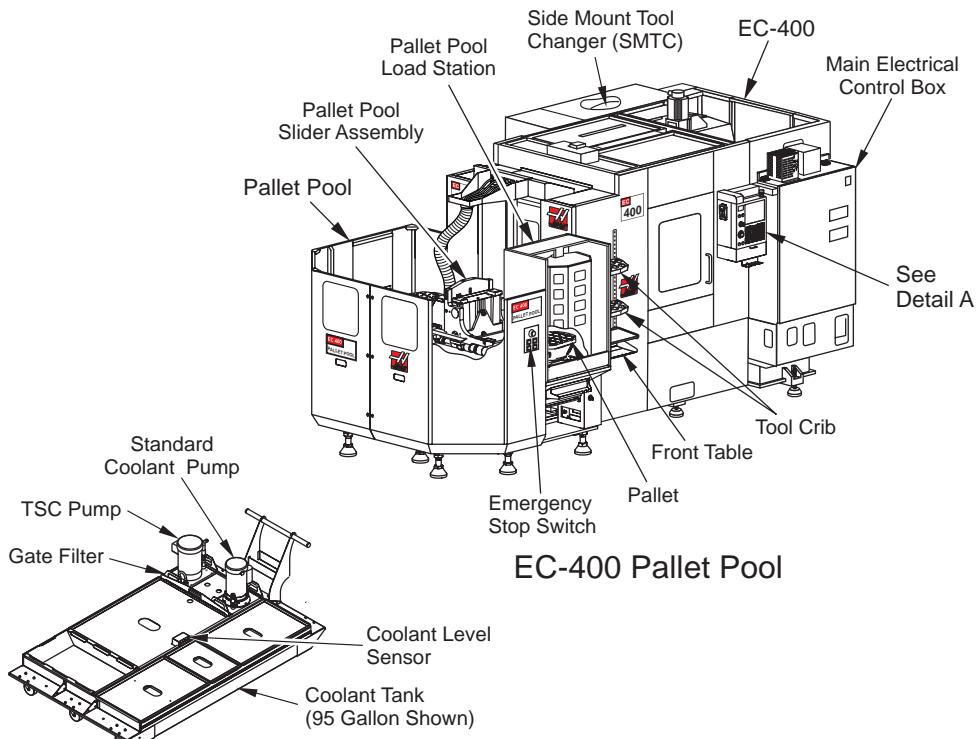
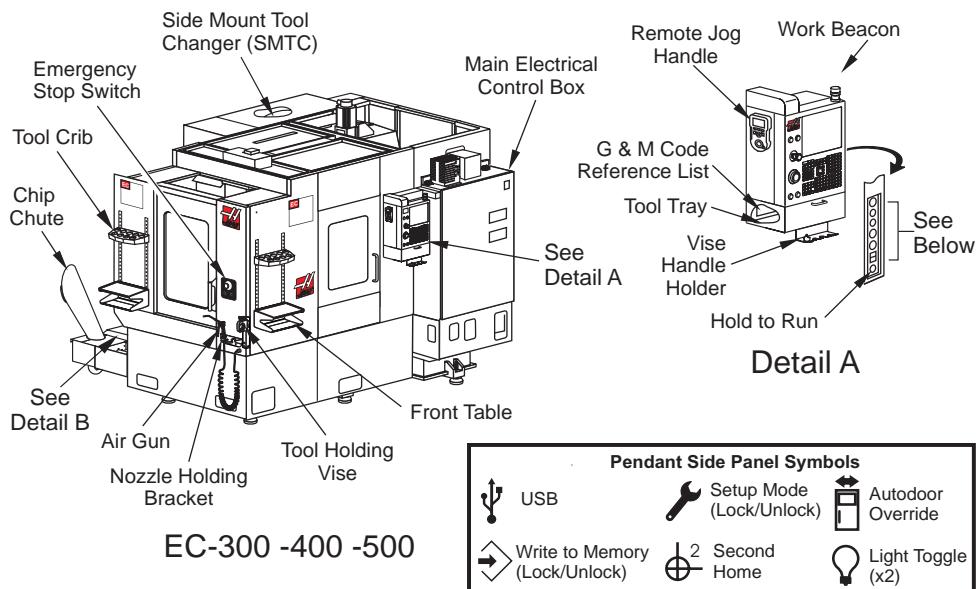
Coolant Level Sensor
Coolant (Optional)
Auxiliary Coolant (Optional)
Washdown (Optional)
Conveyor (Optional)

View Rotated 90° CW





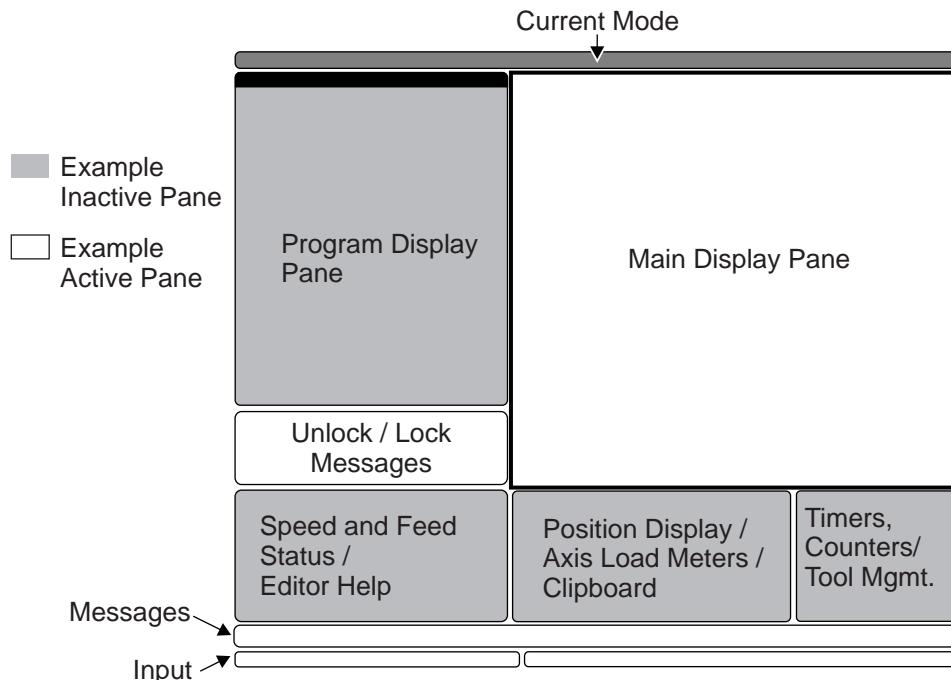
Horizontal Mills





CONTROL DISPLAY AND MODES

The control display is organized into panes that vary depending on the current control mode, and on what display keys are used. The following illustration shows the basic display layout:



Interaction with data can be carried out only within the currently active pane. Only one pane is active at any given time, and it is indicated with a white background. For example, to work with the Tool Offsets table, first make the table active by pressing the Offset key until it displays with a white background. Then make changes to the data. Changing the active pane within a control mode is typically done with the display keys.

Control functions are organized into three modes: **Setup**, **Edit**, and **Operation**. Each mode provides all of the necessary information to perform tasks that fall under the mode, organized to fit in one screen. For example, Setup mode displays both the work and tool offset tables, and position information. Edit mode provides two program editing panes and access to the VQCP and IPS/WIPS systems (if installed).

Access modes using the mode keys as follows:

Setup: ZERO RET, HAND JOG keys. Provides all control features for machine setup.

Edit: EDIT, MDI/DNC, LIST PROG keys. Provides all program editing, management, and transfer functions.



Operation: MEM key. Provides all control features necessary to make a part.

The current mode is shown in the title bar at the top of the display.

Note that functions from other modes can still be accessed from within the active mode by using the display keys. For example, while in Operation mode, pressing OFFSET will display the offsets tables as the active pane; toggle the offset display using the OFFSET key. pressing PROGRM CONVRS in most modes will shift to the edit pane for the current active program.

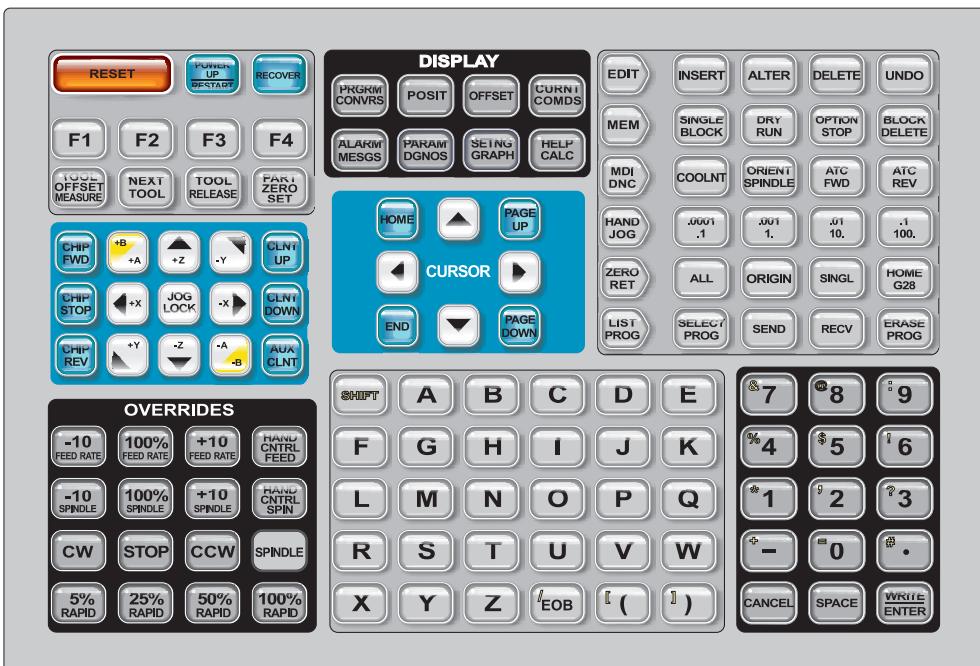
NAVIGATING TABBED MENUS

Tabbed menus are used in several control functions, such as Parameters, Settings, Help, List Prog, and IPS. To navigate these menus, use the arrow keys to select a tab, then press Enter to open the tab. If the selected tab contains sub-tabs, use the arrow keys and Enter to select the appropriate one.

To go up one tab level, press Cancel.

CONTROL PENDANT INTRODUCTION

The keyboard is broken up into eight sections: Function Keys, Jog Keys, Override Keys, Display Keys, Cursor Keys, Alpha Keys, Mode Keys and Number Keys. In addition there are miscellaneous keys and features located on the pendant and keyboard which are described briefly.





PENDANT FRONT PANEL CONTROLS



Power On - Turns the machine on.



Power Off - Turns the machine off.



Emergency Stop - Large red button with a yellow bezel. Press to stop all axis motion, disable servos, stop the spindle and tool changer, and turn off the coolant pump. Turn to reset.



Jog Handle - This is used to jog axes (select in Hand 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. Press Cycle Start to cancel. Note: The spindle will continue to turn during cutting.

PENDANT SIDE PANEL CONTROLS



USB - Plug compatible USB devices into this port.



Memory Lock - Keyswitch. Toggle to the lock position to protect programs and settings from alteration. Unlock to allow changes.



Setup Mode - Keyswitch. Locks and unlocks machine safety features for setup purposes (see the “Setup Mode” in the Safety section of this manual for details)



Second Home - Press this button to rapid all axes to the coordinates specified in G154 P20.



Autodoor Override - Press this button to open or close the Auto-door (if equipped).



Worklight - These switches toggle the internal worklight and High Intensity Lighting (if equipped).

Keyboard Beeper - Located at the top of the parts tray. Adjust the volume by turning the cover.



FUNCTION KEYS

F1- F4 Keys - These buttons have different functions depending on mode of operation. See the specific mode section for further descriptions and examples.

Tool Offset Meas (Tool Offset Measure) - Used to record tool length offsets during part setup.

Next Tool - Used to select the next tool from the tool changer. Used after pressing Tool Offset Measure in Setup.

Tool Release - Releases the tool from the spindle when in MDI mode, zero return mode, or handle jog mode.

Part Zero Set - Used to record work coordinate offsets during part setup (see Setting Offsets in the Operation section).

JOG KEYS

Chip FWD (Chip Auger Forward) - Starts optional chip auger in the "Forward" direction, moving chips out of the machine.

Chip Stop (Chip Auger Stop) - Stops auger movement.

Chip REV (Chip Auger Reverse) - Starts the optional chip auger in the "Reverse" direction, which is useful in clearing jams and debris from the auger.

XI-X, YI-Y, ZI-Z, AI-A and BI-B (axis keys)- Allows manually jogging of axis by holding down the individual button or by pressing the desired axis button and using the jog handle.

Jog Lock - Works with the axes buttons. Press jog lock and then an axis button and the axis will move to maximum travel or until jog lock is pressed again.

CLNT Up (Coolant Up) - Moves the optional Programmable Coolant (P-Cool) nozzle up.

CLNT Down (Coolant Down) - Moves the optional P-Cool nozzle down.

AUX CLNT (Auxiliary Coolant) - Pressing this key while in MDI mode only will turn on the optional Through the Spindle Coolant (TSC) system; pressing it a second time will turn off TSC.

OVERRIDE KEYS

These keys give the user the ability to override the speed of non-cutting (rapid) axes motion, programmed feeds and spindle speeds.

-10 - Decreases current feedrate by 10%.

100% - Sets overridden feedrate to programmed feedrate.

+10 - Increases current feedrate by 10%.



-10 - Decreases current spindle speed by 10%.

100% - Sets overridden spindle speed to programmed speed.

+10 - Increases current spindle speed by 10%.

Hand Cntrl Feed (Handle Control Feedrate) - Pressing this button allows the jog handle to be used to control the feedrate in $\pm 1\%$ increments.

Hand Cntrl Spin (Handle Control Spindle) - Pressing this button allows the jog handle to be used to control spindle speed in $\pm 1\%$ increments.

CW - Starts the spindle in the clockwise direction. This button is disabled on CE (export) machines.

CCW - Starts the spindle in the counterclockwise direction. This button is disabled on CE (export) machines.

The spindle can be started or stopped with the CW or CCW buttons any time the machine is at a Single Block stop or the Feed Hold button has been pressed. When the program is restarted with Cycle Start, the spindle will be turned back on to the previously defined speed.

STOP - Stops the spindle.

5% / 25% / 50% / 100% Rapid - Limits machine rapids to the value on the key. The 100% Rapid button allows maximum rapid.

Override Usage

The feedrate can be varied from 0% to 999% of the programmed value while in operation. This is done with the feedrate +10%, -10% and 100% buttons. The feedrate override is ineffective during G74 and G84 tapping cycles. Feedrate override does not change the speed of any auxiliary axes. During manual jogging, the feedrate override will adjust the rates selected from the keypad. This allows for fine control of the jog speed.

The spindle speed can also be varied, from 0% to 999%, using the spindle overrides. It is also ineffective for G74 and G84. In the Single Block mode, the spindle may be stopped. It will automatically start up upon continuing the program (pressing Cycle Start).

By pressing the Handle Control Feedrate key, the jog handle can be used to control feedrate from 0% to 999% in $\pm 1\%$ increments. By pressing the Handle Control Spindle key, the jog handle can be used to control spindle speed in $\pm 1\%$ increments (from 0% to 999%).

Rapid moves (G00) may be limited to 5%, 25%, or 50% of maximum using the keypad. If the 100% rapid is too fast, it may be set to 50% of maximum by Setting 10.

In the Settings page, it is possible to disable the override keys so that the operator cannot use them. These are Settings 19, 20 and 21.



The Feed Hold button acts as an override, stopping rapid and feed moves when it is pressed. The Cycle Start button must be pressed to proceed after a Feed Hold. The door switch on the enclosure also has a similar result but will display "Door Hold" when the door is opened. When the door is closed, the control will be in Feed Hold and Cycle Start must be pressed to continue. Door Hold and Feed Hold do not stop any auxiliary axes.

The operator can override the coolant setting by pressing the COOLNT button. The pump will remain either on or off until the next M-code or operator action (see Setting 32).

Overrides can be reset to defaults with an M06, M30 and/or pressing RESET (See Settings 83, 87, 88).

DISPLAY KEYS

Display keys provide access to the machine displays, operational information and help pages. They are often used to switch active panes within a function mode. Some of these keys will display additional screens when pressed more than once.

Prgrm/Convrs - Selects the active program pane in most modes. In MDI/DNC mode, press to access VQC and IPS/WIPS (if installed).

Posit (Position) - Selects the positions pane, located in the lower center of most screens. Displays the current axis positions. Toggle between relative positions by pressing the POSIT key. To filter the axes displayed in the pane, type the letter for each axis to display and press WRITE/ENTER. Each axis position is displayed in the order indicated.

Offset - Press to toggle between two offsets tables. Select the Tool Offsets table to display and edit tool length geometry, radius offsets, wear offsets, and coolant position. Select the Work Offsets table to edit the G-code specified work offset locations used in programs.

Curnt Comds (Current Commands) - Press PAGE UP / PAGE DOWN to cycle through menus for Maintenance, Tool Life, Tool Load, Advanced Tool Management (ATM), System Variables, Clock settings and timer / counter settings.

Alarm / Mesgs (Alarms / Messages) - Displays the alarm viewer and message screens. There are three alarm screens, the first shows the currently active alarms (first press of the Alarm/Mesgs button). Press the Right Arrow key to view Alarm History. Use the Up and Down Arrow keys to scroll through alarm history entries, and press F2 to write to a memory device.

Param / Dgnos (Parameters / Diagnostics) - Displays parameters that define the machine's operation. Parameters are organized by category in a tabbed menu, or to find a known parameter, type in the number and press the up or down arrow. Parameters are set at the factory and should not be modified except by authorized Haas personnel.



A second press of the Param / Dgnos key will display the first page of diagnostic data. This information is mainly used for troubleshooting by a certified Haas service technician. The first page of diagnostic data is discrete inputs and outputs. Pressing Page Down will display additional pages of diagnostic data.

Setng / Graph (Settings / Graphics) - Displays and allows changing of user settings. Like Parameters, Settings are organized by category in a tabbed menu. To find a known setting, type in the number and press the up or down arrow.

Pressing the Setng / Graph key a second time enables Graphics mode. In Graphics mode, the generated tool path of the program is seen and, if necessary, the program can be debugged before running it (See Graphics Mode in the Operation section)

Help / Calc (Help / Calculator) - Displays help topics in a tabbed menu. See the “Tabbed Help / Calculator Function” section for more information.

CURSOR KEYS

Use Cursor Keys to move to various screens and fields in the control, and for editing CNC programs.

Home - This button will move the cursor to the top-most item on the screen; in editing, this is the top left block of the program.

Up / Down Arrows - moves up/down one item, block or field.

Page Up / Down - Used to change displays or move up/down one page when viewing a program.

Left Arrow - Used to select individually editable items when viewing a program; moves cursor to the left. It is used to scroll through setting selections.

Right Arrow - Used to select individually editable items when viewing a program; moves cursor to the right. It is used to scroll through setting selections and moves the zoom window right when in graphics mode.

End - This button generally moves the cursor to the bottom-most item on the screen. In editing, this is the last block of the program.

ALPHA KEYS

The alpha keys allow the user to enter the letters of the alphabet along with some special characters. Some of the special characters are entered by first pressing the “Shift” key.

Shift - The shift key provides access to additional characters on the keyboard. The additional characters are seen in the upper left of some of the alpha and



number keys. Pressing Shift and then the character will enter that character on the data entry line. When entering text, UPPER CASE is the default, to enter lower case characters, press and hold the Shift key.

When a control has a fifth axis installed, the B axis is selected for jogging by pressing the Shift button and then the +/-A jog keys.

EOB - This is the End-Of-Block character. It is displayed as a semicolon (;) on the screen and it signifies the end of a program line.

() - Parentheses are used to separate CNC program commands from user comments. They must always be entered as a pair. Note: Any time an invalid line of code is received through the RS-232 port while receiving a program, it is added to the program between parenthesis.

/ - The right slash is used in the Block Delete feature and in Macro expressions. If this symbol is the first symbol in a block and a Block Delete is enabled, then that block is ignored at run time. The symbol is also used for division (divide by) in macro expressions (see the Macro section).

[] - Square brackets are used in macro functions. Macros are an optional software feature.

MODE KEYS

Mode keys change the operational state of the CNC machine tool. Once a mode button is pressed, the buttons in the same row are made available to the user. The current mode is always displayed at the top center of the display.

EDIT - Selects edit mode. This mode is used to edit programs in control's memory. Edit mode provides two editing panes: one for the currently active program, and another for background editing. Switch between the two panes by pressing the EDIT key. Press F1 to access help pop-up menus.

Insert - Pressing this button will enter commands into the program in front of the cursor. This button will also insert the text from the clipboard to the current cursor location, and is also used to copy blocks of code in a program.

Alter - Pressing this button will change the highlighted command or text to the newly entered commands or text. This button will also change the highlighted variables to the text stored in the clipboard, or move a selected block to another location.

Delete - Deletes the item that the cursor is on, or deletes a selected program block.

Undo - Undoes up to the last 9 edit changes, and deselects a highlighted block.

MEM (Memory) - Selects the memory mode. The screen displays the active program and other information necessary when making a part.



Single Block - Turns single block on or off. When single block is on, only one block of the program is executed, for every press of Cycle Start.

Dry Run - This is used to check actual machine movement without cutting a part. (See the Dry Run section in the Operation Chapter)

Opt Stop (Optional Stop) - Turns on and off optional stops. Also see G103 in the G-Code chapter.

When this feature is ON and an M01 (optional stop) code is programmed, the machine will stop when it reaches the M01. The machine will continue once Cycle Start is pressed. However, depending on the look-ahead function (G103), it may not stop immediately (See block look ahead section). In other words, the block look-ahead feature may cause the Optional Stop command to ignore the nearest M01.

If the Optional Stop button is pressed during a program it will take effect on the line after the highlighted line when the Opt Stop button is pressed.

Block Delete - Turns On/Off block delete function. Blocks with a slash ("/") as the first item are ignored (not executed) when this option is enabled. If a slash is within a line of code, the commands after the slash will be ignored if this feature is enabled. Block Delete will take effect two lines after Block Delete is pressed, except when cutter compensation is used, in this case, block delete will not take effect until at least four lines after the highlighted line. Processing will slow down for paths containing block deletes during high-speed machining. Block Delete will stay active when power is cycled.

MDI/DNC - MDI mode is the "Manual Data Input" mode where a program can be written but it is not entered into memory. DNC mode "Direct Numeric Control", allows large programs to be "drip fed" into the control so it can be executed (See DNC mode section).

Coolnt (Coolant) - Turns the optional coolant on and off.

Orient Spindle - Rotates the spindle to a given position and then locks the spindle. Can be used during setup to indicate parts.

ATC FWD / REV - Rotates the tool turret to the next / previous tool. To load a specific tool into the spindle, enter MDI or hand jog mode, type a tool number (T8) and press ATC FWD or ATC REV.

Hand Jog - Selects axis jogging mode .0001, .1 - 0.0001 inches (metric 0.001mm) for each division on the jog handle. For dry run, .1 inches/min.

.0001/.1, .001/1., .01/10., .1/100. - The first number (top number), when in inch mode, selects that amount to be jogged 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 0.001mm). The second number (bottom number) is used for dry run mode and is used to select the speed feedrate and axis motions.



Zero Ret (Zero Return) - Selects Zero Return mode, which displays axis location in four different categories, they are; Operator, Work G54, Machine and Dist (distance) to go. Press POSIT to switch between the categories.

All - Returns all axes to machine zero. This is similar to Power Up/Restart except a tool change will not occur. This can be used to establish the initial axes zero position.

Origin - Sets selected displays and timers to zero.

Singl (Single) - Returns one axis to machine zero. Press the desired axis letter and then press the Singl Axis button. This can be used to move a single axis to the initial axis zero position.

HOME G28 - Returns all axes to zero in rapid motion. Home G28 will also home a single axis in the same manner as if an axis letter is entered and the home G28 button is pressed. CAUTION! There is no warning message to alert the operator of any possible collision. For example, if the Z-axis is in amongst parts, when X or Y is zeroed, a crash can result.

List Prog (List Programs) - Controls all loading and saving of data in the control.

Select Prog - Makes the highlighted program the active program. Note: The active program will have an "A" preceding it in the program list. Manage multiple programs by pressing WRITE/ENTER to place a check mark next to the desired programs, then press F1 to choose a function.

Send - Transmits programs out the RS-232 serial port.

Recv - Receives programs from the RS-232 serial port.

Erase Prog - Erases the cursor selected program in List Prog mode or the entire program when in MDI mode.

NUMERIC KEYS

The numeric keys give the user the ability to enter numbers and a few special characters into the control.

Cancel - The Cancel key is used to delete the last character entered.

Space - Used to format comments placed into programs or in the message area.

Write / Enter - General purpose enter key.

- (Minus sign)- Used to enter negative numbers.

. (Decimal Point)- Used for decimal precision.



SCREEN CAPTURE

The control can capture and automatically save a snapshot of the current screen to an attached USB device or the hard drive. If no USB device is connected and the machine does not have a hard drive, no image will be saved.

Press SHIFT then F1 to capture the screen using the default filename 'snapshot.bmp'. This will overwrite any screen capture previously taken using the default name.

Optionally, you can enter a filename on the input line before capturing the screen. The control adds the *.bmp file extension automatically.

DATE AND TIME

The control contains a clock and date function. To view the time and date, press the CRNT COMDS key, then Page Up or Page Down until the date and time appears.

To make adjustments, press Emergency Stop, type the current date (in MM-DD-YYYY format) or current time (in HH:MM format), and press WRITE/ENTER. Reset Emergency Stop when finished.

TABBED HELP / CALCULATOR FUNCTION

Press HELP/CALC to display the tabbed help menu. If pressing HELP/CALC calls a pop-up help menu, press HELP/CALC again to access the tabbed menu. Navigate tabs using the Cursor Arrow keys. Press WRITE/ENTER to select tabs, and press CANCEL to back up one tab level. The main tab categories and their sub-tabs are described here:

Help

The on-screen help system includes content from the entire operator's manual. Selecting the Help tab displays the table of contents. Highlight a topic using the Cursor Arrow keys and press WRITE/ENTER to view the topic contents. Select from subtopic menus in the same way.

Scroll through the page using the jog handle or the Up/Down Cursor Arrow keys. Use the Left/Right Cursor Arrow keys to move to the next topic. Press HOME to return to the main table of contents.

Press F1 to search manual contents, or press CANCEL to exit the Help tab and select the Search tab.

Search

Use the Search tab to look for help content by keyword. Type your search term in the text field and press F1 to execute the search. The results page displays topics that contain your search term; highlight a topic and press WRITE/ENTER to view.



Drill Table

Displays a drill size table featuring decimal equivalents and tap sizes.

Calculator

Calculator functions are available under the third Help tab. Select the calculator from the lower tabs and press WRITE/ENTER to use it.

All of the Calculator functions will do simple add, subtract, multiply, and divide operations. When one of the functions is selected, a calculator window appears with the possible operations (LOAD, +, -, *, and /). LOAD is initially highlighted, and the other options can be selected with the left and right cursor arrows.

Numbers are entered by typing them in and pressing WRITE/ENTER. When a number is entered and LOAD is selected, that number will be entered into the calculator window directly. When a number is entered when one of the other functions (+ - * /) is selected, that calculation will be performed with the number just entered and any number that was already in the calculator window. The calculator will also accept a mathematical expression such as $23*4-5.2+6/2$, evaluating it (doing multiplication and division first) and placing the result, 89.8 in this case, in the window.

Note that data cannot be entered in any field where the label is highlighted. Clear data in other fields until the label is no longer highlighted in order to change the field directly.

Function Keys: The function keys can be used to copy and paste the calculated results into a section of a program or into another area of the Calculator feature.

F3: In EDIT and MDI modes, F3 will copy the highlighted triangle/circular milling/tapping value into the data entry line at the bottom of the screen. This is useful when the calculated solution will be used in a program.

In the Calculator function, pressing F3 copies the value in the calculator window to the highlighted data entry for Trig, Circular or Milling/Tapping calculations.

F4: In the Calculator function, this button uses the highlighted Trig, Circular or Milling/Tapping data value to load, add, subtract, multiply, or divide with the calculator.

Trigonometry Help Function

The Trigonometry calculator page will help solve a triangular problem. Enter the lengths and the angles of a triangle and when enough data has been entered, the control will solve for the triangle and display the rest of the values. Use the Cursor Up/Down buttons to select the value to be entered with WRITE/ENTER. For inputs that have more than one solution, entering the last data value a second time will cause the next possible solution to be displayed.



HELP (MEM)		O00000 N00000000
CALCULATOR		
[0.000000000]		
LOAD + - × /		
(MACHINE) Y Y Z	ANGLE 1	40.000
	ANGLE 2	72.000
	ANGLE 3	68.000
SIDE 1 SIDE 2 SIDE 3	SIDE 1	10.0000
	SIDE 2	14.7958
	SIDE 3	14.4244
F3 copies calculator value to highlighted field in this or other calculator screens. F3 also copies calculator value to the data entry line of edit screens. F4 copies highlighted data to the calculator field.		

Circular Interpolation Help

The Circular calculator page will help solve a circle problem. You enter the center, radius, angles, start and end points; when enough data has been entered, the control will solve for the circular motion and display the rest of the values. Use the Cursor Up/Down buttons to select the value to be entered with Write. In addition, it will list alternate formats that such a move could be programmed with a G02 or G03. The formats can be selected using the Cursor Up/Down buttons, and press F3 to import the highlighted line into the program being edited.

HELP (MEM)		O00000 N00000000
CALCULATOR		
[0.000000000]		
LOAD + - × /		
(MACHINE) X Y Z	CENTER X	13.0000
	CENTER Y	20.0000
	START X	4.0000
END X END Y RADIUS ANGLE DIRECTION	START Y	10.0000
	END X	7.0000
	END Y	32.0416
16 19. J10. 16 R13.4536 16 19. J10 G91 G2 X3. Y22.0416 R13.4536		

For inputs that have more than one solution, entering the last data value a second time will cause the next possible solution to be displayed. To change the CW value to the CCW value, highlight the CW/CCW column and press WRITE/ENTER.



Circle-Line Tangent Calculator

This feature provides the ability to determine points of intersection where a circle and a line meet as tangent. Enter two points, A and B, on a line and a third point, C, away from that line. The control will calculate the point of intersection. The point is where a normal line from point C will intersect with the line AB, as well as the perpendicular distance to that line.

CIRCLE-CIRCLE TANGENT

CIRCLE1 X	5.0000
CIRCLE1 Y	6.0000
RADIUS 1	4.0000
CIRCLE2 X	0.0000
CIRCLE2 Y	0.0000
RADIUS 2	[2.0000]

TANGT A X	1.3738
	Y 7.6885
TANGT B X	7.3147
	Y 2.7378
TANGT C X	-1.8131
	Y 0.8442
TANGT D X	1.1573
	Y -1.6311

Type: STRAIGHT
.....
Use F and T to form G-code.
F1 for alternate solution

CIRCLE-LINE TANGENT

POINT A X	5.0000
	Y 3.0000
POINT B X	1.0000
	Y 4.0000
POINT C X	0.0000
	Y [0.0000]

RADIUS	4.1231
TANGT PT X	1.0000
TANGT PT Y	4.0000

Circle-Circle Tangent Calculator

This feature provides will determine points of intersection between two circles or points. The user provides the location of two circles and their radii. The control then calculates the intersection points that are formed by lines tangent to both circles. Note that for every input condition (two disjointed circles), there are up to eight intersection points. Four points are from drawing straight tangents and four points by forming cross tangents. The F1 key is used to toggle between the two diagrams. When "F" is pressed, the control will prompt for the from and to points (A, B, C, etc.) that specify a segment of the diagram. If the segment is an arc, the control will also prompt for C or W (CW or CCW). G code is then displayed at the bottom of the screen. When "T" is entered, the previous to point becomes the new From point and the control prompts for a new To point. To enter the solution (line of code), go to MDI or Edit and press F3, as the G-code is already on the input line.

Drill/Tap Chart

A Drill and Tap Chart is available in the tabbed help menu.

SPINDLE WARM-UP PROGRAM

If any spindle has been idle for more than 4 days, it must be thermally cycled prior to operation. This warm-up will prevent possible overheating of the spindle due to the settling of lubrication. A 20-minute warm-up program (number 002020) is supplied with the machine which will bring the spindle up to speed slowly, allowing the spindle to thermally stabilize. This program can be used daily for spindle warm-up prior to high-speed use.



COOLANT LEVEL GAUGE

The coolant level is displayed in the upper right of the screen in MEM mode, or on the CURNT COMDS screen. A vertical bar shows the status of the coolant. The display will flash when the coolant reaches a point that could cause intermittent coolant flow.

WORK BEACON

The beacon light provides quick visual confirmation of the machine's current status. There are four different beacon states:

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.

M-NET

M-Net is a network protocol for communication between machine tools, robots, and shop computers over a Local Area Network (LAN) and the Internet. Commercially developed applications use M-Net to send, receive, compile and report data between devices on the network.

Software vendors can provide more information about their M-Net applications.

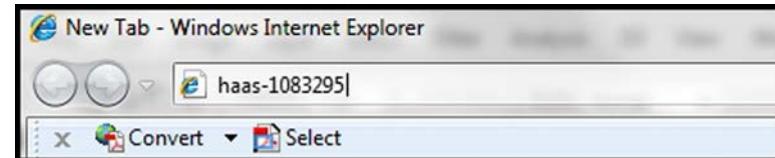
ALARM NOTIFICATIONS

Haas machines include a basic application to send an alert to an email address or cellular telephone when an alarm occurs. Setting up this application requires some knowledge about your network; ask your system administrator or Internet Service Provider (ISP) if you do not know the correct settings.

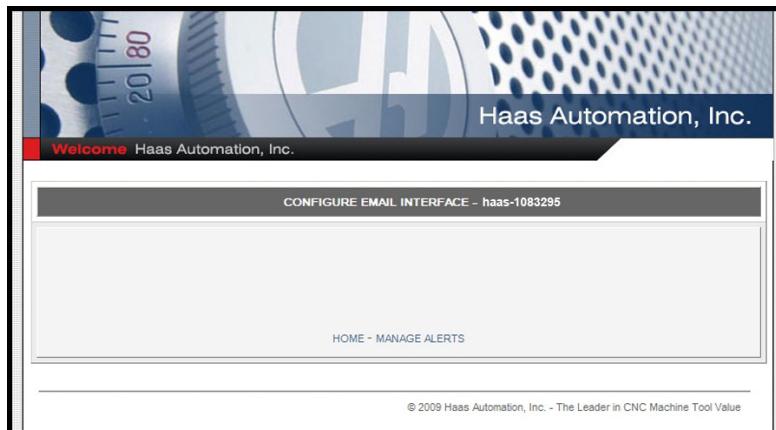
Before setting up alerts, make sure the machine has an established connection to your Local Area Network, and that Setting 900 defines a unique network name for the machine. This feature requires the Ethernet option and software version 18.01 or later.

Setting up Alerts

1. Using an Internet browser on another device connected to the the network, type the machine's network name (Setting 900) into the browser address bar and press Enter.



2. A popup message may appear with a request to set a cookie in your browser. This will happen each time you access the machine using a different computer or browser, or after an existing cookie has expired. Click "OK" to proceed.
3. The home screen displays, with the setup options at the bottom of the screen. Click "Manage Alerts".





4. At the “Manage Alerts” screen, enter the email address and/or cellular phone number where you want to receive alerts. If entering a cellular phone number, select your carrier from the pull-down menu under the cell number field. Click the “submit changes” button when finished.

The screenshot shows the "MANAGE ALERTS - haas-1083295" page. It has fields for "Email alerts to:" and "Text alert cell number:", both with input boxes. Below these is a dropdown menu labeled "Cellular carrier" with the option "Other - enter full URL with cell number". A "SUBMIT CHANGES" button is at the bottom. Navigation links include "HOME - CONFIGURE EMAIL INTERFACE" and "HOME - MANAGE ALERTS". The footer copyright is "© 2009 Haas Automation, Inc. - The Leader in CNC Machine Tool Value".

Note: If your cellular carrier is not listed in the menu, ask your carrier to provide your account's email address through which you can receive text messages. Enter this address in the email field.

5. Click “Configure Email Interface”.

The screenshot shows the "CONFIGURE EMAIL INTERFACE - haas-1083295" page. It has fields for "DNS IP address:", "SMTP server name:", and "SMTP server port:" (set to 25), each with input boxes. Below these is a field for "Authorized EMAIL account:" with an input box. A "SUBMIT CHANGES" button is at the bottom. Navigation links include "HOME - CONFIGURE EMAIL INTERFACE" and "HOME - MANAGE ALERTS". The footer copyright is "© 2009 Haas Automation, Inc. - The Leader in CNC Machine Tool Value".



6. Fill in the fields with your email system's information. Ask your system administrator or ISP if you do not know the correct values. Click the "submit changes" button when finished.

Note: Haas Automation service personnel cannot diagnose or repair problems with your network.

- a. In the first field, enter the IP address for your domain name server (DNS).
 - b. In the second field, enter your simple mail transfer protocol (SMTP) server name.
 - c. The third field, SMTP server port, is already populated with the most common value (25). Change this only if the default setting does not work.
 - d. In the last field, enter an authorized email address, which the application will use to send the alert.
7. Press Emergency Stop to generate an alarm to test the system. An email or text message should arrive at the designated address or telephone number with details about the alarm.

SPINDLE VIBRATION DETECTION

An unbalanced tool in the spindle generates an alarm if it causes vibration beyond parameter-controlled limits. Make sure the tool is correctly balanced and that both the tool holder and tool are undamaged.

MEMORY LOCK KEY SWITCH

Locks memory to prevent accidental or unapproved program editing by unauthorized personnel. It can also be used to lock settings, parameters, offsets and macro variables.

OPTIONS

200 Hour Control Option Try-Out

Options that normally require a unlock code to activate (Rigid Tap, Macros, etc.) are activated and deactivated as desired by entering the number "1" instead of the unlock code to turn it on. Enter a "0" to turn off the option. An option activated in this manner is automatically deactivated after a total of 200 power-on hours. Note that the deactivation only occurs when power to the machine is turned off, not while it is running. An option can be activated permanently by entering the unlock code. Note that the letter "T" will be displayed to the right of the option on the parameter screen during the 200 hour period.

To enter a 1 or 0 into the option, press the Emergency Stop button in and turn setting 7 (Parameter Lock) off. When the option reaches 100 hours the machine will give an alarm warning that the try out period is almost over. To permanently activate an option contact your dealer.



USB and Ethernet

Store and transfer data between your Haas machine(s) and a network. Program files are easily transferred to and from memory, and allows DNC of large files.

Rigid Tapping

Synchronized tapping eliminates the need for expensive, floating tap holders, and prevents lead-thread distortion and start-thread pullout.

Macros

Create subroutines for custom canned cycles, probing routines, operator prompting, math equations or functions, and family-of-parts machining with variables.

Rotation and Scaling

Use rotation in conjunction with work offset probing to speed workpiece setup, or to rotate a pattern to another location or around a circumference, etc. Use scaling to reduce or enlarge a toolpath or pattern.

Spindle Orientation

The Spindle Orientation option allows spindle positioning to a specific, programmed angle, using the standard spindle motor and the standard spindle encoder for feedback. This option provides inexpensive, accurate (0.1 degree) positioning.

High Speed Machining

High speed machining makes it possible for an increase in the removal rate of material, improve surface finish, and reduce cutting forces which will reduce machining costs and extend the life of the tools.

High Speed Machining is most often required for the machining of smoothly sculpted shapes as is typical of mold making. The Haas High Speed Machining option increases the amount of lookahead to 80 blocks and allows full speed (500 inches per minute) blending of feed strokes.

It is important to understand that high speed machining works best with smoothly blended shapes where the feed rate can remain high through the blend of one stroke to the next. If there are sharp corners, the control will always need to slow down or corner rounding will occur.

The affect that blending of strokes can have on feed rate is always to slow down motion. The programmed feed rate (F) is thus a maximum and the control will sometimes go slower than that in order to achieve the required accuracy.

Too short of a stroke length can result in too many data points. Check how the CAD/CAM system generates data points to insure that it does not exceed 1000 blocks per second.

Too few data points can result in either "facetting" or blending angles which



are so great that the control must slow down the feed rate. Facetting is where the desired smooth path is actually made up of short, flat, strokes that are not close enough to the desired smoothness of the path.

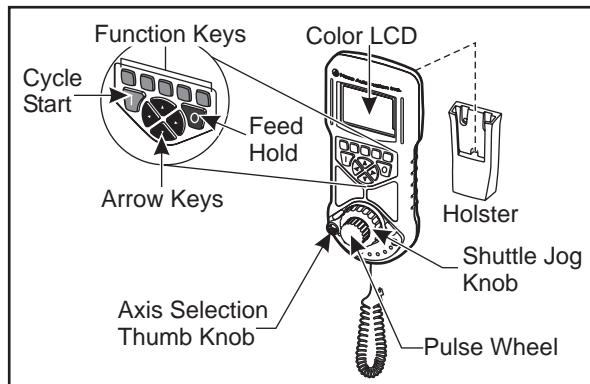
High Speed Tooling – The tool holders should be an AT-3 or better with a nylon back-up screw. The tolerances maintained in the AT-3 design are the minimum that would be recommended for a high speed process. The nylon back-up screw increases collet grip on the tool and creates a better seal to aid in coolant transfer.

Use single angle collet chucks and collets for best grip and concentricity. These collet systems are made up of a long single angle located in the holder. The angle per side should be eight degrees or less for best results. Avoid double angle collet systems when maximum rigidity and close tolerance are dictated. It is recommended that minimum engagement of 2/3 of the full length of the bore in the double split single angle collet. However for better results 3/4 to full engagement is preferred if possible.

High Intensity Lighting - Auxiliary lights provide bright illumination of the work area. The lights operate automatically when doors open and close, or can be activated manually using the switch on the side of the control pendant. Turn the switch on and the lights will turn on when the door is opened and off when the door is closed. Set the switch to off and the lights will not turn on when the door is opened. See setting 238.

REMOTE JOG HANDLE

The Enhanced Color Remote Jog Handle (RJH) features a color liquid crystal display (LCD) and controls for increased functionality. It also features a high-intensity LED flashlight.



Refer to the section on offsets and machine operation for more information on those topics.

LCD – Displays machine data and the RJH-E/C interface.



Function Keys (F1-F5) - Variable-function keys. Each key corresponds to a label at the bottom of the LCD screen. Pressing a function key will perform or toggle the corresponding menu. Toggled functions are highlighted when on.

Cycle Start - Starts programmed axis motion.

Feed Hold - Stops programmed axis motion.

Arrow Keys - Used to navigate between menu fields (up/down) and to select pulse jog rates (left/right).

Pulse Wheel - Jogs a selected axis by the selected increment. Works like the jog handle on the control.

Shuttle Jog - Rotates up to 45 degrees CW or CCW from center, and returns to center when released. Used to jog axes at variable speeds. The farther the shuttle jog is rotated from the center position, the faster the axis moves. Allow the knob to return to the center position to stop motion.

Axis Select - Used to select any of the available axes for jogging. The selected axis is displayed at the bottom of the screen. The far right position of this selector accesses the auxiliary menu.

Removing the unit from the cradle/holster powers it on and turns over jogging control from the pendant to the Remote Jog Handle (The hand wheel on the pendant is disabled).

NOTE: The pendant must be in Hand Jog mode (Setup).

Place the Remote Jog Handle back in its cradle/holster to power it off and return jogging control to the pendant.

The pulse knob and shuttle knob function as scrollers to change the value of a user-definable field such as tool offset, length, wear, etc.

Built-in “Panic” Function — Press any key during axis motion to instantly stop the spindle and all axis motion. Pressing Feed Hold while the spindle is in motion and the control is in Handle Jog mode will stop the spindle. The message “**Button pressed while axis was moving—Reselect Axis**” appears on the display. Move the axis selection knob to a different axis to clear.

If the axis selection knob is moved while the shuttle jog is turned, the message “**Axis selection changed while axis was moving—Reselect Axis**” appears on the display, and all axis motion stops. Move the axis selection knob to a different axis to clear the error.

If the shuttle jog knob is turned from its centered position when the Remote Jog Handle is removed from its cradle/holster, or when the control mode is changed to a mode with motion (for instance, from MDI to Handle Jog mode), the message “**Shuttle off center—No Axis selected**” appears on the display and no axis motion will occur. Move the axis selection knob to clear the error.

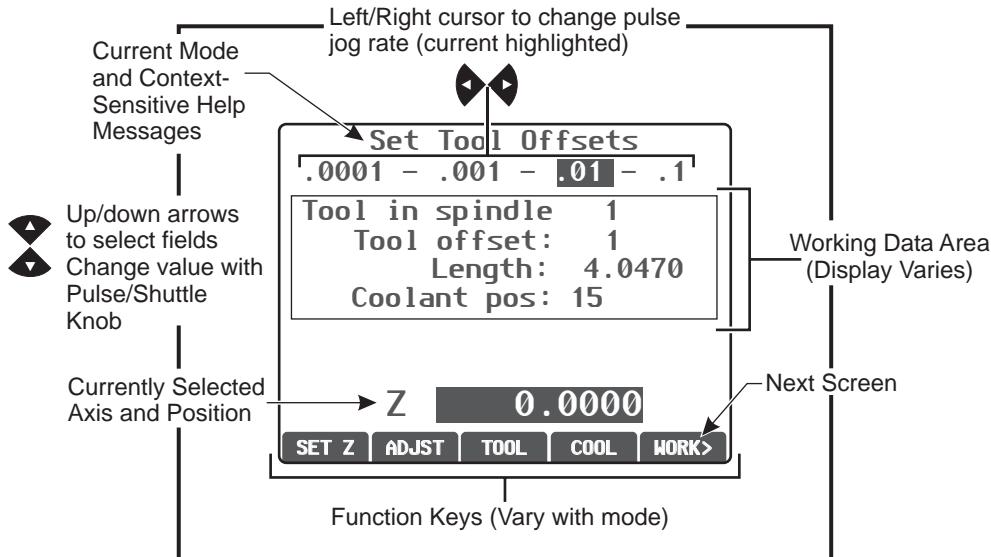


If the pulse jog knob is rotated while the shuttle jog knob is in use, the message “**Conflicting jog commands—Reselect Axis**” appears on the Remote Jog Handle display, and all axis motion stops. Move the axis selection knob to a different axis to clear the error, then back to reselect the previously selected axis.

NOTE: If any of the above errors fail to clear when the axis selection knob is moved, there may be a problem with the shuttle jog knob. Contact Haas service for repair/replacement.

If contact between the Remote Jog Handle and the control is broken for any reason (cable cut or disconnected, etc.), all axis motion stops. When reconnected, the message “**RJH / Control Communication Fault—Reselect Axis**” appears on the Remote Jog Handle display. Move the axis selection knob to clear the error. If the error does not clear, place the unit in its cradle/holster, wait for it to power off, and then remove from the cradle/holster.

RJH Menus



RJH Manual Jogging

This menu contains a large display of the current machine position. Turning the shuttle jog or pulse knob will move the currently selected axis by the currently selected jog increment. Change the jog increment with the left/right arrow keys. Press OPER, WORK, MACH, or TO GO to change the coordinate system (current highlighted). To zero the operator position, press the function key under OPER to select the position, then press the function key again (it now reads ZERO).



Manual Jogging

.0001 - .001 - .01 - .1

X:	0.0000	in
Y:	0.0000	in
Z:	0.0000	in

OPER WORK MACH TO GO TOOL>

Manual Jogging Display

RJH Tool Offsets

Use this menu to set and check tool offsets. Select fields using the function keys and change values using the pulse or shuttle knob. Select axes using the thumb knob. The axis line (at the bottom of the display) must be highlighted to jog that axis. Press ENTER to set the current Z-axis position into the offset table. To make adjustments to table values, press ADJST, use the pulse or shuttle knob to select the amount to increase or decrease the value (use the left and right arrows to change the increment), then press ENTER to apply the adjustment. Press TOOL to change the tools, and press COOL to change the coolant position for the selected tool.

CAUTION: Stay clear of the spindle when changing tools.

Set Tool Offsets

.0001 - .001 - .01 - .1

Tool in spindle	1
Tool offset:	1
Length:	4.0470
Coolant pos:	15

Z 0.0000

SET Z ADJST TOOL COOL WORK>

Setting Tool Offsets Display

RJH WORK OFFSETS

Press WK CS to change the work offset G code. Manually jog the selected axis with the shuttle or pulse knob when the axis field at the bottom of the screen is highlighted. Press SET to set the current position of the current axis into the work offset table. Move the axis selector to the next axis and repeat the process to set that axis. To make adjustments to a set value, move the axis



selector to the desired axis. Press ADJST and use the pulse or shuttle knob to increase or decrease the adjustment value, then press ENTER to apply the adjustment.

Set Work Offsets			
.0001	-.001	-.01	-.1
Work CS G52			
X:	0.0000		
Y:	0.0000		
Z:	0.0000		
X 0.0000			
SET X	ADJST	WK CS	JOG>

Setting Work Offsets Display

Introduction

Auxiliary Menu

The RJH auxiliary menu features controls for machine coolant and the RJH flashlight. Access the menu by moving the axis selector to the far right position (indicated by a page icon molded into the RJH case). Toggle the available features by pressing the corresponding function key.

Auxiliary Menu			
Flash Light: OFF Coolant: OFF			
LIGHT	CLNT	UTIL>	AUX>

Utility Menu			
RJH-C Firmware Version: 0.01g			
RJH-C Font Version: RJH-C			
RJH-C Font ID 5			
Main Build Version: VER M16.02x			
AUX>			

Auxiliary Menu

Utility Menu

UTIL Menu

Accesses information about the current configuration of the RJH. This information is used for diagnostic purposes by service technicians. Press AUX to return to the Auxiliary Menu.

Program Display (Run Mode)

This mode displays the currently running program. Enter run mode by pressing MEM or MDI on the control pendant. The tab options at the bottom of the screen provide controls for coolant on/off, single block, optional stop, and block delete. Toggled commands such as COOL will appear highlighted when turned



on. The CYCLE START and FEED HOLD buttons function just as the buttons on the pendant. Return to jogging by pressing HAND JOG on the control pendant, or place the Remote Jog Handle back in the cradle/holster to continue running the program from the pendant.



OPERATION

MACHINE POWER-UP

Turn the machine on by pressing the Power-On button on the pendant.

The machine will go through a self test and then display either the Messages screen, if a message was left, or the Alarms screen. In either case the lathe will have one or more alarms present (102 SERVOS OFF).

Follow the directions in the ‘mode status box’ on the left side of the display. Generally the door will need to be cycled and E-STOP pressed and cleared before the ‘Power Up’ or ‘Auto All Axes’ operations become available. For more information on safety lock features, refer to the “Safety” section in this manual.

Press the Reset button to clear each alarm. If an alarm cannot be cleared the machine may need servicing; if this is the case call your dealer.

Once the alarms are cleared the machine needs a reference point to start all operations from; this point is called “Home”. To home the machine, press the Power-Up Restart button.

**WARNING! Automatic motion will begin once this button is pressed.
Keep away from the inside of the machine and the tool changer.**

Note that pressing the Power-Up/Reset button will automatically clear alarm 102 if it was present.

After home is found the Current Commands page is displayed, and the machine is now ready to run.

PROGRAMMING INTRODUCTION

Manual Data Input (MDI)

Manual Data Input (MDI) is a means to command automatic CNC moves without using a formal program.

Press the MDI/DNC key to enter this mode. Programming code is entered by typing in the commands and pressing Enter at the end of each line. Note that an End of Block (EOB) will be automatically inserted at the end of each line.

PROGRAM - MDI

```
G97 S1000 M03 ;
G00 X2. Z0.1 ;
G01 X1.8 Z-1. F12 ;
X1.78 ;
X1.76 ;
X1.75 ;
```



To edit the MDI program use the keys to the right of the Edit button. Cursor to the point that is changing, then the different edit functions can be used.

To enter an additional command to the line, enter the command and press Enter.

To change a value use the arrow buttons or the jog handle to highlight the command, enter the new command and press Alter.

To delete a command, highlight the command and press Delete.

The Undo key will reverse changes (up to 9 times) that have been made to the MDI program.

An MDI program can be saved to the memory of the control. To do so cursor to the beginning of the program (or press Home), enter a program name (programs need to be named using the format Onnnnn; the letter "O" followed by up to 5 numbers) and press Alter. This will add the program to the list of programs and clear the MDI page. To re-access the program, press List Prog and select it.

The data in MDI is retained after exiting MDI mode and when the machine is turned off.

To clear the current MDI commands press the Erase Prog button.

NUMBERED PROGRAMS

To create a new program, press LIST PROG to enter the program display and the list of programs mode. Enter a program number (Onnnnn) and press Select Prog key or Enter. If the program exists, it will be selected. If it does not yet exist, it will be created. Press Edit to show the new program. A new program will consist of only the program name and an End of Block (;).

NOTE: Using O09XXX numbers when creating new programs is not recommended. Macro programs often use numbers in this block, and overwriting them may cause machine functions to stop working. (Example: overwriting O09876 will cause G47 operations (engraving) to malfunction).

Numbered programs are retained when the machine is turned off.

Basic Editing of MDI and Numbered Programs

The only difference between an MDI program and a numbered program is the O code. To edit an MDI program, simply press MDI. To edit a numbered program, select it, then press Edit.

The program edit mode includes a type in the program data and press enter. Program data falls into three categories: addresses, comments or EOBS.



EDIT: EDIT	
PROGRAM EDIT 000741	(CYCLE START TO SIMULATE)
<pre>G00 X0 Z0.1 ; G74 Z-0.345 F0.03 K0.1 ; ; G00 X2. Z0.1 ; G74 X1. Z-4. I0.2 K0.75 D255 ; G00 X3. Z0.1</pre>	

To add program code to the existing program, highlight the code that the additional code will go in front of, type in the data and press the Insert key. More than one code, such as X, Y, and Z, can be entered before pressing Insert.

Address data is a letter followed by a numeric value. For example: G04 P1.0. The G04 commands a dwell (pause) and the P1.0 is the length (1 second) of the dwell.

Comments can be either alpha or numeric characters, but must be prefaced with parentheses. For example: (1 second dwell). Comments can be a maximum of 80 characters.

End of Blocks are entered by pressing the EOB button and are displayed as a semicolon (;). These are used like a carriage return at the end of a paragraph. In CNC programming an EOB is entered at the end of a string of program code.

An example of a line of code using the three types of commands would be:

G04 P1. (1 second dwell);

There is no need to put any symbols or spaces between the commands. A space is automatically placed between elements for ease of reading and editing.

To alter characters, highlight the desired portion of the program using the arrow keys or the jog handle, enter the replacement code, and press Alter.

To remove characters or commands, highlight the text and press Delete.

There is no save command, as the program is saved as each line is entered.

Converting an MDI program to a numbered program

An MDI program can be converted to a numbered program and added to the list of programs. To do so, cursor to the beginning of the program (or press Home), enter a program name (programs need to be named using the format Onnnnn; the letter "O" followed by up to 5 numbers) and press Alter. This will add the program to the list of programs and clear MDI. To re-access the program, press List Prog and select it.

Searching the program

While in MDI, EDIT or MEM mode, the cursor up and down keys can be used to search the program for specific codes or text. To search for particular character(s), enter the character(s) on the data entry line (i.e. G40) and press



the cursor up or down keys. The cursor up key will search for the entered item backwards (towards the start of the program) and the cursor down key will search forward (towards the end of the program).

Deleting Programs

To delete a program, press LIST PROG. Use the cursor up or down keys to highlight the program number, then press the ERASE PROG key. Press Y at the prompt to confirm the deletion or N to back out. Or type in the program number and press the ERASE PROG key; however, use this option carefully as there is no Y/N prompt and the program will be immediately deleted.

Select ALL at the end of the list and press the ERASE PROG key to delete all the programs in the list. There are some important programs that are included with the machine; they are O02020 (spindle warm up), O09997 (Visual Quick Code), and O09876 (engraving font file). Save these programs to a memory device or PC before erasing all programs. Turn on Setting 23 to protect O09XXX programs from deletion.

NOTE: The UNDO key will not recover programs that are deleted.

Renaming Programs

A program number can be changed by entering a new number in Edit mode and pressing the Alter button. Take care not to accidentally overwrite important programs such as those listed in the previous section.

Maximum Number of Programs

If the maximum number of programs (500) is in the control's memory, the message "DIR FULL" will display and new programs cannot be created.

Program Selection

Enter the program directory by pressing "List Prog"; this will display the stored programs. Scroll to the desired program and press "Select Prog" to select the program. Entering the program name and pressing "Select Prog" will also select a program.

Once "Select Prog" is pressed, the letter "A" appears next to the program name. This is the program that will be run when the mode is changed to MEM and CYCLE START is pressed. It is also the one displayed on the EDIT display.

When in MEM mode, another program can be selected and displayed quickly by entering the program number (Onnnnn) and pressing the Up/Down arrow, or **F4**.

The selected program will remain selected after the machine is turned off.

Loading Programs to the CNC Control

Numbered programs can be copied from the CNC control to a personal computer (PC) and back again. It is best if the programs are saved to a file that ends in ".txt". That way, they will be recognized by any PC as a simple text file.

Programs can be transferred by many different methods such as USB, and RS-232. Settings, offsets and macro variables can be transferred between the CNC and a PC in a similar manner.

Corrupted program data, if received by the CNC, is converted into a comment, stored in the program, and an alarm is generated. However, the data will still be loaded in to the control.

USB / Hard Drive / Ethernet Device Manager

The Haas control incorporates a device manager which shows the available memory devices on the machine in a tabbed menu.

Enter the Device Manager by pressing "List Prog". Navigate the tabbed menu using the arrow keys to select the appropriate device tab and press Enter.

When browsing a list of programs within a device tab, use the up/down arrow keys to highlight programs and press Enter to add the highlighted program to the selection.

Note: External USB hard drives only work when they are formatted as FAT or FAT32. NTFS formatted devices will not work. To find out what your devices is formatted as; connect it to your PC, right click on the drive in Windows Explorer and select Properties.

The following example shows the directory for the USB device. The selected program in memory is shown with an "A". The selected file will also show in the active program display.

Tabbed Menu Navigation	Active Program	Active Tab	Highlighted Program
Cursor Arrows: Navigate Tabs WRITE/ENTER: Select a Tab CANCEL: Move Back One Tab Level	Program Selection Cursor Arrows: Move Selection cursor WRITE/ENTER: Add Program to Selection (A Check Mark is Placed) SELECT PROG: Makes the Selected Program the Active Program ("A") or Selects Program for FNC INSERT: Create New Folder in Current Directory (Type Folder Name, then Insert) ALTER: Rename Folder or Program	Active Tab MEMORY USB DEVICE HARD DRIVE NET SHARE USB DEVICE 2 CURRENT DIRECTORY: USB DEVICE USB DEVICE Selected Program	Highlighted Program 011133 (WORK ORDER 7) 1153 10-29-07 11:13:25 01234 (WORK ORDER 11) FITTING PROJECT 2 ALL File Size 784 11-12-07 08:20:00 Subdirectory <DIR> Date and Time
Help Mode Press HELP/CALC to Access the Pop-Up Help Menu. Navigate Using Cursor Arrow Keys. Choose Options for Selected Programs (Copy, Delete, Etc.)			Use CURSOR keys to navigate listing and CANCEL to go back to devices. Press HELP for Help listing.
			✓ : FILES IN SELECTION A : ACTIVE PROGRAM (001254)

Navigating Directories

To enter a sub-directory, scroll to the sub-directory and press Enter.

To leave a sub-directory, go to the top of the sub-directory or press Cancel.



Create Directories

Create a new folder by entering a name and pressing “Insert”.

To create a new Sub-directory, go to the directory where the new sub-directory will be located, enter a name and press “Insert”. Sub-directories are displayed with their name followed by <DIR>”.

Copying Files

Highlight a file and press “Enter” to select it. A check mark appears next to the file name.

Navigate to the destination directory with the arrow keys, press “Enter”, and press F2 to copy the file.

Note that files copied from the control’s memory to a device will have the extension “.NC” appended to the file name. However the name can be changed by navigating to the destination directory, entering a new name, and then pressing F2.

Duplicating a File

Press List Prog to access the Device Manager. Select the memory tab. Cursor to the program to duplicate, type a new program number (Onnnnn) and press F2. The highlighted program is duplicated with the new name, and it is made the active program. To duplicate a file to a different device, cursor to the program name and press F2 without entering a new file name. A popup menu lists destination devices. Select a device and press Enter to duplicate the file. To copy multiple files, press Enter to place a check mark at each file name.

File Naming Convention

File names should be kept to a typical eight-dot-three format. For example: program1.txt. However, some CAD/CAM programs use “.NC” as a file type recognition, which is acceptable. File names can also be the same as the program number with no extension, but some PC applications may not recognize the file.

Files developed in the control will be named with the letter “O” followed by 5 digits. For example, O12345.

Renaming

To change the name of a file in USB or the Hard Drive, highlight the file, type a new name and press “Alter”.

Deleting

To delete a program file from a device, highlight the file and press Erase Prog. Delete multiple files by selecting them (press Enter to add a file to the selection and place a check mark next to it; deselect by pressing Enter again), then press Erase Prog to delete all selected files.



On-Screen Help

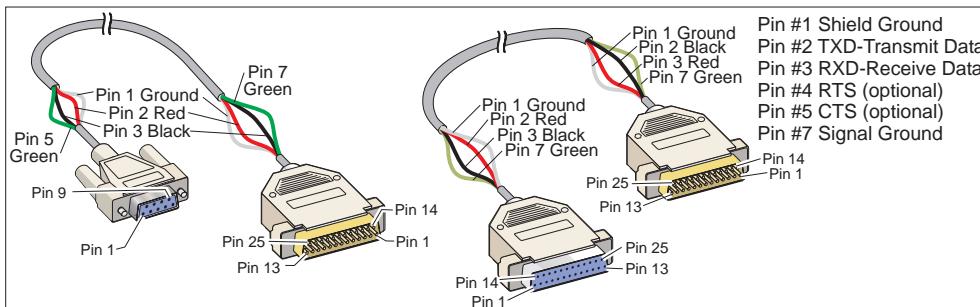
On-Screen help is available by pressing "HELP/CALC". Select functions from the popup menu and press "Enter" to execute, or use the hot key listed. To exit the Help screen, press the "Cancel" button to return to the device manager.

RS-232

RS-232 is one way of connecting the Haas CNC control to another computer. This feature enables the programmer to upload and download programs, settings and tool offsets from a PC.

Programs are sent or received through the RS-232 port (Serial Port 1) located on side of the control box (Not the operator's pendant).

A cable (not included) is necessary to link the CNC control with the PC. There are two styles of RS-232 connections: the 25-pin connector and the 9-pin connector. The 9-pin connector is more commonly used on PCs.



WARNING! One of the biggest causes of electronic damage is a lack of a good ground on both the CNC mill and the computer. The lack of grounding will damage the CNC, or the computer, or both.

Cable length

The following lists baud rate and the respective maximum cable length.

Baud rate	Max. cable length (ft)
19200	50
9600	500
4800	1000
2400	3000

The settings between the Haas control and the other computer must match. To change the settings in the CNC control, go to the Settings page (press Setng/Graph) and scroll to the RS-232 settings (or enter "11" and press the up or down arrow). Use the up/down arrow keys to highlight the settings and the left and right arrow keys to change the values. Press Enter when the proper selection is highlighted.



The settings (and defaults) that control the RS-232 port are:

- | | |
|-----------------------------|---------------------------|
| 11 Baud Rate (9600) | 24 Leader to Punch (None) |
| 12 Parity (Even) | 25 EOB Pattern (CR LF) |
| 13 Stop Bits (1) | 37 Number Data Bits (7) |
| 14 Synchronization Xon/Xoff | |

There are a number of different programs that can link with the Haas control. An example is the Hyper Terminal program that is installed with most Microsoft Windows applications. To change the settings on this program go to the “File” drop down menu at the top left. Choose the “Properties” selection from the menu and then press the “Configure” button. This will open up the port settings; change these to match what is in the CNC control.

To receive a program from the PC, push the LIST PROG key. Move the cursor to the word ALL and push the RECV RS-232 key and the control will receive all main and sub programs until it receives a “%” indicating end of input. All programs sent to the control from the PC must begin with a line containing a single “%” and must end with a line containing a single “%”. Note that when using “ALL”, the programs must have a Haas formatted program number (Onnnnn). If there is not a program number, type in a program number before pushing RECV RS-232 and the program will be stored under that number. Or select an existing program for input and it will be replaced.

To send a program to the PC, use the cursor to select the program and push the SEND RS-232 key. Select “ALL” to send all of the programs in the memory of the control. A setting (Setting 41) can be turned on to add spaces to the RS-232 output and improve the readability of the programs.

Parameters, settings, offsets, and macro variables pages may also be sent individually via RS-232 by selecting the “LIST PROG” mode, selecting the desired display screen, and pushing the SEND key. They can be received by pushing the RECV key and selecting the file on the PC that is to be received.

The file can be viewed on a PC by adding “.txt” to the file name from CNC control. Then open the file on a PC using a program such as Windows Notepad.

If an abort message is received, check the set-up between the mill and the PC and the cable.

File Delete

On the List Prog page, type “DEL <filename>”. Press WRITE and the file will be deleted.

FILE NUMERIC CONTROL (FNC)

A program can be run from its place on the network or from a storage device, such as a USB drive. To run a program from such a location, go to the Device Manager screen (Press List Prog), highlight a program on the selected device and press “Select Prog”. The program will be displayed in the active program pane, and an “FNC” next to the program name in List Prog indicates that it is



the currently active FNC program. Sub-programs can be called using an M98, provided that the subprogram is in the same directory as the main program. Additionally, the subprogram must be named using the Haas naming convention with case sensitivity, e.g. O12345.nc.

CAUTION! The program can be modified remotely, and the change will take affect the next time the program is run. Subprograms may be changed while the CNC program is running.

Program editing is not allowed in FNC. The program is displayed and can be browsed through, but not edited. Editing can be done from a networked computer or by loading the program into memory.

To run a program in FNC:

1. Press List Prog, then navigate to the tabbed menu for the appropriate device (USB, hard drive, Net Share).
2. Cursor down to the desired program and press Select Prog. The program will appear in the Active Program pane and can be run directly from the memory device.

To Exit FNC, highlight the program again and press Select Prog, or select a program in CNC memory.

USING THE HAAS EDITOR (FNC)

The Haas Editor is a program editing application built into the Haas control. In addition the Haas Editor needs either the MainCon board, or the Hard drive/ Enet option (revision 13.03 or later), and parameter 737 Common Switch 5 Bit 9 Enable Haas Editor, must be set to 1.

The Haas Editor provides the same familiar functions as the Advanced Editor, alongside new features to enhance program development on the control, including multiple-document viewing and editing.

LOADING A PROGRAM

1. Highlight a program in the USB, Hard Drive, or Net Share tab of the LIST PROG menu and press SELECT PROG to make it the active program (in the Haas Editor, programs open in FNC, but are editable).
2. With the program loaded, press EDIT to shift focus to the program edit pane.
3. The initial display mode shows the active program on the left, and the program list on the right.



EDITOR File Edit Search Modify

004001.txt

004001
; (WORKING DEMO);
; (24-01-97 TIME, Hr:Min - 13:59);
; (T1-.101 DIA. DRILL);
; (T2-.314 DIA. FACEMILL);
; (T3-.500 DIA. 2 FLT E.M.);
; (T4-.250 DIA. 2 FLT E.M.);
; (T5-.60 DEG. ENGRAVE/CHAMFER TOOL);
; ;
; (W07 P002);
; (CALLS 2ND OP. MILL PROGRAM);
; ;
; (W07 P2003);
; (CALLS 2ND OP. POCKET PROGRAM);
; ;
; (W07 P2004);
; (CALLS 1ST & 2ND OP. CHAMFER PG);
; ;
; (WORK OFFSET #54 UPPER RIGHT);
; (CORNER OF PART.);
; (WORK OFFSET #55 IS THE LARGE);
; (OFFSET THAT IS X=.831 FROM ZERO);
; (IN X-AXIS, AND IS Y=.9157 FROM);
; (ZERO IN Y-AXIS.);
; ;
;

MEMORY USB DEVICE

CURRENT DIRECTORY: USB DEVICE

File	Size	Date
20new.txt	192	00-10-2009 15:24
hd.txt	192	00-10-2009 15:57
Findtext.txt	192	07-29-2009 17:19
FindtextID.txt	192	03-20-2009 14:42
004001.txt	158	04-01-2009 14:17
Mohage.txt	370991	00-10-2009 14:58
004001.txt	13570	00-10-2009 16:20

7 FILES EMPTY FILE SELECTION

✓ : FILES IN SELECTION
■ : ACTIVE PROGRAM (000000)

F1 For Menu TKN USB

MENU NAVIGATION

Press F1 to access the menu. Use the left and right cursor arrow keys or jog handle to move between the menu categories, and use the up and down cursor arrow keys to highlight an option within a category. Press WRITE/ENTER to make a menu selection.

DISPLAY MODES

Three display modes are available. Switch between display modes using the “Change View” command in the File menu or press PRGRM/CONVRS.

List displays the current FNC program alongside the tabbed LIST PROG menu.

Main displays one program at a time in a tabbed pane (switch between tabs using the “Swap Programs” command in the File menu or by pressing F4).

Split displays the current FNC program on the left and the currently open programs in a tabbed pane on the right. Toggle the active pane using the “Switch to Left or Right Side” in the File menu or by pressing EDIT. When the tabbed pane is active, switch between tabs using the “Swap Programs” command in the File menu or by pressing F4.



DISPLAY FOOTER

```
    (CALLS 1ST & 2ND OP. CHAMFER PGM) ;  
    (-----);  
    (WORK OFFSET #54 UPPER RIGHT) ;  
    (CORNER OF PART.) ;  
    (WORK OFFSET #55 IS THE LARGE) ;  
    (DIAMETER THAT IS X5.831 FROM ZERO) ;  
    (IN X-AXIS. AND IS Y-.9157 FROM) ;  
    (ZERO IN Y-AXIS.) ;  
    (-----);
```

F1 For Menu TKN USB

The footer section of the program display shows system messages and other information about the program and current modes. The footer is available in all three display modes.

The first field displays prompts (in red text) and other system messages. For example, if a program has been changed and needs to be saved, the message "PRESS SEND TO SAVE" appears in this field.

The next field displays the current jog handle scroll mode. TKN indicates that the editor is currently scrolling token by token through the program. Continuously jogging through the program will change the scroll mode to LNE, and the cursor will scroll line by line. Continuing to jog through the program will change the scroll mode to PGE, scrolling a page at a time.

The last field indicates which device (HD, USB, NET) on which the active program is saved. This display will be blank when the program is not saved or when the clipboard is being edited.

OPENING MULTIPLE PROGRAMS

Up to three programs can be open simultaneously in Haas Editor. To open an existing program while another program is open in Haas Editor:

1. Press F1 to access the menu.
2. Under the "File" category, select "Open Existing File".
3. The program list is displayed. Select the device tab where the program resides, highlight the program with the up/down arrow keys or the jog handle, and press SELECT PROG. The display will switch to split mode with the FNC program on the left and the newly opened program and the FNC program on the right in a tabbed pane. To change the program in the tabbed pane, select the "Swap Programs" command in the File menu or press F4 while the tabbed pane is active.



DISPLAY LINE NUMBERS

Line numbers independent of the program text can be displayed. Select the "Show Line Numbers" command from the File menu to display them. Note that these are not the same as Nxx line numbers; they are only for reference when viewing the program. To hide the line numbers, reselect the option in the File menu.

FILE MENU

Open Existing File

Opens a file from the LIST PROG menu in a new tab.

Close File

Closes the current active file. If the file has been changed, the control will prompt to save before closing.

Save

Save the current active file under the same filename. Hot Key: SEND (after a change is made)

Note: Programs are not saved automatically. If power is lost or turned off before saving changes, those changes will be lost. Be sure to save your program frequently while editing.

Save As

Save the current active file under a new filename.

Swap Programs

Bring the next program in a tabbed pane to the top of the tab stack. *Hot Key: F4*

Switch to Left or Right Side

Change the active program window (the currently active window has a white background). *Hot Key: EDIT*

Change View

Switch between List, Main, and Split view modes. *Hot Key: PRGRM/CONVRS*

Show Line Numbers

Displays reference-only line numbers independent of program text (They are never saved as part of the program like Nxx numbers would be). Select the option again to hide the line numbers.



EDIT MENU

Undo

Reverse changes made to the active program. Block and global functions cannot be undone.

Select Text

Highlights a block of text for further edit functions. Position the cursor at the first line of the block you wish to select before choosing this menu option. Then scroll up or down using the cursor arrow keys or jog handle to define the selection area. Press WRITE/ENTER or F2 to highlight the block. *Hot Key: F2*

Move/Copy/Delete Selected Text

Use these three menu options to work with selected text.

Move removes the selected text from its current location and places it after the cursor position. *Hot Key: ALTER*

Copy places the selected text after the cursor position without deleting it from its current location. *Hot Key: INSERT*

Delete removes the selected text from the program. *Hot Key: DELETE*

Cut/Copy Selection to Clipboard

Use these menu options to place selected text in the clipboard.

Cut removes the selected text from the current program and moves it to the clipboard.

Copy places the selected text in the clipboard without removing it from the program.

The clipboard is a persistent storage location for program code; text copied to the clipboard is available until overwritten, even after power cycles.

Paste from Clipboard

Places the clipboard contents after the cursor location. Does not delete the clipboard contents.

Hide/Show Clipboard

Hide the clipboard to view the position and timers & counters displays in its place. Restore the clipboard display by selecting Show Clipboard.

Edit Clipboard

Make adjustments to the clipboard contents. When finished, select "Close Clipboard" from the Edit menu.

Note: The Haas Editor clipboard is separate from the Advanced Editor clipboard. Edits made in Haas Editor cannot be pasted into Advanced Editor.



SEARCH MENU

Find Text

Define a search term and a search direction to locate the first occurrence of the search term in the direction indicated. When choosing a search direction, press F to search for the term below the cursor position, and press B to search above the cursor position.

Find Again

Select this function immediately after a "Find Text" search to locate the next occurrence of the search term. Repeat to continue to the next occurrence.

Find and Replace

Define a search term and a term to replace it with, then specify the search direction (forward/back). When the first occurrence of the search term is found, the control will prompt "Replace (Yes/No/All/Cancel)?". Type the first letter of your choice to continue. If you choose "Yes" or "No", the editor will execute your choice and move to the next occurrence of the search term. Choose "All" to automatically replace all occurrences of the search term. Choose "Cancel" to back out of the function without making changes (text already replaced will remain so if you choose this option).

Find Tool

This function searches the program for tool numbers. Select it again to locate the next tool number.

MODIFY MENU

Remove All Line Numbers

This function removes all Nxx line numbers from the program.

Renumber All Lines

Use this function to renumber all program lines with Nxx codes. Choose a starting number and line number increment.

Reverse + and - Signs

Changes all positive values to negative and vice versa.

Reverse X and Y

Changes all X values to Y values and vice versa.

DIRECT NUMERIC CONTROL (DNC)

Direct Numeric Control (DNC) is another method of loading a program into the control. It is the ability to run a program as it is received through the RS-232 port. This feature differs from a program loaded through the RS-232 port in that there is no limit to the size of the CNC program. The program is run by the control as it is sent to the control; it is not stored in the control.

PROGRAM (DNC)	N00000000
WAITING FOR DNC . . .	
DNC RS232	

DNC Waiting for Program

PROGRAM (DNC)	N00000000
<pre> O01000 ; (G-CODE FINAL QC TEST CUT) ; (MATERIAL IS 2x8x8 6061 ALUMINUM) ; ; (MAIN) ; ; M00 ; (READ DIRECTIONS FOR PARAMETERS AND SETTINGS) ; (FOR VF -SERIES MACHINES W/4TH AXIS CARDS) ; (USE / FOR HIS, VR, VB, AND NON - FORTH MACHINES) ; (CONNECT CABLE FOR HASC BEFORE STARTING THE PROGRAM) ; (SETTINGS TO CHANGE) ; (SETTING 31 SET TO OFF) ; ; ; DNC RS232 DNC END FOUND </pre>	

Program received from DNC

DNC is enabled using Parameter 57 bit 18 and Setting 55. Turn the parameter bit on (1) and change Setting 55 to On. It is recommended that DNC be run with Xmodem or parity selected because an error in transmission will then be detected and will stop the DNC program without crashing. The settings between the CNC control and the other computer must match. To change the setting in the CNC control, enter the Settings page (press Setng/Graph) and scroll to the RS-232 settings (or enter 11 and press the up or down arrow). Use the up/down arrows to highlight the variables and the left and right arrows to change the values. Press Enter when the proper selection is highlighted.

The recommended RS-232 settings for DNC are:

- Settings: 11 Baud Rate Select: 19200
- 12 Parity Select: NONE
- 13 Stop Bits:1
- 14 Synchronization: XMODEM
- 37 RS-232 Date Bits: 8

DNC is selected by pressing MDI twice (The DNC page “Program DNC”) at the top of the page. Note: DNC needs a minimum of 8k bytes of user memory available. This can be done by going to the List Programs page and checking the amount of free memory on the bottom of the page.

The program sent to the control must begin and end with a %. The data rate selected (Setting 11) for the RS-232 port must be fast enough to keep up with the rate of block execution of the program. If the data rate is too slow, the tool may stop in a cut.

Start sending the program to the control before the Cycle Start button is pushed. Once the message “DNC Prog Found” is displayed, press Cycle Start.

DNC Notes

While a program is running in DNC, modes cannot be changed. Therefore, editing features such as Background Edit is not available.

DNC supports Drip Mode. The control will perform one block (command) at a time. Each block will be performed immediately with no block look-ahead. The



exception is when Cutter Compensation is commanded. Cutter Compensation requires three blocks of motion commands to be read prior to a compensated block being performed.

Full duplex communication during DNC is possible by using the G102 command or DPRNT to output axes coordinates back to the controlling computer.

ADVANCED TCP/IP

To set up network communications, enter the specific values for your network into the the CNC control's network settings (refer to settings 900-916 in the Settings chapter of this manual). Press F1 once all the settings have been updated for your control and network system, this will initiate the network.

Use only letters (A-Z case insensitive), digits (0-9), dashes (-) and periods for the network names settings 900 and 907).

NETWORKING TROUBLESHOOTING

The most common errors are cause by an incorrect username or password, incorrect permissions or an expired password.

If accessing the NET SHARE tab and the message “COULD NOT CONNECT TO NETWORK” appears, additional troubleshooting information is available in the file “error.log”, located in the ADMIN folder on the hard drive (this file can be viewed in FNC). If no ADMIN folder exists on the hard drive; create it then retry accessing remote share to generate the log file.

Hardware Check

If the software has been updated and you would like to check the Ethernet hardware version; power up the machine and wait for the NOT READY message to disappear from the List/Prog menu. Press PARAM/DGNOS twice , then PAGE DOWN. At the bottom of the page is the FV version; it should be 12.001 or higher.

Microsoft Network Administration

Verify in Network Neighborhood that the file serving computer can be seen on the network from another computer. Double Click on the server name icon in Network Neighborhood. Verify that the folder is seen for that computer name (The folder name should be the name entered in Setting 139) on the CNC. Verify the file serving computer privileges to this folder are shared. (Not READ ONLY; which is a typical default).

Verify the networking is functioning (available only on TCP/IP networks). Turn DHCP to OFF.

Enter the Static IP address-Setting 902 and Subnet Mask-Setting 903 press F1. Go to a computer on the network. Go to DOS (E.g. MS DOS Command Prompt) and at the DOS prompt type in “Ping” and the same information which was typed into Setting 902.



Example: C:>PING 192.168.1.2

Various data times will be displayed. If network time-out error occurs, verify the settings and check the data cable(s).

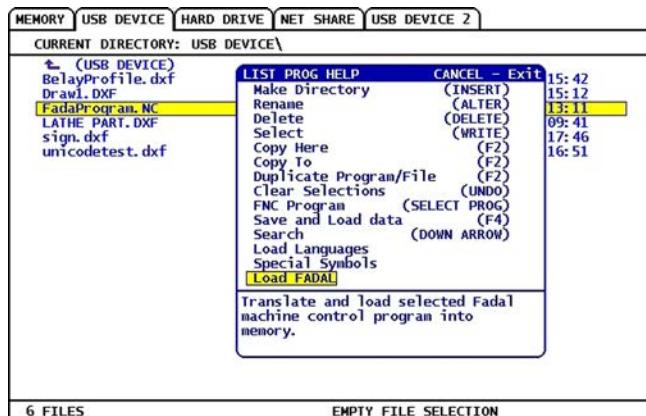
FADAL PROGRAM CONVERTER

Introduction

The Fadal Program Converter quickly converts Fadal code into a Haas program.

Using the Converter

- Access the converter through the LIST PROG menu. Highlight the Fadal program and press F1 or HELP/CALC. Select “LOAD FADAL” from the pop-up menu.



- The converted program is loaded into memory. A copy of the converted program is also saved to I/O device selected, with an “.out” extension. The program will contain “(Converted Fadal Program)” at the top to verify that it is a converted program. Any lines that could not be converted are commented out with an M199, which will give a User Generated Alarm when the program is run. Review these lines and edit them for Haas compatibility.

TIP: You can quickly locate unconverted lines using the search function in edit mode. With the converted program in the active pane (press PROG/CONVRS to change the active pane), press F1 or HELP/CALC and select “Search” from the pop-up menu. Use M199 as the search term.



MEM	000003	N00000000	MEMORY	USB DEVICE	HARD DRIVE	NET SHARE	USB DEVICE 2
CURRENT DIRECTORY: USB DEVICE\							
<pre> 000003 (Converted Fada program) : N10 (N1) G03 (FACE TABLE) ; N11 (N2) G01 Z10.0 F10.0 S5.0 ; N30 (N3) (X0 Y0 IS) ; N40 (N4) (Z0 IS TOP OF THE PART) ; N50 (N5) (*****TOOL_LIST*****); N60 (N6) (T1=3 "SONNET FACE MILL3, DIA. 3"); N70 (N7) (*****END OF TOOL LIST*****) ; N80 (N8) G90 G00 G40 G17 G00 ; N90 (N9) T1 M05 (T1=3 "SONNET FACE MILL 3, DIA. 3"); N10 (N10) G154 P14 G00 X10.5 Y-7.5 S4000 M03 ; N110 (N11) G43 H01 ZL. M08 ; N120 (N12) Z0.1 ; N130 (N13) G01 Z0. F30. ; N140 (N14) G00 X0.5 Y-5.5 ; N150 (N15) Y-7.5 ; N160 (N16) X10.5 ; N170 (N17) Y-7.5 ; N180 (N18) X8.1 Y-5.1 ; N190 (N19) X-8.1 ; N200 (N20) G00 X0.5 Y-5.5 ; N210 (N21) X8.1 ; N220 (N22) Y-5.1 ; N230 (N23) Y-5.7 Y-2.7 ; N240 (N24) X-5.7 ; N250 (N25) Y2.7 ; N260 (N26) X5.7 ; </pre>							
48761 07-29-2009 15:42 1194268 03-31-1999 15:12 553 05-02-2009 13:11 FadaProgram.NC FadaProgram.OUT 931 09-05-2008 09:44 LADIE.PART.DXF 957490 06-01-2006 09:44 signs.dxf 157120 11-16-2007 17:46 unicodestest.dxf 13173 07-28-2009 16:51							
7 FILES				EMPTY FILE SELECTION			
Use CURSOR keys to navigate listing and CANCEL to go back to devices. Press HELP for Help listing.							
▼ : FILES IN SELECTION ■ : ACTIVE PROGRAM (000003)							

MEM	008686	N00000210
<pre> M199 (NOT SUPPORTED: P. 01) : N370 (N934) M97 L1 P9330 ; N380 (N936) G80 ; N390 (N938) M05 ; M09 ; N400 (N940) G90 G43 H00 Z0. ; N410 (N942) M01 ; N420 (N944) (* 1/4-20 TAPRH TOOL - 4 DIA. OF F. - 4 LEN. - 4 DIA - 0.25) ; N430 (N946) T4 M06 ; </pre>		



PROGRAM OPTIMIZER

This feature allows an operator to override spindle speed and axes feed and coolant (P-cool) positions within a program as the program is running. Once the program is finished the changed program lines are highlighted and can be permanently changed or reverted back to the original values.

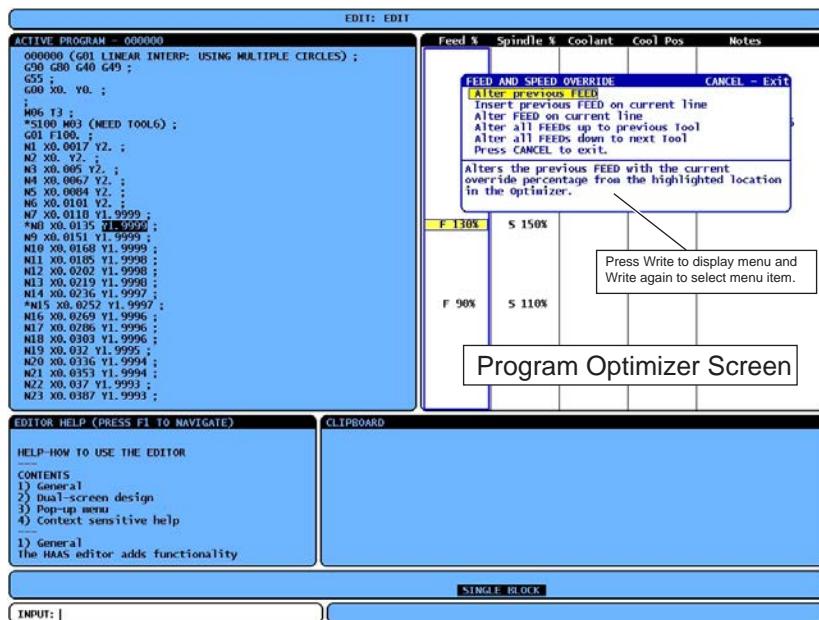
In addition the operator can store notes by typing a comment into the input line and pressing enter.

Operation

As the program is running the operator can type notes, adjust spindle rpm, axis feeds and P-cool positions. At the end of a program (in Memory [MEM] mode) press F4 to go to the Program Optimizer screen.

Use the right/left and up/down arrows, page up/page down and home/end buttons to scroll through overrides and comments. Press Enter on the one to edit and a pop-up window will appear with selections for that column (see figure). The programmer can make a number of changes using the commands in the menu.

In addition a section of code can be highlighted (cursor to start of selection, press F2, scroll to end of selection and press F2). Go back to the Program Optimizer (press Edit) and press enter, this will allow the operator to alter all the feeds or speeds in the highlighted section.



MACHINE DATA COLLECTION

Machine Data Collection is enabled by Setting 143, which allows the user to extract data from the control using a Q command sent through the RS-232 port (or by using an optional hardware package). This feature is software-based and requires an additional computer to request, interpret and store data from the control. Certain Macro variables can also be set by the remote computer.

Data Collection Using the RS-232 Port

The control only responds to a Q command when Setting 143 is ON. The following output format is used:

<STX> <CSV response> <ETB> <CR/LF> <0x3E>

STX (0x02) marks the start of data. This control character is for the remote computer.

CSV is the Comma Separated Variables, one or more data variables separated by commas.

ETB (0x17) is the end of the data. This control character is for the remote computer.

CR/LF tells the remote computer data segment is complete and to move to the next line.

0x3E Displays the “>” prompt.



If the control is busy, it outputs "Status, Busy". If a request is not recognized, the control outputs "Unknown" and a new prompt "> ". The following commands can be used:

Q100 - Machine Serial Number	Q301 - Motion Time (total)
>Q100	>Q301
SOFTWARE, VER M16.01	C.S. TIME, 00003:02:57
Q101 - Control Software Version	Q303 - Last Cycle Time
>Q101	>Q303
SOFTWARE, VER M16.01	LAST CYCLE, 000:00:00
Q102 - Machine Model Number	Q304 - Previous Cycle Time
>Q102	>Q304
MODEL, VF2D	PREV CYCLE, 000:00:00
Q104 - Mode (LIST PROG, MDI, etc.)	Q402 - M30 Parts Counter #1 (resettable at control)
>Q104	>Q402
MODE, (MEM)	M30 #1, 553
Q200 - Tool Changes (total)	Q403 - M30 Parts Counter #2 (resettable at control)
>Q200	>Q403
TOOL CHANGES, 23	M30 #2, 553
Q201 - Tool Number in use	Q500 - Three-in-one (PROGRAM, Oxxxxx, STATUS, PARTS, xxxx)
>Q201	>Q500
USING TOOL, 1	STATUS, BUSY
Q300 - Power-on Time (total)	Q600 Macro or system variable
>Q300	>Q600 801
P.O. TIME, 00027:50:59	ACRO, 801, 333.339996

The user has the ability to request the contents of any macro or system variable by using the Q600 command, for example, "Q600 xxxx". This will display the contents of macro variable xxxx on the remote computer. In addition, macro variables #1-33, 100-199, 500-699, 800-999 and #2001 thru #2800 can be "written to" using an "E" command, for example, "Exxxx yyyy.yyyyy" where xxxx is the macro variable and yyyy.yyyyy is the new value. Note that this command should only be used when there are no alarms present.

Data Collection Using Optional Hardware

This method is used to provide machine status to a remote computer, and is enabled with the installation of an 8 Spare M-code relay board (all 8 become dedicated to below functions and cannot be used for normal M-code operation), a power-on relay, an extra set of Emergency Stop contacts, and a set of special cables. Contact your dealer for pricing information on these parts.



Once installed, output relays 40 thru 47, a power-on relay and the Emergency Stop switch are used to communicate the status of the control. Parameter 315 bit 26, Status Relays, must be enabled. Standard spare M-codes are still available for use.

The following machine statuses will be available:

- * E-STOP contacts. This will be closed when the E-STOP button is pushed.
- * Power ON - 115 VAC. Indicates the control is turned ON. It should be wired to a 115 VAC coil relay for interface.
- * Spare Output Relay 40. Indicates that the control is In-Cycle (running.)
- * Spare Output Relay 41 and 42:

11 = MEM mode & no alarms (AUTO mode.)
10 = MDI mode & no alarms (Manual mode.)
01 = Single Block mode (Single mode)
00 = other modes (zero, DNC, jog, list prog, etc.)

- * Spare Output Relay 43 and 44:

11 = Feed Hold stop (Feed Hold.)
10 = M00 or M01 stop
01 = M02 or M30 stop (Program Stop)
00 = none of the above (could be single block stop or RESET.)

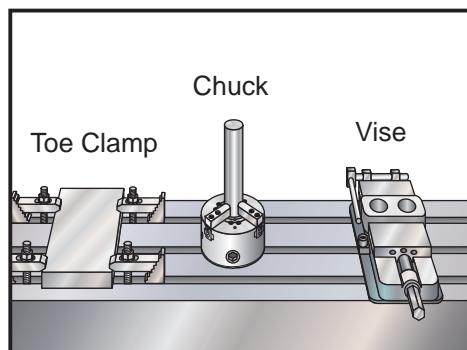
- * Spare Output Relay 45 Feed Rate Override is active (Feed Rate is NOT 100%)

- * Spare Output Relay 46 Spindle Speed Override active (Spindle Speed is NOT 100%)

- * Spare Output Relay 47 Control is in EDIT mode

PART SETUP

It is necessary to properly secure the part to the table. This can be done a number of ways, using vises, chucks or using T-bolts and toe clamps.





TOOLING

Tool Functions (Tnn)

The Tnn code is used to select the next tool to be placed in the spindle from the tool changer. The T address does not start the tool change operation; it only selects which tool will be used next. M06 and will start a tool change operation, for example T1M06 will put tool 1 in the spindle.

Note: There is no X or Y motion required prior to performing a tool change; however, if the work piece or fixture is large, position X or Y prior to a tool change in order to prevent a crash between the tools and the part or fixture.

A tool change can be commanded with the X, Y, and Z axes in any position. The control will bring the Z axis up to the machine zero position. The control will move the Z axis to a position above machine zero during a tool change but will never move below machine zero. At the end of a tool change, the Z axis will be at machine zero.

Tool Holders

There are number of different spindle options for the Haas mills. Each of these types requires a specific tool holder. The most common spindles are the #40 and #50 taper. 40 taper spindles are divided into two types, BT and CT; these are referred to as BT40 and CT40. The spindle and tool changer are only capable of holding one type.

Pull Studs

A pull stud or retention knob is required to secure the tool holder into the spindle. Pull studs are threaded into the top of the tool holder and are specific to the type of spindle. The following chart describes the pull studs used on the Hass mill. Do not use the short shaft or pull studs with a sharp right angle (90-degree) head; they will not work and will cause serious damage to the spindle.

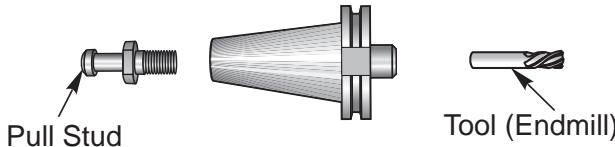


<p>A B C D E</p>	40T CT	5/8-11 Inch Threads	0.990	TSC only	Ø 0.172 Thru.																																				
	24-Piece Kits	JMTBA Standard MAS 403 P40T-1	45°																																						
<ul style="list-style-type: none"> TPS24CT (TSC) PS24CT (Non-TSC) 	50T CT	1"-8 Inch Threads	1.386	TSC only	Ø 0.31 Thru.																																				
	<ul style="list-style-type: none"> TPS24CT50 (TSC) PS24CT50 (Non-TSC) 	JMTBA Standard MAS 403 P50T-1	1.780	45°																																					
<p>A B C D E</p>	40T BT	M16 X 2 Threads	1.104	TSC only	Ø 0.172 Thru.																																				
	<ul style="list-style-type: none"> TPS24BT (TSC) PS24BT (non-TSC) 	JMTBA Standard MAS 403 P40T-1	45°																																						
<p>A B C D E</p>	50T BT	M24 X 3 Threads	1.386	TSC only	Ø 0.31 Thru.																																				
	<ul style="list-style-type: none"> TPS24E50 (TSC) PS24E50 (Non-TSC) 	JMTBA Standard MAS 403 P50T-1	1.780	45°																																					
CT CAT V-Flange <table border="1"> <tr><td>40T</td><td>2.69</td><td>2.50</td><td>.44</td><td>5/8"-11</td><td>1.75</td></tr> <tr><td>50T</td><td>4.00</td><td>3.87</td><td>.44</td><td>1"-8</td><td>2.75</td></tr> </table> BT MAS 403 <table border="1"> <tr><td>40T</td><td>2.57</td><td>2.48</td><td>.65</td><td>M16X2</td><td>1.75</td></tr> <tr><td>50T</td><td>4.00</td><td>3.94</td><td>.91</td><td>M24X3</td><td>2.75</td></tr> </table> DIN / ISO <table border="1"> <tr><td>40T</td><td>2.69</td><td>2.50</td><td>.44</td><td>M16X2</td><td>1.75</td></tr> <tr><td>50T</td><td>4.00</td><td>3.84</td><td>.44</td><td>M24X3</td><td>2.75</td></tr> </table>	40T	2.69	2.50	.44	5/8"-11	1.75	50T	4.00	3.87	.44	1"-8	2.75	40T	2.57	2.48	.65	M16X2	1.75	50T	4.00	3.94	.91	M24X3	2.75	40T	2.69	2.50	.44	M16X2	1.75	50T	4.00	3.84	.44	M24X3	2.75	40T DIN / ISO	M16 X 2 Threads	.990	TSC only	Ø 0.172 Thru.
40T	2.69	2.50	.44	5/8"-11	1.75																																				
50T	4.00	3.87	.44	1"-8	2.75																																				
40T	2.57	2.48	.65	M16X2	1.75																																				
50T	4.00	3.94	.91	M24X3	2.75																																				
40T	2.69	2.50	.44	M16X2	1.75																																				
50T	4.00	3.84	.44	M24X3	2.75																																				
<ul style="list-style-type: none"> TPS24E (TSC) PS24E (Non-TSC) 	JMTBA Standard MAS 403 P40T-1	45°																																							
<p>A B C D E</p>	50T DIN / ISO	M24 X 3 Threads	1.386	TSC only	Ø 0.31 Thru.																																				
		JMTBA Standard MAS 403 P50T-1	1.780	45°																																					

Tool Holder Assembly

Tool holders and pull studs must be in good condition and tightened together with wrenches or they may stick in the spindle. Clean the tool holder body (the part that goes into the spindle) with a lightly oiled rag to leave a film, which will prevent rusting.

40 Taper CT Tool Holder



Install a tool into the tool holder as instructed by the tool manufacturer.

TOOL CHANGER

There are two types of tool changers available for the Haas mills; these are the side mount tool changer and the umbrella style. Both types are commanded in the same manner, but each one is set up differently.

Before loading tools the mill must be zero returned (power up restart button), which should have been done at machine power up.

The tool changer is manually operated using the tool release button and the ATC FWD and ATC REV buttons. There are two tool release buttons; one on the side of the spindle head cover and the second on the keypad.



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.

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

CAUTION! Keep clear of the tool changer during power up, power down, and any tool changer operations.

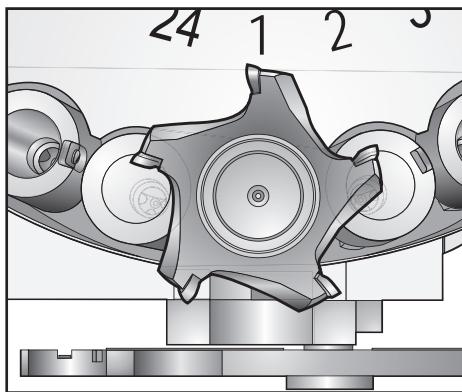
Tools are always loaded into the tool changer by first installing the tool into the spindle. Never load a tool directly into the tool changer.

CAUTION! Tools that make a loud bang when being released indicate a problem and should be checked before serious damage to the tool changer occurs.

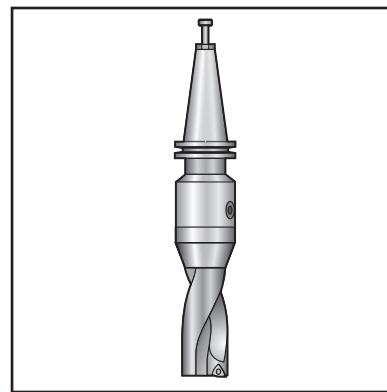
Tool Loading for a Side Mount Tool Changer

NOTE: A normal size tool has a diameter of less than 3" for 40-taper machines, or less than 4" for 50-taper machines. Tools larger than these measurements are considered large size.

1. Ensure the tools loaded have the correct pull stud type for the mill.
2. Press the CURNT COMDS button. From a fresh Power Up/Restart, press the Page Up key once to reach the Tool Pocket Table display. From a normal operating state, press Page Up/Down to reach the Tool Pocket Table.
3. Clear any current "Large" or "Heavy" tool designations. Use the cursor keys to scroll to any tool pocket that has an "L" or "H" next to it. Press Space, then Write/Enter to clear the "Large" or "Heavy" tool designations. Or press 3, then Origin to clear all designation.



Large (and heavy) tool with surrounding pockets empty



Heavy Tool (not large)

Operation

4. Press Origin to reset the Tool Pocket Table to default values. This will place tool 1 in the spindle, tool 2 in pocket 1, tool 3 in pocket 2, etc. This is done to clear the previous Tool Pocket Table settings as well as renumber the Tool Pocket Table for the next program. Another way to reset the Tool Pocket Table is to enter 0 (zero) and press Origin, this will reset all the values to zero.

NOTE: There cannot be two different tool pockets holding the same tool number. Entering a tool number already displayed in the Tool Pocket Table will result in an “Invalid Number” error.

5. Determine if the next program will need any large tools. A large tool has a diameter of greater than 3" for 40-taper machines and greater than 4" for 50-taper machines. If large tools are not used, proceed to Step 10. If large tools are used, proceed to the next step.

6. Organize the tools to match to the CNC program. Determine the numerical positions of large tools and designate those pockets as Large in the Tool Pocket Table. To designate a tool pocket as “Large”, scroll to that pocket, press L, then Write/Enter.

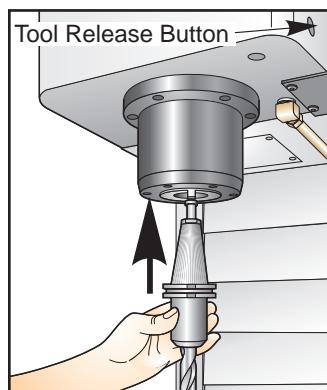
CAUTION! A large tool cannot be placed in the tool changer if one or both of the surrounding pockets already contain tools. Doing so will cause the tool changer to crash. Large tools must have the surrounding pockets empty. However, large tools can share adjoining empty pockets.

7. When all necessary Large and Heavy tool pockets are designated, press the Origin key to renumber the Tool Pocket Table. At this point, the machine is ready to accept tool 1 into the spindle.

8. Take tool 1 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.



9. Press the "Next tool" button.
10. Repeat Steps 9 and 10 until all the tools are loaded.

High-Speed Side Mount Tool Changer

The high-speed tool changer has an additional tool assignment, which is "Heavy". Heavy tools are defined as tools weighing more than 4 pounds. If a tool, heavier than 4 pounds is used, the tool must be entered in the table with an "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 if changing a heavy tool. The pocket up/down speed is not slowed down. The control restores the speed to the current rapid, once the tool change is complete. If problems are encountered changing unusual or extreme tooling, contact your dealer for assistance.

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

L - Empty pockets are required on both sides (large tools are assumed heavy).

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.

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

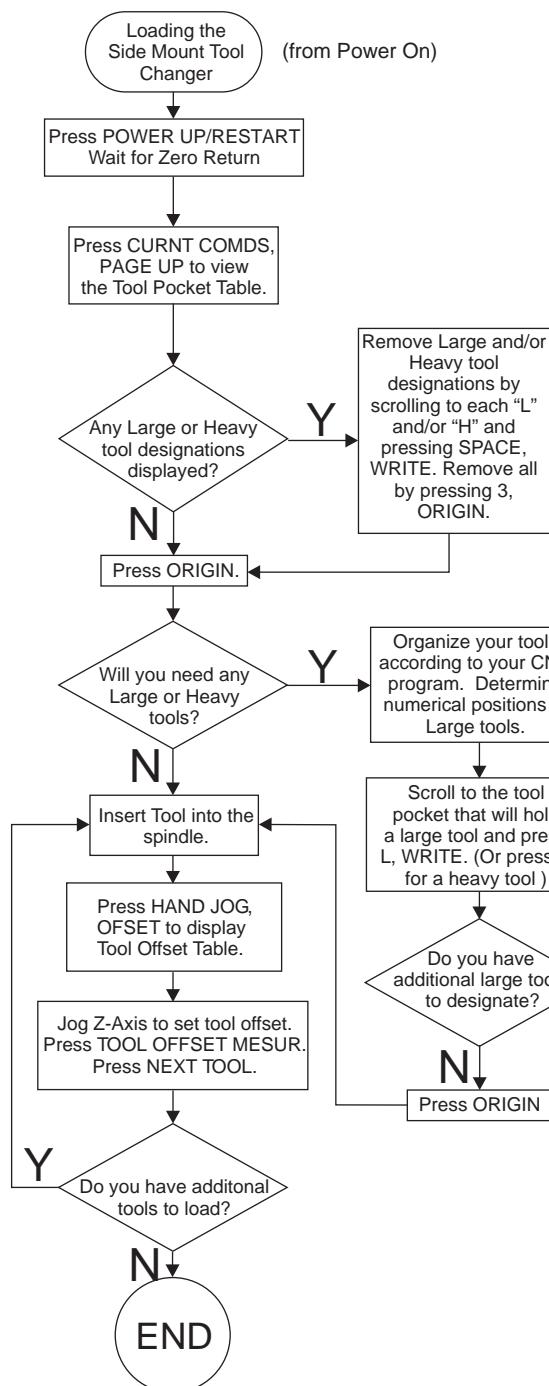
Large tools are assumed to be heavy.

Heavy tools are not assumed to be large.

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



Tool Loading Flowchart



A “Large Tool” has a diameter greater than 3” for 40-taper mills, or greater than 4” for 50-taper mills.

High-speed Toolchangers: A “Heavy Tool” is a tool weighing 4 or more pounds.

Do not enter a lower case “h” or lower case “l”(L) in the tool table.



Using 0 for a Tool Designation

A 0 (number zero) can be inserted in the tool table in place of a tool number. If this is done the tool changer does not “see” this pocket and will never try to install or retrieve a tool from pockets with a “0” designation.

Enter 0 then Origin to zero all pockets, enter 1 then Origin to sequence the pockets, and enter a 3 then Origin to clear all H,h,L,l entries. A 0 cannot be used to designate the tool inserted into the spindle. The spindle must always have a tool number designation.

To designate a pocket as an “always empty” pocket: Use the arrow keys to move to and highlight the pocket to be empty, press the 0 button on the numeric keypad and then press Enter.

Moving Tools in the Carousel

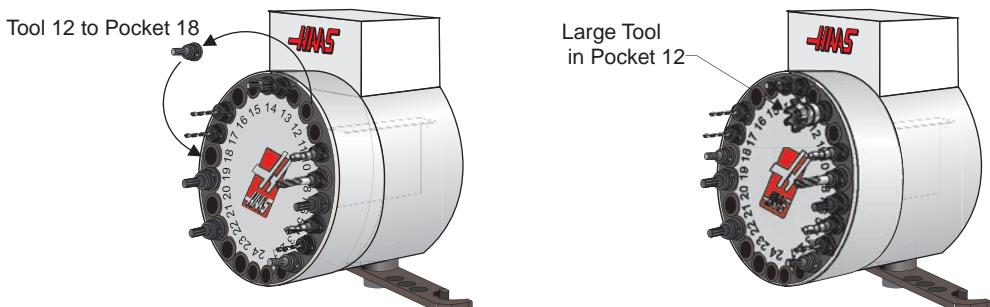
Should tools need moving in the carousel, follow the steps below.

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.

Creating Room for a Large Size Tool

The tool changer pictured has an assortment of normal size tools. For the purposes of this example, tool 12 will be moved to pocket 18 to create room for a large size tool to be placed in pocket 12.

1. Select MDI mode. Press the CURNT COMDS button. Press Page Up/Down (if necessary) to reach the Tool Pocket Table display. Verify which tool number is in pocket 12.
2. Enter Tnn into the control (where Tnn is the tool number from step 1). Press ATC FWD. This will place the tool from pocket 12 into the spindle.
3. Enter P18 into the control, then press ATC FWD to place the tool currently in the spindle into pocket 18.



4. Scroll to pocket 12 in the Tool Pocket Table and press L, Write/Enter to designate that pocket as Large.



5. Enter the tool number into SPNDL (spindle) on the Tool Pocket Table. Insert the tool into the spindle.

NOTE: There cannot be two different tool pockets holding the same tool number. Entering a tool number already displayed in the Tool Pocket table will result in an “Invalid Number” error.

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

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 will cover the tool pocket on either side of the pocket it is installed in. Change bit 3 of parameter 315 to 1, if a tool this size is needed. The tool table must be updated as now two empty pockets are needed between extra large tools.

Umbrella Tool Changer

Tool Loading 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:

1. Ensure the tools loaded have the correct pull stud type for the mill.
2. Enter MDI mode.
3. Organize the tools to match to the CNC program.
4. Take tool 1 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.
5. Press the “ATC FWD” key.
6. 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, push the Emergency Stop button and remove the cause of the jam. Push the RESET button to clear the alarms. Push the Recover button and follow the directions to reset the tool changer.

CAUTION! Never put your hands near the tool changer unless the EMERGENCY STOP button is pressed first.

Side Mount Tool Changer Recovery

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

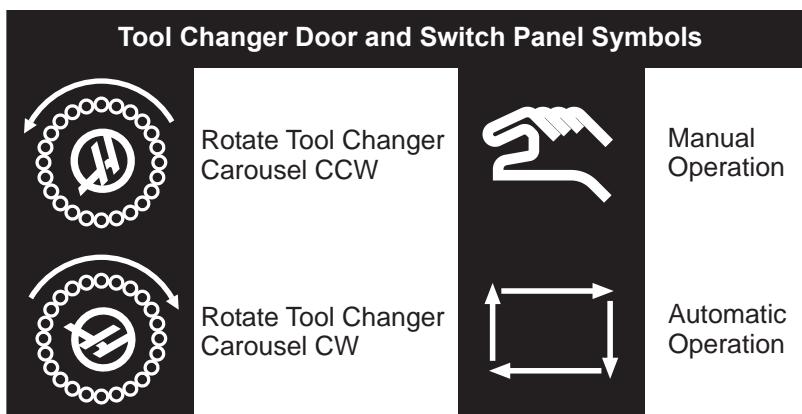


The control will first attempt an automatic recovery. At the tool changer recovery screen, press A to begin automatic recovery or E to exit. If the automatic recovery fails, the option appears for a manual recovery. Press M to continue.

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

Side Mount Tool Changer Door and Switch Panel (if equipped)

Mills such as the MDC, EC-300 and EC-400 have a sub-panel to aid tool loading. The Manual/Auto switch must be set to "Auto" for automatic tool changer operation. If the switch is set to "Manual", the other two buttons, labeled CW and CCW, are enabled and automatic tool changes are disabled. The CW and CCW buttons rotate the tool changer in the clockwise and counterclockwise directions. The door has a switch which detects when the door is open.



Operation

If the cage door is opened while a tool change is in progress, the tool change will stop and not resume until the cage door is closed. However, any machining operations that are in progress will continue.

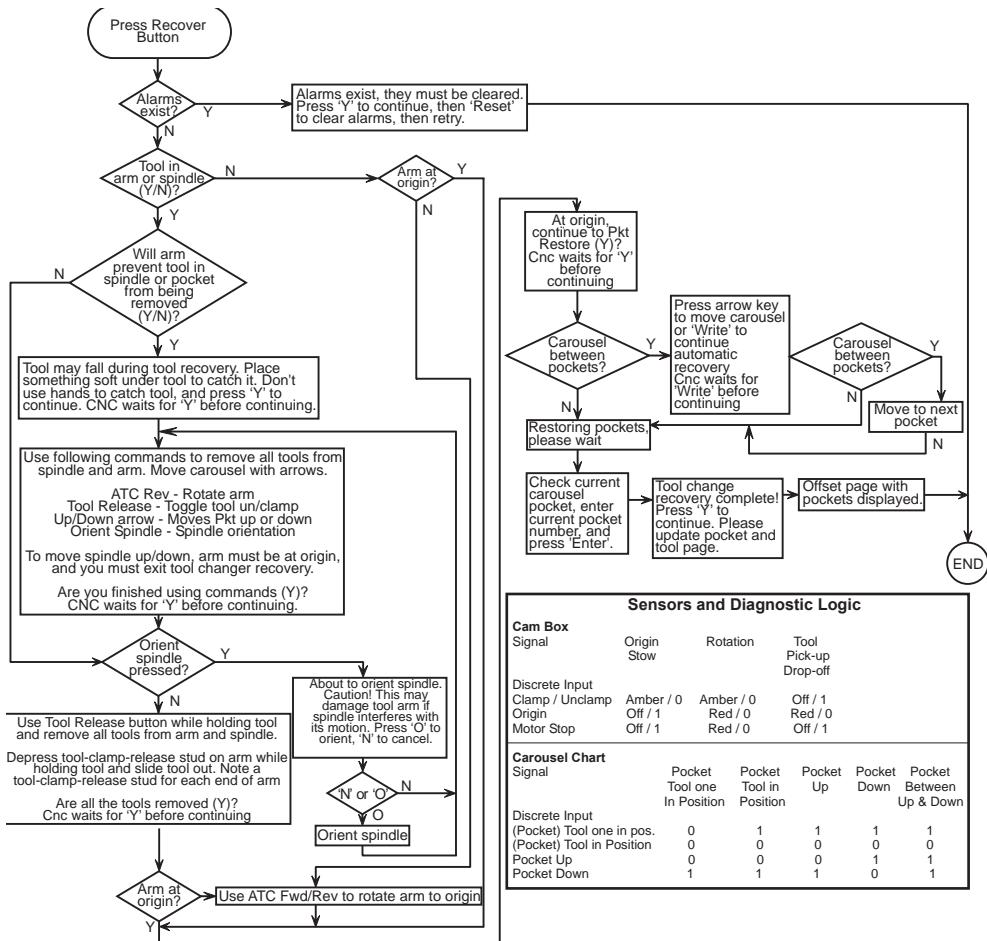
If the switch is turned to "Manual" while a tool change is in progress, current tool changer motion will be completed. The next tool change will not execute until the switch is turned back to "Auto". Any machining operations that are in progress will continue.

The carousel will rotate one position whenever the CW or CCW button is pressed once, while the switch is set to "Manual"

During tool changer recovery, if the cage door is open or the switch is in the "Manual" position and the Recover button 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.



Side Mount Tool Changer Manual Recovery Flow Chart



Operation

Hydraulic Tool Changer Tool Pocket Setup

The Tool pocket table is accessed by pressing the Offset key and then press the right cursor arrow key to reach the tool pocket column. Enter the pocket values for each tool used. This table must be properly setup by the operator to avoid the possibility of damaging tools, the spindle or the tool changer.

Creating a New Tool Table

During the course of operating the machining center it will be necessary to completely reprogram the tool table. There are two useful functions that can be used as an aid in creating a new tool table:

Pressing the '**ORIGIN**' button on the keypad at any time while on the tool table screen will set all the tool pockets to their default values. For example, tool 1 in the spindle, tool 2 in pocket 1, tool 3 in pocket 2, etc.



Pressing ‘0’ and then the ‘**ORIGIN**’ button on the keypad at any time while on the tool table screen will set all the tool pockets to ‘0’.

Tool Numbering System

Powering up the machine for the first time sets the default tool-pocket table. The table is setup such that each pocket contains a tool with the same number as the pocket number. The spindle is initialized to contain Tool 1 (T1). For example, Pocket 1 is dedicated to contain Tool 1 (T1), Pocket 2 is dedicated to Tool 2 (T2), etc. Therefore, the ATC 38 side-mount tool changer has a default of 38 tools, T1 through T38 (T1 in the spindle).

The tool numbers in the tool table assigns a dedicated tool number to each pocket of the tool changer. This number remains in the tool table regardless of where the tool actually is. For example, if tool 5 (T5) is removed from the tool pocket 5 and placed in the spindle, the tool table will indicate that T5 is in the spindle and that pocket 5 is dedicated to tool T5.

A tool called for from the machine’s program commands the control to search the tool table for the tool number and indexes the tool changer to the pocket containing the tool number.

CAUTION! Damage to the machine and/or tooling can occur if the tool called for in the program does not match the tool listed in the tool table and/or installed in the corresponding pocket.

Acceptable Tool Numbers

In general the tool numbers are from T1 to the number of pockets in the tool changer chain (T38 on a 30-pocket tool changer, however, it is possible to use all the numbers in the tool table. This is done to allow for a situation where the operator requires more than available tool pockets to complete the machining job. An example is 55 different tools are required to perform a complete machining job. The first 38 tools would be used and then the machine would be paused while the operator removes 17 of the original tools and replaces them with the 17 tools required to complete the job. After updating the tool table to reflect the new tools in their respective pockets using tool numbers T39 through T55 the machining job is ready to continue.

Using ‘0’ for a Tool Designation

A tool pocket can be labeled as an “always empty” pocket, by entering “0” (zero) for the tool number in the tool table. If this is done the tool changer does not “see” this pocket and will never try to install or retrieve a tool from pockets with a ‘0’ designation.

A zero cannot be used to designate the tool inserted into the spindle. The spindle must always have a tool number designation.



Use of "Large " Tool Designations

The tool changer can accept oversized tools as long as the adjacent pockets are left empty and a designator is added to the tool table. **Oversize (large) tools are tools with a diameter of more than 4.9" (125mm).**

To designate a large tool, scroll to highlight the specific pocket (the one that holds the large tool) and press "L" and then the 'WRITE/ENTER' button. The letter "L" will appear on the screen next to this pocket and the tool numbers on either side will be changed to "-" indicating that tools cannot be inserted into these pockets.

To remove an 'L' designation, highlight the 'L' pocket and press the 'SPACE' button and then the 'WRITE/ENTER' button.

NOTE: Large tools cannot be bigger than 9.8" (250mm).

Heavy tools

Designating a tool as "Heavy" will have no affect on tool changer speed or actions.

Installing/Removing Tools (hydraulic tool changer)

Tools can be installed into the tool changer by inserting them directly into the pockets or by installing them into the spindle and using the ATC FWD/REV buttons on the keypad to stow the tools in the tool changer. During initial setup, using the ATC FWD/REV buttons, the first tool inserted into the spindle will be tool T1 and will be placed into pocket 1.

To install them directly in the tool changer chain, turn switch on the operator's sub-panel to "Manual", open the operator door (it is at the back of the tool changer cage), and install the tool in the chain. Use the CW/CCW buttons to move the chain to the next empty pocket. Remember to leave an empty pocket on either side of a large tool.

Update the tool changer tool table once all the tools are installed.

Tools can be removed, by calling each tool to the spindle and removing them from it, or by taking them directly from the chain at the operator station in the tool cage. To remove tools from the chain, turn the switch to "Manual", open the tool-cage operator door, grasp the tool and press the foot pedal.

ATC FWD, ATC REV Operation (hydraulic tool changer)

Using ATC FWD/REV will cause the tool changer to move to the next tool pocket with respect to the tool in the spindle. For example, if tool T15 is in the spindle and it is dedicated to pocket 20, the tool changer will put tool T15 into pocket 20 and then ATC FORWARD to pocket 21. **It will not retrieve tool 16 (T16).**

The ATC FWD/REV buttons will change the tool, in the spindle, to the next or



previous tool. However, if the next, or previous, tool is a pocket designated with a zero (an empty pocket) the tool changer will skip that pocket and get a tool from a non-zero pocket.

Tool Changer Recovery (hydraulic tool changer)

The tool changer recovery mode is used to manually move the tool changer's arm and carriage into its HOME position.

Press the "Recover" button and follow the on-screen prompts to return the tool changer to the home position.

STICKING TOOLS

Tools may stick in the spindle when a cold tool is put in a hot spindle. To remove a stuck tool:

Let the spindle cool, and try removing the tool with the Tool Release button.

If the tool is still stuck see below.

Note: Two people are necessary to remove a stuck tool from the spindle.

Warning: Wear gloves and eye protection as tools are sharp and can shatter.

Hold the tool with one hand and tap the tool holder (not the spindle) with a piece of aluminum or brass. A small plastic mallet can be used if the other pieces are not available. The other person will press and hold the Tool Release button.

JOG MODE

Jog Mode allows each axes to be jogged to a desired location. Before jogging the axes it is necessary to home (beginning axes reference point) the axes (See the Machine Power-up Section).

To enter jog mode press the hand jog button, then press one of the desired axes (e.g. X, Y, Z, A or B etc.) and either use the handle jog buttons or the jog handle to move the axes. There are different increment speeds that can be used while in jog mode; they are .0001, .001, .01 and .1. The optional Remote Jog Handle (RJH) can also be used to jog the axes.

SETTING OFFSETS

In order to accurately machine a work piece, the mill needs to know where the part is located on the table. Jog the mill with a pointer tool in the spindle, until it reaches the top left corner of the part (see the following illustration); this position is part zero. The values will be entered into G54 on the Work Offset page.

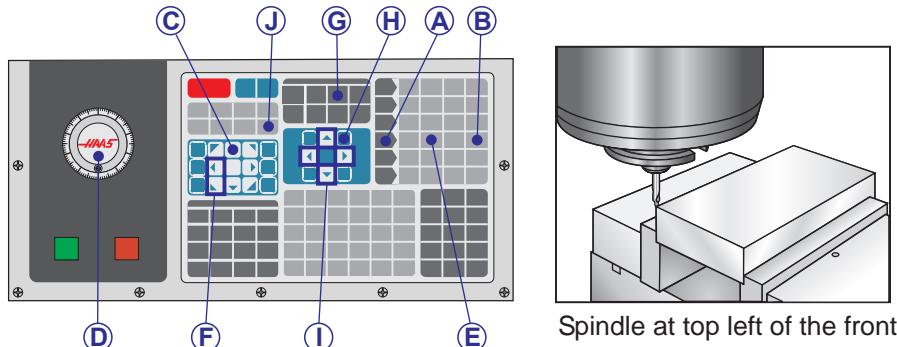


Offsets can also be entered manually by choosing one of the offsets pages, moving the cursor to the desired column, typing a number and pressing Write or F1. Pressing F1 will enter the number in the selected column. Entering a value and pressing Write will add the amount entered to the number in the selected column.

TYPICAL WORK OFFSET SET-UP

1. Place the material in the vise and tighten.
2. Load a pointer tool in the spindle .
3. Press Handle Jog (A).
4. Press .1/100. (B) (The mill will move at a fast speed when the handle is turned).
5. Press +Z (C).
6. Handle jog (D) the Z-axis approximately. 1" above the part.
7. Press .001/1. (E) (The mill will move at a slow speed when the handle is turned).
8. Handle jog (D) the Z-axis approximately. 0.2" above the part.
9. Select between the X and Y axes (F) and handle jog (D) the tool to the upper left corner of the part (See the following illustration).
10. Press Offset (G) until the Work Zero Offset Pane is active.
11. Cursor (I) to G54 Column X.
12. Press Part Zero Set (J) to load the value into the X-axis column. The second press of Part Zero Set (J) will load the value into the Y-axis column.

CAUTION! Do Not Press Part Zero Set a third time; doing so will load a value into the Z-axis. This will cause a crash or Z-axis alarm when the program is run.





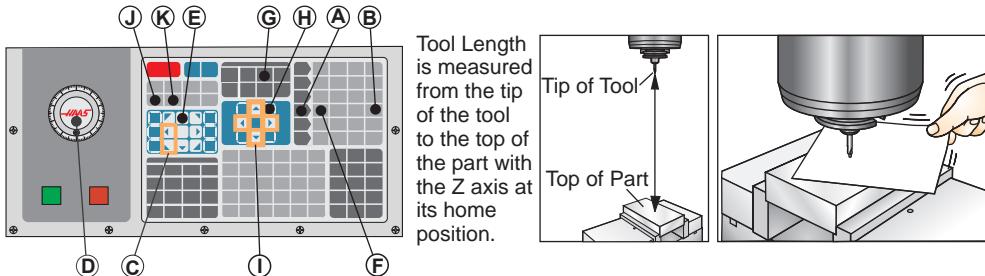
Setting the Tool Offset

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.

1. Load the tool in the spindle.
2. Press Handle Jog (A).
3. Press .1/100. (B) (The mill will move at a fast rate when the handle is turned).
4. Select between the X and Y axes (C) and handle jog (D) the tool near the center of the part.
5. Press +Z (E).
6. Handle jog (D) the Z axis approximately 1" above the part.
7. Press .0001/.1 (F) (The mill will move at a slow rate when the handle is turned).
8. 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.
9. Press Ofset (G).
10. Press Page Up (H) until the page with "Coolant - Length - Radius at the top and scroll to tool #1.
11. Cursor (I) to Geometry for position #1.
12. Press Tool Ofset Mesur (J).

This will take the Z position located in the bottom left of the screen and put it at the tool number position.

CAUTION! The next step will cause the spindle to move rapidly in the Z axis.



13. Press Next Tool (K).



Additional Tooling Set-up

There are other tool set-up pages within the Current Commands. Press Curnt Comds and then use the Page Up/Down buttons to scroll to these pages.

The first is the page with “Spindle Load” and “Vibration” at the top of the page. The programmer can add a tool load limit for spindle load and vibration. The control will reference these values and can be set to do a specific action should the limitations be reached (See setting 84).

The second page is the Tool Life page. On this page there is a column called “Alarm”. The programmer can put a value in this column, causing the machine to stop once the tool has been used that amount of times.

ADVANCED TOOL MANAGEMENT OPERATION

Tool Management Introduction

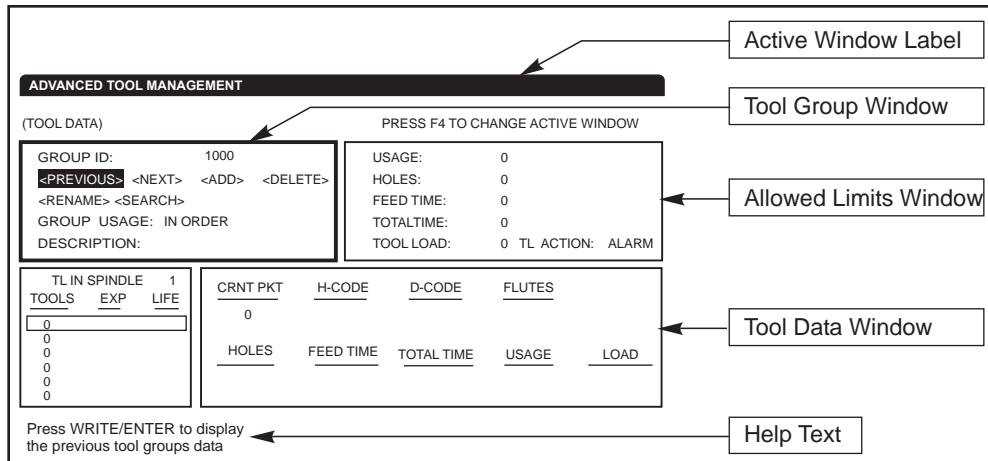
Advanced Tool Management (ATM) allows the programmer to setup and access duplicate tools for the same or a series of jobs. The ATM page is located in the Current Commands mode (Press Current Commands button and page up once). A sample of the ATM screen follows; the screen has “TOOL GROUP” in the title.

Duplicate or backup tools are classified into specific groups. The programmer specifies a group of tools instead of a single tool in the G-code program. ATM will track the usage of individual tools in each tool group and compare it to user defined limits. Once a limit (e.g. number of times used, or tool load) has been reached the mill will automatically choose one of the other tools in the group the next time that tool is needed.

When a tool expires, the beacon flashes orange and the tool life screen automatically displays.

To activate ATM, make sure that Setting 7 (Parameter lock) is off and press E-stop. Under Parameter 315, bit 28, change the value from “0” to “1” and press F4 to switch between windows. Use the cursor keys (left, right, up, down) to move to different items inside the active window. Pressing the Enter key will select, modify or clear values in each item depending on the selection.

The bottom left corner displays simple help information for selected items.



Tool Group - In the Tool Group Window the operator defines the tool groups used in the programs.

Previous – Highlighting <PREVIOUS> and pressing Enter changes the display to the previous group.

Next – Highlighting <NEXT> and pressing Enter changes the display to next group.

Add – Highlight <ADD>, enter a number between 1000 and 2999, and press Enter to add a tool group.

Delete – Use <PREVIOUS> or <NEXT> to scroll to the group to delete. Highlight <DELETE> and press Enter. Confirm the deletion; answering 'Y' will complete the deletion; answering 'N' will cancel the deletion.

Rename - Highlight <RENAME>, enter a number 1000 and 2999 and press enter to renumber the group ID.

Search - To search for a group, highlight <SEARCH>, enter a group number and press enter.

Group Id – Displays the group ID number.

Group Usage – Enter the order in which the tools in the group are called. Use the left and right cursor keys to select how the tools are used.

Description – Enter a descriptive name of the tool group.

Allowed Limits - The Allowed Limits window contains the user defined limits to determine when a tool is worn out. These variables affect every tool in the group. Leaving any variable set to zero causes them to be ignored.

Feed Time – Enter the total amount of time, in minutes, a tool is used in a feed.

Total Time – Enter the total time, in minutes, a tool is used.



Tool Usage – Enter the total times a tool is used (number of tool changes).

Holes – Enter the total number of holes a tool is allowed to drill.

Tool Load – Enter the maximum tool load (in percent) for the tools in the group.

TL Action* – Enter the automatic action to be taken when the maximum tool load percentage is reached. Use the left and right cursor keys to select the automatic action.

Tool Data

TL in Spindle – Tool in the spindle.

Tool – Used to add or remove a tool from a group. To add a tool press F4 until the Tool Data window is outlined. Use the cursor keys to highlight any of the areas under the “Tool” heading and enter a tool number. Entering zero will clear the tool or highlighting the tool number and pressing ORIGIN will reset the H-Code, D-Code and Flutes data to the default values.

EXP (Expire) – Used to manually obsolete a tool in the group. To obsolete a tool, enter a ‘*’, or to clear an obsolete tool, (*), press enter.

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

CRNT PKT – The tool changer pocket the highlighted tool is in.

H-Code – The H-code (tool length) that will be used for the tool. H-code cannot be edited unless Setting 15 H & T Code Agreement is set to Off. The operator can change the H-code by entering a number and pressing Enter. The number entered will correspond to the tool number in the tool offsets display.

D-Code – The D-code that will be used for that tool. D-code can be changed by entering a number and pressing Enter.

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.

Flutes – The number of flutes on the tool. This can be edited by selecting it, entering a new number, and pressing Enter. This is the same as the “Flutes” column listed on the tool offsets page.

Highlighting any of the following sections (Holes through Load) and pressing ORIGIN will clear their values. To change the values, highlight the value in the specific category, enter a new number and press enter.

Load – The maximum load, in percent, exerted on the tool.

Holes – The number of holes the tool has drilled/ tapped/ bored using Group 9 canned cycles .



Feed Time – The amount of time, in minutes , the tool has been in a feed.

Total Time – The total amount of time, in minutes, the tool has been used.

Usage – The number of times the tool has been used.

Tool Group Setup

To add a tool group press F4 until the Tool Group window is outlined. Use the cursor keys until <ADD> is highlighted. Enter a number between 1000 and 2999 (this will be the group ID number). To change a group ID number, highlight the <RENAME> feature, enter a new number and press enter.

Tool Group Usage

A tool group must be setup prior to using a program. To use a tool group in a program first set up a tool group. Next, substitute the tool group ID number for the tool number and for the H-codes and D-codes in the program. See the following program for an example of the new programming format.

Example:

T1000 M06 (tool group 1000)

G00 G90 G55 X0.565 Y-1.875 S2500 M03

G43 H1000 Z0.1 (H-code 1000 same as group ID number)

G83 Z-0.62 F15. R0.1 Q0.175

X1.115 Y-2.75

X3.365 Y-2.875

G00 G80 Z1.0

T2000 M06 (use tool group 2000)

G00 G90 G56 X0.565 Y-1.875 S2500 M03

G43 H2000 Z0.1 (H-code 2000 same as group ID number)

G83 Z-0.62 F15. R0.1 Q0.175

X1.115 Y-2.75

X3.365 Y-2.875

G00 G80 Z1.0

M30

Macros

Tool Management can use macros to obsolete a tool within a tool group.

Macros 8001 to 8200 represent tools 1 through 200. By setting one of these macros to 1, the operator can expire a tool.

Example:

#8001 = 1 (this will expire tool 1 and it will no longer be used)

#8001 = 0 (if tool 1 was expired manually or with a macro, then setting macro 8001 to 0 will make tool 1 available again for use)

Macro variables 8500-8515 enable a G code program to obtain tool group



information. When a tool group ID number is specified using macro 8500, the control will return the tool group information in macro variables 8501 through 8515.

See the variables 8500-8515 in the Macros chapter for the macro variable data label information.

Macro variables 8550-8564 enable a G code program to obtain individual tool information. When an individual tool ID number is specified using macro 8550, the control will return the individual tool information in macro variables 8551-8564. Additionally, a user can specify an ATM group number using macro 8550. In this case, the control will return the individual tool information for the current tool in the specified ATM tool group using macro variables 8551-8564. See the description for variables 8550-8564 in the Macros chapter. The values in these macros provide data that is also accessible from macros starting at 1601, 1801, 2001, 2201, 2401, 2601, 3201, and 3401 and for macros starting at 5401, 5501, 5601, 5701, 5801, and 5901. These first 8 sets provide access for tool data for tools 1-200; the last 6 sets provide data for tools 1-100. Macros 8551-8564 provide access to the same data, but for tools 1-200 for all data items.

Save and Restore Advanced Tool Management tables

The control can save and restore the variables associated with the Advanced Tool Management (ATM) feature to the USB drive and RS-232. These variables hold the data that is entered on the ATM screen. The information can be saved, either as part of an overall backup by using the LIST PROG/POSIT page, or save only the ATM information by bringing up the ATM display screen and pressing F2. When the Advanced Tool Management data is saved as part of an overall backup, the system creates a separate file with a .ATM extension. The ATM data can be saved and restored via the RS232 port by pressing the SENDRS232 and RECV232 buttons while the Advanced Tool Management screen is displayed.

OPTIONAL PROGRAMMABLE COOLANT SPIGOT

The optional programmable coolant (P-cool) directs coolant at the work piece at various angles. The angle of the coolant can be changed within the CNC program.

An additional column on the tool offset page titled "Coolant Position" is displayed, with this option. The spigot will be moved to the position entered for the specific tool when the associated H code and M08 is called.

Programmable Coolant (P-Cool) Set-up

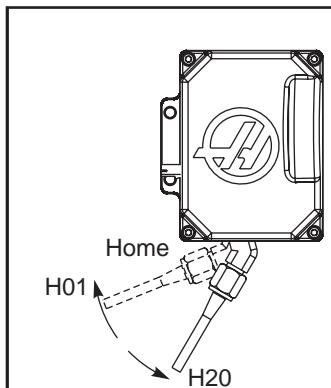
1. Press the OFFSET button to enter the offsets table, press the CLNT UP or CLNT DOWN button to move the P-cool nozzle into the desired position. Press the COOLNT button to turn on the coolant in order to check the P-cool position.
Note: The P-cool position is displayed at the bottom left corner of the screen.



2. Enter the coolant position number for the tool in the Coolant Position column and press F1. Repeat steps 1 and 2 for each tool.

3. Enter the coolant position as an H code in the program. For example H2 will command the nozzle to the position entered in the Tool 2 Coolant Position column.

If setting 15 (H & T Agreement) is on the H code and T code commanded in the program must be the same (e.g. T1 H1 must be used together). If setting 15 is turned off the H code and T code commanded do not need to be the same (e.g. T1 H2 can be commanded).



GRAPHICS MODE

A safe way to troubleshoot a program is to run it in Graphics mode. No movement will occur on the machine, instead the movement will be illustrated on the screen.

Graphics mode can be run from Memory, MDI, DNC or Edit modes. To run a program press the SETNG/GRAFH button until the Graphics page is displayed. Or press Cycle Start from the active program pane in Edit mode to enter Graphics mode. To run DNC in graphics, select DNC, then go to graphics display and send the program to the machine's control (See the DNC section). There are three helpful display features in Graphics mode that can be accessed by pressing one of the function keys (F1 - F4). F1 is the help button, which will give a short description of each of the functions possible in Graphics mode. F2 is the zoom button, which highlights an area using the arrow buttons, Page Up and Page Down to control the zoom level, and pressing the Write button. F3 and F4 are used to control the simulation speed. Note that not all machine functions or motions are simulated in graphics.



DRY RUN OPERATION

The Dry Run function is used to check a program quickly without actually cutting parts. Dry Run is selected by pressing the Dry Run button while in MEM or MDI mode. When in Dry Run, all rapids and feeds are run at the speed selected with the jog speed buttons.

Dry Run can only be turned on or off when a program has completely finished or the Reset button is pressed. Dry Run will still make all of the commanded XYZ moves and requested tool changes. The override keys can be used to adjust the Spindle speeds in Dry Run. Note: Graphics mode is just as useful and may be safer as it does not move the axes of the machine before the program is checked.

RUNNING PROGRAMS

Once a program is loaded on the machine and the offsets are set, run the program by pressing the Cycle Start button. It is suggested that the program is run in Graphics mode before doing any cutting.

BACKGROUND EDIT

Background Edit will allow editing of one program while another program is running. To activate Background Edit while a program is running, press Edit until the background edit pane (on the right side of the screen) is active. Press Select Prog to select a program to background edit (the program must be in Memory) from the list and press Write/Enter to begin background editing. To select a different program to background edit, press Select Prog from the background edit pane and choose a new program from the list.

All of the changes made during Background Edit will not affect the running program, or its subprograms. The changes will go into effect the next time the program is run. To exit background edit and return to the running program, press Prgrm Convrns.

The Cycle Start button may not be used while in Background Edit. If the program contains a programmed stop (M00 or M30) exit Background Edit (press F4) and then press Cycle Start to resume the program.

Note: All keyboard data is diverted to the Background Editor, when a M109 command is active and Background Edit is entered. Once an edit is complete (by pressing Prgrm/Convrns) keyboard input will return to the M109 in the running program.

RUN-STOP-JOG-CONTINUE

This feature allows the operator to stop a running program, jog away from the part, and then resume program execution. The following is an operation procedure:



1. Press Feed Hold to stop the running program
2. Press X, Y or Z followed by the Handle Jog button. The control will store the current X, Y, and Z positions. Note: Axes other than X, Y, and Z cannot be jogged.
3. The control will display the message "Jog Away". Use the jog handle, remote jog handle, jog, or jog-lock buttons to move the tool away from part. Control buttons such as AUX CLNT (TSC), or COOLNT to turn on/off the coolant (AUX CLNT requires that the spindle is rotating and the door is closed). The spindle can be controlled by pressing CW, CCW, Stop, Tool Release. If necessary, tool inserts can be changed. Caution: When the program is continued, the old offsets will be used for the return position. Therefore, it is unsafe and not recommended to change tools and offsets when the program is interrupted.
4. Jog to a position as close as possible to the stored position, or to a position where there will be an unobstructed rapid path back to the stored position.
5. Return to the previous mode by pressing MEM, MDI, or DNC. The control will only continue if the mode that was in effect when stopped is re-entered.
6. Press Cycle Start. The control will display the message Jog Return and rapid X and Y at 5% to the position where Feed Hold was pressed, then return the Z-axis. Caution: The control will not follow the path used to jog away. If Feed Hold is pressed during this motion, the mill axes motion will pause and display the message "Jog Return Hold". Pressing Cycle Start will cause the control to resume the Jog Return motion. When the motion is completed, the control will again go into a feed hold state.
7. Press Cycle Start again and the program resumes normal operation. Also see Setting 36 Program Restart.

AXIS OVERLOAD TIMER

When a spindle or an axes current load is overloaded, a timer will start and display POSITION pane. The starts at 1.5 minutes and counts down to zero. An axis overload alarm (SERVO OVERLOAD) displays when the time has expired to zero.

PALLET CHANGER (EC-SERIES AND MDC-500)

Introduction

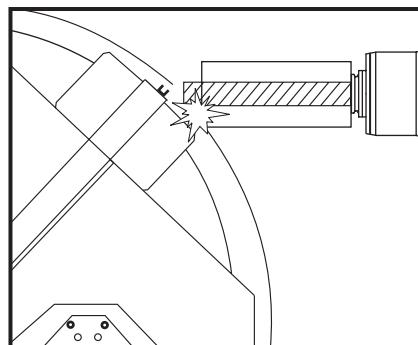
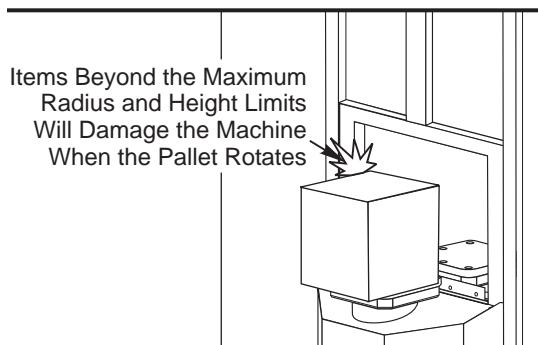
The pallet changer is commanded through a CNC program. The M50 (execute pallet change) function consists of unlocking, lifting and rotating the pallets, then lowering and locking the pallets again. The pallet changer rotates the pallets 180°, then back; it does not continuously rotate in the same direction.

The pallet changer is provided with an audible signaling device to alert nearby personnel when a pallet change is taking place. However, do not depend on the signal to avoid mishaps.



Pallet Changer Warnings and Cautions

- Large workpieces may collide into frame during a pallet change.
- Verify tool length clearance during pallet changes. Long tools may collide with workpiece.



EC-300 Shown

Maximum Pallet Loads

EC-300 550lb (249kg) per station, balanced within 20%
MDC 700lb (318kg) per station, balanced within 20%
EC-400 1 and 45 degree indexer – 1000 lb per pallet
Full 4th Axis 660 lb per pallet

Pallet Changer Operation

The Pallet Changer is commanded using M Codes. M50 determines if a pallet has been scheduled. The pallets will change if a pallet is scheduled or the program will pause and prompt the operator the pallet is not scheduled.

G188 uses the pallet schedule table to load and run the program scheduled for the current pallet. Once the part program completes, the M99 command loops back to the M50 (pallet change) in order to load the next pallet.

M36 is not used in this method, as M50 monitors the pallet scheduling. M36 is provided for backward compatibility, and to program pallet changes without using PST.

Messages aid the operator with loading/unloading and pallet changes. For example, at the beginning of the M50 pallet change, if the load station is not ready, a message is displayed on the screen. The message will not stop flashing and the pallet change will not continue until the load station is ready and the Part Ready button is pressed. At the beginning of a pallet change, if the load station is ready, no messages appears, and no button press is required, and the pallet change starts directly.

M46 – Qn Pmm

Jump to line mm in the current program if pallet n is loaded, otherwise go to the next block.



M48 – Validate that current program is appropriate for loaded pallet

Checks in the Pallet Schedule Table that the current program is assigned to the loaded pallet. If the current program is not in the list or the loaded pallet is incorrect for the program, an alarm is generated. M48 can be in a program listed in the PST, but never in a subroutine of the PST program. An alarm will occur if M48 is incorrectly nested.

M49Pnn Qmm – Sets the status of pallet nn to a value of mm.

Without a P-code, this command sets the status of the currently loaded pallet. The status of each pallet are defined in the pull-down menu of the (PST).

Operator Load Station (EC-300, EC-400, MDC)

To ease loading/unloading parts and to speed production pallet changing mills have an additional load area. The load station is guarded by a door, and a sub-panel includes a few buttons to control the pallet changer. As a safety precaution the load station door must be closed before a pallet change can occur.

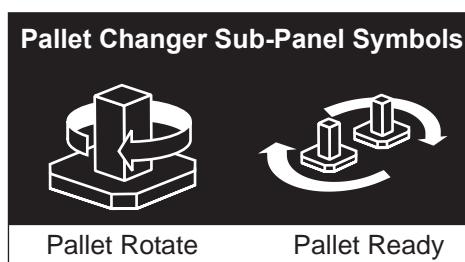
NOTE: The load station pallet must be at home to do a pallet change.

Sub-Panel Controls

Emergency Stop: The button behaves just like the one on the operator's pendant.

Rotary Index: Rotates the load station pallet (see Setting 164).

Part Ready: Used to indicate the pallet is ready. It also contains a light that 1) blinks when the control is waiting for the operator or 2) is on when the operator is ready for a pallet change.



Pallet Changer G-Code

G188 Get Program From PST

Calls the parts program for the loaded pallet based on the PST entry for the pallet

Pallet Changer Programming

The Pallet Changer can be programmed to run the same parts program on both pallets, or a different program on each pallet. See "Sample Programs" for some of the options available for pallet change programming.



Method 1 The following is the preferable method to accomplish a pallet change:

To perform automatic pallet sequencing and part program selection, each pallet must be “scheduled” and must have a part program assigned to it. Scheduling is done in two ways, the first is a pallet can be scheduled with the Part Ready button on the operator’s panel. Pushing the button schedules the pallet that is outside of the machining area.

The second is the pallets can be scheduled from the Pallet Schedule Table (PST). This display can be found by pressing the CURNT COMDS key and then pressing the Page Up or Page Down keys until the Pallet Schedule Table page is reached. Use the arrow keys to highlight the “Load Order” box for the pallet. Enter a pallet number and then the Write/Enter key. If there is already a priority number for that pallet, the “Load Order” numbers of the other pallets will be updated as necessary. A pallet that is on the receiver (in the work area) will have an asterisk in the “Load Order” column; this pallet cannot be scheduled.

Assigning a part program is also done in the PST display. The arrow keys are used to highlight the “Program Number” box for the pallet. The program number is entered by keying the number and then pressing the Write/Enter key. For example, keying “O123”, then Write/Enter will put the program number O00123 into the table.

If a parts program encounters an M50 (without a P code) and the Part Ready button has not been pressed, the control will pause operation, the beacon will blink green, and the message “None Scheduled” is displayed. The mill will wait until the Part Ready button has been pressed, or the PST has been updated, before performing the pallet change. This feature prevents a pallet change from occurring before the operator is ready. The Part Ready button can be pressed at any time and will be recognized when the next pallet change is required.

Method 2

Although the previous method is recommended, the pallet changer can also be operated without automatic sequencing or PST inputs. This is done using M50 with a P code. For proper operation M50 must be preceded by M36. M36 P1 before the M50 P1 will check that pallet #1 is ready.

Pallets can be changed without automatic sequencing or PST inputs. This is done using M50 with a P code. M50 P1 will load pallet #1 without checking to see if it is scheduled. If the PART READY button has been pressed, then pallet #1 will be loaded. If the PART READY button for pallet #1 has not been pressed, the indicator light on the button will flash and the message, “Schedule Pal#1” is displayed.

Pallet Schedule Table

The pallet schedule table incorporates a number of features to help the user with their routine.



Load Order and Pallet Status These two features work together to show which pallet is currently in the machining area.

Pallet Usage This feature gives the number of times the specific pallet has been loaded into the machining area. The counter will turn over to 0 after 32767 pallet changes.

Program Number This detail shows which program number has been assigned to the pallet.

Program Comment This area displays the comments that are written in the part program.

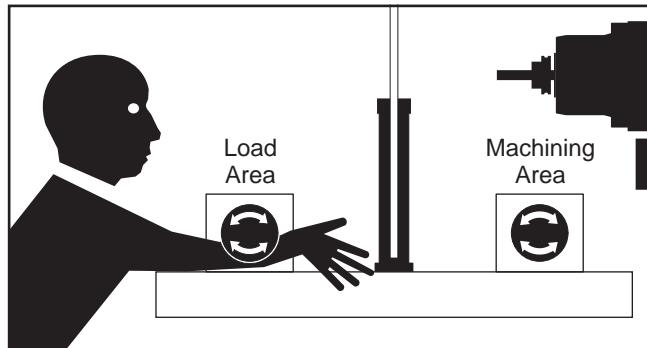
There are 30 different pallet status values to use. The first four: Unscheduled, Scheduled, Loaded, and Completed, are fixed and cannot be changed. The remaining 26 can be modified and used as needed.

Changing or adding status text can be done in the PST. Use the arrow keys to move the cursor to the “Pallet Status” column, and press the F1 key. A selection menu will appear over the “Pallet Status” column (Pressing F1 again or Reset will close the menu.) The number to the left of the text is the status number. This number is used with the M49 command to set the status from the part program. The items in the menu can be selected with the Up and Down arrow keys, or the jog handle. Enter the text, then press F3. Note: All pallets use the same list of status items. Pressing F1 closes the menu without changing the status of any of the pallets.

Resetting a status item to “User” is done by selecting an item from the menu and pressing F4. All status items can be reset at the same time with the Origin key.

Changing the status of an individual pallet can be done from the PST or with the M49 command. In the PST, move the table cursor to the “Pallet Status” column for the pallet desired. Press F1 for the status item menu. Use the arrow keys to select the status, then press F2 or Write/Enter. See the previous description of M49 and in the following examples for setting the pallet status from a program.

CAUTION! The following commands can cause either rotary products to move: Zero Return or Handle Jog



M48 must be placed at the beginning of the program (or section of program) that is run for the current pallet. This will check each time that a program is run, that the program matches the pallet. For example:

```
Oxxxx (Users program)
M48
;
; (Users part program for pallet 1)
;
M30
Oxxxx (Users program)
M48
;
; (Users part program for pallet 2)
;
M30
```

An alarm will be generated, “A (or B) not in Position”, if the pallet in the machine is not the one associated with the part program. If this alarm is generated, verify the correct program is running for the loaded pallet.

Important: Verify that the rotary table on pallet one is plugged into “Connector 1”, and that the rotary table on pallet two is plugged into “Connector 2”.

Sample Programs

Example #1

A basic pallet change program that loads the next scheduled pallet and runs the parts program. The following is a sample of the PST, which indicates that pallet #1 is loaded and pallet #2 is scheduled. Pallet #2 will be loaded next (see



column 2, "Load Order") and program O06012 will be used to cut parts on that pallet (see column 5, "Program Number"). The program comment is captured from the program.

Pallet Schedule Sample Table 1

Pallet Number	Load Order	Pallet Status	Pallet Usage	Program Number	Program Comment
1	*	Loaded	23	O04990	(Rough and Finish)
2	1	Scheduled	8	O06012	(Cut Slot)
O00001		(Program Number)			
M50		(Change to next pallet after Part Ready Button is pressed)			
G188:		(Calls the parts program for the loaded pallet)			
M99		(Loop to top of main program)			
O04990					
Part Program		(User's part program)			
M99		(Return from Subroutine)			
O0612					
Part Program					
M49Q12		(User's Part Program)			
M99		Set current pallet status to 12, string defined by operator. (Return from subroutine)			

Description: The first loop through program O00001 will load pallet #2 (M50) and run program O06012 (G188 selects program from PST for pallet #2). The PST will then resemble Sample Table 2. The asterisk for pallet #2 in the "Load Order" column indicates that this pallet is in the mill.

Pallet Schedule Sample Table 1

Pallet Number	Load Order	Pallet Status	Pallet Usage	Program Number	Program Comment
1	0	Completed	23	O04990	(Rough and Finish Tog)
2	1	Loaded	9	O06012	(Cut Slot)

Description: In the next loop through program O00001, the M50 will detect that no pallets are scheduled. The beacon light will flash green and program O00001 will pause until the operator schedules a pallet or presses Reset. A pallet can be scheduled by pressing the Part Ready button.

Example #2

Basic pallet change program that tracks which part is to be machined on each pallet. Each pallet has a different machining operation. Note that the P code for M46 is a line number in the current program, not a subroutine number.

Oxxxxx Program number



M50	(Perform pallet change after the Part Ready button is pressed or PST is updated)
M46 Q1 Pxx1	This line will check to see if pallet #1 is on the machine. If it is then it will jump to line xx1. If the pallet is not on the machine, then it will continue to the next line. (See description of M46.)
M46 Q2 Pxx2	(If pallet #2 is loaded, program will jump to line xx2, otherwise it will go to the next line.)
M99 Pxxxx	(Jump to line Nxxxx: See the "M Code" section for a more detailed description of M99)
Nxx1	(Line number.)
Part program	(User's part program for Pallet #1)
M99 Pxxxx	(Jump to line Nxxxx)
Nxx2	(Line number)
Part program	(User's part program for Pallet #2)
M99 Pxxxx	(Jump to line Nxxxx)
Nxxxx	(Line number)
M99	(Repeat program)

Example #3

This is an alternate method to Example #2 that uses subroutine calls, but does not jump if the pallet is unscheduled.

NOTE: For proper operation M50 with a P code must be preceded by M36.

M36 P1	(Flash "No Pallet Scheduled" on display, flash green beacon, on Schedule Pallet button #1 until button is pushed or pallet is scheduled in PST)
M50 P1	(Load Pallet #1)
M98 Pxxx1	(Control jumps to program Oxxx1 and runs this program)
M36 P2	(Wait for pallet to be scheduled)
M50 P2	(Load Pallet #2)
M98 Pxxx2	(Control jumps to program Oxxx2 and runs this program)
M99	(Repeat Program)

An M99 at the end of a program will cause continuous operation. M30 at the end of a program will cause the control to wait for the operator to press Cycle Start.



Pallet Changer Recovery (Not For Vertical Mill APC)

EC-300 or MDC - If the pallet change is interrupted another M50 must be run; use M50P1 or M50P2. If this puts the wrong pallet into the mill, then an additional M50 must be run.

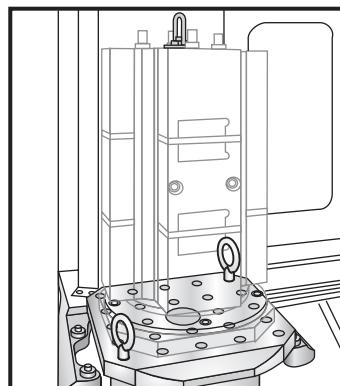
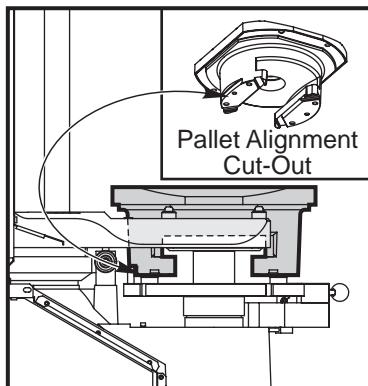
All Other Horizontal Pallet Changing Mills - The control has a pallet changer recovery mode to assist the operator if the pallet changer fails to complete a pallet change. To enter the pallet changer recovery mode press the Recover button and then press the specific function key (F2) for pallet changer recovery mode. Note that if the pallet is in the proper position, the pallet changer recover function is not available.

The most convenient way to recover from a failed pallet change attempt is to press "Y" and follow the on-screen help text. A message will prompt the operator to command an individual step of the pallet change sequence. More than one step may need to be completed, as each step is done, press "Y" for the next step. The control will exit the recovery screen once the pallet changer has recovered.

Pallet Replacement

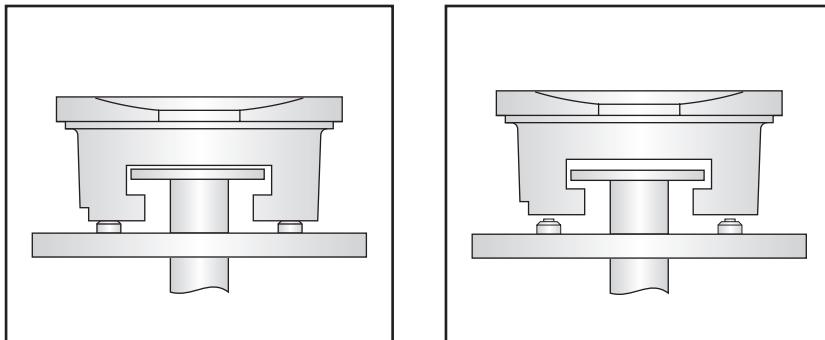
The pallets can only be loaded into the mill through the load station. Note the orientation of the pallet; the pallet can only be loaded one way. A cut-out is machined into the pallet to insure the proper orientation of the pallet.

1. Orient the pallet 90 degrees from home in either direction.



2. Attach a suitable lifting device to the top of the fixture(s) or use eye bolts screwed into the pallet holes.

3. Lift the pallet approximately .25" (6.35mm) to position it above the load station pins, but below the load station lock plate. Pull the pallet towards you until it has cleared the load station.



Pallet Storage

When removing the pallet, be sure to set it on a soft surface such as a wooden pallet. The bottom side of the pallet has machined surfaces that must be protected.

TIPS AND TRICKS

General Tips

Cursor Searching for a Program. When in EDIT or MEM mode, another program can be selected and displayed quickly by entering the program number (Onnnnn) and pressing the Up/Down arrow.

Searching for a Program Command. Searching for a specific command in a program can be done in either MEM or EDIT mode. Enter the address letter code (A, B, C, etc.) or address letter code and value (A1.23), and press the Up/Down arrow. If the address code and no value is entered, the search will stop at the next use of that address code, regardless of the value

Spindle Command. Stop or start the spindle with **CW** or **CCW** any time at a Single Block stop or a Feed Hold. Once the program is restarted with **CYCLE START**, the spindle is turned on.

Saving an MDI Program. Save a program from MDI to the list of programs by positioning the cursor at the beginning of the MDI program, enter a program number (Onnnnn) and press **ALTER**.

To Rapid an Axis Home. Rapid all axes to machine zero by pressing the **HOME G28** key. Move one axis to machine zero in rapid motion by entering the axis letter (e.g. X), then press **HOME G28**. **CAUTION!** There is no warning to alert of any possible collision.

Offset

Entering Offset Values. Pressing **OFFSET** will toggle between the Tool Length Offsets and Work Zero Offsets pages. Pressing Write/Enter will add the number entered to the selected offset value. Pressing F1 will replace the selected offset with the entered number. Pressing F2 will enter the negative value into the offsets.



Coolant Spigot Position. The coolant nozzle position is displayed as the first value after the tool number in the tool offsets table.

Clearing All Offsets and Macro Variables. When in the Tool Length Offset display, clear all the offsets by pressing the Origin key. This also works on the Work Zero Offset and Macro Variables page.

Calculator

Transferring Simple Calculations. The number in the simple calculator box (in the upper left corner) can be transferred to any cursor-selected data line by moving the cursor to the line and pressing **F3**.

Transferring to EDIT or MDI. Pressing **F3** will transfer the number in the calculator box (when the cursor is on the number in the box) to the data entry line in EDIT or MDI mode. Enter the letter (X, Y or Z) to use with the number from the calculator.

Circular Calculator. The Circular Calculator will list four different ways that a circular move can be programmed using the values entered. One of the solutions can be transferred to either EDIT or MDI modes. To do so, cursor onto the program line to use and press either EDIT or MDI. Press the **F3** key, which will transfer the circular move to the data entry line at the bottom of the display. Press Insert to add that circular command line to the program.

One-Line Expressions. The calculator allows a simple, one-line expression without parentheses, such as $23*45.2+6/2$, to be solved. It will be evaluated when the Write/Enter button. Note: Multiplication and division are performed before addition and subtraction.

Programming

Quick Reverse Out of a G84 Rigid Tapping Cycle.

This rigid tapping feature has the tap exit faster than it entered. A J code on the G84 line commands this, for example, J2 retracts twice as fast, J3 retracts three times as fast, etc. up to J9. The J code must be specified in each block.

Duplicating a Program in LIST PROG.

In the List Prog mode, a program can be duplicated by selecting the program number, typing in a new program number (Onnnnn), and pressing **F1**. Select "duplicate program/file" from the popup list and press Enter.

Communications

Receiving Program Files from a Floppy Disk. Program files can be loaded from a floppy disc via a USB Floppy Drive. Use the LIST PROG menu to transfer the files.

Sending Multiple Programs Using Program Numbers. In the LIST PROG menu, highlight each program to send and press WRITE/ENTER at each one to place a check mark. Press F2 and choose the desired function.

Sending a Program File from LIST PROG Display. Files can be sent to a USB device or through the RS-232 port from the LIST PROG display. Use the



cursor arrows and the Enter key to select the program(s), or “ALL” to send all of them under one file name. When F2 is pressed, a popup menu lists the available functions. Choose one and press ENTER or the listed hotkey to execute.

Sending Multiple Programs from LIST PROG Using SEND RS232. Several programs can be sent to the serial port by typing all the program names together on the input line without spaces (e.g., O12345O98765) and pressing SEND RS232.

Send and Receive Offsets, Settings, Parameters and Macro Variables to/ from Disk. Offsets, settings, parameters and macro variables can be saved to a storage device. Press LIST PROG, then select the tab for the device to save to or load from. Press F4 and select the appropriate function, then press WRITE.

Send and Receive Offsets, Settings, Parameters and Macro Variables to/ from RS232. Offsets, settings, parameters and macro variables can be saved to the RS-232 port. Press List Prog, and select a display page (e.g. OFSET, SETNG). Enter a file name and press Send RS232 to send that display page to the RS-232 port. Press RECV RS232 to read the file via RS-232.

Deleting a Program File from a Floppy Disk. A file can be erased from the floppy drive from the LIST PROG display. Type “DEL <file name>” and press Write Enter.

INTUITIVE PROGRAMMING SYSTEM (IPS)

Introduction

The optional Intuitive Programming System (IPS) software simplifies the development of full CNC programs.

To enter the IPS menu press MDI/DNC, then PROGRM/CONVRS. Navigate through the menus using the left and right arrow keys. To select the menu press Write/Enter. Some menus have sub-menus, which again use the left and right arrow keys and Enter to select a sub-menu. Use the arrow keys to navigate through the variables. Key in a variable using the number pad and press Write/Enter. To exit the menu press Cancel.

To exit the IPS menus press any of the Display keys. Press PROGRM/CONVRS in MDI/DNC mode to return to the IPS menus.

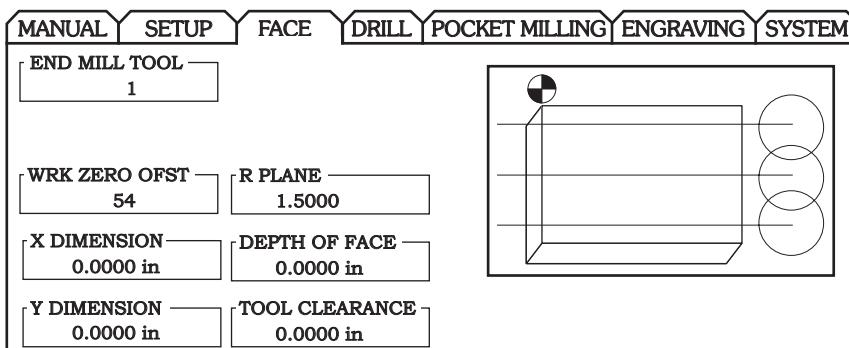
Note that a program entered through the IPS menus is also accessible in MDI mode.

Automatic Mode

Tool and Work offsets must be set before an automatic operation can be run. Enter values for each tool used on the Setup screen. The tool offsets will be referenced when that tool is called in the automatic operation.



On each of the following interactive screens the user is asked to enter data needed to complete common machining tasks. When all the data has been entered, pressing "Cycle Start" will begin the machining process.



Sample IPS Screen

Turning the Option On and Off

The IPS option is toggled off and on using parameter 315 bit 31 (Intuitive Prog Sys). Mills with the option can be returned to the traditional Haas program displays by turning this parameter bit to 0.

To do this, Press the PARAM/DGNOS button, enter "315" and press the down arrow. Use the left and right arrow, or jog handle to scroll to the last parameter bit (Intuitive Prog Sys). Press the Emergency Stop Button, type "0" (zero) and press enter.

To re-activate the IPS option, scroll to the parameter bit as previously described, press the Emergency Stop button, type "1" and press Enter.

IPS Recorder

The IPS recorder provides a simple method to place G-code generated by IPS into new or existing programs.

Operation

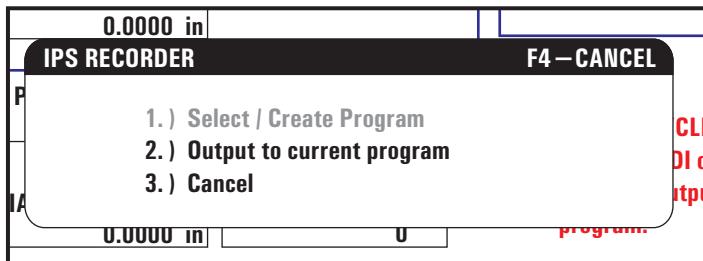
1. To access IPS, press MDI/DNC, then PROGRM/CONVRS.
2. When the recorder is available, a message appears in red in the lower right corner of the tab:



MANUAL	SETUP	FACE	DRILL	POCKET MILLING	ENGRAVING	VQC
CENTER DRILL 0	DRILL TOOL 0	TAP TOOL 0				
CENTER DEPTH 0.0000 in	DRILL DEPTH 0.0000 in	TAP DEPTH 0.0000 in				
CENTER PECK 0.0000 in	DRILL PECK 0.0000 in					
WRK ZERO OFST 54	R PLANE 0.2000 in	NUM OF HOLES 0	Press <CYCLE START> to run in MDI or <F4> to record output to a program.			
X CENTER PT 0.0000 in	DIAMETER 0.0000 in	CENTER HOLE 0				
Y CENTER PT 0.0000 in	ANGLE 0.000 deg					
BOLT CIRCLE		BOLT LINE	SINGLE HOLE	MULTIPLE HOLES		

Operation

3. Press F4 to access the IPS recorder menu. Choose menu option 1 or 2 to continue, or option 3 to cancel and return to IPS. F4 can also be used to return to IPS from any point within IPS recorder.



IPS Recorder Menu

Menu Option 1: Select / Create Program

Select this menu option to choose an existing program in memory or to create a new program into which the G-code will be inserted.

1. To create a new program, input the letter 'O' followed by the desired program number and press the WRITE key. The new program is created, selected, and displayed. Press the WRITE key once more to insert the IPS G-code into the new program.
2. To select an existing program, enter an existing program number using the O format (Onnnnn), then press the WRITE key to select and open the program. To choose from a list of existing programs, press the WRITE key without input. Use the cursor arrow keys to choose a program and press WRITE to open it.



MANUAL	SETUP	FACE	DRILL	POCKET MILLING	ENGRAVING	VQC
<p>Select / Create Program F4 – CANCEL</p> <p>CENTER DRILL 0</p> <p>CENTER DEPTH 0.0000 in</p> <p>CENTER PECK 0.0000 in</p> <p>WRK ZERO OFST 54</p> <p>X CENTER PT 0.0000 in</p> <p>Y CENTER PT 0.0000 in ANGLE 0.000 deg</p> <p>BOLT CIRCLE BOLT LINE SINGLE HOLE MULTIPLE HOLES</p> <p>000000 (PROGRAM A) 000001 (PROGRAM B) 000002 (PROGRAM C) 000003 (PROGRAM D) 000004 (PROGRAM E) 000005 (PROGRAM F) *000006 (PROGRAM G)</p> <p>Choose a program by using the cursor keys and press WRITE to select. or Enter a '0' followed by a new program number and press WRITE to create.</p> <p>< CYCLE START > or < MDI or < F4 > and output to a file.</p>						

3. Using the arrow keys, move the cursor to the desired insertion point for the new code. Press WRITE to insert the code.

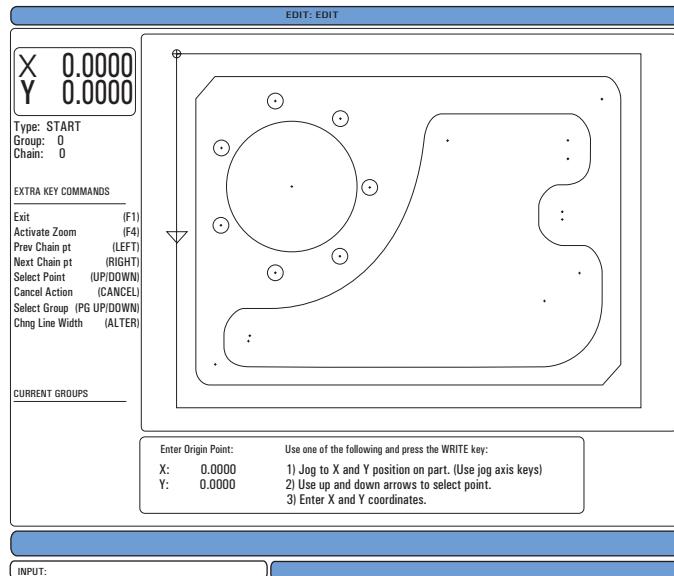
Menu Option 2: Output to Current Program

1. Select this option to open the currently selected program in memory.
2. Use the arrow keys to move the cursor to the desired insertion point for the new code. Press WRITE to insert the code.



DXF FILE IMPORTER

This feature can quickly build a CNC G code program from a .dxf file. This is accomplished with three steps:



Operation

The DXF importer feature provides an on-screen help throughout the process. The step outline box shows which steps are completed by turning the text green at each completed step. The keys needed are defined beside the steps. Additional keys are identified in the left hand column for advanced use. Once a tool-path is completed it can be inserted into any program in memory. This feature will identify repetitive tasks and automatically execute them, for example, finding all the holes with the same diameter. Long contours are also automatically joined.

NOTE: DXF importer is only available with the IPS option.

Start by setting up the cutting tools in IPS. Selecting a .dxf file and press F2. The control will recognize a DXF file and import it into the editor

1. Set the origin of the part.

This can be done by using one of three methods.

- Point Selection
- Jogging
- Enter Coordinates

The jog handle or arrow buttons are used to highlight a point; press "Enter" to accept the highlighted point as the origin. This is used to set the work coordinate information of the raw part.



2. Chain / Group

This step finds the geometry of the shape(s). The auto chaining function will find most part geometry. If the geometry is complex and branches off a prompt will display so the operator can select one of the branches. The automatic chaining will continue once a branch is selected. Similar holes are grouped together for drilling and/or tapping operations.



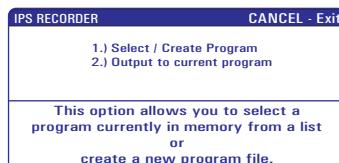
Use the jog handle or arrow buttons to choose the starting point of the tool path. Press F2 to open the dialog box open. Choose the option that best suits the desired application. The Automatic Chaining function is typically the best choice as it will automatically plot the tool path for a part feature. Press "Enter" This will change the color of that part feature and add a group to the register under "Current group" on the left hand side of the window.

3. Select Tool Path

This step applies a tool-path operation to a particular chained group. Select group and Press F3 to choose a tool path. Use the jog handle to bisect an edge of the part feature; this will be used as a entry point for the tool. Once a tool-path is selected, the IPS (Intuitive Programming System) template for that path will display.

Most IPS templates are filled with reasonable defaults. They are derived from tools and materials that have been setup.

Press F4 to save the toolpath once the template is completed; either add the IPS G code segment to an existing program or create a new program. Press EDIT to return to the DXF import feature to create the next tool path.



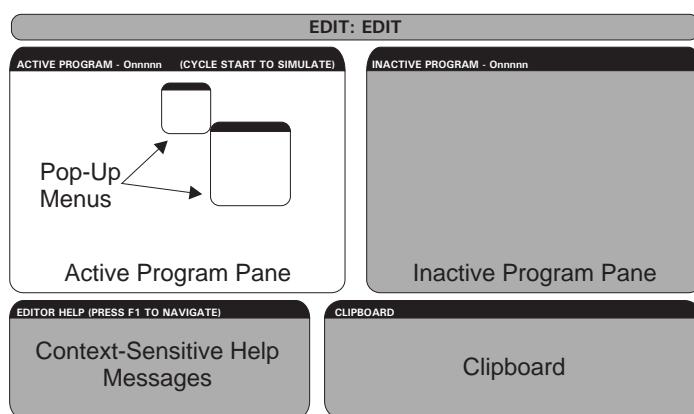


PROGRAMMING

Edit gives the user the ability to edit programs using popup menus.

Press the EDIT key to enter edit mode. Two editing panes are available; an active program pane and an inactive program pane. Switch between the two by pressing the EDIT key.

To edit a program, enter the program name (Onnnnn) from the active program pane and press SELECT PROG; the program will open in the active window. Pressing the F4 button will open another copy of that program in the inactive program pane if there is not a program there already. In addition, a different program can be selected in the inactive program pane by pressing SELECT PROG from the inactive program pane and selecting the program from the list. Press F4 to exchange the programs between the two panes (make the active program inactive and vice versa). Use the jog handle or the down/up arrows buttons to scroll through the program code.



Programming

Basic Edit Mode Layout

Press F1 to access the Popup Menu. Use the Left and Right cursor arrow keys to select from the topic menu (HELP, MODIFY, SEARCH, EDIT, PROGRAM), and use the Up and Down arrow keys or handle jog to select a function. Press Enter to execute from the menu. A context-sensitive help pane in the lower left provides information on the currently selected function. Use Page Up/Down to scroll through the help message. This message also lists hot keys that can be used for some functions.

THE PROGRAM MENU

Create New Program

This menu item will create a new program. To do so, enter a program name (Onnnnn) (that is not already in the program directory) and press Enter to create the program. Hot Key - Select Prog



Select Program From List

Choose this menu item to edit a program that exists in the memory.

When this menu item is selected, the programs in the control are displayed. Scroll through the list by using the cursor buttons or the jog handle. Pressing Enter or Select Prog will select the highlighted program to replace the program list with the selected program. Hot key - Select Prog

Duplicate Active Program

This selection will copy the current program. The user will be prompted to enter a program (Onnnnn) number for the duplicate program.

Delete Program From List

This menu item will delete a program from the program memory. Hot Key - Erase Prog

Swap Editor Programs

Puts the active program in the inactive program pane and the inactive program in the active program pane.

Hot Key-F4

Switch To Left Or Right Side

This will switch between the active and inactive program for editing. Inactive and active programs remain in their respective panes. Hot Key - Edit

THE EDIT MENU

Undo

The last edit operation will be undone, up to the last 9 editing operations. Hot Key - Undo

Select Text

This menu item will select lines of program code to set the start point of the text selection. Then use the cursor keys, home, end, page up/down, or the jog handle to scroll to the last line of code to be selected and Press F2 or Write/Enter. The selected text will be highlighted. To deselect the block, press Undo. Hot Key - F2 to begin selection, F2 or Write to end selection

Move Selected Text

This feature works with the "Select Text" feature. Scroll the cursor arrow to desired part of code and press the Write/Enter button to move the selected text to its new location. The selected text will be moved to the point following the cursor (>).



Copy Selected Text

To select text, scroll the cursor arrow (>) to a portion of text and press the Write/Enter button. The text copied will be highlighted. Scroll the cursor arrow to the part of text to insert the copied text into. Press F2 or Write/Enter to insert the copied text to the point following the cursor (>). Hot Key - Select Text, Position Cursor and Press Write

Delete Selected Text

To select text, scroll the cursor arrow (>) to a portion of text and press the Write/Enter button. The text copied will be highlighted. Once highlighted, press the Write/Enter button to delete the text. If no block is selected, the currently highlighted item is deleted.

Cut Selection To Clipboard

All selected text will be moved from the current program to a new program called the clipboard. Any previous contents of the clipboard are deleted.

Copy Selection To Clipboard

All selected text will be copied from the current program to a new program called the clipboard. Any previous contents of the clipboard are deleted.

Paste From Clipboard

Contents of the clipboard are copied into the current program at the line following the current cursor position.

THE SEARCH MENU

Find Text

This menu item will search for text or program code in the current program.

Find Again

This menu item will search again for the same program code or text.

Find And Replace Text

This menu item will search the current program for specific text or program and optionally replace each (or all) with another G-Code item.

THE MODIFY MENU

Remove All Line Numbers

This menu item will automatically remove all unreferenced N-Codes (line numbers) from the edited program. If a group of lines is selected only these lines will be affected.

Renumber All Lines

This menu item will either renumber all selected blocks in the program or, if a group of lines is selected the renumber feature will affect those lines only.



Renumber By Tool

Searches for T (tool) codes, highlights all the program code up to the next T code and renames the N code (line numbers) in the program code.

Reverse + & - Signs

This menu item will reverse the signs of the numeric values. Press the enter key to start the process and then enter the axes (e.g. X, Y, Z etc.) that is to change. When using this feature be cautious if the program contains a G10 or G92 (See the G Code section for a description).

Reverse X & Y

This feature will change X address codes in the program to Y address codes and Ys to Xs.

OTHER KEYS

INSERT

INSERT can be used to copy selected text in a program to the line after where you place the cursor arrow point.

ALTER

ALTER can be used move selected text in a program to the line after the cursor arrow.

DELETE

DELETE can be used to delete selected text in a program.

UNDO

If a block has been selected, pressing UNDO will simply exit a block definition.



CUTTER COMPENSATION

Cutter compensation is a method of shifting the tool path so that the actual centerline of the tool is moved 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 will select cutter compensation left; that is, the tool is moved to the left of the programmed path to compensate for the amount entered in the offsets page (Refer to setting 40). G42 will select cutter compensation right, which will move the tool to the right of the programmed path. A Dnnn must also be programmed with G41 or G42 to select the correct offset number from the radius/diameter offset column. If the offset contains a negative value, cutter compensation will operate as though the opposite G code was specified. For example, a negative value entered for a G41 will behave as if a positive value was entered for G42. Also, if cutter compensation is selected (G41 or G42), you may only use the X-Y plane for circular motions (G17). Cutter Compensation is limited to compensation in only the X-Y plane.

The code G40 will cancel cutter compensation and is the default condition when a machine is powered on. When canceled, 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 to check the next two blocks containing X or Y motions. Interference checks are performed on these three blocks of information. Setting 58 controls how this part of cutter compensation works. It can be set to Fanuc or Yasnac.

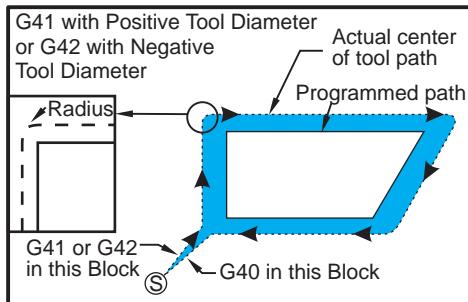
Selecting Yasnac for Setting 58, the control must be able to position the side of the tool along all of the edges of the programmed contour without over cutting the next two motions. A circular motion joins all of the outside angles.

Selecting Fanuc for Setting 58, the control does not require that the tool cutting edge be placed along all edges of the programmed contour, preventing over-cutting. However an alarm will be generated if the cutter's path is programmed such that over-cutting cannot be avoided. Outside angles less than or equal to 270 degrees are joined by a sharp corner and outside angles of more than 270 degrees are joined by an extra linear motion (See the following diagrams).

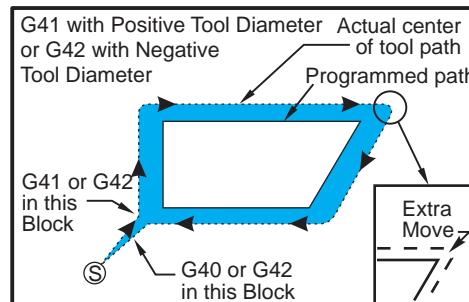


The following diagrams show how cutter compensation works for the two possible values of Setting 58. Note that a small cut of less than tool radius and at a right angle to the previous motion will only work with the Fanuc setting.

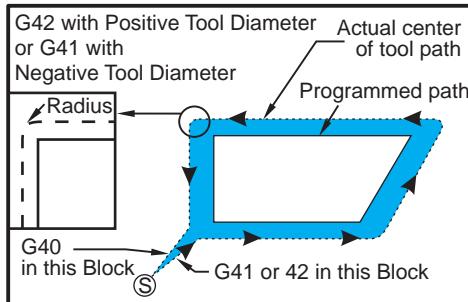
Cutter Compensation - (YASNAC Style)



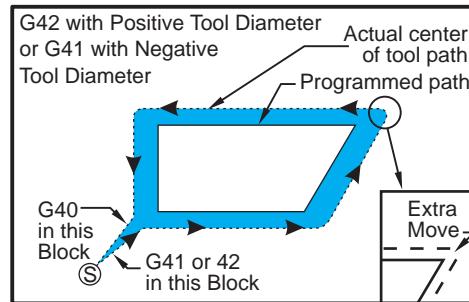
Cutter Compensation - (FANUC Style)



Cutter Compensation - (YASNAC Style)



Cutter Compensation - (FANUC Style)



ENTRY AND EXIT FROM CUTTER COMPENSATION

When entering and exiting cutter compensation or when changing from left side to right side compensation, there are special considerations to be aware of. Cutting should not be performed during any of these moves. To activate cutter compensation, a nonzero D code must be specified with either G41 or G42 and G40 must be specified in the line that cancels cutter compensation. In the block that turns on cutter compensation, the starting position of the move is the same as the programmed position, but the ending position will be offset, to either the left or right of the programmed path, by the amount entered in the, radius/diameter, offset column. In the block that turns off cutter compensation, the starting point is offset and the ending point is not offset. Similarly, when changing from left to right or right to left side compensation, the starting point of the move needed to change cutter compensation direction will be offset to one side of the programmed path and end at a point that is offset to the opposite side of the programmed path. The result of all this is that the tool moves through a path that may not be the same as the intended path or direction (See drawing



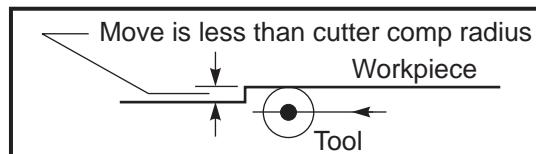
A). If cutter compensation is turned on or off in a block without any X-Y move, there is no change made to cutter compensation until the next X or Y move is encountered. To exit from cutter compensation, you must specify G40.

You should always turn off cutter compensation in a move that clears the tool away from the part being cut. If a program is terminated with cutter compensation still active, an alarm is generated. In addition, you cannot turn cutter compensation on or off during a circular move (G02 or G03); otherwise an alarm will be generated.

An offset selection of D0 will use zero as the offset value and have the same effect as turning off cutter compensation. If a new D value is selected while cutter compensation is already active, the new value will take effect at the end of the proceeding move. You cannot change the D value or change sides during a circular motion block.

When turning on cutter compensation in a move that is followed by a second move at an angle of less than 90 degrees, there are two ways of computing the first motion: cutter compensation type A and type B (Setting 43). Type A is the default in Setting 43 and is what is normally needed; the tool moves directly to the offset start point for the second cut. Type B is used when clearance around a fixture or clamp is needed, or in rare cases when part geometry demands it. The diagrams on the following pages illustrate the differences between type A and type B for both Fanuc and Yasnac settings (Setting 58).

Improper Cutter Compensation Application



Improper Cutter Compensation Application

Note that 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.

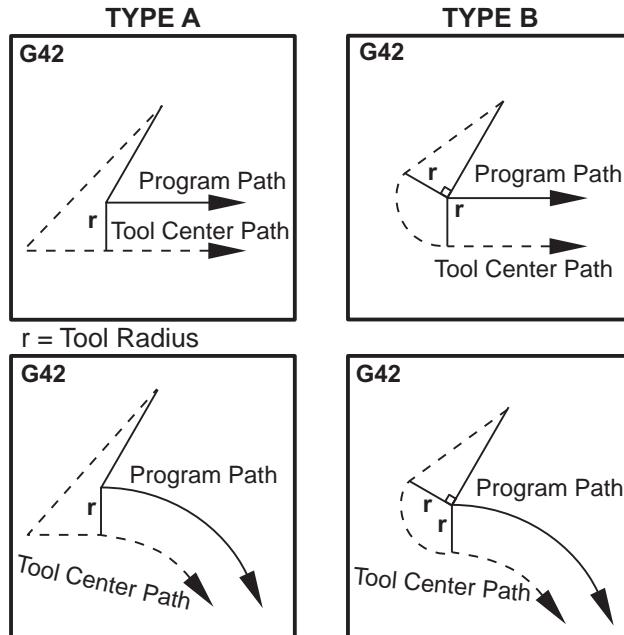
FEED ADJUSTMENTS IN CUTTER COMPENSATION

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.

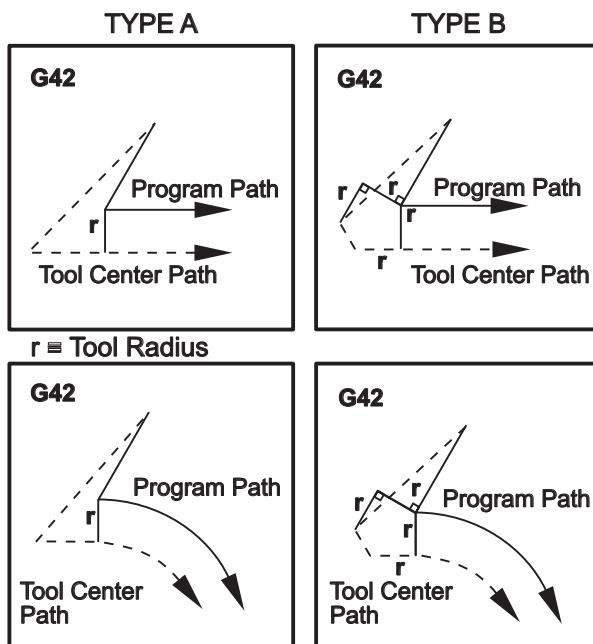


When the cut is on the outside of a circular motion, there is no speed-up adjustment made to the rate feed.

Cutter Compensation Entry (Yasnac)



Cutter Compensation Entry (Fanuc style)





CIRCULAR INTERPOLATION AND CUTTER COMPENSATION

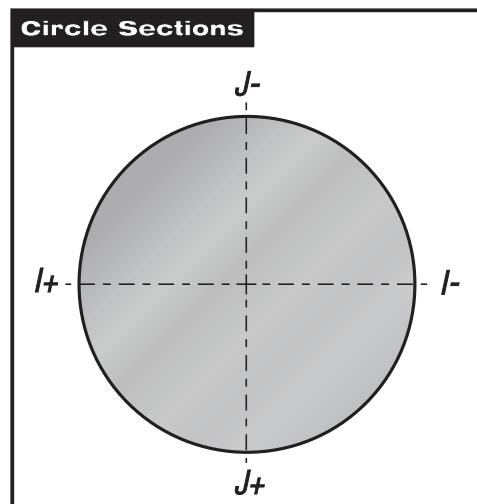
In this section, the usage of G02 (Circular Interpolation Clockwise), G03 (Circular Interpolation Counterclockwise) and Cutter Compensation (G41: Cutter Compensation Left, G42: Cutter Compensation Right) is described.

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°), an I or a J with a value must be specified. The circle section illustration will describe the different sections of a circle.

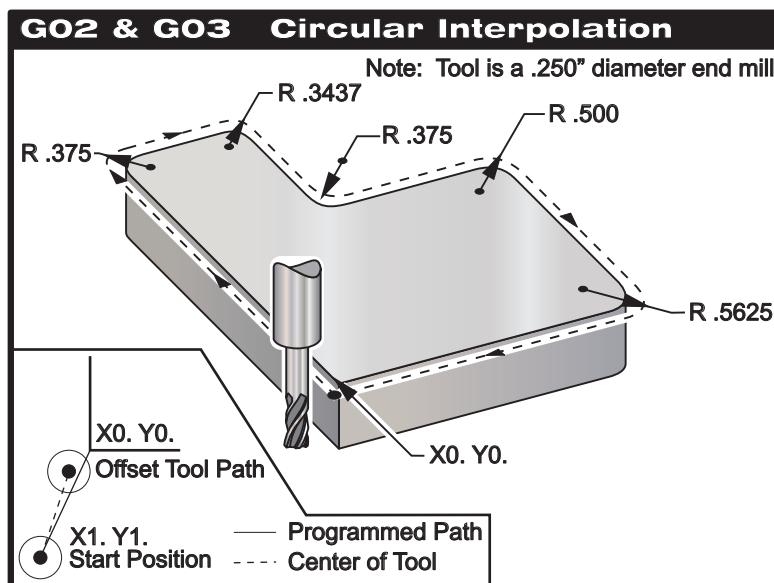
By using cutter compensation in this section, the programmer will be able to shift the cutter by an exact amount and be able to machine a profile or a contour to the exact print dimensions. By using cutter compensation, programming time and the likelihood of a programming calculation error is reduced due to the fact that real dimensions can be programmed, and part size and geometry can be easily controlled.

The following are a few rules about cutter compensation that have to be closely followed in order to perform successful machining operations. Always refer to these rules when programming.

1. Cutter compensation must be turned ON during a G01 X,Y move that is equal to or greater than the cutter radius, or the amount being compensated for.
2. When an operation using cutter compensation is done, the cutter compensation will need to be turned OFF, using the same rules as the turn ON process, i.e., what is put in must be taken out.
3. In most machines, during cutter compensation, a linear X,Y move that is smaller than the cutter radius may not work. (Setting 58 - set to Fanuc - for positive results.)
4. Cutter compensation cannot be turned ON or OFF in a G02 or G03 arc movement.
5. With cutter compensation active, machining an inside arc with a radius less than what is defined by the active D value will cause the machine to alarm.



The following 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.



Programming exercise showing tool path.

The following program uses no cutter compensation. Tool path is programmed to centerline of the cutter. This is also the way the control calculates for cutter compensation.



The following program uses cutter compensation. Tool path is programmed to centerline of the cutter. This is also the way the control calculates for cutter compensation.

```
O6100
T1 M06
G00 G90 G54 X-1. Y-1. S5000 M03
G43 H01 Z.1 M08
G01 Z-1.0 F50.
G41 G01 X0 Y0 D1. F50.
Y4.125
G02 X.250 Y4.375 R.375
G01 X1.6562
G02 X2.0 Y4.0313 R.3437
G01 Y3.125
G03 X2.375 Y2.750 R.375
G01 X3.5
G02 X4.0 Y2.25 R.5
G01 Y.4375
G02 X3.4375 Y-.125 R.5625
G01 X-.125
G40 X-1. Y-1.
G00 Z1.0 M09
G28 G91 Y0 Z0
M30
```



MACROS

INTRODUCTION

This control feature is optional; call your dealer for information.

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 may be run multiple times. A macro statement can assign a value to a variable or 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 program.

Here are a few examples of the applications for Macros.

- Tools For Immediate, On-Table Fixturing Many setup procedures can be semi-automated to assist the machinist. For instance, suppose a standard clamp with a standard bolthole pattern is used. If it is discovered, after setup, that a fixture will need an additional clamp and if a macro subroutine has been programmed for drilling the bolt pattern of the clamp, then the following two-step procedure is all that is needed for adding the clamp to the fixture.

1. Determine X, Y, and Z coordinates and angle where the clamp is to be placed by jogging the machine to the proposed clamp position and reading the position coordinates from the machine display.

2. Execute the following command in MDI mode:

G65 P2000 X??? Y??? Z??? A??? ;

Where “???” Are the values determined in Step 1.

Here, macro 2000 (p2000) does the work since it was designed to drill the clamp bolthole pattern at the specified angle of A. Essentially; the machinist has created a custom canned cycle.

- **Simple Patterns That Are Repeated** Patterns that recur over and over again can be defined using macros and stored. For example:

1. Bolt hole patterns
2. Slotting
3. Angular patterns, any number of holes, at any angle, with any spacing
4. Specialty milling such as soft jaws
5. Matrix Patterns, (e.g. 12 across and 15 down)
6. 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:

1. Profiling of a part to determine unknown dimensions for machining.
2. Tool calibration for offset and wear values.
3. Inspection prior to machining to determine material allowance on castings.
4. 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.

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 (see "M Codes" section)

Settings

There are 3 settings that can affect macro programs (9000 series programs), these are 9xxxx progs Lock (#23), 9xxx Progs Trace (#74) and 9xxx Progs Single BLK (#75).

Lookahead

Lookahead is of great importance to the macro programmer. The control will attempt 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,

```
#1101=1  
G04 P1.  
#1101=0
```

This is intended to turn an output ON, wait 1 second, and then turn it off. However, lookahead will cause the output to turn on then immediately back off while the dwell is being processed. G103 P1 can be used to limit lookahead to 1 blocks. To make this example work properly, it must be modified as follows:



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

```
;  
#1101=1  
G04 P1.  
;  
;  
;  
#1101=0
```

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 #100 may later be read as 7.000001, 7.000000, or 6.999999. If the statement was, "IF [#100 EQ 7]...", it may give a false reading. A safer way of programming this would be, "IF [ROUND [#100] EQ 7]...". This issue is usually only a problem when storing integers in macro variables where it is expected to see a fractional part later.

OPERATION NOTES

Macro variables can be saved or loaded through the RS-232 or USB port much like settings, and offsets.

Variable Display Page

The macro variables are displayed and can be modified through the current commands display. To get to the pages, press CURNT COMDS and use the page up/down key.

As the control interprets a program, the variable changes are displayed on the variable display page and results can be viewed.

The macro variable is set by entering a value and then pressing the Write/Enter button. Macro variables can be cleared by pressing the Origin key, which will clear all variables.

Entering the macro variable number and pressing the up/down arrow will search for that variable.

The variables displayed represent values of the variables during running of the program. At times, this may be up to 15 blocks ahead of actual machine actions. Debugging programs is easier when inserting a G103 at the beginning of a program to limit block buffering and then removing the G103 after debugging is completed.

Macro Arguments

The arguments in a G65 statement are a means of sending values to and setting the local variables of a macro subroutine.

In the previous example 2, the arguments (values of) X and Y are passed to



the macro subroutine local variables. Local variable #24 is associated with X and is set to 0.5. Similarly, Local variable #25 is associated with Y and is set to 0.25.

The following two tables indicate the mapping of the alphabetic address variables to the numeric variables used in a macro subroutine.

Alphabetic Addressing

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

Alternate Alphabetic Addressing

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

Arguments accept any floating-point value to four decimal places. If the control is in metric, it will assume thousandths (.000). In Example 3, local variable #7 will receive .0004. If a decimal is not included in an argument value, such as: G65, P9910, A1, B2, C3, the values are passed to macro subroutines according to the following table:

Integer Argument Passing (no decimal point)

Address:	A	B	C	D	E	F	G
Variable:	.001	.001	.001	1.	1.	1.	-
Address:	H	I	J	K	L	M	N
Variable	1.	.0001	.0001	.0001	1.	1.	-
Address:	O	P	Q	R	S	T	U
Variable:	-	-	.0001	.0001	1.	1.	.0001
Address:	V	W	X	Y	Z		
Variable:	.0001	.0001	.0001	.0001	.0001		

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



Example 3: G65 P2000 I1 J2 K3 I4 J5 K6;

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

Macro Variables

There are three categories of macro variables: system, global, and local.

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 .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 subroutine 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: Address: Alternate:	1 A	2 B	3 C	4 I	5 J	6 K	7 D	8 E	9 F	10 K	11 H
Variable: Address: Alternate:	12 M	13 I	14 J	15 K	16 I	17 J	18 R	10 S	20 T	21 U	22 V
Variable: Address: Alternate:	23 W	24 X	25 Y	26 Z	27 K	28 I	29 J	30 K	31 I	32 J	33 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 subroutine, 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 subroutine, 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 subroutine via an M97 or M98 does not nest the local variables. Any local variables referenced in a subroutine called by an M98 are the same variables and values that existed prior to the M97 or M98 call.



Global Variables

Global variables are variables that are accessible at all times. There is only one copy of each global variable. Global variables occur in three ranges: 100-199, 500-699 and 800-999. The global variables remain in memory when power is turned off.

Occasionally, there have been some macros written for factory installed options that use the global variables. For example, probing, pallet changers, etc. When using global variables, be sure they are not in use by another program on the machine.

System Variables

System variables give the programmer the ability to interact with a variety of control conditions. By setting a system variable, the function of the control can be modified. By reading a system variable, a program can modify its behavior based on the value in the variable. Some system variables have a Read Only status; this means that the programmer cannot modify them. A brief table of currently implemented system variables follows with an explanation of their use.

VARIABLES	USAGE
#0	Not a number (read only)
#1-#33	Macro call arguments
#100-#199	General-purpose variables saved on power off
#500-#699	General-purpose variables saved on power off
#700-#749	Hidden variables for internal use only.
#800-#999	General-purpose variables saved on power off
#1000-#1063	64 discrete inputs (read only)
#1064-#1068	Maximum axis loads for X, Y, Z, A, and B-axes respectively
#1080-#1087	Raw analog to digital inputs (read only)
#1090-#1098	Filtered analog to digital inputs (read only)
#1094	Coolant Level
#1098	Spindle load with Haas vector drive (read only)
#1100-#1139	40 discrete outputs
#1140-#1155	16 extra relay outputs via multiplexed output
#1264-#1268	Maximum axis loads for C, U, V, W, and T-axes respectively
#1601-#1800	Number of Flutes of tools #1 through 200
#1801-#2000	Maximum recorded vibrations of tools 1 through 200
#2201-#2200	Tool length offsets
#2201-#2400	Tool length wear
#2401-#2600	Tool diameter/radius offsets
#2601-#2800	Tool diameter/radius wear
#3000	Programmable alarm
#3001	Millisecond timer
#3002	Hour Timer
#3003	Single block suppression
#3004	Override control
#3006	Programmable stop with message
#3011	Year, month, day



VARIABLES	USAGE
#3002	Hour timer
#3003	Single block suppression
#3004	Override control
#3006	Programmable stop with message
#3011	Year, month, day
#3012	Hour, minute, second
#3020	Power on timer (read only)
#3021	Cycle start timer
#3022	Feed timer
#3023	Present part timer
#3024	Last complete part timer
#3025	Previous part timer
#3026	Tool in spindle (read only)
#3027	Spindle RPM (read only)
#3028	Number of pallet loaded on receiver
#3030	Single Block
#3031	Dry Run
#3032	Block Delete
#3033	Opt Stop
#3201-#3400	Actual Diameter for tools 1 through 200
#3401-#3600	Programmable coolant positions for tools 1 through 200
#3901	M30 count 1
#3902	M30 count 2
#4000-#4021	Previous block G-Code group codes
#4101-#4126	Previous block address codes

Note: 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. The mapping of the axes are x=1, y=2, ... b=5, for example, the Z machine coordinate system variable would be #5023.

VARIABLES	USAGE
#5001-#5005	Previous block end position
#5021-#5025	Present machine coordinate position
#5041-#5045	Present work coordinate position
#5061-#5069	Present skip signal position - X, Y, Z, A, B, C, U, V, W
#5081-#5085	Present tool offset
#5201-#5205	G52 Work Offsets
#5221-#5225	G54 Work Offsets
#5241-#5245	G55 Work Offsets
#5261-#5265	G56 Work Offsets
#5281-#5285	G57 Work Offsets
#5301-#5305	G58 Work Offsets
#5321-#5325	G59 Work Offsets
#5401-#5500	Tool feed timers (seconds)
#5501-#5600	Total tool timers (seconds)
#5601-#5699	Tool life monitor limit
#5701-#5800	Tool life monitor counter



VARIABLES	USAGE
#5801-#5900	Tool load monitor maximum load sensed so far
#5901-#6000	Tool load monitor limit
#6001-#6277	Settings (read only)
#6501-#6999	Parameters (read only)

Note: The low order bits of large values will not appear in the macro variables for settings and parameters.

VARIABLES	USAGE
#7001-#7006 (#14001-#14006)	G110 (G154 P1) additional work offsets
#7021-#7026 (#14021-#14026)	G111 (G154 P2) additional work offsets
#7041-#7046 (#14041-#14046)	G112 (G154 P3) additional work offsets
#7061-#7066 (#14061-#14066)	G113 (G154 P4) additional work offsets
#7081-#7086 (#14081-#14086)	G114 (G154 P5) additional work offsets
#7101-#7106 (#14101-#14106)	G115 (G154 P6) additional work offsets
#7121-#7126 (#14121-#14126)	G116 (G154 P7) additional work offsets
#7141-#7146 (#14141-#14146)	G117 (G154 P8) additional work offsets
#7161-#7166 (#14161-#14166)	G118 (G154 P9) additional work offsets
#7181-#7186 (#14181-#14186)	G119 (G154 P10) additional work offsets
#7201-#7206 (#14201-#14206)	G120 (G154 P11) additional work offsets
#7221-#7226 (#14221-#14221)	G121 (G154 P12) additional work offsets
#7241-#7246 (#14241-#14246)	G122 (G154 P13) additional work offsets
#7261-#7266 (#14261-#14266)	G123 (G154 P14) additional work offsets
#7281-#7286 (#14281-#14286)	G124 (G154 P15) additional work offsets
#7301-#7306 (#14301-#14306)	G125 (G154 P16) additional work offsets
#7321-#7326 (#14321-#14326)	G126 (G154 P17) additional work offsets
#7341-#7346 (#14341-#14346)	G127 (G154 P18) additional work offsets
#7361-#7366 (#14361-#14366)	G128 (G154 P19) additional work offsets
#7381-#7386 (#14381-#14386)	G129 (G154 P20) additional work offsets
#7501-#7506	Pallet priority
#7601-#7606	Pallet status
#7701-#7706	Part program numbers assigned to pallets
#7801-#7806	Pallet usage count
#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.



VARIABLES	USAGE
#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).
#8550	Individual tool ID
#8551	Number of Flutes of tools
#8552	Maximum recorded vibrations
#8553	Tool length offsets
#8554	Tool length wear
#8555	Tool diameter offsets
#8556	Tool diameter wear
#8557	Actual diameter
#8558	Programmable coolant position
#8559	Tool feed timer (seconds)
#8560	Total tool timers (seconds)
#8561	Tool life monitor limit
#8562	Tool life monitor counter
#8563	Tool load monitor maximum load sensed so far
#8564	Tool load monitor limit
#14401-#14406	G154 P21 additional work offsets
#14421-#14426	G154 P22 additional work offsets
#14441-#14446	G154 P23 additional work offsets
#14461-#14466	G154 P24 additional work offsets
#14481-#14486	G154 P25 additional work offsets
#14501-#14506	G154 P26 additional work offsets
#14521-#14526	G154 P27 additional work offsets
#14541-#14546	G154 P28 additional work offsets
#14561-#14566	G154 P29 additional work offsets
#14581-#14586	G154 P30 additional work offsets
•	
•	
#14781-#14786	G154 P40 additional work offsets
•	
•	
#14981-#14986	G154 P50 additional work offsets
•	
•	
#15181-#15186	G154 P60 additional work offsets
•	
•	
#15381-#15386	G154 P70 additional work offsets
•	
•	
#15581-#15586	G154 P80 additional work offsets



VARIABLES	USAGE
•	
•	
•	
#15781-#15786	G154 P90 additional work offsets
•	
•	
15881-15886	G154 P95 additional work offsets
15901-15906	G154 P96 additional work offsets
15921-15926	G154 P97 additional work offsets
15941-15946	G154 P98 additional work offsets
15961-15966	G154 P99 additional work offsets

SYSTEM VARIABLES IN-DEPTH

Variables #750 and #751

These variables collect the input from serial port 2. The programmer can test for data queued in the serial port 2 buffer and collect the data for processing. Variable #750 will inform the programmer of data waiting at the RS232 port 2. A value of 1 means that the receive buffer has data queued, otherwise a value of 0 is returned. Variable 751 collects the first character from the input buffer when data is queued. This means that the buffer content is first checked to see if it is empty; if not the value of the next character queued is returned.

1-Bit Discrete Inputs

Inputs designated as “spare” can be connected to external devices and used by the programmer.

1-Bit Discrete Outputs

The Haas control is capable of controlling up to 56 discrete outputs. However, a number of these outputs are already reserved for use by the Haas controller.

CAUTION! Do not use outputs that are reserved by the system. Using these outputs may result in injury or damage to your equipment.

The user can change the state of these outputs by writing to variables designated as “spare”. If the outputs are connected to relays, then an assignment of “1” sets the relay. An assignment of “0” clears the relay.

Referencing these outputs will return the current state of the output and this may be the last assigned value or it may be the last state of the output as set by some user M code. For example, after verifying that output #1108 is “spare”:

#1108=1; (Turns #1108 relay on)

#101=#3001+1000; (101 is 1 second from now)

WHILE [[#101 GT #3001] AND [#1109 EQ 0]] D01

END1 (Wait here 1 second or until relay #1109 goes high)

#1108=0; (Turns #1108 relay off)



If the control is not equipped with the M-code relay board, then M21 through M28 will be mapped from #1132-#1139. If the M-code relay board is installed, see the 8M-option section for information and instructions.

NOTE: Always test or dry run programs that have been developed for macros that are using new hardware.

Maximum Axis Loads

The following variables are used to contain the maximum load values for each axis. They can be cleared by cycling power to the machine or by setting the macro to zero in a program (for example, #1064=0;).

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 radius (D) along with associated wear values.

#2001-#2200	H geometry offsets (1-200) for length.
#2200-#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.

Programmable 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. The first 34 characters of the comment are used for the alarm message.

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 is updated every 20 milliseconds and thus activities can be timed with an accuracy of only 20 milliseconds.



At Power On, the millisecond timer is reset. The timer has a limit of 497 days. 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

#3003 Variable 3003 is the Single Block Suppression parameter. It overrides the Single Block function in G-code. In the following example Single Block is ignored when #3003 is set equal to 1. After M3003 is set =1, each G-code command (lines 2-5) are executed continuously even though the Single Block function is ON. When #3003 is set equal to zero, Single Block will operate as normal. The user must press Cycle Start to execute each line of code (lines 7-11).

```
#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 while running.

The first bit disables the Feed Hold button. If feed hold is not used during a section of code, set variable #3004 to 1 before the specific lines of code. After that section of code set #3004 to 0 to restore the function of the Feed Hold button. For example:

Approach code	(Feed Hold allowed)
#3004=1;	(Disables Feed Hold button)
Non-stoppable code	(Feed Hold not allowed)
#3004=0;	(Enables Feed Hold button)
Depart code	(Feed Hold allowed)

The following 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

Stops can be programmed which acts like an M00 - The control stops and waits until Cycle Start is pressed. Once Cycle Start is pressed, the program continues with the block after the #3006. In the following example, the first 15 characters of the comment are displayed on the lower left part of the screen.

```
IF [#1 EQ #0] THEN #3006=101 (comment here);
```

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

The grouping of G codes permits more efficient processing. G codes with similar functions are usually under the same group. For example, G90 and G91 are under group 3. These variables store the last or default G code for any of 21 groups. By reading the group code, a macro 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-#5005 Last Target Position

The final programmed point for the last motion block can be accessed through variables #5001-#5005, X, Y, Z, A, and B, respectively. Values are given in the current work coordinate system and can be used while the machine is in motion.

Axis Position Variables

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

#5021-#5025 Current Machine Coordinate Position

The current position in machine coordinates can be obtained through #5021-#5025, X, Y, Z, A, and B, respectively. Values CANNOT be read while machine is in motion. Value of #5023 (Z) has tool length compensation applied to it.



#5041-#5045 Current Work Coordinate Position

The current position in the current work coordinates can be obtained through #5041-5045, X, Y, Z, A, and B, respectively. The values CANNOT be read while the machine is in motion. The value of #5043 (Z) has tool length compensation applied to it.

#5061-#5069 Current Skip Signal Position

The position where the last skip signal was triggered can be obtained through #5061-#5069, X, Y, Z, A, B, C, U, V and W respectively. 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-#5085 Tool Length Compensation

The current total tool length compensation that is being applied to the tool. This includes tool length offset referenced by the current value set in H (#4008) plus the wear value.

NOTE: The mapping of the axes are x=1, y=2, ... b=5. So as an example, the Z machine coordinate system variable would be #5023.

#6996-#6999 Parameter access using macro variables

It is possible for a program to access parameters 1 to 1000 and any of the parameter bits, as follows:

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

NOTE: Variables 6998 and 6999 are read-only.

Usage

To access the value of a parameter, the number of that parameter is copied into variable 6996, after which, the value of that parameter is available using macro variable 6998, as shown:

#6996=601 (Specify parameter 601)

#100=#6998 (Copy the value of parameter 601 to variable #100)

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

#6996=57 (Specify parameter 57)

#6997=0 (Specify bit zero)

#100=#6999 (Copy parameter 57 bit 0 to variable #100)



NOTE: Parameter bits are numbered 0 through 31. 32-bit parameters are formatted, on-screen, with bit 0 at the top-left, and bit 31 at the bottom-right.

Pallet Changer

The status of the pallets, from the Automatic Pallet Changer are checked using the following 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

Offsets

All tool work offsets can be read and set within a macro expression to allow the coordinates to be preset to approximate locations, or set coordinates to values based upon results of skip signal locations and calculations. When any of offsets are read, the interpretation lookahead queue is stopped until that block is executed.

#5201-#5205	G52 X, Y, Z, A, B OFFSET VALUES
#5221-#5225	G54 " " " " "
#5241-#5245	G55 " " " " "
#5261-#5265	G56 " " " " "
#5281-#5285	G57 " " " " "
#5301-#5305	G58 " " " " "
#5321-#5325	G59 " " " " "
#7001-#7005	G110 X, Y, Z, A, B OFFSET VALUES
" "	" " " " "
#7381-#7385	G129 X, Y, Z, A, B OFFSET VALUES

#8550-#8567

These variables provide information on tooling. Set variable #8550 to the tool or tool group number, then access information for the selected tool / tool group using the read-only macros 8551-8564. If specifying a tool group number, the selected tool will be the next tool in that group.

VARIABLE USAGE

All variables are referenced with a pound sign (#) followed by a positive number: #1, #101, and #501.

Variables are decimal values that are represented as floating point numbers. If a variable has never been used, it can take on a special “undefined” value. This indicates that it has not been used. A variable can be set to undefined with the special variable #0. #0 has the value of undefined or 0.0 depending on the context it is used in. Indirect references to variables can be accomplished by enclosing the variable number in brackets: #[<Expression>]



The expression is evaluated and the result becomes the variable accessed. For example:

```
#1=3;  
#[#1]=3.5 + #1;
```

This sets the variable #3 to the value 6.5.

Variables can be used in place of G-code address where “address” refers to the letters A-Z.

In the block: N1 G0 G90 X1.0 Y0; the variables can be set to the following values:

```
#7=0;  
#11=90;  
#1=1.0;  
#2=0.0;
```

and replaced by: N1 G#7 G#11 X#1 Y#2; Values in the variables at runtime are used as the address values.

ADDRESS SUBSTITUTION

The usual method of setting control addresses A-Z is the address followed by a number. For example:

G01 X1.5 Y3.7 F20. ;

Sets addresses G, X, Y and F to 1, 1.5, 3.7 and 20.0 respectively and thus instructs the control to move linearly, G01, to position X=1.5 Y=3.7 at a feed rate of 20 inches per minute. Macro syntax allows the address values to be replaced with any variable or expression.

The previous statement can be replaced by the following code:

```
#1=1;  
#2=.5;  
#3=3.7;  
#4=20;  
G#1 X[#1+#2] Y#3 F#4 ;
```

The permissible syntax on addresses A-Z (exclude N or O) is as follows:

<address><-><variable>	A-#101
<address>[<expression>]	Y[#5041+3.5]
<address><->[<expression>]	Z-[SIN[#1]]

If the variable value does not agree with the address range, the control will generate an alarm. For example, the following code would result in a range error alarm because tool diameter numbers range from 0-50.

```
#1=75;
```



D#1;

When a variable or expression is used in place of an address value, the value is rounded to the least significant digit. If #1=.123456, then G1X#1 would move the machine tool to .1235 on the X axis. If the control is in the metric mode, the machine would be moved to .123 on the X axis.

When an undefined variable is used to replace an address value, then that address reference is ignored. For example, if #1 is undefined then the block

G00 X1.0 Y#1 ;

Becomes

G00 X1.0.

No Y movement takes place.

Macro Statements

Macro statements are lines of code that allow the programmer to manipulate the control with features similar to any standard programming language. Included are functions, operators, conditional and arithmetic expressions, assignment statements, and control statements.

Functions and operators are used in expressions to modify variables or values. The operators are essential to expressions while functions make the programmer's job easier.

Functions

Functions are built-in routines that the programmer has available to use. All functions have the form <function_name>[argument] and return floating-point decimal values. The functions provided in the Haas control are as follows:

Function	Argument	Returns	Notes
SIN[]	Degrees	Decimal	Sine
COS[]	Degrees	Decimal	Cosine
TAN[]	Degrees	Decimal	Tangent
ATAN[]	Decimal	Degrees	Arctangent Same as FANUC ATAN[]/[1]
SQRT[]	Decimal	Decimal	Square root
ABS[]	Decimal	Decimal	Absolute value
ROUND[]	Decimal	Decimal	Round off a decimal
FIX[]	Decimal	Integer	Truncate fraction
ACOS[]	Decimal	Degrees	Arccosine
ASIN[]	Decimal	Degrees	Arcsine
#[]	Integer	Integer	Variable Indirection
DPRNT[]	ASCII text	External Output	

Notes on Functions

The function "Round" works differently depending on the context that it is used. When used in arithmetic expressions, any number with a fractional part greater than or equal to .5 is rounded up to the next whole integer; otherwise, the fractional part is truncated from the number.



```
#1= 1.714 ;
#2= ROUND[#1] ; (#2 is set to 2.0)
#1= 3.1416 ;
#2= ROUND[#1] ; (#2 is set to 3.0)
```

When round is used in an address expression, "Round" is rounded to the significant precision. For metric and angle dimensions, three-place precision is the default. For inch, four-place precision is the default.

```
#1= 1.00333 ;
G0 X[ #1 + #1 ] ;
(Table moves to 2.0067) ;
G0 X[ ROUND[ #1 ] + ROUND[ #1 ] ] ;
(Table moves to 2.0066) ;
G0 A[ #1 + #1 ] ;
(Axis moves to 2.007) ;
G0 A[ ROUND[ #1 ] + ROUND[ #1 ] ] ;
(Axis moves to 2.006) ;
D[1.67]      (Diameter 2 is made current) ;
```

Fix vs. Round

#1=3.54; #2=ROUND[#1]; #3=FIX[#1]. #2 will be set to 4. #3 will be set to 3.

Operators

Operators can be classified into three categories: Arithmetic, Logical and Boolean.

Arithmetic Operators

Arithmetic operators consist of unary and binary operators. They are:

+	- Unary plus	+1.23
-	- Unary minus	-[COS[30]]
+	- Binary addition	#1=#1+5
-	- Binary subtraction	#1=#1-1
*	- Multiplication	#1=#2*#3
/	- Division	#1=#2/4
MOD	- Remainder	#1=27 MOD 20 (#1 contains 7)

Logical Operators

Logical operators are operators that work on binary bit values. Macro variables are floating point numbers. When logical operators are used on macro variables, only the integer portion of the floating point number is used. The logical operators are:

OR - logically OR two values together
 XOR - Exclusively OR two values together
 AND - Logically AND two values together

Examples:



```
#1=1.0; 0000 0001  
#2=2.0; 0000 0010  
#3=#1 OR #2; 0000 0011  
  
#1=5.0;  
#2=3.0;  
IF [#1 GT 3.0] AND [#2 LT 10]] GOTO1
```

Here the variable #3 will contain 3.0 after the OR operation.

Here control will transfer to block 1 because "#1 GT 3.0" evaluates to 1.0 and "#2 LT 10" evaluates to 1.0, thus 1.0 AND 1.0 is 1.0 (TRUE) and the GOTO occurs.

Note that care must be taken when using logical operators so that the desired result is achieved.

Boolean Operators

Boolean operators always evaluate to 1.0 (TRUE) or 0.0 (FALSE). There are six Boolean operators. These operators are not restricted to conditional expressions, but they most often are used in conditional expressions. They are:

- EQ - Equal to
- NE - Not Equal to
- GT - Greater Than
- LT - Less Than
- GE - Greater than or Equal to
- LE - Less Than or Equal to

The following are four examples of how Boolean and Logical operators can be used:

Example

```
IF [#1 EQ 0.0] GOTO100;  
  
WHILE [#101 LT 10] DO1;  
  
#1=[1.0 LT 5.0];  
IF [#1 AND #2 EQ #3] GOTO1
```

Explanation

Jump to block 100 if value in variable #1 equals 0.0.
While variable #101 is less than 10 repeat loop DO1..END1.
Variable #1 is set to 1.0 (TRUE).
If variable #1 logically ANDed with variable #2 is equal to the value in #3 then control jumps to block 1.

Expressions

Expressions are defined as any sequence of variables and operators surrounded by the square brackets "[" and "]". There are two uses for expressions: conditional expressions or arithmetic expressions. Conditional expressions return FALSE (0.0) or TRUE (any non zero) values. Arithmetic expressions use arithmetic operators along with functions to determine a value.

Conditional Expressions

In the HAAS control, ALL expressions set a conditional value. The value is either 0.0 (FALSE) or the value is nonzero (TRUE). The context in which the expression is used determines if the expression is a conditional expression.



Conditional expressions are used in the IF and WHILE statements and in the M99 command. Conditional expressions can make use of Boolean operators to help evaluate a TRUE or FALSE condition.

The M99 conditional construct is unique to the HAAS control. Without macros, M99 in the HAAS control has the ability to branch unconditionally to any line in the current subroutine by placing a P code on the same line. For example: **N50 M99 P10**; branches to line N10. It does not return control to the calling subroutine. With macros enabled, M99 can be used with a conditional expression to branch conditionally. To branch when variable #100 is less than 10 we could code the above line as follows: **N50 [#100 LT 10] M99 P10;**

In this case, the branch occurs only when #100 is less than 10, otherwise processing continues with the next program line in sequence. In the above, the conditional M99 can be replaced with **N50 IF [#100 LT 10] GOTO10;**

Arithmetic Expressions

An arithmetic expression is any expression using variables, operators, or functions. An arithmetic expression returns a value. Arithmetic expressions are usually used in assignment statements, but are not restricted to them.

Examples of Arithmetic expressions:

```
#101=#145*#30;  
#1=#1+1;  
X[#105+COS[#101]];  
#[#2000+#13]=0;
```

Assignment Statements

Assignment statements allow the programmer to modify variables. The format of the assignment statement is:

<expression>=<expression>

The expression on the left of the equal sign must always refer to a macro variable, whether directly or indirectly. The following macro initializes a sequence of variables to any value. Here both direct and indirect assignments are used.

```
O0300  
N1 IF [#2 NE #0] GOTO2  
#3000=1  
N2 IF [#19 NE #0] GOTO3  
#3000=2  
N3 WHILE [#19 GT 0] DO1 ;  
#19=#19-1  
#[#2+#19]=#22  
END1 ;  
M99 ;
```

(Initialize an array of variables) ;
(B=base variable) ;
(Base variable not given) ;
(S=size of array);
(Size of array not given) ;
(Decrement count) ;
(V=value to set array to) ;

The above macro could be used to initialize three sets of variables as follows:



G65 P300 B101. S20 (INIT 101..120 TO #0) ;
G65 P300 B501. S5 V1 (INIT 501..505 TO 1.0) ;
G65 P300 B550. S5 V0 (INIT 550..554 TO 0.0) ;

The decimal point in B101., etc. would be required.

Control Statements

Control statements allow the programmer to branch, both conditionally and unconditionally. They also provide the ability to iterate a section of code based on a condition.

Unconditional Branch (GOTOnnn and M99 Pnnnn)

In the Haas control, there are two methods of branching unconditionally. An unconditional branch will always branch to a specified block. M99 P15 will branch unconditionally to block number 15. The M99 can be used whether or not macros is installed and is the traditional method for branching unconditionally in the Haas control. GOTO15 does the same as M99 P15. In the Haas control, a GOTO command can be used on the same line as other G-codes. The GOTO is executed after any other commands like M codes.

Computed Branch (GOTO#n and GOTO [expression])

Computed branching allows the program to transfer control to another line of code in the same subprogram. The block can be computed as the program is running, using the GOTO [expression] form. Or the block can be passed in through a local variable, as in the GOTO#n form.

The GOTO will round the variable or expression result that is associated with the Computed branch. For instance, if #1 contains 4.49 and GOTO#1 is executed, the control will attempt to transfer to a block containing N4. If #1 contains 4.5, then execution will transfer to a block containing N5.

The following code skeleton could be developed to make a program that adds serial numbers to parts:

O9200	(Engrave digit at current location.)
;	
(D=Decimal digit to engrave);	
IF [#7 NE #0] AND [#7 GE 0] AND [#7 LE 9]	
GOTO99;	
#3000=1	(Invalid digit)
;	
N99	
#7=FIX[#7]	(Truncate any fractional part)
;	
GOTO#7	(Now engrave the digit)
;	
N0	(Do digit zero)



```
...
M99
;
N1                                (Do digit one)
;
M99
;
N2                                (Do digit two)
;
...
;
(etc.,...)
```

With the above subroutine, you would engrave digit five with the following call:
G65 P9200 D5;

Computed GOTOs using expression could be used to branch processing based on the results of reading hardware inputs. An example might look like the following:

```
GOTO [#1030*2]+#1031;
NO (1030=0, 1031=0);
...
M99;
N1 (1030=0, 1031=1);
...
M99;
N2 (1030=1, 1031=0);
...
M99;
N3 (1030=1, 1031=1);
...
M99;
```

The discrete inputs always return either 0 or 1 when read. The GOTO [expression] will branch to the appropriate line of code based on the state of the two discrete inputs #1030 and #1031.

Conditional Branch (IF and M99 Pnnnn)

Conditional branching allows the program to transfer control to another section of code within the same subroutine. Conditional branching can only be used when macros are enabled. The Haas control allows two similar methods for accomplishing conditional branching.

IF [<conditional expression>] GOTO n

As discussed, <conditional expression> is any expression that uses any of the six Boolean operators EQ, NE, GT, LT, GE, or LE. The brackets surrounding the expression are mandatory. In the Haas control, it is not necessary to include these operators. For example: IF [#1 NE 0.0] GOTO5; could also be: IF [#1] GOTO5;



In this statement, if the variable #1 contains anything but 0.0, or the undefined value #0, then branching to block 5 will occur; otherwise, the next block will be executed.

In the Haas control, a conditional expression can also be used with the M99 Pnnn format. For example:

```
G0 X0 Y0 [#1EQ#2] M99 P5;
```

Here, the conditional is for the M99 portion of the statement only. The machine tool is instructed to X0, Y0 whether or not the expression evaluates to True or False. Only the branch, M99, is executed based on the value of the expression. It is recommended that the IF GOTO version is used if portability is desired.

Conditional Execution (IF THEN)

Execution of control statements can also be achieved by using the IF THEN construct. The format is:

```
IF [<conditional expression>] THEN <statement>;
```

Note: To preserve compatibility with FANUC syntax "THEN" may not be used with GOTOn.

This format is traditionally used for conditional assignment statements such as:

```
IF [#590 GT 100] THEN #590=0.0 ;
```

Variable #590 is set to zero when the value of #590 exceeds 100.0. In the Haas control, if a conditional evaluates to FALSE (0.0), then the remainder of the IF block is ignored. This means that control statements can also be conditioned so that we could write something like:

```
IF [#1 NE #0] THEN G1 X#24 Y#26 F#9 ;
```

This executes a linear motion only if variable #1 has been assigned a value. Another example is:

```
IF [#1 GE 180] THEN #101=0.0 M99 ;
```

This says that if variable #1 (address A) is greater than or equal to 180, then set variable #101 to zero and return from the subroutine.

Here is an example of an IF statement that branches if a variable has been initialized to contain any value. Otherwise, processing will continue and an alarm will be generated. Remember, when an alarm is generated, program execution is halted.

```
N1 IF [#9NE#0] GOTO3 (TEST FOR VALUE IN F) ;
N2 #3000=11(NO FEED RATE) ;
N3 (CONTINUE) ;
```



Iteration/Looping (WHILE DO END)

Essential to all programming languages is the ability to execute a sequence of statements a given number of times or to loop through a sequence of statements until a condition is met. Traditional G coding allows this with the use of the L address. A subroutine can be executed any number of times by using the L address.

M98 P2000 L5;

This is limited since you cannot terminate execution of the subroutine on condition. Macros allow flexibility with the WHILE-DO-END construct. For example:

```
WHILE [<conditional expression>] DOn;  
    <statements>;  
    ENDn;
```

This executes the statements between DOn and ENDn as long as the conditional expression evaluates to True. The brackets in the expression are necessary. If the expression evaluates to False, then the block after ENDn is executed next. WHILE can be abbreviated to WH. The DOn-ENDn portion of the statement is a matched pair. The value of n is 1-3. This means that there can be no more than three nested loops per subroutine. A good example of how nesting of WHILE loops can be used is in defining a matrix.

```
#101= 3;  
#102= 4;  
G0 X#101 Y4. ;  
F2.5;  
WH [#101 GT 0] DO1;  
#102= 4;  
WH [#102 GT 0] DO2;  
G81 X#101 Y#102 Z-0.5;  
#102= #102 - 1;  
END2;  
#101= #101 - 1;  
END1;  
;  
M30;
```

This program drills a 3 x 4 matrix hole pattern.

Although nesting of WHILE statements can only be up to three levels, there really is no limit since each subroutine can have up to three levels of nesting. If there is a need to nest to a level greater than 3, then the segment containing the three lowest levels of nesting can be made into a subroutine thus overcoming the limitation.

If two separate WHILE loops are in a subroutine, they can use the same nesting index. For example:

```
#3001=0 (WAIT 500 MILLISECONDS);  
WH [#3001 LT 500] DO1;
```



```
END1;  
<Other statements>  
#3001=0 (WAIT 300 MILLISECONDS) ;  
WH [#3001 LT 300] DO1 ;  
END1 ;
```

You can use GOTO to jump out of a region encompassed by a DO-END, but you cannot use a GOTO to jump into it. Jumping around inside a DO-END region using a GOTO is allowed.

An infinite loop can be executed by eliminating the WHILE and expression. Thus,

```
DO1;  
<statements>  
END1;           executes until the RESET key is pressed.  
CAUTION! The following code can be confusing: WH [#1] D01;  
END1;
```

In this example, an alarm results indicating no “Then” was found; “Then” refers to the D01. Change D01 (zero) to DO1 (letter O).

G65 MACRO SUBROUTINE CALL

G65 is the command that calls a subroutine with the ability to pass arguments to it. The format follows.

G65 Pnnnn [Lnnnn] [arguments];

Anything italicized in square brackets is optional. The G65 command requires a P address corresponding to a program number currently in the control's memory. When the L address is used the macro call is repeated the specified number of times. In Example 1, subroutine 1000 is called once without conditions passed to the subroutine. 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 subroutine 1000 as a macro)  
M30;                (Program stop)  
O1000;              (Macro Subroutine)  
...  
M99;                (Return from Macro Subroutine)
```

In Example 2, subroutine 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 subroutine is called.

Example 2:

```
G00 G90 X1.0 Y1.0 Z.05 S1000 M03;          (Position tool)
G65 P9010 X.5 Y.25 Z.05 F10. T10;          (Call 9010)
G28;
M30;
O9010;                                         (Diagonal hole pattern)
F#9;                                            (F=Feed rate)
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

Aliasing is a means of assigning a G-code or M-code to a G65 P##### sequence. For instance, in Example 2 it would be easier to write: G06 X.5 Y.25 Z.05 F10. T10;

When aliasing, variable can be passed with a G-code; variable 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 parameter associated with subroutine 9010 must be set to 06 (Parameter 91). Note that G00, G65, G66, and G67 cannot be aliased. All other codes between 1 and 255 can be used for aliasing.

Program numbers 9010 through 9019 are reserved for G code aliasing. The following table lists which HAAS parameters are reserved for macro subroutine aliasing.

Haas Parameter	O Code
91	9010
92	9011
93	9012
94	9013
95	9014
96	9015
97	9016
98	9017
99	9018
100	9019

G Code Aliasing

Haas Parameter	M Macro Call
81	9000
82	9001
83	9002
84	9003
85	9004
86	9005
87	9006
88	9007
89	9008
90	9009

M Code Aliasing



COMMUNICATION WITH EXTERNAL DEVICES - DPRNT[]

Macros allow additional capabilities to communicate with peripheral devices. One can do digitizing of parts, provide runtime inspection reports, or synchronize controls with user provided devices. The commands provided for this are POPEN, DPRNT[] and PCLOS.

Communication Preparatory Commands

POPN and PCLOS are not required on the Haas mill. It has been included so that programs from different controls can be sent to the Haas control.

Formatted Output

The DPRNT statement allows the programmer to send formatted text to the serial port. Any text and any variable can be printed to the serial port. The form of the DPRNT statement is as follows: DPRNT [<text> <#nnnn[wf]>...] ;

DPRNT must be the only command in the block. In the previous example, <text> is any character from A to Z or the letters (+,-,/,* and the space). When an asterisk is output, it is converted to a space. The <#nnnn[wf]> is a variable followed by a format. The variable number can be any macro variable. The format [wf] is required and consists of two digits within square brackets. Remember that macro variables are real numbers with a whole part and a fractional part. The first digit in the format designates the total places reserved in the output for the whole part. The second digit designates the total places reserved for the fractional part. The total places reserved for output cannot be equal to zero or greater than eight. Thus the following formats are illegal: [00] [54] [45] [36] /* not legal formats */

A decimal point is printed out between the whole part and the fractional part. The fractional part is rounded to the least significant place. When zero places are reserved for the fractional part, then no decimal point is printed out. Trailing zeros are printed if there is a fractional part. At least one place is reserved for the whole part, even when a zero is used. If the value of the whole part has fewer digits than have been reserved, then leading spaces are output. If the value of the whole part has more digits than has been reserved, then the field is expanded so that these numbers are printed.

A carriage return is sent out after every DPRNT block.

DPRNT[] Examples

Code	Output
N1 #1= 1.5436 ;	
N2 DPRNT[X#1[44]*Z#1[03]*T#1[40]] ;	X1.5436 Z 1.544 T 1
N3 DPRNT[***MEASURED*INSIDE*DIAMETER***] ;	MEASURED INSIDE DIAMETER
N4 DPRNT[] ;	(no text, only a carriage return)
N5 #1=123.456789 ;	
N6 DPRNT[X-#1[25]] ;	X-123.45679 ;



Execution

DPRNT statements are executed at block interpretation time. This means that the programmer must be careful about where the DPRNT statements appear in the program, particularly if the intent is to print out.

G103 is useful for limiting lookahead. If you wanted to limit lookahead interpretation to one block, you would include the following command at the beginning of your program: (This actually results in a two block lookahead.)

G103 P1;

To cancel the lookahead limit, change the command to G103 P0. G103 cannot be used when cutter compensation is active.

Editing

Improperly structured or improperly placed macro statements will generate an alarm. Be careful when editing expressions; brackets must be balanced.

The DPRNT[] function can be edited much like a comment. It can be deleted, moved as a whole item, or individual items within the bracket can be edited. Variable references and format expressions must be altered as a whole entity. If you wanted to change [24] to [44], place the cursor so that [24] is highlighted, enter [44] and press the write key. Remember, you can use the jog handle to maneuver through long DPRNT[] expressions.

Addresses with expressions can be somewhat confusing. In this case, the alphabetic address stands alone. For instance, the following block contains an address expression in X:

G1 G90 X [COS [90]] Y3.0; CORRECT

Here, the **X** and brackets stand-alone and are individually editable items. It is possible, through editing, to delete the entire expression and replace it with a floating-point constant.

G1 G90 X 0 Y3.0 ; WRONG

The above block will result in an alarm at runtime. The correct form looks as follows:

G1 G90 X0 Y3.0; CORRECT

Note that there is no space between the X and the Zero (0). REMEMBER when you see an alpha character standing alone it is an address expression.

FANUC-STYLE MACRO FEATURES NOT INCLUDED IN HAAS CNC

This section lists the FANUC macro features that are not available on the Haas control.

M Aliasing Replace G65 Pnnnn with Mnn PROGS 9020-9029.



G66	Modal call in every motion block
G66.1	Modal call in every motion block
G67	Modal cancel
M98	Alaising, T code PROG 9000, VAR #149, enable bit
M98	Aliasing, S Code PROG 9029, VAR #147, enable bit
M98	Aliasing, B Code PROG 9028, VAR #146, enable bit
SKIP/N	N=1..9
#3007	Mirror image on flag each axis

#4201-#4320

Current block modal data

#5101-#5106

Current servo deviation

Names for Variables for Display Purposes

ATAN []/[]

Arctangent, FANUC version

BIN []

Conversion from BCD TO BIN

BCD []

Conversion from BIN TO BCD

FUP []

Truncate fraction ceiling

LN []

Natural logarithm

EXP []

Base E Exponentiation

ADP []

Re-Scale variable to whole number

BPRNT []

The following can be used as alternative methods for achieving the same results for a few of the unavailable FANUC macro features.

GOTO-nnnn

Searching for a block to jump in the negative direction, i.e. backwards through a program, is not necessary if you use unique N address codes.

A block search is made starting from the current block being interpreted. When the end of the program is reached, searching continues from the top of the program until the current block is encountered.

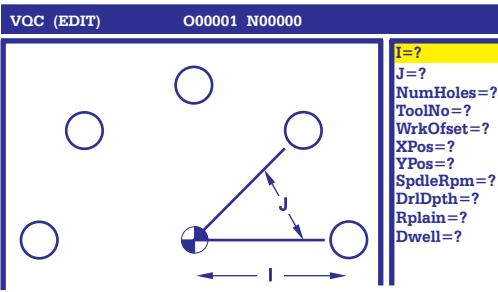
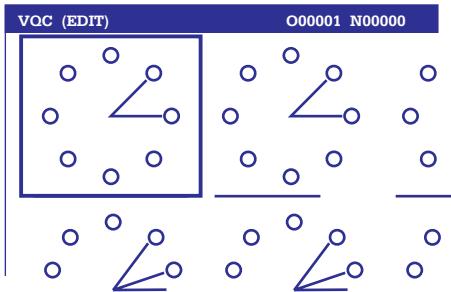


VISUAL QUICK CODE

To start Visual Quick Code (VQC), press MDI/DNC, then press the PRGRM/CONVRS key. Select VQC from the tabbed menu.

SELECTING A CATEGORY

Use the arrow keys to select the parts category whose description closely matches the desired part and press Write. A set of illustrations of the parts in that category will appear.



Selecting a Part Template

Use the arrow keys to select a template on the page. Pressing Write will display an outline of the part and wait for the programmer to enter values to make the selected part.

Entering the Data

The control will prompt the programmer for information about the selected part. Once the information is entered, the control another asks the user where the G-code is to be placed:

1) Select/Create a Program

A window will open prompting the user to select a program name. Highlight the desired name and press Write. This will add the new lines of code to the selected program. If the program already contains code, VQC enter the lines of code to the beginning of the program, before the existing code. The user also has the option to Create a new program by entering a program name and by pressing Write, this will add the lines of code to the new program.

2) Add to Current Program – The code generated by VQC will be added after the cursor.

3) MDI – The code will output to MDI. Note: Anything in MDI will be overwritten.

4) Cancel – The window will close and the program values will be displayed.

NOTE: The program is also available for editing in Edit Mode. It is advised to check the program by running it in Graphics mode.





SUBROUTINES

Subroutines (subprograms) are usually a series of commands that are repeated several times in a program. Instead of repeating the commands many times in the main program, subroutines are written in a separate program. The main program then has a single command that “calls” the subroutine program. A subroutine is called using an M97 and a P address. The P code is the same as the sequence number (Onnnnn) of the subroutine to be called, that is located after an M30. A subprogram is called using an M98 and a P address. The P address with an M98 is for the program number.

Canned Cycles are the most common use of subroutines. The X and Y locations of the holes are placed in a separate program and then called. Instead of writing the X, Y locations once for each tool, the X, Y locations are written once for any number of tools.

The subroutines can include an L or repeat count. If there is an L, the subroutine call is repeated that number of times before the main program continues with the next block.

EXTERNAL SUBROUTINE

An external subroutine is a separate program that is referenced several times by the main program. Local subroutines are commanded (called) using an M98 and a Pnnnn which refers it to the program number of the sub program.

Subroutines

External Subroutine Example

O00104 (sub program with an M98)

T1 M06

G90 G54 G00 X1.5 Y-0.5

S1406 M03

G43 H01 Z1. M08

G81 G99 Z-0.26 R0.1 F7.

M98 P105 (Call Sub-Program O00105)

Sub Program

O00105

X.5 Y-.75

Y-2.25

G98 X1.5 Y-2.5

G99 X3.5

X4.5 Y-2.25

Y-.75

X3.5 Y-.5

G80 G00 Z1.0 M09

G53 G49 Z0. M05

M99

T2 M06

G90 G54 G00 X1.5 Y-0.5

S2082 M03

G43 H02 Z1. M08

G83 G99 Z-.75 Q0.2 R0.1 F12.5

M98 P105 (Call Sub-Program O00105)

T3 M06

G90 G54 G00 X1.5 Y-0.5

S750

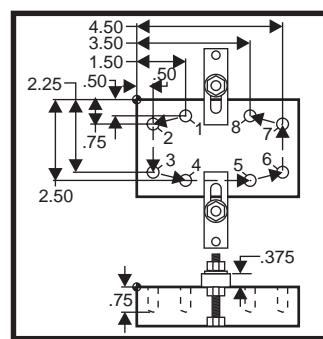
G43 H03 Z1. M08

G84 G99 Z-.6 R0.1 F37.5

M98 P105 (Call Sub-Program O00105)

G53 G49 Y0.

M30 (End Program)





LOCAL SUBROUTINE EXAMPLE

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

The local subroutine format is to end the main program with an M30 then enter the local subroutines after the M30. Each subroutine 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.

Local Subroutine Example

```
O00104 (local sub program with an M97)
T1 M06
G90 G54 G00 X1.5 Y-0.5
S1406 M03
G43 H01 Z1. M08
G81 G99 Z-0.26 R0.1 F7.
M97 P1000 (Call local subroutine at line N1000)
T2 M06
G90 G54 G00 X1.5 Y-0.5
S2082 M03
G43 H02 Z1. M08
G83 G99 Z-.75 Q0.2 R0.1 F12.5
M97 P1000 (Call local subroutine at line N1000)
T3 M06
G90 G54 G00 X1.5 Y-0.5
S750
G43 H03 Z1. M08
G84 G99 Z-.6 R0.1 F37.5
M97 P1000 (Call local subroutine at line N1000)
G53 G49 Y0.
M30 (End Program)
N1000 (Begin local subroutine)
X.5 Y-.75
Y-2.25
G98 X1.5 Y-2.5
G99 X3.5
X4.5 Y-2.25
Y-.75
X3.5 Y-.5
G80 G00 Z1.0 M09
G53 G49 Z0. M05
M99
```



SUBROUTINE CANNED CYCLE EXAMPLE

O1234 (Canned Cycle Example Program)
T1 M06
G90 G54 G00 X.565 Y-1.875 S1275 M03
G43 H01 Z.1 M08
G82 Z-.175 P.03 R.1 F10.
M98 P1000
G80 G00 Z1.0 M09
T2 M06
G00 G90 G54 X.565 Y-1.875 S2500 M03
G43 H02 Z.1 M08
G83 Z-.720 Q.175 R.1 F15.
M98 P1000
G00 G80 Z1.0 M09
T3 M06
G00 G90 G54 X.565 Y-1.875 S900 M03
G43 H03 Z.2 M08
G84 Z-.600 R.2 F56.25
M98 P1000
G80 G00 Z1.0 M09
G28 G91 Y0 Z0
M30

Sub Program
O1000 (X,Y
Locations)
X 1.115 Y-2.750
X 3.365 Y-2.875
X 4.188 Y-3.313
X 5.0 Y-4.0
M99



SUBROUTINES WITH MULTIPLE FIXTURES

Subroutines can be useful when cutting the same part in different X and Y locations within the machine. For example, there are six vises mounted on the table. Each of these vises uses a new X, Y zero. They are referenced in the program using the G54 through G59 work offsets. Use an edge finder or an indicator to establish the zero point on each part. Use the part zero set key in the work coordinate offset page to record each X, Y location. Once the X, Y zero position for each workpiece is in the offset page, the programming can begin.

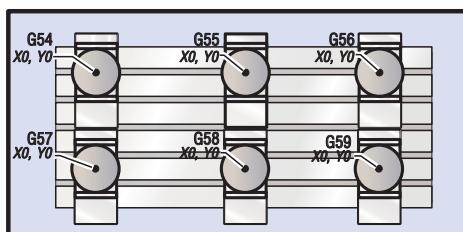
The figure shows what this setup would look like on the machine table. For an example, each of these six parts will need to be drilled at the center, X and Y zero.

Main Program

```
O2000  
T1 M06  
G00 G90 G54 X0 Y0 S1500 M03  
G43 H01 Z.1 M08  
M98 P3000  
G55  
M98 P3000  
G56  
M98 P3000  
G57  
M98 P3000  
G58  
M98 P3000G59  
M98 P3000  
G00 Z1.0 M09  
G28 G91 Y0 Z0  
M30
```

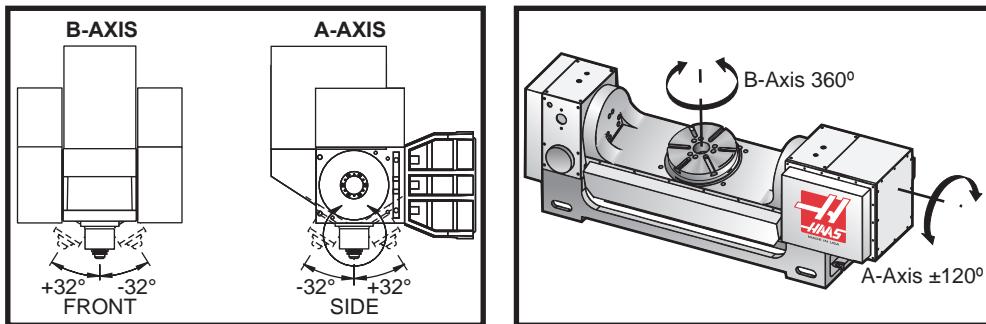
Subroutine

```
O3000  
X0 Y0  
G83 Z-1.0 Q.2 R.1 F15.  
G00 G80 Z.2  
M99
```





4TH AND 5TH AXIS PROGRAMMING

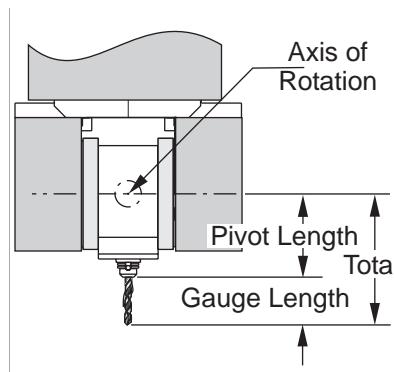


Axis motion on the VR-11 Mill and the Haas TRT 210

CREATING FIVE-AXIS PROGRAMS

Most five-axis programs are rather complex and should be written using a CAD/CAM package. It is necessary to determine the pivot length and gauge length of the machine, and input them into these programs.

Each machine has a specific pivot length. This is the distance from the spindle head's center of rotation to the bottom surface of the master tool holder. The pivot length can be found in Setting 116, and is also engraved into the master tool holder that is shipped with a 5-axes machine.



4th & 5th
Axis

When setting up a program, it is necessary to determine gauge length for each tool. Gauge length is the distance from the bottom flange of the master tool holder to the tip of the tool. This distance can be calculated by setting a magnetic base indicator on the table, indicating the bottom surface of the master tool holder, and setting this point as Z0 in the control. Then, insert each tool and calculate distance from the tool tip to Z0; this is the gauge length.



The total length is the distance from the spindle head center of rotation to the tip of the tool. It can be calculated by adding the gauge length and pivot length. This number is entered into the CAD/CAM program, which will use the value for its calculations.

Offsets

The work-offset display is found on the offset display by pushing the Page Up button. The G54 through G59 or G110 through G129 offsets can be set by using the Part Zero Set button. Position the axes to the work zero point of the workpiece. Using the cursor, select the proper axis and work number. Press the Part Zero Set button and the current machine position will be automatically stored in that address. This will work with only the work zero offsets display selected. Note that entering a nonzero Z work offset will interfere with the operation of an automatically entered tool length offset.

Work coordinate numbers are usually entered as positive numbers. Work coordinates are entered into the table as a number only. To enter an X value of X2.00 into G54, cursor to the X column and enter 2.0.

Five-axis Programming Notes

Use a tight synchronization cut across resolution of geometry in the CAD/CAM system will allow smooth flowing contours and a more accurate part.

Positioning the machine to an approach vector should only be done at a safe distance above or to the side of the workpiece. When in the rapid mode, the axes will arrive at the programmed position at different times; the axis with shortest distance from target will arrive first, and longest distance last. A high feed rate will force the axes to arrive at the commanded position at the same time avoiding the possibility of a crash.

G Codes

Fifth-axis programming is not affected by the selection of inch (G20) or metric (G21), as A and B axes are always programmed in degrees.

G93 inverse time must be in effect for simultaneous 5-axis motion. In G93 mode, the maximum feed rate will include the combination of all axis motion in one block of code. The limit is set by the control and looks at encoder steps programmed for all axes in a block of code.

Limit the post processor (CAD/CAM software) if possible; the maximum speed in G93 mode is 32 degrees per minute. This will result in smoother motion, which may be necessary when fanning around tilted walls.

M Codes

Important! It is highly recommended that the A/B brakes be engaged when doing any non 5-axis motion. Cutting with the brakes off can cause excessive wear in the gear sets.

M10/M11 engages/disengages the A-axis brake



M12/M13 engages/disengages the B-axis brake

When in a 4 or 5 axis cut, the machine will pause between blocks. This pause is due to the A and/or B axis brakes releasing. To avoid this dwell and allow for smoother program execution, program an M11 and/or M13 just before the G93. The M-codes will disengage the brakes, resulting in a smoother motion and an uninterrupted flow of motion. Remember that if the brakes are never re-engaged, they will remain off indefinitely.

Settings

A number of settings are used to program the 4th and 5th axis. See settings 30, 34 and 48 for the 4th axis and 78, 79 and 80 for the 5th axis.

Setting 85 should be set to .0500 for 5-axis cutting. Settings lower than .0500 will move the machine closer to an exact stop and cause uneven motion.

G187 can also be used in the program to slow the axes down.

Caution! When cutting in 5-axis mode poor positioning and over-travel can occur if the tool length offset (H-code) is not canceled. To avoid this problem use G90 G40, H00 and G49 in the first blocks after a tool change. This problem can occur when mixing 3-axis and 5-axis programming, restarting a program or when starting a new job and the tool length offset is still in effect.

Feed Rates

A feed-rate must be commanded for each line of 4 and/or 5 axis code. Limit the feed-rate to less than 75IPM when drilling. The recommended feeds for finish machining in 3-axis work should not exceed 50 to 60 IPM with at least .0500" to .0750" stock remaining for the finish operation.

Rapid moves are not allowed; rapid motions, entering and exiting holes (full retract peck-drill cycle) are not supported.

When programming simultaneous 5-axis motion, less material allowance is required and higher feedrates may be permitted. Depending on finish allowance, length of cutter and type of profile being cut, higher feed rates may be possible. For example, when cutting mold lines or long flowing contours, feedrates may exceed 100 IPM.

4th & 5th
Axis

Jogging the 4th and 5th Axis

All aspects of handle jogging for the fifth axis work as they do for the other axes. The exception is the method of selecting jog between axis A and axis B.

By default the '+A' and '-A' keys, when pressed, will select the A axis for jogging. The B axis can be selected for jogging by pressing the Shift button, and then pressing either the '+A' or '-A' key.

EC-300: Jog mode shows A1 and A2, use "A" to jog A1 and Shift "A" to jog A2.



EC-300 Pallet and 4th axis Operation

The rotary table in the machining area will always appear, and operate, as the A axis. The rotary axis on pallet 1 is referred to by "A1" and the other axis, on pallet 2, by "A2". Operation examples:

To jog axis A1, enter "A1" and press "HAND JOG".

To key jog, use the +/-A jog buttons to jog the A1 axis and +/-B buttons to jog the A2 axis.

To zero return the A axis on pallet #2, enter "A2" press ZERO SINGL AXIS.

Mirroring Feature: If G101 is used to mirror the A axis, then mirroring is turned on for both A axes. When pallet #1 is in the machining area, A1-MIR will be displayed at the bottom of the screen. When pallet #2 is in the machine, A2-MIR will be displayed. The behavior of the Mirror Settings is different, if Setting 48 Mirror Image A-Axis is ON, only the A axis on pallet #1 is mirrored and the message A1-MIR is displayed.

If setting 80 (parameter 315, bit 20 MAP 4TH AXIS is a 1, the name for setting 80 is the same as that for Setting 48, i.e. Mirror Image A-Axis) is ON mirroring will be turned on for the A-axis on pallet #2. When pallet #2 is inside of the mill, A2-MIR will be displayed.

Crash Recovery Procedure

If the machine crashes while cutting a five-axis part, it can often be difficult to clear the tool away from the part due to the angles involved. Do Not immediately press the Recover button or turn the power off. To recover from a crash in which the spindle is stopped while the tool is still in a cut, retract the spindle using the Vector Jog feature. To do this press the letter "V" on the keypad, press "Handle Jog", and use the jog handle to move along that axis. This feature will allow motion along any axes determined by A and/or B axis.

The Vector Jog feature is intended to allow the operator to clear the cutting tool from the part in an extreme situation as a result of a crash or an alarm condition.

G28 is not available in vector jog mode; it is only available for the X, Y, Z, A, and B when selecting single axis.

If there was a loss of power during a cut, vector jog will not work as the control requires a reference position. Other means of clearing the tool from the part will be necessary.

If the tool is not in a cut when it is crashed, press the Recover button and answer the questions that appear on the screen. When the Recover button is pressed, the spindle head will move the A, B and Z-axes simultaneously, in order to retract the tool. If the tool is in a cut at an angle, it will crash when this key is pressed.



INSTALLING AN OPTIONAL FOURTH AXIS

When adding a rotary table to the Haas mill change settings 30 and 34 to the specific rotary table and part diameter currently used. Warning: Failure to match the correct brush or brushless rotary setting to the actual product being installed on the mill may cause motor damage. "B" in the settings denotes a brushless rotary product. Brushless indexers have two cables from table and two connectors at the mill control.

Parameters

In rare cases some parameters may need to be modified to get a specific performance out of the indexer. Do not do this without a list of parameters to change. (If a list of parameters was not included with the indexer, then no change is necessary. DO NOT CHANGE THE PARAMETERS. Doing so will void your warranty.)

To change parameters for a fourth or fifth axis indexer: Press the E-stop button. Turn off the parameter lock (Setting 7). Go to the settings page by pressing the Setting button. Enter "7" and press the down arrow; this will jump to setting 7. With the cursor on Setting 7, use the right or left arrow button to select "Off" and press the Write button, to turn off the parameter lock. Go to the parameter page and enter the number of the parameter to change and press the down arrow button. Enter the new value for the new parameter and press the Write button; change the other parameters similarly. Return to the Setting 7 and turn this setting back on. Reset the E-stop button. Home the indexer and verify that it operates properly by pressing Handle Jog and the "A" button. Jog the A axis using the jog handle, the indexer should move. Check for proper ratio by marking the table, rotate 360 degrees as viewed in the position page and check mark to be in the same location. If it is close (within 10 degrees) then the ratio is correct.

Initial Start-up

Turn on the mill (and servo control, if applicable) and home the indexer. All Haas indexers home in the clockwise direction as viewed from the front. If the indexer homes counter-clockwise, press E-stop and call your dealer.

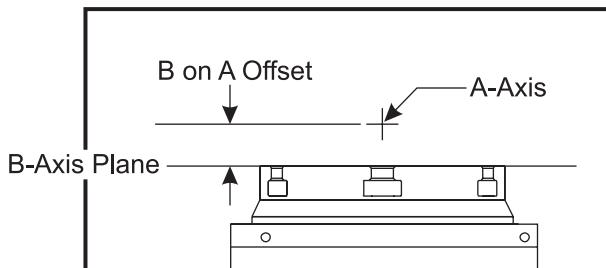
4th & 5th
Axis

INSTALLING AN OPTIONAL FIFTH AXIS

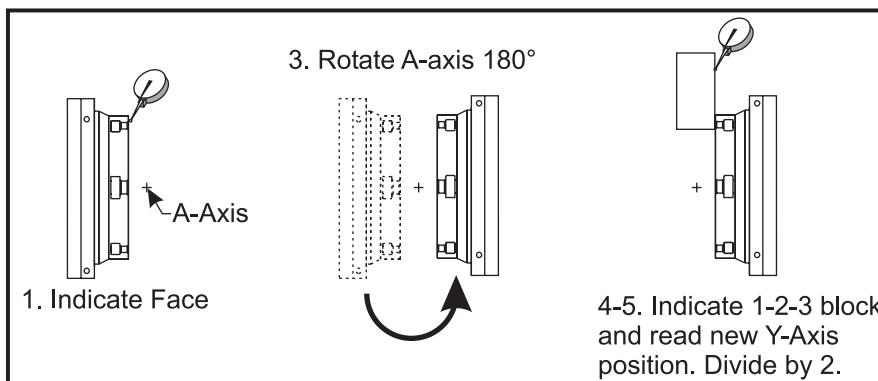
The fifth axis is installed in the same manner as the fourth axis. Settings 78 and 79 govern the 5th axis, and the axis is jogged and commanded using the B address.

B ON A AXIS OFFSET (TILTING ROTARY PRODUCTS)

This procedure determines the distance between the plane of the B-axis platter and the A-axis centerline on tilting rotary products. The offset is required by some CAM software applications.



1. Rotate the A-axis until the B-axis is vertical. Mount a dial indicator on the machine spindle (or other surface independent of table motion) and indicate the platter face. Set the indicator to zero.
2. Set the Y-axis operator position to zero (select the position and press ORIGIN).
3. Rotate the A-axis 180°.
4. The platter face must now be indicated from the same direction as the first indication. Place a 1-2-3 block against the platter face and indicate the face of the block that rests against the platter face. Move the Y-axis to meet the block with the indicator tip. Reset the indicator to zero.
5. Read the new Y-axis position. Divide this value by 2 to determine the B on A axis offset value.



B on A Axis Illustrated Procedure

DISABLING THE AXES

Turn setting 30 for the 4th axis and 78 for the 5th axis off when it is removed from the machine. Do not connect or disconnect any cables with the control on. If the settings are not turned off when the unit is removed, an alarm will be generated.



G CODES / M CODES / SETTINGS

Section Contents

G00 Rapid Motion Positioning (Group 01).....	157
G01 Linear Interpolation Motion (Group 01).....	157
G02 CW / G03 CCW Circular Interpolation Motion (Group 01).....	158
G04 Dwell (Group 00).....	163
G09 Exact Stop (Group 00).....	163
G10 Set Offsets (Group 00).....	163
G12 Circular Pocket Milling CW / G13 Circular Pocket Milling CCW (Group 00).....	164
G17 XY / G18 XZ / G19 YZ plane selection (Group 02).....	166
G20 Select Inches / G21 Select Metric (Group 06)	167
G28 Return to Machine Zero Thru Optional G29 Ref. Point (Group 00)	167
G29 Return From Reference Point (Group 00)	168
G31 Feed Until Skip (Group 00).....	168
G35 Automatic Tool Diameter Measurement (Group 00)	168
G36 Automatic Work Offset Measurement (Group 00).....	169
G37 Automatic Tool Offset Measurement (Group 00)	170
G40 Cutter Comp Cancel (Group 07).....	171
G41 2D Cutter Compensation Left / G42 2D Cutter Comp. Right (Group 07)....	171
G43 Tool Length Comp + / G44 Tool Length Comp - (Group 08).....	171
G47 Text Engraving (Group 00).....	172
G49 G43/G44/G143 Cancel (Group 08).....	176
G50 Cancel Scaling (Group 11).....	176
G51 Scaling (Group 11).....	176
G52 Set Work Coordinate System (Group 00 or 12).....	178
G53 Non-Modal Machine Coordinate Selection (Group 00).....	179
G54-59 Select Work Coordinate System #1 - #6 (Group 12).....	179
G60 Uni-Directional Positioning (Group 00)	179
G61 Exact Stop Mode (Group 15).....	179
G64 G61 Cancel (Group 15)	179
G68 Rotation (Group 16).....	179
G69 Cancel G68 Rotation (Group 16).....	182
G70 Bolt Hole Circle (Group 00).....	182
G71 Bolt Hole Arc (Group 00).....	182
G72 Bolt Holes Along an Angle (Group 00).....	182
G73 High-Speed Peck Drilling Canned Cycle (Group 09).....	188
G74 Reverse Tap Canned Cycle (Group 09).....	190
G76 Fine Boring Canned Cycle (Group 09)	191
G77 Back Bore Canned Cycle (Group 09)	192
G80 Canned Cycle Cancel (Group 09).....	192
G81 Drill Canned Cycle (Group 09).....	193
G82 Spot Drill Canned Cycle (Group 09)	193
G83 Normal Peck Drilling Canned Cycle (Group 09)	194
G84 Tapping Canned Cycle (Group 09)	196
G85 Boring Canned Cycle (Group 09)	197
G86 Bore and Stop Canned Cycle (Group 09).....	197

G & M Codes
Settings



G87 Bore In and Manual Retract Canned Cycle (Group 09).....	198
G88 Bore In, Dwell, Manual Retract Canned Cycle (Group 09).....	198
G89 Bore In, Dwell, Bore Out Canned Cycle (Group 09).....	199
G90 Absolute Position Commands (Group 03)	199
G91 Incremental Position Commands (Group 03)	199
G92 Set Work Coordinate Systems Shift Value (Group 00).....	200
G93 Inverse Time Feed Mode (Group 05).....	200
G94 Feed Per Minute Mode (Group 05).....	201
G95 Feed per Revolution (Group 05).....	201
G98 Canned Cycle Initial Point Return (Group 10)	201
G99 Canned Cycle R Plane Return (Group 10)	201
G100 Cancel Mirror Image (Group 00).....	201
G101 Enable Mirror Image (Group 00).....	201
G102 Programmable Output to RS-232 (Group 00).....	204
G103 Limit Block Buffering (Group 00).....	204
G107 Cylindrical Mapping (Group 00).....	204
G107 Description.....	205
G110-G129 Coordinate System #7-26 (Group 12).....	207
G136 Automatic Work Offset Center Measurement (Group 00).....	207
G141 3D+ Cutter Compensation (Group 07).....	208
G143 5-Axis Tool Length Compensation + (Group 08)	209
G150 General Purpose Pocket Milling (Group 00).....	210
G153 5-Axis High Speed Peck Drilling Canned Cycle (Group 09).....	216
G154 Select Work Coordinates P1-P99 (Group 12).....	216
G155 5-Axis Reverse Tap Canned Cycle (Group 09).....	218
G161 5-Axis Drill Canned Cycle (Group 09).....	218
G162 5-Axis Spot Drill Canned Cycle (Group 09)	219
G163 5-Axis Normal Peck Drilling Canned Cycle (Group 09)	220
G164 5-Axis Tapping Canned Cycle (Group 09)	222
G165 5-Axis Boring Canned Cycle (Group 09)	223
G166 5-Axis Bore and Stop Canned Cycle (Group 09)	224
G169 5-Axis Bore and Dwell Canned Cycle (Group 09)	224
G174 CCW Non-Vertical Rigid Tap (Group 00)	225
G184 CW Non-Vertical Rigid Tap (Group 00)	225
G187 Setting the Smoothness Level (Group 00).....	226
G188 Get Program From PST (Group 00)	226
M Codes	227
Settings.....	239



G CODES (PREPARATORY FUNCTIONS)

G00 Rapid Motion Positioning (Group 01)

- X Optional X-axis motion command
- Y Optional Y-axis motion command
- Z Optional Z-axis motion command
- A Optional A-axis motion command

G00 is used to move the machine's axis at the maximum speed. It is primarily used to quickly position the machine to a given point before each feed (cutting) command (All moves are done at full rapid speed). This G code is modal, so a block with G00 causes all following blocks to be rapid motion until another Group 01 code is specified.

Programming note: Generally, rapid motion will not be in a straight line. Each axis specified is moved at the same speed, but all axes will not necessarily complete their motions at the same time. The machine will wait until all motions are complete before starting the next command.

Incremental or absolute position commands (G90 or G91) will change how those axis motion values are interpreted. Setting 57 (Exact Stop Canned X-Y) can change how closely the machine waits for a precise stop before and after a rapid move.

G01 Linear Interpolation Motion (Group 01)

- F Feedrate in inches (mm) per minute
- X Optional X-axis motion command
- Y Optional Y-axis motion command
- Z Optional Z-axis motion command
- A Optional A-axis motion command
- ,R Radius of the arc
- ,C Chamfer distance

This G code moves the axes at a commanded feed rate. It is primarily used to cut the workpiece. A G01 feed can be a single axis move or a combination of the axes. The rate of axes movement is controlled by feedrate (F) value. This F value can be in units (inch or metric) per minute (G94) or per spindle revolution (G95), or time to complete the motion (G93). The feedrate value (F) can be on the current program line, or a previous line. The control will always use the most recent F value until another F value is commanded.

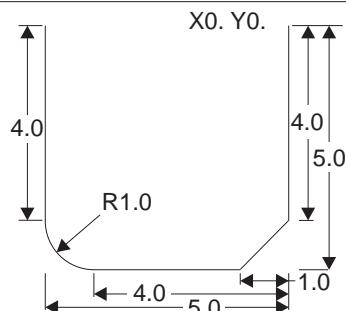
G01 is a modal command, which means that it will stay in effect until canceled by a rapid command such as G00 or a circular motion command like G02 or G03.

Once a G01 is started all programmed axes will move and reach the destination at the same time. If an axis is not capable of the programmed feedrate the control will not proceed with the G01 command and an alarm (max feedrate exceeded) will be generated.



Corner Rounding and Chamfering Example

```
O1234 (Corner Rounding and Chamfering Example);
T1 M6;
G00 G90 G54 X0. Y0. S3000 M3;
G43 H01 Z0.1 M08;
G01 Z-0.5 F20. ;
Y-5. ,C1. ;
X-5. ,R1. ;
Y0. ;
G00 Z0.1 M09;
G53 G49 Z0. ;
G53 Y0. ;
M30;
```



A chamfer block or a corner-rounding block can be automatically inserted between two linear interpolation blocks by specifying ,C (chamfering) or ,R (corner rounding). There must be a terminating linear interpolation block following the beginning block (a G04 pause may intervene).

These two linear interpolation blocks specify a corner of intersection. If the beginning block specifies a C, the value following the C is the distance from the intersection to where the chamfer begins, and also the distance from the intersection to where the chamfer ends. If the beginning block specifies an R, the value following the R is the radius of a circle tangent to the corner at two points: the beginning of the corner-rounding arc and the endpoint of that arc. There can be consecutive blocks with chamfering or corner rounding specified. There must be movement on the two axes specified by the selected plane, whether the active plane is **XY** (G17), **XZ** (G18) or **YZ** (G19).

G02 CW / G03 CCW Circular Interpolation Motion (Group 01)

F	Feedrate in inches (mm) per minute
I	Optional distance along X axis to center of circle
J	Optional distance along Y axis to center of circle
K	Optional distance along Z axis to center of circle
R	Optional radius of circle
X	Optional X-axis motion command
Y	Optional Y-axis motion command
Z	Optional Z-axis motion command
A	Optional A-axis motion command

Using I,J and K is most preferred method for programming a radius. R is suitable for most general radii.

These G codes are used to specify circular motion. Two axes are necessary to complete circular motion and the correct plane, G17-19, must be used.

There are two methods of commanding a G02 or G03, the first is using the I, J, K addresses and the second is using the R address. A chamfer or corner-rounding feature can be added to the program, by specifying ,C (chamfering) or ,R (corner rounding), as described in the G01 definition.



Using I, J, K addresses

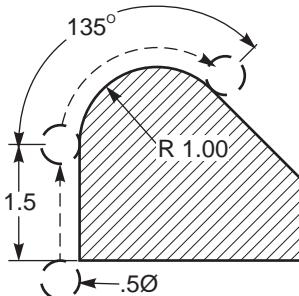
I, J and K address are used to locate the arc center in relation to the start point. In other words, the I, J, K addresses are the distances from the starting point to the center of the circle. Only the I, J or K specific to the selected plane are allowed (G17 uses IJ, G18 uses IK and G19 uses JK). The X, Y, and Z commands specify the end point of the arc. If the X, Y, or Z location for the selected plane is not specified, the endpoint of the arc is the same as the starting point for that axis.

To cut a full circle the I, J, K addresses must be used; using an R address will not work. To cut a full circle, do not specify an ending point (X, Y and Z); program I, J or K to define the center of the circle. For example: G02 I3.0 J4.0 (Assumes G17; XY plane)

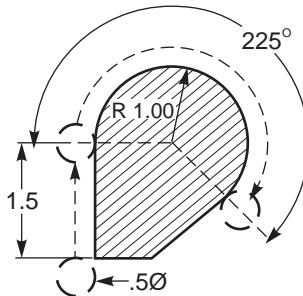
Using the R address

The R-value defines the distance from the starting point to the center of the circle. Use a positive R-value for radii of 180° or less, and a negative R-value for radii more than 180°.

Programming Examples



G90 G54 G00 X-0.25 Y-.25
G01 Y1.5 F12.
G02 X1.884 Y2.384 R1.25



G90 G54 G00 X-0.25 Y-0.25
G01 Y1.5 F12.
G02 X1.884 Y0.616 R-1.25

Corner Rounding and Chamfering example:

```

G00 X1. Y1.
G01 Z-0.125 F30.
G01 X5 ,C0.75 F12
G01 Y1.75
G01 X6. ,C0.25
G01 Y5. ,R06.25
G01 X5.
G01 Y8. ,C0.5
G01 X1. ,R1.
G01 Y1.
G00 X0.75 Y0.75

```



Thread Milling

Thread milling uses a standard G02 or G03 move to create the circular move in X-Y, then adds a Z move on the same block to create the thread pitch. This generates one turn of the thread; the multiple teeth of the cutter generate the rest. Typical line of code: N100 G02 I-1.0 Z-.05 F5. (generates 1-inch radius for 20-pitch thread)

Thread Milling notes: Internal holes smaller than 3/8 inch may not be possible or practical. Always climb cut the cutter.

Use a G03 to cut I.D. threads or a G02 to cut O.D. threads. An I.D. right hand thread will move up in the Z-axis by the amount of one thread pitch. An O.D. right hand thread will move down in the Z-axis by the amount of one thread pitch. PITCH = 1/Threads per inch (Example - 1.0 divided by 8 TPI = .125)

Thread Milling Example:

This program will I.D. thread mill a 1.5 x 8 TPI hole using a .750 diameter x 1.0 thread hob.

To start, take the hole diameter (1.500). Subtract the cutter diameter .750 and then divide by 2. $(1.500 - .75) / 2 = .375$

The result (.375) is the distance the cutter starts from the I.D. of the part.

After the initial positioning, the next step of the program is to turn on cutter compensation and move to the I.D. of the circle.

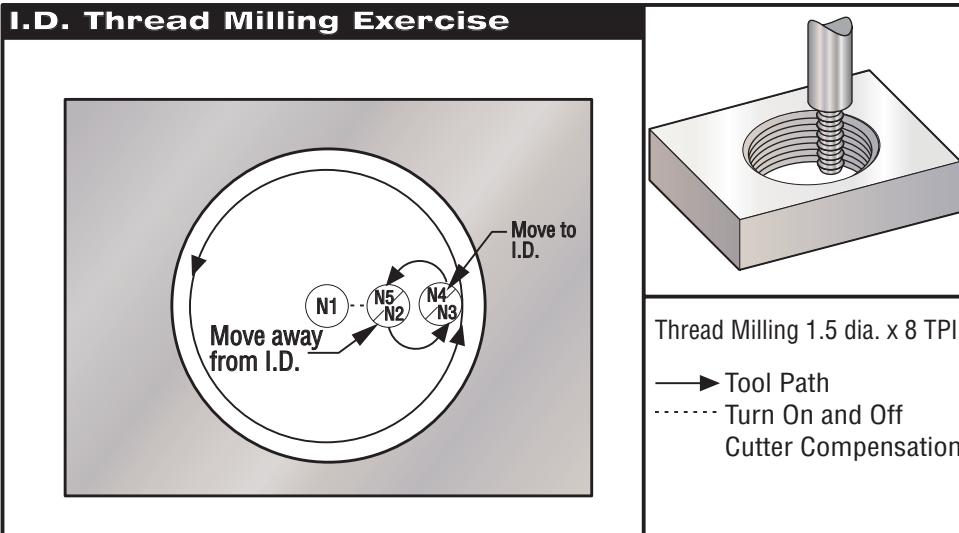
The next step is to program a complete circle (G02 or G03) with a Z-axis command of the amount of one full pitch of the thread (this is called "helical interpolation")

The last step is to move away from the I.D. of the circle and turn off cutter compensation

Cutter compensation cannot be turned off or on during an arc movement. A linear move must be made, either in the X or Y axis to move the tool to and from the diameter to cut. This move will be the maximum compensation amount that can be adjusted.



I.D. Thread Milling Exercise



Thread Milling Example

Program Example

%

O02300

T1 M06 (0.5 DIA 2FLT. THREAD MILL)
G00 G90 G54 X0. Y0. S1910 M03
G43 H01 Z0.1 M08

G00 Z-0.6

N1 G01 G41 D01 X0.125 F30.
N2 G03 X0.75 Y0. R0.3125 F11.5
N3 G03 I-0.75 Z-0.475
N4 G03 X0.125 Y0. R0.3125 F30.
N5 G01 G40 X0. Y0.

G00 Z0.1 M09

G28 G91 Y0. Z0.

M30

%

Description

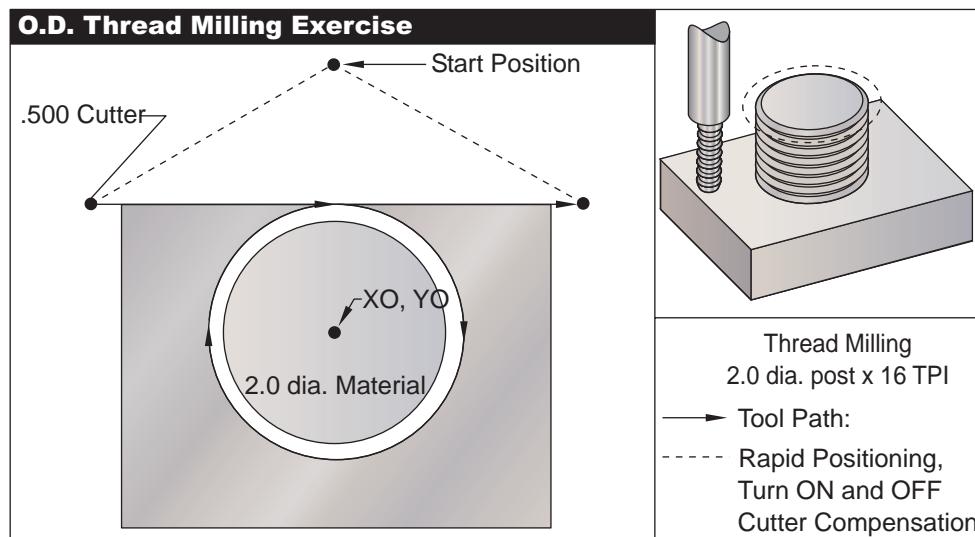
(Thread milling 1.5 x 8 TPI)
(X0. Y0. is at the center of the hole)
(Z0. is at the top of the part - using .5" thick material)

(Turn on Cutter Comp)
(Move to I.D. of bored hole)
(One full revolution with Z moving up .125)
(Move away from the new threads)
(Cancel Cutter Comp)

Note: Maximum cutter compensation adjustability is .175.



O.D. Thread Milling



O.D. Thread Milling Example

Program Example

```
%  
O02400  
T1 M06 (0.5 DIA. 2FLT. THREAD MILL) (Thread milling a 2.0 diameter post X 16 TPI)  
G00 G90 G54 X-0.2 Y1.4 S1910 M30 (X0, Y0 is at the center of the post)  
G43 H01 Z0.1 M08 (Z0 at the top of the part—Post height is 1.125")  
G00 Z-1.  
G01 G41 D01 Y1. F30. (Turn on Cutter Compensation)  
G01 X0. F11.5 (Linear move to the post)  
G02 J-0.962 Z-1.0625 (Circular move; negative Z move)  
G01 X0.2 (Linear move away from the post)  
G01 G40 Y1.4 F30. (Turn off Cutter Compensation)  
G00 Z0.1 M09  
G28 G91 Y0. Z0.  
M30  
%
```

Description

Note: A cutter compensation move can consist of any X or Y move from any position just as long as the move is greater than the amount being compensated for.

Single-Point Thread Milling Example

The program is for a 2.500 diameter hole, with a cutter diameter of .750" a radial value of .875 and a thread pitch of .0833 (12 TPI) and a part thickness of 1.0.



Program Example	Description
%	
O1000	(X0,Y0 is at the center of the hole, Z0 is at the top of the part)
T1 M06	(Tool #1 is a .750 diameter single-point thread tool!)
G00 G90 G54 X0 Y0 S2500 M03	
G43 H01 Z.1 M08	
G01 Z-1.083 F35.	
G41 X.275 D1	(Radial value)
G3	X.875 I.3 F15.
G91 G3 I-.875 Z.0833 L14	(Multiply .0833 pitch x 14 passes = 1.1662 Z-axis move)
G90 G3 X.275 I-.300	
G00 G90 Z1.0 M09	
G1	G40 X0 Y0
G28 G91 Y0 Z0	
M30	
%	

Helical Motion

Helical (spiral) motion is possible with G02 or G03 by programming the linear axis that is not in the selected plane. This third axis will be moved along the specified axis in a linear manner, while the other two axes will be moved in the circular motion. The speed of each axis will be controlled so that the helical rate matches the programmed feedrate.

G04 Dwell (Group 00)

P The dwell time in seconds or milliseconds

G04 is used to cause a delay or dwell in the program. The block containing G04 will delay for the time specified by the P code. For example G04 P10.0. This will delay the program for 10 seconds. Note the use of the decimal point G04 P10. is a dwell of 10 seconds; G04 P10 is a dwell of 10 milliseconds.

G09 Exact Stop (Group 00)

The G09 code is used to specify a controlled axes stop. It only affects the block in which it is commanded; it is non-modal, it does not affect the following blocks. Machine moves will decelerate to the programmed point before another command is processed.

G & M Codes
Settings

G10 Set Offsets (Group 00)

G10 allows the programmer to set offsets within the program. Using G10 replaces the manual entry of offsets (i.e. Tool length and diameter, and work coordinate offsets).

L – Selects offset category.

L2 Work coordinate origin for G52 and G54-G59

L10 Length offset amount (for **H** code)

L1 or L11 Tool wear offset amount (for **H** code)

L12 Diameter offset amount (for **D** code)



L13 Diameter wear offset amount (for **D** code)
L20 Auxiliary work coordinate origin for G110-G129

P – Selects a specific offset.

P1-P100 Used to reference **D** or **H** code offsets (L10-L13)

P0 G52 references work coordinate (L2)

P1-P6 G54-G59 references work coordinates (L2)

P1-P20 G110-G129 references auxiliary coordinates (L20)

P1-P99 G154 P1-P99 reference auxiliary coordinate (L20)

R Offset value or increment for length and diameter.

X Optional X-axis zero location.

Y Optional Y-axis zero location.

Z Optional Z-axis zero location.

A Optional A-axis zero location.

Programming Examples

G10 L2 P1 G91 X6.0 {Move coordinate G54 6.0 to the right};

G10 L20 P2 G90 X10. Y8.0; {Set work coordinate G111 to X10.0 ,Y8.0};

G10 L10 G90 P5 R2.5 {Set offset for Tool #5 to 2.5};

G10 L12 G90 P5 R.375 {Set diameter for Tool #5 to .375"};

G10 L20 P50 G90 X10. Y20. {Set work coordinate G154 P50 to X10. Y20.}

G12 Circular Pocket Milling CW / G13 Circular Pocket Milling CCW (Group 00)

These two G codes are used to mill circular shapes. They are different only in which direction of rotation is used. Both G codes use the default XY circular plane (G17) and imply the use of G42 (cutter compensation) for G12 and G41 for G13. These two G-codes are non-modal.

*D Tool radius or diameter selection

I Radius of first circle (or finish if no K). I value must be greater than Tool Radius, but less than K value.

K Radius of finished circle (if specified)

L Loop count for repeating deeper cuts

Q Radius increment, or stepover (must be used with K)

F Feedrate in inches (mm) per minute

Z Depth of cut or increment

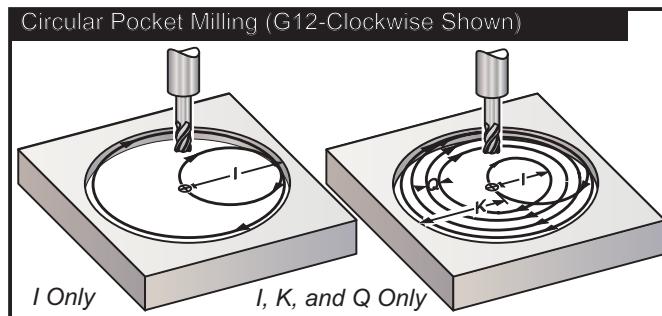
*In order to get the programmed circle diameter, the control uses the selected D code tool size. To program tool centerline select D0.

NOTE: Specify D00 if no cutter compensation is desired. If no D is specified in the G12/G13 block, the last commanded D value will be used, even if it was previously canceled with a G40.

The tool must be positioned at the center of the circle using X and Y. To remove all the material within the circle, use I and Q values less than the tool diameter and a K value equal to the circle radius. To cut a circle radius only, use an I value set to the radius and no K or Q value.



%
000098 (SAMPLE G12 AND G13)
(OFFSET D01 SET TO APPROX. TOOL
SIZE)
(TOOL MUST BE MORE THAN Q IN
DIAM.)
T1M06
G54G00G90X0Y0 (Move to center of G54)
G43Z0.1H01
S2000M03
G12I1.5F10.Z-1.2D01 (Finish pocket clockwise)
G00Z0.1
G55X0Y0 (Move to center of G55)
G12I0.3K1.5Q0.3F10.Z-1.2D01 (Rough and finish clockwise)
G00Z0.1
G56X0Y0 (Move to center of G56)
G13I1.5F10.Z-1.2D01 (Finish pocket counterclockwise)
G00Z0.1
G57X0Y0 (Move to center of G57)
G13I0.3K1.5Q0.3F10.Z-1.2D01 (Rough and finish counterclockwise)
G00Z0.1
G28
M30



These G codes assume the use of cutter compensation, so a G41 or G42 is not required in the program line. However, a D offset number, for cutter radius or diameter, is required to adjust the circle diameter.

The following programming examples show the G12 and G13 format, as well as the different ways these programs can be written.

Single Pass: Use I only.

Applications: One-pass counter boring; rough and finish pocketing of smaller holes, ID cutting of O-ring grooves.

Multiple Pass: Use I, K, and Q.

Applications: Multiple-pass counter boring; rough and finish pocketing of large holes with cutter overlap.



Multiple Z-Depth Pass: Using **I** only, or **I**, **K**, and **Q** (G91 and **L** may also be used).

Applications: Deep rough and finish pocketing.

The previous figures show the tool path during the pocket milling G-codes.

Example G13 multiple-pass using **I**, **K**, **Q**, **L**, and G91:

This program uses G91 and an **L** count of **4**, so this cycle will execute a total of four times. The **Z** depth increment is 0.500. This is multiplied by the **L** count, making the total depth of this hole 2.000.

The G91 and **L** count can also be used in a G13 “I only” line.

Note: If geometry column of control Offsets display has a value inserted, G12/G13 will read the data, whether a D0 is present or not. To cancel cutter compensation insert a D00 in the program line, this will bypass the value in the Offsets geometry column.

Program Example	Description
%	
O4000	(0.500 entered in the Radius/Diameter offset column)
T1 M06	(Tool #1 is a 0.500" diameter endmill)
G00 G90 G54 X0 Y0 S4000 M03	
G43 H01 Z.1 M08	
G01 Z0 F30.	
G13 G91 Z-.5 I.400 K2.0 Q.400 L4	
D01 F20.	
G00 G90 Z1.0 M09	
G28 G91 Y0 Z0	
M30	
%	

G17 XY / G18 XZ / G19 YZ plane selection (Group 02)

The face of the workpiece to have a circular milling operation (G02, G03, G12, G13) done to it must have two of the three main axes (X, Y and Z) selected. One of three G codes is used to select the plane, G17 for XY, G18 for XZ, and G19 for YZ. Each is modal and applies to all subsequent circular motions. The default plane selection is G17, which means that a circular motion in the XY plane can be programmed without selecting G17. Plane selection also applies to G12 and G13, circular pocket milling, (always in the XY plane).

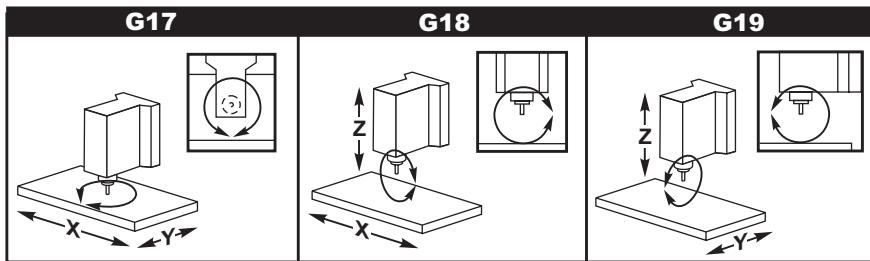
If cutter radius compensation is selected (G41 or G42), only use the XY plane (G17) for circular motion.

G17 Defined - Circular motion with the operator looking down on the XY table from above. This defines the motion of the tool relative to the table.

G18 Defined - Circular motion is defined as the motion for the operator looking from the rear of the machine toward the front control panel.



G19 Defined - Circular motion is defined as the motion for the operator looking across the table from the side of the machine where the control panel is mounted.

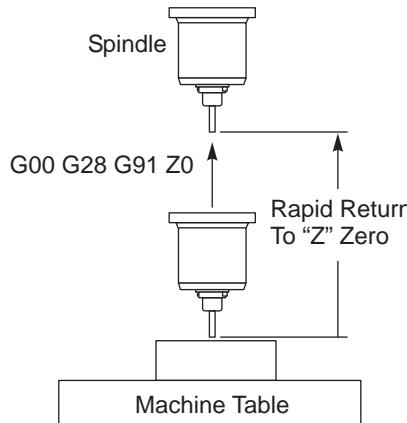


G20 Select Inches / G21 Select Metric (Group 06)

The G codes G20 (inch) and G21 (mm) codes are used to ensure that the inch/metric selection is set correctly for the program. Selection between inch and metric programming should be done using Setting 9.

G28 Return to Machine Zero Thru Optional G29 Reference Point (Group 00)

G28 is used to return all axes to machine zero, unless an axis (or axes) is specified, in which case only that axis (or axes) is returned to machine zero. G28 cancels tool length offsets for the following lines of code.



**G & M Codes
Settings**

Example 1

Work Offset G54: Z = 2.0

Tool 2 Length: 12.0

Program segment:

G90 G54;

G43 H02;

G28 Z0.;

G00 Z1.



The G28 block will move to machine coordinate Z = 14.0 before moving to Z = 0. The following block (G00 Z1.) will move to machine coordinate Z = 1.

Example 2 (same work and tool offsets as Example 1)

Program segment:

```
G54;  
G43 H02;  
G00 G91 G28 Z0
```

The G28 block will move directly to machine coordinate Z = 0 since incremental positioning is in effect.

G29 Return From Reference Point (Group 00)

The G29 code is used to move the axes to a specific position. The axes selected in this block are moved to the G29 reference point saved in G28, and then moved to the location specified in the G29 command.

G31 Feed Until Skip (Group 00)

(This G-code is optional and requires a probe)

F	Feedrate in inches (mm) per minute
X	X-axis absolute motion command
Y	Y-axis absolute motion command
Z	Z-axis absolute motion command
A	A-axis absolute motion command
B	B-axis absolute motion command

This G-code moves the axes to the programmed position. It applies only to the block in which G31 is specified. The specified move is started and continues until the position is reached or the probe receives a signal (skip signal). The control will beep when the end of travel is reached.

Do not use Cutter Compensation with a G31.

Use the assigned M-codes (for example M52 and M62), with a dwell, to turn the table probe on and off

For example:

```
M53  
G04 P100  
M63
```

Also see M75, M78 and M79.

G35 Automatic Tool Diameter Measurement (Group 00)

(This G-code is optional and requires a probe)

F	Feedrate in inches (mm) per minute
D	Tool diameter offset number
X	Optional X-axis command
Y	Optional Y-axis command

Automatic Tool Diameter Offset Measurement function (G35) is used to set the tool diameter (or radius) using two passes of the probe; one on each side of



the tool. The first point is set with a G31 block using an M75, and the second point is set with the G35 block. The distance between these two points is set into the selected (non-zero) Dnnn offset. Setting 63 (Tool Probe Width) is used to reduce the measurement of the tool by the width of the tool probe.

This G-code moves the axes to the programmed position. The specified move is started and continues until the position is reached or the probe sends a signal (skip signal).

Notes:

Also see G31.

Use the assigned M-code (M52) to turn the table probe on.

Use the assigned M-code (M62) to turn the table probe off.

Also see M75, M78, and M79.

Do not use Cutter Compensation with a G35.

Turn on the spindle in reverse (M04), for a right handed cutter.

O1234 (G35)

M52

T1 M06

G00 G90 G54 X0 Y1.

G43 H01 Z0

G01 Z-1. F10.

M04 S200

G31 Y0.49 F5. M75

G01 Y1. F20.

Z0

Y-1.

Z-1.

G35 Y-0.49 D1 F5.

G01 Y-1. F20.

M62

G00 G53 Z0 M05

M30

G36 Automatic Work Offset Measurement (Group 00)

(This G-code is optional and requires a probe)

G & M Codes
Settings

F	Feedrate in inches (mm) per minute
I	Optional offset distance along X-axis
J	Optional offset distance along Y-axis
K	Optional offset distance along Z-axis
X	Optional X-axis motion command
Y	Optional Y-axis motion command
Z	Optional Z-axis motion command

Automatic Work Offset Measurement (G36) is used to command a probe to set work fixture offsets. A G36 will feed the axes of the machine in an effort to probe the workpiece with a spindle mounted probe. The axis (axes) will move until a signal from the probe is received, or the travel limit is reached.

Tool offsets (G41, G42, G43, or G44) must not be active this function is



performed. The currently active work coordinate system is set for each axis programmed. The point where the skip signal is received becomes the zero position.

If an **I**, **J**, or **K** is specified, the appropriate axis work offset is shifted by the amount in the **I**, **J**, or **K** command. This allows the work offset to be shifted away from where the probe actually contacts the part.

Notes:

The points probed are offset by the values in Settings 59 through 62.

Use G91 incremental moves when using a G36.

Use the assigned M-codes (for example M53 and M63) with a dwell to turn the spindle probe on or off.

Example:

```
M53  
G04 P100  
M63
```

Program Example

```
O1234 (G36)  
M53  
G04 P100  
M63  
G00 G90 G54 X1. Y0  
Z-18.  
G91 G01 Z-1. F20.  
G36 X-1. F10.  
G90 G01 X1.  
M53  
G04 P100  
M63  
G00 G90 G53 Z0  
M30
```

G37 Automatic Tool Offset Measurement (Group 00)

(This G-code is optional and requires a probe)

F	Feedrate in inches (mm) per minute
H	Tool offset number
Z	Required Z-axis offset

Automatic Tool Length Offset Measurement (G37) is used to command a probe to set tool length offsets. A G37 will feed the Z-axis in an effort to probe a tool with a table mounted probe. The Z-axis will move until a signal from the probe is received, or the travel limit is reached. A non-zero H code and either G43 or G44 must be active. When the signal from the probe is received (skip signal) the Z position is used to set the specified tool offset (Hnnn). The resulting tool offset is the offset between the work zero point and the point where the probe is touched.



The coordinate system (G54-G59, G110-G129) and the tool length offsets (H01-H200) may be selected in this block or the previous block.

Notes:

Use the assigned M-code (M52) to turn the table probe on.

Use the assigned M-code (M62) to turn the table probe off.

Cutter Compensation may not be active during a skip function.

Also see M78 and M79.

Specify Z0 for no offset.

O1234 (G37)

T1 M06

M52

G00 G90 G110 X0 Y0

G00 G43 H1 Z5.

G37 H1 Z0. F30.

G00 G53 Z0

M62

M30

G40 Cutter Comp Cancel (Group 07)

G40 will cancel G41 or G42 cutter compensation.

G41 2D Cutter Compensation Left / G42 2D Cutter Comp. Right (Group 07)

G41 will select cutter compensation left; that is, the tool is moved to the left of the programmed path to compensate for the size of the tool. A D address must be programmed to select the correct tool radius or diameter offset. If the value in the selected offset is negative, cutter compensation will operate as though G42 (Cutter Comp Right.) was specified.

The right or left side of the programmed path is determined by looking at the tool as it moves away. If the tool needs to be on the left of the programmed path as it moves away, use G41. If it needs to be on the right of the programmed path as it moves away, use G42. Refer to the "Cutter Compensation" section for more information.

G43 Tool Length Compensation + (Add) / G44 Tool Length Comp - (Subtract) (Group 08)

A G43 code selects tool length compensation in the positive direction; the tool length in the offsets page is added to the commanded axis position. A G44 code selects tool length compensation in the negative direction; the tool length in the offsets page is subtracted from the commanded axis position. A non-zero H address must be entered to select the correct entry from the offsets page.



G47 Text Engraving (Group 00)

During a G47 command the control switches to G91 (Incremental mode) while engraving and then switches back to G90 (Absolute mode), when finished. To have the control stay in incremental mode, Setting 29 (G91 Non-Modal) and Setting 73 (G68 Incremental Angle) must be off.

E	Plunge feed rate (units/min)
F	Engraving feedrate (units/min)
I	Angle of rotation (-360. to +360.); default is 0
J	Height of text in inches (minimum = 0.001 inch); default is 1.0 inch
P	0 for literal string engraving 1 for sequential serial number engraving 32-126 for ASCII characters
R	Return plane
X	X start of engraving
Y	Y start of engraving
Z	Depth of cut

Sequential Serial Number Engraving

This method is used to engrave numbers on a series of parts, with the number being increased by one each time. The # symbol is used to select the number of digits in the serial number. For example, G47 P1 (###), will limit the serial number to four digits (P1 (##) will limit the serial number to two digits etc.).

The initial serial number can either be programmed or set manually. For example, if it is programmed, G47 P1 (1234), will set the initial serial number to "1234."

The initial serial number can also be set manually into a macro variable. The Macros option does not have to be enabled to do this. Macro variable #599 is used to hold the initial serial number to be engraved. For example, when macro variable #599 is set to "1234," G47 P1 (###), will produce, 1234. See the Macros section for more information.

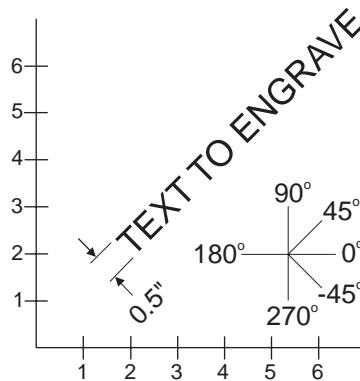
Literal String Engraving

This method is used to engrave desired text on a part. The text should be in the form of a comment on the same line as the P0 statement. For example, G47 P0 (ENGRAVE THIS), will produce, ENGRAVE THIS

Example

This example will create the figure shown.

```
G47 P0 X2.0 Y2.0 I45. J5 R.05 Z-.005 F15.0 E10.0 (TEXT TO  
ENGRAVE)
```



In this example:

G47 P0 select literal string engraving
 X2.0 Y2.0 select 2.0, 2.0 as the starting point for the text.
 I45. places the text at a positive 45° angle
 J.5 sets the text height to 0.5 inch
 R.05 commands the cutter to retract to 0.05 inch above the cutting plane after engraving
 Z-.005 selects a 0.005 inch (mm) deep cut
 F15.0 selects an engraving feedrate of 15 units/min
 E10.0 commands the cutter to plunge at a rate of 10 units/min
 The milling strokes defining each character, i.e., the font, are compiled G-code in the HAAS control. The font characters may be redefined by supplying a different G-code program, giving it the name O09876. This program must conform to the format expected by the HAAS control.

Note: Avoid using program number O09876 for programs other than font defining. Overwriting O09876 with a regular milling program will prevent G47 from functioning properly.

For guidance, some of the code from the built-in font program is shown below. The example below may be used as a template. Code should be written exactly as shown.

P values to engrave specific characters:

32	blank	41)	59	;	93]
33	!	42	*	60	<	94	^
34	"	43	+	61	=	95	-
35	#	44	,	62	>	96	_
36	\$	45	-	63	?	97-122	a-z
37	%	46	.	64	@	123	{
38	&	47	/	65-90	A-Z	124	
39	'	48-57	0-9	91	[125	}
40	(58	:	92	\	126	~



Example

To engrave "\$2.00" two lines of code are necessary. The first uses a P36 to engrave the dollar sign (\$), and the second uses P0 (2.00). Note that the axes will need to be shifted between the first and second line of code in order to make a space between the dollar sign and the 2.

Example O9876 G-Code Program

	Comments
%	% marks the start of the program.
O9876 (Engraving)	The program number must be 9876.
#700= #4003	Save G90/G91
#701= #4001	Save G00/G01 etc.
G00 X#24 Y#25	
Z#18	If R, move there with users G90/G91
#702= #5003 - #26	
IF [#9 EQ #0] #9= #4109	Use present F if none specified
IF [#8 EQ #0] #8= #9	If no E, use F
G91	All incremental from here on
IF [#4 EQ #0] #4= 0.0	
IF [#5 EQ #0] #5= 1.0	
G68 R#4	
G51 P [#5 * 1000]	
N1000	
M97	M97 auto M99 at end of string
GOTO1000	
N125	
M99	
(SPACE)	This section mills a space.
N126	
G00 X0.864 F#8	
M99	
N127	
G#700	Restore G90/G91
G#701	Restore G00/G01 etc.
M99	
N1	This section mills an exclamation point
(!)	
G00 X0.2692	
G01 Z - #702 F#8	
G03 J0.0297 F#9	
G00 Z#702	
G00 Y0.2079	
G01 Z - #702 F#8	
G01 X0.0495 Y0.6732 F#9	
G03 X-0.099 R0.0495	
G01 X0.0495 Y-0.6732	
G00 Z#702	
G00 X0.2692 Y-0.2079	
M99	
N2	This section mills double quotes.
(«)	



G00 X0.2345 Y0.792
 G01 Z - #702 F#8
 G01 X0.0148 Y0.198 F#9
 G01 X-0.0297
 G01 X0.0148 Y-0.198
 G00 Z#702
 G00 X0.1485
 G01 Z - #702 F#8
 G01 X0.0148 Y0.198 F#9
 G01 X-0.0297
 G01 X0.0148 Y-0.198
 G00 Z#702
 G00 X0.2346 Y-0.792
 M99
 N3
 (#)

This section mills a # sign.

G00 X0.4082 Y0.1666
 G01 Z - #702 F#8
 G01 X0.0433 Y0.8086 F#9
 G00 Z#702
 G00 X0.2627 Y0.0148
 G01 Z - #702 F#8
 G01 X-0.0433 Y-0.8234 F#9
 G00 Z#702
 G00 X0.2194 Y0.2374
 G01 Z - #702 F#8
 G01 X-0.6676 F#9
 G00 Z#702
 G00 X0.2167 Y-0.723
 M99
 ...

% marks the end of the program.

For the creation of each character, there is a different label to start the code.
Each section terminates with an M99.

Label	N126	N1	N2	N3	N4	N5	N6	N7	N8	N9
Character	space	!	"	#	\$	%	&	'	()
Label	N10	N11	N12	N13	N14	N15	N16	N17	N18	N19
Character	*	+	,	-	.	/	0	1	2	3
Label	N20	N21	N22	N23	N24	N25	N26	N27	N28	N29
Character	4	5	6	7	8	9	:	;	,	=
Label	N30	N31	N32	N33	N34	N35	N36	N37	N38	N39
Character	>	?	@	A	B	C	D	E	F	G
Label	N40	N41	N42	N43	N44	N45	N46	N47	N48	N49
Character	H	I	J	K	L	M	N	O	P	Q
Label	N50	N51	N52	N53	N54	N55	N56	N57	N58	N59
Character	R	S	T	U	V	W	X	Y	Z	[
Label	N60	N61	N62	N63	N64	N65	N66	N67	N68	N69



Character \]	^	_	'	a	b	c	d	e
Label N70	N71	N72	N73	N74	N75	N76	N77	N78	N79
Character f	g	h	i	j	k	l	m	n	o
Label N80	N81	N82	N83	N84	N85	N86	N87	N88	N89
Character p	q	r	s	t	u	v	w	x	y
Label N90	N91	N92	N93						
Character z	{		}						

G49 G43/G44/G143 Cancel (Group 08)

This G code cancels tool length compensation. Note: An H0, G28, M30, and Reset will also cancel tool length compensation.

G50 Cancel Scaling (Group 11)

G50 cancels the optional scaling feature. Any axis scaled by a previous G51 command is no longer in effect.

G51 Scaling (Group 11)

(This G-code is optional and requires Rotation and Scaling)

X optional center of scaling for the X axis
Y optional center of scaling for the Y axis
Z optional center of scaling for the Z axis
P optional scaling factor for all axes; three-place decimal from 0.001 to 8383.000.

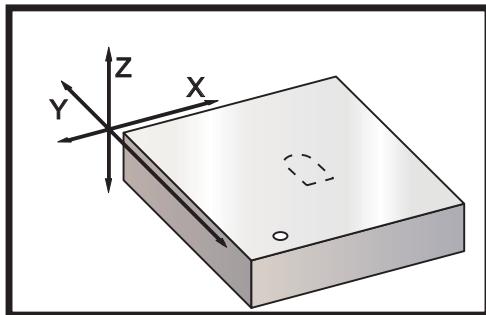
G51 [X...] [Y...] [Z...] [P...]

A scaling center is always used by the control in determining the scaled position. If any scaling center is not specified in the G51 command block, then the last commanded position is used as the scaling center.

When scaling (G51) is commanded, all X, Y, Z, I, J, K, or R values addressing machine motion are multiplied by a scaling factor and are offset relative to a scaling center.

G51 will affect all appropriate positioning values in the blocks following the G51 command. The X, Y and Z axes can be scaled using a P address, if a P address is not entered the Setting 71 scaling factor is used.

The following programs illustrate how scaling is performed when different scaling centers are used.

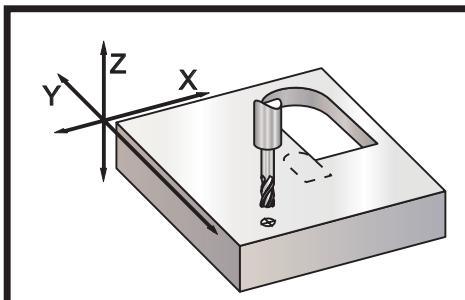


0001 (GOTHIC WINDOW) ;
F20, S500 ;
G00 X1. Y1. ;
G01 X2. ;
Y2. ;
G03 X1. R0.5;
G01 Y1. ;
G00 X0 Y0 ;
M99 ;

○ = Work coordinate origin
No Scaling

G51

The first example illustrates how the control uses the current work coordinate location as a scaling center. Here, it is X0 Y0 Z0.

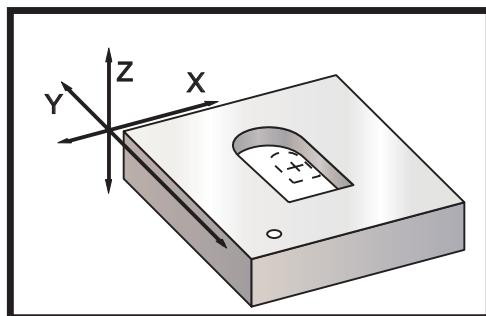


00010 ;
G59 ;
G00 G90 X0 Y0 Z0 ;
G51 P2. (scaling center is X0 Y0 Z0) ;
M98 P1 ;
M30 ;

○ = Work coordinate origin
+ = Center of scaling

G51 Scaling

The next example specifies the center of the window as the scaling center.

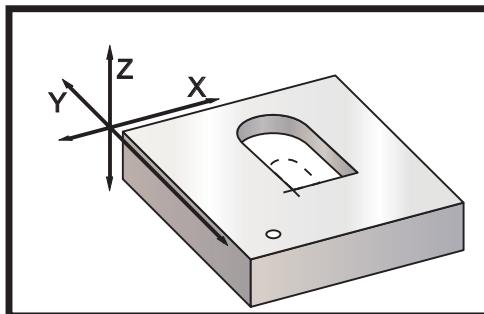


00011 ;
G59 ;
G00 G90 X0 Y0 Z0 ;
M98 P1 ;
G51 X1.5 Y1.5 P2. ;
M98 P1 ;
M30 ;

○ = Work coordinate origin
+ = Center of scaling

G51 Scaling

The last example illustrates how scaling can be placed at the edge of tool paths as if the part was being set against locating pins.



```
00011 ;  
G59 ;  
G00 G90 X0 Y0 Z0 ;  
M98 P1 ;  
G51 X1.0 Y1.0 P2 ;  
M98 P1 ;  
M30 ;
```

○ = Work coordinate origin
+ = Center of scaling

G51 Scaling

Programming notes:

Tool offsets and cutter compensation values are not affected by scaling.

Scaling does not affect canned cycle Z-axis movements such as clearance planes and incremental values.

The final results of scaling are rounded to the lowest fractional value of the variable being scaled.

G52 Set Work Coordinate System (Group 00 or 12)

The G52 command works differently depending on the value of setting 33. Setting 33 selects the Fanuc, Haas, or Yasnac style of coordinates.

If Yasnac is selected, G52 is a group 12 G-code. G52 works the same as G54, G55, etc. All of the G52 values will not be set to zero (0) when powered on, reset is pressed, at the end of the program, or by an M30. When using a G92 (Set Work Coordinate Systems Shift Value), in Yasnac format, the X, Y, Z, A, and B values are subtracted from the current work position, and automatically entered into the G52 work offset.

If Fanuc is selected, G52 is a group 00 G-code. This is a global work coordinate shift. The values entered into the G52 line of the work offset page are added to all work offsets. All of the G52 values in the work offset page will be set to zero (0) when powered on, reset is pressed, changing modes, at the end of the program, by an M30, G92 or a G52 X0 Y0 Z0 A0 B0. When using a G92 (Set Work Coordinate Systems Shift Value), in Fanuc format, the current position in the current work coordinate system is shifted by the values of G92 (X, Y, Z, A, and B). The values of the G92 work offset are the difference between the current work offset and the shifted amount commanded by G92.

If Haas is selected, G52 is a group 00 G-code. This is a global work coordinate shift. The values entered into the G52 line of the work offset page are added to all work offsets. All of the G52 values will be set to zero (0) by a G92. When using a G92 (Set Work Coordinate Systems Shift Value), in Haas format, the current position in the current work coordinate system is shifted by the values



of G92 (X, Y, Z, A, and B). The values of the G92 work offset are the difference between the current work offset and the shifted amount commanded by G92 (Set Work Coordinate Systems Shift Value).

G53 Non-Modal Machine Coordinate Selection (Group 00)

This code temporarily cancels work coordinate offsets and uses the machine coordinate system. In the machine coordinate system, the zero point for each axis is the position where the machine goes when a Zero Return is performed. G53 will revert to this system for the block it is commanded in.

G54-59 Select Work Coordinate System #1 - #6 (Group 12)

These codes select one of the six user coordinate systems. All future references to axes positions will be interpreted using the new (G54 G59) coordinate system.

G60 Uni-Directional Positioning (Group 00)

This G code is used to provide positioning only from the positive direction. It is provided only for compatibility with older systems. It is non-modal, so does not affect the blocks that follow it. Also see Setting 35.

G61 Exact Stop Mode (Group 15)

The G61 code is used to specify an exact stop. It is modal; therefore, it affects the blocks that follow it. The machine axes will come to an exact stop at the end of each commanded move.

G64 G61 Cancel (Group 15)

The G64 code is used to cancel exact stop (G61).

G68 Rotation (Group 16)

(This G-code is optional and requires Rotation and Scaling.)

G17, G18, G19 optional plane of rotation, default is current

- A optional center of rotation for the first axis of the selected plane
- B optional center of rotation for the second axis of the selected plane
- R optional angle of rotation specified in degrees
Three-place decimal -360.000 to 360.000.

A G17, 18 or 19 must be used before G68 to establish the axis plane being rotated. For example: G17 G68 Annn Bnnn Rnnn;

A and B correspond to the axes of the current plane; for the G17 example A is the X-axis and B is the Y-axis.

A center of rotation is always used by the control to determine the positional values passed to the control after rotation. If any axis center of rotation is not specified, the current location is used as the center of rotation.

When rotation (G68) is commanded, all X, Y, Z, I, J, and K values are rotated through a specified angle R using a center of rotation.



G68 will affect all appropriate positional values in the blocks following the G68 command. Values in the line containing G68 are not rotated. Only the values in the plane of rotation are rotated, therefore, if G17 is the current plane of rotation, only X and Y values are affected.

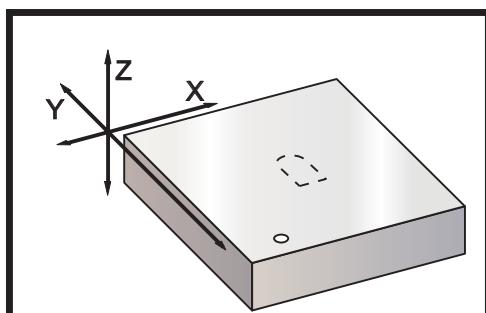
Entering a positive number (angle) for the R address will rotate the feature counterclockwise.

If the angle of rotation (R) is not entered, then the angle of rotation is taken from Setting 72.

In G91 mode (incremental) with Setting 73 ON, the rotation angle is changed by the value in R. In other words, each G68 command will change the rotation angle by the value specified in R.

The rotational angle is set to zero at the beginning of the program, or it can be set to a specific angle using a G68 in G90 mode.

The following examples illustrate rotation using G68.

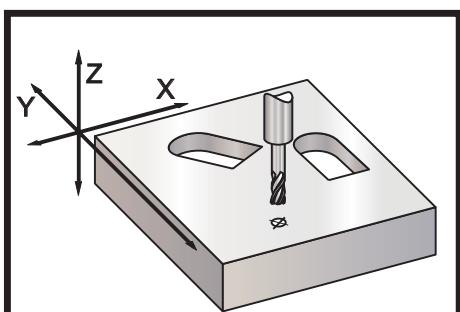


```
0001 (GOTHIC WINDOW) ;  
F20, S500 ;  
G00 X1. Y1. ;  
G01 X2. ;  
Y2. ;  
G03 X1. R0.5  
G01 Y1. ;  
M99 ;
```

○ = Work coordinate origin
No Rotation

G68

The first example illustrates how the control uses the current work coordinate location as a rotation center ($X0\ Y0\ Z0$).

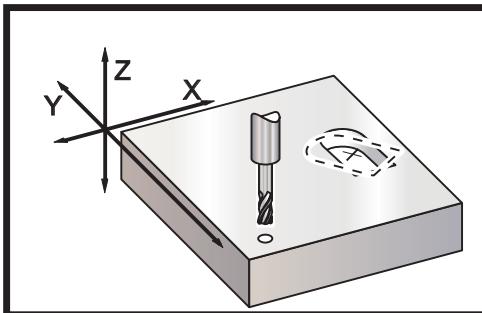


```
00002 ;  
G59 ;  
G00 G90 X0 Y0 Z0 ;  
M98 P1 ;  
G90 G00 X0 Y0 ; (Last Commanded Position)  
G68 R60. ;  
M98 P1 ;  
G69 G90 G00 X0 Y0 ;  
M30 ;
```

○ = Work coordinate origin
+ = Center of rotation

G68 Rotation

The next example specifies the center of the window as the rotation center.

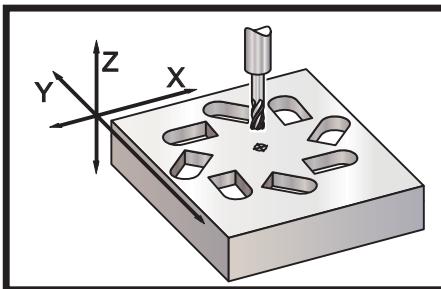


```
00003 ;
G59 ;
G00 G90 X0 Y0 Z0 ;
M98 P1 ;
G00 G90 X0 Y0 Z0 ;
G68 X1.5 Y1.5 R60. ;
M98 P1 ;
G69 G90 G00 X0 Y0 ;
M30 ;
```

= Work coordinate origin
 = Center of rotation

G68 Rotation

This example shows how the G91 mode can be used to rotate patterns about a center. This is often useful for making parts that are symmetric about a given point.



```
00004 ;
G59 ;
G00 G90 X0 Y0 Z0 ;
M98 P10 L8 (SUBROUTINE 00010) ;
M30 ;
```

```
00010 ;
G91 G68 R45. ;
G90 M98 P1 ;
G90 G00 X0 Y0 ;
M99 ;
```

= Work coordinate origin
 = Center of rotation

G68 Rotation

Do not change the plane of rotation while G68 is in effect.

Rotation with Scaling

If scaling and rotation are used simultaneously, it is recommended that scaling be turned on prior to rotation, and that separate blocks be used. Use the following template when doing this.

```
G51 ..... (SCALING) ;
...
G68 ..... (ROTATION) ;
.
.
.
G69 ..... (ROTATION OFF) ;
...
G50 ..... (SCALING OFF) ;
```

Rotation with Cutter Compensation

Cutter compensation should be turned on after the rotation command is issued. Compensation should also be turned off prior to turning rotation off.



G69 Cancel G68 Rotation (Group 16)

(This G-code is optional and requires Rotation and Scaling.)

G69 cancels any rotation specified previously.

G70 Bolt Hole Circle (Group 00)

- I Radius (+CCW / -CW)
- J Starting angle (0 to 360.0 degrees CCW from horizontal; or 3 o'clock position)
- L Number of holes evenly spaced around the circle

This non-modal G code must be used with one of the canned cycles G73, G74, G76, G77, or G81-G89. A canned cycle must be active so that at each position, a drill or tap function is performed.

G71 Bolt Hole Arc (Group 00)

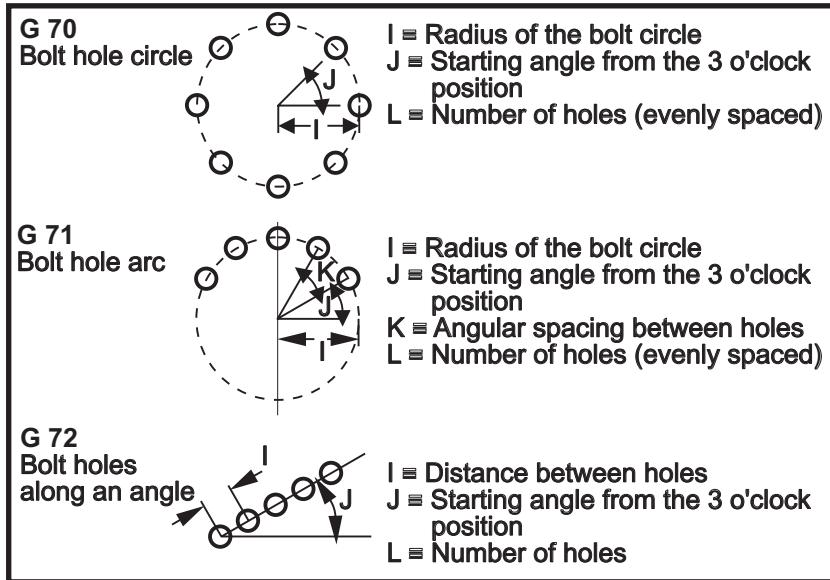
- I Radius (+CCW / -CW)
- J Starting angle (degrees CCW from horizontal)
- K Angular spacing of holes (+ or -)
- L Number of holes

This non-modal G code is similar to G70 except that it is not limited to a complete circle. G71 belongs to Group 00 and thus is non-modal. A canned cycle must be active so that at each position, a drill or tap function is performed.

G72 Bolt Holes Along an Angle (Group 00)

- I Distance between holes (+CCW / -CW)
- J Angle of line (degrees CCW from horizontal)
- L Number of holes

This non-modal G code drills "L" number of holes in a straight line at the specified angle. It operates similarly to G70. For a G72 to work correctly, a canned cycle must be active so that at each position, a drill or tap function is performed.



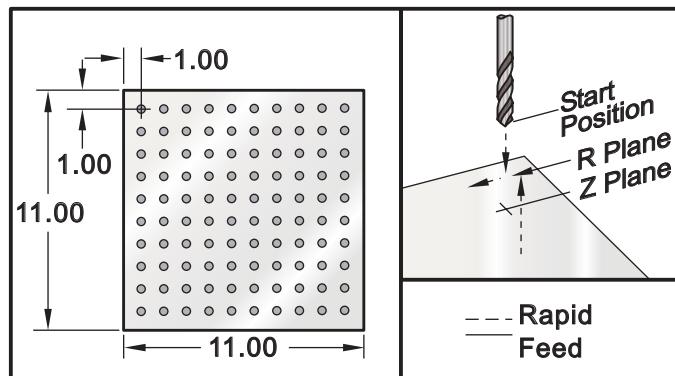
Rules For Bolt Pattern Canned Cycles:

1. The tool must be placed at the center of the bolt pattern before the canned cycle execution. The center is usually X0, Y0.
2. The J code is the angular starting position and is always 0 to 360 degrees counterclockwise from the three o'clock position.

Looping Canned Cycles

The following is an example of a program using a drilling canned cycle that is incrementally looped.

Note: The sequence of drilling used here is designed to save time and to follow the shortest path from hole to hole.



G81 Drilling Canned Cycle (Incremental) and Grid Plate for Multiple-Fixture Subprogram

Program Example

```
%  
O03400  
T1 M06  
G00 G90 G54 X1.0 Y-1.0 S2500 M03  
G43 H01 Z.1 M08  
G81 Z-1.5 F15. R.1  
G91 X1.0 L9  
G90 X-2.0  
G91 X-1.0 L9  
G90 Y-3.0  
G91 X1.0 L9  
G90 Y-4.0  
G91 X-1.0 L9  
G90 Y-5.0  
G91 X1.0 L9  
G90 Y-6.0  
G91 X-1.0 L9  
G90 Y-7.0  
G91 X1.0 L9 |||  
G90 Y-8.0  
G91 X-1.0 L9  
G90 Y-9.0  
G91 X1.0 L9  
G90 Y-10.0  
G91 X-1.0 L9  
G00 G90 G80 Z1.0 M09  
G28 G91 Y0Z0  
M30  
%
```

Description

(Drilling grid plate)

(Or stay in G91 and repeat Y-1.0)



Modifying Canned Cycles

In this section we will cover canned cycles that have to be customized in order to make the programming of difficult parts easier.

Using G98 and G99 to clear clamps – For example, A square part being held to the table with one-inch tall table clamps. A program needs to be written to clear the table clamps.

Program Example	Description
%	
O4500	
T1 M06	
G00 G90 G54 X1.0 Y-1.0 S3500 M03	
G43 H01 Z1.125 M08	
G81 G99 Z-1.500 R.05 F20.	
X2.0 G98	(Will return to starting point after executing cycle)
X6.0 G99	(Will return to reference plane after executing cycle)
X8.0	
X10.0	
X12.0 G98	
X16.0 G99	
X18.0 G98	
G00 G80 Z2.0 M09	
G28 G91 Y0 Z0	
M30	
%	



X, Y Plane Obstacle Avoidance In A Canned Cycle:

To avoid an obstacle in the X, Y plane during a canned cycle place an L0 in a canned cycle line to make an X, Y move without executing the Z-axis canned operation.

For example, having a six-inch square aluminum block, with a one-inch by one-inch deep flange on each side, the print calls for two holes centered on each side of the flange. The program to avoid each of the corners on the block.

Program Example	Description
%	
O4600	(X0,Y0 is at the top left corner, Z0 is at the top of the part)
T1 M06	
G00 G90 G54 X2.0 Y-.5 S3500 M03	
G43 H01 Z-.9 M08	
G81 Z-2.0 R-.9 F15.	
X4.0	
X5.5 L0	(angular corner avoidance)
Y-2.0	
Y-4.0	
Y-5.5 L0	
X4.0	
X2.0	
X.5 L0	
Y-4.0	
Y-2.0	
G00 G80 Z1.0 M09	
G28 G91 Y0 Z0	
M30	
%	



G-CODE CANNED CYCLES

Introduction

Canned cycles are used to simplify programming. They are used for repetitive operations, such as drilling, tapping, and boring. The canned cycle is executed every time an X and/or Y-axis motion is programmed.

Using Canned Cycles

The positioning of a canned cycle in the X and/or Y-axes can be done in either absolute (G90) or incremental (G91). Incremental (G91) motion in a canned cycle is often useful with a loop count (Lnn) which will repeat the canned cycle operation that many times with each incremental X or Y move for the canned cycle.

Example:

G81 G99 Z-0.5 R0.1 F6.5 (This will drill one hole at the present location)

G91 X-0.5625 L9 (This will drill 9 more holes .5625 equally spaced in the minus direction)

If a canned cycle is defined without an X or Y and a loop count of 0 (L0), the cycle will not be performed initially. The operation of the canned cycle will vary according to whether incremental (G91) or absolute (G90) positioning is active. Incremental motion in a canned cycle is often useful as a loop (L) count as it can be used to repeat the operation with an incremental X or Y move between each cycle.

Example:

X1.25 Y-0.75 (center location of bolt hole pattern)

G81 G99 Z-0.5 R0.1 F6.5 L0 (L0 on the G81 line will not drill a hole in the bolt hole circle)

G70 I0.75 J10. L6 (6-hole bolt hole circle)

Once a canned cycle is commanded, that operation is done at every X-Y position listed in a block. Some of the canned cycle numerical values can be changed after the canned cycle is defined. The most important of these are the R plane value and the Z depth value. If these are listed in a block with XY commands, the XY move is done and all subsequent canned cycles are performed with the new R or Z value.

The positioning of the X and Y-axes prior to a canned cycle is done with rapid moves.

G98 and G99 change the way the canned cycles operate. When G98 is active, the Z-axis will return to the initial start plane at the completion of each hole in the canned cycle. This allows for positioning up and around areas of the part and/or clamps and fixtures.



When G99 is active, the Z-axis returns to the R (rapid) plane after each hole in the canned cycle for clearance to the next XY location. Changes to the G98/G99 selection can also be made after the canned cycle is commanded, which will affect all later canned cycles.

A P address is an optional command for some of the canned cycles. This is a programmed pause at the bottom hole to help break chips, provide a smoother finish and relieve any tool pressure to hold closer tolerance. Note that if a value for P is entered for one canned cycle it will be used in others unless cancelled (G00, G01, G80 or the Reset button).

An S (spindle speed) command must be defined in, or before the G-code line of code.

Tapping in a canned cycle needs a feed rate calculated. The Feed Formula is:

Spindle speed divided by Threads per inch of the tap = Feedrate in inches per minute

Canned Cycles also benefit from the use of Setting 57. Turning this setting ON will perform an exact stop between rapids. This is useful to avoid nicking the part at the bottom of the hole.

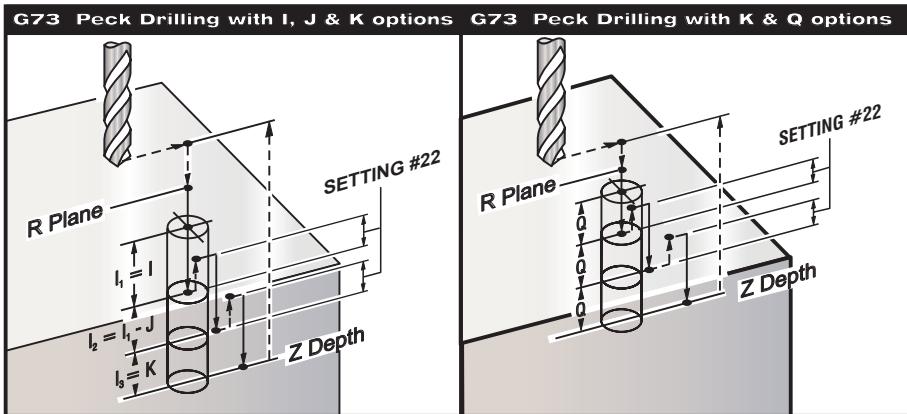
Note: The Z, R, and F addresses are required data for all canned cycles.

Canceling a Canned Cycle

The G80 code is used to cancel all canned cycles; note that a G00 or G01 code will also cancel a canned cycle. Once selected, a canned cycle is active until canceled with G80, G00 or G01.

G73 High-Speed Peck Drilling Canned Cycle (Group 09)

- F Feedrate in inches (mm) per minute
- I First cut depth
- J Amount to reduce cutting depth for pass
- K Minimum depth of cut (The control will calculate the number of pecks)
- L Number of repeats (Number of holes to drill) if G91 (Incremental Mode) is used
- P Pause at the bottom of the hole (in seconds)
- Q Cut Depth (always incremental)
- R Position of the R plane (Distance above part surface)
- X X-axis location of hole
- Y Y-axis location of hole
- Z Position of the Z-axis at the bottom of hole

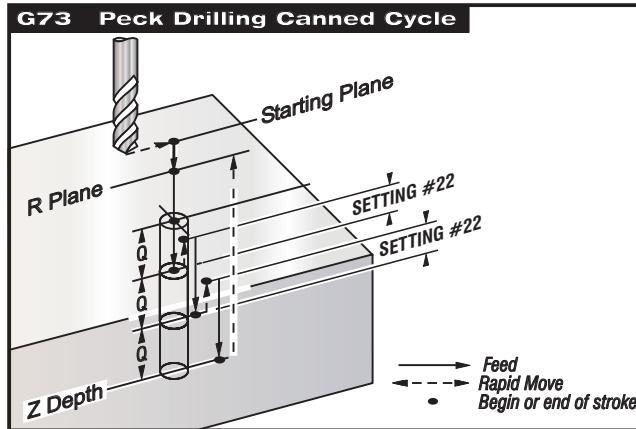


I, J, K, and Q are always positive numbers.

There are two methods to program a G73; first using the **I, J, K** addresses and the second using the **K and Q** addresses.

If **I, J, and K** are specified, The first pass will cut in by the value **I**, each succeeding cut will be reduced by the value of **J**, and the minimum cutting depth is **K**. If **P** is specified, the tool will pause at the bottom of the hole for that amount of time.

If **K** and **Q** are both specified, a different operating mode is selected for this canned cycle. In this mode, the tool is returned to the **R plane** after the number of passes totals up to the **K** amount.

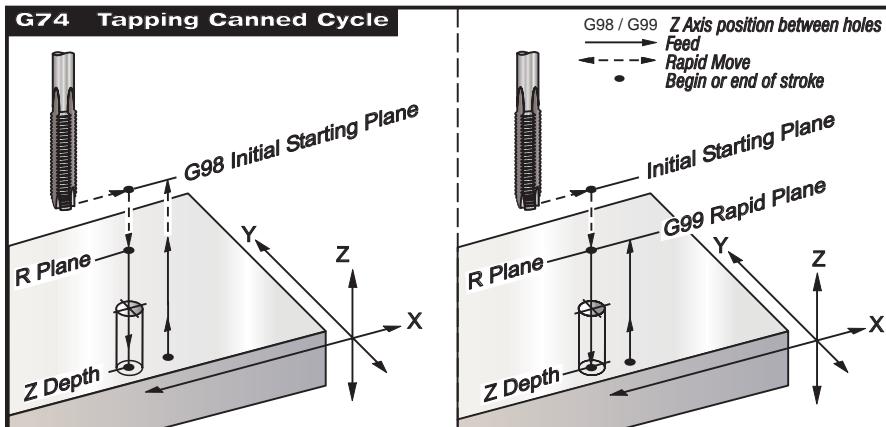


G & M Codes
Settings



G74 Reverse Tap Canned Cycle (Group 09)

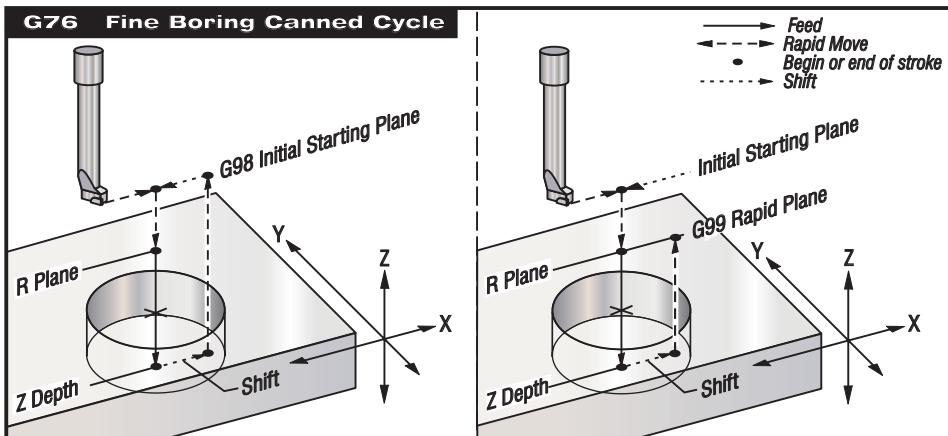
- F Feedrate in inches (or mm) per minute (use the formula, described in the canned cycle introduction, to calculate feed rate and spindle speed)
- J Retract Multiple (How fast to retract - see Setting 130)
- L Number of repeats (How many holes to tap) if G91 (Incremental Mode) is used
- R Position of the R plane (position above the part) where tapping starts
- X X-axis location of hole
- Y Y-axis location of hole
- Z Position of the Z-axis at the bottom of hole





G76 Fine Boring Canned Cycle (Group 09)

- F Feedrate in inches (or mm) per minute
I Shift value along the X-axis before retracting, if Q is not specified
J Shift value along the Y-axis before retracting, if Q is not specified
L Number of holes to bore if G91 (Incremental Mode) is used
P The dwell time at the bottom of the hole
Q The shift value, always incremental
R Position of the R plane (position above the part)
X X-axis location of hole
Y Y-axis location of hole
Z Position of the Z-axis at the bottom of hole



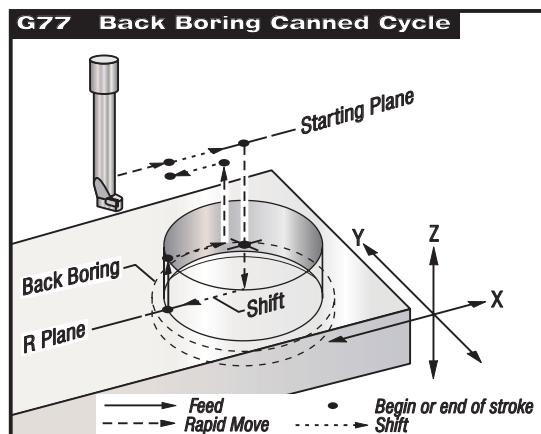
In addition to boring the hole, this cycle will shift the X and/or Y axis prior to retracting in order to clear the tool while exiting the part. If Q is used Setting 27 determines the shift direction. If Q is not specified, the optional I and J values are used to determine the shift direction and distance.



G77 Back Bore Canned Cycle (Group 09)

F Feedrate in inches (or mm) per minute
I Shift value along the X-axis before retracting, if Q is not specified
J Shift value along the Y-axis before retracting, if Q is not specified
L Number of holes to bore if G91 (Incremental Mode) is used
Q The shift value, always incremental
R Position of the R plane (position above the part)
X X-axis location of hole
Y Y-axis location of hole
Z Position of the Z-axis at the bottom of hole

In addition to boring the hole, this cycle shifts the X and/or Y axis prior to and after cutting in order to clear the tool while entering and exiting the part (see G76 for an example of a shift move). Setting 27 determines the shift direction. If Q is not specified, optional I and J values are used to determine shift direction and distance.



G80 Canned Cycle Cancel (Group 09)

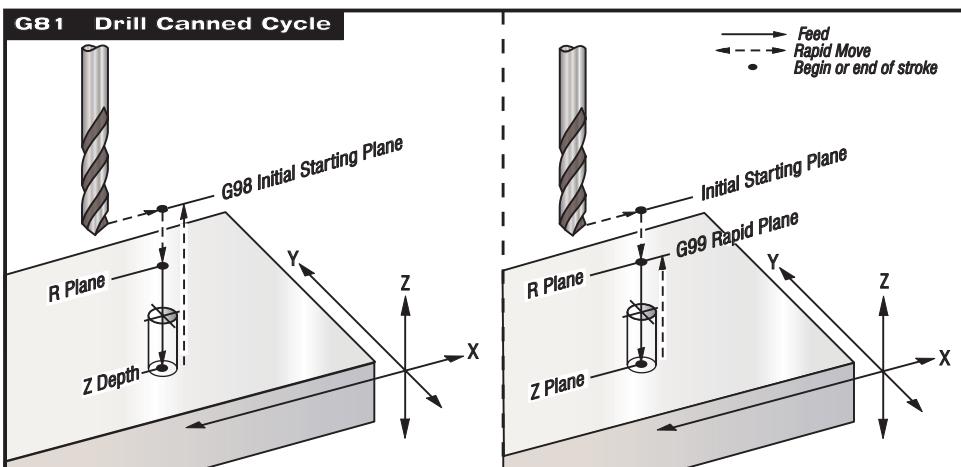
This G code deactivates all canned cycles until a new one is selected. Note that use of G00 or G01 will also cancel a canned cycle.



G81 Drill Canned Cycle (Group 09)

- F Feedrate in inches (or mm) per minute
- L Number of holes to drill if G91 (Incremental Mode) is used
- R Position of the R plane (position above the part)
- X X-axis motion command
- Y Y-axis motion command
- Z Position of the Z-axis at the bottom of hole

Note: The X and Y addresses, in most cases, are the location of the first hole to be drilled.



Program Example

The following is a program to drill through an aluminum plate:

```
T1 M06
G00 G90 G54 X1.125 Y-1.875 S4500 M03
G43 H01 Z0.1
G81 G99 Z-0.35 R0.1 F27.
X2.0
X3.0 Y-3.0
X4.0 Y-5.625
X5.250 Y-1.375
G80 G00 Z1.0
G28
M30
```

**G & M Codes
Settings**

G82 Spot Drill Canned Cycle (Group 09)

- F Feedrate in inches (or mm) per minute
- L Number of holes if G91 (Incremental Mode) is used
- P The dwell time at the bottom of the hole
- R Position of the R plane (position above the part)
- X X-axis location of hole
- Y Y-axis location of hole
- Z Position of bottom of hole



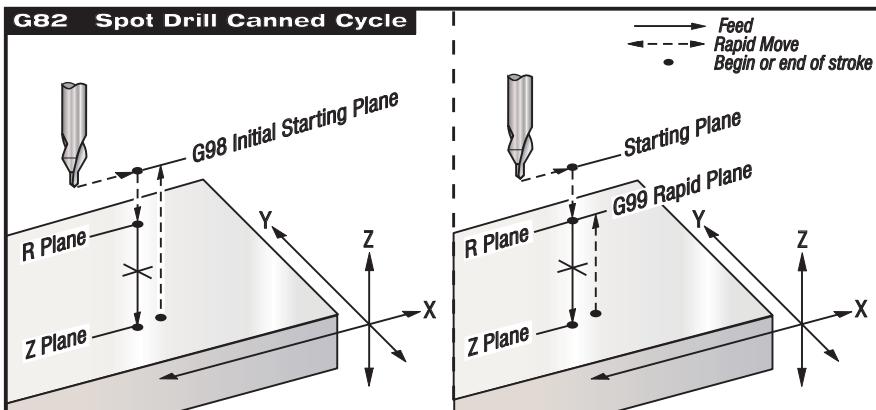
Programming Note: G82 is similar to G81 except that there is the option to program a dwell (P)

Program Example

```
%  
O1234  
T1 M06  
G90 G54 G00 X.565 Y-1.875 S1275 M03  
G43 H01 Z0.1 M08  
G82 Z-0.175 P.3 R0.1 F10.  
X1.115 Y-2.750  
X3.365 Y-2.875  
X4.188 Y-3.313  
X5.0 Y-4.0  
G80 G00 Z1.0 M09
```

Description

(Sample program)
(Tool #1 is a 0.5" x 90-degree spot drill)
(90-degree spot drill; the depth is)
(half of the chamfer diameter)



G82 Spot Drilling Example

G83 Normal Peck Drilling Canned Cycle (Group 09)

- F Feedrate in inches (or mm) per minute
- I Size of first cutting depth
- J Amount to reduce cutting depth each pass
- K Minimum depth of cut
- L Number of holes if G91 (Incremental Mode) is used
- P Pause at end of last peck, in seconds (Dwell)
- Q Cut depth, always incremental
- R Position of the R plane (position above the part)
- X X-axis location of hole
- Y Y-axis location of hole
- Z Position of the Z-axis at the bottom of hole

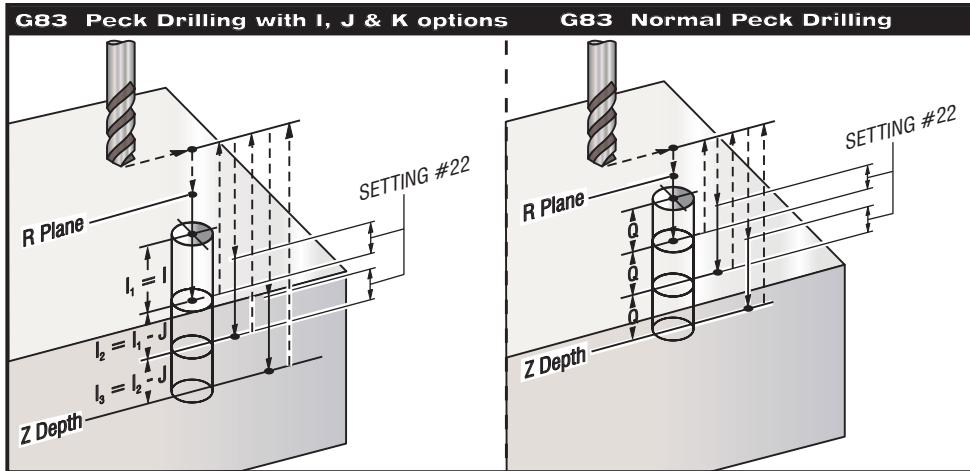
If I, J, and K are specified, the first pass will cut in by the amount of I, each succeeding cut will be reduced by amount J, and the minimum cutting depth is K. Do not use a Q value when programming with I,J,K.



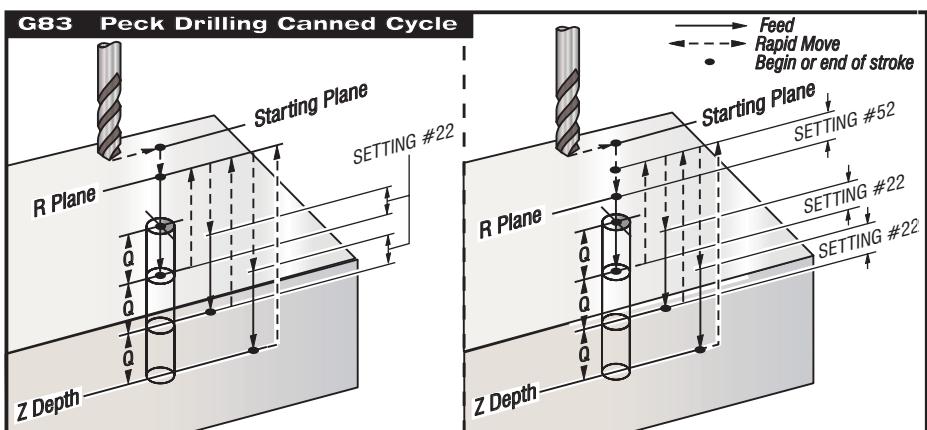
If P is specified, the tool will pause at the bottom of the hole for that amount of time. The following example will peck several times and dwell for 1.5 seconds:

G83 Z-0.62 F15. R0.1 Q0.175 P1.5

The same dwell time will apply to all subsequent blocks that do not specify a dwell time.



Setting 52 changes the way G83 works when it returns to the R plane. Usually the R plane is set well above the cut to ensure that the peck motion allows the chips to get out of the hole. This wastes time as the drill starts by drilling "empty" space. If Setting 52 is set to the distance required to clear chips, the R plane can be put much closer to the part being drilled. When the chip-clearing move to R occurs, the Z axis distance above R is determined by this setting.





Program Example

```
T2 M06  
G90 G54 G00 X0.565 Y-1.875 S2500 M03  
G43 H02 Z0.1 M08  
G83 Z-0.720 Q0.175 R0.1 F15.  
X1.115 Y-2.750  
X3.365 Y-2.875  
X4.188 Y-3.313  
X5.0 Y-4.0  
G80 G00 Z1.0 M09
```

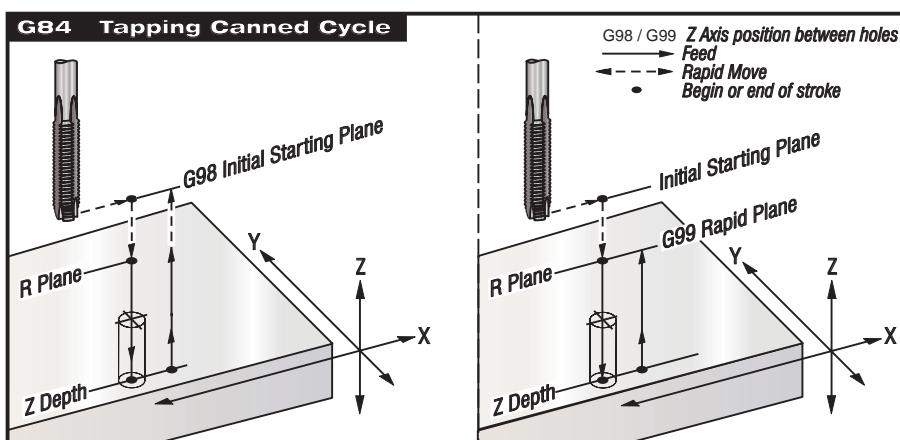
Description

(Tool #2 is a 0.3125" stub drill)

(Drill point is 1/3 of the drill diameter)

G84 Tapping Canned Cycle (Group 09)

- F Feedrate in inches (or mm) per minute
- J Retract Multiple (Example: J2 will retract twice as fast as the cutting speed, also see Setting 130)
- L Number of holes if G91 (Incremental Mode) is used
- R Position of the R plane (Position above the part)
- X X-axis location of hole
- Y Y-axis location of hole
- Z Position of the Z-axis at the bottom of hole



G84 Tapping Canned Cycle Example

Program

```
T3 M06  
G90 G54 G00 X0.565 Y-1.875 S900 M03  
G43 H03 Z0.2 M08  
G84 Z-0.600 R0.2 F56.25  
X1.115 Y-2.750  
X3.365 Y-2.875  
X4.188 Y-3.313  
X5.0 Y-4.0  
G80 G00 Z1.0 M09  
G28 G91 Y0 Z0  
M30  
%
```

Example

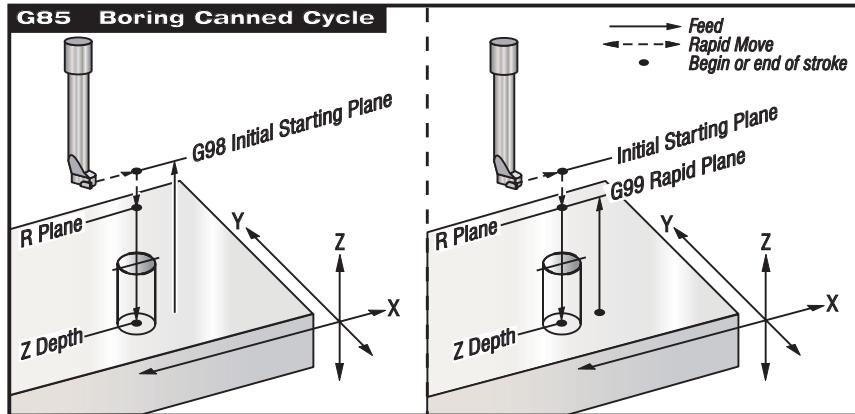
(Tool #3 is a 3/8-16 tap)

(900 rpm divided by 16 tpi = 56.25 ipm)



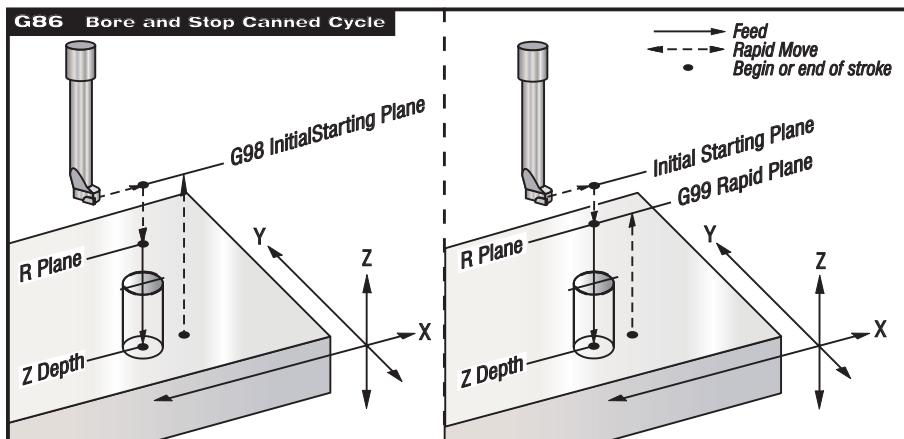
G85 Boring Canned Cycle (Group 09)

- F Feedrate in inches (or mm) per minute
L Number of holes if G91 (Incremental Mode) is used
R Position of the R plane (position above the part)
X X-axis location of holes
Y Y-axis location of holes
Z Position of the Z-axis at the bottom of hole



G86 Bore and Stop Canned Cycle (Group 09)

- F Feedrate in inches (or mm) per minute
L Number of holes if G91 (Incremental Mode) is used
R Position of the R plane (position above the part)
X X-axis location of hole
Y Y-axis location of hole
Z Position of the Z-axis at the bottom of hole

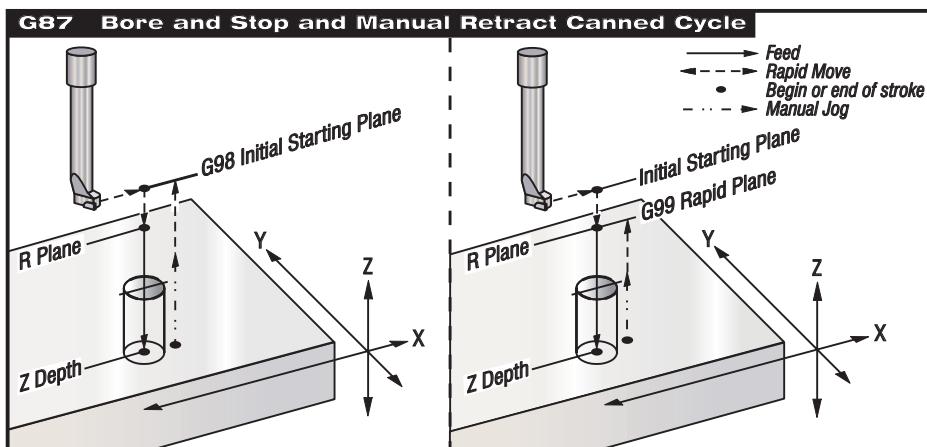




G87 Bore In and Manual Retract Canned Cycle (Group 09)

- F Feedrate in inches (or mm) per minute
- L Number of holes if G91 (Incremental Mode) is used
- R Position of the R plane (position above the part)
- X X-axis location of hole
- Y Y-axis location of hole
- Z Position of the Z-axis at the bottom of hole

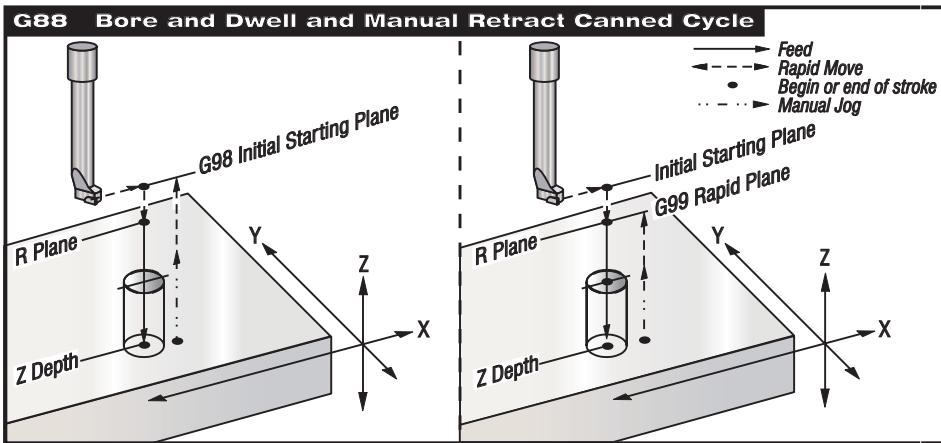
This G code will stop once the hole is bored. At this point the tool is manually jogged out of the hole. The program will continue when Cycle Start is pressed.



G88 Bore In, Dwell, Manual Retract Canned Cycle (Group 09)

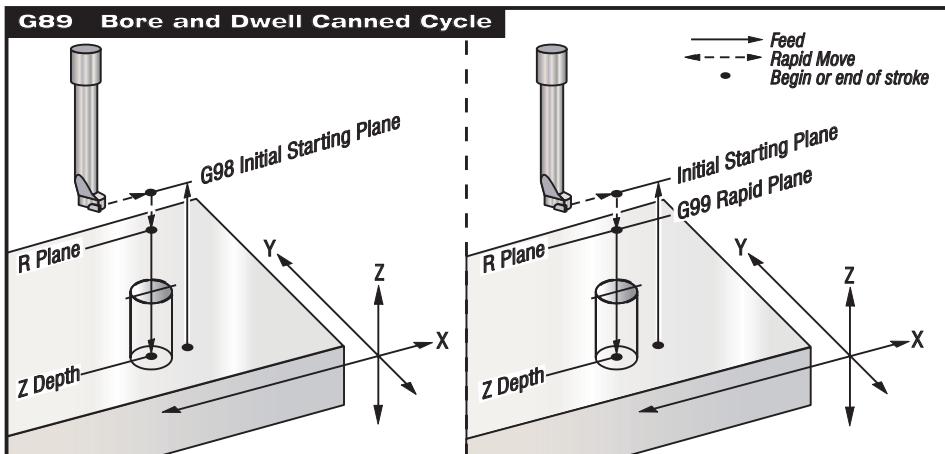
- F Feedrate in inches (or mm) per minute
- L Number of holes if G91 (Incremental Mode) is used
- P The dwell time at the bottom of the hole
- R Position of the R plane (position above the part)
- X X-axis location of hole
- Y Y-axis location of hole
- Z Position of the Z-axis at the bottom of hole

This G code will stop once the hole is bored. At this point the tool is manually jogged out of the hole. The program will continue when Cycle Start is pressed.



G89 Bore In, Dwell, Bore Out Canned Cycle (Group 09)

F Feedrate in inches (or mm) per minute
 L Number of holes if G91 (Incremental Mode) is used
 P The dwell time at the bottom of the hole
 R Position of the R plane (position above the part)
 X X-axis location of holes
 Y Y-axis location of holes
 Z Position of the Z-axis at the bottom of hole

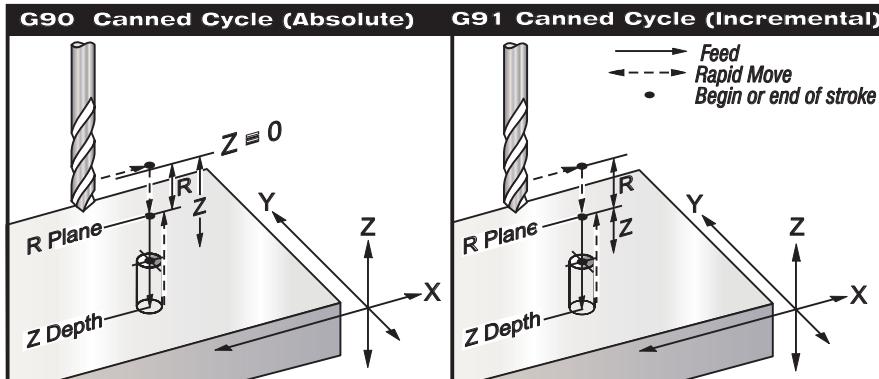


G & M Codes
Settings

G90 Absolute Position Commands (Group 03)

G91 Incremental Position Commands (Group 03)

These G codes change the way the axis commands are interpreted. Axes commands following a G90 will move the axes to the machine coordinate. Axes commands following a G91 will move the axis that distance from the current point. G91 is not compatible with G143 (5-Axis Tool Length Compensation).



G92 Set Work Coordinate Systems Shift Value (Group 00)

This G-code does not move any of the axes; it only changes the values stored as user work offsets. G92 works differently depending on Setting 33, which selects a FANUC, HAAS, or YASNAC coordinate system.

FANUC or HAAS

If setting 33 is set to Fanuc or Haas, a G92 command shifts all work coordinate systems (G54-59, G110-129) so that the commanded position becomes the current position in the active work system. G92 is non-modal.

A G92 command cancels any G52 in effect for the commanded axes. Example: G92 X1.4 cancels the G52 for the X-axis. The other axes are not affected.

The G92 shift value is displayed at the bottom of the Work Offsets page and may be cleared there if necessary. It is also cleared automatically after power-up, and any time ZERO RET and AUTO ALL AXES or ZERO SINGLE AXIS are used.

YASNAC

If setting 33 is set to Yasnac, a G92 command sets the G52 work coordinate system so that the commanded position becomes the current position in the active work system. The G52 work system then automatically becomes active until another work system is selected.

G93 Inverse Time Feed Mode (Group 05)

F Feed Rate (strokes per minute)

This G code specifies that all F (feedrate) values are interpreted as "strokes per minute." In other words the time (in seconds) to complete the programmed motion using G93 is, 60 (seconds) divided by the F value.

G93 is generally used in 4 and 5-axis work when the program is generated using a CAM system. G93 is a way of translating the linear (inches/min)



feedrate into a value that takes rotary motion into account. When G93 is used, the F value will tell you how many times per minute the stroke (tool move) can be repeated.

When G93 is used, feedrate (F) is mandatory for all interpolated motion blocks. Therefore each non-rapid motion block must have its own feedrate (F) specification.

Notes: Pressing RESET will set the machine to G94 (Feed per Minute) mode.

Settings 34 and 79 (4th & 5th axis diameter) are not necessary when using 93.

G94 Feed Per Minute Mode (Group 05)

This code deactivates G93 (Inverse Time Feed Mode) and returns the control to Feed Per Minute mode.

G95 Feed per Revolution (Group 05)

When G95 is active; a spindle revolution will result in a travel distance specified by the Feed value. If the Setting 9 Dimensioning is set to Inch, then the feed value F will be taken as inches/rev (set to MM, then the feed will be taken as mm/Rev). Feed Override and Spindle override will affect the behavior of the machine while G95 is active. When a spindle override is selected, any change in the spindle speed will result in a corresponding change in feed in order to keep the chip load uniform. However, if a feed override is selected, then any change in the feed override will only affect the feed rate and not the spindle.

G98 Canned Cycle Initial Point Return (Group 10)

Using G98, the Z-axis returns to its initial starting point (the Z position in the block before the canned cycle was commanded) between each X and/or Y location. This allows for positioning up and around areas of the part and/or clamps and fixtures.

G99 Canned Cycle R Plane Return (Group 10)

Using G99, the Z-axis will stay at the R plane between each X and/or Y location. When obstructions are not in the path of the tool G99 saves machining time.

G100 Cancel Mirror Image (Group 00)

G101 Enable Mirror Image (Group 00)

X	X-axis command
Y	Y-axis command
Z	Z-axis command
A	A-axis command

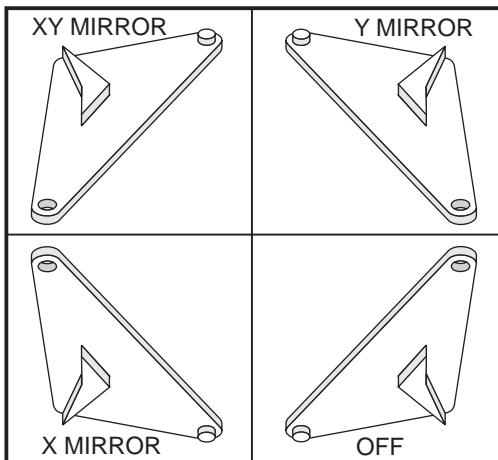
Programmable mirror imaging is used to turn on or off any the axes. When one is ON, axis motion may be mirrored (or reversed) around the work zero point. These G codes should be used in a command block without any other G codes. They do not cause any axis motion. The bottom of the screen will indicate when an axis is mirrored. Also see Settings 45 through 48 for mirror imaging.



The format for turning Mirror Image on and off is:

G101 X09 = Will turn on mirror imaging for the X axis.

G100 X09 = Will turn off mirror imaging for the X axis.



Mirror Image and Cutter Compensation

When using cutter compensation with mirror imaging, follow this guideline:

After turning mirror imaging on or off with G100 or G101, the next motion block should be to a different work coordinate position than the first one. The following code is an example:

Correct:

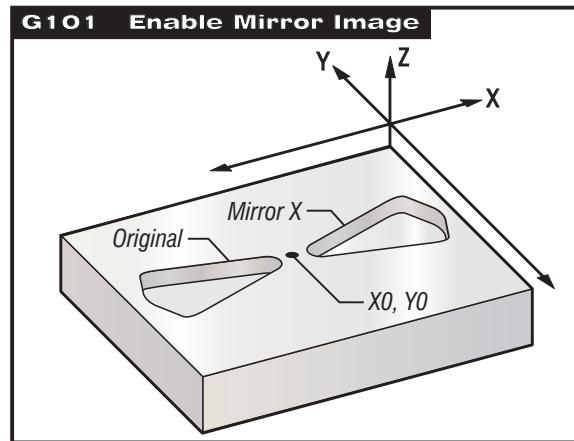
```
G41 X1.0 Y1.0  
G01 X2.0 Y2.0  
G101 X0  
G00 Z1.0  
G00 X1.0  
G00 X2.0 Y2.0  
G40
```

Incorrect:

```
G41 X1.0 Y1.0  
G01 X2.0 Y2.0  
G101 X0  
G00 Z1.0  
G00 X2.0 Y2.0  
G40.
```

Mirroring only one of the **X** or **Y** axes will cause the cutter to move along the opposite side of a cut. In addition, if mirror imaging is selected for only one axis of a circular motion plane (G02, G03) then they are reversed, and left and right cutter compensation commands (G41, G42) are reversed.

Note: When milling a shape with XY motions, turning on Mirror Image for just one of the X or Y axes will change climb milling to conventional milling and/or conventional milling to climb milling. As a result, the type of cut or finish may not be what was desired. Mirror imaging of both X and Y will eliminate this problem.



Mirror Image and Pocket Milling

Program Code for Mirror Imaging in the X-Axis:

Program Example

Description

```
%  
O3600  
T1 M06  
G00 G90 G54 X-.4653 Y.052 S5000 M03  
G43 H01 Z.1 M08  
G01 Z-.25 F5.  
M98 P3601 F20.  
G00 Z.1  
G101 X0.  
X-.4653 Y.052  
G01 Z-.25 F5.  
M98 P3601 F20.  
G00 Z.1  
G100 X0.  
G28 G91 Y0 Z0  
M30  
%  
%  
O3601  
G01 X-1.2153 Y.552  
G03 X-1.3059 Y.528 R.0625  
G01 X-1.5559 Y.028  
G03 X-1.5559 Y-.028 R.0625  
G01 X-1.3059 Y-.528  
G03 X-1.2153 Y-.552 R.0625  
G01 X-.4653 Y-.052  
G03 X-.4653 Y.052 R.0625  
M99  
%
```

(Contour subprogram)

**G & M Codes
Settings**



G102 Programmable Output to RS-232 (Group 00)

X X-axis command
Y Y-axis command
Z Z-axis command
A A-axis command

Commanding a G102 will send the current work coordinates of the axes to the first RS-232 port, from there a computer is used to record the values sent. Each axis listed in the G102 command block is output to the RS-232 port in the same format as values displayed in a program. A G102 should be used in a command block without any other G-codes. It will not cause any axis motion, the value for the axes have no effect.

Also see Setting 41 and Setting 25. The values sent out are always the current axis positions referenced to the current work coordinate system.

This G-code is useful in order to probe a part (also see G31). When the probe touches the part, the next line of code could be a G102 to send the axes position to a computer in order to store the coordinates. This is referred to as digitizing a part, which is taking a tangible part and making an electronic copy of it. Additional software for personal computers is required to complete this function.

G103 Limit Block Buffering (Group 00)

Maximum number of blocks the control will look ahead (Range 0-15), for example:

G103 [P.]

This is commonly referred to, as “Block Look-ahead” which is a term used to describe what the control is doing in the background during machine motions. The control prepares future blocks (lines of code) ahead of time. While the current block is executing, the next block has already been interpreted and prepared for continuous motion.

When G103 P0 is programmed, block limiting is disabled. Block limiting is also disabled if G103 appears in a block without a P address code. When G103 Pn is programmed, look-ahead is limited to n blocks.

G103 is also useful for debugging macro programs. Macro expressions are done during look-ahead time. For example, by inserting a G103 P1 into the program, macro expressions will be performed one block ahead of the currently executing block.

G107 Cylindrical Mapping (Group 00)

X X-axis command
Y Y-axis command
Z Z-axis command
A A-axis command
Q Diameter of cylindrical surface
R Radius of rotary axis



This G code translates all programmed motion occurring in a specified linear axis into the equivalent motion along the surface of a cylinder (as attached to a rotary axis) as shown in the following figure. It is a Group 0 G code, but its default operation is subject to Setting 56 (M30 Restores Default G). The G107 command is used to either activate or deactivate cylindrical mapping.

- Any linear-axis program can be cylindrically mapped to any rotary axis (one at a time).
- An existing linear-axis G-code program can be cylindrically mapped by inserting a G107 command at the beginning of the program.
- The radius (or diameter) of the cylindrical surface can be redefined, allowing cylindrical mapping to occur along surfaces of different diameters without having to change the program.
- The radius (or diameter) of the cylindrical surface can either be synchronized with or be independent of the rotary axis diameter(s) specified in the Settings 34 and 79.
- G107 can also be used to set the default diameter of a cylindrical surface, independently of any cylindrical mapping that may be in effect.

G107 Description

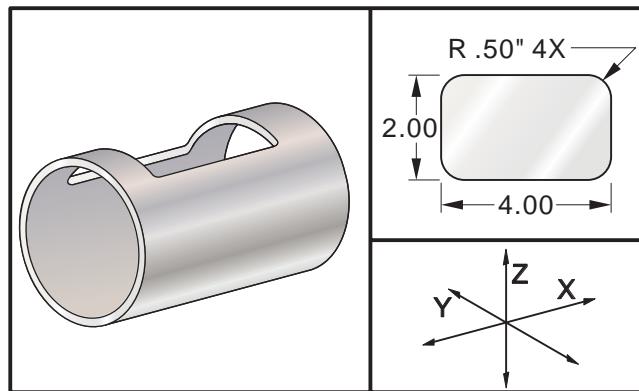
Three address codes can follow a G107: **X**, **Y** or **Z**; **A** or **B**; and **Q** or **R**.

X, **Y**, or **Z**: An X, Y, or Z address specifies the linear axis that will be mapped to the specified rotary axis (A or B). When one of these linear axes is specified, a rotary axis must also be specified.

A or **B**: An A or B address identifies which rotary axis holds the cylindrical surface.

Q or **R**: Q defines the diameter of the cylindrical surface, while R defines the radius. When Q or R is used, a rotary axis must also be specified. If neither Q nor R is used, then the last G107 diameter is used. If no G107 command has been issued since power-up, or if the last value specified was zero, then the diameter will be the value in Setting 34 and/or 79 for this rotary axis. When Q or R is specified, that value will become the new G107 value for the specified rotary axis.

Cylindrical mapping will also be turned off automatically whenever the G-code program ends, but only if Setting 56 is ON. Pressing the RESET key will turn off any cylindrical mapping that is currently in effect, regardless of the status of Setting 56.



While R is suitable for defining the radius, it is recommended that I, J and K are used for more complex G02 and G03 programming.

Example

```
%  
O0079 (G107 TEST)  
T1 M06 (.625 DIA. 2FL E.M.)  
G00 G40 G49 G80 G90  
G28 G91 A0  
G90  
G00 G54 X1.5 Y0 S5000 M03  
G107 A0 Y0 R2. (IF NO R OR Q VALUE, MACHINE WILL USE VALUE IN SETTING 34)  
G43 H01 Z0.25  
G01 Z-0.25 F25.  
G41 D01 X2. Y0.5  
G03 X1.5 Y1. R0.5  
G01 X-1.5  
G03 X-2. Y0.5 R0.5  
G01 Y-0.5  
G03 X-1.5 Y-1. R0.5  
G01 X1.5  
G03 X2. Y-0.5 R0.5  
G01 Y0.  
G40 X1.5  
G00 Z0.25  
M09  
M05  
G91 G28 Z0.  
G28 Y0.  
G90  
G107  
M30  
%
```



G110-G129 Coordinate System #7-26 (Group 12)

These codes select one of the additional work coordinate systems. All subsequent references to axis positions will be interpreted in the new coordinate system. Operation of G110 to G129 is the same as G54 to G59.

G136 Automatic Work Offset Center Measurement (Group 00)

(This G-code is optional and requires a probe)

F	Feedrate in inches (mm) per minute
I	Optional offset distance along X-axis
J	Optional offset distance along Y-axis
K	Optional offset distance along Z-axis
X	Optional X-axis motion command
Y	Optional Y-axis motion command
Z	Optional Z-axis motion command

Automatic Work Offset Center Measurement (G136) is used to command a probe to set work offsets. A G136 will feed the axes of the machine in an effort to probe the workpiece with a spindle mounted probe. The axis (axes) will move until a signal from the probe is received, or the travel limit is reached.

Tool offsets (G41, G42, G43, or G44) must not be active when this function is preformed. The currently active work coordinate system is set for each axis programmed. Use a G31 cycle with an M75 to set the first point. A G136 will set the work coordinates to a point at the center of a line between the probed point and the point set with an M75. This allows the center of the part to be found using two separate probed points.

If an I, J, or K is specified, the appropriate axis work offset is shifted by the amount in the I, J, or K command. This allows the work offset to be shifted away from where the probe actually contacts the part.

Notes:

Also see G31.

The points probed are offset by the values in Settings 59 through 62.

Use G91 incremental moves when using a G36.

Use the assigned M-codes (M53 & M63) with a dwell to turn the spindle probe on or off.

Example:

M53

G04 P100

M63

Programming example to probe the center of a bore:

O1234 (G136)

M53

G04 P100

M63

G00 G90 G54 X0 Y0

Z-17.

G91 G01 Z-1. F20.

G31 X1. F10. M75



G01 X-1.

G136 X-1. F10.

G01 X1.

M53

G04 P100

M63

G00 G90 G53 Z0

M30

Programming example to probe the center of a part:

O1234 (G136)

M53

G04 P100

M63

G00 G90 G54 X0 Y5.

Z-17.

G91 G01 Z-1. F20.

G31 Y-1. F10. M75

G01 Y1. F20.

G00 Z2.

Y-10.

G01 Z-2. F20.

G136 Y1. F10.

G01 Y-1.

M53

G04 P100

M63

G00 G90 G53 Z0

M30

G141 3D+ Cutter Compensation (Group 07)

X X-axis command

Y Y-axis command

Z Z-axis command

A A-axis command (optional)

B B-axis command (optional)

D Cutter Size Selection (modal)

I X-axis cutter compensation direction from program path

J Y-axis cutter compensation direction from program path

K Z-axis cutter compensation direction from program path

F Feed rate in G93 or G94 (modal in G94)

This feature performs three-dimensional cutter compensation.

The form is:

G141 Xnnn Ynnn Znnn Innn Jnnn Knnc Fnnc Dnnn

Subsequent lines can be:

G01 Xnnn Ynnn Znnn Innn Jnnn Knnc Fnnc

Or

G00 Xnnn Ynnn Znnn Innn Jnnn Knnc



Some CAM systems are able to output the X, Y, and Z with values for I, J, K. The I, J, and K values tell the control the direction in which to apply the compensation at the machine.

The I, J, and K specify the normal direction relative to the center of the tool to the contact point of the tool in the CAM system. The I, J, and K vectors are required by the control to be able to shift the tool path in the correct direction. The value of the compensation can be in a positive or negative direction.

The offset amount entered in radius or diameter (Setting 40) for the tool will compensate the path by this amount even if the tool motions are 2 or 3 axes.

Only G00 and G01 can use G141. A Dnn will have to be programmed, the D-code selects which offset to use. G93 feed command is required on each block.

With a unit vector, $I_2 + J_2 + K_2$ must equal 1.

Only the end-point of the commanded block is compensated in the direction of I, J, and K. For this reason this compensation is recommended only for surface tool paths having a tight tolerance (small motion between blocks of code).

For best results program from the tool center using a ball nose end mill.

G141 Example:

T1 M06

G00 G90 G54 X0 Y0 Z0 A0 B0

G141 D01 X0.Y0. Z0. (RAPID POSIT WITH 3 AX C COMP)

G01 G93 X.01 Y.01 Z.01 I.1 J.2 K.9747 F300. (FEED INV TIME)

X.02 Y.03 Z.04 I.15 J.25 K.9566 F300.

X.02 Y.055 Z.064 I.2 J.3 K.9327 F300.

X2.345 Y.1234 Z-1.234 I.25 J.35 K.9028 F200. (LAST MOTION)

G94 F50. (CANCEL G93)

G0 G90 G40 Z0 (Rapid to Zero, Cancel Cutter Comp)

X0 Y0

M30

G143 5-Axis Tool Length Compensation + (Group 08)

(This G-code is optional; it only applies to machines on which all rotary motion is movement of the cutting tool.)

This G code allows the user to correct for variations in the length of cutting tools without the need for a CAD/CAM processor. An H code is required to select the tool length from the existing length compensation tables. A G49 or H00 command will cancel 5-axis compensation. For G143 to work correctly there must be two rotary axes, A and B. G90, absolute positioning mode must be active (G91 cannot be used). Work position 0,0 for the A and B axes must be so the tool is parallel with Z-axis motion.



The intention behind G143 is to compensate for the difference in tool length between the originally posted tool and a substitute tool. Using G143 allows the program to run without having to repost a new tool length.

G143 tool length compensation works only with rapid (G00) and linear feed (G01) motions; no other feed functions (G02 or G03) or canned cycles (drilling, tapping, etc.) can be used. For a positive tool length, the Z-axis would move upward (in the + direction). If one of X, Y or Z is not programmed, there will be no motion of that axis, even if the motion of A or B produces a new tool length vector. Thus a typical program would use all 5 axes on one block of data. G143 may effect commanded motion of all axes in order to compensate for the A and B axes.

Inverse feed mode (G93) is recommended, when using G143. An example follows:

```
T1 M06  
G00 G90 G54 X0 Y0 Z0 A0 B0  
G143 H01 X0. Y0. Z0. A-20. B-20. (RAPID POSIT W. 5AX COMP)  
G01 G93 X.01 Y.01 Z.01 A-19.9 B-19.9 F300. (FEED INV TIME)  
X0.02 Y0.03 Z0.04 A-19.7 B-19.7 F300.  
X0.02 Y0.055 Z0.064 A-19.5 B-19.6 F300.  
X2.345 Y.1234 Z-1.234 A-4.127 B-12.32 F200. (LAST MOTION)  
G94 F50. (CANCEL G93)  
G0 G90 G49 Z0 (RAPID TO ZERO, CANCEL 5 AXS COMP)  
X0 Y0  
M30
```

G150 General Purpose Pocket Milling (Group 00)

- D Tool radius/diameter offset selection
- F Feedrate
- I X-axis cut increment (positive value)
- J Y-axis cut increment (positive value)
- K Finishing pass amount (positive value)
- P Subprogram number that defines pocket geometry
- Q Incremental Z-axis cut depth per pass (positive value)
- R Position of the rapid R-plane location
- S Optional spindle speed
- X X start position
- Y Y start position
- Z Final depth of pocket

The G150 starts by positioning the cutter to a start point inside the pocket, followed by the outline, and completes with a finish cut. The end mill will plunge in the Z-axis. A subprogram P### is called that defines the pocket geometry of a closed area using G01, G02, and G03 motions in the X and Y axes on the pocket. The G150 command will search for an internal subprogram with a N-number specified by the P-code. If that is not found the control will search for an external subprogram. If neither are found, alarm 314 Subprogram Not In Memory will be generated.



NOTE: When defining the G150 pocket geometry in the subprogram, do not move back to the starting hole after the pocket shape is closed.

An I or J value defines the roughing pass amount the cutter moves over for each cut increment. If I is used, the pocket is roughed out from a series of increment cuts in the X-axis. If J is used, the increment cuts are in the Y-axis.

The K command defines a finish pass amount on the pocket. If a K value is specified, a finish pass is performed by K amount, around the inside of pocket geometry for the last pass and is done at the final Z depth. There is no finishing pass command for the Z depth.

The R value needs to be specified, even if it is zero (R0), or the last R value that was specified will be used.

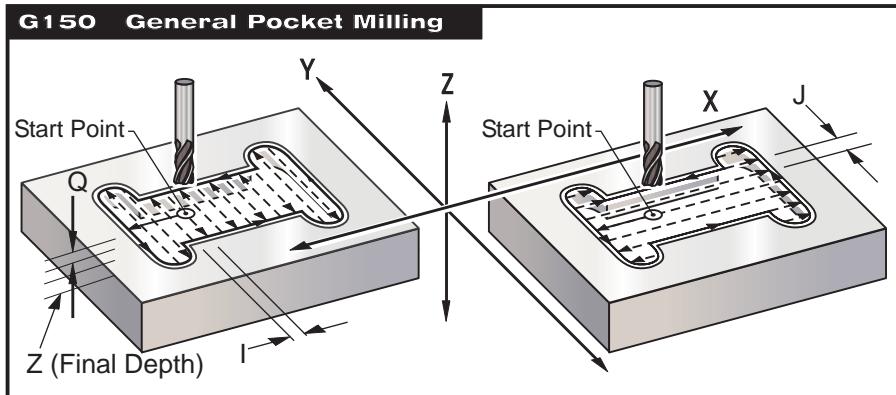
Multiple passes in the pocket area are done, starting from the R plane, with each Q (Z-axis depth) pass to the final depth. The G150 command will first make a pass around pocket geometry, leaving stock with K, then doing passes of I or J roughing out inside of pocket after feeding down by the value in Q until the Z depth is reached.

The Q command must be in the G150 line, even if only one pass to the Z depth is desired. The Q command starts from the R plane.

Notes: The subprogram (P) must not consist of more than 40 pocket geometry moves.

The Q command must be in the G150 line, even if only one pass to the Z depth is desired. The Q command starts from the R plane.

It may be necessary to drill a starting point, for the G150 cutter, to the final depth (Z). Then position the end mill to the start location in the XY axes within the pocket for the G150 command.



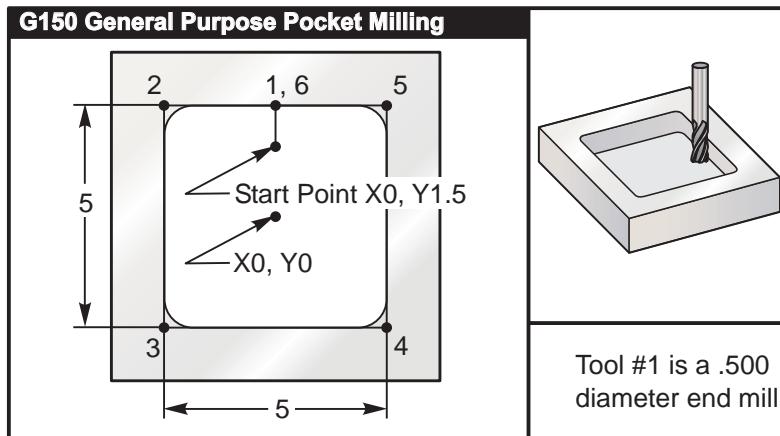


Example

O01001	(G150 Pocket example)
T1 M06	(T1 Drills clearance hole for endmill)
G90 G54 G00 X3.25 Y4.5 S1200	(Pocket start point)
M03	
G43 H01 Z1.0 M08	(Tool length offset, rapid to Z start point, coolant on)
G83 Z-1.5 Q0.25 R0.1 F20.	(Peck drill cycle)
G53 G49 Z0	(Returns Z to home position)
T2 M06 (.5" Endmill)	(T2 Cuts pocket in two passes to Z depth)
G54 G90 G00 X3.25 Y4.5 S1450	(Pocket start point)
M03	
G43 H02 Z1.0 M08	(Tool length offset, rapid to Z start point, coolant on)
G150 X3.25 Y4.5 Z-1.5 G41 J0.35	
K.01 Q0.8 R.1 P2001 D02 F15.	
(0.01" finish pass (K) on sides)	
G40 X3.25 Y4.5	(Cancel cutter comp. and position back to start point)
G53 G49 Y0 Z0	(Returns Z to home position)
M30	(End of main program)
O02001	(Separate program as a subprogram for G150 pocket geometry)
G01 Y7	(The first move onto pocket geometry with a G01)
X1.5	(The following lines define pocket geometry)
G03 Y5.25 R0.875	
G01 Y2.25	
G03 Y0.5 R0.875	
G01 X5.	
G03 Y2.25 R0.875	
G01 Y5.25	
G03 Y7. R0.875	
G01 X3.25	
M99	(Close pocket geometry. Do not go back to start.)
	(Return to Main Program)



Square Pocket



Pocket Milling for G150 Operations

5.0 x 5.0 x 0.500 DP. Square Pocket

Main Program

```
%  
O01001  
T1 M06 (Tool #1 is a 0.500" diameter endmill)  
G90 G54 G00 X0. Y1.5 (XY Start Point)  
S2000 M03  
G43 H01 Z0.1 M08  
G01 Z0.1 F10.  
G150 P1002 Z-0.5 Q0.25 R0.01 J0.3 K0.01 G41 D01 F10. X0. (6) (Close Pocket Loop)  
  
G40 G01 X0. Y1.5  
M99 (Return to Main  
Program)  
%  
G00 Z1. M09  
G53 G49 Y0. Z0.  
M30  
%
```

Subprogram

```
%  
O01002  
G01 Y2.5 (1)  
X-2.5 (2)  
Y-2.5 (3)  
X2.5 (4)  
Y2.5 (5)
```

Absolute and Incremental examples of a subprogram called up by the P### command in the G150 line:

Absolute Subprogram

```
%  
O01002 (G90 Subprogram for G150)  
G90 G01 Y2.5 (1)  
X-2.5 (2)  
Y-2.5 (3)  
X2.5 (4)  
Y2.5 (5)  
X0. (6)  
M99  
%
```

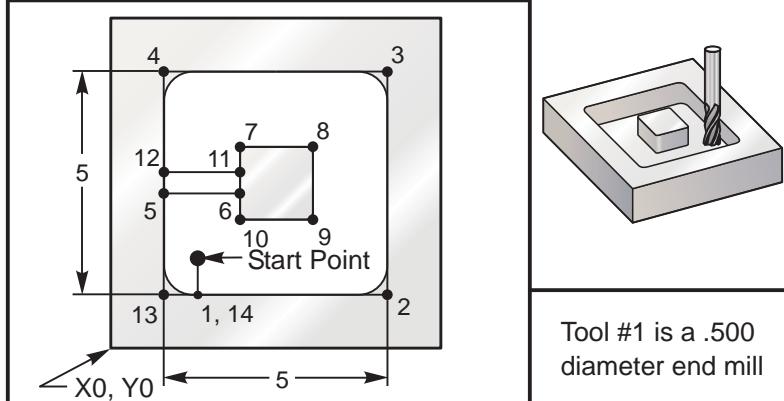
Incremental Subprogram

```
%  
O01002 (G91 Subprogram for G150)  
G91 G01 Y0.5 (1)  
X-2.5 (2)  
Y-5. (3)  
X5. (4)  
Y5. (5)  
X-2.5 (6)  
G90  
M99  
%
```



Square Island

G150 Pocket Milling (Square Island)



G150 Pocket Milling Program with a Square Island

5.0 x 5.0 x 0.500 DP. Square Pocket with Square Island

Main Program

%
O02010
T1 M06 (Tool is a 0.500" diameter endmill)

G90 G54 G00 X2. Y2. (XY Start Point)
S2500 M03
G43 H01 Z0.1 M08
G01 Z0.01 F30.
G150 P2020 X2. Y2. Z-0.5 Q0.5 R0.01 I0.3 X2.75 (6)
K0.01 G41 D01 F10.
G40 G01 X2. Y2.
G00 Z1.0 M09
G53 G49 Y0. Z0.
M30

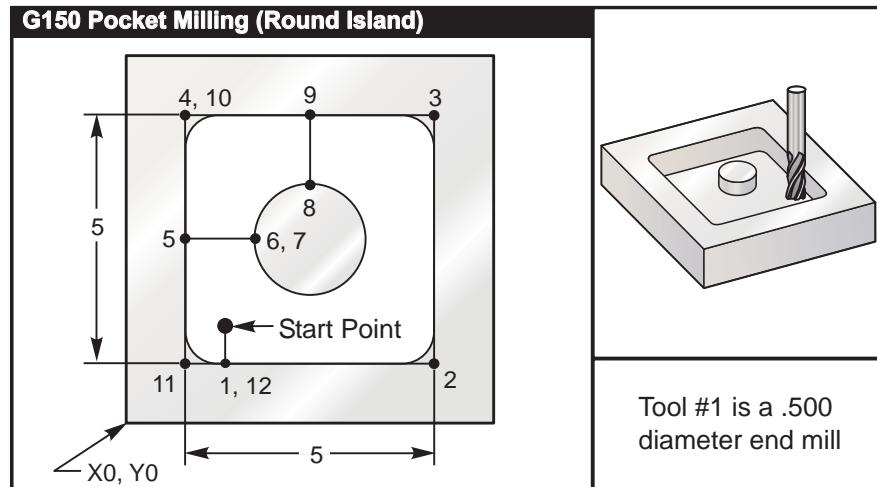
X6. (2)
Y6. (3)
X1. (4)
Y3.2 (5)
X2.75 (6)
Y4.25 (7)
X4.25 (8)
Y2.75 (9)
X2.75 (10)
Y3.8 (11)
X1. (12)
Y1. (13)
X2. (14) (Close Pocket Loop)
M99 (Return to Main Program)
%

Subprogram

%
O02020 (Subprogram for G150 in O02010)
G01 Y1. (1)



Round Island



G150 Pocket Milling Program with a Round Island

5.0 x 5.0 x 0.500 DP. Square Pocket with Round Island

Main Program

```
%  
O03010  
T1 M06 (Tool is a 0.500" diameter endmill)  
G90 G54 G00 X2. Y2. (XY Start Point)  
S2500 M03  
G43 H01 Z0.1 M08  
G01 Z0. F30.  
G150 P3020 X2. Y2. Z-0.5 Q0.5 R0.01 J0.3  
K0.01 G41 D01 F10.  
G40 G01 X2. Y2.  
G00 Z1. M09  
G53 G49 Y0. Z0.  
M30  
%
```

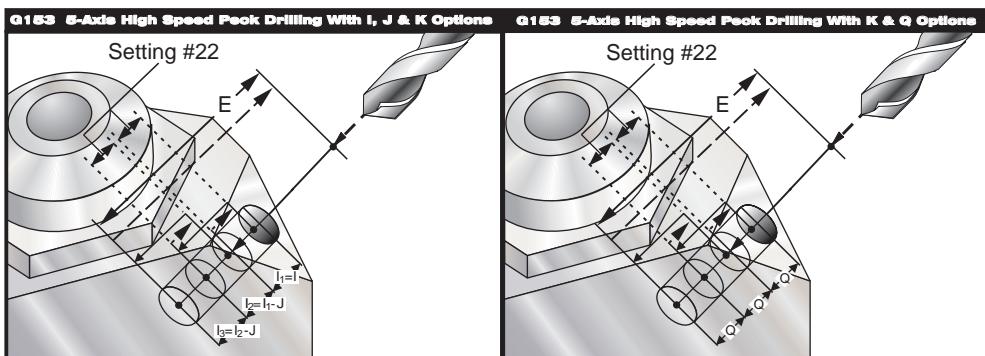
Subprogram

```
%  
O03020 (Subprogram for G150 in O03010)  
G01 Y1. (1)  
X6. (2)  
Y6. (3)  
X1. (4)  
Y3.5 (5)  
X2.5 (6)  
G02 I1. (7)  
G02 X3.5 Y4.5 R1. (8)  
G01 Y6. (9)  
X1. (10)  
Y1. (11)  
X2. (12) (Close Pocket Loop)  
M99 (Return to Main Program)  
%
```



G153 5-Axis High Speed Peck Drilling Canned Cycle (Group 09)

- E Specifies the distance from the start position to the bottom of the hole
- F Feedrate in inches (mm) per minute
- I Size of first cutting depth (must be a positive value)
- J Amount to reduce cutting depth each pass (must be a positive value)
- K Minimum depth of cut (must be a positive value)
- L Number of repeats
- P Pause at end of last peck, in seconds
- Q The cut-in value (must be a positive value)
- A A-axis tool starting position
- B B-axis tool starting position
- X X-axis tool starting position
- Y Y-axis tool starting position
- Z Z-axis tool starting position



This is a high-speed peck cycle where the retract distance is set by Setting 22.

If **I**, **J**, and **K** are specified, a different operating mode is selected. The first pass will cut in by amount **I**, each succeeding cut will be reduced by amount **J**, and the minimum cutting depth is **K**. If **P** is used, the tool will pause at the bottom of the hole for that amount of time.

Note that the same dwell time applies to all subsequent blocks that do not specify a dwell time.

G154 Select Work Coordinates P1-P99 (Group 12)

This feature provides 99 additional work offsets. G154 with a P value from 1 to 99 activates additional work offsets. For example G154 P10 selects work offset 10 from the list of additional work offsets. Note that G110 to G129 refer to the same work offsets as G154 P1 through P20; they can be selected by using either method. When a G154 work offset is active, the heading in the upper right work offset will show the G154 P value.



G154 work offsets format

#14001-#14006 G154 P1 (also #7001-#7006 and G110)
#14021-#14026 G154 P2 (also #7021-#7026 and G111)
#14041-#14046 G154 P3 (also #7041-#7046 and G112)
#14061-#14066 G154 P4 (also #7061-#7066 and G113)
#14081-#14086 G154 P5 (also #7081-#7086 and G114)
#14101-#14106 G154 P6 (also #7101-#7106 and G115)
#14121-#14126 G154 P7 (also #7121-#7126 and G116)
#14141-#14146 G154 P8 (also #7141-#7146 and G117)
#14161-#14166 G154 P9 (also #7161-#7166 and G118)
#14181-#14186 G154 P10 (also #7181-#7186 and G119)
#14201-#14206 G154 P11 (also #7201-#7206 and G120)
#14221-#14221 G154 P12 (also #7221-#7226 and G121)
#14241-#14246 G154 P13 (also #7241-#7246 and G122)
#14261-#14266 G154 P14 (also #7261-#7266 and G123)
#14281-#14286 G154 P15 (also #7281-#7286 and G124)
#14301-#14306 G154 P16 (also #7301-#7306 and G125)
#14321-#14326 G154 P17 (also #7321-#7326 and G126)
#14341-#14346 G154 P18 (also #7341-#7346 and G127)
#14361-#14366 G154 P19 (also #7361-#7366 and G128)
#14381-#14386 G154 P20 (also #7381-#7386 and G129)
#14401-#14406 G154 P21
#14421-#14426 G154 P22
#14441-#14446 G154 P23
#14461-#14466 G154 P24
#14481-#14486 G154 P25
#14501-#14506 G154 P26
#14521-#14526 G154 P27
#14541-#14546 G154 P28
#14561-#14566 G154 P29
#14581-#14586 G154 P30
#14781-#14786 G154 P40
#14981-#14986 G154 P50
#15181-#15186 G154 P60
#15381-#15386 G154 P70
#15581-#15586 G154 P80
#15781-#15786 G154 P90
#15881-#15886 G154 P95
#15901-#15906 G154 P96
#15921-#15926 G154 P97
#15941-#15946 G154 P98
#15961-#15966 G154 P99



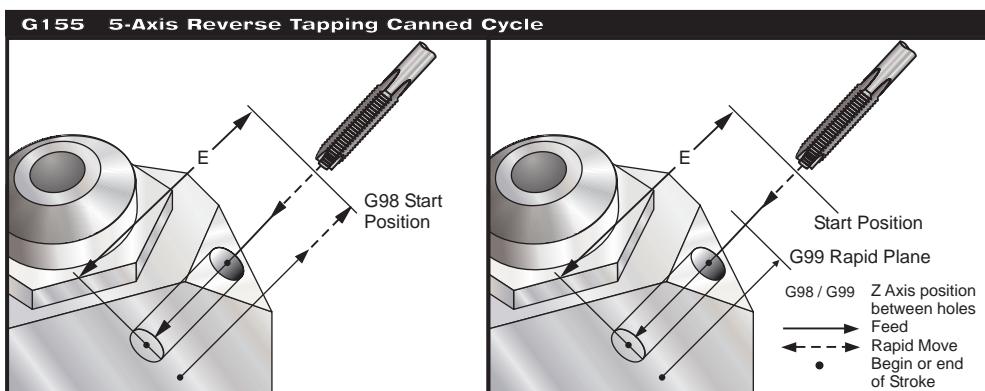
G155 5-Axis Reverse Tap Canned Cycle (Group 09)

G155 only performs floating taps. G174 is available for 5-axis reverse rigid tapping.

- E Specifies the distance from the start position to the bottom of the hole
- F Feedrate in inches (mm) per minute
- L Number of repeats
- A A-axis tool starting position
- B B-axis tool starting position
- X X-axis tool starting position
- Y Y-axis tool starting position
- Z Z-axis tool starting position
- S Spindle Speed

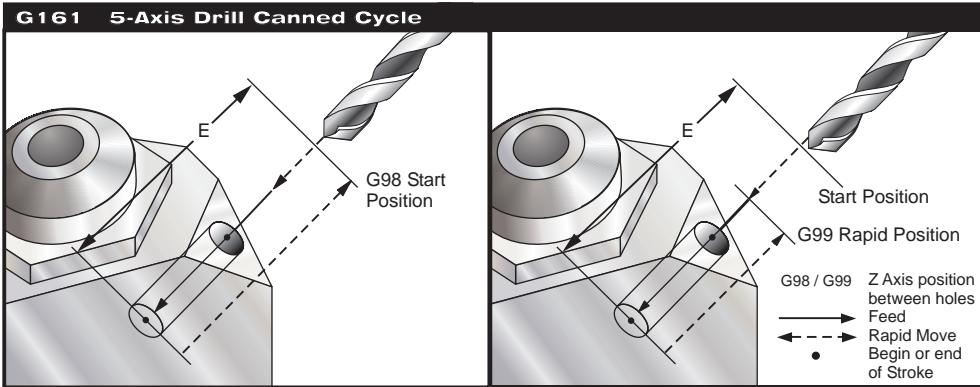
A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded. This position is used as the "Initial Start position".

The control will automatically start the spindle CCW before this canned cycle.



G161 5-Axis Drill Canned Cycle (Group 09)

- E Specifies the distance from the start position to the bottom of the hole
- F Feedrate in inches (mm) per minute
- A A-axis tool starting position
- B B-axis tool starting position
- X X-axis tool starting position
- Y Y-axis tool starting position
- Z Z-axis tool starting position



A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded.

Example

```
( DRILL RIGHT, FRONT )
T4 M6
G01 G54 G90 X8.4221 Y-8.4221 B23. A21.342 S2200 M3 F360. (Clearance Position)
G143 H4 Z14.6228 M8
G1 X6.6934 Y-6.6934 Z10.5503 F360. (Initial Start position)
G161 E.52 F7. (Canned Cycle)
G80
X8.4221 Y-8.4221 B23. A21.342 Z14.6228 (Clearance Position)
M5
G1 G28 G91 Z0.
G91 G28 B0. A0.
M01
```

G162 5-Axis Spot Drill Canned Cycle (Group 09)

- E Specifies the distance from the start position to the bottom of the hole
- F Feedrate in inches (mm) per minute
- P The dwell time at the bottom of the hole
- A A-axis tool starting position
- B B-axis tool starting position
- X X-axis tool starting position
- Y Y-axis tool starting position
- Z Z-axis tool starting position

A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded.



Example

(COUNTER DRILL RIGHT, FRONT)

T2 M6

G01 G54 G90 X8.4221 Y-8.4221 B23. A21.342 S2200 M3 F360. (Clearance Position)

G143 H2 Z14.6228 M8

G1 X6.6934 Y-6.6934 Z10.5503 F360. (Initial Start position)

G162 E.52 P2.0 F7. (Canned Cycle)

G80

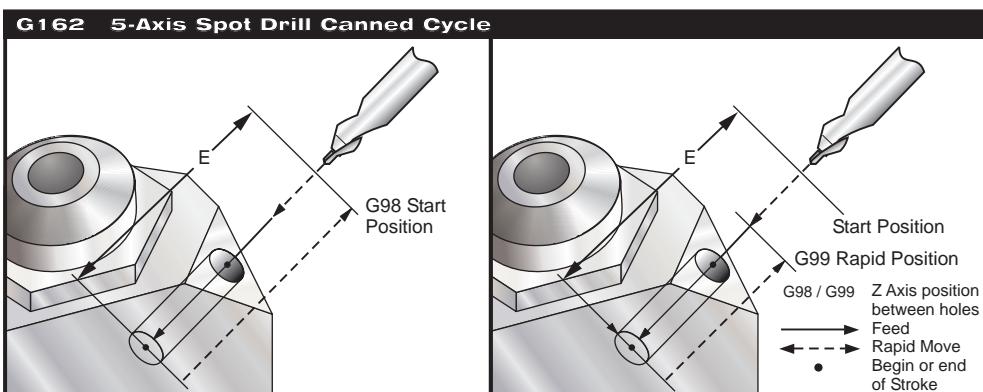
X8.4221 Y-8.4221 B23. A21.342 Z14.6228 (Clearance Position)

M5

G1 G28 G91 Z0.

G91 G28 B0. A0.

M01



G163 5-Axis Normal Peck Drilling Canned Cycle (Group 09)

- E Specifies the distance from the start position to the bottom of the hole
- F Feedrate in inches (mm) per minute
- I Optional size of first cutting depth
- J Optional amount to reduce cutting depth each pass
- K Optional minimum depth of cut
- P Optional pause at end of last peck, in seconds
- Q The cut-in value, always incremental
- A A-axis tool starting position
- B B-axis tool starting position
- X X-axis tool starting position
- Y Y-axis tool starting position
- Z Z-axis tool starting position

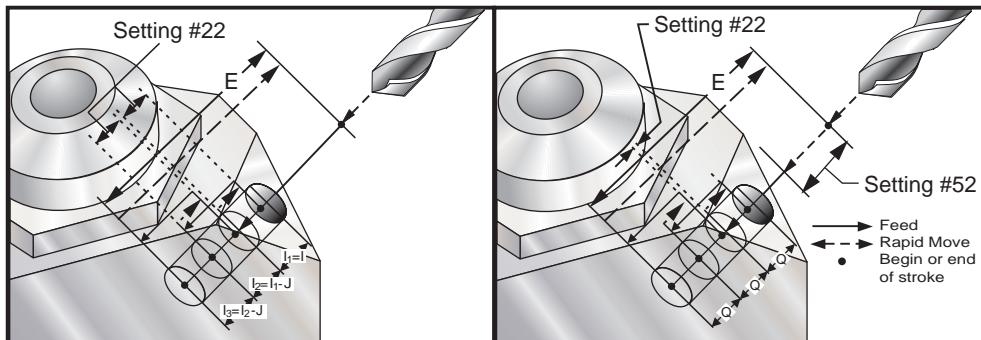
A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded.

If I, J, and K are specified the first pass will cut in by amount I, each succeeding cut will be reduced by amount J, and the minimum cutting depth is K.



A **P** value is used the tool will pause at the bottom of the hole after the last peck for that amount of time. The following example will peck several times and dwell for one and a half seconds at the end: G163 E0.62 F15. Q0.175 P1.5.

Note that the same dwell time applies to all subsequent blocks that do not specify a dwell time.



Setting 52 also changes the way G163 works when it returns to the **start position**. Usually the **R** plane is set well above the cut to ensure that the peck motion allows the chips to get out of the hole. This wastes time as the drill starts by drilling "empty" space. If Setting 52 is set to the distance required to clear chips, the **start position** can be put much closer to the part being drilled. When the chip-clearing move to the **start position** occurs, the **Z** axis will be moved above the **start position** by the amount given in this setting.

Example

(PECK DRILL RIGHT, FRONT)

T5 M6

G01 G54 G90 X8.4221 Y-8.4221 B23. A21.342 S2200 M3 F360. (Clearance Position)

G143 H5 Z14.6228 M8

G1 X6.6934 Y-6.6934 Z10.5503 F360. (Initial Start position)

G163 E1.0 Q.15 F12. (Canned Cycle)

G80

X8.4221 Y-8.4221 B23. A21.342 Z14.6228 (Clearance Position)

M5

G1 G28 G91 Z0.

G91 G28 B0. A0.

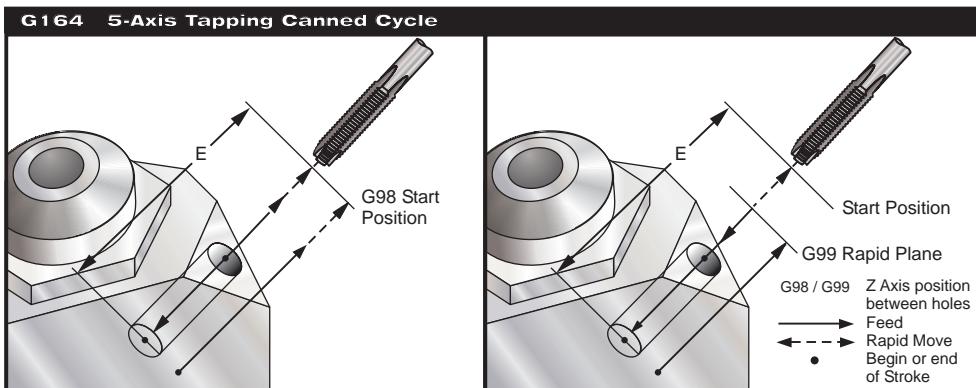
M01



G164 5-Axis Tapping Canned Cycle (Group 09)

G164 only performs floating taps. G174/184 is available for 5-axis rigid tapping.

- E Specifies the distance from the start position to the bottom of the hole
- F Feedrate in inches (mm) per minute
- A A-axis tool starting position
- B B-axis tool starting position
- X X-axis tool starting position
- Y Y-axis tool starting position
- Z Z-axis tool starting position
- S Spindle Speed



A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded. The control will automatically start the spindle CW before this canned cycle.

Example

(1/2-13 TAP)

T5 M6

G01 G54 G90 X8.4221 Y-8.4221 B23. A21.342 S500M3 F360. (Clearance Position)

G143 H5 Z14.6228 M8

G1 X6.6934 Y-6.6934 Z10.5503 F360. (Initial Start position)

G164 E1.0 F38.46 (Canned Cycle)

G80

X8.4221 Y-8.4221 B23. A21.342 Z14.6228 (Clearance Position)

M5

G1 G28 G91 Z0.

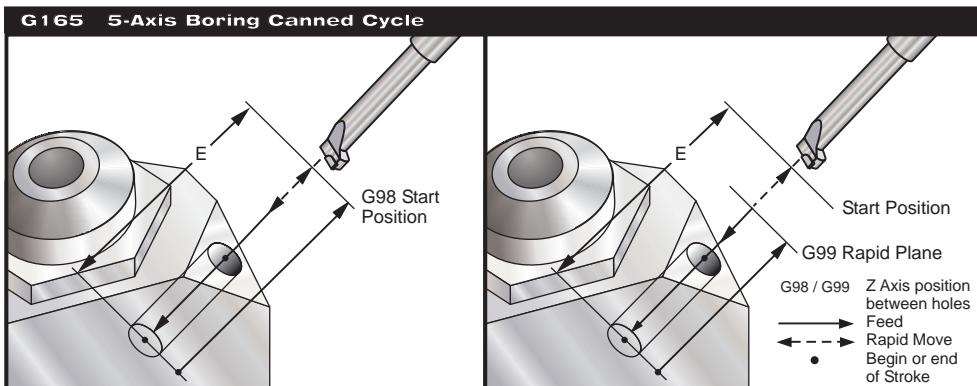
G91 G28 B0. A0.

M01



G165 5-Axis Boring Canned Cycle (Group 09)

- E Specifies the distance from the start position to the bottom of the hole
- F Feedrate in inches (mm) per minute
- A A-axis tool starting position
- B B-axis tool starting position
- X X-axis tool starting position
- Y Y-axis tool starting position
- Z Z-axis tool starting position



A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded.

Example

(Boring Cycle)

T5 M6

G01 G54 G90 X8.4221 Y-8.4221 B23. A21.342 S2200 M3 F360. (Clearance Position)

G143 H5 Z14.6228 M8

G1 X6.6934 Y-6.6934 Z10.5503 F360. (Initial Start position)

G165 E1.0 F12. (Canned Cycle)

G80

X8.4221 Y-8.4221 B23. A21.342 Z14.6228 (Clearance Position)

M5

G1 G28 G91 Z0.

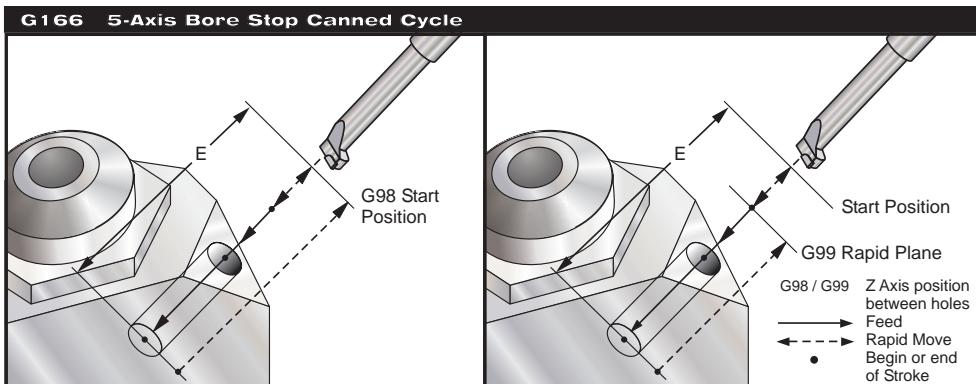
G91 G28 B0. A0.

M01



G166 5-Axis Bore and Stop Canned Cycle (Group 09)

- E Specifies the distance from the start position to the bottom of the hole
- F Feedrate in inches (mm) per minute
- A A-axis tool starting position
- B B-axis tool starting position
- X X-axis tool starting position
- Y Y-axis tool starting position
- Z Z-axis tool starting position



A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded.

Example

(Bore and Stop Cycle)

T5 M6

G01 G54 G90 X8.4221 Y-8.4221 B23. A21.342 S2200 M3 F360. (Clearance Position)

G143 H5 Z14.6228 M8

G1 X6.6934 Y-6.6934 Z10.5503 F360. (Initial Start position)

G166 E1.0 F12. (Canned Cycle)

G80

X8.4221 Y-8.4221 B23. A21.342 Z14.6228 (Clearance Position)

M5

G1 G28 G91 Z0.

G91 G28 B0. A0.

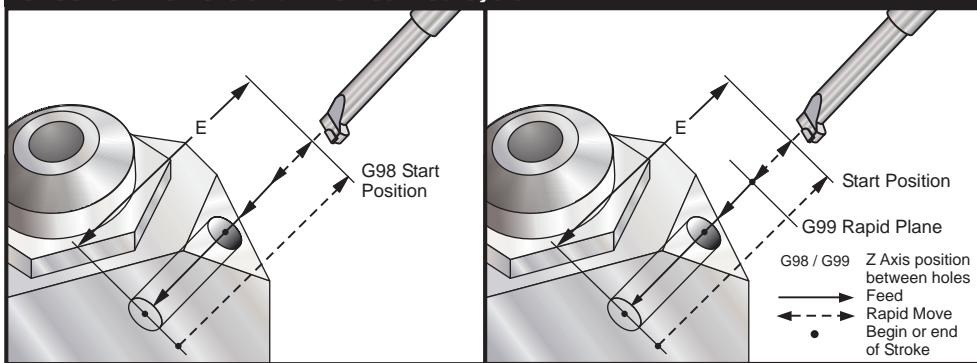
M01

G169 5-Axis Bore and Dwell Canned Cycle (Group 09)

- E Specifies the distance from the start position to the bottom of the hole
- F Feedrate in inches (mm) per minute
- P The dwell time at the bottom of the hole
- A A-axis tool starting position
- B B-axis tool starting position
- X X-axis tool starting position
- Y Y-axis tool starting position
- Z Z-axis tool starting position



G169 5-Axis Bore and Dwell Canned Cycle



A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded.

Example

(Bore and Dwell Cycle)

T5 M6

G01 G54 G90 X8.4221 Y-8.4221 B23. A21.342 S2200 M3 F360. (Clearance Position)

G143 H5 Z14.6228 M8

G1 X6.6934 Y-6.6934 Z10.5503 F360. (Initial Start position)

G169 E1.0 P5.0 F12. (Canned Cycle)

G80

X8.4221 Y-8.4221 B23. A21.342 Z14.6228 (Clearance Position)

M5

G1 G28 G91 Z0.

G91 G28 B0. A0.

M01

G174 CCW Non-Vertical Rigid Tap (Group 00)

G184 CW Non-Vertical Rigid Tap (Group 00)

F Feedrate in inches per minute

X X position at bottom of hole

Y Y position at bottom of hole

Z Z position at bottom of hole

S Spindle Speed

A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded. This position is used as the "Start position".

This G code is used to perform rigid tapping for non-vertical holes. It may be used with a right-angle head to perform rigid tapping in the X or Y axis on a three-axis mill, or to perform rigid tapping along an arbitrary angle with a five-axis mill. The ratio between the feedrate and spindle speed must be precisely the thread pitch being cut.

It is not necessary to start the spindle before this canned cycle; the control does this automatically.



G187 Setting the Smoothness Level (Group 00)

G-187 is an accuracy command that can set and control both the smoothness and max corner rounding value when cutting a part. The format for using G187 is **G187 Pn Ennnn**.

P Controls the smoothness level, P1(rough), P2(medium), or P3(finish).

E Sets the max corner rounding value, temporarily overriding Setting 85.

Setting 191 sets the default smoothness to the user specified “rough,” “medium,” or “finish” when G187 is not active. The “medium” setting is the factory default setting. **NOTE:** Changing setting 191 to “Finish” will take longer to machine a part. Use this setting only when needed for the best finish.

G187 Pm Ennnn sets both the smoothness and max corner rounding value.

G187 Pm sets the smoothness but leaves max corner rounding value at its current value. **G187 Ennnn** sets the max corner rounding but leaves smoothness at its current value. **G187** by itself cancels the E value and sets smoothness to the default smoothness specified by Setting 191. G187 will be cancelled whenever “Reset” is pressed, M30 or M02 is executed, the end of program is reached, or E-stop is pressed.

G188 Get Program From PST (Group 00)

Calls the parts program for the loaded pallet based on the Pallet Schedule Table entry for the pallet.



M CODES (MISCELLANEOUS FUNCTIONS)

M Code Introduction

M-Codes are non axes-moving commands for the machine. The format for an M code is the letter "M" followed by two numbers, for example M03.

Only one M code may be programmed per line of code. All M codes take effect at the end of the block.

M00 Stop Program

The M00 code is used to stop a program. It stops the axes, spindle, turns off the coolant (including Through Spindle Coolant). The next block (block after the M00) will be highlighted when viewed in the program editor. Pressing Cycle Start will continue program operation from the highlighted block.

M01 Optional Program Stop

M01 works the same as M00, except the Optional Stop feature must be on.

M02 Program End

The M02 code is used to end a program. Note that the most common way of ending a program is with an M30.

M03 / M04 / M05 Spindle Commands

M03 turns spindle on in the forward direction

M04 turns spindle on in the reverse direction

M05 Stops the spindle

Spindle speed is controlled with an S address code, for example, S5000 will command a spindle speed of 5000RPM.

NOTE: It is not recommended to run an M04 command with Through-Spindle Coolant (TSC).

M06 Tool Change

The M06 code is used to change tools, for example M06 T12 this will put tool 12 into the spindle. If the spindle is running, the spindle and coolant (including TSC) will be stopped by the M06 command.

M07 Shower Coolant

This M code activates the optional shower coolant pump. The pump is turned off by M09, which also turns off standard coolant. The optional shower coolant is automatically turned off before a tool change or a pallet change, and it will automatically restart after a tool change if it was ON prior to a tool change sequence.



M08 Coolant On / M09 Coolant Off

The M08 code will turn on the optional coolant supply and an M09 code will turn it off. Also see M34/M35 for optional P-Cool and M88/89 for optional Through-the-spindle coolant.

NOTE: Coolant status is checked only at the start of a program, so a low coolant condition will not stop a running program.

M10 Engage 4th Axis Brake/ M11 Release 4th Axis Brake

These codes will apply and release the brake to the optional 4th axis. The brake is normally engaged, so the M10 command is only required when an M11 has been used to release the brake.

M12 Engage 5th Axis Brake / M13 Release 5th Axis Brake

These codes will apply and release the brake to the optional 5th axis. The brake is normally engaged, so the M12 command is only required when an M13 has been used to release the brake.

M16 Tool Change

This M code behaves the same as M06. However M06 is the preferred method for commanding tool changes.

M17 Unclamp APC Pallet and Open APC Door/ M18 Clamp Pallet and Close Door

This M-code is used on vertical machining centers with pallet changers. It is used as a maintenance/test function only. Pallet changes should be commanded with an M50 command only.

M19 Orient Spindle (P and R values are an optional feature)

This code is used to adjust the spindle to a fixed position. However, the spindle will only orient to the zero position unless equipped with this optional feature

The optional Orient Spindle function allows P and R address codes. For example, M19 P270 will orient the spindle to 270 degrees. The R-value allows the programmer to specify up to four decimal places, for example, M19 R123.4567.

M21-M28 Optional User M Function with M-Fin

The M codes M21 through M28 are optional for user-defined relays. Each M code activates one of the optional relays. The Reset button will stop any operation that is waiting for a relay-activated accessory to finish. Also see M51-58 and M61-68.

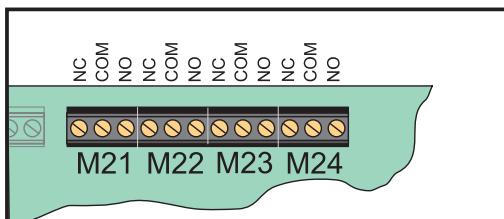
Some or all of the M21-25 (M21-M22 on Toolroom, Office and Mini mills) on the I/O PCB may be used for factory-installed options. Inspect the relays for existing wires to determine which have been used. Contact your dealer for more details.



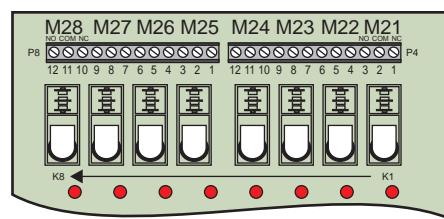
Only one relay is activated at a time. A typical operation is to command a rotary product. The sequence is: Run the machining portion of a CNC part program. Stop CNC motion and command rotary motion through the relay. Wait for a finish (stop) signal from the rotary product. Continue the CNC part program.

M-Code Relays

These outputs can be used to activate probes, auxiliary pumps or clamping devices etc. The auxiliary devices are electrically connected to the terminal strip for the individual relay. The terminal strip has a position for, Normally Open (NO), Normally Closed (NC) and Common (COM).



Main I/O PCB M-Code Relays



Optional M-Code Relay Board
(Mounted above Main I/O PCB)

Optional 8M-Code Relays

Additional M-Code relays can be purchased in banks of 8. A total of 4 banks of 8 relays are possible in the Haas system these are numbered from 0-3. Banks 0 and 1 are internal to the main I/O PCB. Bank 1 includes the M21-25 relays at the top of the IOPCB. Bank 2 addresses the first 8M option PCB. Bank 3 addresses the second 8M option PCB.

NOTE: Bank 3 may be used for some Haas-installed options and may not be available. Contact your dealer for more details.

Only one bank of outputs may be addressable with M-codes at a time. This is controlled by parameter 352 "Relay Bank Select". Relays in the non-activated banks are only accessible with macro variables or M59/69. Parameter 352 is shipped set to "1" as standard.

NOTE: With any probing option, Parameter 352 must be set to '1'. When the 8M option is installed, access its relays using M59/69

M30 Program End and Reset

The M30 code is used to stop a program. It stops the spindle and turns off the coolant (including TSC) and the program cursor will return to the start of the program. M30 cancels tool length offsets.



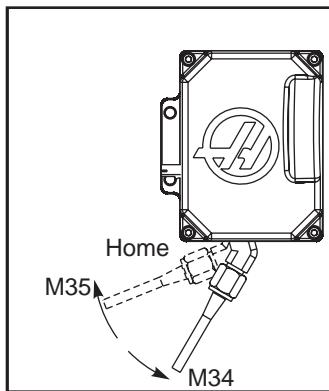
M31 Chip Conveyor Forward / M33 Chip Conveyor Stop

M31 starts the optional chip conveyor motor in the forward direction; the direction that moves the chips out of the machine. The conveyor will not turn if the door is open. It is recommended that the chip auger be used intermittently. Continuous operation will cause the motor to overheat.

Starting and stopping the chip conveyor will also run the optional conveyor washdown.

M33 Stops Conveyor motion.

M34 Coolant Increment / M35 Coolant Decrement



M34 moves the optional P-Cool spigot one position away from the current position (farther from home).

M35 moves the coolant spigot one position towards the home position.

Do not rotate the coolant spigot by hand. Serious motor damage will occur.

M36 Pallet Part Ready

Used on machines with pallet changers. This M code delays the pallet change until the Part Ready button is depressed. A pallet change will occur after the pallet ready button is pressed (and the doors are closed). For example:

Onnnnn (program number)

M36 (Flash "Part Ready" light, wait until the button is pressed)

M50 (Perform pallet change after Part Ready button is pressed)

(Part Program)

M30

M39 Rotate Tool Turret

Tool changes should be commanded using M06. M39 is not normally required but is useful for diagnostic purposes or to recover from a tool changer crash.



The M39 code is used to rotate the side mount tool changer without performing a tool change. The desired tool pocket number (Tn) must be programmed previous to the M39.

M41 / M42 Low / High Gear Override

On machines with a transmission the M41 command is used to hold the machine in low gear and an M42 will hold the machine in high gear. Normally the spindle speed (Snnn) will determine which gear the transmission should be in.

M46 Jump if Pallet Loaded

This M code causes control to be transferred to the line number specified by the P code if the pallet specified by the Q code is currently loaded.

Example: M46Qn Pnn

Jump to line nn in the current program if pallet n is loaded, otherwise go to the next block.

M48 Check Validity of Current Program

This M code is used as a safeguard for pallet changing machines. Alarm 909 (910) will display if the current program (pallet) is not listed in the Pallet Schedule Table.

M49 Set Status of Pallet

This M code sets the status of the pallet specified by the P code to the value specified by the Q code. The possible Q codes are 0-Unscheduled 1-Scheduled 2-Loaded 3-Completed 4 through 29 are user definable. The pallet status is for display purposes only. The control does not depend upon it being any particular value, but if it is 0, 1, 2 or 3, the control will update it as appropriate.

Example: M49Pnn Qmm Sets the status of pallet nn to a value of mm.

Without a P-code, this command sets the status of the currently loaded pallet.

M50 Execute Pallet Change

Used with a P value or pallet schedule table to perform a pallet change. Also see the pallet changer section.

M51-M58 Set Optional User M Codes

The M51 through M58 codes are optional for user interfaces. They will activate one of the relays and leave it active. Use M61-M68 to turn these off. The RESET key will turn off all of these relays.

See M21-M28 for details on the M-Code relays.

G & M Codes
Settings

M59 Set Output Relay

This M code turns on a relay. An example of its usage is **M59 Pnn**, where "nn" is the number of the relay being turned on. An M59 command can be used to turn on any of the discrete output relays in the range from 1100 to 1155. When using Macros, M59 P1103 does the same thing as using the optional macro command #1103=1, except that it is processed at the end of the line of code.



NOTE: 8M #1 uses addresses 1140-1147.

M61-M68 Clear Optional User M Codes

The M61 through M68 codes are optional for user interfaces. They will turn off one of the relays. Use M51-M58 to turn these on. The Reset key will turn off all of these relays. See M21-M28 for details on the M-Code relays.

M69 Clear Output Relay

This M code turns off a relay. An example of its usage is **M69 Pnn**, where "nn" is the number of the relay being turned off. An M69 command can be used to turn off any of the output relays in the range from 1100 to 1155. When using Macros, M69 P1103 does the same thing as using the optional macro command #1103=0, except that it is processed at the end of the line of code.

M75 Set G35 or G136 Reference Point

This code is used to set the reference point for G35 and G136 commands. It must be used after a probing function.

M76 Control Display Inactive / M77 Control Display Active

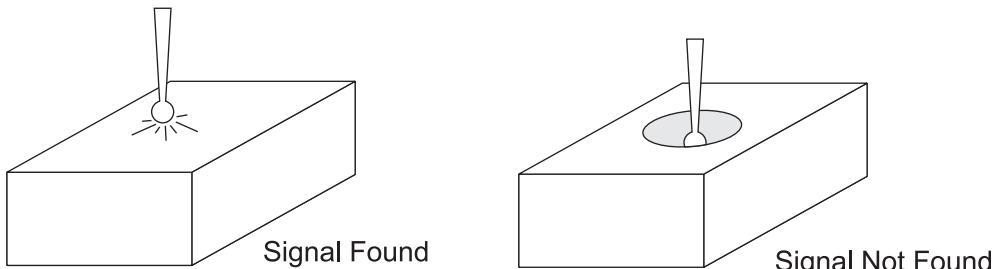
These codes are used to disable and enable the screen display. This M-Code is useful during the running of a large complicated program as refreshing the screen takes processing power that otherwise may be necessary to command the moves of the machine.

M78 Alarm if Skip Signal Found

This M-code is used with a probe. An M78 will generate an alarm if a programmed skip function (G31, G36 or G37) receives a signal from the probe. This is used when a skip signal is not expected, and may indicate a probe crash. This code can be placed on the same line as the skip G-code or in any block after.

M79 Alarm if Skip Signal Not Found

This M-code is used with a probe. An M79 will generate an alarm if a programmed skip function (G31, G36, or G37) did not receive a signal from the probe. This is used when the lack of the skip signal means a probe positioning error. This code can be placed on the same line as the skip G-code or in any block after.





M80 Auto Door Open / M81 Auto Door Close

The M80 opens the Auto Door and the M81 closes the close the Auto Door. The control pendant will beep when the door is in motion.

M82 Tool Unclamp

This code is used to release the tool from the spindle. It is used as a maintenance/test function only. Tool changes should be done using an M06.

M83 Auto Air Gun On / M84 Auto Air Gun Off

An M83 will turn the Air Gun on, and an M84 will turn it off. Additionally, an M83 Pnnn (where nnn is in milliseconds) will turn it on for the specified time, then off automatically. The Auto Air Gun is also manual toggled on and off by pressing "Shift" followed by the "Coolant" button.

M86 Tool Clamp

This code will clamp a tool into the spindle. It is used as a maintenance/test function only. Tool changes should be done using an M06.

M88 Through-Spindle Coolant On / M89 Through-Spindle Coolant Off

The M88 code is used to turn on the through-spindle coolant (TSC) option, an M89 turns the coolant off.

Proper tooling, with a through-hole, must be in place before using the TSC system. Failure to use proper tooling will flood the spindle head with coolant and void the warranty. Running an M04 (Spindle Reverse) command with TSC on is not recommended.

Sample Program

Note: The M88 command should be before the spindle speed command.

```
T1 M6; (TSC Coolant Through Drill)
G90 G54 G00 X0 Y0;
G43 H06 Z.5;
M88; (Turn TSC on)
S4400 M3;
G81 Z-2.25 F44. R.03;
M89 G80; (Turn TSC off)
G91 G28 Z0;
M30;
```

G & M Codes
Settings

M95 Sleep Mode

Sleep mode is essentially a long dwell (pause). Sleep mode can be used when the user wants the machine to begin warming itself up so it can be ready for use upon the operator's arrival. The format of the M95 command is: M95 (hh:mm)

The comment immediately following the M95 must contain the hours and minutes that the machine is to sleep for. For example, if the current time were 6 p.m. and the user wanted the machine to sleep until 6:30 a.m. the next day, the following command would be used:



M95 (12:30)

The line(s) following the M95 should be axis moves and spindle warm-up commands.

M96 Jump If No Input

P Program block to go to when conditional test is met

Q Discrete input variable to test (0 to 63)

This code is used to test a discrete input for 0 (off) status. This is useful for checking the status of automatic work holding or other accessories that will generate a signal for the control. The Q value must be in the range 0 to 63, which corresponds to the inputs found on the diagnostic display (The upper left input is 0 and the lower right is input 63. When this program block is executed and the input signal specified by Q has a value of 0, the program block Pnnnn is performed (the Pnnnn line must be in the same program)).

M96 Example:

N05 M96 P10 Q8 (Test input #8, Door Switch, until closed);

N10 (Start of program loop);

. (Program that machines part);

N85 M21 (Execute an external user function)

N90 M96 P10 Q27 (Loop to N10 if spare input [#27] is 0);

N95 M30 (If spare input is 1 then end program);

M97 Local Sub-Program Call

This code is used to call a subroutine referenced by a line number (N) within the same program. A code is required and must match a line number within the same program. This is useful for simple subroutines within a program; does not require a separate program. The subroutine must end with an M99. An Lnn code in the M97 block will repeat the subroutine call that nn times.

M97 Example:

O00011 (M97 CALL)

T1 M06

G00 G90 G54 X0 Y0 S1000 M03

G43 H01 Z1.

G01 Z0 F20.

M97 P1000 L5

(L5 will cause the program to run the N1000 line five times)

G00 G90 Z1.

M30

N1000

(N line that will run after M97 P1000 is run)

G01 G91 Z-0.1

G90 X2.

G91 Z-0.1

G90 X0

M99



M98 Sub Program Call

This code is used to call a subroutine, the format is M98 Pnnnn (Pnnnn is the number of the program being called). The subprogram must be in the program list, and it must contain an M99 to return to the main program. An Lnn count can be put on the line containing the M98 and will cause the subroutine to be called nn times before continuing to the next block.

O00012 (M98 CALL)	(Main Program number)
T1 M06	
G00 G90 G54 X0 Y0 S1000 M03	
G43 H01 Z1.	
G01 Z0 F20.	
M98 P1000 L5	(Call Sub-program, Sub-program Number, Loop 5 times)
G00 G90 Z1.	
M30	(End of program)
O01000 (M98 SUB)	(Sub Program Number)
G01 G91 Z-0.1	
G90 X2.	
G91 Z-0.1	
G90 X0	
M99	

M99 Sub-Program Return or Loop

This code is used to return to the main program from a subroutine or macro, the format is M99 Pnnnn (Pnnnn is the line in the main program to return to). It will cause the main program to loop back to the beginning without stopping when used in the main program.

Programming Notes - Fanuc behavior is simulated by using the following code:

calling program:	Haas O0001 ... N50 M98 P2 N51 M99 P100 ... N100 (continue here) ...	Fanuc O0001 ... N50 M98 P2 ... N100 (continue here) ...
subroutine:	M30 O0002 M99	M30 O0002 M99 P100

G & M Codes
Settings

M99 With Macros - If the machine is equipped with the optional macros, use a global variable and specify a block to jump to by adding **#nnn=ddd** in the subroutine and then using **M99 P#nnn** after the subroutine call.

M104 Extend Probe Arm / M105 Retract Probe Arm

Used in a program to extend or retract the probe arm.



M109 Interactive User Input

This M code allows a G-code program to place a short prompt (message) on the screen. A macro variable in the range 500 through 599 must be specified by a P code. The program can check for any character that can be entered from the keyboard by comparing with the decimal equivalent of the ASCII character (G47, Text Engraving, has a list of ASCII characters).

The following sample program will ask the user a Yes or No question, then wait for either a "Y" or an "N" to be entered. All other characters will be ignored.

```
N1 #501= 0.           (Clear the variable)
N5 M109 P501          (Sleep 1 min?)
IF [ #501 EQ 0. ] GOTO5 (Wait for a key)
IF [ #501 EQ 89. ] GOTO10 (Y)
IF [ #501 EQ 78. ] GOTO20 (N)
GOTO1                 (Keep checking)
N10                  (A Y was entered)
M95 (00:01)
GOTO30
N20                  (An N was entered)
G04 P1.               (Do nothing for 1 second)
N30                  (Stop)
M30
```

The following sample program will ask the user to select a number, then wait for a 1, 2, 3, 4 or a 5 to be entered; all other characters will be ignored.

```
%  
O01234 (M109 Program)  
N1 #501= 0 (Clear Variable #501)  
(Variable #501 will be checked)  
(Operator enters one of the following selections)  
N5 M109 P501 (1,2,3,4,5)  
IF [ #501 EQ 0 ] GOTO5  
(Wait for keyboard entry loop until entry)  
(Decimal equivalent from 49-53 represent 1-5)  
IF [ #501 EQ 49 ] GOTO10 (1 was entered go to N10)  
IF [ #501 EQ 50 ] GOTO20 (2 was entered go to N20)  
IF [ #501 EQ 51 ] GOTO30 (3 was entered go to N30)  
IF [ #501 EQ 52 ] GOTO40 (4 was entered go to N40)  
IF [ #501 EQ 53 ] GOTO50 (5 was entered go to N50)  
GOTO1 (Keep checking for user input loop until found)  
N10  
(If 1 was entered run this sub-routine)  
(Go to sleep for 10 minutes)  
#3006= 25 (Cycle start sleeps for 10 minutes)  
  
M95 (00:10)  
GOTO100  
N20  
  
(If 2 was entered run this sub routine)  
(Programmed message)
```



```
#3006= 25 (Programmed message cycle start)
GOTO100
N30
(If 3 was entered run this sub routine)
(Run sub program 20)
#3006= 25 (Cycle start program 20 will run)
G65 P20 (Call sub-program 20)
GOTO100
N40
(If 4 was entered run this sub routine)
(Run sub program 22)
#3006= 25 (Cycle start program 22 will be run)
M98 P22 (Call sub program 22)
GOTO100
N50
(If 5 was entered run this sub-routine)
(Programmed message)
#3006= 25 (Reset or cycle start will turn power off)
#1106= 1
N100
M30
%
```





SETTINGS

The setting pages contain values that control machine operation and that the user may need to change. Most settings can be changed by the operator. They are preceded by a short description on the left and the value on the right.

The on-screen settings are organized into pages of functionally similar groupings. This will make it easier for the user to remember where the settings are located and reduces the amount of time spent maneuvering through the settings display. The following list is separated into page groups with the page title as the heading.

Use the vertical cursor keys to move to the desired setting. Depending on the setting, it may be changed by entering a new number or, if the setting has specific values, press the horizontal cursor keys to display the choices. Press the Write key to enter or change the value. The message near the top of the screen displays how to change the selected setting.

The following is a detailed description of each of the settings:

1 - Auto Power Off Timer

This setting is used to automatically power-down the machine after a period of idle time. The value entered in this setting is the number of minutes the machine will remain idle until it is powered down. The machine will not be powered down while a program is running, and the time (number of minutes) will start back at zero anytime a button is pressed or the jog handle is used. The auto-off sequence gives the operator a 15-second warning before power down, at which time pressing any button will stop the power down.

2 - Power Off at M30

Powers down the machine at the end of a program (M30) if this setting is set to "On". The machine will give the operator a 30-second warning once an M30 is reached. Pressing any button will interrupt the sequence.

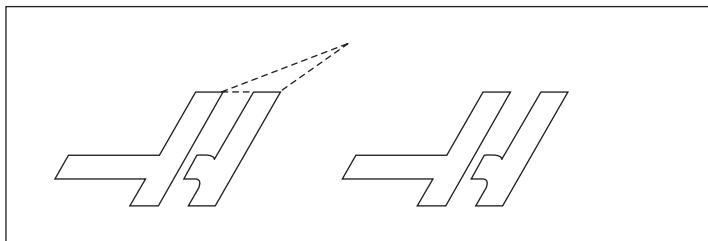
3 - 3D Graphics

3D Graphics.

4 - Graphics Rapid Path

This setting changes the way a program is viewed in the Graphics mode. When it is Off, rapid, non-cutting tool motions do not leave a path. When it is On, rapid tool motions leave a dashed line on the screen.

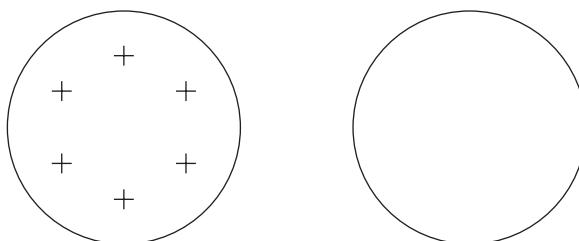
G & M Codes
Settings





5 - Graphics Drill Point

This setting changes the way a program is viewed in Graphics mode. When it is On, motion in the Z-axis will leave an X mark on the screen. When it is Off, no additional marks are shown on the graphics display.



6 - Front Panel Lock

When set to "On", this Setting disables the Spindle CW / CCW keys and ATC FWD / ATC REV keys.

7 - Parameter Lock

Turning this setting On will stop the parameters from being changed, except for parameters 81-100. Note that when the control is powered up, this setting is on.

8 - Prog Memory Lock

This setting locks out the memory editing functions (Alter, Insert, etc.) when it is set to On.

9 - Dimensioning

This setting selects between inch and metric mode. When it is set to Inch, the programmed units for X, Y, and Z are inches, to 0.0001". When it is set to Metric, programmed units are millimeters, to 0.001mm. All offset values are converted when this setting is changed from inches to metric, or vice versa. However, changing this setting will not automatically translate a program stored in memory; the programmed axis values must be changed for the new units.

When set to Inch, the default G code is G20, when set to Metric, the default G code is G21.



	INCH	METRIC
Feed	inches/min.	mm/min.
Max Travel	+/- 15400.0000	+/-39300.000
Min. Programmable Dimension	.0001	.001
Feed Range	.0001 to 300.000 in/min.	.001 to 1000.000

Axis Jog Keys		
.0001 Key	.0001 in/jog click	.001 mm/jog click
.001	.001 in/jog click	.01 mm/jog click
.01	.01 in/jog click	.1 mm/jog click
.1 Key	.1 in/jog click	1 mm/jog click

10 - Limit Rapid at 50%

Turning this setting On will limit the machine to 50% of its fastest non-cutting axis motion (rapids). This means, if the machine can position the axes at 700 inches per minute (ipm), it will be limited to 350 ipm when this setting is On. The control will display a 50% rapid override message, when this setting is on. When it is Off, the highest rapid speed of 100% is available.

11 - Baud Rate Select

This setting allows the operator to change the rate at which data is transferred to/from the first serial port (RS-232). This applies to the upload/download of programs, etc., and to DNC functions. This setting must match the transfer rate from the personal computer.

12 - Parity Select

This setting defines parity for the first serial port (RS-232). When set to none, no parity bit is added to the serial data. When set to zero, a 0 bit is added. Even and Odd work like normal parity functions. Make sure the system needs are understood, for example, XMODEM must use 8 data bits and no parity (set to "None"). This setting must match the transfer rate from the personal computer.

13 - Stop Bit

This setting designates the number of stop bits for the first serial port (RS-232). It can be 1 or 2. This setting must match the transfer rate from the personal computer.

14 - Synchronization

This changes the synchronization protocol between sender and receiver for the first serial port (RS-232). This setting must match the transfer rate from the personal computer. When set to RTS/CTS, the signal wires in the serial data cable are used to tell the sender to temporarily stop sending data while the receiver catches up. When set to XON/XOFF, the most common setting, ASCII character codes are used by the receiver to tell the sender to temporarily stop.



The selection DC Codes is like XON/XOFF, except that paper tape punch or reader start/stop codes are sent. XMODEM is a receiver-driven communications protocol that sends data in blocks of 128 bytes. XMODEM has added reliability as each block is checked for integrity. XMODEM must use 8 data bits and no parity.

15 H & T Code Agreement

Turning this setting ON has the machine check to ensure that the H offset code matches the tool inthe spindle. This check can help to prevent crashes. Note that this setting will not generate an alarm with an H00. H00 is used to cancel the tool legh offset.

Settings 16-21

These settings can be turned on in order to stop unfamiliar operators from altering the machine's functions and causing damage to the machine or work-piece.

16 - Dry Run Lock Out

The Dry Run feature will not be available when this setting is turned On.

17 - Opt Stop Lock Out

The Optional Stop feature will not be available when this setting is On.

18 - Block Delete Lock Out

The Block Delete feature will not be available when this setting is On.

19 - Feedrate Override Lock

The feedrate override buttons will be disabled when this setting is turned On.

20 - Spindle Override Lock

The spindle speed override buttons will be disabled when this setting is turned On.

21 - Rapid Override Lock

The axis rapid override buttons are disabled when this setting is turned On.

22 - Can Cycle Delta Z

This setting specifies the distance the Z-axis is retracted to clear chips during a G73 canned cycle. The range is 0.0 to 29.9999 inches (0-760 mm).

23 - 9xxx Progs Edit Lock

Turning this setting on will stop the 9000 series of programs from being viewed, edited or deleted. 9000 series programs cannot be uploaded or downloaded with this setting on. Note that 9000 series programs are usually macro programs.

24 - Leader To Punch

This setting is used to control the leader (the blank tape at the beginning of a program) sent to a paper tape punch device connected to the first RS-232 port.



25 - EOB Pattern

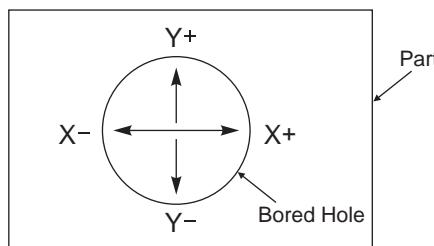
This setting controls the EOB (End of Block) pattern when data is sent and received to/from serial port 1 (RS-232). This setting must match the transfer rate from the personal computer.

26 - Serial Number

This is the serial number of the machine. It cannot be changed.

27 - G76/G77 Shift Dir.

This setting controls the direction the tool is shifted (moved) to clear a boring tool during a G76 or G77 canned cycle. Selections are X+, X-, Y+, or Y-. For more information on how this setting works see the G76 and G77 cycle in the G code section.



28 - Can Cycle Act w/o X/Z

Turning this setting On will cause the commanded canned cycle to complete without an X or Z command. The preferred method of operation is with this setting On.

When this setting is Off, the control will stop if a canned cycle is programmed without an X or Z axis move.

29 - G91 Non-modal

Turning this setting On will use the G91 command only in the program block it is in (non-modal). When it is OFF, and a G91 is commanded, the machine will use incremental moves for all axis positions.

30 - 4th Axis Enable

This setting initializes the control for a specific 4th axis. When this setting is OFF, the fourth axis is disabled; no commands can be sent to that axis. See Setting 78 for 5th axis. Note that there are two selections: "USER1" and "USER2" that can be used to set-up a unique rotary table.

31 - Reset Program Pointer

When this setting is Off, the Reset button will not change the position of the program pointer. When it is On, the Reset button will move the program pointer to the beginning of the program.



32 - Coolant Override

This setting controls how the coolant pump operates. The “Normal” selection allows the operator to turn the pump on and off manually or with M-codes. The “Off” selection will generate an alarm if an attempt is made to turn the coolant on manually or from a program. The “Ignore” selection will ignore all programmed coolant commands, but the pump can be turned on manually.

33 - Coordinate System

This setting changes the way the Haas control recognizes the work offset system when a G52 or G92 is programmed. It can be set to FANUC, HAAS, or YASNAC.

Set to YASNAC

G52 becomes another work offset; like G55.

Set to FANUC with G52

Any values in the G52 register will be added to all work offsets (global coordinate shift). This G52 value can be entered either manually or through a program. When FANUC is selected, pressing RESET, commanding an M30, or machine power down will clear out the value in G52.

Set to HAAS with G52

Any values in the G52 register will be added to all work offsets. This G52 value can be entered either manually or through a program. The G52 coordinate shift value is set to zero (zeroed) by manually entering zero, or by programming it with G52 X0, Y0, and/or Z0.

Set to YASNAC with G92:

Selecting YASNAC and programming a G92 X0 Y0, the control will enter the current machine location as a new zero point (Work Zero Offset), and that location will be entered into and viewed in the G52 list.

Set to FANUC or HAAS with G92:

Selecting FANUC or HAAS with a G92, it will work like the YASNAC setting, except that the new Work Zero location value will be loaded as new G92. This new value in the G92 list will be used, in addition to, the presently recognized work offset to define the new work zero location.

34 - 4th Axis Diameter

This is used to set the diameter of the A-axis (0.0 to 50 inches), which the control will use to determine the angular feedrate. The feedrate in a program is always inches per minute (or mm per minute), therefore, the control must know the diameter of the part being machined in the A-axis in order to compute angular feedrate. See Setting 79 for 5th axis diameter.

35 - G60 Offset

This is a numeric entry in the range 0.0 to 0.9999 inches. It is used to specify the distance an axis will travel past the target point prior to reversing. Also see G60.



36 - Program Restart

When this setting is On, restarting a program from a point other than the beginning will direct the control to scan the entire program to ensure that the tools, offsets, G and M codes, and axis positions are set correctly before the program starts at the block where the cursor is positioned. The following M codes will be processed when Setting 36 is enabled:

M08 Coolant On	M42 High Gear
M09 Coolant Off	M51-58 Set User M
M41 Low Gear	M61-68 Clear User M

When it is Off the program will start without checking the conditions of the machine. Having this setting Off may save time when running a proven program.

37 - RS-232 Data Bits

This setting is used to change the number of data bits for serial port 1 (RS-232). This setting must match the transfer rate from the personal computer. Normally 7 data bits should be used but some computers require 8. XMODEM must use 8 data bits and no parity.

38 - Aux Axis Number

This is a numeric entry between 0 and 1. It is used to select the number of external auxiliary axes added to the system. If it is set to 0, there are no auxiliary axes. If it is set to 1, there is a V-axis.

39 - Beep @ M00, M01, M02, M30

Turning this setting On will cause the keyboard beeper to sound when an M00, M01 (with Optional Stop active), M02 or an M30 is found. The beeper will continue until a button is pressed.

40 - Tool Offset Measure

This setting selects how tool size is specified for cutter compensation. Set to either Radius or Diameter.

41 - Add Spaces RS-232 Out

When this setting is On, spaces are added between address codes when a program is sent out via RS-232 serial port 1. This can make a program much easier to read/edit on a personal computer (PC). When it is set to Off, programs sent out the serial port have no spaces and are more difficult to read.

42 - M00 After Tool Change

Turning this setting On will stop the program after a tool change and a message will be displayed stating this. The Cycle Start button must be pressed to continue the program.

43 - Cutter Comp Type

This controls how the first stroke of a compensated cut begins and the way the tool is cleared from the part. The selections can be A or B; see the cutter compensation section.



44 - Min F in Radius TNC %

(Minimum feedrate in radius tool nose compensation percent) This setting affects the feed rate when cutter compensation moves the tool towards the inside of a circular cut. This type of cut will slow down to maintain a constant surface feed rate. This setting specifies the slowest feed rate as a percentage of the programmed feed rate (range 1-100).

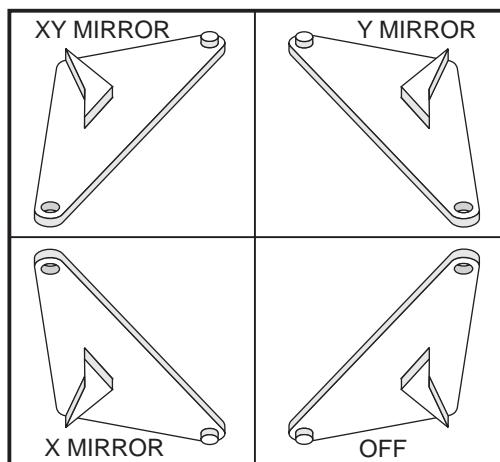
45 - Mirror Image X-axis

46 - Mirror Image Y-axis

47 - Mirror Image Z-axis

48 - Mirror Image A-axis

When one or more of these settings is On, axis motion will be mirrored (reversed) around the work zero point. See also G101, Enable Mirror Image.



49 - Skip Same Tool Change

In some program, the same tool may be called in the next section of a program or a subroutine. The control will do two changes and finish with the same tool in the spindle. Turning this setting ON will skip same tool, tool changes; a tool change will only occur if a different tool will be placed in the spindle.

50 - Aux Axis Sync

This changes the synchronization between sender and receiver for the second serial port. The second serial port is used for auxiliary axes. The settings between the CNC control and the auxiliary axes must be the same.

Selecting "RTS/CTS" will tell the sender to temporarily stop sending data while the receiver catches up.

Selecting "XON/XOFF" uses ASCII character codes from the receiver to tell the sender to temporarily stop. XON/XOFF is the most common setting.

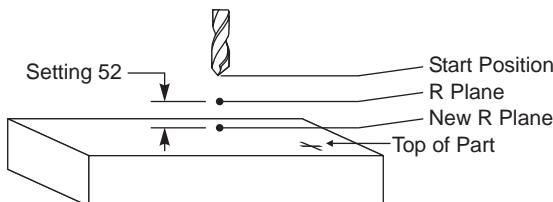
The "DC Codes" selection is like XON/XOFF, with the exception that start/stop codes are sent.



The “XMODEM” selection is receiver-driven, which sends data in blocks of 128 bytes. XMODEM gives the RS-232 communication added reliability because each block is checked for integrity.

52 - G83 Retract Above R

Range 0.0 to 30.00 inches or 0-761mm). This setting changes the way G83 (peck drilling cycle) behave. Most programmers set the reference (R) plane well above the cut to ensure that the chip clearing motion actually allows the chips to get out of the hole. However this wastes time as the machine will “drill” through this empty distance. If Setting 52 is set to the distance required to clear chips, the R plane can be put much closer to the part being drilled.



53 - Jog w/o Zero Return

Turning this setting On allows the axes to be jogged without zero returning the machine (finding machine home). This is a dangerous condition as the axis can be run into the mechanical stops and the possibly damage the machine. When the control is powered up, this setting automatically returns to Off.

54 - Aux Axis Baud Rate

This setting allows the operator to change the data rate for the second serial port (Auxiliary axis). This setting needs to match the value in the auxiliary axis control.

55 - Enable DNC from MDI

Turning this setting “On” will make the DNC feature available. DNC is selected, in the control by pressing the MDI/DNC button twice.

The DNC Direct Numeric Control feature is not available when set to “Off”.

56 - M30 Restore Default G

When this setting is On, ending a program with M30 or pressing Reset returns all modal G codes to their defaults.

57 - Exact Stop Canned X-Z

When this setting is OFF, the axes may not get to the programmed X, Y position before the Z-axis starts moving. This may cause problems with fixtures, fine part details or workpiece edges.

Turning this setting ON ensures the mill will reach the programmed X,Y position before the Z-axis moves.



58 - Cutter Compensation

This setting selects the type of cutter compensation used (FANUC or YAS-NAC). See the cutter compensation section.

59 - Probe Offset X+

60 - Probe Offset X-

61 - Probe Offset Z+

62 - Probe Offset Z-

These settings are used to define the displacement and size of the spindle probe. They specify the travel distance and direction from where the probe is triggered to where the actual sensed surface is located. These settings are used by G31, G36, G136, and M75 codes. The values entered for each setting can be either positive or negative numbers. Macros can be used to access these settings, see the Macro section for more information.

63 - Tool Probe Width

This setting is used to specify the width of the probe used to test tool diameter. This setting only applies to the probing option; it is used by G35.

64 - T. Ofs Meas Uses Work

This setting changes the way the Tool Ofset Mesur (Tool Offset Measure) button works. When this is On, the entered tool offset will be the measured tool offset plus the work coordinate offset (Z-axis). When it is Off, the tool offset equals the Z machine position.

65 - Graph Scale (Height)

This setting specifies the height of the work area that is displayed on the Graphics mode screen. The default value for this setting is the maximum height, which is the entire machine work area. Using the following formula can set a specific scale:

$$\text{Total Y travel} = \text{Parameter 20}/\text{Parameter 19}$$

$$\text{Scale} = \text{Total Y travel}/\text{Setting 65}$$

66 - Graphics X Offset

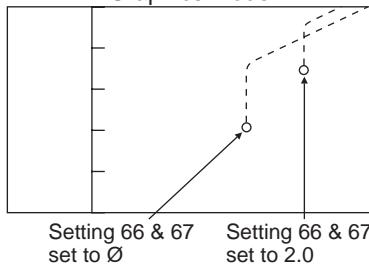
This setting locates the right side of the scaling window relative to the machine X zero position (see the Graphics section). Its default is zero.

67 - Graphics Y Offset

This setting locates the top of the zoom window relative to the machine Y zero position (see the Graphics section). Its default is zero.



Graphics Mode



68 - Graphics Z Offset

Reserved for future use.

69 - DPRNT Leading Spaces

This is an On/Off setting. When set to Off, the control will not use leading spaces generated by a macro DPRNT format statement. Conversely, when set to On, the control will use leading spaces. The following example illustrates control behavior when this setting is OFF or ON.

#1 = .0 ;	OUTPUT	
G0 G90 X#1 ;	OFF	ON
DPRNT[X#1[44]] ;	X3.0000	X 3.0000

Notice the space between the "X" and the 3 when the setting is On. Information can be easier to read when this setting is On.

70 - DPRNT Open/CLOS DCode

This setting controls whether the POPEN and PCLOS statements in macros send DC control codes to the serial port. When the setting is On, these statements will send DC control codes. When it is Off, the control codes are suppressed. Its default value is On.

71 Default G51 Scaling

This specifies the scaling for a G51 (See G-Code Section, G51) command when the P address is not used. The default is 1.000 (Range 0.001 to 8380.000).

72 Default G68 Rotation

This specifies the rotation, in degrees, for a G68 command when the R address is not used. It must be in the range 0.0000 to 360.0000°.

73 G68 Incremental Angle

This setting allows the G68 rotation angle to be changed for each commanded G68. When this switch is ON and a G68 command is executed in the Incremental mode (G91), then the value specified in the R address is added to the previous rotation angle. For example an R value of 10 will cause the rotation of the feature to be 10° the first time commanded, 20° the next time etc.



74 - 9xxx Progs Trace

This setting, along with Setting 75, is useful for debugging CNC programs. When Setting 74 is On, the control will display the code in the macro programs (O9xxxx). When the setting is Off, the control will not display the 9000 series code.

75 - 9xxxx Progs Singls BLK

When Setting 75 is On and the control is operating in Single Block mode, then the control will stop at each block of code in a macro program (O9xxxx) and wait for the operator to press Cycle Start. When Setting 75 is Off, the macro program is run continuously, the control will not pause at each block, even if Single Block is on. The default setting is On.

When Setting 74 and Setting 75 are both On, the control acts normally. That is, all blocks executed are highlighted and displayed, and when in Single Block mode there is a pause before each block is executed.

When Setting 74 and Setting 75 are both Off, the control will execute 9000 series programs without displaying the program code. If the control is in Single Block mode, no single-block pause will occur during the running of the 9000 series program.

When Setting 75 is On and Setting 74 is Off, then 9000 series programs are displayed as they are executed.

76 - Tool Release Lock Out

When this setting is ON, the tool release key on the keyboard is disabled.

77 - Scale Integer F

This setting allows the operator to select how the control interprets an F value (feedrate) that does not contain a decimal point. (It is recommended that programmers always use a decimal point.) This setting helps operators run programs developed on a control other than Haas. For Example F12 becomes:

0.0012units/minute with Setting 77 Off

12.0 units/minute with Setting 77 On

There are 5 feedrate settings:

INCH	MILLIMETER
DEFAULT(.0001)	DEFAULT (.001)
INTEGERF1 = F1	INTEGER F1 = F1
.1 F1 = F.0001	.1 F1 = F.001
.01 F10 = F.001	.01 F10 = F.01
.001 F100 = F.01	.001 F100 = F.1
.0001 F1000 = F.1	.0001 F1000 = F1



78 - 5th axis Enable

When this setting is OFF, the fifth axis is disabled and no commands can be sent to that axis. See setting 30 for 4th axis. Note that there are two selections "USER1" and "USER2" that can be used to set-up a unique rotary table.

79 - 5th-axis Diameter

This is used to set the diameter of the B-axis (0.0 to 50 inches), which the control will use to determine the angular feedrate. The feedrate in a program is always inches per minute (or mm per minute), therefore, the control must know the diameter of the part being machined in the B-axis in order to compute angular feedrate. See setting 34 for 4th axis diameter.

80 - Mirror Image B-axis

This is an On/Off setting. When it is OFF, axis motions occur normally. When it is on, B axis motion may be mirrored (or reversed) around the work zero point. Also see settings 45-48 and G101.

81 - Tool At Power Up

When the Power Up/Restart key is pressed, the control will change to the tool specified in this setting. If zero (0) is specified, no tool change occurs at power up. The default setting is 1.

Setting 81, will cause one of the following actions to occur during a Power-up/Restart:

- A. If Setting 81 is set to zero, the carousel will be rotated to pocket #1. No tool change is performed.
- B. If Setting 81 contains the tool #1, and the tool currently in the spindle is tool #1, and ZERO RET - ALL is pressed, the carousel will remain at the same pocket and no tool change will be performed.
- C. If Setting 81 contains the tool number of a tool not currently in the spindle, the carousel will be rotated to pocket #1 and then to the pocket containing the tool specified by Setting 81. A tool change will be performed to change the specified tool into the spindle.

82 - Language

Languages other than English are available in the Haas control. To change to another language, choose a language and press Enter.

83 - M30/Resets Overrides

When this setting is On, an M30 restores any overrides (feedrate, spindle, rapid) to their default values (100%).

84 - Tool Overload Action

This setting causes the specified action (Alarm, Feedhold, Beep, Autofeed) to occur anytime a tool becomes overloaded (see the Tooling section).

Choosing "Alarm" will cause the machine to stop when the tool is overloaded.



When set to "Feedhold", the message "Tool Overload" will be displayed and the machine will stop in a feedhold situation when this condition occurs. Pressing any key will clear the message.

Selecting "Beep" will cause an audible noise (beep) from the control when the tool is overloaded.

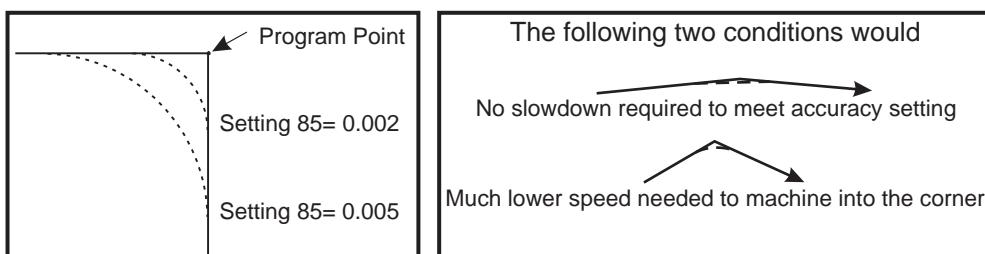
When set to "Autofeed", the lathe automatically limits the feedrate based on the tool load.

Autofeed Notes: When tapping (rigid or floating), the feed and spindle overrides will be locked out, so the Autofeed feature will be ineffective (the control will appear to respond to the override buttons, by displaying the override messages). The Autofeed feature should not be used when thread milling or auto reversing tapping heads, as it may cause unpredictable results or even a crash.

The last commanded feedrate would be restored at the end of the program execution, or when the operator presses Reset or turns off the Autofeed feature. The operator may use the keyboard feedrate override buttons while the Autofeed feature is selected. These buttons will be recognized by the Autofeed feature as the new commanded feedrate as long as the tool load limit is not exceeded. However, if the tool load limit has already been exceeded, the control will ignore the feedrate override buttons.

85 - Maximum Corner Rounding

Defines the machining accuracy of rounded corners within a selected tolerance. The initial default value is 0.05 inch. If this setting is zero (0), the control acts as if an exact stop is commanded in each motion block



86 - M39 Lockout

This is an On/Off setting. When it is ON, M39 commands are ignored.

87 - M06 Resets Override

This is an On/Off setting. When this setting is ON and an M06 is commanded, any overrides are canceled and set to their programmed values or defaults.

88 - Reset Resets Overrides

This is an On/Off setting. When it is On and the Reset key is pressed, any overrides are canceled and set to their programmed values or defaults.



90 - Max Tools To Display

This setting limits the number of tools displayed on the Tool Geometry screen. The range of this setting is 1 to 200.

91 - Advanced Jog

Turning this setting ON enables the Index Jog and Jog Travel Limits features. This setting is used on the Tool Room series of machines. See the Toolroom Mill addendum for more information on these features.

100 - Screen Saver Delay

When the setting is zero, the screen saver is disabled. If setting is set to some number of minutes, then after that time with no keyboard activity the IPS screen will be displayed. After the second screen saver delay, the Haas logo will be displayed that will change position every 2 seconds (deactivate with any key press, handle jog or alarm). The screen saver will not activate if the control is in Sleep, Jog, Edit, or Graphics mode.

101 - Feed Overide -> Rapid

Turning this setting On and pressing Handle Control Feedrate will cause the jog handle to affect both the feedrate and the rapid rate overrides. Setting 10 affects the maximum rapid rate.

103 - CYC START/FH Same Key

The Cycle Start button must be pressed and held to run a program when this setting is On. When Cycle Start is released, a feed hold is generated.

This setting cannot be turned on while Setting 104 is on. When one of them is set to On, the other will automatically turn off.

104 - Jog Handle to SNGL BLK

The jog handle can be used to single-step through a program when this setting is On. Reversing the jog handle direction will generate a feed hold.

This setting cannot be turned On while Setting 103 is on. When one of them is set to On, the other will automatically turn off.

108 - Quick Rotary G28

Turning this setting ON will return the rotary unit back to zero using the shortest distance.

For example if the rotary unit is at 10° and a zero return is commanded, the rotary table will rotate 350° if this setting is OFF. If the setting is ON the table will rotate -10°.

In order to use Setting 108, the parameter bit CIRC. WRAP. (10) must be set to 1 on parameter 43 for the A axis and Parameter 151 for the B-axis. If parameter bit(s) are not set to 1 the control will ignore setting 108.



109 - Warm-Up Time in MIN.

This is the number of minutes (up to 300 minutes from power-up) during which the compensations specified in Settings 110-112 are applied.

Overview – When the machine is powered on, if Setting 109, and at least one of Settings 110, 111, or 112, are set to a nonzero value, the following warning will be displayed:

CAUTION! Warm up Compensation is specified!

Do you wish to activate

Warm up Compensation (Y/N)?

If a 'Y' is entered, the control immediately applies the total compensation (Setting 110, 111, 112), and the compensation begins to decrease as the time elapses. For instance, after 50% of the time in Setting 109 has elapsed, the compensation distance will be 50%.

To "restart" the time period, it is necessary to power the machine off and on, and then answer "yes" to the compensation query at start-up.

CAUTION! Changing Setting 110, 111, or 112 while compensation is in progress can cause a sudden movement of up to 0.0044 inch.

The amount of remaining warmup time is displayed on the bottom right hand corner of the Diagnostics Inputs 2 screen using the standard hh:mm:ss format.

110 - Warmup X Distance

111 - Warmup Y Distance

112 - Warmup Z Distance

Settings 110, 111 and 112 specify the amount of compensation (max = $\pm 0.0020"$ or ± 0.051 mm) applied to the axes. Setting 109 must have a value entered for settings 110-112 to have an affect.

114 - Conveyor Cycle (minutes)

115 - Conveyor On-time (minutes)

These two settings control the optional chip conveyor. Setting 114 (Conveyor Cycle Time) is the interval that the conveyor will turn on automatically. Setting 115 (Conveyor On-Time) is the amount of time the conveyor will run. For example, if setting 114 is set to 30 and setting 115 is set to 2, the chip conveyor will turn on every half an hour, run for 2 minutes, then turn off.

On-time should be set no greater than 80% of cycle time. Note the following:

The CHIP FWD button (or M31) will start the conveyor in the forward direction and activate the cycle.

The CHIP STOP button (or M33) will stop the conveyor and cancel the cycle.

116 - Pivot Length

Setting 116 is set when the machine is first built and never changed. Only a qualified service technician should modify this setting.



117 - G143 Global Offset

This setting is provided for customers who have several 5-axis Haas mills and want to transfer the programs and tools from one to another. The pivot length difference (difference between Setting 116 for each machine) can be entered into this setting, and it will be applied to the G143 tool length compensation.

118 - M99 Bumps M30 CNTRS

When this setting is On, an M99 will add one to the M30 counters (these are visible on the Curnt Comnds displays). Note that an M99 will only increase the counters as it occurs in a main program, not a sub-program.

119 - Offset Lock

Turning the setting On will not allow the values in the Offset display to be altered. However, programs that alter offsets will still be able to do so.

120 - Macro Var Lock

Turning this setting On will not allow the macro variables to be altered. However, programs that alter macro variables will still be able to do so.

121 - APC Pal. One Load X

122 - APC Pal. One Load Y

123 - APC Pal. One Unload X

124 - APC Pal. One Unload Y

125 - APC Pal. Two Load X

126 - APC Pal. Two Load Y

127 - APC Pal. Two Unload X

128 - APC Pal. Two Unload Y

129 - APC Pal. 1 and 2 Safe X Pos

Settings 121-129 are the pallet positions for the X and Y axis. They are set at the time the APC is installed and should not need altering.

130 - Tap Retract Speed

This setting affects the retract speed during a tapping cycle (The mill must have the Rigid Tapping option). Entering a value, such as 2, will command the mill to retract the tap twice as fast as it went in. If the value is 3, it will retract three times as fast. A value of 0 or 1 will have no affect on the retract speed. (Range 0-4)

G & M Codes
Settings

Entering a value of 2, is the equivalent of using a J code of 2 for G84 (Tapping canned cycle). However, specifying a J code for a rigid tap will override setting 130.

131 - Auto Door

This setting supports the Auto Door option. It should be set to On for machines with an autodoor. Also see M80/81 (Autodoor Open/Close M-codes).

The door will close when Cycle Start is pressed and will open when the program reaches an M00, M01 (with Optional Stop turned on) or M30 and the spindle has stopped turning.



133 - REPT Rigid Tap

This setting ensures that the spindle is oriented so that the threads will line up when a second tapping pass, in the same hole, is programmed.

142 - Offset Chng Tolerance

This setting generates a warning message if an offset is changed by more than the amount entered for this setting. The following prompt will be displayed: "XX changes the offset by more than Setting 142! Accept (Y/N)?" if an attempt is made to change an offset by more than the entered amount (either positive or negative). If "Y" is entered, the control updates the offset as usual; otherwise, the change is rejected.

If "Y" is entered, the control updates the offset as usual, otherwise, the change is rejected.

143 Machine Data Collect

This setting enables the user to extract data from the control using one or more Q commands sent through the RS-232 port, and to set Macro variables by using an E command. This feature is software-based and requires an additional computer to request, interpret and store data from the control. A hardware option also allows the reading of machine status. See the CNC Data Transfer section for detailed information.

144 - Feed Overide->Spindles

This setting is intended to keep the chip load constant when an override is applied. When this setting is On, any feedrate override will also be applied to the spindle speed, and the spindle overrides will be disabled.

146 - APC Pallet 3 Load X

147 - APC Pallet 3 Load Y

148 - APC Pallet 3 Unload X

149 - APC Pallet 3 Unload Y

150 - APC Pallet 4 Load X

151 - APC Pallet 4 Load Y

152 - APC Pallet 4 Unload X

153 - APC Pallet 4 Unload Y

154 - APC Pallet 3 & 4 Safe X

See Settings 121-129

155 - Load Pocket Tables

This setting should only be used when a software upgrade is performed and/or memory has been cleared and/or the control is re-initialized. In order to replace the contents of the side-mount tool changer pocket tool table with the data from the file, the setting must be ON.



If this setting is OFF when loading an Offset file from a USB drive or RS-232, the contents of the Pocket Tool table will be unaltered. Setting 155 automatically defaults to OFF when the machine is turned on.

156 - Save Offset with PROG

Turning this setting On will have the control save the offsets in the same file as the programs, but under the heading O999999. The offsets will appear in the file before the final % sign.

157 - Offset Format Type

This setting controls the format in which offsets are saved with programs.

When it is set to A the format looks like what is displayed on the control, and contains decimal points and column headings. Offsets saved in this format can be more easily edited on a PC and later reloaded.

When it is set to B, each offset is saved on a separate line with an N value and a V value.

158,159,160 - XYZ Screw Thermal COMP%

These settings can be set from -30 to +30 and will adjust the existing screw thermal compensation by -30% to +30% accordingly.

162 - Default To Float

When this setting is On, the control will add a decimal point to values entered without a decimal point (for certain address codes.) When the setting is Off, values following address codes that do not include decimal points are taken as machinists notation (i.e., thousandths or ten-thousandths.) This setting will exclude the A value (tool angle) in a G76 block. Thus, the feature applies to the following address codes: X, Y, Z, A, B, C, E, F, I, J, K, U, W

A (except with G73) If a G76 A value containing a decimal point is found during program execution, alarm 605 Invalid Tool Nose Angle is generated.

D (except with G73)

R (except with G71 in YASNAC mode)

	Value entered	With Setting Off	With Setting On
In Inch mode	X-2	X-.0002	X-2.
In MM mode	X-2	X-.002	X-2.

Note that this setting affects the interpretation of all programs entered either manually or from disk or via RS-232. It does not alter the effect of setting 77 Scale Integer F.

163 - Disable .1 Jog Rate

This setting disables the highest jog rate. If the highest jog rate is selected, the next lower rate is automatically selected instead.



164 - Rotary Increment

This setting applies to the Rotary Index button on the EC300. It specifies the rotation for the rotary table in the load station. It should be set to a value from -360 to 360 degrees. For example, entering "90" will rotate the pallet 90° each time the rotary index button is pressed. If it is set to zero, the rotary table will not rotate.

167-186 Periodic Maintenance

There are 14 items that can be monitored, as well as six spare items, in the periodic maintenance settings. These settings will allow the user to change the default number of hours for each item when it is initialized during use. If the number of hours is set to zero, the item will not appear in the list of items shown in the maintenance page of current commands.

- 167 Coolant Replacement default in power-on hours
- 169 Oil Filter Replacement default in power-on hours
- 170 Gearbox Oil Replacement default in power-on hours
- 171 Coolant Tank Level Check default in power-on hours
- 172 Way Lube Level Check default in motion-time hours
- 173 Gearbox Oil Level Check default in power-on hours
- 174 Seals/Wipers Inspection default in motion-time hours
- 175 Air Supply Filter Check default in power-on hours
- 176 Hydraulic Oil Level Check default in power-on hours
- 177 Hydraulic Filter Replacement default in motion-time hours
- 178 Grease Fittings default in motion-time hours
- 179 Grease Chuck default in motion-time hours
- 180 Grease Tool Changer Cams default in tool-changes
- 181 Spare Maintenance Setting #1 default in power-on hours
- 182 Spare Maintenance Setting #2 default in power-on hours
- 183 Spare Maintenance Setting #3 default in motion-time hours
- 184 Spare Maintenance Setting #4 default in motion-time hours
- 185 Spare Maintenance Setting #5 default in tool-changes
- 186 Spare Maintenance Setting #6 default in tool-changes

187 - Machine Data Echo

This setting can be set ON or OFF. When set to ON, the data collection Q commands issued from the user's PC will be displayed on the PC screen. When set to OFF, these commands will not be displayed.

188/189/190 - G51 X/Y/Z SCALE

The axes can be scaled individually using the following new settings (must be a positive number).

- Setting 188 = G51 X SCALE
- Setting 189 = G51 Y SCALE
- Setting 190 = G51 Z SCALE

However, if setting 71 has a value, then settings 188-190 are ignored, and the value in setting 71 is used for scaling. If the value for setting 71 is zero, then



settings 188-190 are used. Note that when settings 188-190 are in effect only linear interpolation, G01, is allowed. If G02 or G03 is used, alarm 467 will be generated.

191 - Default Smoothness

This setting may be set to ROUGH, MEDIUM, or FINISH and uses parameters 302, 303, 314, 749, and 750-754 and G187 to set the smoothness and a maximum corner rounding factor. The default values are used when not overridden by a G187 command.

196 - Conveyor Shutdown

This specifies the amount of time to wait without activity prior to turning off the chip conveyor (and washdown coolant, if installed). Units are minutes.

197 - Coolant Shutdown

This specifies the amount of time to wait without activity prior to Flood, Shower, and Through-Spindle Coolant turn off in mills. Units are minutes.

198 - Background Color

Specifies the background color for inactive display panes. Range is 0 to 254.

199 - Backlight Timer

Specifies the time in minutes after which the machine display backlight will turn off when there is no input at the control (except in JOG, GRAPHICS, or SLEEP mode or when an alarm is present). Press any key to restore the screen (CAN-CEL is preferred).

201 - Show Only Work and Tool Offsets In Use

Turning this setting on will display only the Work and Tool offsets used by the running program. The program must be run in the graphics mode first to activate this feature.

216 - Servo and Hydraulic Shutoff

This setting will turn the servomotors and hydraulic pump, if equipped, off after the specified number of minutes has elapsed without activity, such as running a program, jogging, button presses, etc. The default is 0.

238 - High Intensity Light Timer (minutes)

Specifies the duration in minutes that the High Intensity Light option (HIL) remains turned on. It can be turned on if the door is opened and the work light switch is on. If this value is zero, then the light will remain turned on.

239 - Worklight Off Timer (minutes)

Specifies the duration in minutes that the internal worklight remains turned on when the worklight switch is toggled. If this value is zero, then the light will not turn off automatically.

900 - CNC Network Name

The control name you would like to show up on the network.



901 - Obtain Address Automatically

Retrieves a TCP/IP address and subnet mask from a DHCP server on a network (Requires a DHCP server). When DHCP is on, TCP/IP, SUBNET MASK and GATEWAY entries are no longer required and will have “***” entered. Also note the ADMIN section at the end to get the IP address from DHCP. The machine must be turned off and back on for changes to this setting to take effect.

NOTE: To get IP settings from DHCP: At the control, go to List Prog. Arrow down to the Hard Drive. Press the right arrow key for the Hard Drive directory. Type in ADMIN and press Insert. Select ADMIN Folder and press Write. Copy the IPConfig.txt file to disk or USB and read it on a Windows computer.

902 - IP Address

Used on a network with static TCP/IP addresses (DHCP off). The network administrator will assign an address (example 192.168.1.1). The machine must be turned off and back on for changes to this setting to take effect.

NOTE: The address format for Subnet Mask, Gateway and DNS is XXX.XXX.XXX.XXX (example 255.255.255.255) do not end the address with a period. The max address is 255.255.255.255; no negative numbers.

903 - Subnet Mask

Used on a network with static TCP/IP addresses. The network administrator will assign a mask value. The machine must be turned off and back on for changes to this setting to take effect.

904 - Gateway

Used to gain access through routers. The network administrator will assign an address. The machine must be turned off and back on for changes to this setting to take effect.

905 - DNS Server

The Domain Name Server or Domain Host Control Protocol IP address on the network. The machine must be turned off and back on for changes to this setting to take effect.

906 - Domain/Workgroup Name

Tells the network which workgroup or domain the CNC control belongs to. The machine must be turned off and back on for changes to this setting to take effect.

907 - Remote Server Name

For Haas machines with WINCE FV 12.001 or higher, enter the NETBIOS name from the computer where the share folder resides. IP address is not supported.



908 - Remote Share Path

The name of the shared network folder. After a host name is selected, to re-name the path, enter the new and press the WRITE button.

NOTE: Do not use spaces in the PATH field.

909 - User Name

This is the name used to logon to the server or domain (using a user domain account). The machine must be turned off and back on for changes to this setting to take effect. **User Names are case sensitive and cannot contain spaces.**

910 - Password

This is the password used to logon to the server. The machine must be turned off and back on for changes to this setting to take effect. **Passwords are case sensitive and cannot contain spaces.**

911 - Access to CNC Share (Off, Read, Full)

Used for the CNC hard drive read/write privileges. OFF stops the hard drive from being networked. READ allows read- only access to the hard drive. FULL allows read/write access to the drive from the network. Turning off both this setting and Setting 913 will disable network card communication.

912 - Floppy Tab Enabled

This turns access to the USB floppy drive off/on. When set to OFF, the USB floppy drive will not be accessible.

913 - Hard Drive Tab Enabled

Turns access to the hard drive off/on. When set to OFF, hard drive will not be accessible. Turning off both this setting and CNC Share (Setting 911) will disable network card communication.

914 - USB Tab Enabled

Turns access to the USB port off/on. When set to OFF the USB port will not be accessible.

915 - Net Share

Turns access to the server drive off/on. When set to OFF access to the sever from the CNC control is not possible.

916 - Second USB Tab Enabled

Turns access to the secondary USB port off/on. When set to OFF the USB port will not be accessible.





Maintenance

General Requirements

Operating Temperature Range: 41°F to 104°F (5 to 40°C)

Storage Temperature Range: -4°F to 158°F (-20 to 70°C)

Ambient Humidity: 20% – 95% relative humidity, non-condensing

Altitude: 0-7000 ft.

ELECTRICITY REQUIREMENTS

All Machines Require:

AC input power is three phase Delta or Wye power, except that the power source must be grounded (e.g. leg or center leg for delta, neutral for Wye)

Frequency range of 47-66 Hz

Line voltage that does not fluctuate more than ± 10%

Harmonic distortion not to exceed 10% of the total RMS voltage

20-15 HP System (Standard VF and 10K, EC300, EC400)

	195-260V Voltage	54-488V High-Voltage
Power Supply 1	50 AMP	25 AMP
Haas Circuit Breaker	40 AMP	20 AMP
If service run from elec. panel is less than 100' use:	8 GA. WIRE	12 GA. WIRE
If service run from elec. panel is more than 100' use:	6 GA. WIRE	10 GA. WIRE

40-30 HP System (50 Taper, 40 Taper HT 10K, VF Super Speed, EC-300, EC-400 12K, VM)

	195-260V Voltage	354-488V High-Voltage2
Power Supply1	100 AMP	50 AMP
Haas Circuit Breaker	80 AMP	40 AMP
If service run from elec. panel is less than 100' use:	4 GA. WIRE	8 GA. WIRE
If service run from elec. panel is more than 100' use:	2 GA. WIRE	6 GA. WIRE

40-30 HP System (VS 1/3, HS 3-7 incl R models)

	195-260V Voltage
Power Supply	125 AMP
Haas Circuit Breaker	100 AMP
If service run from elec. panel is less than 100' use:	2 GA. WIRE
If service run from elec. panel is more than 100' use:	0 GA. WIRE



WARNING! A separate earth ground wire of the same conductor size as the input power is required to be connected to the chassis of the machine. This ground wire is required for operator safety and for proper operation. This ground must be supplied from the main plant ground at the service entrance, and should be routed in the same conduit as the input power to the machine. A local cold water pipe or ground rod adjacent to the machine cannot be used for this purpose.

Input power to the machine must be grounded. For wye power, the neutral must be grounded. For delta power, a central leg ground or one leg ground should be used. The machine will not function properly on ungrounded power. (This is not a factor with the External 480V Option.)

The rated horsepower of the machine may not be achieved if the imbalance of the incoming voltage is beyond an acceptable limit. The machine may function properly, yet may not deliver the advertised power. This is noticed more often when using phase converters. A phase converter should only be used if all other methods cannot be used.

The maximum leg-to-leg or leg-to-ground voltage should not exceed 260 volts, or 504 volts for high-voltage machines with the Internal High Voltage Option.

1 The current requirements shown in the table reflect the circuit breaker size internal to the machine. This breaker has an extremely slow trip time. It may be necessary to size the external service breaker up by 20-25%, as indicated by "power supply", for proper operation.

2 The high-voltage requirements shown reflect the Internal 400V configuration which is standard on European machines. Domestic and all other users must use the External 480V option.

AIR REQUIREMENTS

The mill requires a minimum of 100 psi at the input to the pressure regulator on the back of the machine. A volume of 4 scfm (9scfm for EC and HS mills) is also necessary. This should be supplied by at least a two-horsepower compressor, with a minimum 20-gallon tank, that turns on when the pressure drops to 100 psi.

Note: Add 2 scfm to the minimum air requirements (below) if the operator will be using the air nozzle during pneumatic operations.

Machine Type	Main Air Regulator	Input Airline Hose Size
EC-300	85 psi	1/2" I.D.
EC-400	85 psi	1/2" I.D.
EC-1600	85 psi	1/2" I.D.
HS 3/4/6/7 incl R models	85 psi	1/2" I.D.
VF-1 - VF-11 (40Taper), VM	85 psi	3/8" I.D.
VF-5 - VF-11 (50 Taper)	85 psi	1/2" I.D.
VR Series	85 psi	1/2" I.D.
VS 1/3	85 psi	1/2" I.D.



The recommended method of attaching the air hose is to the barb fitting at the back of the machine with a hose clamp. If a quick coupler is desired, use a 1/2" coupler.

NOTE: Excessive oil and water in the air supply will cause the machine to malfunction. The air filter/regulator has an automatic bowl dump that should be empty before starting the machine. This must be checked for proper operation monthly. Also, excessive contaminants in the air line may clog the dump valve and cause oil and/or water to pass into the machine.

NOTE: Auxiliary air connections should be made on the unregulated side of the air filter/regulator.

WARNING! When the machine is operating and the pressure gauge (on the machine regulator) drops by more than 10 psi during tool changes or pallet changes, insufficient air is being supplied to the machine.

MAINTENANCE SCHEDULE

The following is a list of required regular maintenance for the machining center. These required specifications must be followed in order to keep the machine in good working order and protect your warranty.

Interval	Maintenance Performed
Daily	<ul style="list-style-type: none">• Check coolant level each eight-hour shift (especially during heavy TSC usage).• Check way lube lubrication tank level.• Clean chips from way covers and bottom pan.• Clean chips from tool changer.• Wipe spindle taper with a clean cloth rag and apply light oil.
Weekly	<ul style="list-style-type: none">• Check Through the Spindle Coolant (TSC) filters. Clean or replace if needed.• Check for proper operation of auto drain on filter regulator.• On machines with the TSC option, clean the chip basket on the coolant tank. Remove the tank cover



and remove any sediment inside the tank. Disconnect the coolant pump from the cabinet and **power off** the machine before working on the coolant tank. **Do this monthly for machines without the TSC option.**

- Check air gauge/regulator for 85 psi. Check the spindle air pressure regulator for 15 psi for vertical mills. 25 psi for horizontal mills.
- For machines with the TSC option, place a dab of grease on each tool holder pull stud. **Do this monthly for machines without the TSC option.**
- Clean exterior surfaces with mild cleaner. **DO NOT** use solvents.
- Check the hydraulic counterbalance pressure according to the machine's specifications.

Monthly

- Check oil level in gear box. **For 40 taper spindles:** Remove inspection cover beneath spindle head. Add oil slowly from top until oil begins dripping from overflow tube at bottom of sump tank. **For 50 taper spindles:** Check oil level in sight glass. Add from side of gearbox if necessary.
- Inspect way covers for proper operation and lubricate with light oil, if necessary.
- Place a dab of grease on the outside edge of the guide rails of the tool changer and run through all tools.
- Check SMTC oil level in sight glass, (see Side Mount Tool Changer Oil Level in this section).
- **EC-400** Clean the locating pads on the A-axis and the load station. This requires removing the pallet.
- For machines with umbrella-style tool changers, grease the V-flange of each tool holder.
- Check for dust buildup on electrical cabinet vector drive vents (beneath power switch). If buildup exists, open cabinet and wipe vents with a clean cloth rag. Apply compressed air as necessary to remove dust accumulation.



Six Months

- Replace coolant and thoroughly clean the coolant tank.
- Check all hoses and lubrication lines for cracking.
- Check the rotary A-axis. If necessary add oil (Mobil SHC-630). The correct oil level is halfway on the sight glass.

Annually

- Replace the gearbox oil. Drain the oil from the bottom of the gearbox. Remove inspection cover beneath spindle head. Add oil slowly from top until oil begins dripping from overflow tube at bottom of sump tank. **For 50 taper spindles**, add oil from the side of the transmission.
- Clean oil filter inside the Lube Air Panel Oil Reservoir and clean out residue at bottom of filter.
- **VR Machines** Replace A and B Axis gear oil (Mobil SHC 634).

2 years

- **EC-400** Replace the Rotary A-axis oil.
- **VR Machines** Replace the A-axis counterbalance.

PERIODIC MAINTENANCE

Press CURNT COMDS and use Page Up or Page Down to scroll to the periodic maintenance screen.

Use the up and down cursor arrow keys to highlight an item on the list, then press Origin to activate or deactivate the item. If an item is active, the remaining hours will be displayed. A deactivated item displays “—” instead.

The maintenance item time is adjusted by using the left and right arrows. Pressing the Origin key will reinstate the default time.

Items are tracked either by the time accumulated while power is on (ON-TIME) or by cycle-start time (CS-TIME). When the time reaches zero the message “Maintenance Due” is displayed at the bottom of the screen (a negative number of hours indicates the hours past due).

This message is not an alarm and does not interfere with machine operation in any way. After the necessary maintenance has been performed, select that item on the “Maintenance” screen, press the Origin button to deactivate it, then press Origin again to reactivate it with the default number of hours remaining.



Refer to settings 167-186 for additional maintenance defaults. Note that settings 181-186 are used as spare maintenance alerts by keying-in a number. The maintenance number will display on the Current Commands page once a value (time) is added to the setting.

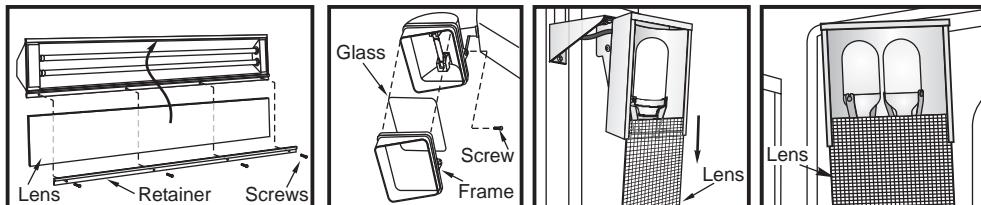
WINDOWS / GUARDING

Polycarbonate windows and guarding can be weakened by exposure to cutting liquids and chemicals that contain amines. It is possible to lose up to 10% of the remaining strength annually. If degradation is suspected, replace the window. It is recommended that the windows be replaced every two years.

Windows and guarding should be replaced if damaged or severely scratched. Replace damaged windows immediately.

WORKLIGHT

There are four types of worklights for the Haas mills. Turn off power to the machine at the main breaker before doing any work on the mill.



The power for the worklight comes from the GFI circuit. If the worklight does not turn on, check this first, it can be reset at the side of the control panel.

CHIP AUGER

During operation, most chips are collected from the machine at the discharge tube. However, very small chips may flow through the drain and collect in the coolant tank strainer. To prevent drain blockage, clean this strainer regularly. Should the drain become clogged causing coolant to collect in the pan, stop the machine, loosen the chips blocking the drain, and allow the coolant to drain. Empty the coolant tank strainer, then resume operation.



SPINDLE AIR PRESSURE

Verify Spindle air pressure using the gauge located behind the main air regulator. VF, VR, and VS mills should be set to 17 psi. EC-series and HS Series should be set to 25psi. Adjust if necessary.

12K & 15K Spindle

The air pressure for 12K &15K Spindles is 20 psi. The 12K &15K Spindles require higher pressure to slightly reduce the delivery speed and amount of oil on the bearings.

LUBRICATION CHART

System	Lubricant	Quantity
Vertical Mills		
Way lube and pneumatics	Mobilith SHC 007	2-2.5 qts
Transmission	Mobil SHC 625	40 Taper 34 oz
	Mobil SHC 625	50 Taper 51 oz
A and B axis (VR-Series)	Mobil SHC 634	A-axis 5qts, B-axis 4qts
EC-Series		
Way lube and pneumatics	Mobil Vactra #2*	2-2.5 qts
Transmission	Mobil SHC 625 (40T)	34 oz
	Mobil SHC 625 (50T)	34 oz
Rotary Table	Mobil SHC 634	Cover sight glass
HS 3/4/6/7 incl R		
Way lube and pneumatics	Mobil DTE 25	2-2.5 qts
Transmission	Mobil SHC 625	34 oz
Rotary Table	Mobil SHC 634	Cover sight glass

*All 50-Taper Machines, 30,000 RPM 40-taper spindles and 15,000 RPM spindles in GR-Series mills use SHC 625.

MINIMAL LUBRICATION SYSTEM

The Minimal Lubrication System consists of two sub-systems to optimize the amount of lubrication to the machine components. The system supplies lubrication only when it is needed; thus reducing both the amount of lubrication oil required for a machine, as well as the chance of excess oil contaminating coolant.

- (1) A grease system to lubricate the linear guides and ballscrews
- (2) An air/oil system to lubricate the spindle bearings.

The Minimal Lubrication system is located next to the control cabinet. A locked door protects the system.



Operation

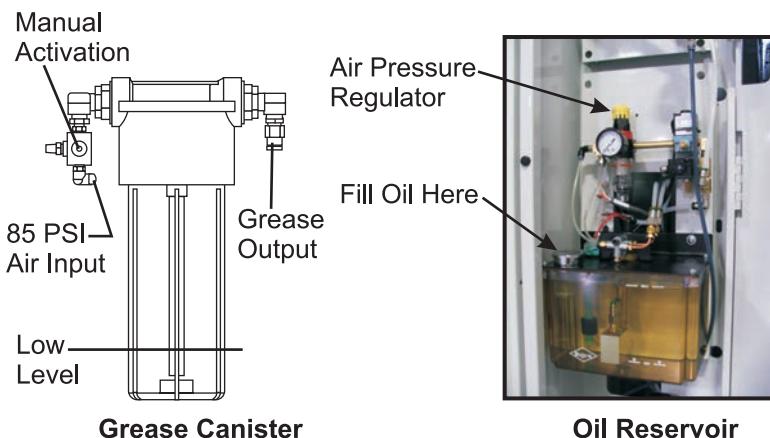
Grease System - The minimal lubrication for the linear guides and ballscrews is a grease system.

The grease system injects lubrication based on the distance of axis travel instead of time. Grease is injected once any of the axes has traveled the distance defined in parameter 811. This grease is distributed equally to each of the lubrication points for all axes.

A full grease canister should last approximately one year.

Air/Oil System - The minimal lubrication system for the spindle is an air/oil mixture. The air/oil system injects lubrication based on the number of actual revolutions of the spindle. A timed air/oil injection cycle is also used for low speed spindle operation to ensure an adequate amount of lubrication to the spindle.

A single tank of oil should last a minimum of 1 year of continuous spindle operation.



Maintenance

Grease Canister Refill:

Note: Make sure to check the grease level regularly. If the reservoir becomes completely empty, do not refill the reservoir and do not operate the machine. Contact your dealer to have the system purged and primed before you operate the machine.

Use Mobilith SHC 007 grease to refill the grease canister.

You can also order a refill bag using Haas part number 93-1933.

1. Disconnect machine air.



2. Use the provided wrench to loosen and remove the grease canister.
3. Cut a corner off of the grease refill bag to use all of the grease, or tear off the top of the bag at the perforation to use some of the grease and then reseal the bag.



Tear the bag at the perforation (resealable) or cut a corner

4. Refill the canister from the grease bag.
5. Screw the reservoir back in and tighten with the wrench. The head has a stop to prevent over-tightening.
6. Connect machine air.

If a grease system alarm occurs, take steps to resolve the problem in a reasonable amount of time. If the alarm is ignored for a long period of time, damage to the machine will result.

Filling the Oil Reservoir:

1. Clean the top of the tank.
2. Open the fill cap and pour SHC 625 oil into the reservoir until the level reaches the maximum fill line.

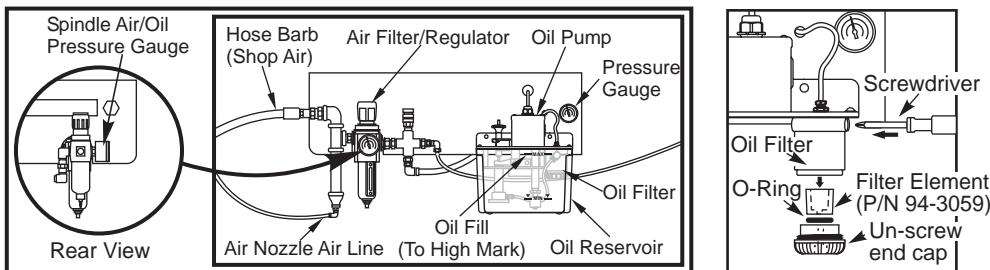
Oil system alarms: Alarm 805 is the oil system alarm. If an alarm occurs take steps to resolve the problem in a reasonable amount of time. If the alarm is ignored for a long period of time, damage to the machine will result.

Air/oil system: Validation of the oiling system: While the spindle is turning at low speed, depress the manual override button on the solenoid operated air valve for 5 seconds, then release. Oil will be seen in very small amounts at the fitting between the air mixer copper line to the air hose. It may take several seconds before being able to see traces of the oil.



AIR/LUBRICATION SYSTEM

Machine lubrication is supplied by the external lubrication system. Current lube level is visible in the reservoir; Add oil as necessary to maintain proper oil level. Warning! Do not add oil above the "high" line marked on the reservoir. Do not allow the oil level to go below the "low" line marked on the reservoir as machine damage could result.



External Lubrication System

Lube Oil Filter

The way lube oil filter element is a 25-micron porous metal filter (94-3059). It is recommended that the filter should be replaced annually or every 2000 hours of machine operation. The filter element is housed in the filter body, which is located in the oil pump reservoir (internal filters).

To change the filter element follow these steps:

1. Remove the screws that hold the oil reservoir to the pump body, carefully lower the reservoir and set aside.
2. Use a strap wrench, pipe wrench or adjustable pliers to unscrew the end cap (see the figure). **Caution:** Use a screwdriver or similar tool to stop the filter from turning while the end cap is removed.
3. Remove the oil filter element from the filter body once the end cap is removed.
4. Clean the inside of the filter housing and the filter end cap as required.
5. Install the new oil filter element (p/n 94-3059), O-ring and the end cap. Use the same tools that were used to remove the filter end cap, to tighten it - Do Not Over Tighten.
6. Replace the oil reservoir; ensure the gasket seats properly between the reservoir and the top flange.

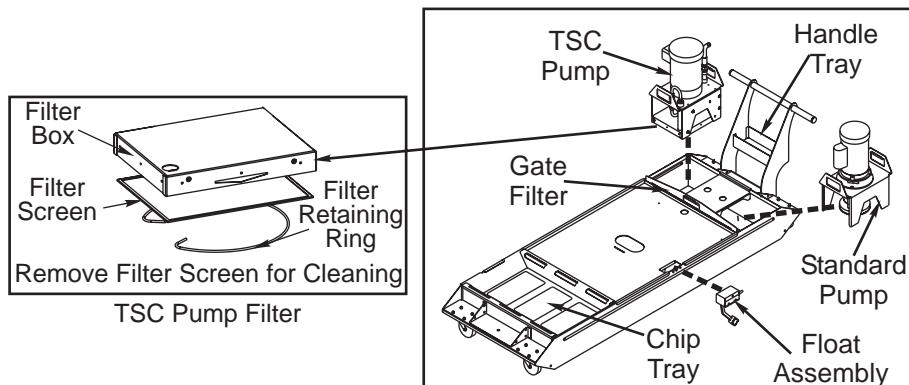


Coolant System Maintenance

Chip Tray Cleaning

The most frequent interaction with the coolant tank will be with the chip tray. Depending upon the type of material being milled, the chip tray may need to be removed and cleaned a few times each day.

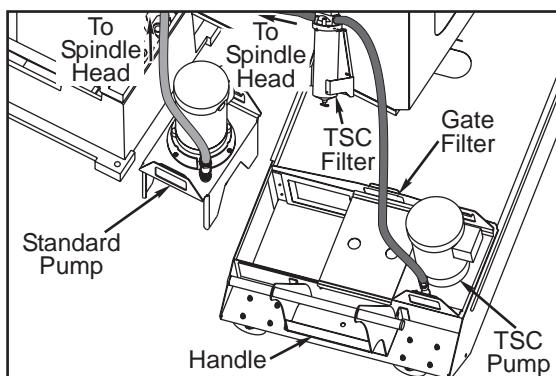
If the level sensor reads full, but the pumps begin to cavitate, the gate filter needs to be cleaned. Pull the gate filter from the tank and tap it in the chip barrel or use an air hose to remove excess chips.



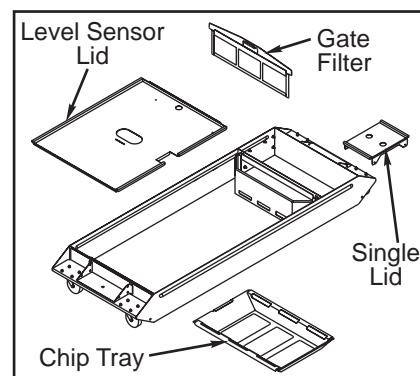
Pump removal (55 Gallon Tank Shown)

NOTE: The TSC pump has a filter, located in the bottom of the pump, that also needs weekly cleaning.

The coolant tank should be cleaned monthly (weekly for TSC). The tank may be pulled out from under the machine for better access by either lifting the pumps from the tank (grasping the pump platform by the handles and lifting) or removing power and disconnecting the hoses (keep a wrench in the handle tray) and power cables, whichever is most convenient.



Tank Movement (55 Gal. Tank shown)



*Tank Component Removal
(55 Gal. Tank shown)*



Tank component removal consists of grasping the lid and lifting it from the coolant tank. Lids are not secured to the tank.

The tank may be cleaned by using a standard shop-vac. If excessive chip build-up is present, use a scoop to remove the chips.

Coolant and Coolant Tank Considerations

As the machine runs the water will evaporate which will change the concentration of the coolant. Coolant is also carried out with the parts.

A proper coolant mixture is between 6% and 7%. To top-off coolant only more coolant or deionized water should be used. Be sure that the concentration is still within the range. A refractometer can be used to check the concentration.

Coolant should be replaced at regular intervals. A schedule should be set and held to. This will avoid a build up of machine oil and will ensure that coolant is within the proper concentration and lubricity.

Before moving the coolant tank for maintenance, lift out the coolant pump(s) and set them aside. **Do not** disconnect them from the machine, and **do not** attempt to move the tank away from the machine with the pump(s) installed and connected.

Machine coolant must be water-soluble, synthetic oil based or synthetic based coolant/lubricant. **Using mineral cutting oils will damage rubber components throughout the machine and void the warranty.**

Coolant must have rust inhibitors. Do not use pure water as a coolant; machine components will rust.

Do not use flammable liquids as coolant.

Acidic and high alkaline fluids will damage components throughout the machine.

TSC Maintenance

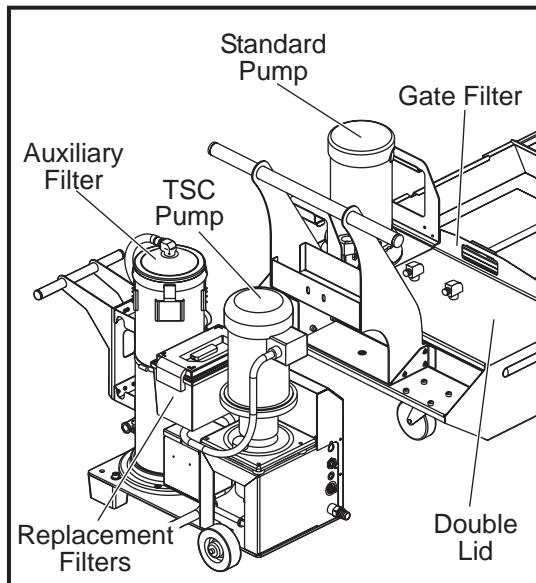
The TSC pump is a precision gear pump and will wear out faster and lose pressure if abrasive particles are present in the coolant.

- Check the TSC filter with the system running and no tool in the spindle. Change the filter when it becomes clogged.
- After changing or cleaning filter elements, run the TSC system with no tool in spindle for at least one minute to prime system.
- Coolant will be used more quickly when the TSC system is in use. Make sure to keep the coolant level up and check the level more frequently (check after every eight hour shift). **Premature wear of the pump can result from running with a low coolant level in the tank.**



TSC1000 Maintenance

Before doing any maintenance to the 1000psi system, disconnect the power source; unplug it from the power supply.



TSC1000 / HPC1000

Check the oil level on a daily basis. If the oil is low, add oil through the fill cap on the reservoir. Fill the reservoir about 25% full with 5-30W synthetic oil.

Auxiliary Filter Element Replacement

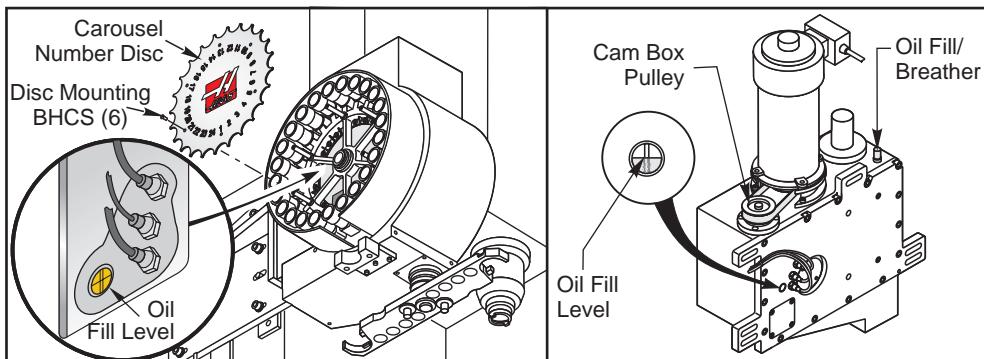
Change the filter bag when the filter gauge displays a vacuum level of -5 in. Hg or more. Do not allow the suction to exceed -10 in. Hg or pump damage may occur. Replace with a 25-micron rated filter bag (Haas P/N 93-9130).

Loosen the clamps and open the lid. Use the handle to remove the basket (the filter element will be removed with the basket). Remove the filter element from the basket and discard. Clean the basket. Install a new filter element and replace the basket (with element). Close the lid and secure the clamps.



VMC SMT/Transmission Oil

Checking the SMT gearbox oil level



Vertical Mills

Vertical Mills: View the oil level sight glass in the location shown. The proper level is half way on the sight glass. If more oil is needed, remove the bronze muffler at the top of the cam box. Pour oil in slowly to the proper level (capacity is 6 quarts), then replace the muffler.

Checking the Spindle gearbox oil level

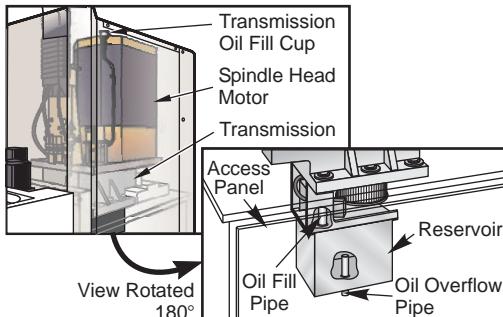
50 Taper Spindles - Remove the inspection plate from the right side of the column to access a spindle gearbox oil level dipstick.

40 Taper Spindles - There is no way to determine the spindle gearbox oil level. Drain, then fill the gearbox.

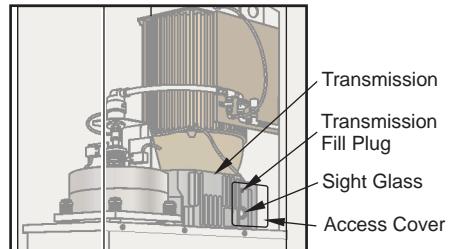
Vertical Mill 40-Taper Transmission Oil Replacement

There is no visible indicator for the level of transmission oil in the VF 1-6/40T models.

To add transmission oil, remove the access panel located directly behind the spindle head. This will expose the Transmission Oil Overflow Pipe. Place a container on the table, beneath this outlet. Manually jog the Z axis to its full -Z travel. Power down the machine. Locate the transmission oil fill cup, accessed from the top of the motor housing. There is a cut-out provided in the top of the motor housing sheetmetal for filling. Slowly pour in Mobil SHC 625 oil until oil starts to come out of the overflow pipe; this overflow indicates the reservoir is full. Close the transmission oil fill cup, wipe off the overflow pipe and replace the access cover. Consider any overflow oil to be used and dispose of properly.



VF 1-6 40 Taper



VF 6 through 11 50T

NOTE: The VF-5 50 taper does not have a sight glass; the oil is circulated directly onto the gears by a hose.

Oil Change

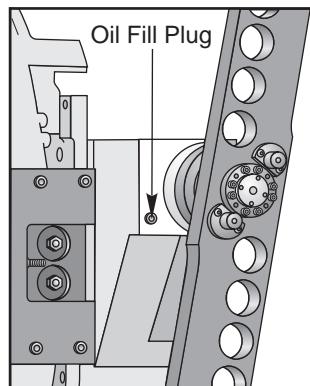
1. Remove the sheet metal from the spindle head.
2. Remove the encoder and the encoder mounting plate.
3. Remove the oil drain plug. When the oil drains, check the magnet for metal particles.
4. Replace the oil drain plug and fill the gear case with 1½ liters of Mobil SHC 625 gear oil at the fill cup on top.
5. Replace the oil overflow plug, put a small amount of thread sealant on the threads. (Do not use thread locking compound) install encoder and insure that the spindle orientation is correct.
6. Install the sheet metal and run a spindle warm up and check for leaks.



HMC SMTC/TRANSMISSION OIL

Checking the SMTC gearbox oil level

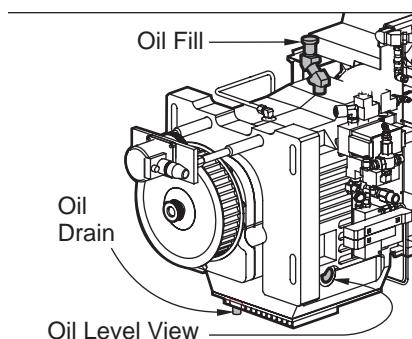
Remove the plug and feel for oil with your finger. If no oil is felt, add oil until it starts to come out of the hole (capacity is 8 quarts). Replace plug.



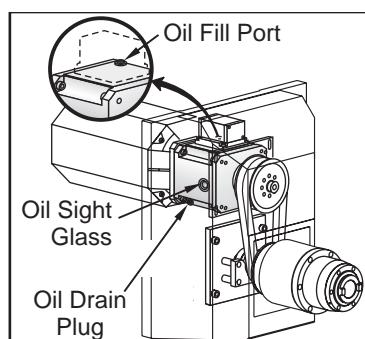
CHECKING THE SPINDLE GEARBOX OIL LEVEL

EC-300/400/500 & ES-5 Series - The smaller horizontal machines have no spindle gearbox.

EC-630/1600/2000/3000 - Remove the sheet metal necessary to gain access to the transmission. View the sight glass on the side of the transmission box as shown. The oil level should be half way on the sight glass. Fill as needed.



EC-1600



HS-3/4/6/7

Spindle Gearbox Oil Change

1. Remove the sheet metal from the spindle head.
2. Remove the drain plug as shown. Inspect the magnetic drainplug for metal particles.
3. Blow downward with an air hose in the vicinity of the fill hole to prevent dirt and metal particles from entering the gear case. Remove the fill plug.



4. Add Mobil SHC 625 gear oil until the oil level is half way up the sight glass.
5. Run a spindle warm-up and check for leaks.

HS 3/4/6/7 38-Tool Tool Changer Maintenance

Six Months

- Lubricate the Magazine Drive Gear, Tool Pot and Changer Slide Rack using red grease:
- Lubricate the Arm Shaft using Moly grease.

Annually

- Lubricate the Changer Slide Linear Guide with red grease.

Tool Pot Chain Tension

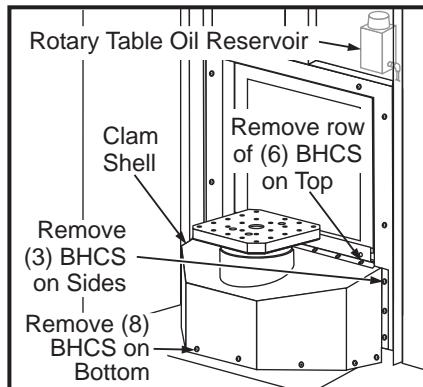
The tool pot chain tension should be checked regularly. Chain tension adjustment is performed in the lower left area of the magazine. Loosen the four M12x50 SHCS from the front of the magazine. This will allow the plate to move. Loosen the hex lock nut on the shaft and tighten the shaft using the hex bolt. Lock in the adjustment with the hex lock nut, and retighten the four 12x50 SHCS. Tensioning will not change the indexed pot location, but check the alignment between the manual tool push cylinder and the tool pot.

EC-Series Pallet Changer Rotary Table

Oil Replacement

EC-300

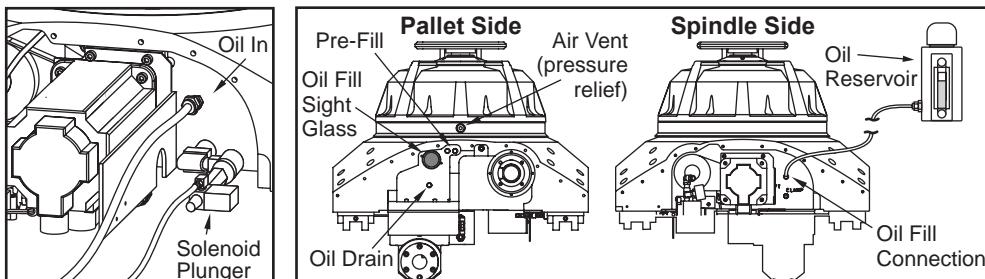
Periodically check the oil level in the reservoir and keep it filled. It is not necessary to replace the oil.



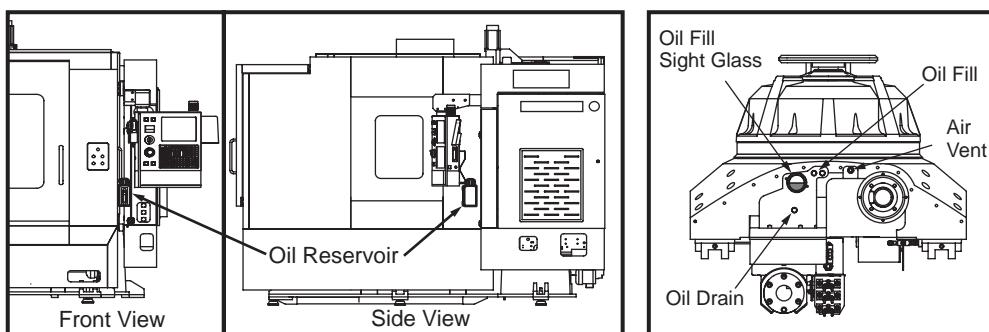
Maintenance



EC-400 Full Fourth Axis Rotary Table (Perform Every 2 years)



1. Remove the fourteen (14) BHCS on the right Z-axis way cover at the receiver end and slide it toward the column.
2. Remove the left Z-axis way cover: Jog the Z-axis all the way toward the column and rotate the H-frame 45° counter clockwise. Remove the thirteen (13) BHCS that fasten the waycover to the receiver and remove it through the door at the control pendant.
3. Disconnect the reservoir at the rotary indexer end and plug the end of the hose.
4. Remove the drain plug on the opposite side of the rotary indexer. Replug the hole when the oil has drained.
5. Remove the air escape hole plug on the side of the platter.
6. Fill the rotary table until oil begins to escape from the air escape hole and plug it.
7. Replace the reservoir hose and the way covers. Command the receiver 180° to 0° repeatedly for fifteen minutes. The reservoir will drop in level as it continues to replace the oil. Add oil as needed to the reservoir to just below the full line.



EC-400 Rotary Indexer

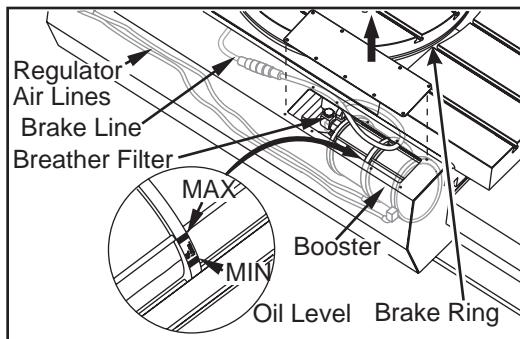


EC-400 Rotary Indexer

1. Remove the drain plug located on the left side of the rotary indexer. Replace the plug when the oil has drained.
2. Remove the air vent plug located to the upper right of the drain hole.
3. Fill the rotary indexer at the oil fill hole shown in the illustration. Replace the plug when oil begins to seep from the air vent.
4. Command a 180° to 0° rotation for fifteen minutes. This will remove air remaining in the system. The rotary indexer is full when the oil level is half way on the sight glass. Fill as needed.
5. Replace the waycover.

HYDRAULIC BRAKE (EC-1600-3000, HS3-7R)

Check the brake fluid level by viewing the fluid level in the booster. To check the EC 1600-3000 remove the brake booster cover. The cover/booster is located at the right, front of the machine. The HS 3-7R brake booster is located on the operator pendant side of the machine. Remove the way cover from the table and slide the way cover away from the table.



Adding Oil

Use only Mobil SHC 525. Remove the breather filter from the brake booster assembly and add oil. The proper oil level is between the minimum and maximum marks on the booster.



COUNTERBALANCE CHARGING

Attach the CGA 580 fitting end of the charge/discharge kit to the source pressure. Ensure T-handle of the gas chuck is turned completely counterclockwise. Attach charge/discharge kit by tightening gas chuck to the Schrader valve finger tight, then use a wrench to lightly tighten. Pressurize the system to required pressure as listed in the following tank pressure requirements chart.

NOTE: For VF-6/8, follow installation procedure for each hydraulic tank.

NOTE: Use regulated dry nitrogen gas (welding grade acceptable) that accepts a right-hand thread CGA 580 fitting. Do not use compressed air, oxygen, or flammable gas. Refer to the table below and verify pressure according to machine and spindle head position, and verify cylinder is seated in counterbore.

Machine	Tank Pressure at Top of Travel
VF-3/4	1150 psi
VF-3YT/50	1100 psi
VF-5/40	875 psi
VF-5/50	1100 psi
VF-6/7/10 50T	1150 psi
VF-8/9/11 50T	1550 psi
VR	1025 psi
VS	1250 psi
HS	1250 psi
EC-630/1600/2000/3000	800 psi

VR-SERIES

The following items must be performed in addition to the regular maintenance items.

Interval Maintenance Performed

Monthly

- Grease all pivot points on the tool changer assembly.
- Check the oil in the three (3) areas of the head. The A-axis covers need to be removed to access the filler cap and the sight glass. The B-axis filler is on the outside of the casting. Add Mobil SHC-634 to the fill port at the top of the casting.

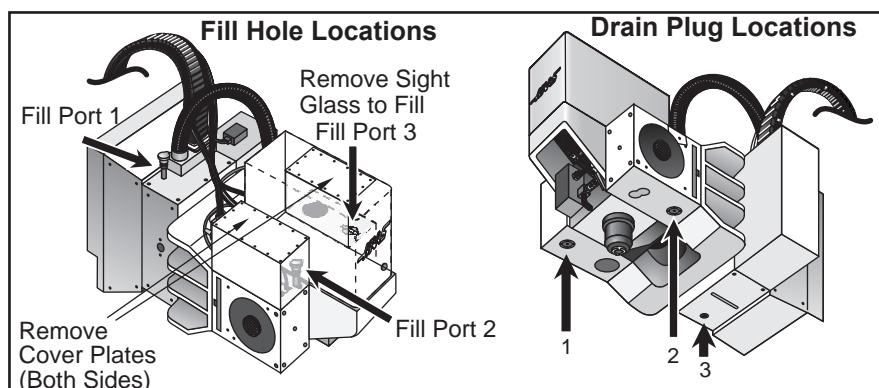


Annually

- Replace the oil in the three (3) areas of the head:

For the areas on either side of the spindle head (A-axis), remove the drain plug (4 BHCS) and drain the oil. Note: Remove the plug closest to the front on the left side of the head, and the plug towards the rear of the right side of the head. Fill the two areas with Mobil SHC-634 as described in the "Monthly" section above.

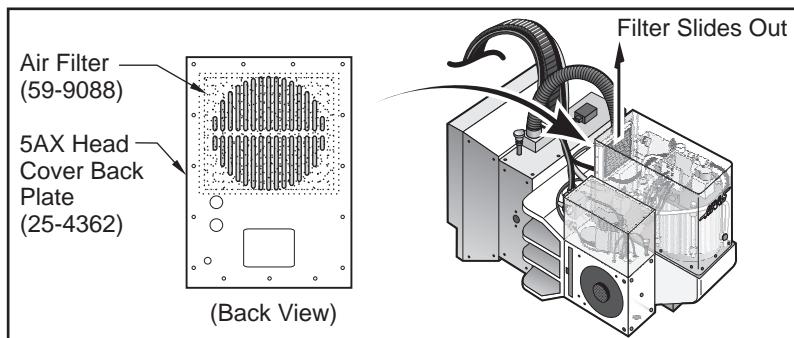
B-Axis For the area at the rear of the spindle head, remove the 1/4" NPT pipeplug with an Allen wrench and drain the oil. **Note:** The plug is near the center of this rear area. Fill with Mobil SHC-634 as described in the "Monthly" section above.



VR-SERIES AIR FILTER

The VR mills are equipped with an air filter (P/N 59-9088) for the motor housing. The recommended replacement interval is monthly, or sooner depending on the machining environment.

The air filter is located on the rear of the head cover. To remove the air filter, simply pull up on the filter; the filter will slide upward out of its bracket. To replace the filter, slide in the new air filter, oriented properly to filter air into the motor housing. Filter air-flow direction is determined by a sticker on the replacement filter.

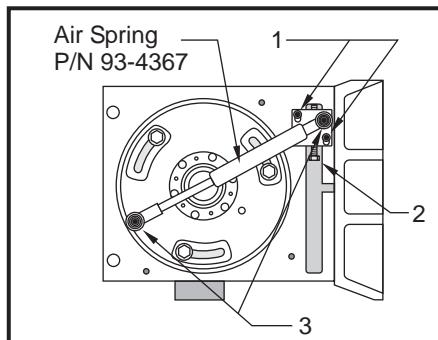


VR-11 Air Filter Location

VR-SERIES AIR SPRING COUNTERBALANCE REPLACEMENT

The counterbalance air spring and rod ends should be replaced every two (2) years.

1. Verify that the Axis is at 0 degrees before beginning. Press E-stop before doing any disassembly.
2. Remove sheet metal cover and loosen the two 3/8-16 SHCS (1).
3. Back out the 1/4-20 SHCS (2), and tighten the two 3/8-16 SHCS (1) this will keep the pre-load cam secure while the next step is accomplished.
4. Remove 3/8-16 SHCS that mount Air Spring and Rod Ends (3).
5. Tighten the rod ends onto the Air Spring and secure the Air Spring using the two 3/8-16 SHCS removed in step 4.



6. Slightly loosen the 3/8-16 SHCS (1). Screw in the 1/4-20 SHCS to force pre-load cam counterbalance down (this will push the air spring inward). Tighten this adjusting bolt until the slots in the cam have contacted the tops of the mounting bolts. Tightening the two 3/8-16 SHCS (1), these will hold the pre-load cam in place.

7. Replace the sheet metal, reset the E-stop and clear the alarms.



Index

Symbols

- 4th and 5th Axis Programming 149
- 4th-axis Operation 152
- 200 Hour Demo 35
- (VQC) Visual Quick Code 143

A

- Advanced Tool Management 79
- Air Requirements 264
- Alarm Notifications 32
- Alarms 23
- Auto Door 17, 233, 255
- AUTOFEED 252
- Auxiliary Coolant 21
- Auxiliary Coolant Filter 275

B

- Background Edit 85
 - Block Delete 26
- ## C
- Calculator 28
 - Canned Cycles 187
 - Chip Auger 21, 230
 - Chip Conveyor 230
 - Conveyor/auger 254
 - Coolant 228
 - Coolant Override 244
 - Coolant Up/Down 21
 - Current Commands 23
 - Cursor Keys 24
 - Cutter Compensation 245, 248
 - Cylindrical Mapping 204

D

- Date 28
- Deleting Programs 46
- Direct Numeric Control 51
- DIRECTORY LISTING 57
- Display Keys 23
- DNC 56
- DXF File Importer 101

E

- Electricity Requirements 263
- Emergency Stop 20
- End-Of-Block 25
- Ethernet , 36

F

- Fadal Program Converter 59
- Feedrate Override Lock 242
- Fifth Axis Installation 153
- Five-Axis Programs 149
- Fourth Axis Installation 153
- Fourth axis Operation 152
- Function Keys 21

G

- G65 Macro Call 138
- General Requirements 263
- Graphics 24, 239, 248
- Guarding 268

H

- Haas Editor (FNC) 51
- Handle Control Feedrate 22
- Handle Control Spindle 22
- Hard Drive 47
- Help 24, 28
- High Intensity Lights (HIL) 259
- High Speed Machining 36
- High-Speed SMTC 68
- HIL (High Intensity Lighting) 37
- HOME G28 27
- Hydraulic Tool Changer 73

I

- Intuitive Programming System 97

J

- Jog Handle 20
- Jog Keys 21
- Jog Lock 21
- Jog Mode 76
- Jog w/o Zero Return 247

K

- Key Switch 35

L

- Languages 251
- Loading Programs 46
- Lookahead, Macros 115
- Lubrication System 269

M

- Machine Data Collect 256
- Machine Power-Up 43
- Macros 114
- Macro Subroutine Call (G65) 138



- Maintenance 258, 263
Maintenance Schedule 265
Maximum Corner Rounding 252
M-Code Relays 229
M codes 227
Memory Lock 240
Memory Lock Key Switch 35
Messages 23
M-Fin 228
Mirror Image 201, 246
M-Net 32
- O**
Operator Load Station 88
Optional Stop 26
Option Try-Out 35
Override 22
Override Keys 21
- P**
Pallet Changer 86, 152
Pallet Changer Recovery 94
Pallet Changer Warnings 87
Pallet Schedule Table 89
Pallet Storage 95
Parameter Lock 240
Parameters 23
Parentheses () 25
P-Cool 21, 83, 230
Program Lock 240
Program Optimizer 60
Program Restart 245
Program Selection 46
Pull Studs 64
- R**
Rapid Override Lock 242
Remote Jog Handle 37
Renaming Programs 46
right slash / 25
Rigid Tapping 190, 196, 255
Rotation and Scaling 176, 179
RS-232 61, 204, 245, 256
Run-Stop-Jog-Continue 32, 85
- S**
Screen Capture 28
Screen Saver 253
Searching the program 45
Serial Number Engraving 172
- Service 263
Settings 24
Settings, Macros 115
Shift key 24
Side Mount Tool Changer 66
Side Mount Tool Changer Recovery 73
Specifications, Tool Changer 66
Spindle Air Pressure 282
Spindle Orient 228
Spindle Orientation 36
Spindle Override Lock 242
Square brackets [] 25
Sub Panel 17
Sub-Panel Controls 88
SubProgram 145, 235
Subroutine 235
System Variables 119
- T**
Thread Milling 160
Time 28
Tips and Tricks 95
Tool Changer 65
Tool Changer, High Speed Side Mount 68
Tool Changer, Hydraulic 73
Tool Changer Recovery 71, 76
SMTC Recovery Flow Chart, 73
Tool Changer, Side Mount 66
Tool Changer Specifications 66
Tool Changer, Umbrella 71
Tool Holders 64
Tooling 64
Tool Loading Flowchart 69
TOOL OFSET MESUR 248
Tool Overload 251
transmission 231
TSC (Through-Spindle Coolant) 233
- U**
Umbrella Tool Changer 71
USB 47
- W**
Warm-Up 254
Worklight 268
- Z**
Zero Return 27