



Haas Automation, Inc.

Mill Operator's Manual

96-8200
Revision D
February 2016
English
Original Instructions

To get translated versions of this Manual:

1. Go to **www.HaasCNC.com**
2. See *Owner Resources* (bottom of page)
3. Select *Manuals and Documentation*

Haas Automation Inc.
2800 Sturgis Road
Oxnard, CA 93030-8933
U.S.A. | HaasCNC.com

© 2016 Haas Automation, Inc.

All rights reserved. No part of this publication may be reproduced, stored in a retrieval system, or transmitted, in any form, or by any means, mechanical, electronic, photocopying, recording, or otherwise, without the written permission of Haas Automation, Inc. No patent liability is assumed with respect to the use of the information contained herein. Moreover, because Haas Automation strives constantly to improve its high-quality products, the information contained in this manual is subject to change without notice. We have taken every precaution in the preparation of this manual; nevertheless, Haas Automation assumes no responsibility for errors or omissions, and we assume no liability for damages resulting from the use of the information contained in this publication.

LIMITED WARRANTY CERTIFICATE

Haas Automation, Inc.

Covering Haas Automation, Inc. CNC Equipment

Effective September 1, 2010

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

Limited Warranty Coverage

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

Repair or Replacement Only

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

Disclaimer of Warranty

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

Limits and Exclusions of Warranty

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

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

Limitation of Liability and Damages

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

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

Entire Agreement

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

Transferability

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

Miscellaneous

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

Customer Feedback

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

You can find an electronic copy of this manual and other useful information on our website in the "Resource Center". Join Haas owners online and be a part of the greater CNC community at these sites:

-  diy.haascnc.com
The Haas Resource Center: Documentation and Procedures
-  atyourservice.haascnc.com
At Your Service: The Official Haas Answer and Information Blog
-  haasparts.com
Your Source for Genuine Haas Parts
-  www.facebook.com/HaasAutomationInc
Haas Automation on Facebook
-  www.twitter.com/Haas_Automation
Follow us on Twitter
-  www.linkedin.com/company/haas-automation
Haas Automation on LinkedIn
-  www.youtube.com/user/haasautomation
Product videos and information
-  www.flickr.com/photos/haasautomation
Product photos and information

Customer Satisfaction Policy

Dear Haas Customer,

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

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

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

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

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

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

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

International:

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

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

Declaration of Conformity

Product: Mill (Vertical and Horizontal)*

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

Manufactured By: Haas Automation, Inc.

2800 Sturgis Road, Oxnard, CA 93030 **805-278-1800**

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

- Machinery Directive 2006 / 42 / EC
- Electromagnetic Compatibility Directive 2014 / 30 / EU
- Additional Standards:
 - EN 60204-1:2006 / A1:2009
 - EN 614-1:2006+A1:2009
 - EN 894-1:1997+A1:2008
 - CEN 13849-1:2015

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, aluminum, and copper

Person authorized to compile technical file:

Jens Thing

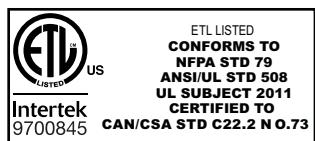
Address: Haas Automation Europe
Mercuriusstraat 28, B-1930
Zaventem, Belgium

USA: Haas Automation certifies this machine to be in compliance with the OSHA and ANSI design and manufacturing standards listed below. Operation of this machine will be compliant with the below-listed standards only as long as the owner and operator continue to follow the operation, maintenance, and training requirements of these standards.

- *OSHA 1910.212 - General Requirements for All Machines*
- *ANSI B11.5-1983 (R1994) Drilling, Milling, and Boring Machines*
- *ANSI B11.19-2003 Performance Criteria for Safeguarding*
- *ANSI B11.23-2002 Safety Requirements for Machining Centers and Automatic Numerically Controlled Milling, Drilling, and Boring Machines*
- *ANSI B11.TR3-2000 Risk Assessment and Risk Reduction - A Guideline to Estimate, Evaluate, and Reduce Risks Associated with Machine Tools*

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

Further, this document satisfies the notice-in-writing provision for exemption from Pre-Start inspection for the listed machinery as outlined in the Ontario Health and Safety Guidelines, PSR Guidelines dated April 2001. The PSR Guidelines allow that notice in writing from the original equipment manufacturer declaring conformity to applicable standards is acceptable for the exemption from Pre-Start Health and Safety Review.



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



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.

Original Instructions

How to Use This Manual

To get the maximum benefit of your new Haas machine, read this manual thoroughly and refer to it often. The content of this manual is also available on your machine control under the HELP function.

IMPORTANT: Before you operate the machine, read and understand the Operator's Manual Safety chapter.

Declaration of Warnings

Throughout this manual, important statements are set off from the main text with an icon and an associated signal word: "Danger," "Warning," "Caution," or "Note." The icon and signal word indicate the severity of the condition or situation. Be sure to read these statements and take special care to follow the instructions.

Description	Example
Danger means that there is a condition or situation that will cause death or severe injury if you do not follow the instructions given.	 DANGER: No step. Risk of electrocution, bodily injury, or machine damage. Do not climb or stand on this area.
Warning means that there is a condition or situation that will cause moderate injury if you do not follow the instructions given.	 WARNING: Never put your hands between the tool changer and the spindle head.
Caution means that minor injury or machine damage could occur if you do not follow the instructions given. You may also have to start a procedure over if you do not follow the instructions in a caution statement.	 CAUTION: Power down the machine before you do maintenance tasks.
Note means that the text gives additional information, clarification, or helpful hints .	 NOTE: Follow these guidelines if the machine is equipped with the optional extended Z-clearance table.

Text Conventions Used in this Manual

Description	Text Example
Code Block text gives program examples.	G00 G90 G54 X0. Y0.;
A Control Button Reference gives the name of a control key or button that you are to press.	Press [CYCLE START] .
A File Path describes a sequence of file system directories.	Service > <i>Documents and Software</i> >...
A Mode Reference describes a machine mode.	MDI
A Screen Element describes an object on the machine's display that you interact with.	Select the SYSTEM tab.
System Output describes text that the machine control displays in response to your actions.	PROGRAM END
User Input describes text that you should enter into the machine control.	G04 P1.;
Variable n indicates a range of non-negative integers from 0 to 9.	Dnn represents D00 through D99.

Contents

Chapter 1	Safety	1
1.1	General Safety Notes	1
1.1.1	Read Before Operating	1
1.2	Unattended Operation	3
1.3	Setup Mode	3
1.3.1	Machine Behavior with the Door Open	4
1.3.2	Robot Cells	5
1.4	Modifications to the Machine	5
1.5	Improper Coolants	5
1.6	Safety Decals	6
1.6.1	Decal Symbols Reference	7
1.7	More Information Online	10
Chapter 2	Introduction	11
2.1	Vertical Mill Orientation	11
2.2	Horizontal Mill Orientation	16
2.3	Control Pendant	24
2.3.1	Pendant Front Panel	24
2.3.2	Pendant Right Side, Top, and Bottom Panels	25
2.3.3	Keyboard	27
2.3.4	Control Display	38
2.3.5	Screen Capture	52
2.4	Tabbed Menu Basic Navigation	53
2.5	Help	53
2.5.1	The Help Tabbed Menu	54
2.5.2	Search Tab	55
2.5.3	Help Index	55
2.5.4	Drill Table Tab	55
2.5.5	Calculator Tab	55
2.6	More Information Online	62
Chapter 3	Control Icons	63
3.1	Introduction	63
3.2	Control Icon Guide	64
3.3	More Information Online	72

Chapter 4 Operation	73
4.1 Machine Power-On	73
4.2 Spindle Warm-Up	73
4.3 Device Manager	74
4.3.1 File Directory Systems.	75
4.3.2 Program Selection	76
4.3.3 Program Transfer	76
4.3.4 Deleting Programs.	77
4.3.5 Maximum Number of Programs	77
4.3.6 File Duplication	78
4.3.7 Changing Program Numbers	78
4.4 Backing Up Your Machine.	79
4.4.1 Making a Backup	79
4.4.2 Restoring From a Backup	80
4.5 Basic Program Search.	81
4.6 RS-232	82
4.6.1 Cable Length	82
4.6.2 Machine Data Collection.	82
4.7 File Numerical Control (FNC)	85
4.8 Direct Numerical Control (DNC)	85
4.8.1 DNC Notes.	87
4.9 Tooling.	87
4.9.1 Tool Holders	87
4.9.2 Advanced Tool Management Introduction.	88
4.10 Tool Changers	93
4.10.1 Loading the Tool Changer	94
4.10.2 Umbrella Tool Changer Recovery	98
4.10.3 SMTC Programming Notes	99
4.10.4 SMTC Recovery	99
4.10.5 SMTC Door Switch Panel	100
4.11 Part Setup	101
4.11.1 Setting Offsets	101
4.12 Features	104
4.12.1 Graphics Mode	104
4.12.2 Dry Run Operation.	106
4.12.3 Axis Overload Timer	106
4.13 Running Programs	106
4.14 Run-Stop-Jog-Continue	106
4.15 More Information Online	108
Chapter 5 Programming	109
5.1 Numbered Programs	109
5.2 Program Editors	109

5.2.1	Basic Program Editing	109
5.2.2	Background Edit	111
5.2.3	Manual Data Input (MDI).	111
5.2.4	Advanced Editor	112
5.2.5	The File Numerical Control (FNC) Editor	121
5.3	Fadal Program Converter	132
5.4	Program Optimizer	133
5.4.1	Program Optimizer Operation	133
5.5	DXF File Importer	134
5.5.1	Part Origin	135
5.5.2	Part Geometry Chain and Group	135
5.5.3	Toolpath Selection	136
5.6	Basic Programming	137
5.6.1	Preparation	138
5.6.2	Cutting	139
5.6.3	Completion.	139
5.6.4	Absolute vs. Incremental Positioning (G90, G91)	140
5.7	Tool and Work Offset Calls	144
5.7.1	G43 Tool Offset	144
5.7.2	G54 Work Offsets	144
5.8	Miscellaneous Codes	145
5.8.1	Tool Functions (Tnn).	145
5.8.2	Spindle Commands	146
5.8.3	Program Stop Commands	146
5.8.4	Coolant Commands	146
5.9	Cutting G-codes	147
5.9.1	Linear Interpolation Motion	147
5.9.2	Circular Interpolation Motion.	147
5.10	Cutter Compensation	149
5.10.1	General Description of Cutter Compensation	149
5.10.2	Entry and Exit from Cutter Compensation	152
5.10.3	Feed Adjustments in Cutter Compensation	153
5.10.4	Circular Interpolation and Cutter Compensation.	155
5.11	Canned Cycles	158
5.11.1	Drilling Canned Cycles.	158
5.11.2	Tapping Canned Cycles	158
5.11.3	Boring and Reaming Cycles	159
5.11.4	R Planes	159
5.12	Special G-codes	159
5.12.1	Engraving	160
5.12.2	Pocket Milling	160
5.12.3	Rotation and Scaling.	160
5.12.4	Mirror Image	160

5.13	Subprograms	161
5.13.1	External Subprogram (M98)	161
5.13.2	Local Subprogram (M97)	164
5.13.3	External Subprogram Canned Cycle Example (M98) . .	165
5.13.4	External Subprograms With Multiple Fixtures (M98) . .	167
5.14	More Information Online	168
Chapter 6	Options Programming	169
6.1	Introduction	169
6.2	4th and 5th Axis Programming	169
6.2.1	Creating Five-Axis Programs	169
6.2.2	Installing an Optional 4th Axis	173
6.2.3	Installing an Optional 5th Axis	175
6.2.4	Tilt Axis Center-of-Rotation Offset (Tilting Rotary Products)	
	175	
6.2.5	Disabling 4th and 5th Axes	177
6.3	Macros (Optional)	178
6.3.1	Macros Introduction	178
6.3.2	Operation Notes	181
6.3.3	System Variables In-Depth	194
6.3.4	Variable Usage	202
6.3.5	Address Substitution	203
6.3.6	G65 Macro Subprogram Call Option (Group 00)	214
6.3.7	Communication With External Devices - DPRNT[] . .	216
6.3.8	Fanuc-Style Macros Not Included	218
6.4	More Information Online	219
Chapter 7	G-codes	221
7.1	Introduction	221
7.1.1	List of G-codes	221
7.2	More Information Online	330
Chapter 8	M-codes	331
8.1	Introduction	331
8.1.1	List of M-codes	331
8.2	More Information Online	349
Chapter 9	Settings	351
9.1	Introduction	351
9.1.1	List of Settings	351
9.2	More Information Online	387

Chapter 10 Maintenance	389
10.1 Introduction	389
10.2 Maintenance Monitor	389
10.2.1 Maintenance Settings	389
10.2.2 The Maintenance Monitor Page	390
10.2.3 Start, Stop, or Adjust Maintenance Monitoring	391
10.3 More Information Online.	392
Chapter 11 Other Equipment	393
11.1 Introduction	393
11.2 Mini Mills.	393
11.3 VF-Trunnion Series	393
11.4 Gantry Routers	393
11.5 Office Mill	393
11.6 EC-400 Pallet Pool	393
11.7 UMC-750	393
11.8 More Information Online.	394
Index	395

Chapter 1: Safety

1.1 General Safety Notes



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

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



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

All CNC machines present hazards from rotating cutting tools, belts and pulleys, high voltage electricity, noise, and compressed air. When you use CNC machines and their components, you must always follow basic safety precautions to reduce the risk of personal injury and mechanical damage.

1.1.1 Read Before Operating



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

Basic safety:

- This machine can cause severe bodily injury.
- This machine is automatically controlled and may start at any time.
- Consult your local safety codes and regulations before you operate the machine. Contact your dealer if you have questions about safety issues.

General Safety Notes

- It is the machine owner's responsibility to make sure that everyone who is involved in installing and operating the machine is fully acquainted with the operation and safety instructions provided with the machine, BEFORE they work with the machine. The ultimate responsibility for safety rests with the machine owner and the individuals who work with the machine.
- Use appropriate eye and ear protection when you operate the machine.
- Replace windows immediately if they are damaged or severely scratched.
- Keep the side windows locked during operation (if available).

Electrical safety:

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

Operation Safety:

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

Follow these guidelines when you work with the machine:

- Normal operation - Keep the door closed and guards in place (for non-enclosed machines) while the machine operates.
- Part loading and unloading – An operator opens the door, completes the task, closes the door, and then presses **[CYCLE START]** (starting automatic motion).

- Machining job set-up – Press [**EMERGENCY STOP**] before you add or remove machine fixtures.
- Maintenance / Machine Cleaner– Press [**EMERGENCY STOP**] or [**POWER OFF**] on the machine before you enter the enclosure.

1.2 Unattended Operation

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

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

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

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

1.3 Setup Mode

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

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

When unlocked, setup mode allows a skilled machinist more access to the machine to set up jobs. In this mode, machine behavior is dependent on whether the doors are opened or closed. Opening the doors when the machine is in cycle stops motion and reduces spindle speed. The machine allows several functions in setup mode with the doors opened, usually at reduced speed. The following charts summarize the modes and allowed functions.



DANGER:

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

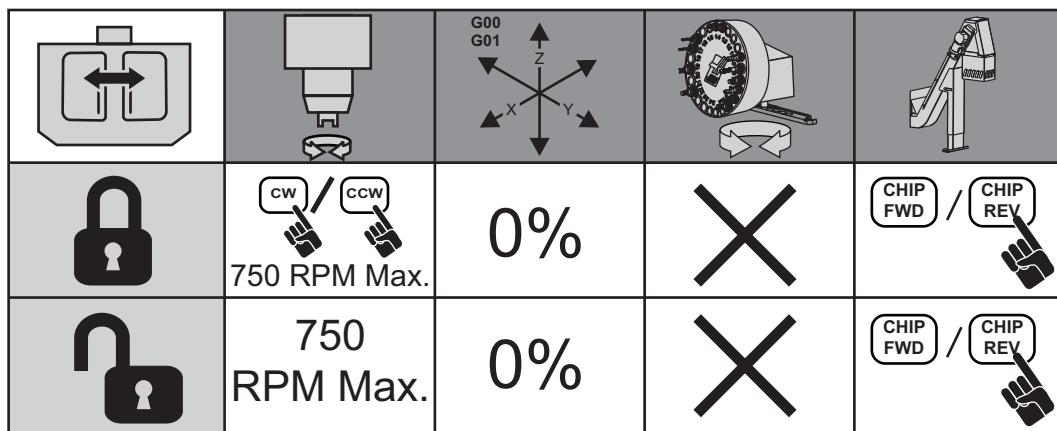
Setup Mode

1.3.1 Machine Behavior with the Door Open

For safety, machine operations stop when the door is open and the setup keyswitch is locked. The unlock position allows limited machine functions with the door open.

T1.1: Setup / Run Mode Limited Overrides with the Machine Doors Open

Machine Function	Keyswitch Locked (Run Mode)	Keyswitch Unlocked (Setup Mode)
Maximum Rapid	Not allowed.	Not allowed.
Cycle Start	Not allowed. No machine motion or program execution.	Not allowed. No machine motion or program execution.
Spindle [CW] / [CCW]	Allowed, but you must press and hold [CW] or [CCW]. Maximum 750 RPM.	Allowed, but maximum 750 RPM.
Tool Change	Not allowed.	Not allowed.
Next Tool	Not allowed.	Not allowed.
Opening the doors while a program runs	Not allowed. The door is locked.	Allowed, but axis motion stops and the spindle slows to a maximum of 750 RPM. The doors lock during tool changes and some canned cycles.
Conveyor motion	Allowed, but you must press and hold [CHIP REV] to run in reverse.	Allowed, but you must press and hold [CHIP REV] to run in reverse.



1.3.2 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 allowed only while a robot communicates with the CNC machine. Typically, an interface between the robot and the CNC machine addresses the safety of both machines.

Robot cell setup is beyond the scope of this manual. Work with a robot-cell integrator and your HFO to correctly set up a safe robot cell.

1.4 Modifications to the Machine

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

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

1.5 Improper Coolants

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

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

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

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

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

Safety Decals

Ask your HFO or your coolant dealer if you have questions about the specific coolant that you plan to use. The Haas Resource Center website has videos and other general information about coolant use and maintenance. You can also scan the code below with your mobile device to directly access this information.



1.6 Safety Decals

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



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

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

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

F1.1: Example Safety Decal Symbols: [1] Hazard Description, [2] Prohibited Action, [3] Recommended Action.



1.6.1 Decal Symbols Reference

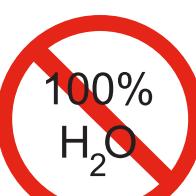
This section gives explanations and clarifications for the safety symbols you will see on your machine.

T1.2: Hazard Symbols – Yellow Triangles

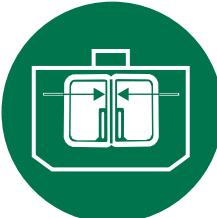
Symbol	Description
	<p>Moving parts can entangle, trap, crush, and cut. Keep all parts of your body away from machine parts when they move, or whenever motion is possible. Motion is possible when the power is on and the machine is not in [EMERGENCY STOP]. Secure loose clothing, hair, etc. Remember that automatically controlled devices can start at any time.</p>
	<p>Do not touch rotating tools. Keep all parts of your body away from machine parts when they move, or whenever motion is possible. Motion is possible when the power is on and the machine is not in [EMERGENCY STOP]. Sharp tools and chips can easily cut skin.</p>
	<p>Long tools are dangerous, especially at spindle speeds higher than 5000 RPM. The tools can break and eject from the machine. Remember that machine enclosures are intended to stop coolant and chips. Enclosures may not stop broken tools or thrown parts. Always check your setup and tooling before you start machining.</p>
	<p>Materials can create hazardous dust or fumes during machining. The machine enclosure alone is not designed to contain dust or fumes. Many materials are harmful, especially when airborne. This can include, but is not limited to: coolant mist, fine particles, fumes, and chips. When necessary, use devices such as breathing apparatus and dust/fume removal systems. Read and understand the Safety Data Sheet (SDS) for the materials, and follow the safety recommendations.</p>

Safety Decals

T1.3: Prohibited Action Symbols – Red Circles with Slash-Through

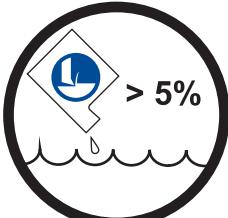
Symbol	Description
	<p>Do not enter the machine enclosure when the machine is capable of automatic motion.</p> <p>When you must enter the enclosure to complete tasks, press [EMERGENCY STOP] or power off the machine. Put a safety tag on the control pendant to alert other people that you are inside the machine, and that they must not turn on or operate the machine.</p>
	<p>Do not machine ceramics.</p>
	<p>Do not attempt to load tools with the spindle dogs misaligned with the cutouts in the toolholder V-Flange.</p>
	<p>Do not machine flammable materials.</p> <p>Do not use flammable coolants.</p> <p>Flammable materials in particulate or vapor form can become explosive.</p> <p>The machine enclosure is not designed to contain explosions or extinguish fire.</p>
	<p>Do not use pure water as coolant. This will cause machine components to rust.</p> <p>Always use a rust-inhibitive coolant concentrate with water.</p>

T1.4: Recommended Action Symbols – Green Circles

Symbol	Description
	Keep the machine doors closed.
	Always wear safety glasses or goggles when you are near a machine. Airborne debris can cause eye damage.
	Make sure the spindle dogs are correctly aligned with the cutouts in the toolholder V-flange.
	Note the location of the tool release button. Press this button only when you are holding the tool. Some tools are very heavy. Handle these tools carefully; use both hands and have someone press the tool release button for you.

More Information Online

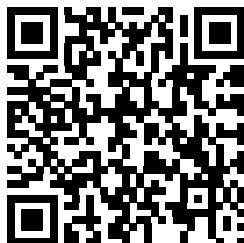
T1.5: Informational Symbols – Black Circles

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

1.7 More Information Online

For updated and supplemental information, including tips, tricks, maintenance procedures, and more, go to DIY.HaasCNC.com.

You can also scan this code with your mobile device to directly access the “Best Practices” page on the Resource Center, which includes information about safety.

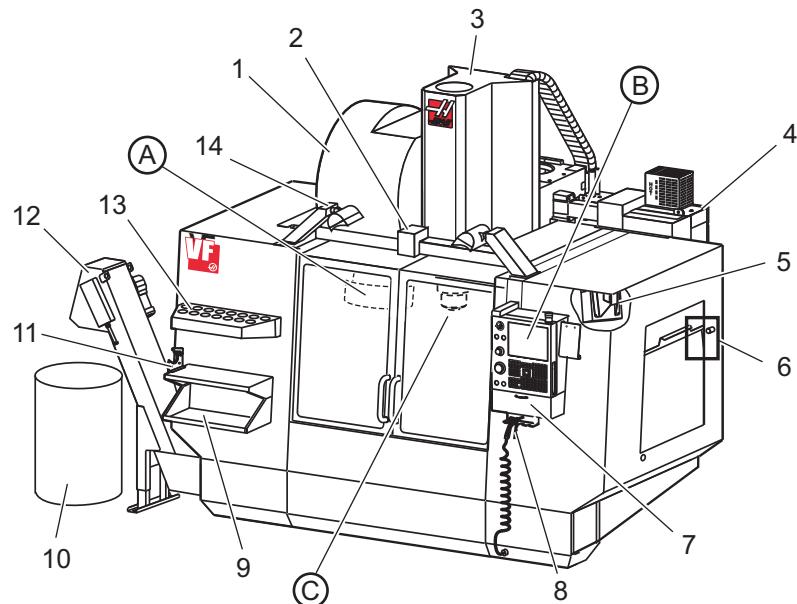


Chapter 2: Introduction

2.1 Vertical Mill Orientation

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

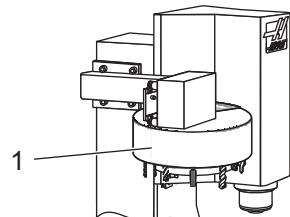
F2.1: Vertical Mill Features (front view)



- | | |
|---|--------------------------------------|
| 1. Side Mount Tool Changer (optional) | A. Umbrella Tool Changer (not shown) |
| 2. Servo Auto Door (optional) | B. Control Pendant |
| 3. Spindle Assembly | C. Spindle Head Assembly |
| 4. Electrical Control Box | |
| 5. Work Light (2X) | |
| 6. Window Controls | |
| 7. Storage Tray | |
| 8. Air Gun | |
| 9. Front Work Table | |
| 10. Chip Container | |
| 11. Tool Holding Vise | |
| 12. Chip Conveyor (optional) | |
| 13. Tool Tray | |
| 14. High Intensity Lights (2X) (optional) | |

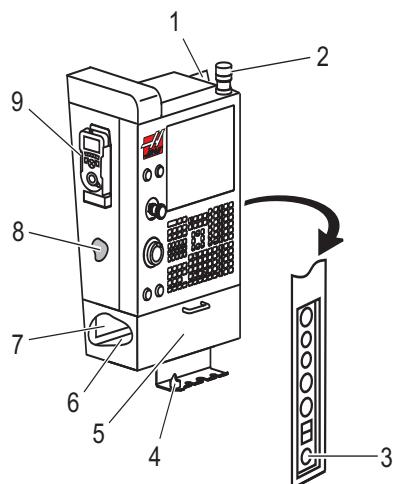
Vertical Mill Orientation

F2.2: Detail A



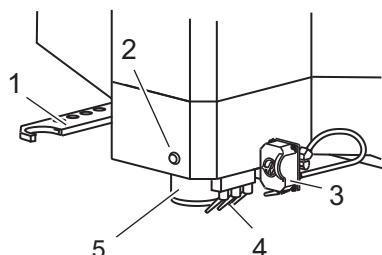
1. Umbrella-Style Tool Changer

F2.3: Detail B



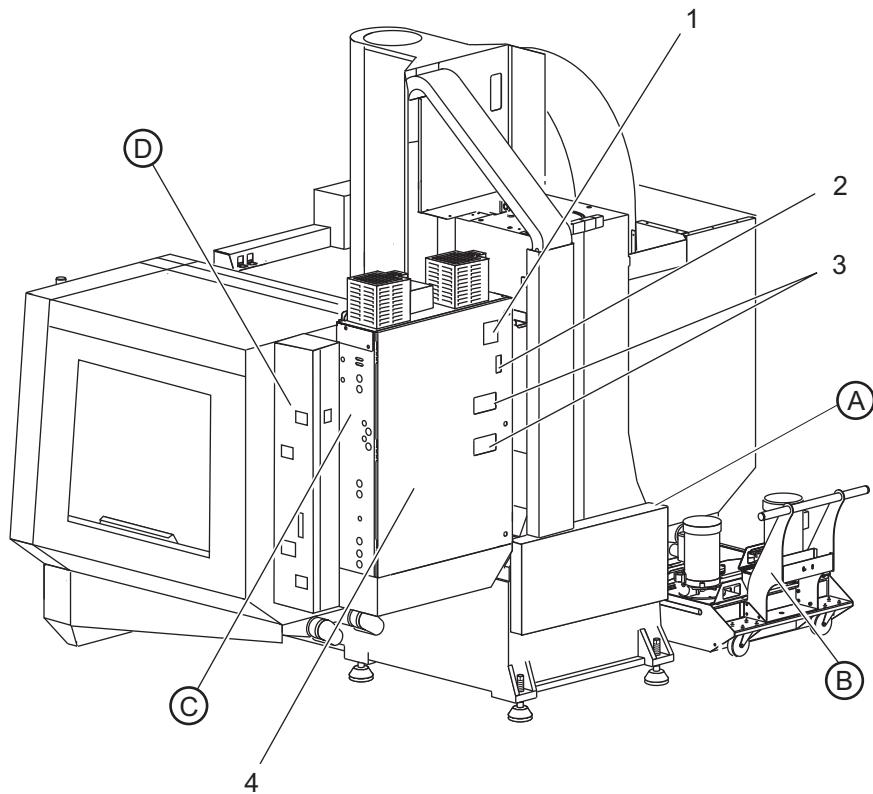
1. Clipboard
2. Work Beacon
3. Hold to Run (where equipped)
4. Vise Handle Holder
5. Tool Tray
6. G- and M-code Reference List
7. Assembly Data (stored inside)
8. Operator's Manual (stored behind console)
9. Remote Jog Handle

F2.4: Detail C



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

F2.5: Vertical Mill Features (rear view)

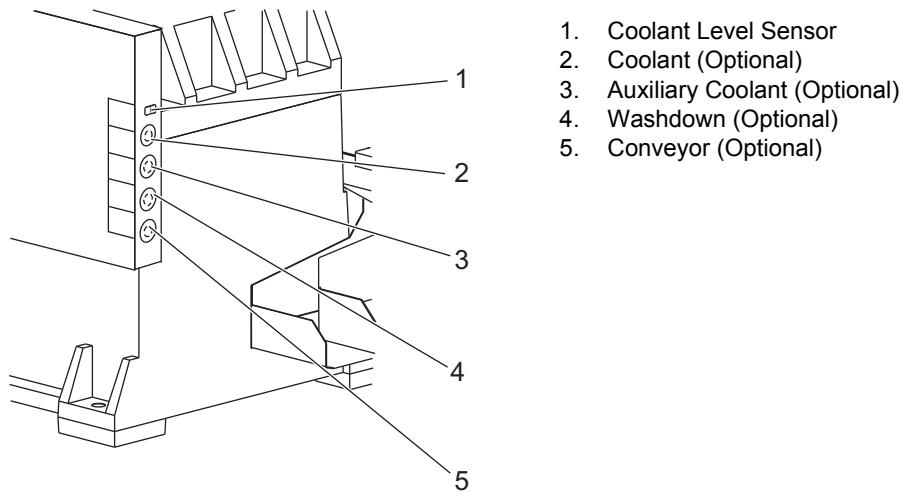


1. Data Plate
2. Main Circuit Breaker Switch
3. Vector Drive Fan (runs intermittently)
4. Control Cabinet
5. Smart Lube Panel Assembly

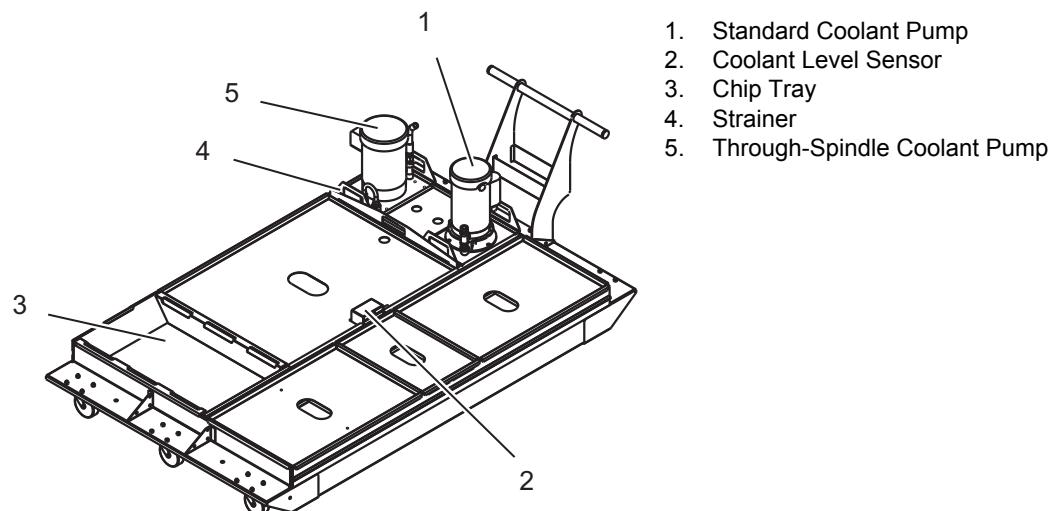
A Electrical Connectors
B Coolant Tank Assembly (movable)
C Electrical Control Cabinet Side Panel

Vertical Mill Orientation

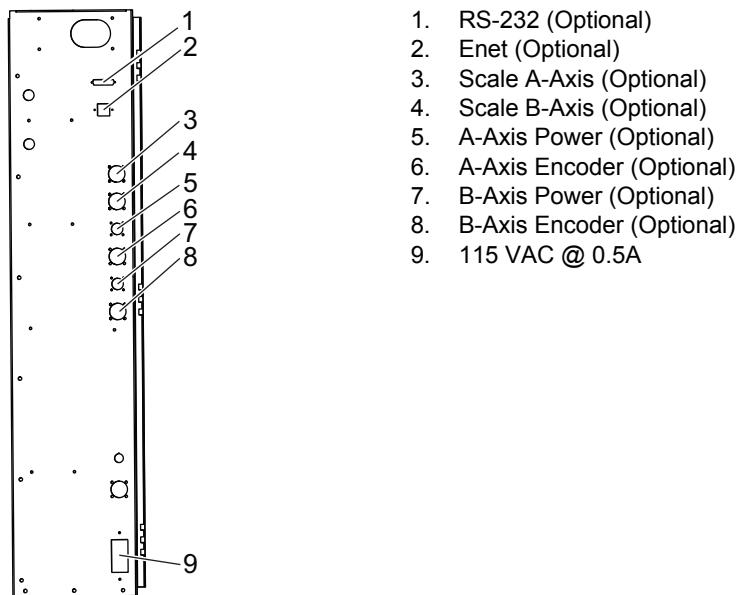
F2.6: Detail A - Electrical Connectors



F2.7: Detail B



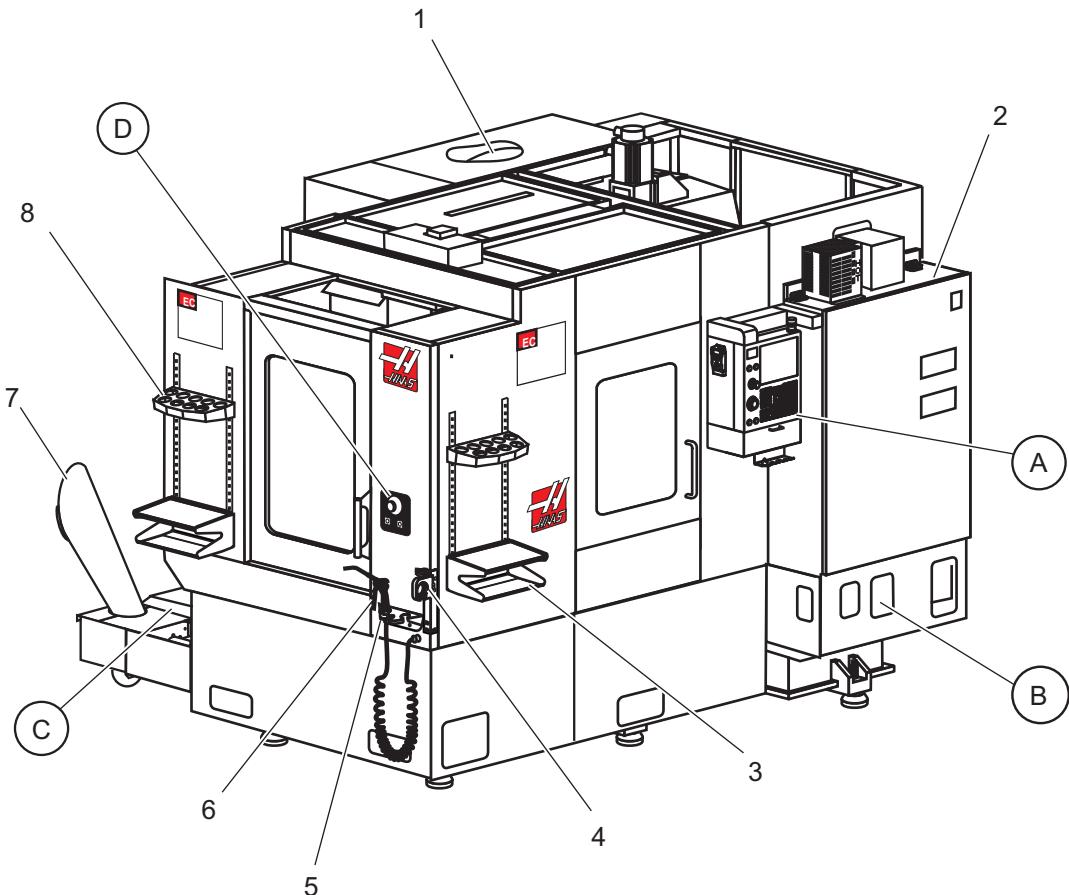
F2.8: Detail C



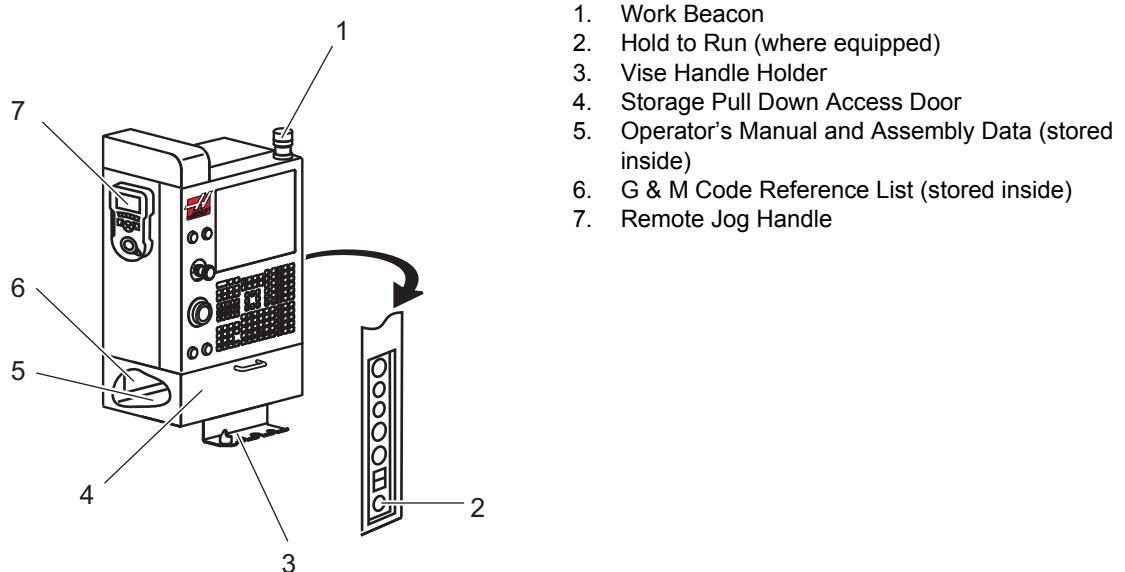
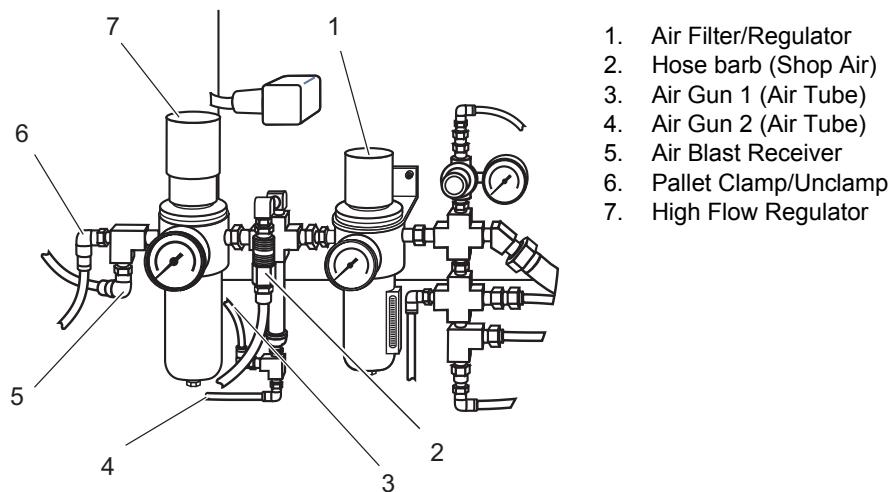
2.2 Horizontal Mill Orientation

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

F2.9: Horizontal Mill Features (EC-400 to EC-500, front view)

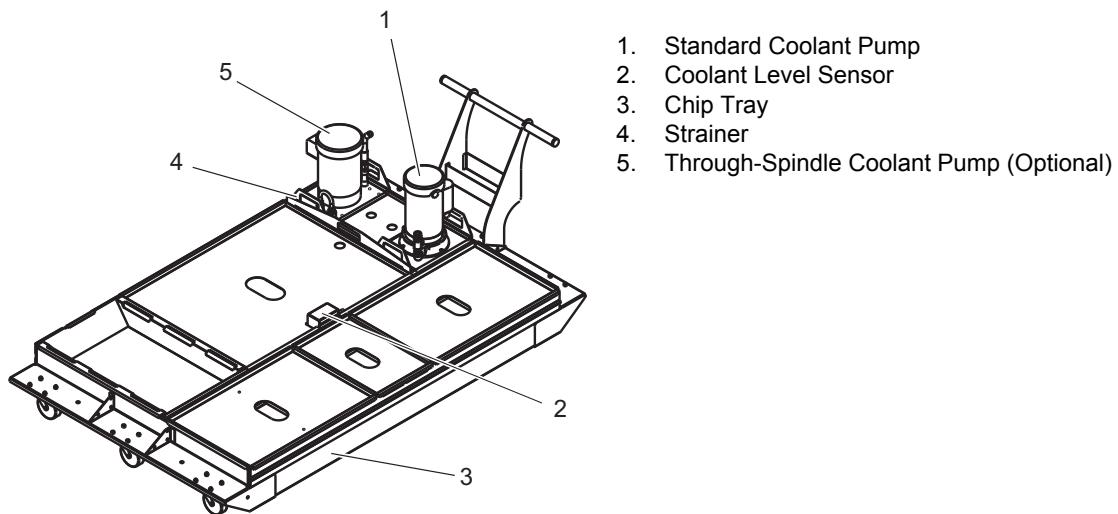


- | | |
|--|---------------------------|
| 1. Side Mount Tool Changer SMTc (optional) | A Control Pendant |
| 2. Electrical Control Box | B Air Supply Assembly |
| 3. Front Work Table | C Coolant Tank Assembly |
| 4. Tool Holding Vise | D Pallet Changer Controls |
| 5. Storage Tray | |
| 6. Air Gun | |
| 7. Chip Conveyor (optional) | |
| 8. Tool Tray | |

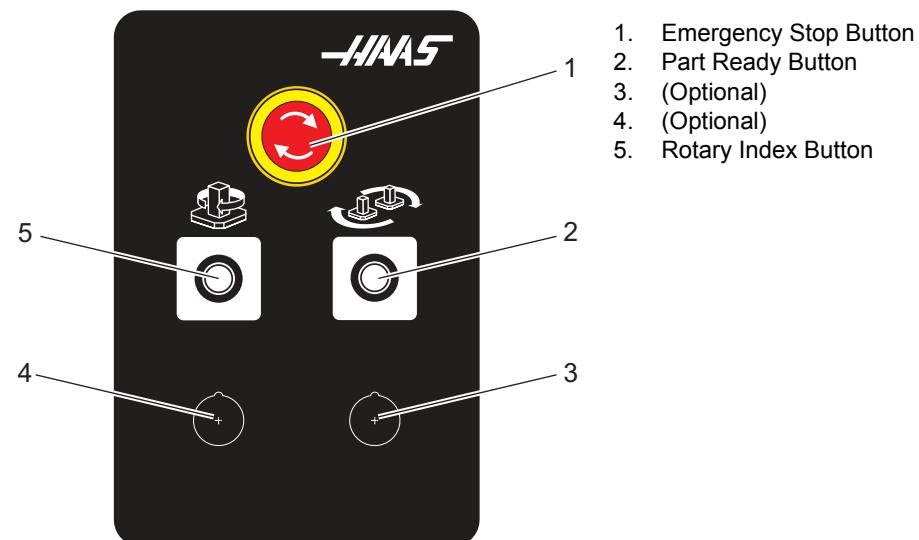
F2.10: Detail A**F2.11:** Detail B

Horizontal Mill Orientation

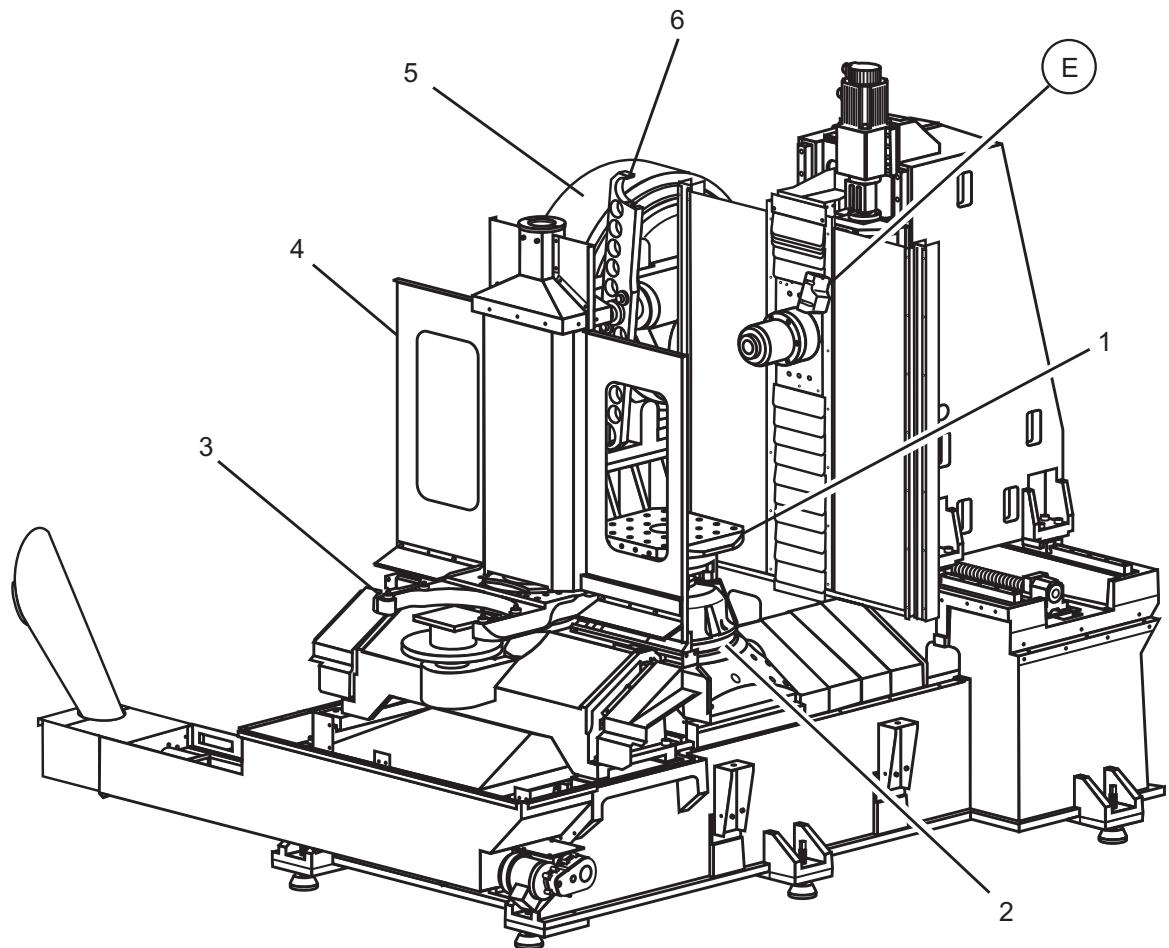
F2.12: Detail C



F2.13: Detail D



F2.14: Horizontal Mill Features (EC-400 covers removed)

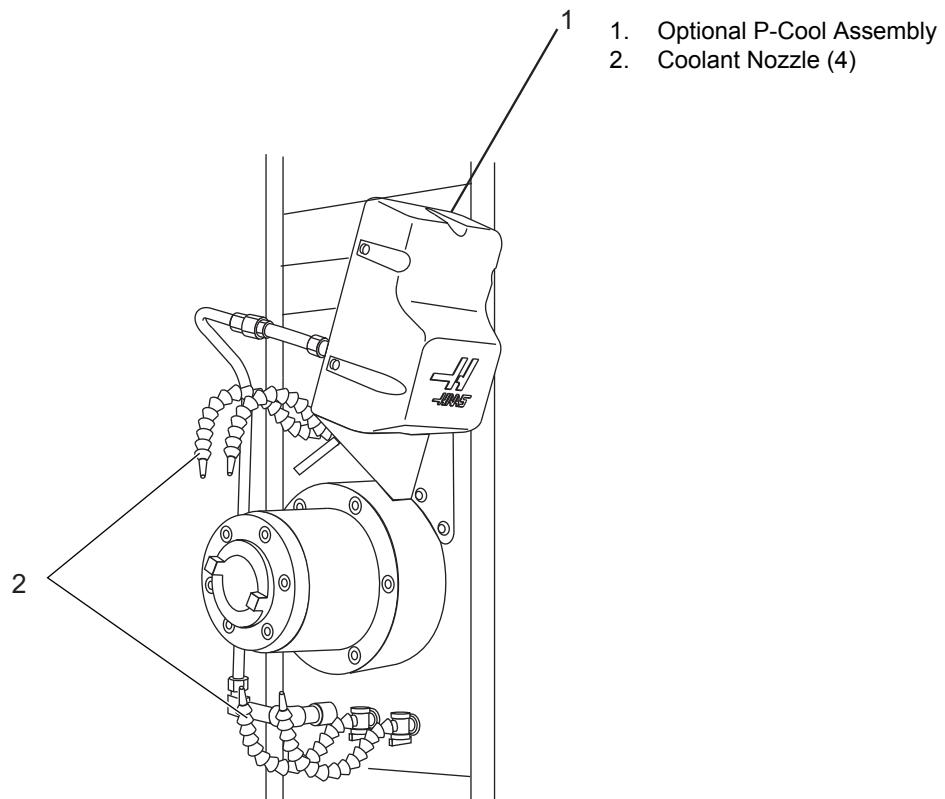


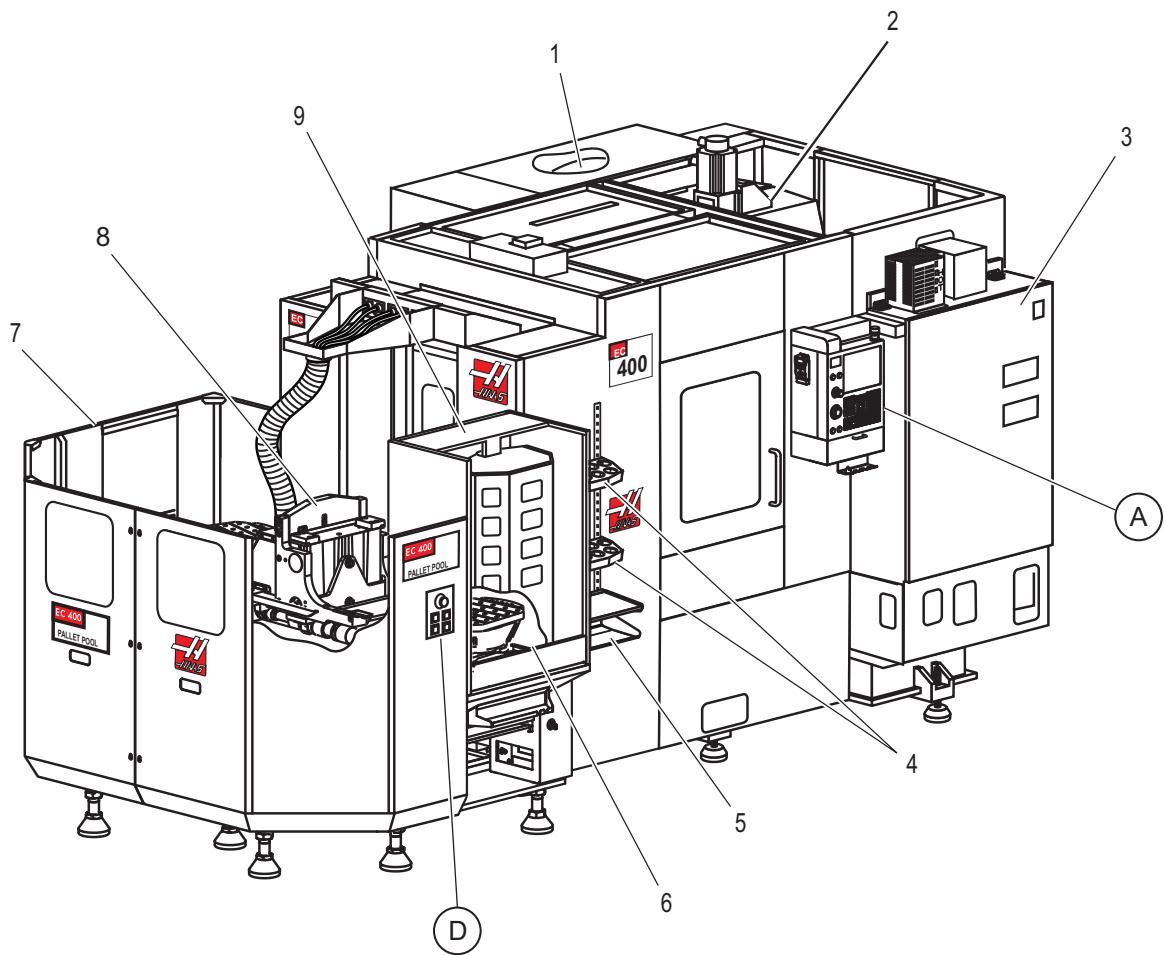
1. Pallet (2)
2. Rotary
3. Pallet Support Arms (pallet removed)
4. Pallet Doors
5. SMTCA
6. SMTCB

E EC-400 Coolant Nozzles

Horizontal Mill Orientation

F2.15: Detail E

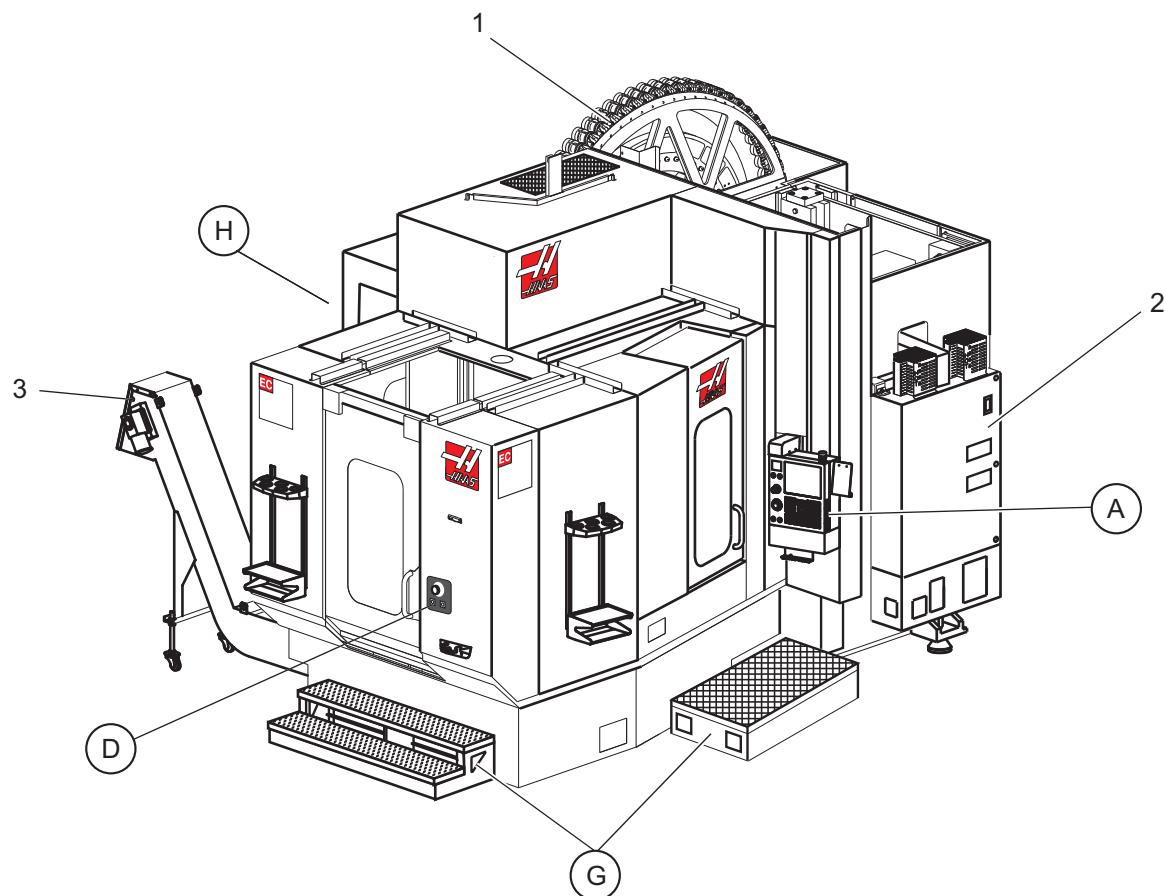


F2.16: Horizontal Mill Features (EC-400 with Pallet Pool)

- | | |
|------------------------------------|---------------------------|
| 1. SMT ^C | A Control Pendant |
| 2. X-axis and Y-axis column | D Pallet Changer Controls |
| 3. Main Electrical Control Cabinet | |
| 4. Tool Crib | |
| 5. Front Table | |
| 6. Load Station | |
| 7. Pallet Pool | |
| 8. Pallet Pool Slider Assembly | |
| 9. Pallet Pool Load Station | |

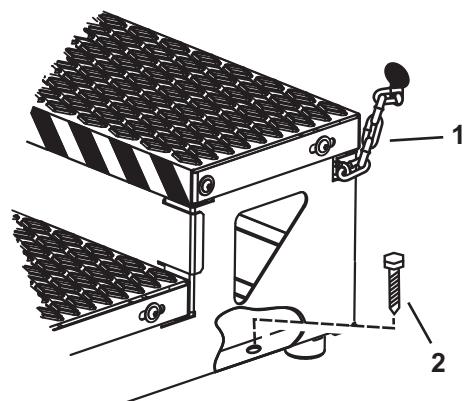
Horizontal Mill Orientation

F2.17: Horizontal Mill Features (EC-550-630)

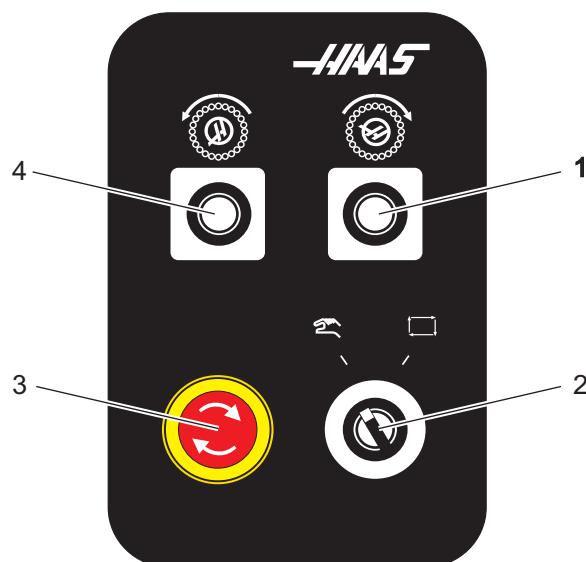


1. SMTC
2. Control Cabinet
3. Chip Conveyor

- A Control Pendant
- D Pallet Changer Controls
- G Stair/Step
- H Remote Tool Changer Controls

F2.18: Detail G

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

F2.19: Detail H

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

2.3 Control Pendant

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

- Pendant front panel
- Pendant right side, top, and bottom
- Keyboard
- Control display

2.3.1 Pendant Front Panel

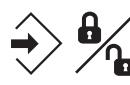
T2.1: Front Panel Controls

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

2.3.2 Pendant Right Side, Top, and Bottom Panels

The following tables describe the right side, top, and bottom of the pendant.

T2.2: Right Side Panel Controls

Name	Image	Function
USB		Plug compatible USB devices into this port. It has a removable dust cap.
Memory Lock		In the locked position, this keyswitch prevents alterations to programs, settings, parameters, offsets, and macro variables.
Setup Mode		In the locked position, this keyswitch enables all machine safety features. Unlock allows setup (refer to "Setup Mode" in the Safety section of this manual for details).
Second Home		Press to rapid all axes to the coordinates specified in G154 P20 (if equipped).
Servo Auto Door Override		Press this button to open or close the Servo Auto Door (if equipped).
Worklight		These buttons toggle the internal worklight and High Intensity Lighting (if equipped).

Control Pendant

T2.3: Pendant Top Panel

Beacon Light	
Provides quick visual confirmation of the machine's current status. There are five different beacon states:	
Light Status	Meaning
Off	The machine is idle.
Solid Green	The machine is running.
Flashing Green	The machine is stopped, but is in a ready state. Operator input is required to continue.
Flashing Red	A fault has occurred, or the machine is in Emergency Stop.
Flashing Yellow	A tool has expired, and the tool life screen automatically displays.

T2.4: Pendant Bottom Panel

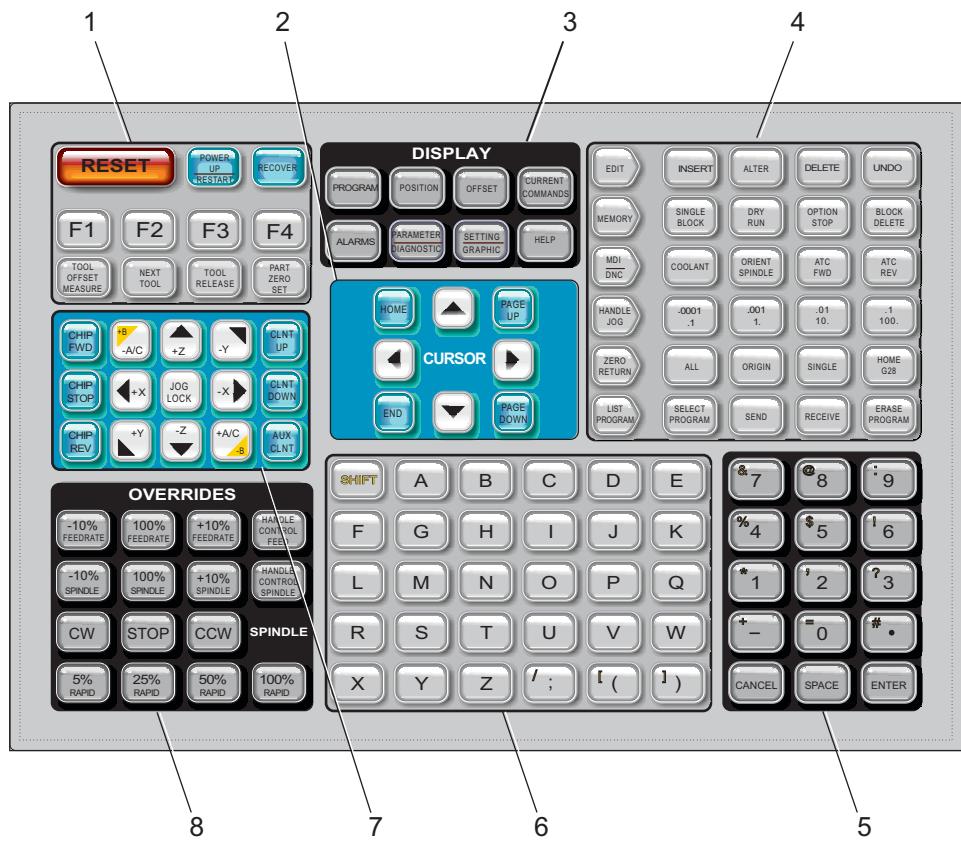
Name	Function
Keyboard Beeper	Located on the bottom of the control pendant. Turn the cover to adjust the volume.

2.3.3 Keyboard

Keyboard keys are grouped into these functional areas:

1. Function
2. Cursor
3. Display
4. Mode
5. Numeric
6. Alpha
7. Jog
8. Overrides

F2.20: Mill Keyboard: [1] Function Keys, [2] Cursor Keys, [3] Display Keys, [4] Mode Keys, [5] Numeric Keys, [6] Alpha Keys, [7] Jog Keys, [8] Override Keys.



Function Keys

T2.5: List of Function Keys and How They Operate

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

Cursor Keys

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

T2.6: Cursor Key List

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

Display Keys

You use the Display to see the machine displays, operational information, and help pages. You also use some of these keys to switch between active panes within a function mode. Some of these keys display additional screens if you press them more than once.

T2.7: List of Display Keys and How They Operate

Name	Key	Function
Program	[PROGRAM]	Selects the active program pane in most modes.
Position	[POSITION]	Selects the positions display.
Offsets	[OFFSET]	Press to toggle between the two offsets tables.
Current Commands	[CURRENT COMMANDS]	Displays menus for Maintenance, Tool Life, Tool Load, Advanced Tool Management (ATM), System Variables, Clock settings, and timer/counter settings.
Alarms / Messages	[ALARMS]	Displays the alarm viewer and message screens.

Control Pendant

Name	Key	Function
Parameter / Diagnostics	[PARAMETER / DIAGNOSTIC]	Displays parameters that define the machine's operation. Parameters are set at the factory and should not be modified except by authorized Haas personnel.
Settings / Graphics	[SETTING / GRAPHIC]	Displays and allows changing of user settings, and enables Graphics mode.
Help	[HELP]	Displays help information.

Mode Keys

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

T2.8: List of [EDIT] Mode Keys and How They Operate

Name	Key	Function
Edit	[EDIT]	Selects EDIT mode to edit programs in the control's memory. Shows <i>EDIT:EDIT</i> in upper left display.
Insert	[INSERT]	Enters text from the input line or the clipboard into the program at the cursor position.
Alter	[ALTER]	Replaces the highlighted command or text with text from the input line or the clipboard.  NOTE: [ALTER] does not work for offsets.
Delete	[DELETE]	Deletes the item that the cursor is on, or deletes a selected program block.
Undo	[UNDO]	Undoes up to the last 9 edit changes, and deselects a highlighted block.  NOTE: [UNDO] does not work for deleted highlighted blocks or to recover a deleted program.

T2.9: List of [MEMORY] Mode Keys and How They Operate

Name	Key	Function
Memory	[MEMORY]	Selects memory mode. Programs are run from this mode, and the other keys in the MEM row control the ways in which the program is run. Shows <i>OPERATION:MEM</i> in upper left display.
Single Block	[SINGLE BLOCK]	Toggles single block on or off. When single block is on, the control runs only one program block each time you press [CYCLE START].
Dry Run	[DRY RUN]	Checks actual machine movement without cutting a part.
Optional Stop	[OPTION STOP]	Toggles optional stop on or off. When optional stop is on, the machine will stop when it reaches M01 commands.
Block Delete	[BLOCK DELETE]	Toggles block delete on or off. Program ignores (does not execute) items with a slash ("/") when this option is enabled.

T2.10: List of [MDI/DNC] Mode Keys and How They Operate

Name	Key	Function
Manual Data Input / Direct Numerical Control	[MDI/DNC]	In MDI mode, you can run programs or blocks of code without saving them. DNC mode allows large programs to be "drip fed" into the control as they run. Shows <i>EDIT:MDI/DNC</i> in upper left display.
Coolant	[COOLANT]	Turns the optional coolant on and off.
Orient Spindle	[ORIENT SPINDLE]	Rotates the spindle to a given position and then locks the spindle.
Automatic Tool Changer Forward / Reverse	[ATC FWD] / [ATC REV]	Rotates the tool turret to the next / previous tool.

Control Pendant

T2.11: List of [HANDLE JOG] Mode Keys and How They Operate

Name	Key	Function
.0001/.1	[.0001 / .1], [.001 / 1], [.01 / 10], [.1 / 100]	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 bottom number is used for dry run mode. Shows <i>SETUP: JOG</i> in upper left display.

T2.12: List of [ZERO RETURN] Mode Keys and How They Operate

Name	Key	Function
Zero Return	[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 [POSITION] or [PAGE UP]/[PAGE DOWN] to switch between the categories. Shows <i>SETUP: ZERO</i> in upper left display.
All	[ALL]	Returns all axes to machine zero. This is similar to [POWER UP/RESTART], except a tool change does not occur.
Origin	[ORIGIN]	Sets selected values to zero.
Single	[SINGLE]	Returns one axis to machine zero. Press the desired axis letter on the Alpha keyboard and then press [SINGLE].
Home G28	[HOME G28]	Returns all axes to zero in rapid motion. [HOME G28] will also home a single axis in the same manner as [SINGLE].



CAUTION: All axes move immediately when you press this key. To prevent a crash, make sure the axis motion path is clear.

T2.13: List of [LIST PROGRAM] Mode Keys and How They Operate

Name	Key	Function
List Programs	[LIST PROGRAM]	Accesses a tabbed menu to load and save programs. Shows <i>EDIT:LIST</i> in upper left display.
Select Programs	[SELECT PROGRAM]	Makes the highlighted program the active program.
Send	[SEND]	Transmits programs out the optional RS-232 serial port.
Receive	[RECEIVE]	Receives programs from the optional RS-232 serial port.
Erase Program	[ERASE PROGRAM]	Deletes the selected program in List Program mode. Deletes the entire program in MDI mode.

Numeric Keys

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

T2.14: List of Numeric Keys and How They Operate

Name	Key	Function
Numbers	[0]-[9]	Types numbers.
Minus sign	[$-$]	Adds a minus (-) sign to the input line.
Decimal point	[$.$]	Adds a decimal point to the input line.
Cancel	[CANCEL]	Deletes the last character typed.
Space	[SPACE]	Adds a space to input.
Enter	[ENTER]	Answers prompts and writes input.
Special Characters	Press [SHIFT] , then a numeric key	Inserts the yellow character on the upper-left of the key. These characters are used for comments, macros, and certain special features.
+	[SHIFT], then [$-$]	Inserts +
=	[SHIFT], then [0]	Inserts =

Control Pendant

Name	Key	Function
#	[SHIFT], then [.]	Inserts #
*	[SHIFT], then [1]	Inserts *
'	[SHIFT], then [2]	Inserts '
?	[SHIFT], then [3]	Inserts ?
%	[SHIFT], then [4]	Inserts %
\$	[SHIFT], then [5]	Inserts \$
!	[SHIFT], then [6]	Inserts !
&	[SHIFT], then [7]	Inserts &
@	[SHIFT], then [8]	Inserts @
:	[SHIFT], then [9]	Inserts :

Alpha Keys

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

T2.15: List of Alpha Keys and How They Operate

Name	Key	Function
Alphabet	[A]-[Z]	Uppercase letters are the default. Press [SHIFT] and a letter key for lowercase.
End-of-block (EOB)	[;]	This is the end-of-block character, which signifies the end of a program line.
Parentheses	[(,)]	Separate CNC program commands from user comments. They must always be entered as a pair.
Shift	[SHIFT]	Accesses additional characters on the keyboard, or shifts to lower case alpha characters. The additional characters are seen in the upper left of some of the alpha and number keys.

Name	Key	Function
Special Characters	Press [SHIFT], then an alpha key	Inserts the yellow character on the upper-left of the key. These characters are used for comments, macros, and certain special features.
	[SHIFT], then [:]	Inserts /
	[SHIFT], then [(Inserts [
	[SHIFT], then ()]	Inserts]

Jog Keys

T2.16: List of Jog Keys and How They Operate

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

Control Pendant

Override Keys

T2.17: List of Override Keys and How They Operate

Name	Key	Function
-10% Feedrate	[-10% FEEDRATE]	Decreases the current feedrate by 10%.
100% Feedrate	[100% FEEDRATE]	Sets an overridden feedrate back to the programmed feed rate.
+10% Feedrate	[+10% FEEDRATE]	Increases the current feedrate by 10%.
Handle Control Feed Rate	[HANDLE CONTROL FEED]	Lets you use the jog handle to adjust the feedrate in 1% increments.
-10% Spindle	[-10% SPINDLE]	Decreases the current spindle speed by 10%.
100% Spindle	[100% SPINDLE]	Sets the overridden spindle speed back to the programmed speed.
+10% Spindle	[+10% SPINDLE]	Increases the current spindle speed by 10%.
Handle Control Spindle	[HANDLE CONTROL SPINLE]	Lets you use the jog handle to adjust the spindle speed in 1% increments.
Clockwise	[CW]	Starts the spindle in the clockwise direction.
Stop	[STOP]	Stops the spindle.
Counterclockwise	[CCW]	Starts the spindle in the counterclockwise direction.
Rapids	[5% RAPID] / [25% RAPID] / [50% RAPID] / [100% RAPID]	Limits machine rapids to the value on the key.

Override Usage

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

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

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

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

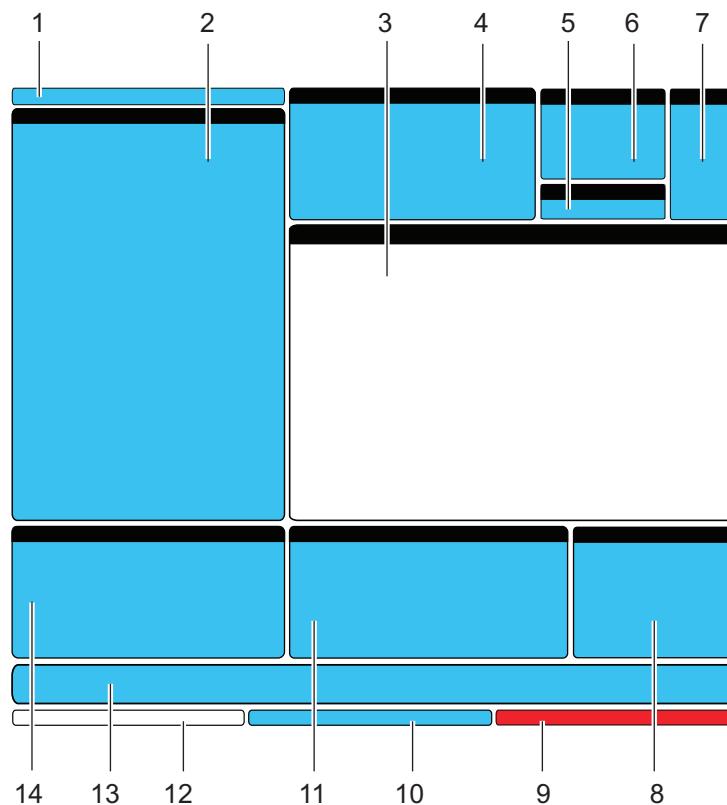
You can override the standard coolant setting by pressing **[COOLANT]**. The coolant pump remains either on or off until the next M-code or operator action (see Setting 32).

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

2.3.4 Control Display

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

F2.21: Basic Control Display Layout



- | | |
|---------------------------------------|----------------------------------|
| 1. Mode and Active Display Bar | 8. Alarm Status |
| 2. Program Display | 9. System Status Bar |
| 3. Main Display (size varies) | 10. Position Display / Axis Load |
| 4. Active Codes | Meters / Clipboard |
| 5. Active Tool | 11. Input Bar |
| 6. Coolant | 12. Icon Bar |
| 7. Timers, Counters / Tool Management | 13. Spindle Status / Editor Help |

The currently active pane has a white background. You can work with data in a pane only when that pane is active, and only one pane is active at any given time. For example, if you want to work with the **Program Tool Offsets** table, press **[OFFSET]** until the table displays with a white background. You can then make changes to the data. In most cases, you change the active pane with the display keys.

Mode and Active Display Bar

Machine 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 optional Visual Quick Code system (VQC), Intuitive Programming System (IPS), and the optional Wireless Intuitive Probing System (WIPS) if installed. Operation mode includes MEM, the mode in which you run programs.

F2.22: The Mode and Display bar shows [1] the current mode and [2] the current display function.



T2.18: Mode, Key Access, and Bar Display

Mode	Mode Key	Bar Display	Function
Setup	[ZERO RETURN]	SETUP: ZERO	Provides all control features for machine setup.
	[HANDLE JOG]	SETUP: JOG	
Edit	[EDIT]	EDIT: EDIT	Provides all program editing, management, and transfer functions.
	[MDI/DNC]	EDIT: MDI	
	[LIST PROGRAM]	EDIT: LIST	
Operation	[MEMORY]	OPERATION: MEM	Provides all control features necessary to run a program.

Offsets Display

There are two offset tables, the Program Tool Offsets table and the Active Work Offset table. Depending on the mode, these tables may appear in two separate display panes, or they may share a pane; press [OFFSET] to toggle between tables.

T2.19: Offset Tables

Name	Function
Program Tool Offsets	This table displays tool numbers and tool length geometry.
Active Work Offset	This table displays the values entered so that each tool knows where the part is located.

Current Commands

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

To access this display, press [**CURRENT COMMANDS**], then press [**PAGE UP**] or [**PAGE DOWN**] to cycle through the pages.

Operation Timers and Setup Display - This page shows:

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

These timers and counters appear in the lower right section of the display in the **OPERATION:MEM** and **SETUP:ZERO** modes.

Macro Variables Display -This page shows a list of the macro variables and their current values. The control updates these variables as programs run. You can also modify the variables in this display; Refer to the Macros section on page **181** in Optional Programming.

Active Codes - This page lists the currently active program codes. A smaller version of this display is included on the **OPERATION:MEM** mode screen.

Positions - This page shows a larger view of the current machine positions, with all of the position reference points (operator, machine, work, distance to go) on the same screen.

**NOTE:**

*You can handle jog the machine axes from this screen if the control is in **SETUP: JOG** mode.*

Tool Life Display - This page shows information that the control uses to predict tool life.

Tool Load Monitor and Display - On this page, you can enter the maximum tool load percentage that is expected for each tool.

Maintenance - On this page, you can activate and deactivate a series of maintenance checks.

Advanced Tool Management - This feature allows you to create and manage tool groups. For more information, refer to the Advanced Tool Management section in the Operation chapter of this manual.

Timer and Counter Reset

To reset the timers and counters on the **CURRENT COMMANDS/TIMERS AND COUNTERS** page:

1. Press the cursor arrow keys to highlight the name of the timer or counter that you want to reset.
2. Press **[ORIGIN]** to reset the timer or counter.

**TIP:**

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

Date and Time Adjustment

To adjust the Date and Time:

1. Press **[CURRENT COMMANDS]**.
2. Press **[PAGE UP]** or **[PAGE DOWN]** until you see the **DATE AND TIME** screen.
3. Press **[EMERGENCY STOP]**.
4. Type the current date (in MM-DD-YYYY format) or current time (in HH:MM:SS format).

**NOTE:**

You must include the dash (-) or colon (:) when you enter a new date or time.

5. Press [ENTER]. Make sure the new date or time is correct. Repeat step 4 if it is not correct.
6. Reset [EMERGENCY STOP] and clear Alarm.

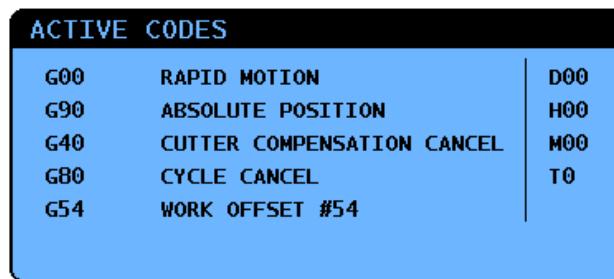
Setting/Graphic Display Function

Press [SETTING/GRAFIC] until you see the Setting shows. Settings change the way the mill behaves; refer to the “Settings” section starting on page 351 for a more detailed description.

To use Graphics mode, press [SETTING/GRAFIC] until you see the Graphics screen. Graphics shows a visual dry run of your part program without the need to move the axes and risk tool or part damage from programming errors. This function is more useful than the Dry Run mode, because you can check all of your work offsets, tool offsets, and travel limits before running the machine. The risk of a crash during setup is greatly reduced. Refer to Graphics Mode on page 104 for a more detailed description.

Active Codes

F2.23: Active Codes Display Example



This display gives read-only, real-time information about the codes that are currently active in the program; specifically, the codes that define the current motion type (rapid vs linear feed vs circular feed), positioning system (absolute vs incremental), cutter compensation (left, right or off), active canned cycle, and work offset. This display also gives the active Dnn, Hnn, Tnn, and most recent Mnnn code.

Active Tool

F2.24: Active Tool Display Example



This display gives information on the current tool in the spindle, including the type of tool (if specified), the maximum tool load the tool has seen and the percentage of tool life remaining (if using Advanced Tool Management).

Coolant Display

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

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

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

In the coolant gauge, an arrow shows the coolant level. Full is **1/1** and empty is **0/1**. To avoid possible coolant flow problems, keep the coolant level above the red range. This gauge is also displayed in **DIAGNOSTICS** mode under the **GAUGES** tab.

Timers & Counters Display

The timer section of this display (located above the lower right of the screen) provides information about cycle times (This Cycle, Last Cycle, and Remaining).

The counter section also includes two M30 counters as well as a Loops Remaining display.

- M30 Counter #1: and M30 Counter #2: every time a program reaches an **M30** command, the counters increase by one. If Setting 118 is on, the counters also increment every time a program reaches an M99 command.
- If you have macros , you can clear or change M30 Counter #1 with #3901 and M30 Counter #2 with #3902 (#3901=0).
- Refer to page **41** for information on how to reset the timers and counters.
- Loops Remaining: shows the number of subprogram loops remaining to complete the current cycle.

Alarm Display

You can use this display to learn more about machine alarms when they occur, to view your machine's entire alarm history, or to read about alarms that can occur.

Press **[ALARMS]** until the ALARMS display appears. Press the **[RIGHT]** and **[LEFT]** cursor arrow keys to cycle between the (3) different alarm display screens:

- The Active Alarm screen shows the alarms that currently affect machine operation. You can use the **[UP]** and **[DOWN]** cursor arrow keys to see the next alarm; they display one-at-a-time.
- The Alarm History screen shows a list of the alarms that have recently affected machine operation.
- The Alarm Viewer screen shows the detailed description of the most recent alarm. You can also enter any alarm number and press **[ENTER]** to read its description.

Messages

You can add a message to the **MESSAGES** screen and it will be saved there until it is removed or changed. The **MESSAGES** screen appears during power-up if there are no new alarms present. To read, add, correct, or clear messages:

1. Press **[ALARMS]** until the **MESSAGES** screen appears.
2. Use the keypad to type your message.

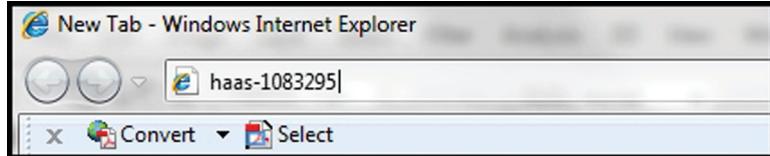
Press **[CANCEL]** or **[SPACE]** to delete existing characters. Press **[DELETE]** to delete an entire line. Your message data is automatically stored and maintained even in a power-off state.

Alarm Alerts

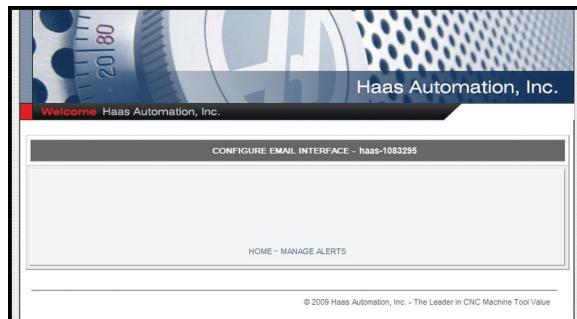
Haas machines include a basic application to send an alert to an email address or cellular telephone when an alarm occurs. Setting up this application requires some knowledge about your network; ask your system administrator or Internet Service Provider (ISP) if you do not know the correct settings.

Before you set 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.

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



2. A 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**.
3. The home screen displays, with the setup options at the bottom of the screen. Click **Manage Alerts**.



Control Pendant

4. At the Manage Alerts screen, enter the email address and/or cellular phone number where you want to receive alerts. If you enter a cellular phone number, select your carrier from the pull-down menu under the cell number field. Click **SUBMIT CHANGES**.

The screenshot shows the 'MANAGE ALERTS' page. It has a header 'Haas Automation, Inc.' and a sub-header 'MANAGE ALERTS - haas-1083295'. There are two input fields: 'Email alerts to:' and 'Text alert cell number:'. Below the second field is a dropdown menu labeled 'Cellular carrier' with the option 'Other - enter full URL with cell number'. At the bottom is a 'SUBMIT CHANGES' button.



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' page. It has a header 'Haas Automation, Inc.' and a sub-header 'CONFIGURE EMAIL INTERFACE - haas-1083295'. There are four input fields: 'DNS IP address:', 'SMTP server name:', 'SMTP server port:' (set to 25), and 'Authorized EMAIL account:'. At the bottom is a 'SUBMIT CHANGES' button.



NOTE:

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

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.
 - 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.

System Status Bar

The System Status Bar is the read-only section of the screen located in the bottom, center. It displays messages for the user about actions they have taken.

Position Display

The Position display usually appears near the lower center of the screen. It shows the current axis position relative to four reference points (Operator, Work, Machine and Distance-to-go). In **SETUP: JOG** mode, this display shows all of the relative positions at the same time. In other modes, press **[POSITION]** to cycle through the different reference points.

Control Pendant

T2.20: Axis Position Reference Points

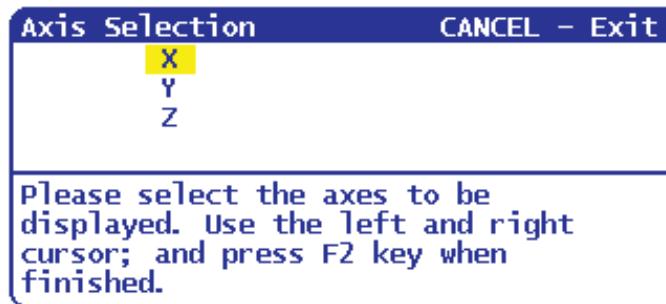
Coordinate Display	Function
OPERATOR	This position shows the distance you have jogged the axes. This does not necessarily represent the actual distance the axis is from machine zero, except when the machine is first powered on.
WORK (G54)	This displays the axis positions relative to part zero. On power-up,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

Position Display Axis Selection

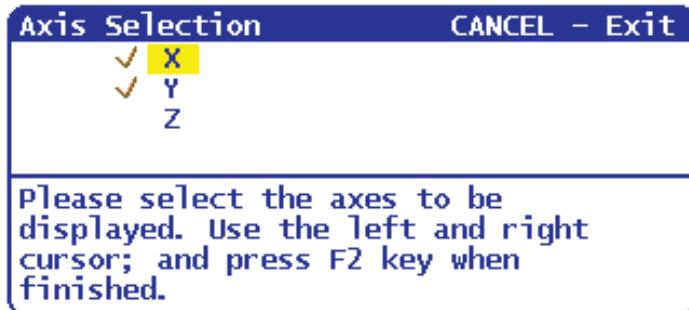
Use this function to change the axis positions that show in the display.

1. With a position display active, press **[F2]**. The **Axis Selection** pop-up menu appears.

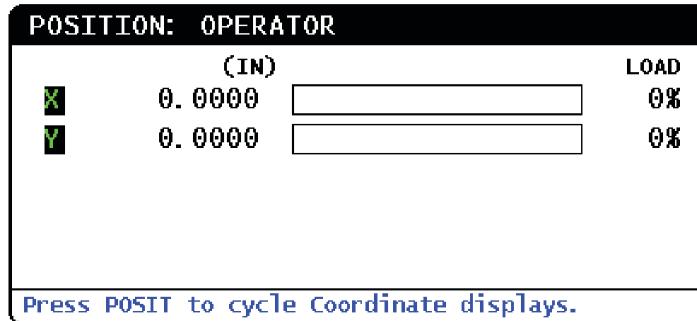
F2.25: The Axis Selection Pop-up Menu



2. Press the **[LEFT]**, **[RIGHT]**, **[UP]**, or **[DOWN]** cursor arrow keys to highlight an axis letter.
3. Press **[ENTER]** to place a check mark next to the highlighted axis letter. This mark means that you want to include that axis letter in the position display.

F2.26: The X and Y Axes Selected in the Axis Selection Menu

4. Repeat steps 2 and 3 until you have selected all of the axes you want to display.
5. Press [F2]. The position display updates with your selected axes.

F2.27: The Updated Position Display

Input Bar

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

F2.28: Input Bar

Special Symbol Input

Some special symbols are not on the keypad.

T2.21: Special Symbols

Symbol	Name
-	underscore
^	caret
~	tilde
{	open curly brackets
}	closed curly brackets
\	backslash
	pipe
<	less than
>	greater than

Do these steps to enter special symbols:

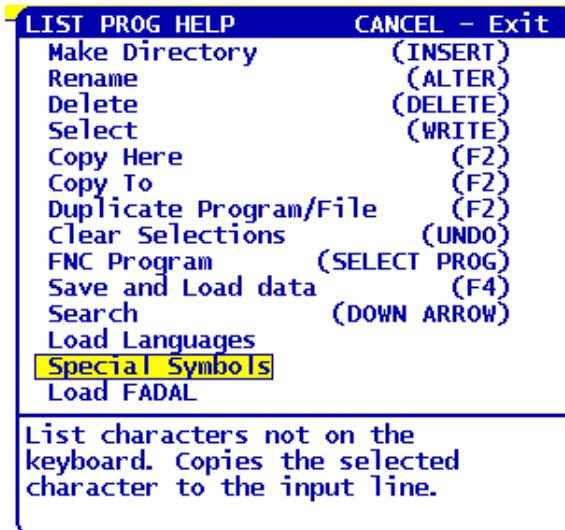


NOTE:

You must have a USB device connected to the control pendant, or an optional hard drive, to access the SPECIAL SYMBOLS menu.

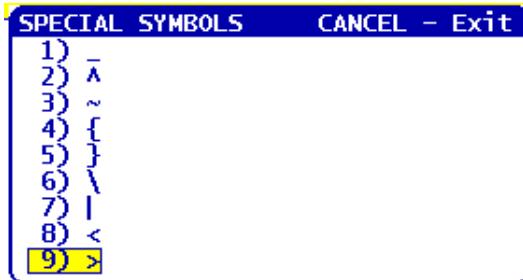
1. Press [LIST PROGRAMS] and select **USB DEVICE** or optional **HARD DRIVE**.
2. Press **[F1]**.

The **LIST PROG HELP** menu shows:



3. Select **Special Symbols** and press **[ENTER]**.

The **SPECIAL SYMBOLS** pick list shows:



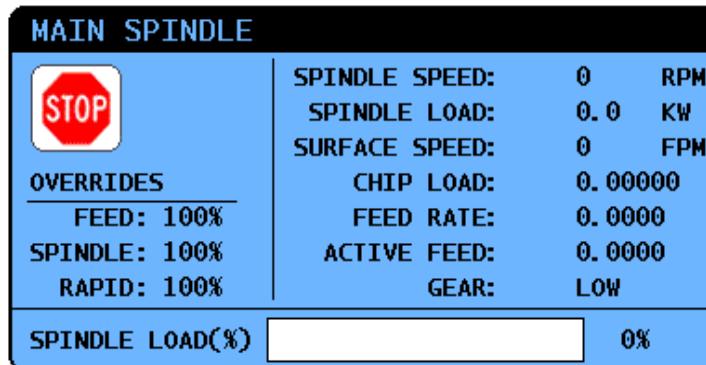
4. Select the symbol and press **[ENTER]** to copy the symbol to the **INPUT:** bar.

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

1. Highlight the directory with the name that you want to change.
2. Type **MY**.
3. Press **[F1]**.
4. Select **Special Symbols** and press **[ENTER]**.
5. Highlight _ (underscore) and press **[ENTER]**.
6. Type **DIRECTORY**.
7. Press **[ALTER]**.

Main Spindle Display

F2.29: Main Spindle (Speed and Feed Status) Display



This first column of this display gives you information about spindle status and the current override values for spindle, feed, and rapids.

The second column displays actual motor load in kW. This value reflects the actual spindle power to the tool. It also displays current programmed and actual spindle speed as well as programmed and actual feed rate.

The bar-graph spindle load meter indicates the current spindle load as a percentage of motor capacity.

2.3.5 Screen Capture

The control can capture and save an image 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.

1. If you want to save the screen capture under a particular filename, type it first. The control adds the *.bmp file extension automatically.



NOTE:

If you do not specify a filename, the control will use the default filename snapshot.bmp. This will overwrite any screen capture taken previously with the default name. Be sure to specify a filename each time if you want to save a series of screen captures.

2. Press [SHIFT].
3. Press [F1].

The screen capture is saved to your USB device or the machine's hard drive, and the control displays the message *Snapshot saved to HDD/USB* when the process is finished.

2.4 Tabbed Menu Basic Navigation

Tabbed menus are used in several control functions, such as Parameters, Settings, Help, List Programs, and IPS. To navigate these menus:

1. Use the [LEFT] and [RIGHT] cursor arrows to select a tab.
2. Press [ENTER] to open the tab.
3. If the selected tab contains sub-tabs, use the cursor arrows, then press [ENTER] to select the sub-tab you want. Press [ENTER] again to open the sub-tab.



NOTE:

In the tabbed menus for parameters and settings, and in the ALARM VIEWER section of the Alarm / Messages display, you can type the number of a parameter, setting, or alarm that you want to view, then press the [UP] or [DOWN] cursor arrow to view it.

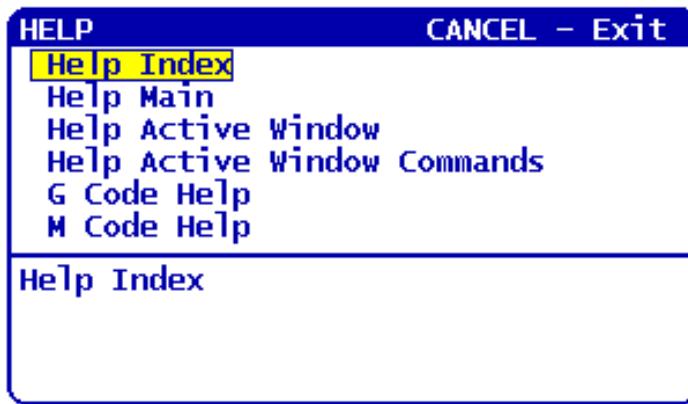
4. Press [CANCEL] if you want to close a sub-tab and return to the higher tab level.

2.5 Help

Use the help function when you need information about machine functions, commands, or programming. The content of this manual is also available on the control.

When you press [HELP], a pop-up menu appears with options for different help information. If you want to directly access the help tabbed menu, press [HELP] again. Refer to page 54 for information on that menu. Press [HELP] again to exit the help function.

F2.30: The Pop-up Help Menu



Use the [UP] and [DOWN] cursor arrow keys to highlight an option, then press [ENTER] to select it. The options available from this menu are:

- **Help Index** - Gives a list of available help topics that you can choose from. For more information, refer to the “Help Index” section on page **55**.
- **Help Main** - Gives the table of contents for the Operator’s Manual on the control. Use the **[UP]** and **[DOWN]** cursor arrow keys to select a topic and press **[ENTER]** to see that topic’s contents.
- **Help Active Window** - Gives the help system topic that relates to the currently active window.
- **Help Active Window Commands** - Gives a list of the available commands for the active window. You can use the hot keys listed in parentheses, or you can select a command from the list.
- **G Code Help** - Gives a list of G-codes you can select from in the same manner as the **Help Main** option for more information.
- **M Code Help** - Gives a list of M-codes that you can select from in the same manner as the **Help Main** option for more information.

2.5.1 The Help Tabbed Menu

To access the help tabbed menu, press **HELP** until you see the **Operator’s Manual Table of Contents**. You can then navigate the Operator’s Manual content that is saved on the control.

You can access other help functions from the tabbed menu; press **[CANCEL]** to close the **Operator’s Manual Table of Contents** tab and access the rest of the menu. For information on navigating tabbed menus, refer to page **53**.

These are the available tabs. They are described in more detail in the sections that follow.

- **Search** - Allows you to enter a keyword to find in the Operator’s Manual content that is saved on the control.
- **Help Index** - Gives a list of available help topics that you can choose from. This is the same as the **Help Index** menu option described on page **55**.
- **Drill Table** - Gives a reference table of drill and tap sizes with decimal equivalents.
- **Calculator** - This sub-tabbed menu provides options for several geometric and trigonometric calculators. Refer to the “Calculator Tab” section, starting on page **55** for more information.

2.5.2 Search Tab

Use the Search tab to look for help content by keyword.

1. Press **[F1]** to search manual contents, or press **[CANCEL]** to exit the Help tab and select the Search tab.
2. Type your search term in the text field.
3. Press **[F1]** to execute the search.
4. The results page displays topics that contain your search term; highlight a topic and press **[ENTER]** to view.

2.5.3 Help Index

This option provides a list of manual topics that link to the information in the on-screen manual. Use the cursor arrows to highlight a topic of interest, and then press **[ENTER]** to access that section of the manual.

2.5.4 Drill Table Tab

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

1. Select the Drill Table tab. Press **[ENTER]**.
2. Use **[PAGE UP]** or **[PAGE DOWN]** and the **[UP]** and **[DOWN]** cursor arrows to read the table.

2.5.5 Calculator Tab

The **CALCULATOR** tab has sub-tabs for different calculator functions. Highlight the sub-tab you want and press **[ENTER]**.

Calculator

All of the Calculator sub-tabs perform simple add, subtract, multiply, and divide operations. When one of the sub-tabs is selected, a calculator window appears with the possible operations (LOAD, +, -, *, and /). Numbers are entered for calculation from the input bar after pressing **[ENTER]**.

1. **LOAD** and the calculator window are initially highlighted. The other options can be selected with **[LEFT]/[RIGHT]** cursors. Numbers are entered by typing them and pressing **[ENTER]**. When a number is entered and **LOAD** and the calculator window are highlighted, that number is entered into the calculator window.
2. When a number is entered after one of the other functions (+, -, *, /) is selected, the calculation is performed with the number just entered and any number that was already in the calculator window.

3. The calculator also accepts a mathematical expression in the input bar. For example, type $23*4 - 5.2 + 6/2$ and press **[ENTER]**. The control evaluates this expression by doing multiplication and division first and then subtraction and addition. The result, 89.8, displays in the window. No exponents are allowed.



NOTE:

Data cannot be entered in any field where the label is highlighted.

*Clear data in other fields (by pressing **[F1]** or **[ENTER]**) until the label is no longer highlighted in order to change the field directly.*

4. **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.
5. **[F3]:** In EDIT and MDI modes, **[F3]** copies 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 is used in a program.
6. 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.
7. **[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.

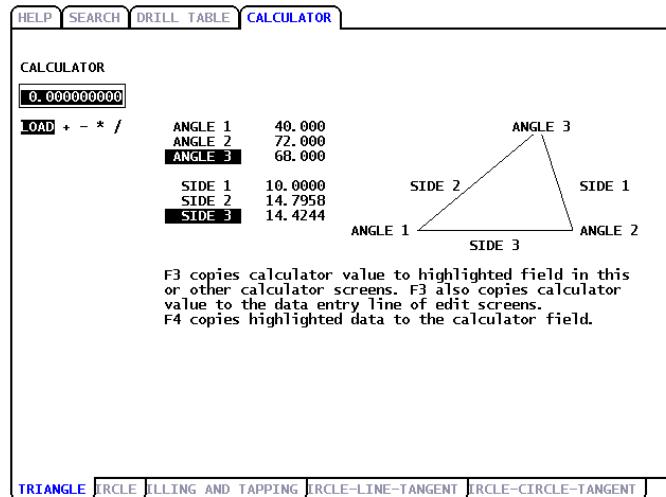
Triangle Sub-tab

The triangle calculator page takes a few triangle measurements and solves for the rest of the values. 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.

1. Use the **[UP]** and **[DOWN]** cursor arrows to select the field for the value to be entered.
2. Type a value, then press **[ENTER]**.
3. Enter the known lengths and angles of a triangle.

When enough data has been entered, the control solves the triangle and displays the results.

F2.31: Calculator Triangle Example



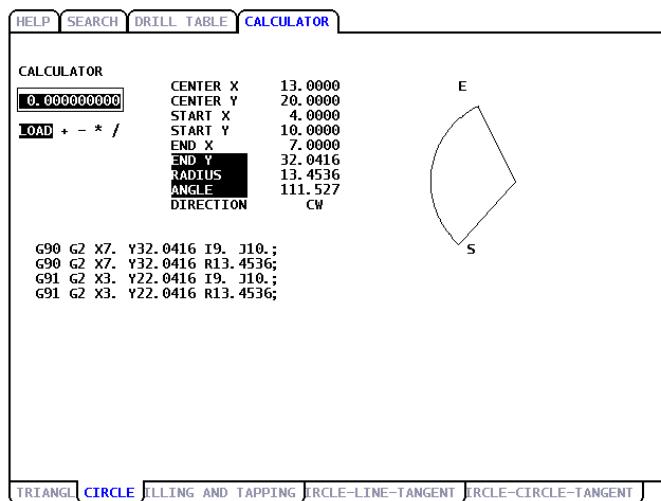
Circle Sub-tab

This calculator page will help solve a circle problem.

1. Use the [UP] and [DOWN] cursor arrows to select the field for the value to be entered.
2. Type the center, radius, angles, start and end points. Press [ENTER] after each entry.

When enough data has been entered, the control solves for the circular motion and displays the rest of the values. Press [ENTER] in the DIRECTION field to change cw/ccw. The control also lists alternate formats that such a move could be programmed with a G02 or G03. Select the format you want and press [F3] to import the highlighted line into the program being edited.

F2.32: Calculator Circle Example



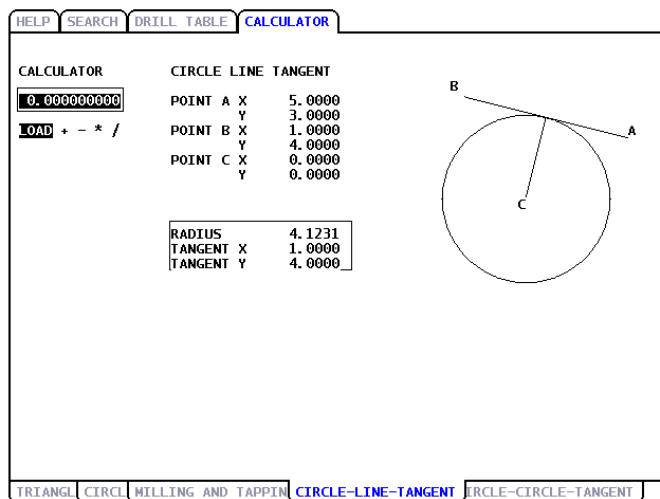
Circle-Line-Tangent Sub-tab

This feature provides the ability to determine points of intersection where a circle and a line meet as tangent.

1. Use the [UP] and [DOWN] cursor arrows to highlight the data field for the value you want to enter.
2. Type the value and press [ENTER].
3. Enter two points, A and B, on a line and a third point, C, away from that line.

The control calculates 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.

F2.33: Calculator Circle-Line-Tangent Example



Circle-Circle-Tangent Sub-tab

This feature determines points of intersection between two circles or points. You provide the location of two circles and their radii. The control calculates the intersection points that are formed by lines tangent to both circles.



NOTE:

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.

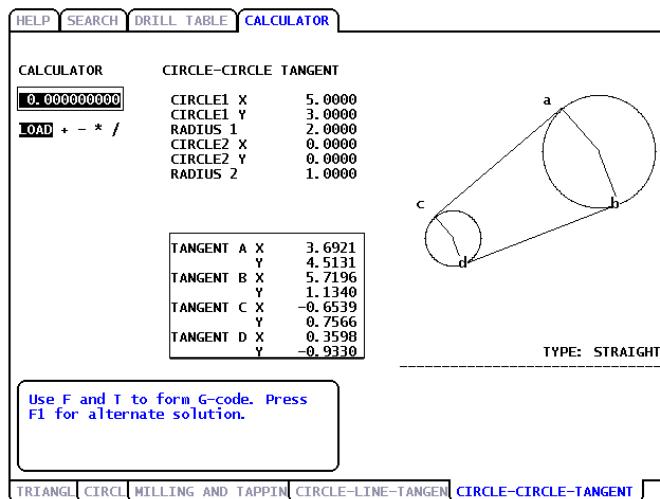
1. Use the UP and DOWN cursor arrows to highlight the data field for the value you want to enter.
2. Type the value and press **[ENTER]**.

After you enter the required values, the control displays the tangent coordinates and associated straight type diagram.

3. Press **[F1]** to toggle between straight and cross tangent results.
4. Press **[F]** and the control prompts 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). To quickly change segment selection, press **[T]** to make the previous To point become the new From point and the control prompts for a new To point.

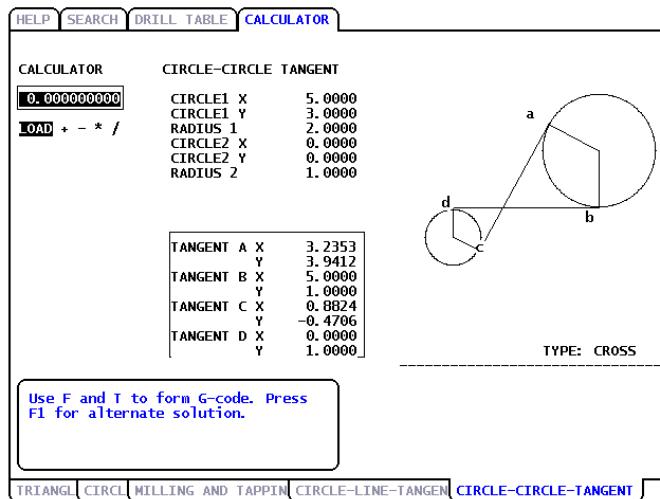
The Input Bar displays the G code for the segment. Solution is in G90 mode. Press M to toggle to G91 mode.

5. Press **[MDI DNC]** or **[EDIT]** and press **[INSERT]** to enter the G-code from the Input Bar.

F2.34: Calculator Circle-Circle-Tangent Type: Straight Example

This example creates this G-code on the input line. From: A To: C generates:

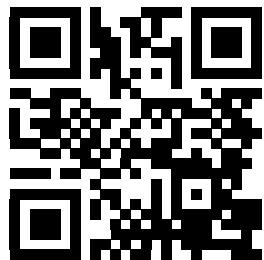
G01 X-4.346 Y-3.7565

F2.35: Calculator Circle-Circle-Tangent Type: Cross Example

More Information Online

2.6 More Information Online

For updated and supplemental information, including tips, tricks, maintenance procedures, and more, visit the Haas Resource Center at diy.HaasCNC.com. You can also scan the code below with your mobile device to go directly to the Resource Center:



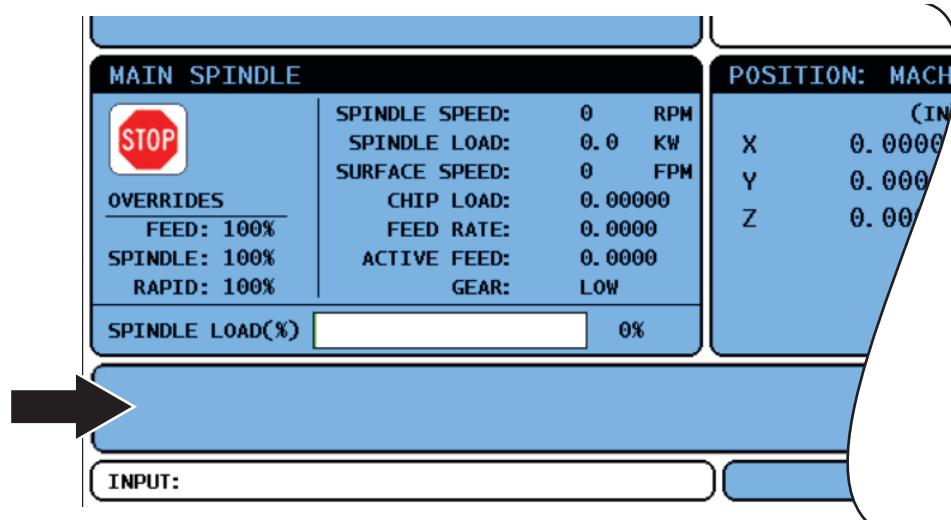
Chapter 3: Control Icons

3.1 Introduction

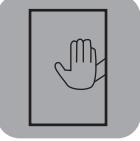
The control screen shows icons to quickly give you information about machine status. Icons tell you about current machine modes, your program as it runs, and machine maintenance status.

The icon bar is near the bottom of the control pendant display, above the input and status bars.

F3.1: Icon Bar Location



3.2 Control Icon Guide

Name	Icon	Meaning
SETUP KEY LOCKED		Setup mode is locked; the control is in "Run" mode. Most machine functions are disabled or limited while the machine doors are open.
SETUP KEY UNLOCKED		Setup mode is unlocked; the control is in "Setup" mode. Most machine functions are available, but may be limited, while the machine doors are open.
DOOR HOLD		Machine motion has stopped because of door rules.
RUNNING		The machine is running a program.
JOGGING		An axis is jogging at the current jog rate.
POWER SAVING SERVOS OFF		The power-saving servos off feature is active. Servos are turned off. Press a key to activate the servos.

Control Icons

Name	Icon	Meaning
JOG RETURN		This icon appears while the control returns to the workpiece during a run-stop-jog-continue operation.
JOG HOLD		You have pressed [FEED HOLD] during the return portion of a run-stop-jog-continue operation.
JOG AWAY		This icon prompts you to jog away during a run-stop-jog-continue operation.
RESTART		The control scans the program before a restart if Setting 36 is ON.
SINGBK STOP		SINGLE BLOCK mode is active, and the control needs a command to continue.
FEEDHOLD		The machine is in feed hold. Axis motion has stopped, but the spindle continues to turn.
FEED		The machine is executing a cutting move.

Control Icon Guide

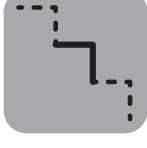
Name	Icon	Meaning
RAPID		The machine is executing a non-cutting axis move (G00) at the fastest possible rate.
DWELL		The machine is executing a dwell (G04) command.
JOG LOCK ON		Jog lock is active. If you press an axis key, that axis moves at the current jog rate until you press [JOG LOCK] again, or the axis reaches its limit.
REMOTE JOG		The optional remote jog handle is active.
VECTOR JOG		For gimbaled spindle mills, the tool will jog along the vector defined by the position of the spindle rotary positions.
X MIRROR		Mirroring mode (G101) is active in the positive direction. The icon message includes the currently mirrored axes.
AXIS UNCLAMPED		A rotary axis, or a combination of rotary axes, is unclamped. The icon message includes the axes that are currently unclamped.

Control Icons

Name	Icon	Meaning
WARNING LOW VOLTAGE		Power Fault Detect Module (PFDM) Incoming Voltage is under the nominal operating level.
WARNING HIGH VOLTAGE		PFDM Incoming Voltage is above the nominal operating level.
ALARM HIGH VOLTAGE		PFDM Incoming Voltage is above the nominal operating level.
ALARM LOW AIR PRESSURE		System air pressure is critically low.
WARNING LOW AIR PRESSURE		System air pressure is low.
WARNING HIGH AIR PRESSURE		System air pressure is high.
ALARM HIGH AIR PRESSURE		System air pressure is critically high

Control Icon Guide

Name	Icon	Meaning
LOW GEAR BOX OIL FLOW LOW GEAR BOX OIL LEVEL		The spindle gear box oil level is low.
CHECK ROTARY LUBRICATION LEVEL		The rotary table lubrication oil reservoir needs service, or the rotary table brake fluid needs service.
DIRTY TSC FILTER		The Through Spindle Coolant filter needs service.
LOW COOLANT CONCENTRATE		The concentrate reservoir for the coolant refill system needs service.
LOW SPINDLE OIL LEVEL LOW SECOND SPINDLE OIL LOW GREASE LEVEL		The spindle lubrication oil system detected a low oil condition, or the axis ball screw lubrication system detected a low grease or low pressure condition.
LOW ROTARY BRAKE FLUID		The rotary brake fluid reservoir needs service.
MAINTENANCE DUE		A maintenance procedure is due, based on information in the MAINTENANCE page. The maintenance page is part of Current Commands.

Name	Icon	Meaning
EMERGENCY STOP, PENDANT	 1	[EMERGENCY STOP] on the pendant has been pressed. This icon disappears when [EMERGENCY STOP] is released.
EMERGENCY STOP, PALLET	 2	[EMERGENCY STOP] on the pallet changer has been pressed. This icon disappears when [EMERGENCY STOP] is released.
EMERGENCY STOP, TC CAGE	 3	[EMERGENCY STOP] on the tool changer cage has been pressed. This icon disappears when [EMERGENCY STOP] is released.
EMERGENCY STOP, AUXILIARY	 4	[EMERGENCY STOP] on an auxiliary device has been pressed. This icon disappears when [EMERGENCY STOP] is released.
SINGLE BLOCK		SINGLE BLOCK mode is active. The control executes programs (1) block at a time, and you need to press [CYCLE START] to execute the next block.
DRY RUN		DRY RUN mode is active.
OPTION STOP		OPTIONAL STOP is active. The control stops the program at each M01 command.

Control Icon Guide

Name	Icon	Meaning
BLOCK DELETE		BLOCK DELETE is active. The control skips program blocks that begin with a slash (/).
CAGE OPEN		The side-mount tool changer door is open.
TOOL CHANGER MANUAL CCW		The side-mount tool changer carousel is turning counter-clockwise as commanded by a manual carousel rotation button.
TOOL CHANGER MANUAL CW		The side-mount tool changer carousel is turning clockwise as commanded by a manual carousel rotation button.
TOOL CHANGE		A tool change is in progress.
TOOL UNCLAMPED		The tool in the spindle is unclamped.
CONVEYOR FORWARD		The conveyor is active and currently moving forward.

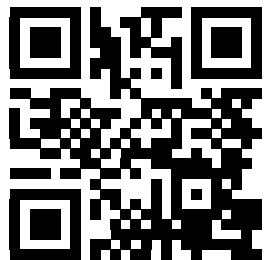
Control Icons

Name	Icon	Meaning
CONVEYOR REVERSE		The conveyor is active and currently moving in reverse.
TSC ON		The Through-Spindle Coolant (TSC) system is active.
TAB ON		The Tool Air Blast (TAB) system is active.
AIR BLAST ON		The Auto Air Gun is active.
COOLANT ON		The main coolant system is active.
COOLANT REFILL ON		The Coolant Refill feature is mixing and adding coolant to the tank.

More Information Online

3.3 More Information Online

For updated and supplemental information, including tips, tricks, maintenance procedures, and more, visit the Haas Resource Center at diy.HaasCNC.com. You can also scan the code below with your mobile device to go directly to the Resource Center:



Chapter 4: Operation

4.1 Machine Power-On

This section tells you how to power-on a machine and establish the axis home positions.

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

The startup screen gives basic instructions to start the machine. Press **[CANCEL]** to dismiss the screen.

2. Turn **[EMERGENCY STOP]** to the right to reset it.
3. Press **[RESET]** to clear the startup alarms. If you cannot clear an alarm, the machine may need service. Contact your Haas Factory Outlet (HFO) for assistance.
4. If your machine is enclosed, close the doors.



WARNING:

*Before you do the next step, remember that automatic motion begins immediately when you press **[POWER UP/RESTART]**. Make sure the motion path is clear. Stay away from the spindle, machine table, and tool changer.*

5. Press **[POWER UP/RESTART]**.



The axes rapid toward their home positions. The axes then move slowly until the machine finds the home switch for each axis. This establishes the machine home position.

The control is now in **OPERATION:MEM** mode. You can now press **[CYCLE START]** to run the active program, or you can use other control functions.

4.2 Spindle Warm-Up

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

Your machine includes a 20-minute warm-up program (002020) in the program list. If you use the spindle at consistent high speeds, you should run this program every day.

4.3 Device Manager

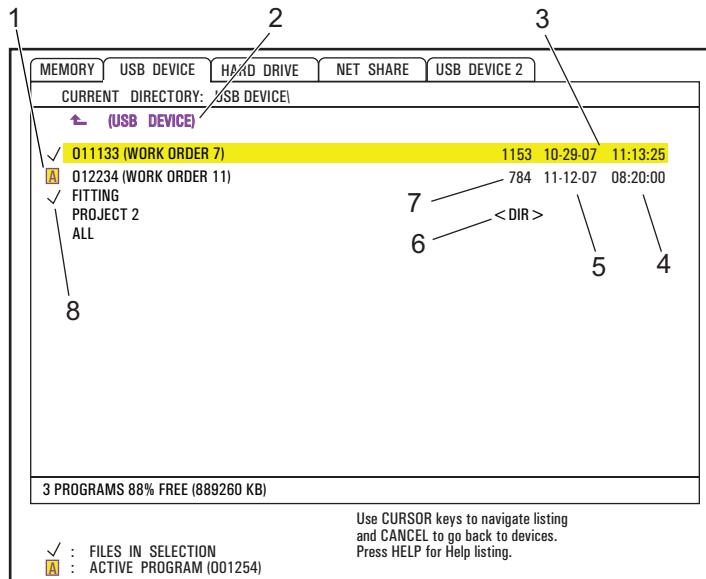
The Device Manager shows you the available memory devices and their contents in a tabbed menu. For information on navigating tabbed menus in the Haas control, refer to page 53.



NOTE: *External USB hard drives must be FAT or FAT32 formatted. Do not use NTFS formatted devices.*

This example shows the directory for the USB device in the device manager.

F4.1: USB Device Menu



1. Active Program
2. Active Directory
3. Highlighted Program
4. Time
5. Date
6. Subdirectory
7. File Size
8. Selected Program

4.3.1 File Directory Systems

Data storage devices such as USB sticks or hard disks usually have a directory structure (sometimes called a “folder” structure), with a root that contains directories and possibly sub-directories, many levels deep. You can navigate and manage directories on these devices in the device manager.

**NOTE:**

The MEMORY tab in the device manager gives a flat list of programs saved in the machine’s memory. There are no further directories in this list.

Navigating Directories

To navigate directories:

1. Highlight the directory you want to open (Directories have a <DIR> designation in the file list). Press **[ENTER]**.
2. To return to the previous directory level, highlight the directory name at the top of the file list. Press **[ENTER]** to go to that directory level.

Directory Creation

You can add directories to the file structure of USB memory devices, hard drives, and your net share directory.

1. Navigate to the device tab and the directory where you want to place your new directory.
2. Type the new directory name and press **[INSERT]**.

The new directory appears in the file list with the <DIR> designation.

4.3.2 Program Selection

When you select a program, it becomes active. The active program appears in the main **EDIT:EDIT** mode window, and it is the program that the control runs when you press **[CYCLE START]** in **OPERATION:MEM** mode.

1. Press **[LIST PROGRAM]** to display the programs in memory. You can also use the tabbed menus to select programs from other devices in the device manager. Refer to page 53 for more information on tabbed menu navigation.
2. Highlight the program you want to select and press **[SELECT PROGRAM]**. You can also type an existing program number and press **[SELECT PROGRAM]**.
The program becomes the active program.
3. In **OPERATION:MEM** mode, you can type an existing program number and press the **[UP]** or **[DOWN]** cursor arrow to quickly change programs.

4.3.3 Program Transfer

You can transfer programs, settings, offsets, and macro variables between machine memory and connected USB, hard drive, or net share devices.

The programs sent to the control from a PC must begin and end with a %.

File Name Convention

Files intended for transfer to and from the machine control should be named with an (8)-character filename and (3)-character extension; for example: `program1.txt`. Some CAD/CAM programs use “.NC” as a file extension, which is also acceptable.

File extensions are for the benefit of PC applications; the CNC control ignores them. You can name program files with no extension, but some PC applications may not recognize the file without the extension.

Files developed in the control are named with the letter “O” followed by 5 digits. For example, `o12345`.

Copying Files

1. Highlight a file and press **[ENTER]** to select it. A check mark appears next to the file name. You can select multiple files this way.
2. If you want to change the name of the file at the destination, type the new name. Skip this step if you do not want to change the name of the file.
3. Press **[F2]**.
4. In the **Copy To** window, use the cursor arrows to select the destination.
5. Press **[ENTER]** to copy the program.

4.3.4 Deleting Programs

**NOTE:**

You cannot undo this process. Be sure to have backups of data that you may want to load on the control again. You cannot press [UNDO] to recover a deleted program.

1. Press [**LIST PROGRAM**] and select the device tab that contains the programs you want to delete.
2. Use the [**UP**] or [**DOWN**] cursor arrows to highlight the program name.
3. Press [**ERASE PROGRAM**].

**NOTE:**

You cannot delete the active program.

4. Press [**Y**] at the prompt to delete the program, or [**N**] to cancel the process.
5. To delete multiple programs:
 - a. highlight each program you want to delete and press [**ENTER**]. This places a check mark next to each program name.
 - b. Press [**ERASE PROGRAM**].
 - c. Answer the **Y/N** prompt for each program.
6. If you want to delete all of the programs in the list, select **ALL** at the end of the list and press [**ERASE PROGRAM**].

**NOTE:**

There are some important programs that may be included with the machine, such as O02020 (spindle warm-up) or macro programs (O09XXX). Save these programs to a memory device or PC before you erase all programs. You can also use Setting 23 to protect O09XXX programs from deletion.

4.3.5 Maximum Number of Programs

The program list in MEMORY can contain up to 500 programs. If the control contains 500 programs and you try to create a new program, the control returns the message **DIR FULL**, and your new program is not created.

Remove some programs from the program list to create new programs.

4.3.6 File Duplication

To duplicate a file:

1. Press **[LIST PROGRAM]** to access the Device Manager.
2. Select the **Memory** tab.
3. Cursor to the program to duplicate.
4. Type a new program name (Onnnnn) and press **[F2]**.
The highlighted program is duplicated with the new name, and it is made the active program.
5. To duplicate a program to a different device, highlight the program and press **[F2]**.
Do not type a program number.
A popup menu lists destination devices.
6. Select a device and press **[ENTER]** to duplicate the file.
7. To copy multiple files, press **[ENTER]** to place a check mark at each file name.

4.3.7 Changing Program Numbers

To change a program number:

1. Highlight the file in LIST PROGRAM mode.
2. Type a new program number in the Onnnnn format.
3. Press **[ALTER]**.

Program Number Change (in Memory)

To change a program number in **MEMORY**:

1. Make the program the active program. Refer to page **76** for more information on the active program.
2. Type the new program number in **EDIT** mode.
3. Press **[ALTER]**.

The program number changes to the name you specified.

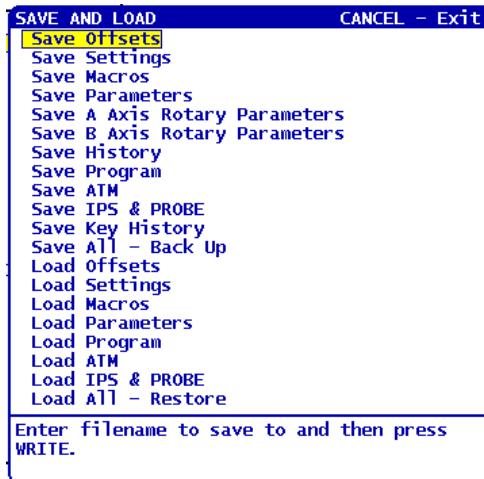
If a program in memory already has the new program number, the control returns the message *Prog exists*. The program number does not change.

4.4 Backing Up Your Machine

The backup function makes a copy of your machine's settings, parameters, programs, and other data so that you can easily restore it in the event of data loss.

You create and load backup files with the **SAVE AND LOAD** pop-up menu. To access the popup menu, press **[LIST PROG]**, then select the **USB**, **Network**, or **Hard Drive** tab, and then press **[F4]**.

F4.2: Save and Load Popup



4.4.1 Making a Backup

The backup function saves your files with a name that you designate. Your designated name gets an associated extension for each data type:

Save File Type	File Extension
Offsets	.OFS
Settings	.SET
Macros - Variables	.VAR
Parameters	.PAR
Parameters - Pallet Positions (Mill)	.PAL
Parameters - Linear Screw Compensation	.LSC

Backing Up Your Machine

Save File Type	File Extension
A Axis Rotary Parameters (Mill)	.ROT
B Axis Rotary Parameters (Mill)	.ROT
History	.HIS
Program	.PGM
ATM - Advanced Tool Management	.ATM
IPS & Probe	.IPS
Key History	.KEY
All - Backup	

To back up the information from your machine:

1. Insert a USB memory device into the USB port on the right side of the control pendant.
2. Select the **USB** tab in the Device Manager.
3. Open the destination directory. If you want to create a new directory for your backup data, refer to page **75** for instructions.
4. Press **[F4]**.
The **Save and Load** popup menu appears.
5. Highlight the option you want.
6. Type a name for the backup. This name is attached to a unique extension for each backup option you picked. Press **[ENTER]**.

The control saves the data you chose, under the name you typed (plus extensions), in the current directory on the USB memory device.

4.4.2 Restoring From a Backup

This procedure tells you how to restore your machine data from the backup on a USB memory device.

1. Insert the USB memory device with the backup files into the USB port on the right side of the control pendant.
2. Select the **USB** tab in the Device Manager.
3. Press **[EMERGENCY STOP]**.
4. Open the directory that contains the files you want to restore.

5. Press **[F4]**.
The **Save and Load** pop-up menu appears.
6. Highlight **Load All - Restore** to load all file types (settings, parameters, programs, macros, tool offsets, variables, etc.)
7. Type the backup name with no extension (e.g., 28012014) that you want to restore, and press **[ENTER]**.
All the files with the typed backup name are loaded on the machine. The message "Disk Done" displays after loading is complete.
8. To load a specific file type (like **name.PAR** for parameters), press **[F4]**, highlight the file type (in this case, **Load Parameters**), type the backup name with no extension, and then press **[ENTER]**.
The file with the typed backup name (in this case **name.PAR**) is loaded on the machine. The message "Disk Done" displays after loading is complete.

4.5 Basic Program Search

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

1. Type the text you want to find in the active program.
2. Press the **[UP]** or **[DOWN]** cursor arrow key.

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

4.6 RS-232

RS-232 is one way to connect the Haas CNC control to a computer (PC). This feature lets you upload and download programs, settings, and tool offsets from a PC.

You need a 9-pin to 25-pin null modem cable (not included) or a 9-pin to 25-pin straight Through cable with a null modem adapter 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. Plug the 25-pin connector end into the connector on the Haas machine located on the side panel of the control cabinet at the back of the machine.



NOTE: *Haas Automation does not supply null modem cables.*

4.6.1 Cable Length

This table lists baud rate and the respective maximum cable length.

T4.1: Cable Length

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

4.6.2 Machine Data Collection

Machine Data Collection lets you extract a Q command through the RS-232 port (or with an optional hardware package). Setting 143 enables the feature. It is a software-based feature that requires an additional computer to request, interpret, and store data from the control. The remote computer can also set certain Macro variables.

Data Collection Using the RS-232 Port

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

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

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

- *CSV response* is Comma Separated Variables, one or more data variables separated by commas.
- *ETB* (0x17) marks the end of the data. This control character is for the remote computer.
- *CR/LF* tells the remote computer the 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 >. These commands are available:

T4.2: Remote Q Commands

Command	Definition	Example
Q100	Machine Serial Number	>Q100 SERIAL NUMBER, 3093228
Q101	Control Software Version	>Q101 SOFTWARE, VER M18.01
Q102	Machine Model Number	>Q102 MODEL, VF2D
Q104	Mode (LIST PROG, MDI, etc.)	>Q104 MODE, (MEM)
Q200	Tool Changes (total)	>Q200 TOOL CHANGES, 23
Q201	Tool Number in use	>Q201 USING TOOL, 1
Q300	Power-on Time (total)	>Q300 P.O. TIME, 00027:50:59
Q301	Motion Time (total)	>Q301 C.S. TIME, 00003:02:57
Q303	Last Cycle Time	>Q303 LAST CYCLE, 000:00:00
Q304	Previous Cycle Time	>Q304 PREV CYCLE, 000:00:00
Q402	M30 Parts Counter #1 (resettable at control)	>Q402 M30 #1, 553
Q403	M30 Parts Counter #2 (resettable at control)	>Q403 M30 #2, 553
Q500	Three-in-one (PROGRAM, Oxxxxx, STATUS, PARTS, xxxx)	>Q500 STATUS, BUSY
Q600	Macro or system variable	>Q600 801 MACRO, 801, 333.339996

You can request the contents of any macro or system variable with the **Q600** command; for example, **Q600 xxxx**. This shows the contents of macro variable **xxxx** on the remote computer. In addition, macro variables #1–33, 100–199, 500–699 (note that variables #550–580 are unavailable if the mill has a probing system), 800–999 and #2001 through #2800 can be written to using an **E** command, for example, **Exxxxx yyyy.yyyyyy** where **xxxxx** is the macro variable and **yyyy.yyyyyy** is the new value.

**NOTE:**

Use this command only when there are no alarms.

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 through 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.

These machine statuses are available:

- E-STOP contacts. This will be closed when the [**EMERGENCY STOP**] 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 program, 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

4.7 File Numerical Control (FNC)

You can run a program directly from its place on your network or from a storage device, such as a USB drive. From the Device Manager screen, highlight a program on the selected device and press **[SELECT PROGRAM]**.

You can call subprograms in an FNC program, but those subprograms must be in the same file directory as the main program.

If your FNC program calls G65 macros or aliased G/M subprograms, they must be in **MEMORY**.



CAUTION: *You can edit subprograms while the CNC program runs. Be careful when you run an FNC program that might have changed since the last time it ran.*

4.8 Direct Numerical Control (DNC)

Direct Numerical Control (DNC) is a way to load a program into the control through the RS-232 port. You can also run the program as the control receives it. Because the control runs the program while it receives the program, there is no limit to the size of the CNC program.

F4.3: DNC Waiting and Received Program

PROGRAM (DNC)	N00000000	PROGRAM (DNC)	N00000000
<pre>WAITING FOR DNC . . .</pre> <p>DNC RS232</p>		<pre>001000 ; (G-CODE FINAL QC TEST CUT) ; (MATERIAL IS 2x8x8 6061 ALUMINUM) ; ; (MAIN) ; ; M00 ; (READ DIRECTIONS FOR PARAMETERS AND SETTINGS) ; (FOR VF-SERIES MACHINES W/ETH AXIS CARDS) ; (USE / FOR HS, 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>	

Direct Numerical Control (DNC)

T4.3: Recommended RS-232 Settings for DNC

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



CAUTION: You should run DNC with XMODEM or parity enabled. This allows the system to detect transmission errors and stop the machine before it crashes.

The data transmission settings must be the same in the CNC control and the computer. To change the

1. [SETTING/GRAFIC] and scroll to the RS-232 settings (or enter 11 and press the up or down arrow).
2. Use the [UP] and [DOWN] cursor arrows to highlight the variables and the left and right arrows to change the values.
3. Press [ENTER] to confirm a selection.
4. DNC is selected by pressing [MDI/DNC] twice. 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.
5. 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.
6. Start sending the program to the control before [CYCLE START] is pushed. After the message *DNC Prog Found* shows, Press [CYCLE START].

4.8.1 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 does (1) block (command) at a time. Each block is 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 with the **G102** command or **DPRNT** to output axis coordinates back to the controlling computer. Refer to page **298**.

4.9 Tooling

This section describes tool management in the Haas control: commanding tool changes, loading tools into holders, and Advanced Tool Management.

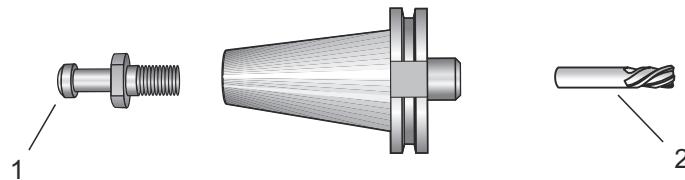
4.9.1 Tool Holders

There are several different spindle options for Haas mills. Each of these types requires a specific tool holder. The most common spindles are 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 in a given machine are capable of holding only one tool type.

Tool Holder Care

1. Make sure that tool holders and pull studs are in good condition and tightened together securely or they may stick in the spindle.

F4.4: Tool Holder Assembly, 40-Taper CT Example: [1] Pull Stud, [2] Tool (Endmill).



2. Clean the tool holder taper body (the part that goes into the spindle) with a lightly oiled rag to leave a film, which helps prevent rusting.

Pull Studs

A pull stud (sometimes called a retention knob) secures 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. Refer to the 30, 40, and 50-taper spindle and tooling information on the Haas Resource Center website for descriptions of the pull studs you need.



CAUTION: *Do not use 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.*

4.9.2 Advanced Tool Management Introduction

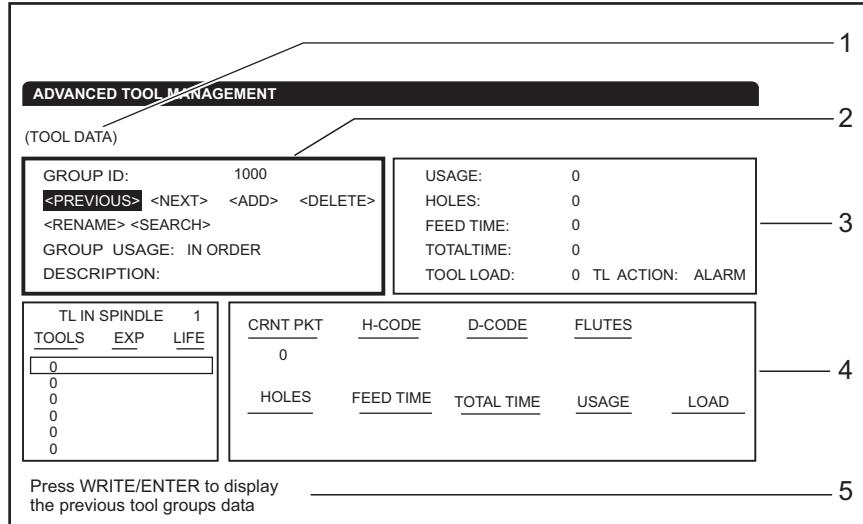
Advanced Tool Management (ATM) lets you set up groups of duplicate tools for the same or a series of jobs.

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

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

The ATM page is located in the Current Commands mode. Press **[CURRENT COMMANDS]**, and then **[PAGE UP]** until you see the ATM screen.

F4.5: Advanced Tool Management Window: [1] Active window label, [2] Tool group window, [3] Allowed limits window, [4] Tool data window, [5] Help text



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] completes the deletion; answering [N] cancels 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.

Tooling

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. You can type zero clear the tool, or highlight the tool number and press **[ORIGIN]** to 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, press **[*]** (**[SHIFT]**, then **[1]**). To remove an obsolete tool (indicated with an asterisk), 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 that contains the highlighted tool.

H-Code (Tool Length) – You cannot edit the H-code unless Setting 15 is set to **OFF**. To change an H-code (if allowed), type a number and press **[ENTER]**. The number entered corresponds to the tool number in the tool offsets display.

D-Code (Tool Diameter) – To change a D-code, type a number and press **[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. To edit this, type a new number and press **[ENTER]**. This is the same as the **Flutes** column listed on the tool offsets page.

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

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

Highlight the Holes or Load field, and then press [**ORIGIN**] to clear their values. To change the values, highlight the value you want to change, type a new number, and press [**ENTER**].

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

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

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

Tool Group Setup

To add a tool group:

1. Press [**F4**] until the Tool Group window is outlined.
2. Use the cursor keys to highlight <**ADD**>.
3. Enter a number between 1000 and 2999 (this will be the group ID number).
4. Press [**ENTER**].
5. To change a group ID number, highlight <**RENAME**>.
6. Enter a new number.
7. Press [**ENTER**].

Tool Group Usage

You must set up a tool group before you run a program with ATM. To use a tool group in a program:

1. Set up a tool group.
2. Substitute the tool group ID number for the tool number and for the H-codes and D-codes in the program. Refer to this program for an example of the new programming format. Be sure to have your work offsets adjusted properly to avoid Alarms (316, 317, 318) due to X, Y, or Z being commanded to move farther than your machine is capable of moving.

```
%  
O30001 (Tool change ex-prog);  
(G54 X0 Y0 is top right corner of part) ;  
(Z0 is on top of the part) ;  
(Group 1000 is a drill) ;  
(T1000 PREPARATION BLOCKS) ;  
T1000 M06 (Select tool group 1000) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
X0 Y0 (Rapid to 1st position) ;
```

```
S1000 M03 (Spindle on CW) ;
G43 H1000 Z0.1 (Tool group offset 1000 on) ;
M08 (Coolant on) ;
(T1000 CUTTING BLOCKS) ;
G83 Z-0.62 F15. R0.1 Q0.175 (Begin G83) ;
X1.115 Y-2.75 (2nd hole) ;
X3.365 Y-2.87 (3rd hole) ;
G80 ;
(T1000 COMPLETION BLOCKS) ;
G00 Z1. M09 (Rapid retract, coolant off) ;
G53 G49 Z0 M05 (Z home, spindle off) ;
M01 (Optional stop) ;
(T2000 PREPARATION BLOCKS)
T2000 M06 (Select tool group 2000) ;
G00 G90 G40 G49 G54 (Safe startup) ;
G00 G54 X0.565 Y-1.875 (Rapid to 4th position) ;
S2500 M03 (Spindle on CW) ;
G43 H2000 Z0.1 (Tool group offset 2000 on) ;
M08 (Coolant on) ;
(T2000 CUTTING BLOCKS) ;
G83 Z-0.62 F15. R0.1 Q0.175 (Begin G83) ;
X1.115 Y-2.75 (5th hole) ;
X3.365 Y2.875 (6th hole) ;
(T2000 COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%
```

Advanced Tool Management Macros

Tool Management can use macros to obsolete a tool within a tool group. Macros 8001 to 8200 represent tools 1 through 200. You can set one of these macros to 1 to expire a tool. For 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 get information about a tool group. If you specify a tool group ID number with macro 8500, the control returns the tool group information in macro variables #8501 through #8515.

Refer to the variables #8500-#8515 in the Macros section for information about macro variable data labels.

Macro variables #8550-#8564 enable a G-code program to get information about individual tools. When you specify an individual tool ID number with macro #8550, the control returns 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.

1. The information can be saved, either as part of an overall backup program by using the **[LIST PROGRAM]/ Save/Load window ([F4])**.
When the Advanced Tool Management data is saved as part of an overall backup, the system creates a separate file with a .ATM extension.
2. The ATM data can be saved and restored via the RS-232 port by pressing **[SEND]** and **[RECEIVE]** while the Advanced Tool Management screen is displayed.

4.10 Tool Changers

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

1. Make sure the machine is zero returned. If it is not, press **[POWER UP/RESTART]**.
2. Use **[TOOL RELEASE]**, **[ATC FWD]**, and **[ATC REV]** to manually command the tool changer. There are (2) tool release buttons; one on the spindle head cover and another on the keyboard.

4.10.1 Loading the Tool Changer



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



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



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

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



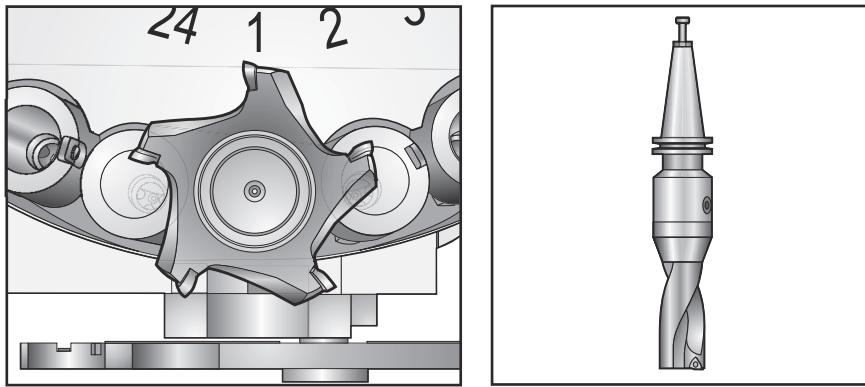
CAUTION: *Tools that make a loud noise when released indicate a problem and should be checked before serious damage occurs to the tool changer or spindle.*

Tool Loading for a Side-Mount Tool Changer

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

1. Make sure your tool holders have the correct pull stud type for the mill.
2. Press **[CURRENT COMMANDS]**. Press **[PAGE UP]** or **[PAGE DOWN]** until you see the **POCKET TOOL TABLE**.
3. Clear the "Large" or "Heavy" tool designations from the pocket tool table. Use the cursor keys to scroll to a tool pocket with an **L** or **H** next to it. Press **[SPACE]**, then **[ENTER]** to clear the designation. To clear all designations, press **[ORIGIN]** and select the **CLEAR CATEGORY FLAGS** option.

F4.6: A Large and Heavy Tool (left), and a Heavy (not Large) Tool (right)



4. Press **[ORIGIN]** to reset the tool pocket table to default values. This places tool 1 in the spindle, tool 2 in pocket 1, tool 3 in pocket 2, etc. This clears the previous tool pocket table settings, and it resets the tool pocket table for the next program. You can also press **[ORIGIN]** and select **SEQUENCE ALL POCKETS** to reset the tool pocket table.



NOTE:

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

5. Determine if your 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 your program does not need large tools, skip to Step 7.
6. Organize the tools to match your 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 **[ENTER]**.



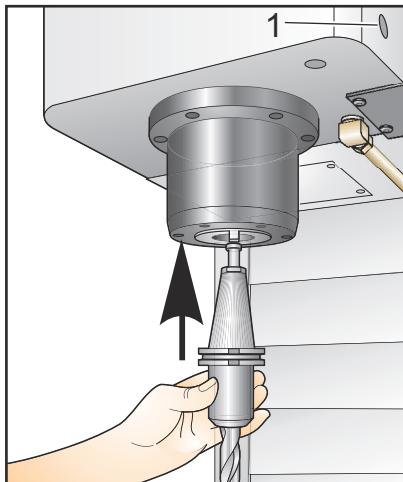
CAUTION:

You cannot place a large tool 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. Insert tool 1 (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 and press the tool release button. When the tool is fitted into the spindle, release the tool release button.

Tool Changers

F4.7: Inserting a Tool Into the Spindle: [1] Tool release button.



High-Speed Side-Mount Tool Changer

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

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

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

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

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.

Using '0' for a Tool Designation

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

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

Moving Tools in the Carousel

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



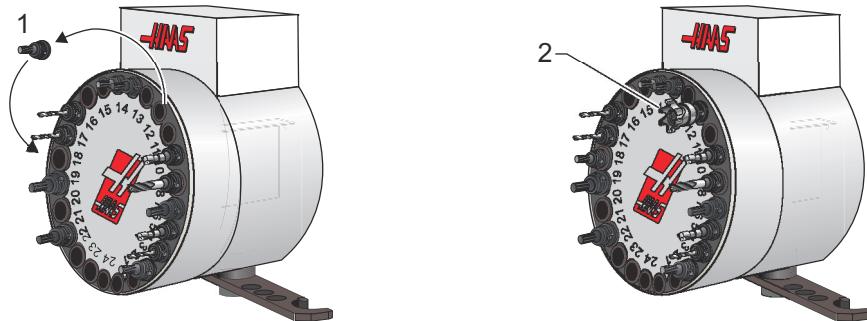
CAUTION:

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

Moving Tools

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

F4.8: Making Room for Large Tools: [1] Tool 12 to Pocket 18, [2] Large tool in Pocket 12.



1. Select **MDI** mode. Press [**CURRENT COMMANDS**] and scroll to the **POCKET TOOL TABLE** display. Identify the tool number that is in pocket 12.
2. Type **Tnn** (where nn is the tool number from step 1). Press [**ATC FWD**]. This places the tool from pocket 12 into the spindle.
3. Type **P18** then press [**ATC FWD**] to put the tool in the spindle into pocket 18.
4. Scroll to pocket 12 in the tool pocket table and press **L**, [**ENTER**] to designate that pocket as large.

Tool Changers

5. Enter the tool number into **SPNDL** (spindle) on the **tool pocket table**. Insert the tool into the spindle.



NOTE:

Extra-large tools can be programmed as well. An “extra-large” tool is one that takes up three pockets; the diameter of the tool covers the tool pocket on either side of the pocket it is installed in. Have your HFO change Parameter 315:3 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.

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

Umbrella Tool Changer

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. Press **[MDI/DNC]** for MDI mode.
3. Organize the tools to match to the CNC program.
4. Take tool in hand and insert the tool (pull stud first) into the spindle. Turn the tool so that the two cutouts in the tool holder line up with the tabs of the spindle. Push the tool upward while pressing the Tool Release button. When the tool is fitted into the spindle, release the Tool Release button.
5. Press **[ATC FWD]**.
6. Repeat Steps 4 and 5 with the remaining tools until all the tools are loaded.

4.10.2 Umbrella Tool Changer Recovery

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



WARNING:

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

1. Press **[EMERGENCY STOP]**.
2. Remove the cause of the jam.
3. Press **[RESET]** to clear the alarms.
4. Press **[RECOVER]** and follow the directions to reset the tool changer.

4.10.3 SMTCA Programming Notes

Tool Pre-Call

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

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

Program commands that stop look-ahead:

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

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

```
T01 M06 (TOOL CHANGE) ;  
T02 (PRE-CALL THE NEXT TOOL) ;
```

4.10.4 SMTCA Recovery

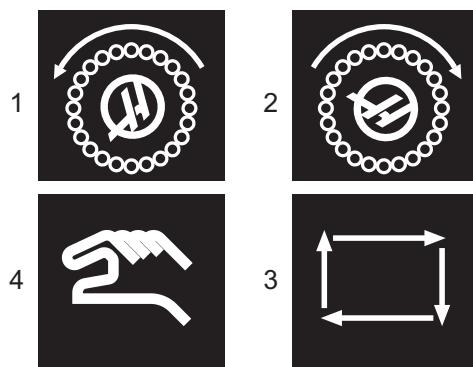
If a problem occurred during a tool change, a tool changer recovery needs to be performed. Enter the tool changer recovery mode by:

1. Press [RECOVER]. The control will first attempt an automatic recovery.
2. 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.
3. Press [M] to continue.
4. 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.

4.10.5 SMTc Door Switch Panel

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

- F4.9:** Tool Changer Door Switch Panel Symbols: [1] Rotate Tool changer Carousel Counter-Clockwise, [2] Rotate Tool Changer Carousel Clockwise, [3] Tool Change Switch - Automatic Operation, [4] Tool Change Switch - Manual Operation Selection.



SMTc Door Operation

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

If the switch is turned to manual while a tool change is in progress, current tool changer motion is completed. The next tool change will not execute until the switch is turned back to automatic. Any machining operations that are in progress remain uninterrupted.

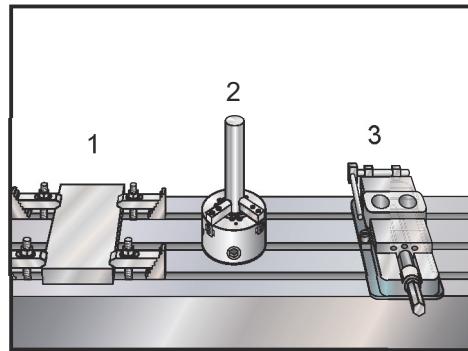
The carousel rotates one position whenever a clockwise or counter-clockwise button is pressed once, while the switch is set to manual.

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

4.11 Part Setup

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

F4.10: Part Setup Examples: [1] Toe clamp, [2] Chuck, [3] Vise.



4.11.1 Setting Offsets

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

To manually enter offsets:

1. Choose one of the offsets pages.
2. Move the cursor to the desired column.
3. Type the offset value you want to use.
4. Press **[ENTER]** or **[F1]**.

The value is entered into the column.

5. Enter a positive or negative value and press **[ENTER]** to add the amount entered to the number in the selected column; press **[F1]** to replace the number in the column.

Jog Mode

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

To enter jog mode:

1. Press **[HANDLE JOG]**.
2. Press the desired axis (**[+X]**, **[-X]**, **[+Y]**, **[-Y]**, **[+Z]**, **[-Z]**, **[+A/C]** or **[-A/C]**, **[+B]**, or **[-B]**).

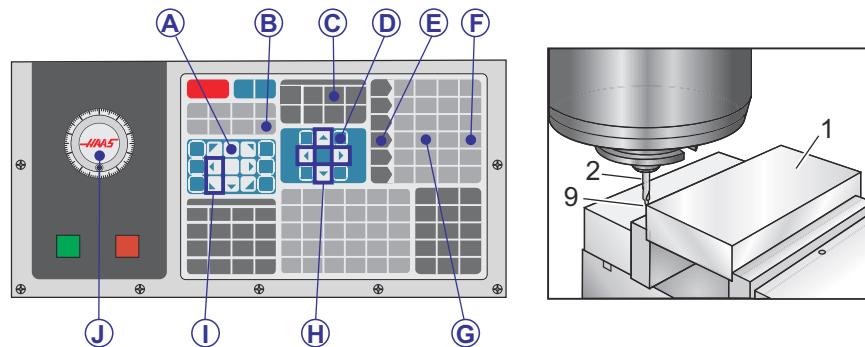
Part Setup

3. There are different increment speeds that can be used while in jog mode; they are [.0001], [.001], [.01] and [.1]. Each click of the jog handle moves the axis the distance defined by the current jog rate. You can also use an optional Remote Jog Handle (RJH) to jog the axes.
4. Press and hold the handle jog buttons or use the jog handle control to move the axis.

Setting Part Zero Offset

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

F4.11: Part Zero Set



1. Place the material [1] in the vise and tighten.
2. Load a pointer tool [2] in the spindle.
3. Press **[HANDLE JOG]** [E].
4. Press **[.1/100.]** [F] (The mill will move at a fast speed when the handle is turned).
5. Press **[+Z]** [A].
6. Handle jog [J] the Z-Axis approximately 1" above the part.
7. Press **[.001/1.]** [G] (The mill will move at a slow speed when the handle is turned).
8. Handle jog [J] the Z-Axis approximately 0.2" above the part.
9. Select between the X and Y axes [I] and handle jog [J] the tool to the upper left corner of the part (See illustration [9]).
10. Press **[OFFSET]** [C] until the Active Work Offset pane is active.
11. Cursor [H] to G54 X-Axis column.

**CAUTION:**

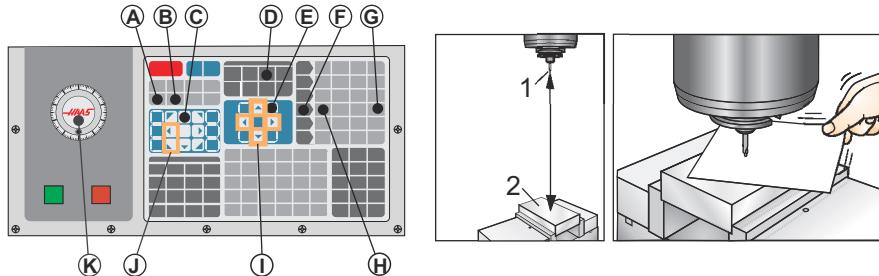
In the next step, do not press [PART ZERO SET] a third time; this loads a value into the Z Axis column. This causes a crash or a Z-Axis alarm when the program is run.

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

Setting 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.

- F4.12:** Setting Tool Offset. With the Z Axis at its home position, Tool Length Offset is measured from the tip of the tool [1] to the top of the part [2].



1. Load the tool in the spindle [1].
2. Press [HANDLE JOG] [F].
3. Press [.1/100.] [G] (The mill moves at a fast rate when the handle is turned).
4. Select between the X and Y axes [J] and handle jog [K] the tool near the center of the part.
5. Press [+Z] [C].
6. Handle jog [K] the Z Axis approximately 1" above the part.
7. Press [.0001/.1] [H] (The mill moves 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 [OFFSET] [D].
10. Press [PAGE UP] [E] until you display the **Program Tool Offsets** window. Scroll to tool #1.
11. Cursor [I] to Geometry for position #1.
12. Press [TOOL OFFSET MEASURE] [A].



CAUTION: *The next step causes the spindle to move rapidly in the Z Axis.*

13. Press **[NEXT TOOL]** [B].
14. Repeat the offset process for each tool.

Additional Tooling Setup

There are other tool setup pages within the Current Commands.

1. Press **[CURRENT COMMANDS]** and then use **[PAGE UP]/[PAGE DOWN]** to scroll to these pages.
2. The first is the page with Tool Load at the top of the page. You can add a tool load limit. The control references these values and can be set to do a specific action should the limitations be reached. Refer to Setting 84 (page 373) for more information on tool limit actions.
3. 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.

4.12 Features

Haas operation features:

- Graphics Mode
- Dry-Run Operation
- Background Edit
- Axis Overload Timer

4.12.1 Graphics Mode

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

The Graphics display has a number of available features:

- **Key Help Area** The lower left of the graphics display pane is the function key help area. Function keys that are currently available are displayed here with a brief description of their usage.
- **Locater Window** The lower right part of the pane displays the whole table area and indicates where the tool is currently located during simulation.
- **Tool Path Window** In the center of the display is a large window that represents a view of the work area. It displays a cutting tool icon and tool paths during a graphics simulation of the program.

**NOTE:**

Feed motion is displayed as fine continuous lines. Rapid moves are displayed as dotted lines. Setting 4 disables the dotted-line display. The places where a drilling canned cycle is used are marked with an X. Setting 5 disables the X display.

- **Adjusting Zoom** Press **[F2]** to display a rectangle (zoom window) indicating the area to be magnified. Use **[PAGE DOWN]** to decrease the size of the zoom window (zooming in), and use **[PAGE UP]** to increase the size of the zoom window (zooming out). Use the Cursor Arrow keys to move the zoom window to the desired location and press **[ENTER]** to complete the zoom and rescale the tool path window. The locator window (small view at the bottom right) shows the entire table with an outline of where the Tool Path window is zoomed. Tool Path window is cleared when zoomed, and the program must be re-run to view the tool path. Press **[F2]** and then **[HOME]** to expand the Tool Path window to cover the entire work area.
- **Z-Axis Part Zero Line** The horizontal line on the Z-axis bar at the top-right corner of the graphics screen indicates the position of the current Z-axis work offset plus the length of the current tool. While a program is running, the shaded portion of the bar indicates the depth of Z-axis motion. You can watch the position of the tool tip relative to the Z-axis part zero position as the program runs.
- **Control Status** The lower left portion of the screen displays control status. It is the same as the last four lines of all other displays.
- **Position Pane** The position pane displays axes locations just as it would during a live part run.
- **Simulation Speed** **[F3]** decreases simulation speed and **[F4]** increases simulation speed.

Graphics mode is run from Memory, MDI, DNC, FNC, or Edit modes. To run a program:

1. Press **[SETTING/GRAFIC]** until the **GRAFICS** page is displayed. Or press **[CYCLE START]** from the active program pane in Edit mode to enter Graphics mode.
2. To run DNC in Graphics mode, press **[MDI/DNC]** until DNC mode is active, then go to **GRAFICS** page and send the program to the machine's control (See the DNC section).
3. Press **[CYCLE START]**.

**NOTE:**

Not all machine functions or motions are simulated in graphics.

4.12.2 Dry Run Operation



CAUTION: *The machine executes all motions exactly as programmed. Do not use a work piece in the machine while dry run is operating.*

The Dry Run function is used to check a program quickly without actually cutting parts. To select Dry Run:

1. While in MEM or MDI mode, press [**DRY RUN**].
When in Dry Run, all rapids and feeds are run at the speed selected with the jog speed buttons.
2. Dry Run can only be turned on or off when a program has finished or [**RESET**] is pressed. Dry Run makes all of the commanded X Y Z moves and requested tool changes. The override keys can be used to adjust the Spindle speeds.



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.*

4.12.3 Axis Overload Timer

When a spindle or an axes current load is 180% load, a timer starts and displays in the **POSITION** pane. The timer starts at 1.5 minutes and counts down to zero. An axis overload alarm **SERVO OVERLOAD** displays when the time has expired to zero.

4.13 Running Programs

Once a program is loaded on the machine and the offsets are set, to run the program:

1. Press [**CYCLE START**].
2. It is suggested that you run the program in Dry Run or Graphics mode before doing any cutting.

4.14 Run-Stop-Jog-Continue

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

1. Press [**FEED HOLD**].
Axis motion stops. The spindle continues to turn.

-
2. Press [X], [Y] or [Z], then press [**HANDLE JOG**]. The control stores the current X, Y, and Z positions.

**NOTE:**

You can jog only the X, Y, and Z Axes in this mode.

3. The control gives the message *Jog Away*. Use the jog handle or jog keys to move the tool away from the part. You can command coolant with [**AUX CLNT**] or [**COOLANT**]. You can start or stop the spindle with [**CW**], [**CCW**], or [**STOP**]. You can also release the tool to change inserts.

**CAUTION:**

When you start the program again, the control uses the previous offsets for the return position. Therefore, it is unsafe and not recommended to change tools and offsets when you interrupt a program.

4. Jog to a position as close as possible to the stored position, or to a position where there is an unobstructed rapid path back to the stored position.
5. Press [**MEMORY**] or [**MDI/DNC**] to return to run mode. The control continues only if you return to the mode that was in effect when you stopped the program.
6. Press [**CYCLE START**]. The control gives the message *Jog Return* and rapids X and Y at 5% to the position where you pressed [**FEED HOLD**]. It then returns the Z Axis. If you press [**FEED HOLD**] during this motion, axis motion pauses and the control gives the message *Jog Return Hold*. Press [**CYCLE START**] to resume the Jog Return motion. The control goes into a feed hold state again when the motion is finished.

**CAUTION:**

The control does not follow the same path that you used to jog away.

7. Press [**CYCLE START**] again and the program resumes operation.

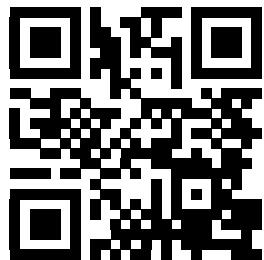
**CAUTION:**

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

More Information Online

4.15 More Information Online

For updated and supplemental information, including tips, tricks, maintenance procedures, and more, visit the Haas Resource Center at diy.HaasCNC.com. You can also scan the code below with your mobile device to go directly to the Resource Center:



Chapter 5: Programming

5.1 Numbered Programs

To create a new program:

1. Press [LIST PROGRAM] to enter the program display and the list of programs mode.
2. Enter a program number (Onnnnn) and press [SELECT PROGRAM] or [ENTER].



NOTE:

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

If the program exists, the control sets it as the active program (refer to page 76 for more information on the active program). If it does not yet exist, the control creates it and sets it as the active program.

3. Press [EDIT] to work with the new program. A new program has only the program number and an end-of-block character (semicolon).

5.2 Program Editors

The Haas control features (3) different program editors: The MDI Editor, the Advanced Editor, and the FNC Editor.

5.2.1 Basic Program Editing

This section describes the basic program editing controls. For information on more advanced program editing functions, refer to page 112.

F5.1: Edit Program Screen Example

EDIT: EDIT	EDITOR
ACTIVE PROGRAM - 099997	
<pre>099997 ; (HAAS VQC Mill, English, Inch, V1.4A) ; (11/14/01) ; ; N100 ; (CATEGORY) ; (NAME G73 HIGH SPEED PECK DRILLING) ; ; N101 ; (TEMPLATE) ; (NAME G73 High Speed Peck Drill Using Q, 1-Hole) ;</pre>	

Program Editors

1. You write or make changes to programs in an active **EDIT:EDIT** or **EDIT:MDI** window.
 - a. To edit a program in MDI, press **[MDI/DNC]**. This is **EDIT:MDI** mode.
 - b. To edit a numbered program, select it, then press **[EDIT]**. This is **EDIT:EDIT** mode. Refer to page **76** to learn how to select a program.
2. To highlight code in Edit mode:
 - a. Use the cursor arrow keys or the **[HANDLE JOG]** control to highlight a single piece of code. That code appears with white text on a black background.
 - b. If you want to highlight an entire block or multiple blocks of code, press **[F2]** at the program block where you want to start, then use the cursor arrow keys or the **[HANDLE JOG]** control to move the cursor arrow (>) to the first or last line you want to highlight. Press **[ENTER]** or **[F2]** to highlight all of that code. Press **[CANCEL]** to exit data selection.
3. To add code to the program in Edit mode:
 - a. Highlight the code that your new code will go in front of.
 - b. Type the code that you want to add to the program.
 - c. Press **[INSERT]**. Your new code appears in front of the block you highlighted.
4. To replace code in Edit mode:
 - a. Highlight the code you want to replace.
 - b. Type the code you want to replace the highlighted code with.
 - c. Press **[ALTER]**. Your new code takes the place of the code you highlighted.
5. To remove characters or commands in Edit mode:
 - a. Highlight the text you want to delete.
 - b. Press **[DELETE]**. The code you highlighted is removed from the program.



NOTE:

*The control saves programs in **MEMORY** as you enter each line. To save programs in **USB**, **HD**, or **Net Share**, refer to the **Haas Editor (FNC)** section on page **121**.*

6. Press **[UNDO]** to reverse up to the last (9) changes.

5.2.2 Background Edit

Background Edit allows you to edit a program while another program runs.

1. Press **[EDIT]** until the background edit pane (Inactive Program) on the right side of the screen is active.
2. Press **[SELECT PROGRAM]** to select a program to background edit (the program must be in Memory) from the list.
3. Press **[ENTER]** to begin background editing.
4. To select a different program to background edit, press **[SELECT PROGRAM]** from the background edit pane and choose a new program from the list.
5. 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 **[PROGRAM]**.
6. **[CYCLE START]** may not be used while in Background Edit. If the program contains a programmed stop (M00 or M01), exit Background Edit (press **[PROGRAM]**) 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 [PROGRAM]) keyboard input will return to the M109 in the running program.

5.2.3 Manual Data Input (MDI)

Manual Data Input (MDI) lets you command automatic CNC moves without a formal program. Your input stays on the MDI input page until you delete it.

F5.2: MDI Input Page Example

MDI	N000000000
G97 S1000 M03 ; G00 X2. Z0.1 ; G01 X1.8 Z-1. F12 ; X1.78 ; X1.76 ; X1.75 ; ;	

1. Press **[MDI/DNC]** to enter **MDI** mode.
2. Type program commands in the window. Press **[CYCLE START]** to execute the commands.

Program Editors

3. If you want to save the program you created in MDI as a numbered program:
 - a. Press **[HOME]** to place the cursor at the beginning of the program.
 - b. Type a new program number. Program numbers must follow standard program number format (Onnnnn).
 - c. Press **[ALTER]**.

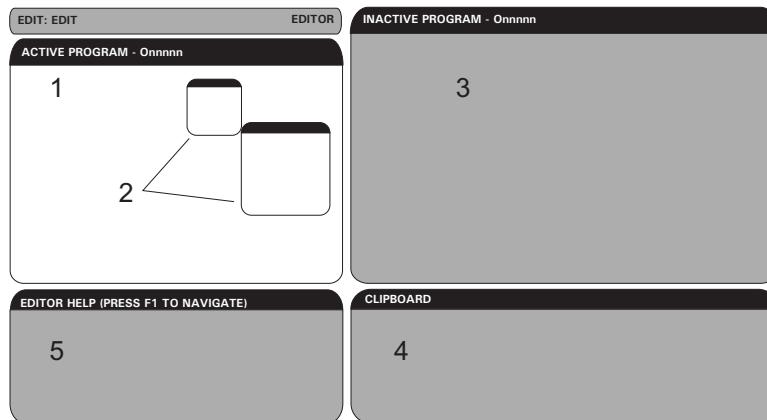
The control saves your program to memory and clears the MDI input page. You can find the new program in the **MEMORY** tab in the Device Manager menu (press **[LIST PROGRAM]**).

- 4. Press **[ERASE PROGRAM]** to delete everything from the MDI input page.

5.2.4 Advanced Editor

The advanced editor allows you to edit programs using pop-up menus.

F5.3: Advanced Editor Display: [1] Active Program Pane, [2] Pop-up menus, [3] Inactive Program Pane, [4] Clipboard, [5] Context-sensitive help messages.



1. Press **[EDIT]** to enter edit mode.
2. Two editing panes are available; an active program pane and an inactive program pane. Press **[EDIT]** to switch between the two panes.
3. Press **[SELECT PROGRAM]**.
The active window lists programs in memory with the active program marked with an asterisk (*) before the name.
4. To edit a program, type the program number (Onnnnn) or select it from the program list, and press **[SELECT PROGRAM]**.
The program opens in the active window.
5. Press **[F4]** to open another copy of that program in the inactive program pane if there is not a program there already.

6. You can also select a different program for the inactive program pane. Press **[SELECT PROGRAM]** from the inactive program pane and select the program from the list.
7. Press **[F4]** to exchange the programs between the two panes (make the active program inactive and vice versa).
8. Use the jog handle or cursor keys to scroll through the program code.
9. Press **[F1]** to access the pop-up menu.
10. Use the **[LEFT]** and **[RIGHT]** cursor arrows to select from the topic menu (HELP, MODIFY, SEARCH, EDIT, PROGRAM), and use the **[UP]** and **[DOWN]** cursor arrows or the jog handle to select a function.
11. Press **[ENTER]** to execute a command from the menu.

**NOTE:**

A context-sensitive help pane in the lower left provides information on the currently selected function.

12. Use **[PAGE UP]/[PAGE DOWN]** to scroll through the help message. This message also lists hot keys that you can use for some functions.

The Advanced Editor Pop-up Menu

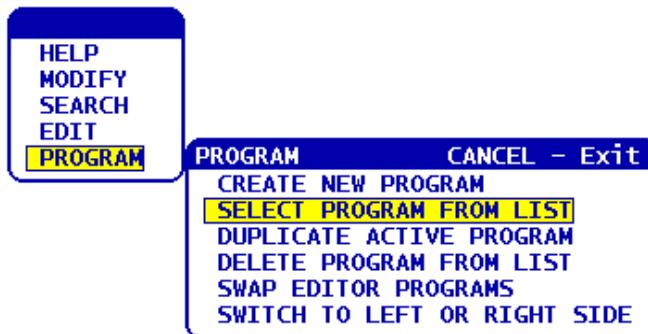
The pop-up menu provides easy access to editor functions in 5 categories: **HELP**, **MODIFY**, **SEARCH**, **EDIT**, and **PROGRAM**. This section describes each category and the options available when you select it.

Press **[F1]** to access the menu. Use the **[LEFT]** and **[RIGHT]** cursor arrows to select from the list of categories, and the **[UP]** and **[DOWN]** cursor arrows to select a command in the category list. Press **[ENTER]** to execute the command.

The Program Menu

The program menu provides options for program creation, deletion, naming, and duplicating, as described in the basic program editing section.

F5.4: The Advanced Editor Program Menu



Create New Program

1. Select the **CREATE NEW PROGRAM** command from the **PROGRAM** pop-up menu category. The letter O is supplied on the INPUT: field.
2. Type a program number (nnnnn) that is not already in the program directory.
3. Press **[ENTER]** to create the program.

Select Program From List

1. Press **[F1]**.
2. Select the **SELECT PROGRAM FROM LIST** command from the **PROGRAM** pop-up menu category.
When you select this menu item, a list appears of programs in the control memory.
3. Highlight the program you want to select.
4. Press **[ENTER]**.

Duplicate Active Program

1. Select the **DUPLICATE ACTIVE PROGRAM** command from the **PROGRAM** pop-up menu category.
2. At the prompt, type a new program number (Onnnnn) and press **[ENTER]** to create the program.

Delete Program From List

1. Select the **DELETE PROGRAM FROM LIST** command from the **PROGRAM** pop-up menu category.
When you select this menu item, a list appears of programs in the control memory.
2. Highlight a program, or highlight **ALL** to select all programs in memory for deletion.
3. Press **[ENTER]** to delete the selected programs.

Swap Editor Programs

This menu option puts the active program in the inactive program pane and the inactive program in the active program pane.

1. Select the **SWAP EDITOR PROGRAMS** command from the **PROGRAM** pop-up menu category.
2. Press **[ENTER]** to swap the programs.
3. You can also press **[F4]** to do this.

Switch to Left or Right Side

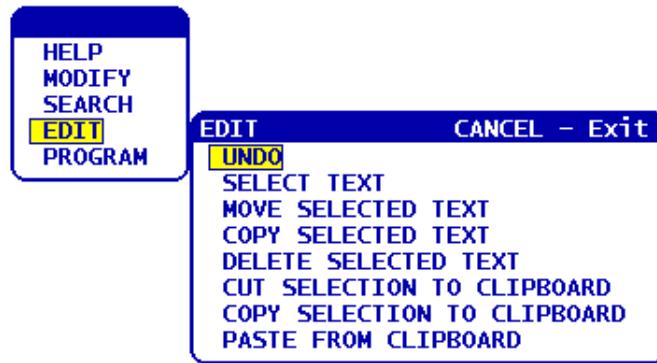
This switches edit control between the active and inactive program. Inactive and active programs remain in their respective panes.

1. Select the **SWITCH TO LEFT OR RIGHT SIDE** command from the **PROGRAM** pop-up menu.
2. Press **[ENTER]** to switch between the active and inactive programs.

The Edit Menu

The edit menu provides advanced edit options over the quick-edit functions described in the basic program editing section.

F5.5: Advanced Edit Pop-up Menu



Undo

Reverses the last edit operation, up to the last 9 edit operations.

1. Press **[F1]**. Select the **UNDO** command from the **EDIT** pop-up menu category.
2. Press **[ENTER]** to undo the last edit operation. You can also use the hot key - **[UNDO]**.

Select Text

This menu item will select lines of program code:

1. Select the **SELECT TEXT** command from the **EDIT** pop-up menu category.
2. Press **[ENTER]** or use the hot key - **[F2]** to set the start point of the text selection.
3. Use the cursor keys, **[HOME]**, **[END]**, **[PAGE UP]** / **[PAGE DOWN]**, or the jog handle to scroll to the last line of code to be selected.
4. Press **[F2]** or **[ENTER]**.

The selected text is highlighted, and you can now move, copy, or delete it.

5. To deselect the block, press **[UNDO]**.

Move Selected Text

After you select a section of text, you can use this menu command to move it to another part of the program.

1. Move the cursor (>) to the program line where you want to move the selected text.
2. Select the **MOVE SELECTED TEXT** command from the **EDIT** pop-up menu category.
3. Press **[ENTER]** to move the selected text to the point after the cursor (>).

Copy Selected Text

After you select a section of text, you can use this menu command to copy it to another location in your program.

1. Move the cursor (>) to the program line where you want to copy the selected text.
2. Select the **COPY SELECTED TEXT** command from the **EDIT** pop-up menu category.
3. Press **[F2]** or **[ENTER]** to copy the selected text to the point after the cursor (>).
4. Hot Key - Select the text, position the cursor, and press **[ENTER]**.

Delete Selected Text

To delete selected text:

1. Press **[F1]**. Select the **DELETE SELECTED TEXT** command from the **EDIT** pop-up menu category.
2. Press **[F2]** or **[ENTER]** to delete the selected text to the point after the cursor (>).
If no block is selected, the currently highlighted item is deleted.

Cut Selection to Clipboard

After you select a section of text, you can use this menu command to remove it from the program and place it in the clipboard.

1. Select the **CUT SELECTION TO CLIPBOARD** command from the **EDIT** pop-up menu category.
2. Press **[F2]** or **[ENTER]** to cut the selected text.

The selected text is removed from the current program and placed in the clipboard. This replaces any content in the clipboard.

Copy Selection To Clipboard

After you select a section of text, you can use this menu command to place a copy of the text in the clipboard.

1. Select the **COPY SELECTION TO CLIPBOARD** command from the **EDIT** pop-up menu category.
2. Press **[ENTER]** to copy the selected text to the clipboard.

The selected text is placed in the clipboard. This replaces any content in the clipboard. The text is not removed from the program.

Paste From Clipboard

To copy the contents of the clipboard to the line after the cursor position:

1. Move the cursor (>) to the program line where you want to insert the clipboard text.
2. Select the **PASTE FROM CLIPBOARD** command from the **EDIT** pop-up menu category.
3. Press **[ENTER]** to insert the clipboard text at the point after the cursor (>).

The Search Menu

The search menu provides advanced search options over the quick-search function described in the basic program editing section.

F5.6: Advanced Search Popup



Find Text

To search for text or program code in the current program:

1. Select the **FIND TEXT** command from the **SEARCH** pop-up menu category.
2. Type the text you want to find.
3. Press **[ENTER]**.
4. Press **[F]** to search for your text below the cursor position. Press **[B]** to search above the cursor position.

The control searches your program in the direction you specified, then it highlights the first occurrence of your search term found. If your search returns no results, the message *NOT FOUND* appears in the system status bar.

Find Again

This menu option allows you to quickly repeat your last **FIND** command. This is a quick way to continue searching the program for more occurrences of a search term.

1. Select the **FIND AGAIN** command in the **SEARCH** pop-up menu category.
2. Press **[ENTER]**.

The control searches again, from the current cursor position, for the last search term you used, in the same direction that you specified.

Find And Replace Text

This command searches the current program for specific text or program, and replaces each occurrence (or all) with different text.

1. Press **[F1]**. Select the **FIND AND REPLACE TEXT** command in the **SEARCH** pop-up menu category.
2. Type your search term.
3. Press **[ENTER]**.

4. Type the text with which you want to replace the search term.
5. Press [ENTER].
6. Press [F] to search for the text below the cursor position. Press [B] to search above the cursor position.
7. When the control finds each occurrence of the search term, it gives the 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.

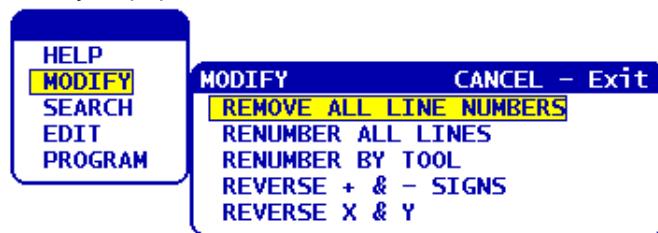
Choose **All** to automatically replace all occurrences of the search term.

Choose **Cancel** to back out of the function without making changes (text already replaced will remain so if you choose this option).

The Modify Menu

The modify menu category contains functions for quick changes to the entire program.

F5.7: Advanced Modify Popup



Remove All Line Numbers

This command automatically removes all unreferenced N-code line numbers from the edited program. If you have selected a group of lines (refer to page 116), this command affects only those lines.

1. Select the **REMOVE ALL LINE NUMBERS** command from the **MODIFY** pop-up menu category.
2. Press [ENTER].

Renumber All Lines

This command numbers all of the blocks in the program. If you have selected a group of lines (refer to page 116), this command affects only those lines.

1. Select the **RENUMBER ALL LINES** from the **MODIFY** pop-up menu category.
2. Enter the starting N-code number.
3. Press **[ENTER]**.
4. Enter the N-code increment.
5. Press **[ENTER]**.

Renumber By Tool

This command searches the program for T (tool) codes, highlights all the program code up to the next T-code, and rennumbers the N-code (line numbers) in the program code.

1. Select the **RENUMBER BY TOOL** command from the **MODIFY** pop-up menu category.
2. For each T-code found, answer the prompt *Renumber (Yes/No/All/Cancel)?* If you answer **[A]**, the process will continue as if you pressed Y for each T-code. The prompt will not appear again during this operation.
3. Enter the starting N-code number.
4. Press **[ENTER]**.
5. Enter the N-code increment.
6. Press **[ENTER]**.
7. Answer *Resolve outside references (Y/N)?* with **[Y]** to change outside code (like GOTO line numbers) with the proper number, or **[N]** to ignore outside references.

Reverse + and - Signs

This menu item reverses the signs of the numeric values in a program. Be cautious with this function if the program contains a G10 or G92 (refer to the G-code section for a description).

1. Select the **REVERSE + & - SIGNS** command from the **MODIFY** pop-up menu category.
2. Enter the letter address code of the value you want to change.
X, Y, Z, etc.



NOTE: *D, F, G, H, L, M, N, O, P, Q, S, and T address codes are not allowed.*

3. Press [**ENTER**].

Reverse X and Y

This feature changes the letter X in the program to the letter Y, and the letter Y to the letter X. Effectively switching the X values to Y values and the Y values to X values.

1. Select the **REVERSE X & Y** command from the **MODIFY** pop-up menu category.
2. Press [**ENTER**].

5.2.5 The File Numerical Control (FNC) Editor

The FNC Editor does the same functions as the Advanced Editor, along with new features to enhance program development on the control, including multiple-document viewing and editing.

In general, you use the Advanced Editor with programs in MEM, while you use the FNC Editor with programs on drives other than MEM (ie., HDD, USB, and Net Share). Refer to Basic Program Editing on page **109** and Advanced Editor on page **112** for information on those editors.

To save a program after editing with the FNC Editor:

1. Press [**SEND**] when prompted.
2. Wait for the program to finish writing to the drive.

Loading a Program (FNC)

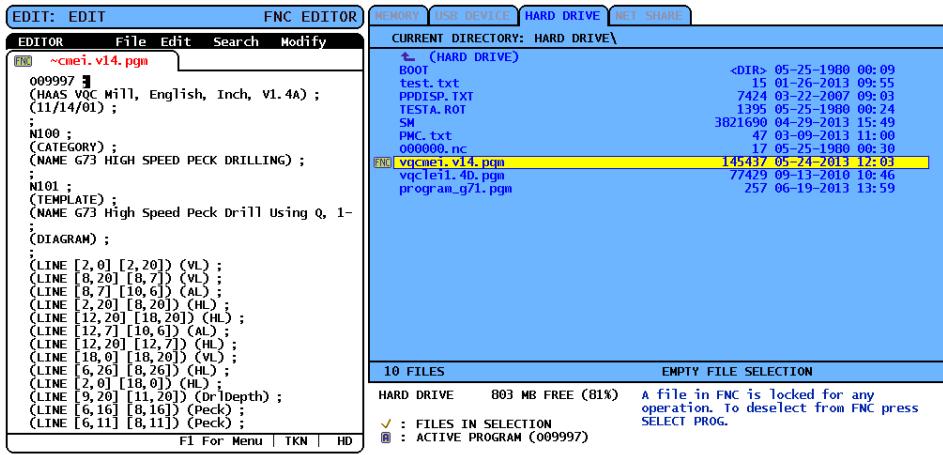
To load a program:

1. Press [**LIST PROGRAM**].
2. Highlight a program in the **USB**, **HARD DRIVE**, or **NET SHARE** tab of the **LIST PROGRAM** window.
3. Press [**SELECT PROGRAM**] to make it the active program (in the FNC Editor, programs open in FNC, but are editable).
4. With the program loaded, press [**EDIT**] to shift focus to the program edit pane.

The initial display mode shows the active program on the left, and the program list on the right.

Program Editors

F5.8: Edit: Edit Display



Menu Navigation (FNC)

To access the menu.

1. Press [F1].
2. Use the left and right cursor arrow keys or jog handle to move between the menu categories, and use the [UP] and [DOWN] cursor arrows to highlight an option within a category.
3. Press [ENTER] to make a menu selection.

Display Modes (FNC)

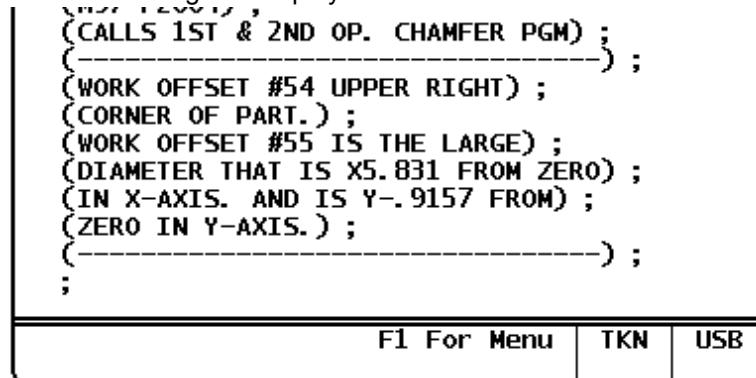
Three display modes are available. To switch between display modes:

1. Press [F1] for the popup File menu.
2. Use the Change View command.
3. Press [ENTER].
4. List displays the current FNC program alongside the tabbed LIST PROG menu.
5. 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]).
6. 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 [F1] popup File menu or by pressing [F4].

Display Footer (FNC)

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.

F5.9: Footer Section of Program Display



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 (FNC)

You can open up to three programs simultaneously in FNC Editor. To open an existing program while another program is open in FNC 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 PROGRAM]**. 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 (FNC)

To display line numbers independent of the program text:

1. Select the **Show Line Numbers** command from the File menu to display them.



NOTE:

These are not the same as Nxx line numbers; they are only for reference when viewing the program.

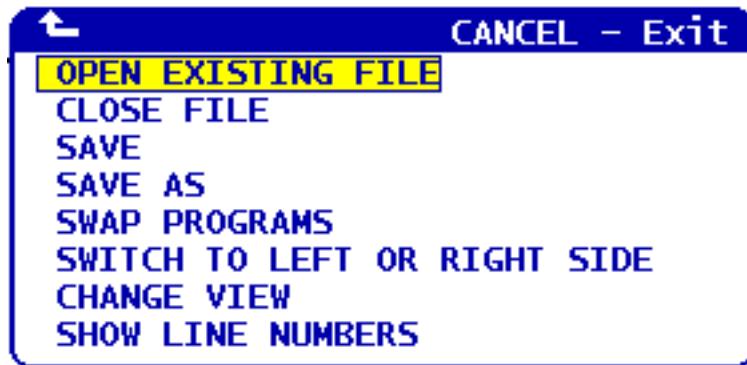
2. To hide the line numbers, reselect the option in the File menu.

File Menu (FNC)

To access the File menu:

1. When in FNC EDITOR mode, press **[F1]**.
2. Select the File menu.

F5.10: File Menu



Open Existing File

When in FNC EDITOR mode,

1. Press **[F1]** and select the File menu.
2. Select Open Existing File.
3. Press up or down on the cursor button to move to the file. Press **[SELECT PROGRAM]**.

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

Close File

When in FNC EDITOR mode,

1. Press **[F1]** and select the File menu.
2. Select Close File.

Closes the current active file. If the file has been modified, the control will prompt to save before the file is closed.

Save



NOTE:

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

Hot Key: **[SEND]** (after a change is made)

1. Press **[F1]** and select the File Menu.
2. Select **Save**.

Saves the current active file under the same filename.

Save As

When in FNC EDITOR mode,

1. Press **[F1]** and navigate to the File Menu.
2. Select Save As.

Saves the current active file under a new filename. Follow prompts to name the file.
Displays in new tab.

Swap Programs

When in FNC EDITOR mode and in a tabbed stack of programs, use Hot Key: **[F4]** or,

1. Press **[F1]** and select the File menu.
2. Select Swap Programs.

Displays the next program in a tabbed pane at the top of the tab stack.

Switch to Left or Right Side

To change the active program window (the currently active window has a white background) in FNC EDITOR mode and in a tabbed stack of programs:

1. Press **[F1]** or use Hot Key: **[EDIT]**.
2. If you pressed **[F1]**, move the cursor to the File menu and select Switch to Left or Right Side.

Change View

When in FNC EDITOR mode, use Hot Key: **[PROGRAM]** or,

1. Press **[F1]** and select the File menu.
2. Select Change View.

Switches between List, Main, and Split view modes.

Show Line Numbers

When in FNC EDITOR mode,

1. Press **[F1]** and select the File menu.
2. Select Show Line Numbers.

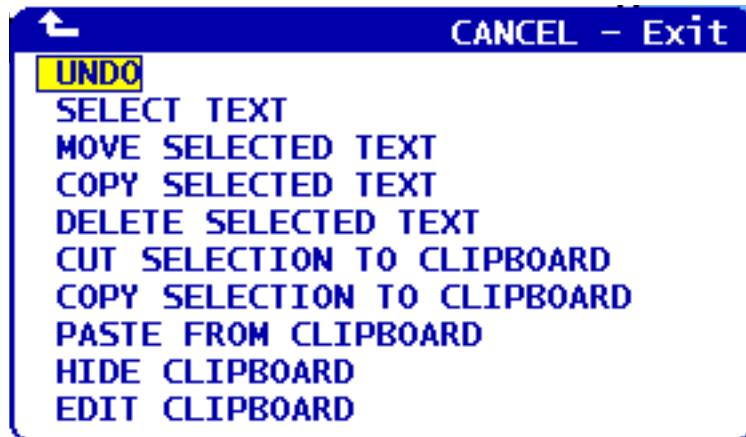
Displays reference-only line numbers independent of program text. They are never saved as part of the program like Nxx numbers would be. Select the option again to hide the line numbers.

Edit Menu (FNC)

To access the Edit menu:

1. When in FNC EDITOR mode, press **[F1]**.
2. Move the cursor to the Edit menu.

F5.11: Edit Menu



Undo

Reverses changes made to the active program in FNC EDITOR mode:



NOTE:

Block and global functions cannot be undone.

1. Press **[F1]**.
2. Select the **EDIT** menu, then select **UNDO**.

Select Text

Highlights a block of text in FNC EDITOR mode:

1. Before you choose this menu option or use the hot key **[F2]**, position the cursor at the first line of the block to select.
2. Press **[F2]** (hot key) or press **[F1]**.
3. If the hot key is used, skip to step 4. Otherwise, move the cursor to the **EDIT** menu and choose **SELECT TEXT**.
4. Use the cursor arrows or jog handle to make the selection area.
5. Press **[ENTER]** or **[F2]** to highlight the block.

Move/Copy/Delete Selected Text

Removes selected text from its current location and places it after the cursor position (Hot Key: **[ALTER]**), places selected text after the cursor position without it being deleted from its current location (Hot Key: **[INSERT]**), or removes the selected text from the program (Hot Key: **[DELETE]**) in FNC EDITOR mode:

1. Before you choose this menu option or use Hot Keys: **[ALTER]**, **[INSERT]**, or **[DELETE]**, position the cursor at the line above where you want to paste the selected text. **[DELETE]** removes the selected text and closes up the program listing.
2. If you did not use the hot keys, press **[F1]**.
3. Move the cursor to the Edit menu and select Move Selected Text, Copy Selected Text, or Delete Selected Text.

Cut/Copy Selection to Clipboard

Removes the selected text from the current program and moves it to the clipboard or places the selected text in the clipboard without removing it from the program in FNC EDITOR mode:



NOTE:

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

1. Press **[F1]**.
2. Move the cursor to the Edit menu and select Cut Selection to Clipboard or Copy Selection to Clipboard.

Paste from Clipboard

Places the clipboard contents after the cursor location in FNC EDITOR mode:



NOTE:

Does not delete the clipboard contents.

1. Before choosing this menu option, move the cursor to the line you want the clipboard contents to follow.
2. Press **[F1]**.
3. Move the cursor to the Edit menu and select Paste from Clipboard.

Hide/Show Clipboard

Hides the clipboard to view the position or timers and counters display in its place or to restore the clipboard display in FNC EDITOR mode:

1. Press **[F1]**.
2. Move the cursor to the Edit menu and select Show Clipboard. To hide the clipboard, repeat this with the menu changed to Hide Clipboard.

Edit Clipboard

To make adjustments to the clipboard contents in FNC EDITOR mode:



NOTE:

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

1. Press **[F1]**.
2. Move the cursor to the Edit menu and select Edit Clipboard.
3. When finished, press **[F1]**, move the cursor to the Edit menu, and select Close Clipboard.

Search Menu (FNC)

Accesses the Search menu:

1. When in FNC EDITOR mode, press **[F1]**.
2. Move the cursor to the Search menu.

F5.12: Search Menu



Find Text

Defines a search term, search direction, and locates the first occurrence of the search term in the direction indicated in FNC EDITOR mode:

1. Press **[F1]**.
2. Move the cursor to the Search menu and select Find Text.
3. Enter the search text.
4. Enter the search direction. 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

Locates the next occurrence of the search term in FNC EDITOR mode:

1. Press **[F1]**.
2. Move the cursor to the Search menu and select Find Again.
3. Select this function immediately after a “Find Text” search. Repeat to continue to the next occurrence.

Find and Replace Text

Defines a search term, a term to replace it with, the search direction, and chooses Yes/No/All/Cancel in FNC EDITOR mode:

1. Press **[F1]**.
2. Move the cursor to the Search menu and select Find and Replace Text.
3. Enter text to locate.
4. Enter replacement text.
5. Enter the search direction. When you choose a search direction, press F to search for the term below the cursor position, and press B to search above the cursor position.
6. When the first occurrence of the search term is found, the control will prompt *Replace (Yes/No/All/Cancel)?*. Type the first letter of the selection to continue. If **Yes** or **No** is chosen, the editor will execute the 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 changes (text that is already replaced will remain if this option is chosen).

Find Tool

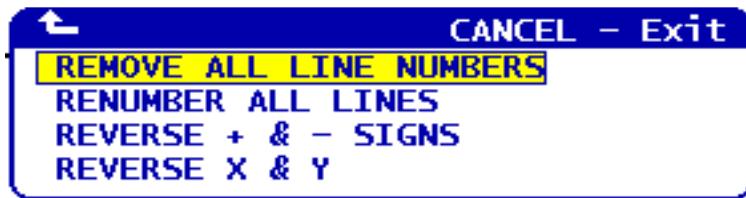
Searches the program for tool numbers in FNC EDITOR mode:

1. Press **[F1]**.
2. Move the cursor to the Search menu and select Find Tool.
3. Select it again to locate the next tool number.

Modify Menu (FNC)

Accesses the Modify menu:

1. When in FNC EDITOR mode, press **[F1]**.
2. Move the cursor to the Modify menu.

F5.13: Modify Menu**Remove All Line Numbers**

Removes all Nxx line numbers from the program in FNC EDITOR mode:

1. Press **[F1]**.
2. Move the cursor to the Modify menu and select **Remove All Line Numbers**.

Renumber All Lines

Renumbers all program lines with Nxx codes in FNC EDITOR mode:

1. Press **[F1]**.
2. Move the cursor to the Modify menu and select **Renumber All Lines**.
3. Choose a starting number.
4. Choose a line number increment.

Reverse + and - Signs

Changes all positive values to negative or negative to positive in FNC EDITOR mode:

1. Press **[F1]**.
2. Move the cursor to the Modify menu and select **Reverse + and - Signs**.
3. Enter the address code(s) to change the value. Letter addresses not allowed: D, F, G, H, L, M, N, O, P, Q, S, and T.

Reverse X and Y

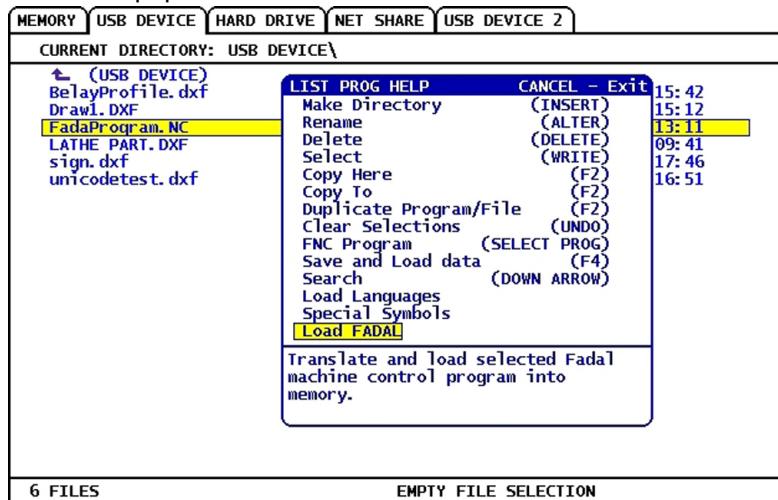
To change all X values to Y values and vice versa in FNC EDITOR mode:

1. Press **[F1]**.
2. Cursor to the Modify menu and select Reverse X and Y.

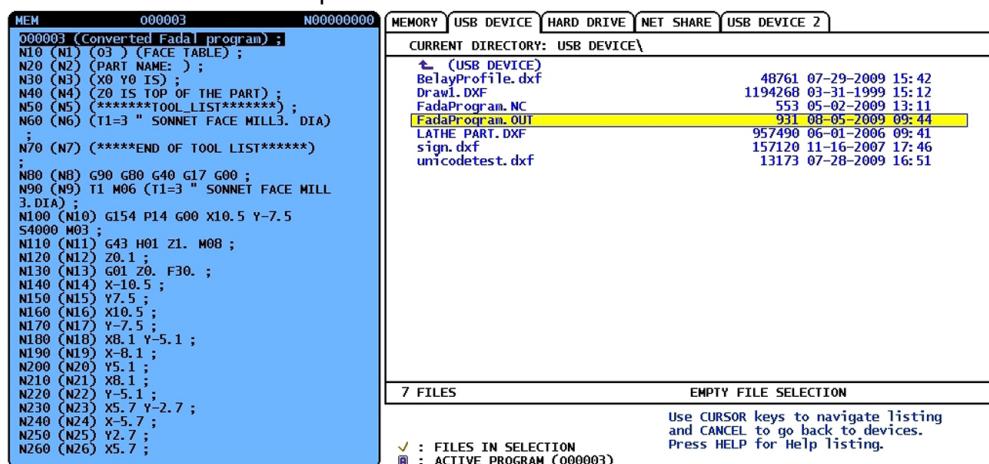
5.3 Fadal Program Converter

If you need to convert a program from Fadal format to Haas, The Fadal Program Converter lets you do so quickly.

F5.14: Load FADAL Popup



F5.15: Fadal Conversion Complete



F5.16: Fadal Conversion Errors

MEM	008686	N00000210
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 ;		

JUSER GENERATED ALARM
NOT SUPPORTED: P. 01

1. Press **[LIST PROGRAM]** to access the converter.
2. Highlight the Fadal program.
3. Press **[F1]**.
4. Select **Load FADAL** from the pop-up menu.

The control loads the converted program into memory. A copy of the converted program is also saved to the current storage device, with an “.out” extension. This file has *Converted Fadal Program* at the top to confirm 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. Edit these lines for Haas compatibility.

**TIP:**

You can use the search function in **EDIT** mode to quickly find unconverted lines. With the converted program in the active pane (press **[PROGRAM]** to change the active pane), press **[F1]** or **[HELP]** and select **Search** from the pop-up menu. Use **M199** as the search term.

5.4 Program Optimizer

This feature lets you override spindle speed, axis feed, and coolant positions (for a mill) in a program, while the program runs. Once the program is finished, the Program Optimizer highlights the program blocks that you changed and then lets you make the changes permanent or revert to the original values.

You can type comments into the input line and press **[ENTER]** to save your input as program notes. You can see the Program Optimizer during a program run by pressing **[F4]**.

5.4.1 Program Optimizer Operation

To go to the Program Optimizer screen:

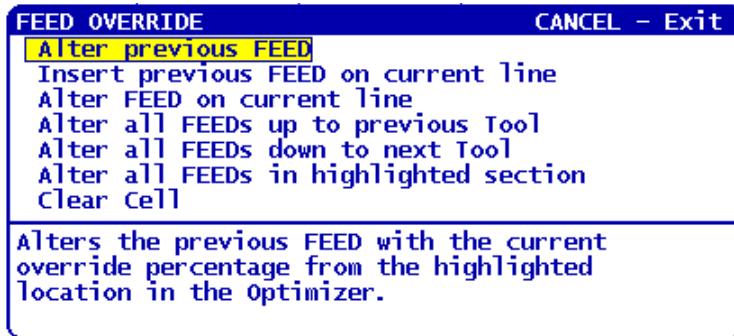
1. At the end of a program run, press **[MEMORY]**.
2. Press **[F4]**.

DXF File Importer

3. Use the right/left and up/down arrows, [PAGE UP]/[PAGE DOWN] and [HOME]/[END] to scroll through **Overrides** and **Notes** columns.
4. On the column topic to edit, press [ENTER].

A pop-up window appears with selections for that column. The programmer can make a number of changes using the commands in the menu.

F5.17: Program Optimizer Screen: Feed Override Popup Example (Mill Screen Shown)

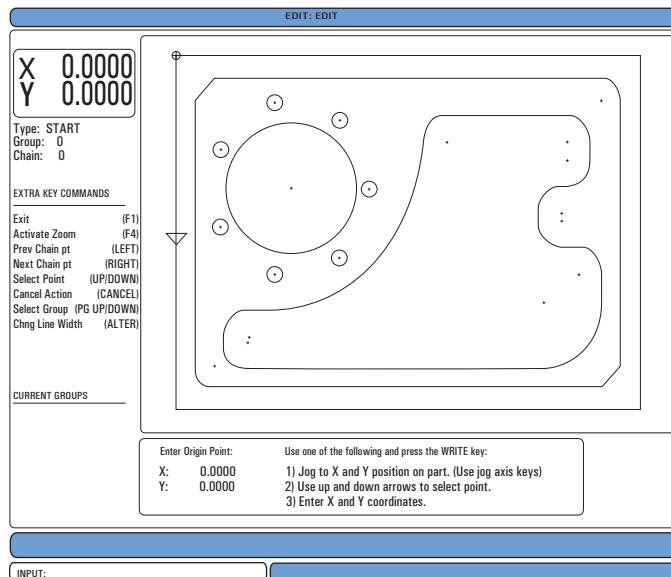


5. In addition, you can highlight a section of code (place the cursor at the start of your selection, press [F2], scroll to the end of your selection and press [F2]). Go back to the Program Optimizer (press [EDIT]) and press [ENTER] to alter all the feeds or speeds in the highlighted section.

5.5 DXF File Importer

This feature can quickly build a G-code program from a .dxf file.

F5.18: DXF File Import



The DXF importer feature gives on-screen help throughout the process. As you complete each step, the text turns green in the step outline box. After you have completed a toolpath, you can put it into any program in memory. The DXF importer can identify and automatically do repetitive tasks. It also automatically combines long contours.

**NOTE:**

Your machine must have the Intuitive Programming System (IPS) option to use the DXF importer.

1. Set up the tools in IPS. Select a .dxf file.
2. Press **[F2]**.
3. Select **[MEMORY]** and press **[ENTER]**. The control recognizes the .dxf file and imports it into the editor.

5.5.1 Part Origin

Use one of these methods to set the part origin.

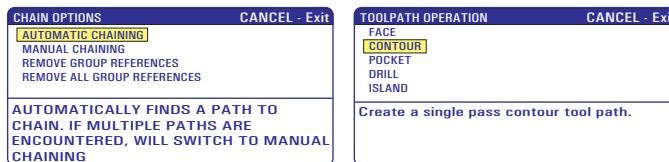
- Point Selection
- Jogging
- Enter Coordinates

1. Use the jog handle or cursor arrows to highlight a point.
2. Press **[ENTER]** to accept the highlighted point as the origin. The control uses this point to set the work coordinate information of the raw part.

5.5.2 Part Geometry Chain and Group

This step finds the geometry of the shape(s). The auto chaining function finds most part geometry. If the geometry is complex and branches off, a prompt displays so you can select one of the branches. The automatic chaining continues after you select a branch. DXF importer groups together holes for drilling and tapping operations.

F5.19: DXF Import Chain/Group Menus



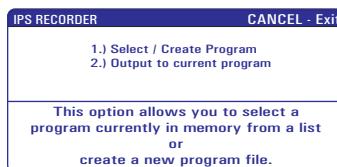
1. Use the jog handle or cursor arrows to choose the toolpath starting location.
2. Press **[F2]** to open the dialog box.

3. Choose the option that best suits the desired application. Most times, the Automatic Chaining function is the best choice, because it automatically plots the toolpath for a part feature.
4. Press **[ENTER]**. This changes the color of that part feature and adds a group to the register under **Current group** on the left side of the window.

5.5.3 Toolpath Selection

This step applies a toolpath to a particular chained group.

F5.20: DXF IPS Recorder Menu



1. Select the group and press **[F3]** to choose a toolpath.
2. Use the jog handle to bisect an edge of the part feature. The control uses this as an entry point for the tool.

After you select a toolpath, you see the IPS (Intuitive Programming System) template for that path.

Most IPS templates are filled with reasonable defaults based on the tools and materials that you set up.

3. Press **[F4]** to save the toolpath once the template is completed. You can add the IPS G-code segment to a program, or create a new program. Press **[EDIT]** to return to the DXF import feature to create the next toolpath.

5.6 Basic Programming

A typical CNC program has (3) parts:

1. **Preparation:** This portion of the program selects the work and tool offsets, selects the cutting tool, turns on the coolant, sets spindle speed, and selects absolute or incremental positioning for axis motion.
2. **Cutting:** This portion of the program defines the tool path and feed rate for the cutting operation.
3. **Completion:** This portion of the program moves the spindle out of the way, turns off the spindle, turns off the coolant, and moves the table to a position from where the part can be unloaded and inspected.

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



NOTE:

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

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

5.6.1 Preparation

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

Preparation Code Block	Description
%	Denotes the beginning of a program written in a text editor.
O40001 (Basic program) ;	O40001 is the name of the program. Program naming convention follows the Onnnnn format: The letter “O”, or “o” is followed by a 5-digit number.
(G54 X0 Y0 is top right corner of part) ;	Comment
(Z0 is on top of the part) ;	Comment
(T1 is a 1/2" end mill) ;	Comment
(BEGIN PREPARATION BLOCKS) ;	Comment
T1 M06 (Select tool 1) ;	Selects tool T1 to be used. M06 commands the tool changer to load Tool 1 (T1) into the spindle.
G00 G90 G17 G40 G49 G54 (Safe startup) ;	This is referred to as a safe startup line. It is good machining practice to place this block of code after every tool change. G00 defines axis movement following it to be completed in Rapid Motion mode. G90 defines axis movements following it to be completed in absolute mode (refer to page 140 for more information). G17 defines the cutting plane as the XY plane. G40 cancels Cutter Compensation. G49 cancels tool length compensation. G54 defines the coordinate system to be centered on the Work Offset stored in G54 on the Offset display.
X0 Y0 (Rapid to 1st position) ;	X0 Y0 commands the table to move to the position X = 0.0 and Y = 0.0 in the G54 coordinate system.
S1000 M03 (Spindle on CW) ;	M03 turns the spindle on in a clockwise direction. It takes the address code Snnnn, where nnnn is the desired spindle RPM. On machines with a gearbox, the control automatically selects high gear or low gear, based on the commanded spindle speed. You can use an M41 or M42 to override this. Refer to page 340 for more information on these M-codes.

Preparation Code Block	Description
G43 H01 Z0.1 (Tool offset 1 on) ;	G43 H01 turns on Tool Length Compensation +. The H01 specifies to use the length stored for Tool 1 in the Tool Offset display. Z0.1 commands the Z Axis to Z=0.1.
M08 (Coolant on) ;	M08 commands the coolant to turn on.

5.6.2 Cutting

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

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

5.6.3 Completion

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

Completion Code Block	Description
G00 Z0.1 M09 (Rapid retract, Coolant off) ;	G00 commands the axis motion to be completed in rapid motion mode. Z0.1 Commands the Z Axis to Z = 0.1. M09 commands the coolant to turn off.
G53 G49 Z0 M05 (Z home, Spindle off) ;	G53 defines axis movements after it to be with respect to the machine coordinate system. G49 cancels tool length compensation. Z0 is a command to move to Z = 0.0. M05 turns the spindle off.
G53 Y0 (Y home) ;	G53 defines axis movements after it to be with respect to the machine coordinate system. Y0 is a command to move to Y = 0.0.

Completion Code Block	Description
M30 (End program) ;	M30 ends the program and moves the cursor on the control to the top of the program.
%	Denotes the end of a program written in a text editor.

5.6.4 Absolute vs. Incremental Positioning (G90, G91)

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

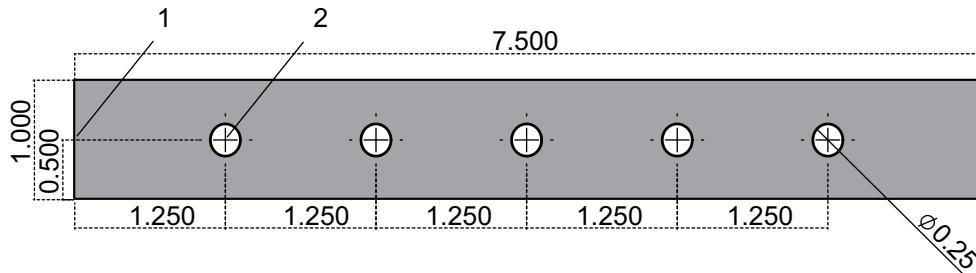
When you command axis motion after a G90 code, the axes move to that position relative to the origin of the coordinate system currently in use.

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

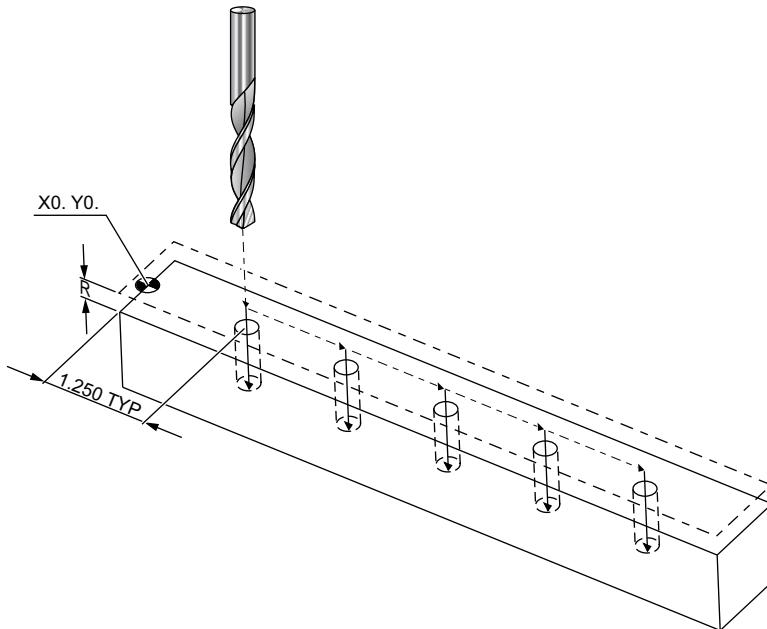
Absolute programming is useful in most situations. Incremental programming is more efficient for repetitive, equally spaced cuts.

Figure F5.21 shows a part with 5 equally spaced $\varnothing 0.25"$ (13 mm) diameter holes. The hole depth is 1.00" (25.4 mm) and the spacing is 1.250" (31.75 mm) apart.

F5.21: Absolute / Incremental Sample Program. G54 X0. Y0. for Incremental [1], G54 for Absolute [2]



Below are two example programs that drill the holes as shown in the drawing, with a comparison between absolute and incremental positioning. We start the holes with a center drill, and finish drilling the holes with a 0.250" (6.35 mm) drill bit. We use a 0.200" (5.08 mm) depth of cut for the center drill and 1.00" (25.4 mm) depth of cut for the 0.250" drill. G81, Drill Canned Cycle, is used to drill the holes.

F5.22: Mill Incremental Positioning Example.

```

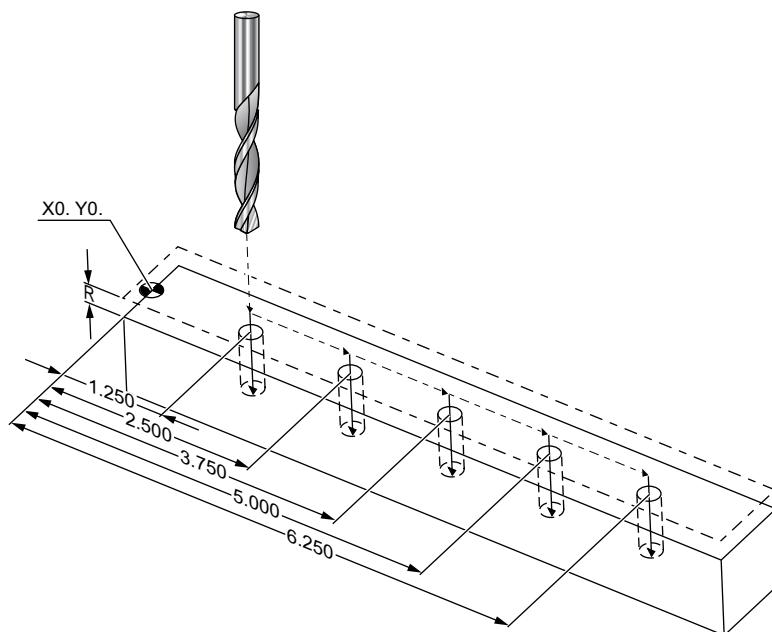
%
O40002 (Incremental ex-prog) ;
N1 (G54 X0 Y0 is center left of part) ;
N2 (Z0 is on top of the part) ;
N3 (T1 is a center drill) ;
N4 (T2 is a drill) ;
N5 (T1 PREPARATION BLOCKS) ;
N6 T1 M06 (Select tool 1) ;
N7 G00 G90 G40 G49 G54 (Safe startup) ;
N8 X0 Y0 (Rapid to 1st position) ;
N9 S1000 M03 (Spindle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

```

Basic Programming

```
N24 G43 H02 Z0.1(Tool offset 2 on) ;
N25 M08 (Coolant on) ;
N26 (T2 CUTTING BLOCKS) ;
N27 G99 G91 G81 F21.4 X1.25 Z-1.1 L5 ;
N28 G80 (Cancel G81) ;
N29 (T2 COMPLETION BLOCKS) ;
N30 G00 Z0.1 M09 (Rapid retract, clnt off) ;
N31 G53 G90 G49 Z0 M05 (Z home, spindle off) ;
N32 G53 Y0 (Y home) ;
N33 M30 (End program) ;
%
```

F5.23: Mill Absolute Positioning Example



```
%  
O40003 (Absolute ex-prog) ;  
N1 (G54 X0 Y0 is center left of part) ;  
N2 (Z0 is on top of the part) ;  
N3 (T1 is a center drill) ;  
N4 (T2 is a drill) ;  
N5 (T1 PREPARATION BLOCKS) ;  
N6 T1 M06 (Select tool 1) ;  
N7 G00 G90 G40 G49 G54 (Safe startup) ;  
N8 X1.25 Y0 (Rapid to 1st position) ;  
N9 S1000 M03 (Spindle on CW) ;  
N10 G43 H01 Z0.1 (Tool offset 1 on) ;  
N11 M08 (Coolant on) ;  
N12 (T1 CUTTING BLOCKS) ;
```

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

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

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

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

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

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

5.7 Tool and Work Offset Calls

5.7.1 G43 Tool Offset

The G43 Hnn Tool Length Compensation command should be used after every tool change. It adjusts the Z-Axis position to account for the length of the tool. The Hnn argument specifies which tool length to use.



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

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

5.7.2 G54 Work Offsets

Work Offsets define where a work piece is located on the table. Work Offsets available are G54-G59, G110-G129, and G154 P1-P99. G110-G129 and G154 P1-P20 refer to the same Work Offsets. A useful feature is to set up multiple work pieces on the table and machining multiple parts in one machine cycle. This is accomplished by assigning each work piece to a different Work Offset. For more information, reference the G-code section of this manual. Below is an example of machining multiple parts in one cycle. The program uses M97 Local Sub-Program Call in the cutting operation.

```
%  
O40005 (Work offsets ex-prog) ;  
(G54 X0 Y0 is center left of part) ;  
(Z0 is on top of the part) ;  
(T1 is a drill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54(Safe startup) ;  
X0 Y0 ;  
(Move to first work coordinate position-G54) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Tool offset 1 on) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
M97 P1000 (Call local Subprogram) ;
```

```

G00 Z3. (Rapid retract) ;
G90 G110 G17 G40 G80 X0. Y0. ;
(Move to second work coordinate position-G110) ;
M97 P1000 (Call local Subprogram) ;
G00 Z3. (Rapid Retract) ;
G90 G154 P22 G17 G40 G80 X0. Y0. ;
(Move to third work coordinate position-G154 P22) ;
M97 P1000 (Call local Subprogram) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
N1000 (Local subprogram) ;
G81 F41.6 X1. Y2. Z-1.25 R0.1 (Begin G81) ;
(1st hole) ;
X2. Y2. (2nd hole) ;
G80 (Cancel G81) ;
M99 ;
%

```

5.8 Miscellaneous Codes

This section lists frequently used M-codes. Most programs have at least one M-code from each of the following families. Refer to the M-code section of this manual.



NOTE:

You can use only one M-code on each line of the program.

5.8.1 Tool Functions (Tnn)

The Tnn code selects 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 starts a tool change operation; for example, T1M06 puts tool 1 in the spindle.



CAUTION:

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

You can command a tool change 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 moves the Z Axis to a position above machine zero during a tool change, but it never moves below machine zero. At the end of a tool change, the Z Axis is at machine zero.

5.8.2 Spindle Commands

There are (3) primary spindle M-code commands:

- M03 Snnnn commands the spindle to turn clockwise.
- M04 Snnnn commands the spindle to turn counter-clockwise.



NOTE:

The Snnnn address commands the spindle to turn at nnnn RPM, up to the maximum spindle speed.

- M05 commands the spindle to stop.



NOTE:

When you command an M05, the control waits for the spindle to stop before the program continues.

5.8.3 Program Stop Commands

There are (2) main M-codes and (1) subprogram M-code to denote the end of a program or subprogram:

- M30 - Program End and Rewind ends the program and resets to the beginning of the program. This is the most common way to end a program.
- M02 - Program End ends the program and remains at the location of the M02 block of code in the program.
- M99 - Subprogram Return or Loop exits the subprogram and resumes the program that called it.



NOTE:

If your subprogram does not end with M99, the control gives Alarm 312 - Program End.

5.8.4 Coolant Commands

Use M08 to command standard coolant on. Use M09 to command standard coolant off. Refer to page 336 for more information on these M-codes.

If your machine has Through-Spindle Coolant (TSC), use M88 to command it on, and M89 to command it off.

5.9 Cutting G-codes

The main cutting G-codes are categorized into interpolation motion and canned cycles. Interpolation motion cutting codes are broken down into:

- G01 - Linear Interpolation Motion
- G02 - Clockwise Circular Interpolation Motion
- G03 - Counter-Clockwise Circular Interpolation Motion
- G12 - Clockwise Circular Pocket Milling
- G13 - Counter-Clockwise Circular Pocket Milling

5.9.1 Linear Interpolation Motion

G01 Linear Interpolation Motion is used to cut straight lines. It requires a feedrate, specified with the Fnnn.nnnn address code. Xnn.nnnn, Ynn.nnnn, Znn.nnnn, and Ann.nnn are optional address codes to specify cut. Subsequent axis motion commands will use the feed rate specified by G01 until another axis motion, G00, G02, G03, G12, or G13 is commanded.

Corners can be chamfered using the optional argument Cnn.nnnn to define the chamfer. Corners can be rounded using the optional address code Rnn.nnnn to define the radius of the arc. Refer to page 233 for more information on G01.

5.9.2 Circular Interpolation Motion

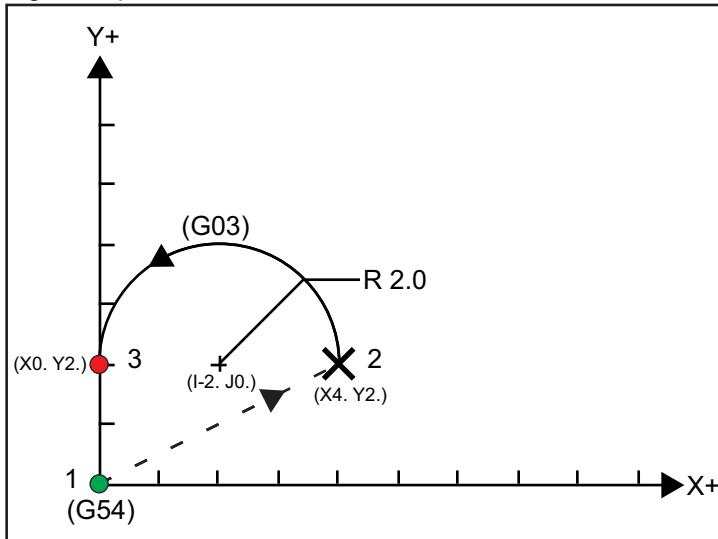
G02 and G03 are the G-codes for circular cutting motions. Circular Interpolation Motion has several optional address codes to define the arc or circle. The arc or circle begins cutting from the current cutter position [1] to the geometry specified within the G02/ G03 command.

Arcs can be defined using two different methods. The preferred method is to define the center of the arc or circle with I, J and/or K and to define the end point [3] of the arc with an X, Y and/or Z. The I J K values define the relative X Y Z distances from the starting point [2] to the center of the circle. The X Y Z values define the absolute X Y Z distances from the starting point to the end point of the arc within the current coordinate system. This is also the only method to cut a circle. Defining only the I J K values and not defining the end point X Y Z values will cut a circle.

The other method to cut an arc is to define the X Y Z values for the end point and to define the radius of the circle with an R value.

Below are examples of using the two different methods to cut a 2" (or 2 mm) radius, 180 degree, counter-clockwise arc. The tool starts at X0 Y0 [1], moves to the starting point of the arc [2], and cuts the arc to the end point [3]:

F5.24: Arc Cutting Example



Method 1:

```
%  
T01 M06 ;  
...  
G00 X4. Y2. ;  
G01 F20.0 Z-0.1 ;  
G03 F20.0 I-2.0 J0. X0. Y2. ;  
...  
M30 ;  
%
```

Method 2:

```
%  
T01 M06 ;  
...  
G00 X4. Y2. ;  
G01 F20.0 Z-0.1 ;  
G03 F20.0 X0. Y2. R2. ;  
...M30 ;  
%
```

Below is an example of how to cut a 2" (or 2 mm) radius circle:

```
%  
T01 M06 ;  
...  
G00 X4. Y2. ;  
G01 F20.0 Z-0.1 ;
```

```
G02 F20.0 I2.0 J0. ;
...
M30 ;
%
```

5.10 Cutter Compensation

Cutter compensation is a method of shifting the tool path so that the actual centerline of the tool moves to either the left or right of the programmed path. Normally, cutter compensation is programmed to shift the tool in order to control feature size. The offset display is used to enter the amount that the tool is to be shifted. The offset can be entered as either a diameter or radius value, depending on Setting 40, for both the geometry and wear values. If diameter is specified, the shift amount is half of the value entered. The effective offset values are the sum of the geometry and wear values. Cutter compensation is only available in the X Axis and the Y Axis for 2D machining (G17). For 3D machining, cutter compensation is available in the X Axis, Y Axis, and Z Axis (G141).

5.10.1 General Description of Cutter Compensation

G41 selects cutter compensation left. This means that the control moves the tool to the left of the programmed path (with respect to the direction of travel) to compensate for the tool radius or diameter defined in the tool offsets table (Refer to Setting 40). G42 selects cutter compensation right, which moves the tool to the right of the programmed path, with respect to the direction of travel.

A G41 or G42 command must have a Dnnn value to select the correct offset number from the radius / diameter offset column. The number to use with D is in the far-left column of the tool offsets table. The value that the control uses for cutter compensation is in the GEOMETRY column under D (if Setting 40 is DIAMETER) or R (if Setting 40 is RADIUS).

If the offset value is negative, cutter compensation operates as though the program specifies the opposite G code. For example, a negative value entered for a G41 will behave as if a positive value was entered for G42. Also, when cutter compensation is active (G41 or G42), you may use only the X-Y plane (G17) for circular motions. Cutter compensation is limited to compensation in only the X-Y plane.

G40 cancels cutter compensation and is the default condition when you power on your machine. When cutter compensation is not active, the programmed path is the same as the center of the cutter path. You may not end a program (M30, M00, M01, or M02) with cutter compensation active.

The control operates on one motion block at a time. However, it will look ahead at the next (2) blocks that have X or Y motions. The control checks these (3) blocks of information for interference. Setting 58 controls how this part of cutter compensation works. Available Setting 58 values are Fanuc or Yasnac.

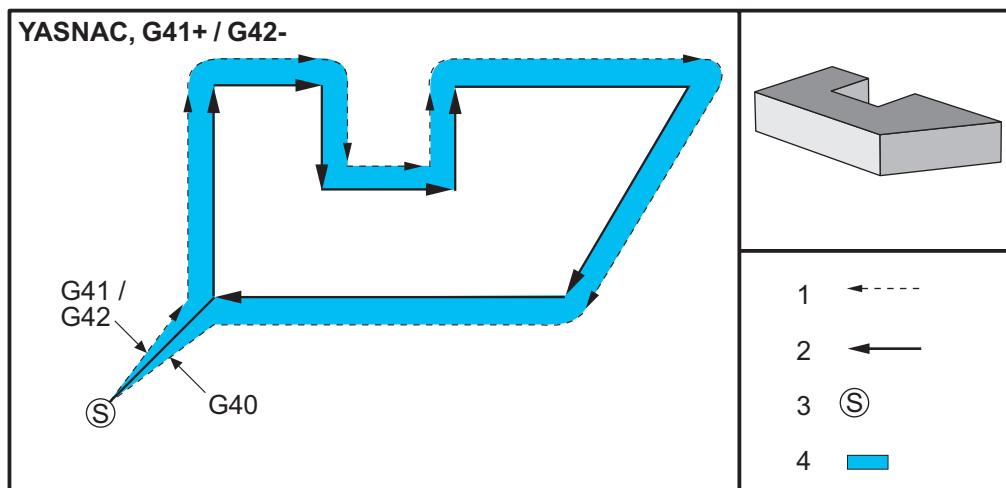
If Setting 58 is set to Yasnac, the control must be able to position the side of the tool along all of the edges of the programmed contour without overcutting the next two motions. A circular motion joins all of the outside angles.

Cutter Compensation

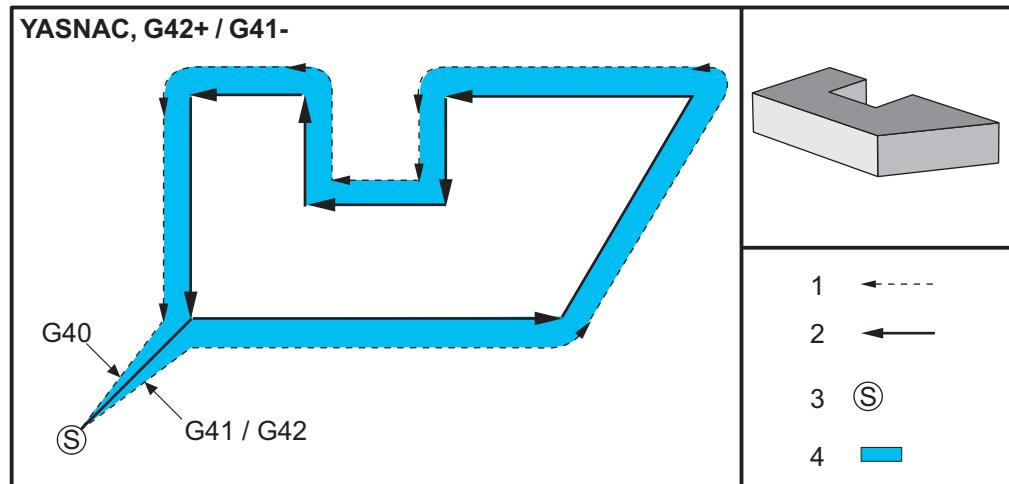
If Setting 58 is set to Fanuc, the control does not require that the tool cutting edge be placed along all edges of the programmed contour, preventing overcutting. However the control will generate an alarm if the cutter's path is programmed so that it will overcut. The control joins outside angles less than or equal to 270 degrees with a sharp corner. It joins outside angles of more than 270 degrees with an extra linear motion.

These diagrams show how cutter compensation works for the possible values of Setting 58. Note that a small cut of less than the tool radius and at a right angle to the previous motion will work only with the Fanuc setting.

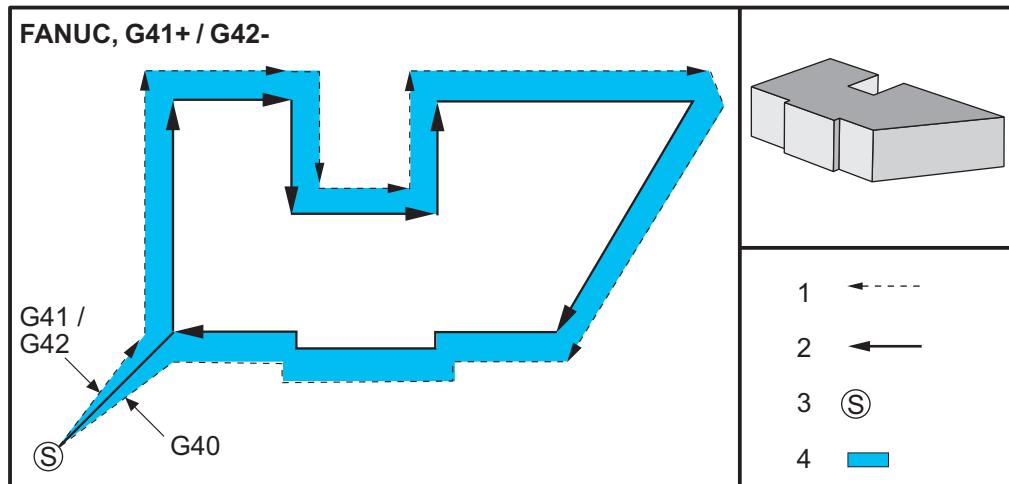
- F5.25:** Cutter Compensation, YASNAC Style, G41 with a Positive Tool Diameter or G42 with a Negative Tool Diameter: [1] Tool Path Actual Center, [2] Programmed Tool Path, [3] Start Point, [4] Cutter Compensation. G41 / G42 and G40 are commanded at the start and end of the tool path.



- F5.26:** Cutter Compensation, YASNAC Style, G42 with a Positive Tool Diameter or G41 with a Negative Tool Diameter: [1] Tool Path Actual Center, [2] Programmed Tool Path, [3] Start Point, [4] Cutter Compensation. G41 / G42 and G40 are commanded at the start and end of the tool path.

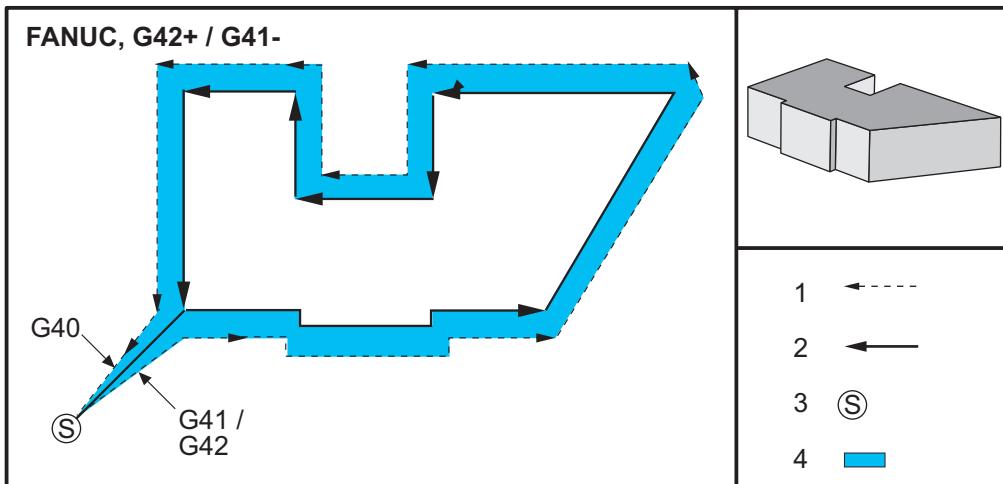


- F5.27:** Cutter Compensation, FANUC Style, G41 with a Positive Tool Diameter or G42 with a Negative Tool Diameter: [1] Tool Path Actual Center, [2] Programmed Tool Path, [3] Start Point, [4] Cutter Compensation. G41 / G42 and G40 are commanded at the start and end of the tool path.



Cutter Compensation

F5.28: Cutter Compensation, FANUC Style, G42 with a Positive Tool Diameter or G41 with a Negative Tool Diameter: [1] Tool Path Actual Center, [2] Programmed Tool Path, [3] Start Point, [4] Cutter Compensation. G41 / G42 and G40 are commanded at the start and end of the tool path.



5.10.2 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.

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**.

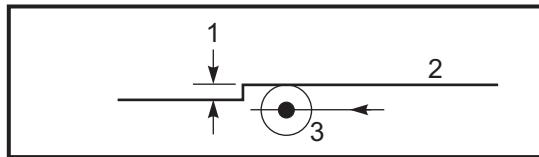
You should always turn off cutter compensation in a move that clears the tool away from the part being cut. If a program is terminated with cutter compensation still active, an alarm is generated. In addition, you cannot turn cutter compensation on or off during a circular move (**G02** or **G03**); 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 in this section illustrate the differences between type A and type B for both Fanuc and Yasnac settings (Setting 58).

Improper Cutter Compensation Application

- F5.29:** Improper Cutter Compensation: [1] Move is less than cutting comp radius, [2] Workpiece, [3] Tool.



NOTE:

A small cut of less than tool radius and at a right angle to the previous motion will only work with the Fanuc setting. A cutter compensation alarm will be generated if the machine is set to the Yasnac setting.

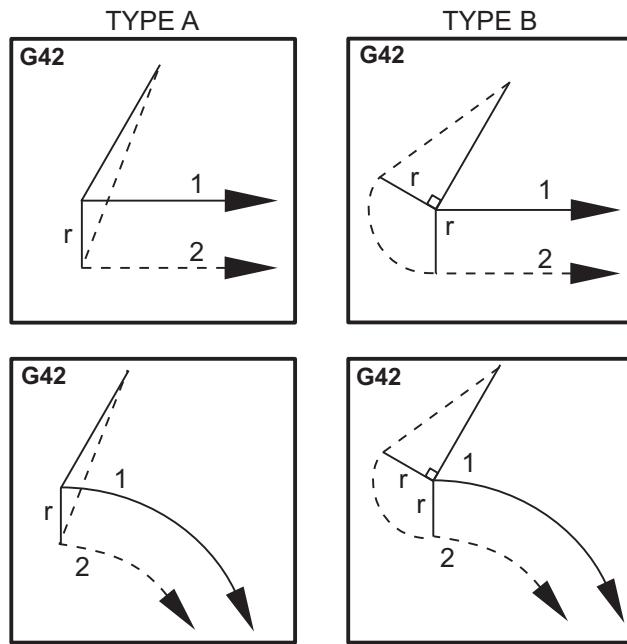
5.10.3 Feed Adjustments in Cutter Compensation

When using cutter compensation in circular moves, there is the possibility of speed adjustments to what has been programmed. If the intended finish cut is on the inside of a circular motion, the tool should be slowed down to ensure that the surface feed does not exceed what was intended by the programmer. There are problems, however, when the speed is slowed by too much. For this reason, Setting 44 is used to limit the amount by which the feed is adjusted in this case. It can be set between 1% and 100%. If set to 100%, there will be no speed changes. If set to 1%, the speed can be slowed to 1% of the programmed feed.

When the cut is on the outside of a circular motion, there is no speed-up adjustment made to the feed rate.

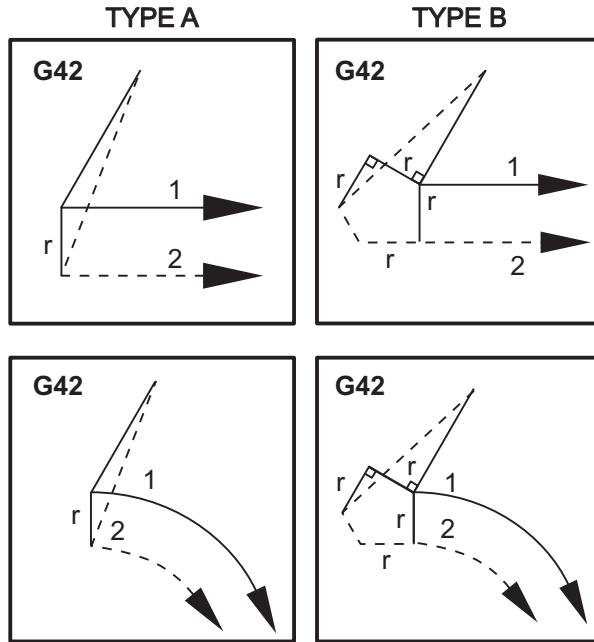
Cutter Compensation Entry (Yasnac)

F5.30: Cutter Compensation Entry (Yasnac) Type A and B: [1] Programmed Path, [2] Tool Center Path, [r] Tool Radius



Cutter Compensation Entry (Fanuc style)

F5.31: Cutter Compensation Entry (Fanuc style) Type A and B: [1] Programmed Path, [2] Tool Center Path, [r] Tool Radius



5.10.4 Circular Interpolation and Cutter Compensation

In this section, the usage of G02 (Circular Interpolation Clockwise), G03 (Circular Interpolation Counterclockwise) and Cutter Compensation (G41: Cutter Compensation Left, G42: Cutter Compensation Right) is described.

Using G02 and G03, we can program the machine to cut circular moves and radii. Generally, when programming a profile or a contour, the easiest way to describe a radius between two points is with an R and a value. For complete circular moves (360 degrees), an I or a J with a value must be specified. The circle section illustration will describe the different sections of a circle.

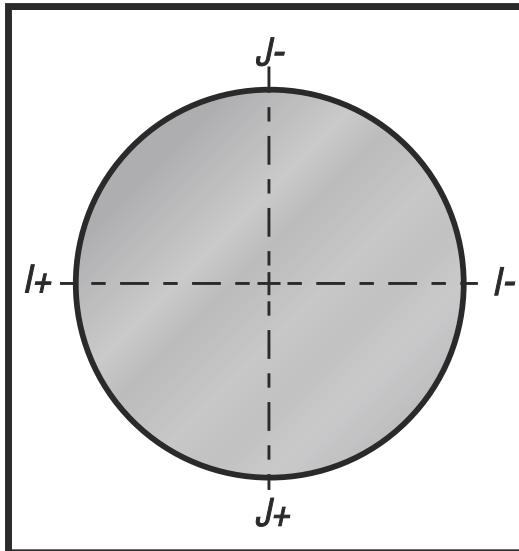
By using cutter compensation in this section, the programmer will be able to shift the cutter by an exact amount and be able to machine a profile or a contour to the exact print dimensions. By using cutter compensation, programming time and the likelihood of a programming calculation error is reduced due to the fact that real dimensions can be programmed, and part size and geometry can be easily controlled.

Cutter Compensation

Here are a few rules about cutter compensation that you must follow closely for successful machining operations. Always refer to these rules when you write your programs.

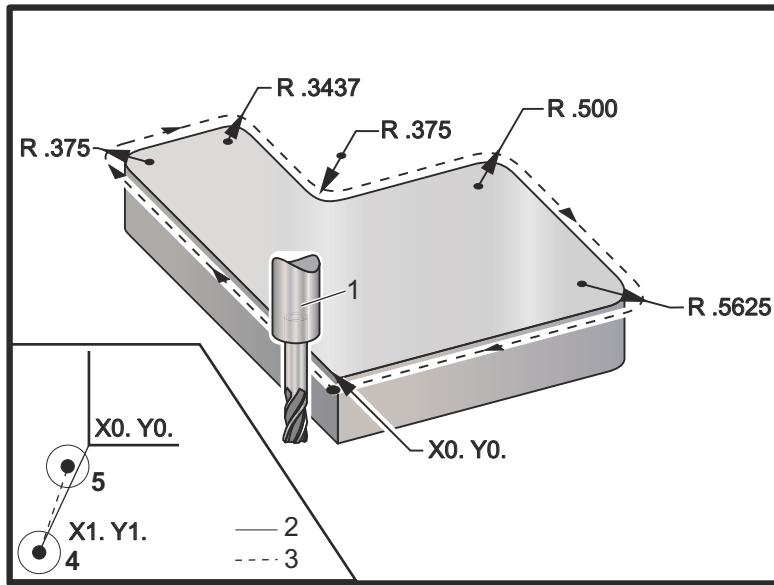
1. Cutter compensation must be turned ON during a G01 X, Y move that is equal to or greater than the cutter radius, or the amount being compensated.
2. When an operation using cutter compensation is done, the cutter compensation will need to be turned OFF, using the same rules as the turn ON process, 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 causes the machine to alarm. Can not have too big of a tool diameter if the radius of arc is too small.

F5.32: Circle Sections



This illustration shows how the tool path is calculated for the cutter compensation. The detail section shows the tool in the starting position and then in the offset position as the cutter reaches the workpiece.

F5.33: Circular Interpolation G02 and G03: [1] 0.250" diameter endmill, [2] Programmed path, [3] Center of Tool, [4] Start Position, [5] Offset Tool Path.



Programming exercise showing tool path.

This program uses cutter compensation. The toolpath is programmed to the centerline of the cutter. This is also the way the control calculates for cutter compensation.

```
%  
O40006 (Cutter comp ex-prog) ;  
(G54 X0 Y0 is at the lower left of part corner) ;  
(Z0 is on top of the part) ;  
(T1 is a .250 dia endmill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
X-1. Y-1. (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Tool offset 1 on) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 Z-1. F50. (Feed to cutting depth) ;  
G41 G01 X0 Y0 D01 F50. (2D Cutter Comp left on) ;  
Y4.125 (Linear motion) ;  
G02 X0.25 Y4.375 R0.375 (Corner rounding) ;  
G01 X1.6562 (Linear motion) ;  
G02 X2. Y4.0313 R0.3437 (Corner rounding) ;  
G01 Y3.125 (Linear motion) ;  
G03 X2.375 Y2.75 R0.375 (Corner rounding) ;
```

```
G01 X3.5 (Linear motion) ;
G02 X4. Y2.25 R0.5 (Corner rounding) ;
G01 Y0.4375 (Linear motion) ;
G02 X3.4375 Y-0.125 R0.5625 (Corner rounding) ;
G01 X-0.125 (Linear motion) ;
G40 X-1. Y-1. (Last position, cutter comp off) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%
```

5.11 Canned Cycles

Canned cycles are G-codes that do repetitive operations such as drilling, tapping, and boring.

5.11.1 Drilling Canned Cycles

All four drill canned cycles can be looped in G91, Incremental Programming mode.

- The G81 Drill Canned Cycle is the basic drilling cycle. It is used for drilling shallow holes or for drilling with Through Spindle Coolant (TSC).
- The G82 Spot Drill Canned Cycle is the same as the G81 Drill Canned Cycle except that it can dwell at the bottom of the hole. The optional argument Pn.nnn specifies the duration of the dwell.
- The G83 Normal Peck Drilling Canned Cycle is typically used for drilling deep holes. Peck depth can be variable or constant and always incremental. Qnn.nnn. Do not use a Q value when programming with I, J, and K.
- The G73 High-Speed Peck Drilling Canned Cycle is the same as the G83 Normal Peck Drilling Canned Cycle except that tool peck retraction is specified with Setting 22 - Can Cycle Delta Z. Peck drilling cycles are advised for hole depths greater than 3 times the diameter of the drill bit. The initial peck depth, defined by I, should generally be a depth of 1 tool diameter.

5.11.2 Tapping Canned Cycles

There are two tapping canned cycles. All tapping canned cycles can be looped in G91, Incremental Programming mode.

- The G84 Tapping Canned Cycle is the normal tapping cycle. It is used for tapping right-hand threads.
- G74 Reverse Tap Canned Cycle is the reverse thread tapping cycle. It is used for tapping left-hand threads.

5.11.3 Boring and Reaming Cycles

There are seven boring canned cycles. All boring canned cycles can be looped in G91, Incremental Programming mode.

- The G85 Boring Canned Cycle is the basic boring cycle. It will bore down to the desired height and return to the specified height.
- The G86 Bore and Stop Canned Cycle is the same as the G85 Boring Canned Cycle except that the spindle will stop at the bottom of the hole before returning to the specified height.
- The G87 Bore In and Manual Retract Canned Cycle is also the same except that the spindle will stop at the bottom of the hole, the tool is manually jogged out of the hole, and the program will resume again when Cycle Start is pressed.
- The G88 Bore In, Dwell, Manual Retract Canned Cycle is the same as G87 except that there is a dwell before the operator can manually jog the tool out of the hole.
- The G89 Bore In, Dwell, Bore Out Canned Cycle is the same as G85 except that there is a dwell at the bottom of the hole, and the hole continues to be bored at the specified feed rate as the tool returns to the specified position. This differs from other boring canned cycles where the tool either moves in Rapid Motion or hand jog to return to the return position.
- The G76 Fine Boring Canned Cycle bores the hole to the specified depth and after boring the hole, moves to clear the tool from hole before retracting.
- The G77 Back Bore Canned Cycle works similar to G76 except that before beginning to bore the hole, it moves the tool to clear the hole, moves down into the hole, and bores to the specified depth.

5.11.4 R Planes

R Planes, or return planes, are G-code commands that specify the Z-Axis return height during canned cycles. The R Plane G-codes remain active for the duration of the canned cycle it is used with. G98 Canned Cycle Initial Point Return moves the Z axis to the height of the Z axis prior to the canned cycle. G99 Canned Cycle R Plane Return moves the Z axis to the height specified by the Rnn.nnnn argument specified with the canned cycle. For additional information, refer to the G and M-code section.

5.12 Special G-codes

Special G-codes are used for complex milling. These include:

- Engraving (G47)
- Pocket Milling (G12, G13, and G150)
- Rotation and Scaling (G68, G69, G50, G51)
- Mirror Image (G101 and G100)

5.12.1 Engraving

The G47 Text Engraving G-code lets you engrave text (including some ASCII characters) or sequential serial numbers with a single block of code.

Refer to page 255 for more information on engraving.

5.12.2 Pocket Milling

There are two types of pocket milling G-codes on the Haas control:

- Circular Pocket Milling is performed with the G12 Clockwise Circular Pocket Milling Command and the G13 Counter-Clockwise Circular Pocket Milling Command G-codes.
- The G150 General Purpose Pocket Milling uses a subprogram to machine user-defined pocket geometries.

Make sure that the subprogram geometry is a fully closed shape. Make sure that the X-Y starting point in the G150 command is within the boundary of the fully closed shape. Failure to do so may cause Alarm 370 - Pocket Definition Error.

Refer to page 244 for more information on the pocket milling G-codes.

5.12.3 Rotation and Scaling



NOTE:

You must purchase the rotation and scaling option to use these features. A 200-hour option tryout is also available.

G68 Rotation is used to rotate the coordinate system in the desired plane. You can use this feature together with G91 Incremental Programming mode to machine symmetrical patterns. G69 cancels rotation.

G51 applies a scaling factor to the positioning values in blocks after the G51 command. G50 cancels scaling. You can use scaling together with rotation, but be sure to command scaling first.

Refer to page 269 for more information on the rotation and scaling G-codes.

5.12.4 Mirror Image

G101 Enable Mirror Image will mirror axis motion about the specified axis. Settings 45-48, 80 and 250 enable mirror imaging about the X, Y, Z, A, B, and C axes. The mirror pivot point along an axis is defined by the Xnn.nn argument. This can be specified for a Y Axis that is enabled on the machine and in the settings by using the axis to mirror as the argument. G100 cancels G101.

Refer to page 296 for more information on the mirror image G-codes.

5.13 Subprograms

Subprograms (subprograms):

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

When you use M97:

- The P code (nnnnn) is the same as the program location (Onnnnn) of the subprogram.
- The subprogram must be within the main program

When you use M98:

- The P code (nnnnn) is the same as the program number (Onnnnn) of the subprogram.
- The subprogram must reside in the control memory or hard drive (optional).

Canned Cycles are the most common use of subprograms. For example, you might put the X and Y locations of a series of holes in a separate program. Then you can call that program as a subprogram with a canned cycle. Instead of writing the locations once for each tool, you write the locations only once for any number of tools.

5.13.1 External Subprogram (M98)

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

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

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

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

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

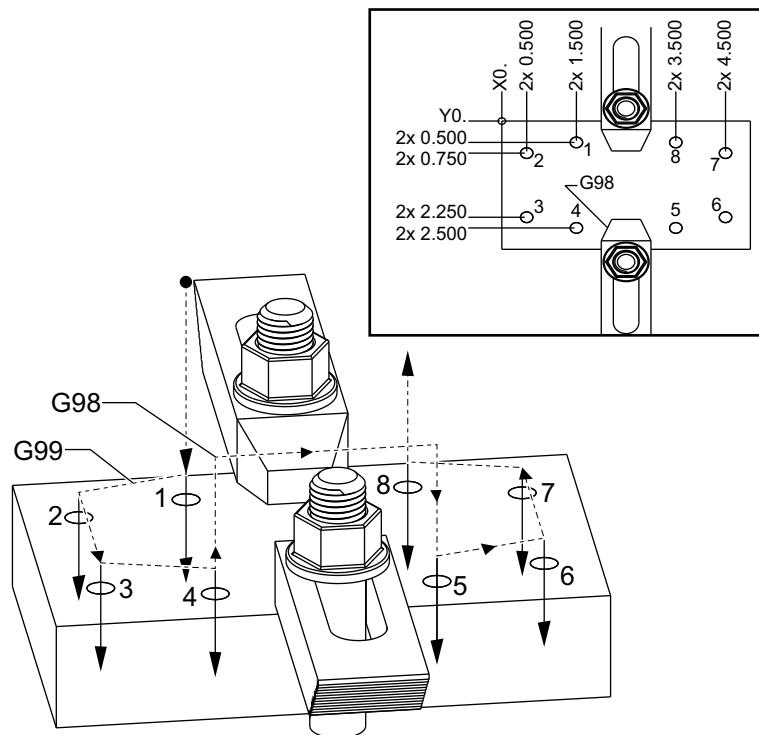
```
%  
O40007 (External subprogram ex-prog) ;  
(G54 X0 Y0 is center left of part) ;
```

Subprograms

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

```
M30 (End program) ;  
%
```

F5.34: Subprogram Pattern



Subprogram

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

5.13.2 Local Subprogram (M97)

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

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

Local Subprogram Example

```
%  
O40009 (Local subprogram ex-prog) ;  
(G54 X0 Y0 is at the top left corner of part) ;  
(Z0 is on top of the part) ;  
(T1 is a spot drill) ;  
(T2 is a drill) ;  
(T3 is a tap) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54(Safe startup) ;  
X1.5 Y-0.5 (Rapid to 1st position) ;  
S1406 M03 (Spindle on CW) ;  
G43 H01 Z1.(Tool offset 1 on) ;  
M08(Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G81 G99 Z-0.26 R0.1 F7. (Begin G81) ;  
M97 P1000 (Call local subprogram) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
M01 (Optional stop) ;  
(BEGIN PREPARATION BLOCKS) ;  
T2 M06 (Select tool 2) ;  
G00 G90 G40 G49 (Safe startup) ;  
G54 X1.5 Y-0.5 (Rapid back to 1st position) ;  
S2082 M03 (Spindle on CW) ;  
G43 H02 Z1. (Tool offset 2 on) ;  
M08(Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G83 G99 Z-0.75 Q0.2 R0.1 F12.5 (Begin G83) ;  
M97 P1000 (Call local subprogram) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
M01 (Optional stop) ;
```

```

(BEGIN PREPARATION BLOCKS) ;
T3 M06 (Select tool 3) ;
G00 G90 G40 G49 (Safe startup) ;
G54 X1.5 Y-0.5 ;
(Rapid back to 1st position) ;
S750 M03 (Spindle on CW) ;
G43 H03 Z1. (Tool offset 3 on) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G84 G99 Z-0.6 R0.1 F37.5 (Begin G84) ;
M97 P1000 (Call local subprogram) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
(LOCAL subprogram) ;
N1000 (Begin local subprogram) ;
X0.5 Y-0.75 (2nd position) ;
Y-2.25 (3rd position) ;
G98 X1.5 Y-2.5 (4th position) ;
(Initial point return) ;
G99 X3.5 (5th position) ;
(R-plane return) ;
X4.5 Y-2.25 (6th position) ;
Y-0.75 (7th position) ;
X3.5 Y-0.5 (8th position) ;
M99 ;
%

```

5.13.3 External Subprogram Canned Cycle Example (M98)

```

%
O40010 (M98_External sub canned cycle ex) ;
(G54 X0 Y0 is at the top left of the part) ;
(Z0 is on top of the part) ;
(T1 is a spot drill) ;
(T2 is a drill) ;
(T3 is a tap) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54(Safe startup) ;
X0.565 Y-1.875 (Rapid to 1st position) ;
S1275 M03 (Spindle on CW) ;
G43 H01 Z0.1 (Tool offset 1 on) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;

```

Subprograms

```
G82 Z-0.175 P0.03 R0.1 F10. (Begin G82) ;
M98 P40011 (Call external subprogram) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z1. M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
M01 (optional stop) ;
(BEGIN PREPARATION BLOCKS) ;
T2 M06 (Select tool 2) ;
G00 G90 G40 G49 (Safe startup) ;
G54 X0.565 Y-1.875 ;
(Rapid back to 1st position) ;
S2500 M03 (Spindle on CW) ;
G43 H02 Z0.1 (Tool offset 2 on) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G83 Z-0.72 Q0.175 R0.1 F15. (Begin G83) ;
M98 P40011 (Call external subprogram) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z1. M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
M01 (optional stop) ;
(BEGIN PREPARATION BLOCKS) ;
T3 M06 (Select tool 3) ;
G00 G90 G40 G49 (Safe startup) ;
G54 X0.565 Y-1.875 ;
(Rapid back to 1st position) ;
S900 M03 (Spindle on CW) ;
G43 H03 Z0.1 (Tool offset 3 on) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G84 Z-0.6 R0.2 F56.25 (Begin G84) ;
M98 P40011 (Call external subprogram) ;
G80 G00 Z1. M09 (Cancel canned cycle) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z1. M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%
```

Subprogram

```
% 
O40011 (M98_Subprogram X,Y Locations) ;
X1.115 Y-2.75 (2nd position) ;
X3.365 Y-2.875 (3rd position) ;
X4.188 Y-3.313 (4th position) ;
X5. Y-4. (5th position) ;
```

```
M99 ;
%
```

5.13.4 External Subprograms With Multiple Fixtures (M98)

Subprograms 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 in absolute coordinates. Use an edge finder or an indicator to establish the zero point on each part. Use the part zero set key in the work 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.

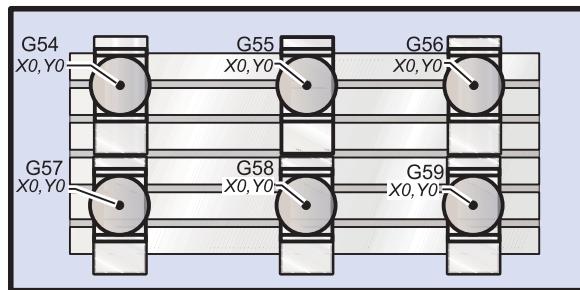
The figure shows what this setup would look like on the machine table. For an example, each of these six parts will need to be drilled at the center, X and Y zero.

Main Program

```
%  
O40012 (M98_External sub multi fixture);  
(G54-G59 X0 Y0 is center of each part) ;  
(G54-G59 Z0 is on top of the part) ;  
(T1 is a drill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54(Safe startup) ;  
X0 Y0 (Rapid to 1st position) ;  
S1500 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Tool offset 1 on) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
M98 P40013 (Call external subprogram) ;  
G55 (Change work offset) ;  
M98 P40013 (Call external subprogram) ;  
G56 (Change work offset) ;  
M98 P40013 (Call external subprogram) ;  
G57 (Change work offset) ;  
M98 P40013 (Call external subprogram) ;  
G58 (Change work offset) ;  
M98 P40013 (Call external subprogram) ;  
G59 (Change work offset) ;  
M98 P40013 (Call external subprogram) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

More Information Online

F5.35: Subprogram Multiple Fixture Drawing



Subprogram

```
%  
O40013 (M98_Subprogram) ;  
X0 Y0 (Move to zero of work offset) ;  
G83 Z-1. Q0.2 R0.1 F15. (Begin G83) ;  
G00 G80 Z0.2 M09 (Cancel canned cycle) ;  
M99 ;  
%
```

5.14 More Information Online

For updated and supplemental information, including tips, tricks, maintenance procedures, and more, visit the Haas Resource Center at diy.HaasCNC.com. You can also scan the code below with your mobile device to go directly to the Resource Center:



Chapter 6: Options Programming

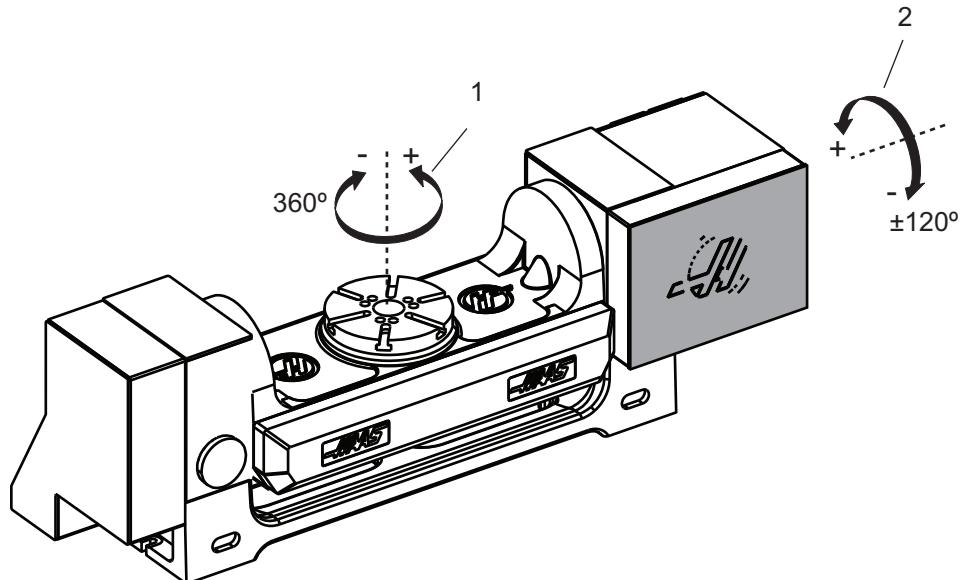
6.1 Introduction

In addition to the standard functions included with your machine, you may also have optional equipment with special programming considerations. This section tells you how to program these options.

You can contact your HFO to purchase most of these options, if your machine did not come equipped with them.

6.2 4th and 5th Axis Programming

F6.1: Axis Motion on an Example Rotary Trunnion Unit: [1] Rotary Axis, [2] Tilt Axis



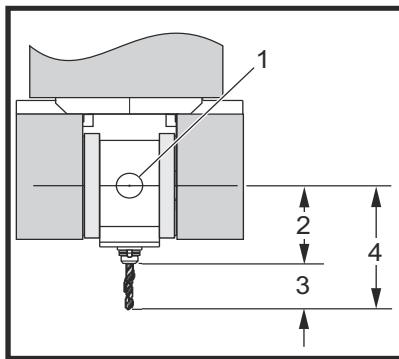
6.2.1 Creating Five-Axis Programs

Most five-axis programs are rather complex and should be written using a CAD/CAM software 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 that is shipped with a 5-axis machine. The pivot length can be found in Setting 116, and is also engraved into the master tool holder.

4th and 5th Axis Programming

F6.2: Pivot and Gauge Length Diagram: [1] Axis of Rotation, [2] Pivot Length, [3] Gauge Length, [4] Total



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 tool holder to the tip of the tool. This distance can be calculated:

1. Set magnetic base indicator on the table.
2. Indicate the bottom surface of the tool holder.
3. Set this point as Z_0 in the control.
4. Insert each tool and calculate distance from the tool tip to Z_0 ; this is the gauge length.
5. 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 uses the value for its calculations.

Offsets

The work-offset display is found on the offset display. The G54 through G59 or G110 through G129 offsets can be set by using the **[PART ZERO SET]** button. This will work with only the work zero offsets display selected.

1. Press **[OFFSET]** until the Work Zero Offset (from all modes except MEM) displays.
2. Position the axes to the work zero point of the workpiece.
3. Using the cursor, select the proper axis and work number.
4. Press **[PART ZERO SET]** and the current machine position will be automatically stored in that address.

**NOTE:**

Entering a nonzero Z work offset will interfere with the operation of an automatically entered tool length offset.

5. 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

Program approach vectors (moving tool paths) to the workpiece at a safe distance above or to the side of the workpiece. This is important when you program the approach vectors with a rapid move (G00), because the axes arrive at the programmed position at different times; the axis with shortest distance from target arrives first, and longest distance last. However, a linear move at a high feed rate forces the axes to arrive at the commanded position at the same time, avoiding the possibility of a crash.

G-codes

5th-axis programming is not affected by the selection of inch (G20) or metric (G21), because the A and B Axes are always programmed in degrees.

G93 inverse time must be in effect for simultaneous 4- or 5-axis motion; however, if your mill supports Tool Center Point Control (G234), you can use G94. Refer to “G93” on page **293** for more information.

Limit the post processor (CAD/CAM software) to a maximum G93 F value of 45000. This results 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 and uninterrupted flow of motion. Remember that if the brakes are never re-engaged, they remain off indefinitely.

Settings

A number of settings are used to program the 4th and 5th axis.

For the 4th axis:

- Setting 30 - 4th Axis Enable
- Setting 34 - 4th Axis Diameter
- Setting 48 - Mirror Image A-Axis

For the 5th axis:

- Setting 78 - 5th Axis Enable
- Setting 79 - 5th-Axis Diameter
- Setting 80 - Mirror Image B-Axis

Setting 85 - Maximum Corner Rounding should be set to .0500 for 5-axis cutting. Settings lower than .0500 move the machine closer to an exact stop and cause uneven motion.

You can also use G187 Pn Ennnn to set the smoothness level in the program to slow the axes down. G187 temporarily overrides Setting 85.



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.

Feedrates

You can command a feed in a program with G01 for the axis assigned to the rotary unit. For example,

G01 A90. F50. ;

turns the A Axis 90 degrees.

Every line of 4th/5th axis code must specify a feedrate. Limit the feedrate to less than 75 IPM when drilling. The recommended feeds for finish machining in 3-axis work should not exceed 50 to 60 IPM with at least 0.0500" to 0.0750" 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 feedrates may be possible. For example, when cutting mold lines or long flowing contours, feedrates may exceed 100 IPM.

Jogging the 4th and 5th Axis

All aspects of handle jogging for the 5th axis work as they do for the other axes. The exception is the method of selecting jog between axis A and axis B.

1. Press **[+A]** or **[-A]** to select the A Axis for jogging.
2. Press **[SHIFT]**, and then press either **[+A]** or **[-A]** to jog the B Axis.
3. EC-300: Jog mode shows A1 and A2, press **[A]** to jog A1 and press **[SHIFT] [A]** to jog A2.

6.2.2 Installing an Optional 4th Axis

Settings 30 and 34 must be changed when adding a rotary table to a Haas mill. Setting 30 specifies the rotary table model and Setting 34 specifies the part diameter.

Changing Setting 30

Setting 30 (and Setting 78 for the 5th axis) specifies a parameter set for a given rotary unit. These Settings allow you to select your rotary unit from a list, which then automatically sets the parameters necessary to allow your mill to interact with the rotary unit.

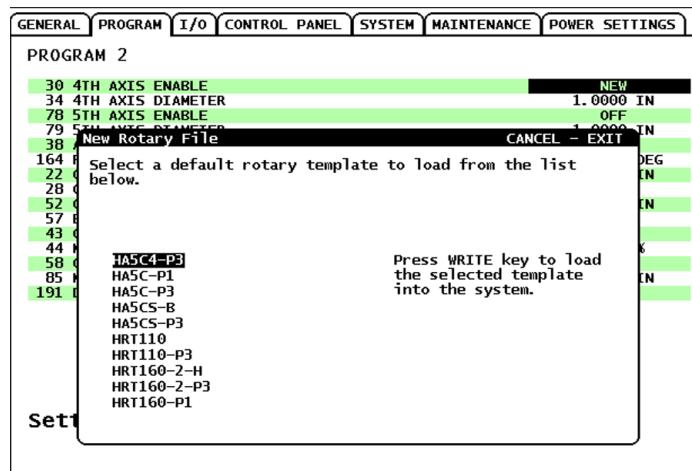


WARNING:

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 the table and two connectors at the mill control for each rotary axis.

4th and 5th Axis Programming

F6.3: New Rotary File Selection Menu



1. Highlight Setting 30 and press the left or right cursor arrow.
2. Press [EMERGENCY STOP].
3. Select NEW and then press [ENTER].
The list of available rotary parameter sets appears.
4. Press the [UP] or [DOWN] cursor arrow to select the correct rotary unit. You can also start typing the name of the rotary unit to reduce the list before making a selection.
The rotary model highlighted in the control must match the model engraved on the rotary unit's identification plate.
5. Press [ENTER] to confirm your choice.
The parameter set is then loaded into the machine. The name of the current parameter set appears for Setting 30.
6. Reset [EMERGENCY STOP].
7. Do not attempt to use the rotary until you cycle machine power.

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.



NOTE:

DO NOT CHANGE THE PARAMETERS if you did not receive a list of parameters with the indexer. Doing so will void your warranty.

Initial Start-up

To start up the indexer:

1. Turn on the mill (and servo control, if applicable).
2. Home the indexer.
3. All Haas indexers home in the clockwise direction as viewed from the front. If the indexer homes counter-clockwise, press [**EMERGENCY STOP**] and call your dealer.

6.2.3 Installing an Optional 5th Axis

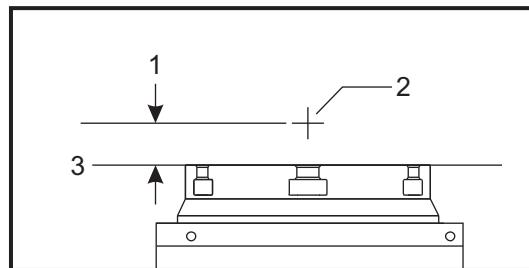
The 5th axis is installed in the same manner as the 4th axis:

1. Use Setting 78 to specify the rotary table model and 79 to define the 5th axis diameter which determines the angular feedrate.
2. Jog and command the 5th axis using the B address.

6.2.4 Tilt Axis Center-of-Rotation Offset (Tilting Rotary Products)

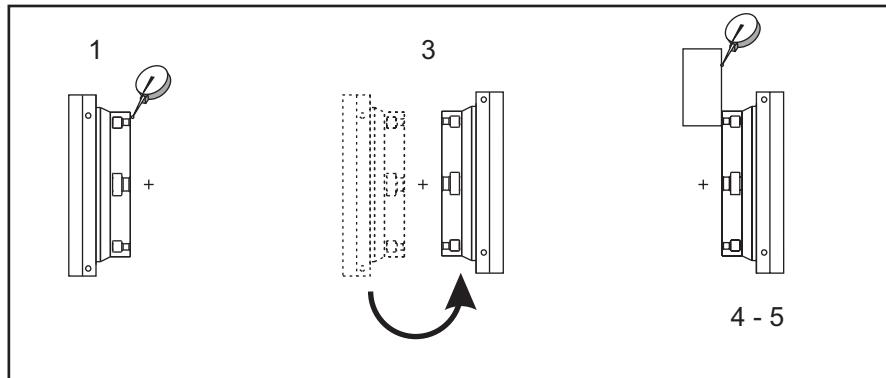
This procedure determines the distance between the plane of the rotary axis platter and the tilt axis centerline on tilting rotary products. Some CAM software applications require this offset value.

- F6.4:** Tilt Axis Center-of-Rotation Offset Diagram (side view): [1] Tilt Axis Center-of-Rotation Offset, [2] Tilt Axis, [3] Plane of the Rotary Axis Platter.



4th and 5th Axis Programming

F6.5: Tilt Axis Center-of-Rotation Illustrated Procedure. Numeric labels in this diagram correspond to the step numbers in the procedure.



1. Jog the tilt axis until the rotary platter is vertical. Attach a dial indicator to the machine spindle (or other surface independent of table motion) and indicate the platter face. Set the dial indicator to zero.



NOTE:

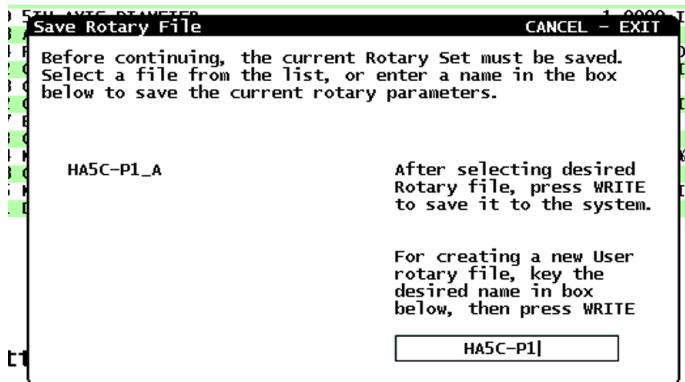
The rotary unit's orientation on the table determines which linear axis to jog in these steps. If the tilt axis is parallel to the X Axis, use the Y Axis in these steps. If the tilt axis is parallel to the Y Axis, use the X Axis in these steps.

2. Set the X- or Y-Axis operator position to zero.
3. Jog the tilt Axis 180 degrees.
4. Indicate the platter face from the same direction as the first indication:
 - a. Hold a 1-2-3 block against the platter face.
 - b. Indicate the face of the block that rests against the platter face.
 - c. Jog the X or Y Axis to zero the indicator against the block.
5. Read the new X- or Y-Axis operator position. Divide this value by 2 to determine the tilt axis center-of-rotation offset value.

6.2.5 Disabling 4th and 5th Axes

To disable 4th and 5th axes:

F6.6: Save Rotary Parameter Set



1. Turn off Setting 30 for the 4th axis and/or 78 for the 5th axis when you remove the rotary unit from the machine.

When you turn off Setting 30 or 78, a prompt appears to save the parameter set.



CAUTION: *Do not connect or disconnect any cables with the control on.*

2. Select a file using the up and down cursor arrows and press [ENTER] to confirm. The name of the currently selected parameter set appears in the box. You can change this filename to save a custom parameter set.
3. The machine generates an alarm if these settings are not turned off when the unit is removed.

6.3 Macros (Optional)

6.3.1 Macros Introduction



NOTE:

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

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

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

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

- **Tools For Immediate, On-Table Fixturing** - You can semi-automate many setup procedures to assist the machinist. You can reserve tools for immediate situations that you did not anticipate in your application design. For instance, suppose a company uses a standard clamp with a standard bolt hole pattern. If you discovered, after setup, that a fixture needs an additional clamp, and suppose that you programmed macro subprogram 2000 to drill the bolt pattern of the clamp, then you only need this two-step procedure to add the clamp to the fixture:
 - a) Jog the machine to the X, Y, and Z coordinates and angle where you want to place the clamp. Read the position coordinates from the machine display.
 - b) Execute this command in MDI mode:
G65 P2000 Xnnn Ynnn Znnn Ann ;
where nnn are the coordinates determined in Step a). Here, macro 2000 (P2000) does the work since it was designed to drill the clamp bolt hole pattern at the specified angle of A. Essentially, this is a custom canned cycle.
- **Simple Patterns That Are Repeated** - You can define and store repeated patterns with macros. For example:
 - a) Bolt hole patterns
 - b) Slotting
 - c) Angular patterns, any number of holes, at any angle, with any spacing
 - d) Specialty milling such as soft jaws
 - e) Matrix Patterns, (e.g. 12 across and 15 down)
 - f) Fly-cutting a surface, (e.g. 12 inches by 5 inches using a 3 inch fly cutter)

- **Automatic Offset Setting Based On The Program** - With macros, coordinate offsets can be set in each program so that setup procedures become easier and less error-prone (macro variables #2001-2800).
- **Probing** - Using a probe enhances the capabilities of the machine, some examples are:
 - a) Profiling of a part to determine unknown dimensions for machining.
 - b) Tool calibration for offset and wear values.
 - c) Inspection prior to machining to determine material allowance on castings.
 - d) Inspection after machining to determine parallelism and flatness values as well as location.

Useful G and M Codes

M00, M01, M30 - Stop Program

G04 - Dwell

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

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

Settings

There are 3 settings that affect macro programs (9000 series programs), these are **9xxx Progs Edit Lock** (Setting 23), **9xxx Progs Trace** (Setting 74), and **9xxx Progs Single BLK** (Setting 75).

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 your 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]... ;
```

Macros (Optional)

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

Look-ahead

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

```
#1101 = 1 ;
G04 P1. ;
#1101 = 0 ;
```

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

```
G103 P1 (See the G-code section of the manual for a further
explanation of G103) ;
;
#1101=1 ;
G04 P1. ;
;
;
;
#1101=0 ;
```

Block Look-Ahead and Block Delete

The Haas control uses block look-ahead to read and prepare for blocks of code that come after the current block of code. This lets the control transition smoothly from one motion to the next. G103 limits how far ahead the control looks at blocks of code. The Pnn address code in G103 specifies how far ahead the control is allowed to look. For additional information, refer to [G103 on page 299](#).

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

Using a

```
/ M99 (Sub-Program Return) ;
```

before a block with

```
M30 (Program End and Rewind) ;
```

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

6.3.2 Operation Notes

You can save or load macro variables through the RS-232 or USB port, much like settings, and offsets.

Variable Display Page

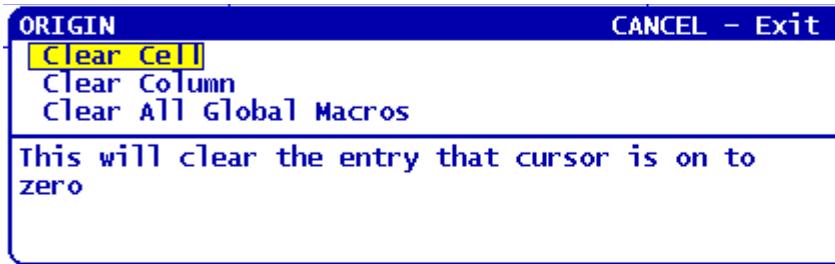
The macro variables #1 - #999 are displayed and modified through the Current Commands display.

1. Press **[CURRENT COMMANDS]** and use **[PAGE UP]/[PAGE DOWN]** to get to the **Macro Variables** page.

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

2. Enter a value and then press **[ENTER]** to set the macro variable. Press **[ORIGIN]** to clear macro variables, this displays the ORIGIN Clear entry popup. Make a selection from the choices and press **[ENTER]**.

- F6.7:** **[ORIGIN]** Clear Entry Popup. `clear Cell` - Clears the highlighted cell to zero.
`Clear Column` - Clears the active cursor column entries to zero.
`Clear All Global Macros` - Clears Global Macro entries (Macro 100-199, Macro 500-699, and Macro 800-999) to zero.



3. Entering the macro variable number and pressing up or down arrow searches for that variable.
4. The variables displayed represent values of the variables when the program runs. At times, this may be up to 15 blocks ahead of actual machine actions. Debugging programs is easier when a G103 P1 is inserted at the beginning of a program to limit block buffering and when G103 P1 is removed after debugging is completed.

Display User Defined Macros 1 and 2

You can display the values of any two user-defined macros (**Macro Label 1**, **Macro Label 2**).



NOTE:

The names Macro Label 1 and Macro Label 2 are changeable labels. just highlight the name, key in new name, and press [ENTER].

To set which two macro variables will display under **Macro Label 1** and **Macro Label 2** on the **Operation Timers & Setup** display window:

1. Press **[CURRENT COMMANDS]**.
2. Press **[PAGE UP]** or **[PAGE DOWN]** to reach the **Operation Timers & Setup** page.
3. Use arrow keys to pick the **Macro Label 1** or **Macro Label 2** entry field (to the right of the label).
4. Key in the variable number (without #) and press **[ENTER]**.

The field to the right of the entered variable number displays the current value.

Macro Arguments

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

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

Alphabetic Addressing

Address	Variable	Address	Variable
A	1	N	-
B	2	O	-
C	3	P	-
D	7	Q	17
E	8	R	18
F	9	S	19

Address	Variable	Address	Variable
G	-	T	20
H	11	U	21
I	4	V	22
J	5	W	23
K	6	X	24
L	-	Y	25
M	13	Z	26

Alternate Alphabetic Addressing

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

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

Macros (Optional)

G65 P9910 A1 B2 C3 ;

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

Integer Argument Passing (no decimal point)

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

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

Example:

G65 P2000 I1 J2 K3 I4 J5 K6;

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

Macro Variables

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

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

Local Variables

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

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

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

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

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

Global Variables

Global variables are accessible at all times. 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.

Sometimes, factory-installed options use global variables. For example, probing, pallet changers, etc.



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

System Variables

System variables let you interact with a variety of control conditions. System variable values can change the function of the control. When a program reads a system variable, it can modify its behavior based on the value in the variable. Some system variables have a Read Only status; this means that you cannot modify them. A brief table of system variables follows with an explanation of their use.

Variables	Usage
#0	Not a number (read only)
#1-#33	Macro call arguments
#100-#155	General-purpose variables saved on power off
#156-#199	Used by the probe (if installed)
#500-#549	General-purpose variables saved on power off
#556-#599	Probe calibration data (if installed)
#600-#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

Variables	Usage
#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
#2001-#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
#3012	Hour, minute, second
#3020	Power on timer (read only)
#3021	Cycle start timer

Macros (Optional)

Variables	Usage
#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.

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
#5801-#5900	Tool load monitor maximum load sensed so far
#5901-#6000	Tool load monitor limit

Macros (Optional)

VARIABLES	USAGE
#6001-#6277	Settings (read only)  NOTE: <i>The low order bits of large values will not appear in the macro variables for settings.</i>
#6501-#6999	Parameters (read only)  NOTE: <i>The low order bits of large values will not appear in the macro variables for parameters.</i>

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

VARIABLES	USAGE
#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.
#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

Macros (Optional)

VARIABLES	USAGE
#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

VARIABLES	USAGE
#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
⋮	
#15781 - #15786	G154 P90 additional work offsets
⋮	
#15881 - #15886	G154 P95 additional work offsets

Macros (Optional)

VARIABLES	USAGE
#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

6.3.3 System Variables In-Depth

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

Variables #550 through #580

These variables store probe calibration data. If these variables are overwritten, you will need to calibrate the probe again.

1-Bit Discrete Inputs

You can connect inputs designated as spare to external devices.

1-Bit Discrete Outputs

The Haas control is capable of controlling up to 56 discrete outputs. However, a number of these outputs are reserved for the Haas control to use.

Maximum Axis Loads

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

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

Tool Offsets

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

#2001-#2200	H geometry offsets (1-200) for length.
#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

Variable #3003 overrides the Single Block function in G-code. When #3003 has a value of 1, the control executes each G-code command continuously even though the Single Block function is ON. When #3003 has a value of zero, Single Block operates as normal. You must press [CYCLE START] to execute each line of code in single block mode.

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

Variable #3004

Variable #3004 overrides specific control features during operation.

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

```
...
(Approach code - [FEED HOLD] allowed) ;
#3004=1 (Disables [FEED HOLD]) ;
(Non-stoppable code - [FEED HOLD] not allowed) ;
#3004=0 (Enables [FEED HOLD]) ;
(Depart code - [FEED HOLD] allowed) ;
...
```

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

E = Enabled D = Disabled

#3004	Feed Hold	Feed Rate Override	Exact Stop Check
0	E	E	E
1	D	E	E

#3004	Feed Hold	Feed Rate Override	Exact Stop Check
2	E	D	E
3	D	D	E
4	E	E	D
5	D	E	D
6	E	D	D
7	D	D	D

#3006 Programmable Stop

You can add stops to the program that act like an M00 - The control stops and waits until you press [CYCLE START], then the program continues with the block after the #3006. In this example, the control displays the first 15 characters of the comment on the lower-left part of the screen.

```
#3006=1 (comment here) ;
```

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

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

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

Example:

G81 Drill Canned Cycle (Group 09)

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

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

Address codes A-Z (excluding G) are maintained as modal values. The information represented by the last line of code interpreted by the lookahead process is contained in variables #4101 through #4126. The numeric mapping of variable numbers to alphabetic addresses corresponds to the mapping under alphabetic addresses. For example, the value of the previously interpreted D address is found in #4107 and the last interpreted I value is #4104. When aliasing a macro to an M-code, you may not pass variables to the macro using variables #1 - #33. Instead, use the values from #4101 - #4126 in the macro.

#5001-#5006 Last Target Position

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

Axis Position Variables

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

#5021-#5026 Current Machine Coordinate Position

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


NOTE:

Values CANNOT be read while the machine is in motion.

The value of #5023 (Z) has tool length compensation applied to it.

#5041-#5046 Current Work Coordinate Position

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


NOTE:

The values CANNOT be read while the machine is in motion.

The value of #5043 (Z) has tool length compensation applied to it.

#5061-#5069 Current Skip Signal Position

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

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

#5081-#5086 Tool Length Compensation

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

#6996-#6999 Parameter Access With Macro Variables

These macro variables can access parameters 1 to 1000 and any of the parameter bits, as follows:

#6996: Parameter Number

#6997: Bit Number (optional)

#6998: Contains the value of the parameter number specified in variable #6996

#6999: Contains the bit value (0 or 1) of the parameter bit specified in variable #6997.



NOTE:

Variables #6998 and #6999 are read-only.

Usage

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

```
%  
#6996=601 (Specify parameter 601) ;  
#100=#6998 (Copy the value of parameter 601 to variable #100)  
;  
%
```

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

```
%  
#6996=57 (Specify parameter 57) ;  
#6997=0 (Specify bit zero) ;  
#100=#6999 (Copy parameter 57 bit 0 to variable #100) ;
```

Macros (Optional)

%



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 Variables

The status of the pallets from the Automatic Pallet Changer is checked with these variables:

#7501-#7506	Pallet priority
#7601-#7606	Pallet status
#7701-#7706	Part program numbers assigned to pallets
#7801-#7806	Pallet usage count
#3028	Number of pallet loaded on receiver

Work Offsets

Macro expressions can read and set all work offsets. This lets you preset coordinates to approximate locations, or set coordinates to values based upon the results of skip signal locations and calculations. When any of the offsets are read, the interpretation look-ahead queue is stopped until that block is executed.

#5201- #5206	G52 X, Y, Z, A, B, C OFFSET VALUES
#5221- #5226	G54 X, Y, Z, A, B, C OFFSET VALUES
#5241- #5246	G55 X, Y, Z, A, B, C OFFSET VALUES
#5261- #5266	G56 X, Y, Z, A, B, C OFFSET VALUES
#5281- #5286	G57 X, Y, Z, A, B, C OFFSET VALUES
#5301- #5306	G58 X, Y, Z, A, B, C OFFSET VALUES
#5321- #5326	G59X, Y, Z, A, B, C OFFSET VALUES
#7001- #7006	G110 X, Y, Z, A, B, C OFFSET VALUES

#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

Macros (Optional)

#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
#7381- #7386	G129 X, Y, Z, A, B, C OFFSET VALUES

#8550-#8567 Tooling

These variables give information on tooling. Set variable #8550 to the tool or tool group number, then access information for the selected tool/tool group with the read-only macros #8551-#8567. If you specify a tool group number, the selected tool is the next tool in that group.



NOTE: Macro variables #1801-#2000 give access to the same data as #8550-#8567.

6.3.4 Variable Usage

All variables are referenced with a number 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 its context. 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.

A variable can be used in place of a 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.

6.3.5 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 (in/mm). Macro syntax allows the address values to be replaced with any variable or expression.

The previous statement can be replaced by this code:

```
#1=1 ;  
#2=1.5 ;  
#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><-><variable>	A-#101
<address>[<expression>]	Z[#5041+3.5]
<address><->[<expression>]	Z-[SIN[#1]]

If the variable value does not agree with the address range, the control generates an alarm. For example, this code causes a range error alarm because the tool diameter numbers range from 0 to 200.

```
#1=250 ;
```

Macros (Optional)

```
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 G01 X#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, that address reference is ignored. For example, if #1 is undefined, then the block

```
G00 X1.0 Y#1 ;
```

becomes

```
G00 X1.0 ;
```

and 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

Function	Argument	Returns	Notes
ROUND[]	Decimal	Decimal	Round off a decimal
FIX[]	Decimal	Integer	Truncate fraction
ACOS[]	Decimal	Degrees	Arc cosine
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, the result 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 ;  
G00 X[ #1 + #1 ] ;  
(Table X Axis moves to 2.0067) ;  
G00 X[ ROUND[ #1 ] + ROUND[ #1 ] ] ;  
(Table X Axis moves to 2.0066) ;  
G00 A[ #1 + #1 ] ;  
(Axis rotates to 2.007) ;  
G00 A[ ROUND[ #1 ] + ROUND[ #1 ] ] ;  
(Axis rotates to 2.006) ;  
D[1.67] (Diameter 2 is made current) ;  
%
```

Fix vs. Round

```
%
```

Macros (Optional)

```
#1=3.54 ;
#2=ROUND[#1] ;
#3=FIX[#1].
%
```

#2 will be set to 4. #3 will be set to 3.

Operators

Operators have (3) categories: Boolean, Arithmetic, and Logical.

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

Here are four examples of how Boolean and Logical operators can be used:

Example	Explanation
IF [#1 EQ 0.0] GOTO100 ;	Jump to block 100 if value in variable #1 equals 0.0.
WHILE [#101 LT 10] DO1 ;	While variable #101 is less than 10 repeat loop DO1..END1.
#1=[1.0 LT 5.0] ;	Variable #1 is set to 1.0 (TRUE).
IF [#1 AND #2 EQ #3] GOTO1 ;	If variable #1 AND variable #2 are equal to the value in #3 then control jumps to block 1.

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 ;  
#2=2.0 ;  
#3=#1 OR #2 ;  
%
```

Here the variable #3 will contain 3.0 after the OR operation.

```
%  
#1=5.0 ;  
#2=3.0 ;  
IF [[#1 GT 3.0] AND [#2 LT 10]] GOTO1 ;  
%
```

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: *To achieve your desired results, be very careful when you use logical operators.*

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.

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 ;  
%
```

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 subprogram 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 subprogram. 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 ;
```

Assignment Statements

Assignment statements let you 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. This macro initializes a sequence of variables to any value. This example uses both direct and indirect assignments.

```
%  
O50001 (INITIALIZE A SEQUENCE OF VARIABLES) ;  
N1 IF [#2 NE #0] GOTO2 (B=base variable) ;  
#3000=1 (Base variable not given) ;  
N2 IF [#19 NE #0] GOTO3 (S=size of array) ;  
#3000=2 (Size of array not given) ;  
N3 WHILE [#19 GT 0] DO1 ;  
#19=#19-1 (Decrement count) ;  
#[#2+#19]=#22 (V=value to set array to) ;  
END1 ;  
M99 ;  
%
```

You could use the above macro 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 control can compute the block while the program runs, using the GOTO [expression] form, or it can pass the block in through a local variable, as in the GOTO#n form.

The GOTO rounds the variable or expression result that is associated with the Computed branch. For instance, if variable #1 contains 4.49 and the program contains a GOTO#1 command, the control attempts to transfer to a block that contains N4. If #1 contains 4.5, then the control transfers to a block that contains N5.

Example: You could develop this code skeleton into a program that adds serial numbers to parts:

```
%  
O50002 (COMPUTED BRANCHING) ;  
(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 ;  
%
```

With the above subprogram, you would use this call to engrave the fifth digit:

```
G65 P9200 D5 ;
```

Computed GOTOS using expression could be used to branch processing based on the results of reading hardware inputs. For example:

```
%  
GOTO [[#1030*2]+#1031] ;  
N0(1030=0, 1031=0) ;  
...M99 ;  
N1(1030=0, 1031=1) ;  
...M99 ;  
N2(1030=1, 1031=0) ;  
...M99 ;  
N3(1030=1, 1031=1) ;  
...M99 ;  
%
```

#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 subprogram. Conditional branching can only be used when macros are enabled. The Haas control allows two similar methods for accomplishing conditional branching:

```
IF [<conditional expression>] GOTOn
```

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 occurs; otherwise, the next block is executed.

In the Haas control, a <conditional expression> is also used with the M99 Pnnnn format. For example:

```
G00 X0 Y0 [#1EQ#2] M99 P5;
```

Here, the conditional is for the M99 portion of the statement only. The machine tool is instructed to go 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 be 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 GOTO.

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 G01 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 subprogram.

Here is an example of an IF statement that branches if a variable has been initialized to contain any value. Otherwise, processing continues and an alarm is 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 subprogram 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 subprogram 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 subprogram. A nest is a loop within a loop.

Although nesting of `WHILE` statements can only be up to three levels, there really is no limit since each subprogram 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 subprogram thus overcoming the limitation.

If two separate `WHILE` loops are in a subprogram, 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).

6.3.6 G65 Macro Subprogram Call Option (Group 00)

G65 is the command that calls a subprogram with the ability to pass arguments to it. The format follows:

```
G65 Pnnnn [Lnnnn] [arguments] ;
```

Arguments italicized in square brackets are optional. See the Programming section for more details on macro arguments.

The G65 command requires a **P** address corresponding to a program number currently in the control's memory. When the **L** address is used the macro call is repeated the specified number of times.

In Example 1, subprogram 1000 is called once without conditions passed to the subprogram. G65 calls are similar to, but not the same as, M98 calls. G65 calls can be nested up to 9 times, which means, program 1 can call program 2, program 2 can call program 3 and program 3 can call program 4.

Example 1:

```
%  
G65 P1000 (Call subprogram 1000 as a macro) ;  
M30 (Program stop) ;  
O01000 (Macro Subprogram) ;  
...  
M99 (Return from Macro Subprogram) ;  
%
```

In Example 2, subprogram 9010 is designed to drill a sequence of holes along a line whose slope is determined by the **X** and **Y** arguments that are passed to it in the G65 command line. The **Z** drill depth is passed as **Z**, the feed rate is passed as **F**, and the number of holes to be drilled is passed as **T**. The line of holes is drilled starting from the current tool position when the macro subprogram is called.

Example 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 ;  
O09010 (Diagonal hole pattern) ;  
F#9 (F=Feedrate) ;  
WHILE [#20 GT 0] D01 (Repeat T times) ;  
G91 G81 Z#26 (Drill To Z depth) ;  
#20=#20-1 (Decrement counter) ;  
IF [#20 EQ 0] GOTO5 (All holes drilled) ;  
G00 X#24 Y#25 (Move along slope) ;  
N5 END1 ;
```

```
M99 (Return to caller) ;
%
```

Aliasing

Aliased codes are user defined G and M-codes that reference a macro program. There are 10 G alias codes and 10 M alias codes available to users.

Aliasing is a means of assigning a G-code or M-code to a G65 P##### sequence. For instance, in the previous Example 2, it would be easier to write:

```
G06 X.5 Y.25 Z.05 F10. T10 ;
```

When aliasing, variables can be passed with a G-code; variables cannot be passed with an M-code.

Here, an unused G code has been substituted, G06 for G65 P9010. In order for the previous block to work, the parameter associated with subprogram 9010 must be set to 06 (Parameter 91).



NOTE:

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. This table lists which Haas parameters are reserved for macro subprogram aliasing.

F6.8: G- and M-code Aliasing

Haas Parameter	O Code	Haas Parameter	O Code
81	9000	91	9010
82	9001	92	9011
83	9002	93	9012
84	9003	94	9013
85	9004	95	9014
86	9005	96	9015
87	9006	97	9016
88	9007	98	9017
89	9008	99	9018
90	9009	100	9019

Setting an aliasing parameter to 0 disables aliasing for the associated subprogram. If an aliasing parameter is set to a G-code and the associated subprogram is not in memory, then an alarm will be given. When a G65 macro, Aliased-M or Aliased-G code is called, the control first looks for the sub-program in **MEM**. If it is not found in **MEM**, the control then looks for the sub-program on the active drive (**USB**, **HDD**). An alarm occurs if the sub-program is not found.

Macros (Optional)

When a G65 macro, Aliased-M or Aliased-G code is called, the control looks for the sub-program in memory and then in any other active drive if the sub-program cannot be located. The active drive may be memory, USB drive or hard drive. An alarm occurs if the control does not find the sub-program in either memory or an active drive.

6.3.7 Communication With External Devices - DPRNT[]

Macros allow additional capabilities to communicate with peripheral devices. With user provided devices you can digitize parts, provide runtime inspection reports, or synchronize controls.

Formatted Output

The DPRNT statement lets programs send formatted text to the serial port. DPRNT can print any text and any variable 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. These 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
#1= 1.5436 ;	
DPRNT [X#1[44]*Z#1[03]*T#1[40]] ;	X1.5436 Z 1.544 T 1
DPRNT [***MEASURED* INSIDE*DIAMETER** *] ;	MEASURED INSIDE DIAMETER
DPRNT [] ;	(no text, only a carriage return)
#1=123.456789 ;	
DPRNT [X-#1[35]] ;	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 look-ahead interpretation to one block, you would include this command at the start of your program: This causes the control to look ahead (2) blocks.

```
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 [**ENTER**]. Remember, you can use the [**HANDLE JOG**] control to maneuver through long DPRNT [] expressions.

Addresses with expressions can be somewhat confusing. In this case, the alphabetic address stands alone. For instance, this block contains an address expression in X:

```
G01 G90 X [COS [90]] Y3.0 (CORRECT) ;
```

Macros (Optional)

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.

G01 G90 X 0 Y3.0 (WRONG) ;

The above block will result in an alarm at runtime. The correct form looks as follows:

G01 G90 X0 Y3.0 (CORRECT) ;



NOTE:

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.

6.3.8 Fanuc-Style Macros Not Included

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	Aliasing, T code PROG 9000, VAR #149, 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 []	

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.

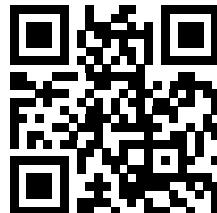
6.4 More Information Online

You can find programming information for other optional equipment in the online Haas Resource Center, including:

- Programmable Coolant Spigot (P-Cool)
- 300- and 1000-psi Through-Spindle Coolant (TSC)
- Intuitive Programming System (IPS)
- Wireless Intuitive Probing System (WIPS)

To access the site, go to www.HaasCNC.com and select the **Haas Resource Center**.

You can also scan this QR code with your mobile device to go directly to the options programming section of the Resource Center.



More Information Online

Chapter 7: G-codes

7.1 Introduction

This chapter gives detailed descriptions of the G-codes that you use to program your machine.


CAUTION:

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


NOTE:

The sample programs in this manual represent a very conservative programming style. The samples are intended to demonstrate safe and reliable programs, and they are not necessarily the fastest or most efficient way to operate a machine. The sample programs use G-codes that you might choose not to use in more efficient programs.

7.1.1 List of G-codes

Code	Description	Group	Page
G00	Rapid Motion Positioning	01	232
G01	Linear Interpolation Motion	01	233
G02	Circular Interpolation Motion CW	01	235
G03	Circular Interpolation Motion CCW	01	235
G04	Dwell	00	242
G09	Exact Stop	00	242
G10	Set Offsets	00	243
G12	Circular Pocket Milling CW	00	244

Introduction

Code	Description	Group	Page
G13	Circular Pocket Milling CCW	00	244
G17	XY Plane Selection	02	246
G18	XZ Plane Selection	02	246
G19	YZ Plane Selection	02	246
G20	Select Inches	06	247
G21	Select Metric	06	247
G28	Return To Machine Zero Point	00	247
G29	Return From Reference Point	00	247
G31	Feed Until Skip	00	248
G35	Automatic Tool Diameter Measurement	00	249
G36	Automatic Work Offset Measurement	00	251
G37	Automatic Tool Offset Measurement	00	253
G40	Cutter Compensation Cancel	07	254
G41	2D Cutter Compensation Left	07	254
G42	2D Cutter Compensation Right	07	254
G43	Tool Length Compensation + (Add)	08	254
G44	Tool Length Compensation - (Subtract)	08	254
G47	Text Engraving	00	255
G49	G43/G44/G143 Cancel	08	263
G50	Cancel Scaling	11	263
G51	Scaling	11	263
G52	Set Work Coordinate System	00 or 12	268
G53	Non-Modal Machine Coordinate Selection	00	268

Code	Description	Group	Page
G54	Select Work Coordinate System #1	12	269
G55	Select Work Coordinate System #2	12	269
G56	Select Work Coordinate System #3	12	269
G57	Select Work Coordinate System #4	12	269
G58	Select Work Coordinate System #5	12	269
G59	Select Work Coordinate System #6	12	269
G60	Uni-Directional Positioning	00	269
G61	Exact Stop Mode	15	269
G64	G61 Cancel	15	269
G65	Macro Subprogram Call Option	00	269
G68	Rotation	16	269
G69	Cancel G68 Rotation	16	273
G70	Bolt Hole Circle	00	273
G71	Bolt Hole Arc	00	274
G72	Bolt Holes Along an Angle	00	274
G73	High-Speed Peck Drilling Canned Cycle	09	275
G74	Reverse Tap Canned Cycle	09	276
G76	Fine Boring Canned Cycle	09	277
G77	Back Bore Canned Cycle	09	278
G80	Canned Cycle Cancel	09	281
G81	Drill Canned Cycle	09	281
G82	Spot Drill Canned Cycle	09	282
G83	Normal Peck Drilling Canned Cycle	09	284

Introduction

Code	Description	Group	Page
G84	Tapping Canned Cycle	09	286
G85	Boring Canned Cycle	09	287
G86	Bore and Stop Canned Cycle	09	288
G87	Bore In and Manual Retract Canned Cycle	09	289
G88	Bore In, Dwell, Manual Retract Canned Cycle	09	290
G89	Bore In, Dwell, Bore Out Canned Cycle	09	291
G90	Absolute Position Command	03	292
G91	Incremental Position Command	03	292
G92	Set Work Coordinate Systems Shift Value	00	292
G93	Inverse Time Feed Mode	05	293
G94	Feed Per Minute Mode	05	293
G95	Feed per Revolution	05	293
G98	Canned Cycle Initial Point Return	10	287
G99	Canned Cycle R Plane Return	10	295
G100	Cancel Mirror Image	00	296
G101	Enable Mirror Image	00	296
G102	Programmable Output to RS-232	00	298
G103	Limit Block Buffering	00	299
G107	Cylindrical Mapping	00	299
G110	#7 Coordinate System	12	301
G111	#8 Coordinate System	12	301
G112	#9 Coordinate System	12	301
G113	#10 Coordinate System	12	301

Code	Description	Group	Page
G114	#11 Coordinate System	12	301
G115	#12 Coordinate System	12	301
G116	#13 Coordinate System	12	301
G117	#14 Coordinate System	12	301
G118	#15 Coordinate System	12	301
G119	#16 Coordinate System	12	301
G120	#17 Coordinate System	12	301
G121	#18 Coordinate System	12	301
G122	#19 Coordinate System	12	301
G123	#20 Coordinate System	12	301
G124	#21 Coordinate System	12	301
G125	#22 Coordinate System	12	301
G126	#23 Coordinate System	12	301
G127	#24 Coordinate System	12	301
G128	#25 Coordinate System	12	301
G129	#26 Coordinate System	12	301
G136	Automatic Work Offset Center Measurement	00	301
G141	3D+ Cutter Compensation	07	303
G143	5-Axis Tool Length Compensation +	08	306
G150	General Purpose Pocket Milling	00	308
G153	5-Axis High Speed Peck Drilling Canned Cycle	09	316
G154	Select Work Coordinates P1-P99	12	317
G155	5-Axis Reverse Tap Canned Cycle	09	319

Introduction

Code	Description	Group	Page
G161	5-Axis Drill Canned Cycle	09	320
G162	5-Axis Spot Drill Canned Cycle	09	321
G163	5-Axis Normal Peck Drilling Canned Cycle	09	322
G164	5-Axis Tapping Canned Cycle	09	324
G165	5-Axis Boring Canned Cycle	09	325
G166	5-Axis Bore and Stop Canned Cycle	09	326
G169	5-Axis Bore and Dwell Canned Cycle	09	327
G174	CCW Non-Vertical Rigid Tap	00	328
G184	CW Non-Vertical Rigid Tap	00	328
G187	Setting the Smoothness Level	00	328
G188	Get Program From PST	00	329
G234	Tool Center Point Control (TCPC) (UMC)	08	329
G254	Dynamic Work Offset (DWO) (UMC)	23	329
G255	Cancel Dynamic Work Offset (DWO) (UMC)	23	329

About G-codes

G-codes tell the machine tool what type of action to do, such as:

- Rapid moves
- Move in a straight line or arc
- Set tool information
- Use letter addressing
- Define axis and beginning and ending positions
- Pre-set series of moves that bore a hole, cut a specific dimension, or a contour (canned cycles)

G-code commands are either modal or non-modal. A modal G-code stays in effect until the end of the program or until you command another G-code from the same group. A non-modal G-code affects only the line it is in; it does not affect the next program line. Group 00 codes are non-modal; the other groups are modal.

For a description of basic programming, refer to the basic programming section of the Programming chapter, starting on page 137.

**NOTE:**

The Intuitive Programming System (IPS) is an optional programming mode that lets you program part features without G-code.

**NOTE:**

A program block can contain more than one G-code, but you cannot put two G-codes from the same group in the same program block.

Canned Cycles

Canned cycles are G-codes that do repetitive operations such as drilling, tapping, and boring. You define a canned cycle with alphabetic address codes. While the canned cycle is active, the machine does the defined operation every time you command a new position, unless you specify not to.

Using Canned Cycles

You can program canned cycle X and Y positions in either absolute (G90) or incremental (G91).

Example:

```
%  
G81 G99 Z-0.5 R0.1 F6.5 (This drills one hole);  
    (at the present location) ;  
G91 X-0.5625 L9 (This drills 9 more holes 0.5625);  
    (equally spaced in the X-negative direction) ;  
%
```

There are (3) possible ways for a canned cycle to behave in the block in which you command it:

- If you command an X/Y position in the same block as the canned cycle G-code, the canned cycle executes. If Setting 28 is **OFF**, the canned cycle executes in the same block only if you command an X/Y position in that block.
- If Setting 28 is **ON**, and you command a canned cycle G-code with or without an X/Y position in the same block, the canned cycle executes in that block—either at the position where you commanded the canned cycle, or at the new X/Y position.
- If you include a loop count of zero (**L0**) in the same block as the canned cycle G-code, the canned cycle does not execute in that block. The canned cycle does not execute regardless of Setting 28 and whether or not the block also contains an X/Y position.

**NOTE:**

Unless otherwise noted, the program examples given here assume that Setting 28 is ON.

When a canned cycle is active, it repeats at every new X/Y position in the program. In the example above, with each incremental move of -0.5625 in the X axis, the canned cycle (G81) drills a 0.5" deep hole. The L address code in the incremental position command (G91) repeats this operation (9) times.

Canned cycles operate differently depending on whether incremental (G91) or absolute (G90) positioning is active. Incremental motion in a canned cycle is often useful, because it lets you use a loop (L) count to repeat the operation with an incremental X or Y move between cycles.

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) ;  
G70 I0.75 J10. L6 (6-hole bolt hole circle) ;  
%
```

The R plane value and the Z depth value are important canned cycle address codes. If you specify these addresses in a block with XY commands, the control does the XY move, and it does all of the subsequent canned cycles with the new R or Z value.

The X and Y positioning in 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 canned cycles. This is a programmed pause at the bottom of the hole to help break chips, provide a smoother finish, and relieve any tool pressure to hold closer tolerance.

**NOTE:**

A P address used for one canned cycle is used in others unless canceled (G00, G01, G80 or the [RESET] button).

You must define an S (spindle speed) command in or before the canned cycle G-code block.

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

The metric version of the feed formula is:

RPM times metric pitch = feedrate in mm per minute

Canned cycles also benefit from the use of Setting 57. If this setting is **ON**, the machine stops after the X/Y rapids before it moves the Z Axis. This is useful to avoid nicking the part when the tool exits the hole, especially if the R plane is close to the part surface.

**NOTE:**

The Z, R, and F addresses are required data for all canned cycles.

Canceling a Canned Cycle

G80 cancels all canned cycles. G00 or G01 code also cancel a canned cycle. A canned cycle stays active until G80, G00, or G01 cancels it.

Looping Canned Cycles

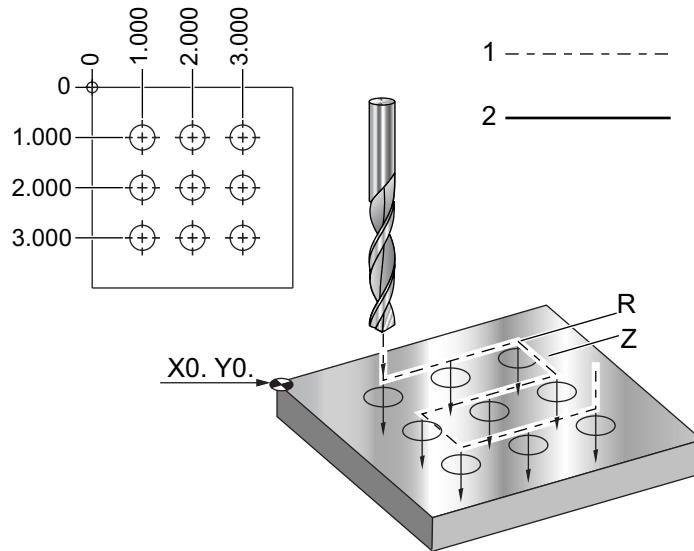
This is an example of a program that uses an incrementally looped drilling canned cycle.

**NOTE:**

The sequence of drilling used here is designed to save time and to follow the shortest path from hole to hole.

Introduction

F7.1: G81 Drilling Canned Cycle: [R] R Plane, [Z] Z Plane, [1] Rapid, [2] Feed.



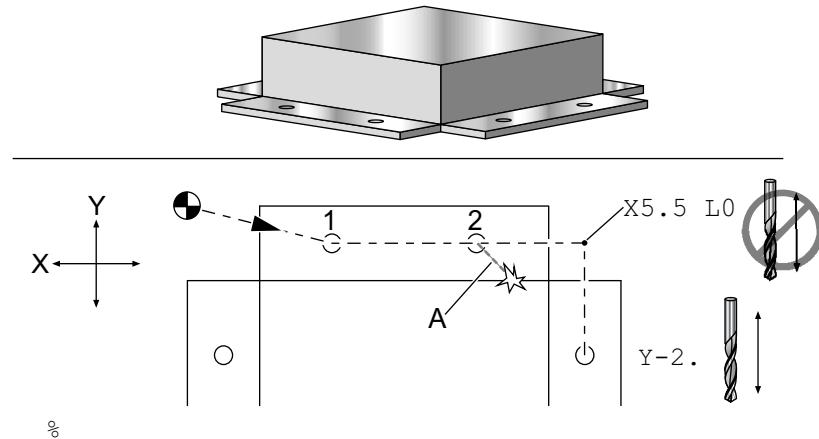
```
%  
O60810 (Drilling grid plate 3x3 holes) ;  
(G54 X0 Y0 is at the top-left of part) ;  
(Z0 is at the top of the part) ;  
(T1 is a drill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X1.0 Y-1.0 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G81 Z-1.5 F15. R.1 (Begin G81 & drill 1st hole) ;  
G91 X1.0 L2 (Drill 1st row of holes) ;  
G90 Y-2.0 (1st hole of 2nd row) ;  
G91 X-1.0 L2 (2nd row of holes) ;  
G90 Y-3.0 (1st hole of 3rd row) ;  
G91 X1.0 L2 (3rd row of holes) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

X/Y Plane Obstacle Avoidance in a Canned Cycle

If you put an `L0` on a canned cycle line, you can make an X, Y move without the Z-Axis canned operation. This is a good way to avoid obstacles in the X/Y plane.

Consider a 6" square aluminum block, with a 1" by 1" deep flange on each side. The print calls for two holes centered on each side of the flange. You use a `G81` canned cycle to make the holes. If you simply command the hole positions in the drill canned cycle, the control takes the shortest path to the next hole position, which puts the tool through the corner of the workpiece. To prevent this, command a position past the corner, so the move to the next hole position does not go through the corner. The drill canned cycle is active, but you do not want a drill cycle at that position, so use `L0` in this block.

- F7.2:** Canned Cycle Obstacle Avoidance. The program drills holes [1] and [2], then moves to X5.5. Because of the `L0` address in this block, there is no drill cycle in this position. Line [A] shows the path that the canned cycle would follow without the obstacle avoidance line. The next move is in the Y Axis only to the position of the third hole, where the machine does another drill cycle.



```
%  
O60811 (X Y OBSTACLE AVOIDANCE) ;  
(G54 X0 Y0 is at the top-left of part) ;  
(Z0 is at the top of the part) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X2. Y-0.5(Rapid to first position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 M08 (Activate tool offset 1) ;  
(Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G81 Z-2. R-0.9 F15. (Begin G81 & Drill 1st hole) ;  
X4. (Drill 2nd hole) ;  
X5.5 L0 (Corner avoidance) ;  
Y-2. (3rd hole) ;  
Y-4. (4th hole) ;
```

```
Y-5.5 L0 (Corner avoidance) ;  
X4. (5th hole) ;  
X2. (6th hole) ;  
X0.5 L0 (Corner avoidance) ;  
Y-4. (7th hole) ;  
Y-2. (8th hole) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

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
- ***B** - Optional B-Axis motion command
- ***C** - Optional C-axis motion command

*indicates optional

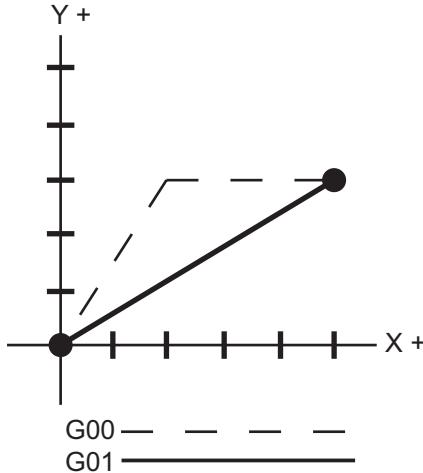
G00 is used to move the machine axes at the maximum speed. It is primarily used to quickly position the machine to a given point before each feed (cutting) command. This G code is modal, so a block with G00 causes all following blocks to be rapid motion until another Group 01 code is specified.

A rapid move also cancels an active canned cycle, just like G80 does.



NOTE:

Generally, rapid motion will not be in a single 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.

F7.3: G00 Multi-linear Rapid Motion

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

- * **X** - X-Axis motion command
- * **Y** - Y-Axis motion command
- * **Z** - Z-Axis motion command
- * **A** - A-Axis motion command
- * **B** - B-Axis motion command
- * **C** - C-axis motion command
- * **,R** - Radius of the arc
- * **,C** - Chamfer distance

*indicates optional

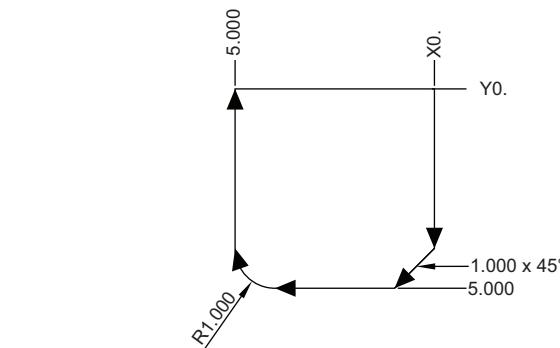
G01 moves the axes at a commanded feed rate. It is primarily used to cut the workpiece. A G01 feed can be a single axis move or a combination of the axes. The rate of axes movement is controlled by feedrate (F) value. This F value can be in units (inch or metric) per minute (G94) or per spindle revolution (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. If in G93, an F value is used on each line. Refer also to G93.

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 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

F7.4: Corner Rounding and Chamfering Example #1



```
%  
O60011 (G01 CORNER ROUNDING & CHAMFER) ;  
(G54 X0 Y0 is at the top-right of part) ;  
(Z0 is on top of the part) ;  
(T1 is an end mill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X0 Y0 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 Z-0.5 F20. (Feed to cutting depth) ;  
Y-5. ,C1. (Chamfer) ;  
X-5. ,R1. (Corner-round) ;  
Y0 (Feed to Y0.) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

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 after 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 after 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 after 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

- ***I** - Distance along X Axis to center of circle
- ***J** - Distance along Y Axis to center of circle
- ***K** - Distance along Z Axis to center of circle
- ***R** - Radius of circle
- ***X** - X-Axis motion command
- ***Y** - Y-Axis motion command
- ***Z** - Z-Axis motion command
- ***A** - A-Axis motion command

*indicates optional



NOTE:

I, J and K is the preferred method to program a radius. *R* is suitable for general radii.

These G codes are used to specify circular motion. Two axes are necessary to complete circular motion and the correct plane, G17-G19, 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, and 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:

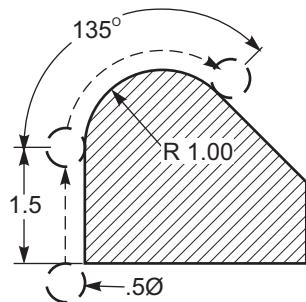
```
G02 I3.0 J4.0 (Assumes G17; XY plane) ;
```

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°.

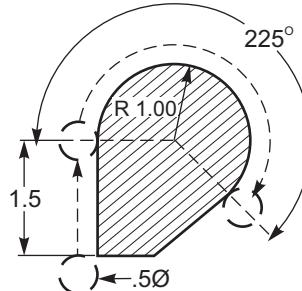
Programming Examples

F7.5: Positive R Address Programming Example



```
%  
O60021 (G02 POSITIVE R ADDRESS) ;  
(G54 X0 Y0 is at the bottom-left of part) ;  
(Z0 is on top of the part) ;  
(T1 is a .5 in dia endmill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X-0.25 Y-0.25 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 Z-0.5 F20. (Feed to cutting depth) ;  
G01 Y1.5 F12. (Feed to Y1.5) ;  
G02 X1.884 Y2.384 R1.25 (CW circular motion) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

F7.6: Negative R Address Programming Example



```
%  
O60022 (G02 NEGATIVE R ADDRESS) ;  
(G54 X0 Y0 is at the bottom-left of part) ;  
(Z0 is on top of the part) ;  
(T1 is a .5 in dia endmill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G54 (Safe startup) ;  
G00 G54 X-0.25 Y-0.25 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 Z-0.5 F20. (Feed to cutting depth) ;  
G01 Y1.5 F12. (Feed to Y1.5) ;  
G02 X1.884 Y0.616 R-1.25 (CW circular motion) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

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 block 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.

Introduction

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)

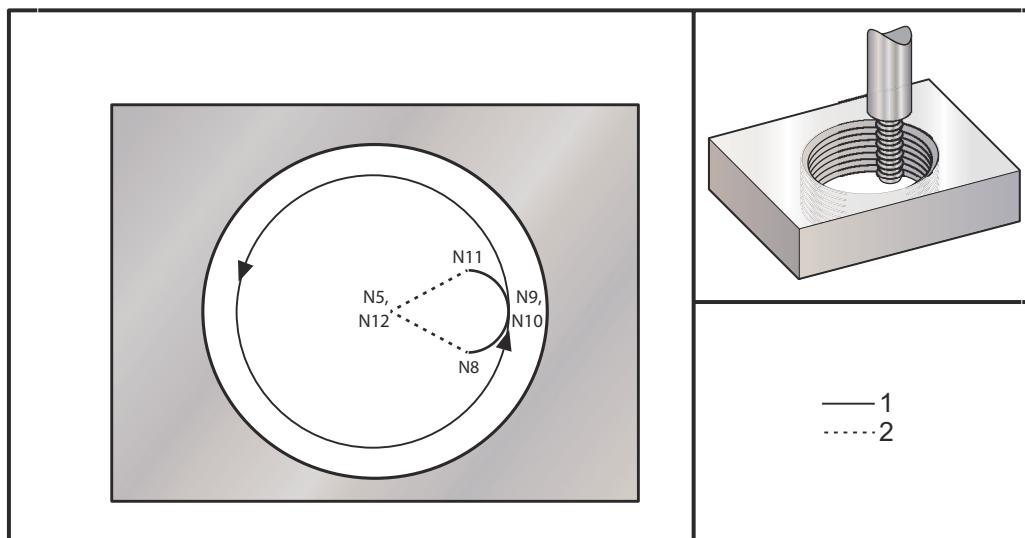
Thread Milling Example:

This program I.D. thread mills a 1.5 diameter x 8 TPI hole with a 0.750" diameter x 1.0" thread hob.

1. To start, take the hole diameter (1.500). Subtract the cutter diameter .750 and then divide by 2. $(1.500 - .750) / 2 = .375$
The result (.375) is the distance the cutter starts from the I.D. of the part.
2. 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.
3. 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).
4. The last step is to move away from the I.D. of the circle and turn off cutter compensation.

You cannot turn cutter compensation off or on during an arc movement. You must program a linear move, 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 you can adjust.

F7.7: Thread Milling Example, 1.5 Diameter X 8 TPI: [1]Tool Path, [2] Turn on and off cutter compensation.



**NOTE:**

Many thread mill manufacturers offer free online software to help you create your threading programs.

```
%  
O60023 (G03 THREAD MILL 1.5-8 UNC) ;  
(G54 X0 Y0 is at the center of the bore) ;  
(Z0 is on top of the part) ;  
(T1 is a .5 in dia thread mill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X0 Y0 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 Z-0.5156 F50. (Feed to starting depth) ;  
(Z-0.5 minus 1/8th of the pitch = Z-0.5156) ;  
G41 X0.25 Y-0.25 F10. D01 (cutter comp on) ;  
G03 X0.5 Y0 I0 J0.25 Z-0.5 (Arc into thread) ;  
(Ramps up by 1/8th of the pitch) ;  
I-0.5 J0 Z-0.375 F20. (Cuts full thread) ;  
(Z moving up by the pitch value to Z-0.375) ;  
X0.25 Y0.25 I-0.25 J0 Z-0.3594 (Arc out of thread) ;  
(Ramp up by 1/8th of the pitch) ;  
G40 G01 X0 Y1 (cutter comp off) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

N5 = XY at the center of the hole

N7 = Thread depth, minus 1/8 pitch

N8 = Enable Cutter Compensation

N9 = Arcs into thread, ramps up by 1/8 pitch

N10 = Cuts full thread, Z moving up by the pitch value

N11 = Arcs out of thread, ramps up 1/8 pitch

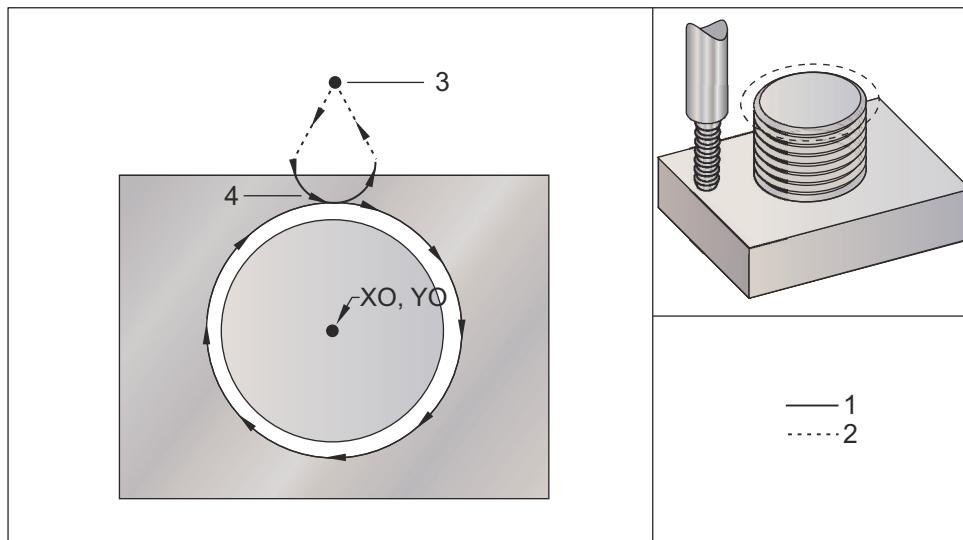
N12 = Cancel Cutter Compensation



NOTE: Maximum cutter compensation adjustability is .175.

O.D. Thread Milling

F7.8: O.D. Thread Milling Example, 2.0 diameter post x 16 TPI: [1] Tool Path [2] Rapid Positioning, Turn on and off cutter compensation, [3] Start Position, [4] Arc with Z.



```
%  
O60024 (G02 G03 THREAD MILL 2.0-16 UNC) ;  
(G54 X0 Y0 is at the center of the post) ;  
(Z0 is on top of the opost) ;  
(T1 is a .5 in dia thread mill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X0 Y2.4 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G00 Z-1. (Rapids to Z-1.) ;  
G01 G41 D01 X-0.5 Y1.4 F20. (Linear move) ;  
(Cutter comp on) ;  
G03 X0 Y0.962 R0.5 F25. (Arc into thread) ;  
G02 J-0.962 Z-1.0625 (Cut threads while lowering Z) ;  
G03 X0.5 Y1.4 R0.5 (Arc out of thread) ;  
G01 G40 X0 Y2.4 F20. (Linear move) ;  
(Cutter comp off) ;
```

```
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%
```

**NOTE:**

A cutter compensation move can consist of any X or Y move from any position as long as the move is greater than the amount being compensated.

Single-Point Thread Milling Example

This program is for a 1.0" diameter hole with a cutter diameter of .500" and a thread pitch of .125 (8TPI). This program positions itself in Absolute G90 and then switches to G91 Incremental mode on line N7.

The use of an Lxx value on line N10 allows us to repeat the thread milling arc multiple times, with a Single-Point Thread Mill.

```
%  
O60025 (G03 SNGL PNT THREAD MILL 1.5-8 UNC) ;  
(G54 X0 Y0 is at the center of the bore) ;  
(Z0 is on top of the part) ;  
(T1 is a .5 in dia thread mill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X0 Y0 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G91 G01 Z-0.5156 F50. (Feed to starting depth) ;  
(Z-0.5 minus 1/8th of the pitch = Z-0.5156) ;  
G41 X0.25 Y-0.25 F20. D01 (Cutter comp on) ;  
G03 X0.25 Y0.25 I0 J0.25 Z0.0156 (Arc into thread) ;  
(Ramps up by 1/8th of the pitch) ;  
I-0.5 J0 Z0.125 L5 (Thread cut, repeat 5 times) ;  
X-0.25 Y0.25 I-0.25 J0 Z0.0156 (Arc out of thread) ;  
(Ramps up by 1/8th of the pitch) ;  
G40 G01 X-0.25 Y-0.25 (Cutter comp off) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;
```

Introduction

```
M30 (End program) ;  
%
```

Specific line description:

N5 = XY at the center of the hole

N7 = Thread depth, minus 1/8 pitch. Switches to G91

N8 = Enable Cutter Compensation

N9 = Arcs into thread, ramps up by 1/8 pitch

N10 = Cuts full thread, Z moving up by the pitch value

N11 = Arcs out of thread, ramps up 1/8 pitch

N12 = Cancel Cutter Compensation

N13 = Switches back to G90 Absolute positioning

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 specifies a delay or dwell in the program. The block with G04 delay for the time specified by the P address code. For example:

```
G04 P10.0. ;
```

Delays the program for 10 seconds.



NOTE:

G04 P10. is a dwell of 10 seconds; G04 P10 is a dwell of 10 milliseconds. Make sure you use decimal points correctly so that you specify the correct dwell time.

G09 Exact Stop (Group 00)

G09 code is used to specify a controlled axes stop. It affects only the block in which it is commanded. It is non-modal and does not affect the blocks that come after the block where it is commanded. Machine moves decelerate to the programmed point before the control processes the next command.

G10 Set Offsets (Group 00)

G10 lets you set offsets within the program. G10 replaces manual offset entry (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)

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** X-Axis zero location.

* **Y** Y-Axis zero location.

* **Z** Z-Axis zero location.

* **A** A-Axis zero location.

* **B** B-Axis zero location.

* **C** C-Axis zero location.

*indicates optional

```
%  
O60100 (G10 SET OFFSETS) ;  
G10 L2 P1 G91 X6.0 ;  
    (Move coordinate G54 6.0 to the right) ;  
;  
G10 L20 P2 G90 X10. Y8. ;  
    (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 G-codes mill circular shapes. They are different only in that G12 uses a clockwise direction and G13 uses a counterclockwise direction. Both G-codes use the default XY circular plane (G17) and imply the use of G42 (cutter compensation) for G12 and G41 for G13. G12 and G13 are non-modal.

*D - Tool radius or diameter selection**

F - Feedrate

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)

Z - Depth of cut or increment

*indicates optional

**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 you do not want to use cutter compensation. If you do not specify a D value in the G12/G13 block, the control uses the last commanded D value, even if it was previously canceled with a G40.

Rapid-position the tool to the center of the circle. To remove all the material inside 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.

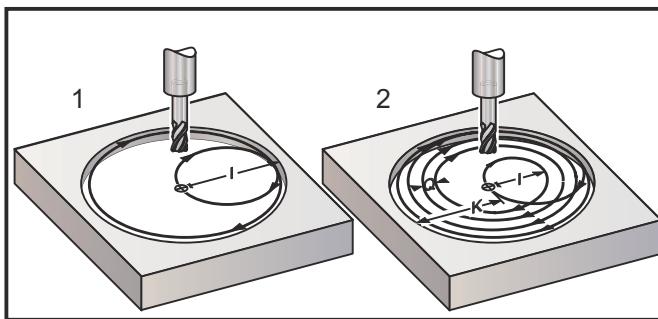
```
%  
O60121 (SAMPLE G12 AND G13) ;  
(G54 X0 Y0 is center of first pocket) ;  
(Z0 is on top of the part) ;  
(T1 is a .25 in. dia endmill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X0 Y0 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Tool offset 1 on) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G12 I0.75 F10. Z-1.2 D01 (Finish pocket CW) ;  
G00 Z0.1 (Retract) ;  
X5. (Move to center of next pocket) ;
```

```

G12 I0.3 K1.5 Q1. F10. Z-1.2 D01 ;
(Rough & finish CW) ;
G00 Z0.1 (Retract) ;
X10. (Move to center of next pocket) ;
G13 I1.5 F10. Z-1.2 D01 (Finish CCW) ;
G00 Z0.1 (Retract) ;
X15. (Move to center of the last pocket) ;
G13 I0.3 K1.5 Q0.3 F10. Z-1.2 D01 ;
(Rough & finish CCW) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%

```

F7.9: Circular Pocket Milling, G12 Clockwise shown: [1] I only, [2] I, K and Q only.



These G codes assume cutter compensation, so you do not need to program G41 or G42 in the program block. However, you must include a **D** offset number, for cutter radius or diameter, to adjust the circle diameter.

These program examples show the G12 and G13 format, and the different ways that you can write these programs.

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.

```
%  
O60131 (G13 G91 CCW EXAMPLE) ;  
(G54 X0 Y0 is center of 1st pocket) ;  
(Z0 is on top of the part) ;  
(T1 is a 0.5 in. dia endmill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X0 Y0 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G13 G91 Z-.5 I.400 K2.0 Q.400 L4 D01 F20. ;  
(Rough & finish CCW) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 G90 Z0.1 M09 (Rapid retract, coolant off) ;  
G53 G49 Z0 M05 (Z home, spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

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).

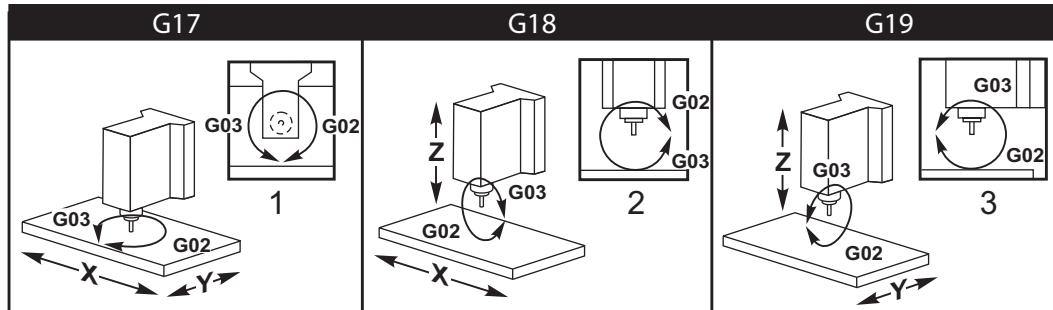
If cutter radius compensation is selected (G41 or G42), only use the XY plane (G17) for circular motion.

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.

F7.10: G17, G18, and G19 Circular Motion Diagrams: [1] Top view, [2] Front view, [3] Right view.



G20 Select Inches / G21 Select Metric (Group 06)

Use G20 (inch) and G21 (mm) codes are to make sure that the inch/metric selection is set correctly for the program. Use Setting 9 to select between inch and metric programming. G20 in a program causes an alarm if Setting 9 is not set to inch.

G28 Return to Machine Zero Point (Group 00)

The G28 code returns all axes (X, Y, Z, A and B) simultaneously to the machine zero position when no axis is specified on the G28 line.

Alternatively, when one or more axes locations are specified on the G28 line, G28 will move to the specified locations and then to machine zero. This is called the G29 reference point; it is saved automatically for optional use in G29.

G28 also cancels tool length offsets.

Setting 108 affects the way that rotary axes return when you command a G28.

```
%  
G28 G90 X0 Y0 Z0 (moves to X0 Y0 Z0) ;  
G28 G90 X1. Y1. Z1. (moves to X1. Y1. Z1.) ;  
G28 G91 X0 Y0 Z0 (moves directly to machine zero) ;  
G28 G91 X-1. Y-1. Z-1 (moves incrementally -1.) ;  
%
```

G29 Return From Reference Point (Group 00)

G29 moves 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)

This G-code is used to record a probed location to a macro variable.

F - Feedrate

***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

***C** - C-axis absolute motion command (UMC)

*indicates optional

This G-code moves the programmed axes while looking for a signal from the probe (skip signal). The specified move is started and continues until the position is reached or the probe receives a skip signal. If the probe receives a skip signal during the G31 move, the control will beep and the skip signal position will be recorded to macro variables. The program will then execute the next line of code. If the probe does not receive a skip signal during the G31 move, the control will not beep and the skip signal position will be recorded at the end of the programmed move. The program will continue.

Macro variables #5061 through #5066 are designated to store skip signal positions for each axis. For more information about these skip signal variables see the macro section of this manual.

Notes:

This code is non-modal and only applies to the block of code in which G31 is specified.

Do not use Cutter Compensation (G41, G42) with a G31.

The G31 line must have a Feed command. To avoid damaging the probe, use a feed rate below F100. (inch) or F2500. (metric).

Turn on the probe before using G31.

If your mill has the standard Renishaw probing system, use the following commands to turn on the probe.

Use the following code to turn on the spindle probe.

```
M59 P1134 ;
```

Use the following code to turn on the tool-setting probe.

```
%  
M59 P1133 ;  
G04 P1.0 ;  
M59 P1134 ;
```

%

Use the following code to turn off either probe.

M69 P1134 ;

Also see M75, M78 and M79 ;

Sample program:

This sample program measures the top surface of a part with the spindle probe traveling in the Z negative direction. To use this program, the G54 part location must be set at, or close to the surface to be measured.

```
%  
O60311 (G31 SPINDLE PROBE) ;  
(G54 X0. Y0. is at the center of the part) ;  
(Z0. is at, or close to the surface) ;  
(T1 is a Spindle probe) ;  
(PREPARATION) ;  
T1 M06 (Select Tool 1) ;  
G00 G90 G54 X0 Y0 (Rapid to X0. Y0.) ;  
M59 P1134 (Spindle probe on) ;  
G43 H1 Z1. (Activate tool offset 1) ;  
(PROBING) ;  
G31 Z-0.25 F50. (Measure top surface) ;  
Z1. (Retract to Z1.) ;  
M69 P1134 (Spindle probe off) ;  
(COMPLETION) ;  
G00 G53 Z0. (Rapid retract to Z home) ;  
M30 (End program) ;  
%
```

G35 Automatic Tool Diameter Measurement (Group 00)

(This G-code is optional and requires a probe)

This G-code is used to set a tool diameter offset.

F - Feedrate

***D** - Tool diameter offset number

***X** - X-Axis command

***Y** - Y-Axis command

*indicates optional

Automatic Tool Diameter Offset Measurement function (G35) is used to set the tool diameter (or radius) using two touches 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.

Introduction

Setting 63 Tool Probe Width is used to reduce the measurement of the tool by the width of the tool probe. See the settings section of this manual for more information about Setting 63.

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:

This code is non-modal and only applies to the block of code in which G35 is specified.

Do not use Cutter Compensation (G41, G42) with a G35.

To avoid damaging the probe, use a feed rate below F100. (inch) or F2500. (metric).

Turn on the tool-setting probe before using G35.

If your mill has the standard Renishaw probing system, use the following commands to turn on the tool-setting probe.

```
%  
M59 P1133 ;  
G04 P1.0 ;  
M59 P1134 ;  
%
```

Use the following commands to turn off the tool-setting probe.

```
M69 P1134 ;
```

Turn on the spindle in reverse (M04), for a right handed cutter.

Also see M75, M78, and M79.

Also see G31.

Sample program:

This sample program measures the diameter of a tool and records the measured value to the tool offset page. To use this program, the G59 Work Offset location must be set to the tool-setting probe location.

```
%  
O60351 (G35 MEASURE AND RECORD TOOL DIA OFFSET) ;  
(G59 X0 Y0 is the tool setting probe location) ;  
(Z0 is at the surface of tool-setting probe) ;  
(T1 is a spindle probe) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G59 X0 Y-1. (Rapid tool next to probe) ;  
M59 P1133 (Select tool-setting probe) ;  
G04 P1. (Dwell for 1 second) ;
```

```

M59 P1134 (Probe on) ;
G43 H01 Z1. (Activate tool offset 1) ;
S200 M04 (Spindle on CCW) ;
(BEGIN PROBING BLOCKS) ;
G01 Z-0.25 F50. (Feed tool below surface of probe) ;
G31 Y-0.25 F10. M75 (Set reference point) ;
G01 Y-1. F25. (Feed away from the probe) ;
Z0.5 (Retract above the probe) ;
Y1. (Move over the probe in Y-axis) ;
Z-0.25 (Move tool below surface of the probe) ;
G35 Y0.205 D01 F10. ;
(Measure & record tool diameter) ;
(Records to tool offset 1);
G01 Y1. F25. (Feed away from the probe) ;
Z1. (Retract above the probe) ;
M69 P1134 (Probe off) ;
(BEGIN COMPLETION BLOCKS) ;
G00 G53 Z0. (Rapid retract to Z home) ;
M30 (End program) ;
%

```

G36 Automatic Work Offset Measurement (Group 00)

(This G-code is optional and requires a probe)

This G-code is used to set work offsets with a probe.

F - Feedrate

- ***I** - Offset distance along X-Axis
- ***J** - Offset distance along Y-Axis
- ***K** - Offset distance along Z-Axis
- ***X** - X-Axis motion command
- ***Y** - Y-Axis motion command
- ***Z** - Z-Axis motion command

*indicates optional

Automatic Work Offset Measurement (G36) is used to command a probe to set work coordinate offsets. A G36 will feed the axes of the machine in an effort to probe the work piece with a spindle mounted probe. The axis (axes) will move until a signal from the probe is received or the end of the programmed move is reached. Tool compensation (G41, G42, G43, or G44) must not be active when this function is performed. The point where the skip signal is received becomes the zero position for the currently active work coordinate system of each axis programmed.

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:

This code is non-modal and only applies to the block of code in which G36 is specified.

The points probed are offset by the values in Settings 59 through 62. See the settings section of this manual for more information.

Do not use Cutter Compensation (G41, G42) with a G36.

Do not use tool length Compensation (G43, G44) with G36

To avoid damaging the probe, use a feed rate below F100. (inch) or F2500. (metric).

Turn on the spindle probe before using G36.

If your mill has the standard Renishaw probing system, use the following commands to turn on the spindle probe.

```
M59 P1134 ;
```

Use the following commands to turn off the spindle probe.

```
M69 P1134 ;
```

Also see M78, and M79.

```
%  
O60361 (G36 AUTO WORK OFFSET MEASUREMENT) ;  
(G54 X0 Y0 is at the top-center of the part) ;  
(Z0 is at the surface of part) ;  
(T1 is a Spindle probe) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 20) ;  
G00 G90 G54 X0 Y1. (Rapid to 1st position) ;  
(BEGIN PROBING BLOCKS) ;  
M59 P1134 (Spindle probe on) ;  
Z-.5 (Move the probe below surface of part) ;  
G01 G91 Y-0.5 F50. (Feed towards the part) ;  
G36 Y-0.7 F10. (Measure and record Y offset) ;  
G91 Y0.25 F50. (Move incrementally away from part) ;  
G00 Z1. (Rapid retract above part) ;  
M69 P1134 (Spindle probe off) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 G90 G53 Z0. (Rapid retract to Z home) ;  
M30 (End program) ;  
%
```

G37 Automatic Tool Offset Measurement (Group 00)

(This G-code is optional and requires a probe)

This G-code is used to set tool length offsets.

F - Feedrate

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 tool-setting 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 distance between the current work coordinate zero point and the point where the probe is touched. If a non-zero Z value is on the G37 line of code the resulting tool offset will be shifted by the non-zero amount. Specify Z0 for no offset shift.

The work coordinate system (G54, G55, etc.) and the tool length offsets

(H01-H200) may be selected in this block or the previous block.

NOTES:

This code is non-modal and only applies to the block of code in which G37 is specified.

A non-zero H code and either G43 or G44 must be active.

To avoid damaging the probe, use a feed rate below F100. (inch) or F2500. (metric).

Turn on the tool-setting probe before using G37.

If your mill has the standard Renishaw probing system, use the following commands to turn on the tool-setting probe.

```
%  
M59 P1133 ;  
G04 P1. ;  
M59 P1134 ;  
%
```

Use the following command to turn off the tool-setting probe.

```
M69 P1134 ;
```

Also see M78 and M79.

Sample program:

This sample program measures the length of a tool and records the measured value on the tool offset page. To use this program, the G59 work offset location must be set to the tool-setting probe location.

```
%  
O60371 (G37 AUTO TOOL OFFSET MEASUREMENT) ;  
(G59 X0 Y0 is center of tool-setting probe) ;  
(Z0 is at the surface of tool-setting probe) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G59 X0 Y0 (Rapid to center of the probe) ;  
G00 G43 H01 Z5. (Activate tool offset 1) ;  
(BEGIN PROBING BLOCKS) ;  
M59 P1133 (Select tool-setting probe) ;  
G04 P1. (Dwell for 1 second) ;  
M59 P1134 (Probe on) ;  
G37 H01 Z0 F30. (Measure & record tool offset) ;  
M69 P1134 (Probe off) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 G53 Z0. (Rapid retract to Z home) ;  
M30 (End program) ;  
%
```

G40 Cutter Comp Cancel (Group 07)

G40 cancels 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. For more information, refer to the Cutter Compensation section.

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)

G47 lets you engrave a line of text, or sequential serial numbers, with a single G-code. To use G47, Settings 29 (G91 Non-Modal) and 73 (G68 Incremental Angle) must be **OFF**.



NOTE: *Engraving along an arc is not supported.*

- ***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 in/mm (minimum = 0.001 inch); default is 1.0 inch
- P** - 0 for literal text 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

*indicates optional

Literal Text Engraving

This method is used to engrave text on a part. The text should be in the form of a comment on the same line as the G47 command. For example, G47 P0 (TEXT TO ENGRAVE), will engrave *TEXT TO ENGRAVE* on the part.



NOTE: *Corner rounding can cause engraved text to appear rounded and make them harder to read. To improve the sharpness and readability of engraved text, consider lowering the corner-rounding values with a G187 E.xxx value before the G47 command. Suggested starting E values are E0.002 (inch) or E0.05 (metric). Command a G187 alone after the engraving cycle to restore the default corner-rounding level. Refer to the example below:*

```
G187 E.002 (PREFACE ENGRAVING WITH A G187 E.xxx)
G47 P0 X.15 Y0. I0. J.15 R.1 Z-.004 F80. E40. (Engraving Text)
G00 G80 Z0.1
G187 (RESTORE NORMAL CORNER ROUNDING FOR SMOOTHNESS)
```

The characters available for engraving are:

A-Z, a-z 0-9, and ` ~ ! @ # \$ % ^ & * - _ = + [] { } \ | ; : ' " , . / < > ?

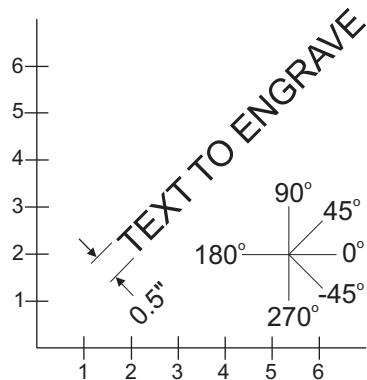
Introduction

Not all of these characters can be entered from the control. When programming from the mill keypad, or engraving parenthesis (), refer to the following Engraving Special Characters section.

This example creates the figure shown.

```
%  
O60471 (G47 TEXT ENGRAVING) ;  
(G54 X0 Y0 is at the bottom-left of part) ;  
(Z0 is on top of the part) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X2. Y2. (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G47 P0 (TEXT TO ENGRAVE) X2. Y2. I45. J0.5 R0.05 Z-0.005 F15.  
E10. ;  
(Starts at X2. Y2., engraves text at 45 deg) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 G80 Z0.1 (Cancel canned cycle) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

F7.11: Engraving Program Example



In this example, G47 P0 selects literal string engraving. X2.0 Y2.0 sets the starting point for the text at the bottom left corner of first letter. I45. places the text at a positive 45° angle. J.5 sets the text height to 0.5 units-in/mm. R.05 retracts cutter to 0.05 units above part after engraving. Z-.005 sets an engraving depth of -.005 units. F15.0 sets an engraving, XY move, feedrate of 15 units per minute. E10.0 sets a plunge, -Z move, feedrate of 10 units per minute.

Initial Serial Number

There are two ways to set the initial serial number to be engraved. The first requires replacing the # symbols within the parenthesis with the first number to be engraved. With this method, nothing is engraved when the G47 line is executed (it is only setting the initial serial number). Execute this once and then change the value within the parenthesis back to # symbols to engrave normally.

The following example will set the initial serial number to be engraved to 0001. Run this code once and then change (0001) to (####).

```
G47 P1 (0001) ;
```

The second method for setting the initial serial number to be engraved is to change the Macro Variable where this value is stored (Macro Variable 599). The Macros option does not need to be enabled.

Press **[CURRENT COMMANDS]** then press **[PAGE UP]** or **[PAGE DOWN]** as needed to display the **MACRO VARIABLES** page. From that screen, enter 599 and press Down cursor.

Once 599 is highlighted on the screen, type in the initial serial number to engrave, **[1]** for example, then press **[ENTER]**.

The same serial number can be engraved multiple times on the same part with the use of a macro statement. The macros option is required. A macro statement as shown below could be inserted between two G47 engraving cycles to keep the serial number from incrementing to the next number. For more details, see the Macros section of this manual.

Macro Statement: #599=[#599-1]

Engraving Sequential Serial Numbers

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 set the number of digits in the serial number. For example, G47 P1 (####), limits the number to four digits while (#) would limit the serial number to two digits.

This program engravings a four digit serial number.

```
%  
000037 (SERIAL NUMBER ENGRAVING) ;  
T1 M06 ;  
G00 G90 G98 G54 X0. Y0. ;  
S7500 M03 ;
```

Introduction

```
G43 H01 Z0.1 ;
G47 P1 (###) X2. Y2. I0. J0.5 R0.05 Z-0.005 F15. E10. ;
G00 G80 Z0.1 ;
M05 ;
G28 G91 Z0 ;
M30 ;
%
```

Engraving Around the Outside of a Rotary Part (G47, G107)

You can combine a G47 Engraving cycle with a G107 Cylindrical Mapping cycle to engrave text (or a serial number) along the outside diameter of a rotary part.

This code engravess a four digit serial number along the outer diameter of a rotary part.

```
%  
O60472 (G47 SERIAL NUMBER ENGRAVING) ;  
(G54 X0 Y0 is at the bottom left of the part) ;  
(Z0 is on top of the part) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X2. Y2. (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G47 P1 (###) X2. Y2. J0.5 R0.05 Z-0.005 F15. E10. ;  
(Engraves serial number) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

For more details on this cycle, refer to the G107 section.

Literal String Engraving (G47 P0)

This method is used to engrave text on a part. The text should be in the form of a comment on the same line as the G47 command. For example, G47 P0 (TEXT TO ENGRAVE), will engrave TEXT TO ENGRAVE on the part.



NOTE:

Engraving along an arc is not supported.

The characters available for engraving, using this method are:

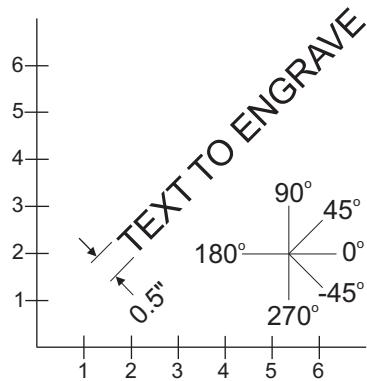
A-Z, a-z 0-9, and ` ~ ! @ # \$ % ^ & * - _ = + [] { } \ | ; : ' " , . / < > ?

Not all of these characters can be entered from the control. When programming from the mill keypad, or engraving parenthesis (), refer to the following Engraving Special Characters section.

This example creates the figure shown.

```
%  
O60471 (G47 TEXT ENGRAVING) ;  
(G54 X0 Y0 is at the bottom-left of part) ;  
(Z0 is on top of the part) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X2. Y2. (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G47 P0 (TEXT TO ENGRAVE) X2. Y2. I45. J0.5 R0.05 Z-0.005 F15.  
E10. ;  
(Starts at X2. Y2., engraves text at 45 deg) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 G80 Z0.1 (Cancel canned cycle) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

F7.12: Engraving Program Example



Introduction

In this example, G47 P0 selects literal string engraving. X2.0 Y2.0 sets the starting point for the text at the bottom left corner of first letter. I45. places the text at a positive 45° angle. J.5 sets the text height to 0.5 units-in/mm. R.05 retracts cutter to 0.05 units above part after engraving. Z-.005 sets an engraving depth of -.005 units. F15.0 sets an engraving, XY move, feedrate of 15 units per minute. E10.0 sets a plunge, -Z move, feedrate of 10 units per minute.

Engraving Special Characters

Engraving Special Characters involves using G47 with specific P values (G47 P32-126).

P- values to engrave specific characters

T7.1: G47 P Values for Special Characters

32		space	59	;	semicolon
33	!	exclamation mark	60	<	less than
34	"	double quotation mark	61	=	equals
35	#	number sign	62	>	greater than
36	\$	dollar sign	63	?	question mark
37	%	percent sign	64	@	at sign
38	&	ampersand	65-90	A-Z	capitol letters
39	,	closed single quote	91	[open square bracket
40	(open parenthesis	92	\	backslash
41)	close parenthesis	93]	closed square bracket
42	*	asterisk	94	^	carrot
43	+	plus sign	95	_	underscore
44	,	comma	96	'	open single quote
45	-	minus sign	97-122	a-z	lowercase letters
46	.	period	123	{	open curly bracket
47	/	slash	124		vertical bar

48-57	0-9	numbers	125	}	closed curly bracket
58	:	colon	126	~	tilde

Example:

To engrave \$2.00, you need (2) blocks of code. The first block uses a P36 to engrave the dollar sign (\$), and the second block uses P0 (2.00).



NOTE:

Shift the X/Y start location between the first and second line of code to make a space between the dollar sign and the 2.

This is the only method to engrave parenthesis () .

Setting Initial Serial Number to be Engraved

There are two ways to set the initial serial number to be engraved. The first requires replacing the # symbols within the parenthesis with the first number to be engraved. With this method, nothing is engraved when the G47 line is executed (it is only setting the initial serial number). Execute this once and then change the value within the parenthesis back to # symbols to engrave normally.

The following example will set the initial serial number to be engraved to 0001. Run this code once and then change (0001) to #####.

```
G47 P1 (0001) ;
```

The second method for setting the initial serial number to be engraved is to change the Macro Variable where this value is stored (Macro Variable 599). The Macros option does not need to be enabled.

Press [CURRENT COMMANDS] then press [PAGE UP] or [PAGE DOWN] as needed to display the MACRO VARIABLES page. From that screen, enter 599 and press Down cursor.

Once 599 is highlighted on the screen, type in the initial serial number to engrave, [1] for example, then press [ENTER].

The same serial number can be engraved multiple times on the same part with the use of a macro statement. The macros option is required. A macro statement as shown below could be inserted between two G47 engraving cycles to keep the serial number from incrementing to the next number. For more details, see the Macros section of this manual.

Macro Statement: #599=[#599-1]

Sequential Serial Number Engraving (G47 P1)

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 set the number of digits in the serial number. For example, G47 P1 (####), limits the number to four digits while (##) would limit the serial number to two digits.



NOTE: *Engraving along an arc is not supported.*

This program engraves a four digit serial number.

```
%  
O00037 (SERIAL NUMBER ENGRAVING) ;  
T1 M06 ;  
G00 G90 G98 G54 X0. Y0. ;  
S7500 M03 ;  
G43 H01 Z0.1 ;  
G47 P1 (####) X2. Y2. I0. J0.5 R0.05 Z-0.005 F15. E10. ;  
G00 G80 Z0.1 ;  
M05 ;  
G28 G91 Z0 ;  
M30 ;  
%
```

Engraving Around the Outside of a Rotary Part (G47, G107)

With the Haas Control, it is possible to combine a G47 Engraving cycle with a G107 Cylindrical Mapping cycle to engrave text (or a serial number) along the Outside Diameter of a rotary part.

This code engraves a four digit serial number along the outer diameter of a rotary part.

```
%  
O60472 (G47 SERIAL NUMBER ENGRAVING) ;  
(G54 X0 Y0 is at the bottom left of the part) ;  
(Z0 is on top of the part) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X2. Y2. (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G47 P1 (####) X2. Y2. J0.5 R0.05 Z-0.005 F15. E10. ;  
(Engraves serial number) ;  
(BEGIN COMPLETION BLOCKS) ;
```

```
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%
```

For more details on this cycle see the G107 section.

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)



NOTE:

You must purchase the Rotation and Scaling option to use this G-code.

*X - Center of scaling for the X Axis

*Y - Center of scaling for the Y Axis

*Z - Center of scaling for the Z Axis

*P - Scaling factor for all axes; three-place decimal from 0.001 to 999.999

*indicates optional

```
G51 [X...] [Y...] [Z...] [P...] ;
```

The control always uses a scaling center to determine the scaled position. If you do not specify a scaling center in the G51 command block, then the control uses the last commanded position as the scaling center.

With a scaling (G51) command, the control multiplies by a scaling factor (P) all X, Y, Z, A, B, and C end points for rapids, linear feeds, and circular feeds. G51 also scales I, J, K, and R for G02 and G03. The control offsets all of these positions relative to a scaling center.

There are (3) ways to specify the scaling factor:

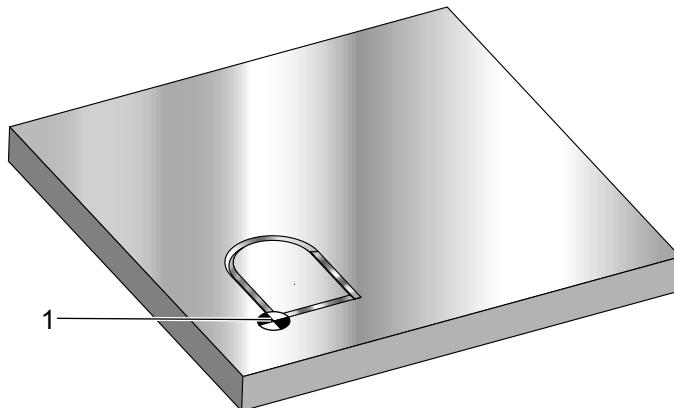
- A P address code in the G51 block applies the specified scaling factor to all axes.

- Setting 71 applies its value as a scaling factor to all axes if it has a nonzero value and you do not use a P address code.
- Settings 188, 189, and 190 apply their values as scaling factors to the X, Y, and Z axes independently if you do not specify a P value and Setting 71 has a value of zero. These settings must have equal values to use them with G02 or G03 commands.

G51 affects all appropriate positioning values in the blocks after the G51 command.

These example programs show how different scaling centers affect the scaling command.

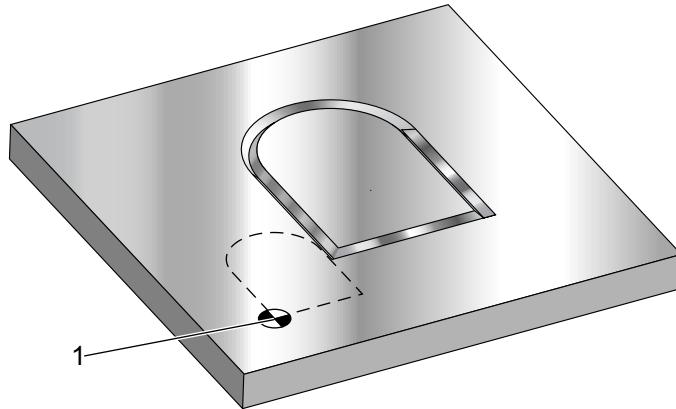
F7.13: G51 No Scaling Gothic Window: [1] Work coordinate origin.



```
%  
O60511 (G51 SCALING SUBPROGRAM) ;  
(G54 X0 Y0 is at the bottom left of window) ;  
(Z0 is on top of the part) ;  
(Run with a main program) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 X2. ;  
Y2. ;  
G03 X1. R0.5 ;  
G01 Y1. ;  
M99 ;  
%
```

The first example illustrates how the control uses the current work coordinate location as a scaling center. Here, it is $X0 Y0 Z0$.

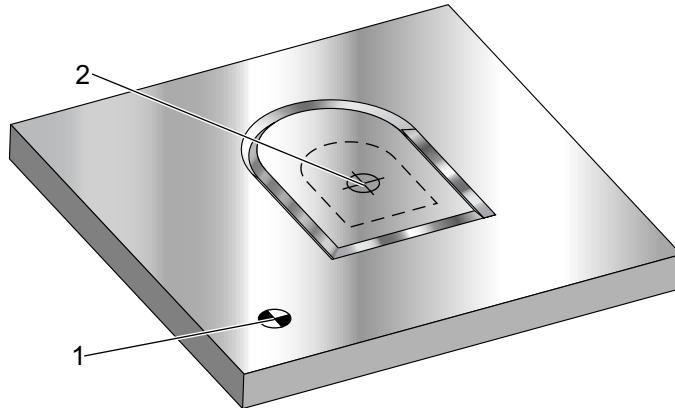
F7.14: G51 Scaling Current Work Coordinates: The Origin [1] is the work origin and the center of scaling.



```
%  
o60512 (G51 SCALING FROM ORIGIN) ;  
(G54 X0 Y0 is at the bottom left of part) ;  
(Z0 is on top of the part) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G54 (Safe startup) ;  
G00 G54 X0 Y0 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 M08 (Activate tool offset 1) ;  
(Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 Z-0.1 F25. (Feed to cutting depth) ;  
M98 P60511 (Cuts shape without scaling) ;  
G00 Z0.1 (Rapid Retract) ;  
G00 X2. Y2. (Rapid to new scale position) ;  
G01 Z-.1 F25. (Feed to cutting depth) ;  
G51 X0 Y0 P2. (2x scale from origin) ;  
M98 P60511 (run subprogram) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09(Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

The next example specifies the center of the window as the scaling center.

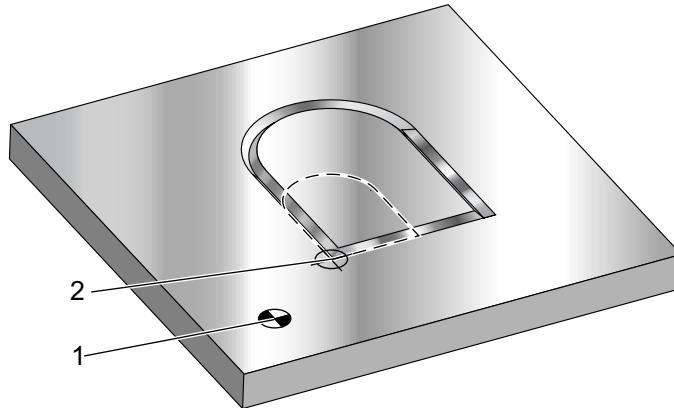
F7.15: G51 Scaling Center of Window: [1] Work coordinate origin, [2] Center of scaling.



```
%  
o60513 (G51 SCALING FROM CENTER OF WINDOW) ;  
(G54 X0 Y0 is at the bottom left of part) ;  
(Z0 is on top of the part) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X0 Y0 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 M08 (Activate tool offset 1) ;  
(Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 Z-0.1 F25. (Feed to cutting depth) ;  
M98 P60511 (Cuts shape without scaling) ;  
G00 Z0.1 (Rapid Retract) ;  
G00 X0.5 Y0.5 (Rapid to new scale position) ;  
G01 Z-.1 F25. (Feed to cutting depth) ;  
G51 X1.5 Y1.5 P2. (2x scale from center of window) ;  
M98 P60511 (run subprogram) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09(Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

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.

F7.16: G51 Scaling Edge of Tool Path: [1] Work coordinate origin, [2] Center of scaling.



```
%  
O60514 (G51 SCALING FROM EDGE OF TOOLPATH) ;  
(G54 X0 Y0 is at the bottom left of part) ;  
(Z0 is on top of the part) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X0 Y0 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 M08 (Activate tool offset 1) ;  
(Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 Z-0.1 F25. (Feed to cutting depth) ;  
M98 P60511 (Cuts shape without scaling) ;  
G00 Z0.1 (Rapid Retract) ;  
G00 X1. Y1. (Rapid to new scale position) ;  
G01 Z-.1 F25. (Feed to cutting depth) ;  
G51 X1. Y1. P2. (2x scale from edge of toolpath) ;  
M98 P60511 (run subprogram) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09(Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

Tool offsets and cutter compensation values are not affected by scaling.

For canned cycles, G51 scales the initial point, depth, and return plane relative to the center of scaling.

To retain the functionality of canned cycles, G51 does not scale these:

- In G73 and G83:

- Peck depth (Q)
- Depth of first peck (I)
- Amount to reduce peck depth per pass (J)
- Minimum peck depth (K)
- In G76 and G77:
 - The shift value (Q)

The control rounds the final results of scaling to the lowest fractional value of the variable being scaled.

G52 Set Work Coordinate System (Group 00 or 12)

G52 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 in which it is commanded.

G54-59 Select Work Coordinate System #1 - #6 (Group 12)

These codes select one of more than six user coordinate systems. All future references to axes positions will be interpreted using the new (G54 G59) coordinate system. See also [317](#) for additional work offsets.

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 refer to 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)

G64 code cancels exact stop (G61).

G65 Macro Subprogram Call Option (Group 00)

G65 is described in the Macros programming section.

G68 Rotation (Group 16)



NOTE:

You must purchase the Rotation and Scaling option to use this G-code. A 200-hour option tryout is also available.

***G17, G18, G19** - Plane of rotation, default is current

***X/Y, X/Z, Y/Z** - Center of rotation coordinates on the selected plane**

***R** - Angle of rotation, in degrees. Three-place decimal, -360.000 to 360.000.

*indicates optional

**The axis designation you use for these address codes corresponds to the axes of the current plane. For example, in the G17 (XY plane), you would use X and Y to specify the center of rotation.

When you command a G68, the control rotates all X, Y, Z, I, J, and K values about a center of rotation to a specified angle (R)..

You can designate a plane with G17, G18, or G19 before G68 to establish the axis plane to rotate. For example:

```
G17 G68 Xnnn Ynnn Rnnn ;
```

Introduction

If you do not designate a plane in the G68 block, the control uses the currently active plane.

The control always uses a center of rotation to determine the positional values after rotation. If you do not specify a center of rotation, the control uses the current location.

G68 affects all appropriate positional values in the blocks after the G68 command. Values in the line that contains the G68 command are not rotated. Only the values in the plane of rotation are rotated; therefore, if G17 is the current plane of rotation, the command affects only the X and Y values.

A positive number (angle) in the R address rotates the feature counterclockwise.

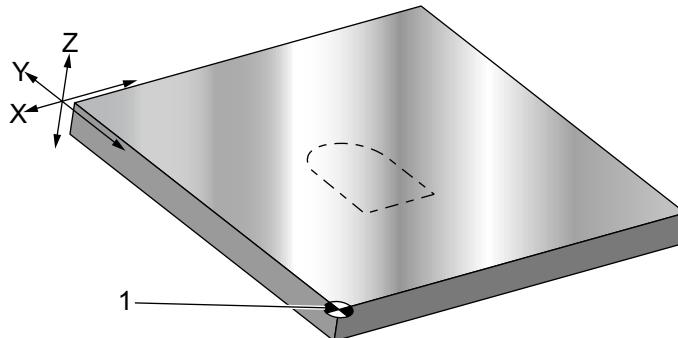
If you do not specify the angle of rotation (R), then the control uses the value in Setting 72.

In G91 mode (incremental) with Setting 73 ON, the rotation angle changes by the value in R. In other words, each G68 command changes the rotation angle by the value specified in R.

The rotational angle is set to zero at the beginning of the program, or you can set it to a specific angle with G68 in G90 mode.

These examples illustrate rotation with G68. The first program defines a Gothic window shape to cut. The rest of the programs use this program as a subprogram.

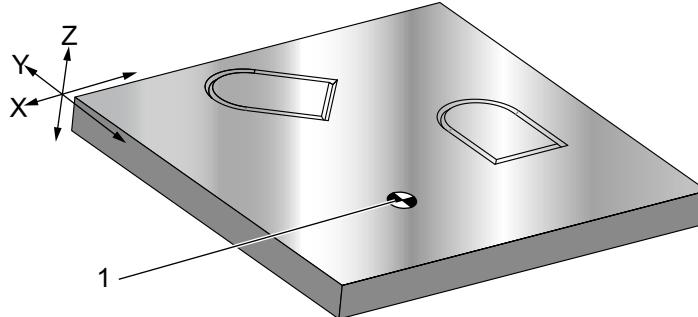
F7.17: G68 Start Gothic Window, No rotation: [1] Work coordinate origin.



```
%  
O60681 (GOTHIC WINDOW SUBPROGRAM) ;  
F20 S500 (SET FEED AND SPINDLE SPEED) ;  
G00 X1. Y1. (RAPID TO LOWER-LEFT WINDOW CORNER) ;  
G01 X2. (BOTTOM OF WINDOW) ;  
Y2. (RIGHT SIDE OF WINDOW) ;  
G03 X1. R0.5 (TOP OF WINDOW) ;  
G01 Y1. (FINISH WINDOW) ;  
M99;  
&
```

The first example illustrates how the control uses the current work coordinate location as a rotation center (X0 Y0 Z0).

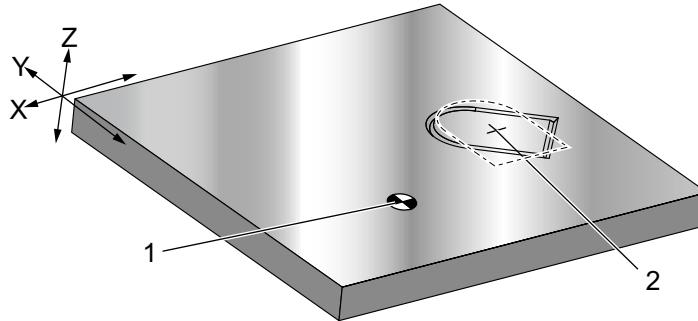
F7.18: G68 Rotation Current Work Coordinate: [1] Work coordinate origin and center of rotation.



```
O60682 (ROTATE ABOUT WORK COORDINATE) ;
G59 (OFFSET) ;
G00 G90 X0 Y0 Z-0.1 (WORK COORDINATE ORIGIN) ;
M98 P60681 (CALL SUBPROGRAM) ;
G90 G00 X0 Y0 (LAST COMMANDED POSITION) ;
G68 R60. (ROTATE 60 DEGREES) ;
M98 P60681 (CALL SUBPROGRAM) ;
G69 G90 X0 Y0 (CANCEL G68) ;
M30
%
```

The next example specifies the center of the window as the rotation center.

F7.19: G68 Rotation Center of Window: [1] Work coordinate origin, [2] Center of rotation.

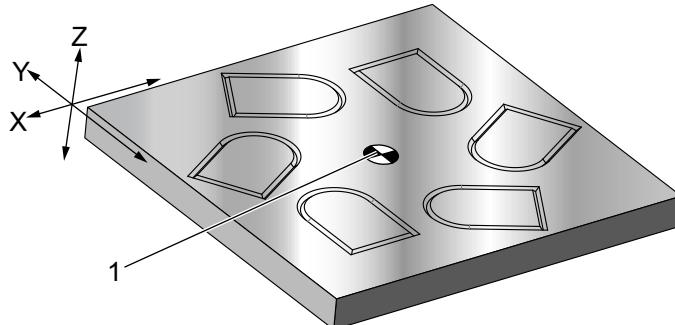


```
% O60683 (ROTATE ABOUT CENTER OF WINDOW) ;
G59 (OFFSET) ;
G00 G90 X0 Y0 Z-0.1 (WORK COORDINATE ORIGIN) ;
G68 X1.5 Y1.5 R60. ;
(ROTATE SHAPE 60 DEGREES ABOUT CENTER) ;
M98 P60681 (CALL SUBPROGRAM) ;
G69 G90 G00 X0 Y0 ;
(CANCEL G68, LAST COMMANDED POSITION) ;
M30 ;
%
```

Introduction

This next 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.

F7.20: G68 Rotate Patterns About Center: [1] Work coordinate origin and center of rotation.



```
%  
O60684 (ROTATE PATTERN ABOUT CENTER) ;  
G59 (OFFSET) ;  
G00 G90 X0 Y0 Z-0.1 (WORK COORDINATE ORIGIN) ;  
M98 P1000 L6 (CALL SUBPROGRAM, LOOP 6 TIMES) ;  
M30 (END AFTER SUBPROGRAM LOOP) ;  
N1000 (BEGIN LOCAL SUBPROGRAM) ;  
G91 G68 R60. (ROTATE 60 DEGREES) ;  
G90 M98 P60681 (CALL WINDOW SUBPROGRAM) ;  
G90 G00 X0 Y0 (LAST COMMANDED POSITION) ;  
M99;  
%
```

Do not change the plane of rotation while G68 is in effect.

Rotation with Scaling:

If you use scaling and rotation at the same time, you should turn on scaling before rotation, and use separate blocks. Use this template:

```
%  
G51 ... (SCALING) ;  
... ;  
G68 ... (ROTATION) ;  
... program ;  
G69 ... (ROTATION OFF) ;  
... ;  
G50 ... (SCALING OFF) ;  
%
```

Rotation with Cutter Compensation:

Turn on cutter compensation after the rotation command. Turn off cutter compensation before you turn off rotation.

G69 Cancel G68 Rotation (Group 16)

(This G-code is optional and requires Rotation and Scaling.)

G69 cancels rotation mode.

G70 Bolt Hole Circle (Group 00)

I - Radius

*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

*indicates optional

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. See also G-code Canned Cycles section.

```
%  
O60701 (G70 BOLT HOLE CIRCLE) ;  
(G54 X0 Y0 is center of the circle) ;  
(Z0 is on the top of the part) ;  
(T1 is a drill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X0 Y0 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G81 G98 Z-1. R0.1 F15. L0 (Begin G81) ;  
(L0 skip drilling X0 Y0 position) ;  
G70 I5. J15. L12 (Begin G70) ;  
(Drills 12 holes on a 10.0 in. diameter circle) ;  
G80 (Canned Cycles off) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home and Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

G71 Bolt Hole Arc (Group 00)

I - Radius

*J - Starting angle (degrees CCW from horizontal)

K - Angular spacing of holes (+ or -)

L - Number of holes

*indicates optional

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

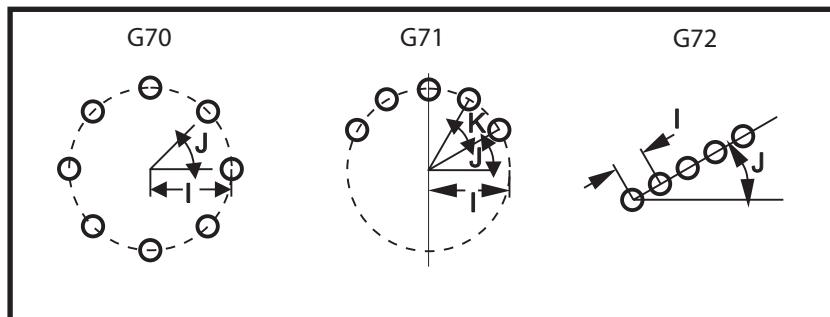
*J - Angle of line (degrees CCW from horizontal)

L - Number of holes

*indicates optional

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.

F7.21: G70, G71, and G72 Bolt Holes: [I] Radius of bolt circle (G70, G71), or distance between holes (G72), [J] Starting angle from the 3 o'clock position, [K] Angular spacing between holes, [L] Number of holes.



Rules For Bolt Pattern Canned Cycles

1. Place the tool at the center of the bolt pattern (for G70 or G71), or at the starting hole location (for G72), before the canned cycle execution.
2. The J code is the angular starting position and is always 0 to 360 degrees counterclockwise from the three o'clock position.
3. For G70 and G71 cycles, put an L0 on the initial canned cycle line to skip drilling at the center of the hole pattern. You can also turn off Setting 28 to prevent a hole from

being drilled at the initial X/Y position. Refer to page **363** for more information on Setting 28.



NOTE: *L0* is the preferred method.

G73 High-Speed Peck Drilling Canned Cycle (Group 09)

F - Feedrate

***I** - First peck depth

***J** - Amount to reduce pecking depth for pass

***K** - Minimum peck depth (The control calculates the number of pecks)

***L** - Number of loops (Number of holes to drill) if G91 (Incremental Mode) is used

***P** - Pause at the bottom of the hole (in seconds)

***Q** - Peck Depth (always incremental)

***R** - Position of the R plane (Distance above part surface)

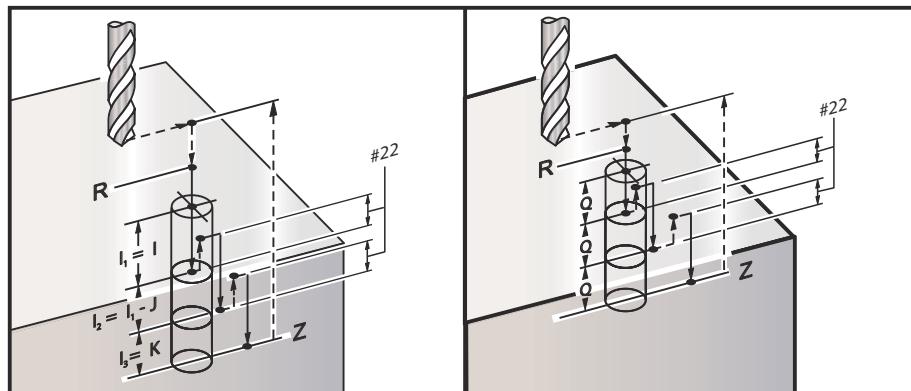
***X** - X-Axis location of hole

***Y** - Y-Axis location of hole

***Z** - Position of the Z-Axis at the bottom of hole

* indicates optional

F7.22: G73 Peck Drilling. Left: Using I, J, and K Addresses. Right: Using Only the Q Address. [#22] Setting 22.



I, J, K, and Q are always positive numbers.

There are three methods to program a G73: using the I, J, K addresses, using the K and Q addresses, and using only a Q address.

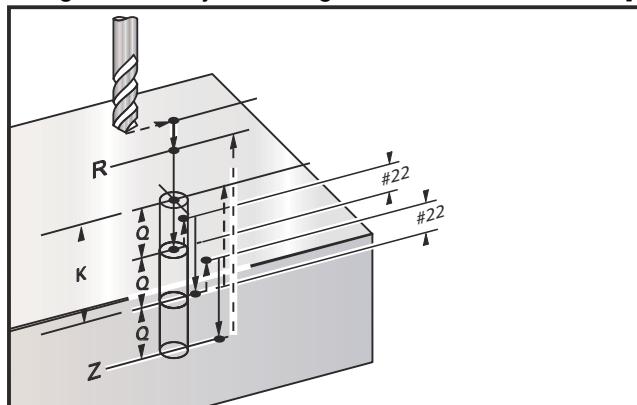
If I, J, and K are specified, The first pass will cut in by the value I, each succeeding cut will be reduced by the value of J, and the minimum cutting depth is K. If P is specified, the tool will pause at the bottom of the hole for that amount of time.

Introduction

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.

If only **Q** is specified, a different operating mode is selected for this canned cycle. In this mode, the tool is returned to the **R** plane after all pecks are completed, and all pecks will be equal to the **Q** value.

F7.23: G73 Peck Drilling Canned Cycles using the **K** and **Q** Addresses: [#22] Setting 22.



G74 Reverse Tap Canned Cycle (Group 09)

F - Feedrate. Use the formula described in the canned cycle introduction to calculate feedrate and spindle speed.

* **J** - Retract Multiple (How fast to retract - see Setting 130)

* **L** - Number of loops (How many holes to tap) if G91 (Incremental Mode) is used

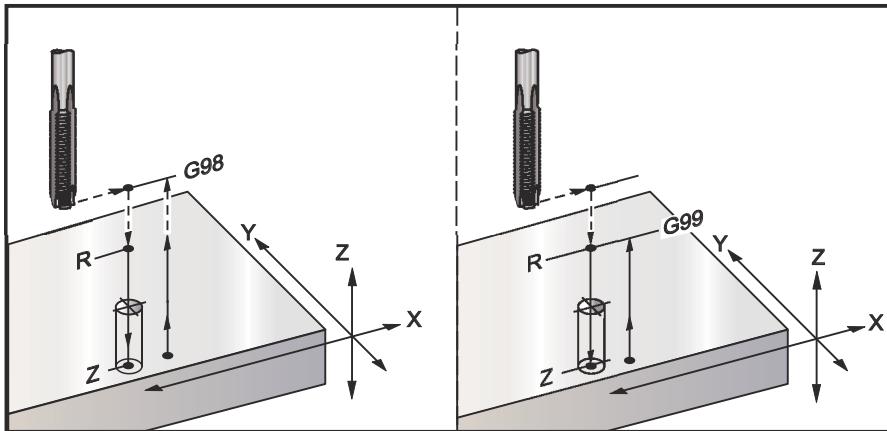
* **R** - Position of the R plane (position above the part) where tapping starts

* **X** - X-Axis location of hole

* **Y** - Y-Axis location of hole

Z - Position of the Z-Axis at the bottom of hole

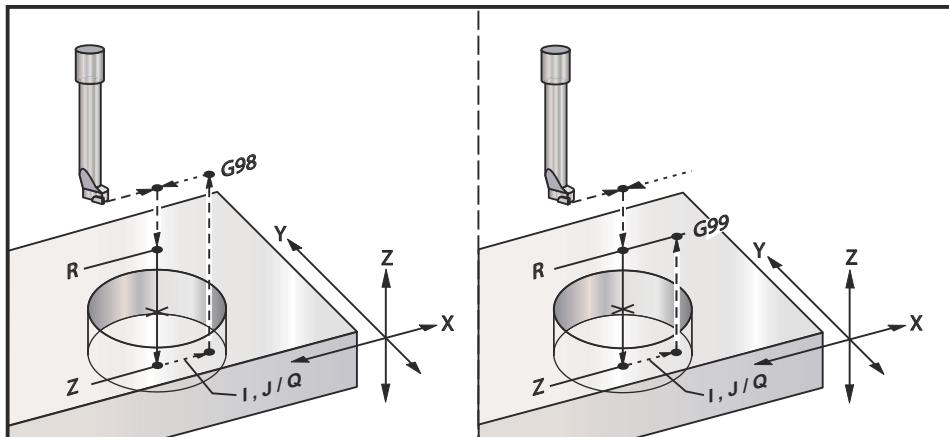
* indicates optional

F7.24: G74 Tapping Canned Cycle**G76 Fine Boring Canned Cycle (Group 09)****F** - Feedrate***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

* indicates optional

**CAUTION:**

Unless you specify otherwise, this canned cycle uses the most recently commanded spindle direction (M03, M04, or M05). If the program did not specify a spindle direction before it commands this canned cycle, the default is M03 (clockwise). If you command M05, the canned cycle will run as a "no-spin" cycle. This lets you run applications with self-driven tools, but it can also cause a crash. Be sure of the spindle direction command when you use this canned cycle.

F7.25: G76 Fine Boring Canned Cycles

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

***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

***X** - X-Axis location of hole

***Y** - Y-Axis location of hole

***Z** - Z-Axis position to cut to

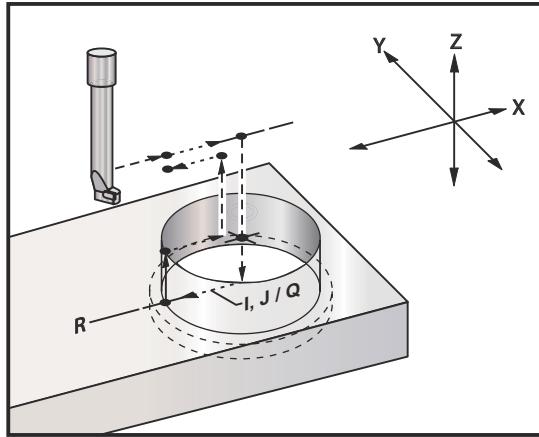
* indicates optional

**CAUTION:**

Unless you specify otherwise, this canned cycle uses the most recently commanded spindle direction (M03, M04, or M05). If the program did not specify a spindle direction before it commands this canned cycle, the default is M03 (clockwise). If you command M05, the canned cycle will run as a "no-spin" cycle. This lets you run applications with self-driven tools, but it can also cause a crash. Be sure of the spindle direction command when you use this canned cycle.

In addition to boring the hole, this cycle shifts the X and Y Axis before and after the cut, to clear the tool while it enters and exits the workpiece (refer to G76 for an example of a shift move). Setting 27 defines the shift direction. If you do not specify a *Q* value, the control uses the optional *I* and *J* values to determine the shift direction and distance.

F7.26: G77 Back Boring Canned Cycle Example

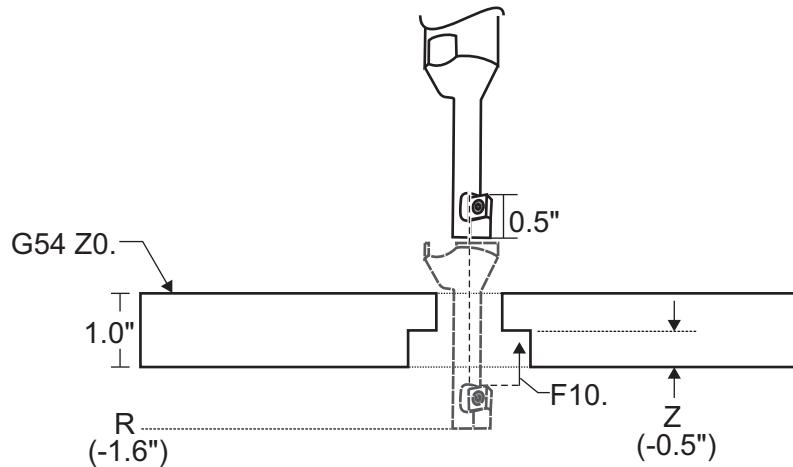


Program Example

```
%  
O60077 (G77 CYCLE-WORKPIECE IS 1.0" THICK) ;  
T5 M06 (BACK COUNTERBORE TOOL) ;  
G90 G54 G00 X0 Y0 (INITIAL POSITION) ;  
S1200 M03 (SPINDLE START) ;  
G43 H05 Z.1 (TOOL LENGTH COMPENSATION) ;  
G77 Z-1. R-1.6 Q0.1 F10. (1ST HOLE) ;  
X-2. (2ND HOLE) ;  
G80 G00 Z.1 M09 (CANCEL CANNED CYCLE) ;  
G28 G91 Z0. M05 ;  
M30 ;  
%
```

Introduction

F7.27: G77 Approximate Toolpath Example. This example shows the entrance motion only. Dimensions are not to scale.



NOTE:

For this example, the “top” of the workpiece is the surface defined as Z0. in the current work offset. The “bottom” of the workpiece is the opposite surface.

In this example, when the tool reaches the R depth, it then moves 0.1" in X (the Q value and Setting 27 define this movement; in this example, Setting 27 is $x+$). The tool then feeds to the Z value at the given feedrate. When the cut is finished, the tool shifts back toward the center of the hole and retracts out of it. The cycle repeats at the next commanded position until the G80 command.



NOTE:

The R value is negative, and it must go past the bottom of the part for clearance.



NOTE:

The Z value is commanded from the active Z work offset.



NOTE:

You do not need to command an initial point return (G98) after a G77 cycle; the control assumes this automatically.

G80 Canned Cycle Cancel (Group 09)

G80 cancels all active canned cycles.



NOTE:

G00 or G01 also cancel canned cycles.

G81 Drill Canned Cycle (Group 09)

F - Feedrate

***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

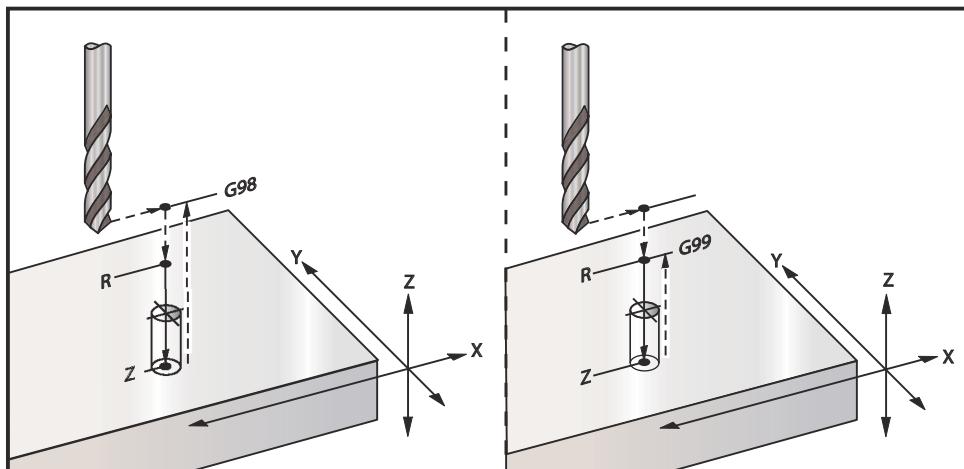
* indicates optional



CAUTION:

Unless you specify otherwise, this canned cycle uses the most recently commanded spindle direction (M03, M04, or M05). If the program did not specify a spindle direction before it commands this canned cycle, the default is M03 (clockwise). If you command M05, the canned cycle will run as a "no-spin" cycle. This lets you run applications with self-driven tools, but it can also cause a crash. Be sure of the spindle direction command when you use this canned cycle.

F7.28: G81 Drill Canned Cycle



This is a program to drill through an aluminum plate:

```
%  
O60811 (G81 DRILLING CANNED CYCLE) ;  
(G54 X0 Y0 is at the top-left of part) ;  
(Z0 is on top of the part) ;  
(T1 is a .5 in drill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X2. Y-2. (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G81 Z-0.720 R0.1 F15. (Begin G81) ;  
(Drill 1st hole at current X Y location) ;  
X2. Y-4. (2nd hole) ;  
X4. Y-4. (3rd hole) ;  
X4. Y-2. (4th hole) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 G90 Z1. M09 (Rapid retract, coolant off) ;  
G53 G49 Z0 M05 (Z home, spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

G82 Spot Drill Canned Cycle (Group 09)

F - Feedrate

***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

* indicates optional



CAUTION:

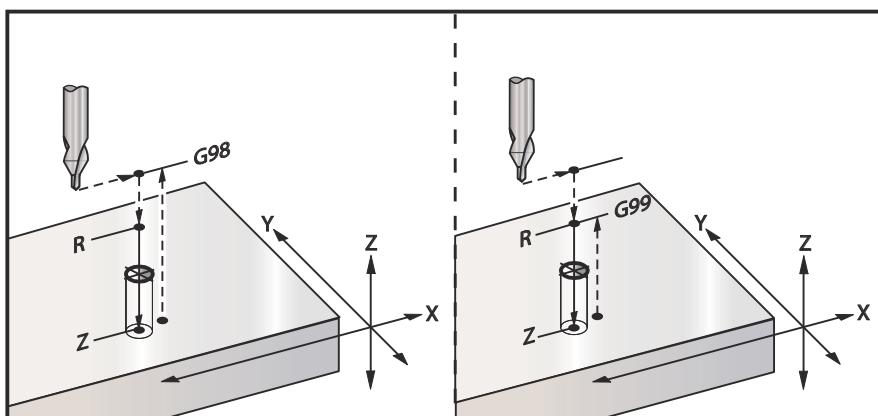
Unless you specify otherwise, this canned cycle uses the most recently commanded spindle direction (M03, M04, or M05). If the program did not specify a spindle direction before it commands this canned cycle, the default is M03 (clockwise). If you command M05, the canned cycle will run as a “no-spin” cycle. This lets you run applications with self-driven tools, but it can also cause a crash. Be sure of the spindle direction command when you use this canned cycle.

**NOTE:**

G82 is similar to G81 except that there is the option to program a dwell (P).

```
%  
O60821 (G82 SPOT DRILLING CANNED CYCLE) ;  
(G54 X0 Y0 is at the top-left of part) ;  
(Z0 is on top of the part) ;  
(T1 is a 0.5 in 90 degree spot drill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X2. Y-2. (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G82 Z-0.720 P0.3 R0.1 F15. (Begin G82) ;  
(Drill 1st hole at current X Y location) ;  
X2. Y-4. (2nd hole) ;  
X4. Y-4. (3rd hole) ;  
X4. Y-2. (4th hole) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z1. M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

F7.29: G82 Spot Drilling Example



G83 Normal Peck Drilling Canned Cycle (Group 09)

F - Feedrate

***I** - Size of first peck depth

***J** - Amount to reduce peck depth each pass

***K** - Minimum depth of peck

***L** - Number of holes if G91 (Incremental Mode) is used, also G81 through G89.

***P** - Pause at end of last peck, in seconds (Dwell)

***Q** - Peck 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

* indicates optional

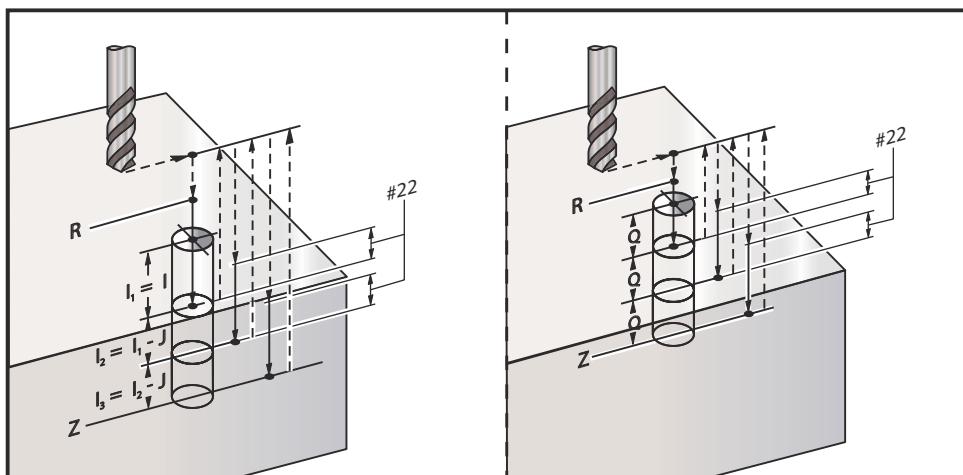
If I, J, and K are specified, the first pass will cut in by the amount of I, each succeeding cut will be reduced by amount J, and the minimum cutting depth is K. Do not use a Q value when programming with I, J, and K.

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 ;
```

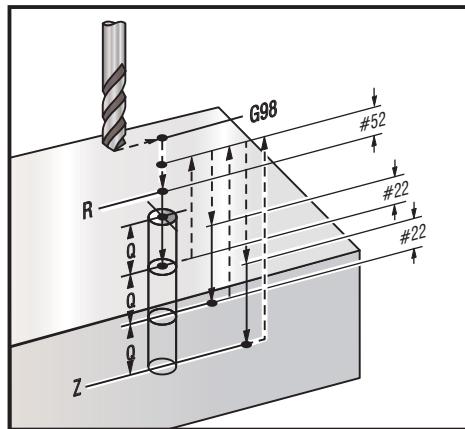
The same dwell time will apply to all subsequent blocks that do not specify a dwell time.

F7.30: G83 Peck Drilling with I, J, K and Normal Peck Drilling: [#22] Setting 22.



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, you can set the R plane much closer to the part. When the chip-clearing move to R occurs, Setting 52 determines the Z-Axis distance above R.

F7.31: G83 peck Drilling Canned Cycle with Setting 52 [#52]



```
%  
O60831 (G83 PECK DRILLING CANNED CYCLE) ;  
(G54 X0 Y0 is at the top-left of part) ;  
(Z0 is on top of the part) ;  
(T1 is a 0.3125 in. stub drill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X2. Y-2. (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G83 Z-0.720 Q0.175 R0.1 F15. (Begin G83) ;  
(Drill 1st hole at current X Y location) ;  
X2. Y-4. (2nd hole) ;  
X4. Y-4. (3rd hole) ;  
X4. Y-2. (4th hole) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z1. M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

G84 Tapping Canned Cycle (Group 09)

F - Feedrate

* **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

* **S** - Spindle speed

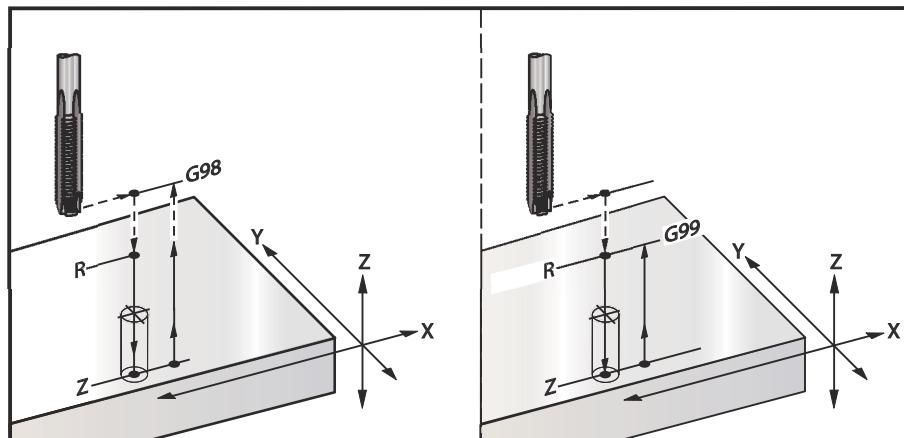
* indicates optional



NOTE:

You do not need to command a spindle start (M03 / M04) before G84. The canned cycle starts and stops the spindle as needed.

F7.32: G84 Tapping Canned Cycle



%
O60841 (G84 TAPPING CANNED CYCLE) ;
(G54 X0 Y0 is at the top-left of part) ;
(Z0 is on top of the part) ;
(T1 is a 3/8-16 tap) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54 (Safe startup) ;
G00 G54 X2. Y-2. (Rapid to 1st position) ;
G43 H01 Z0.1 (Activate tool offset 1) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G84 Z-0.600 R0.1 F56.25 S900 (Begin G84) ;

```
(900 rpm divided by 16 tpi = 56.25 ipm) ;
(Drill 1st hole at current X Y location) ;
X2. Y-4. (2nd hole) ;
X4. Y-4. (3rd hole) ;
X4. Y-2. (4th hole) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z1. M09 (Canned cycle off, rapid retract) ;
(Coolant off) ;
G53 G49 Z0 (Z home) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%
```

G85 Bore In, Bore Out Canned Cycle (Group 09)

F - Feedrate

***L** - Number of holes if G91 (Incremental Mode) is used

***R** - Position of the R plane (position above the part)

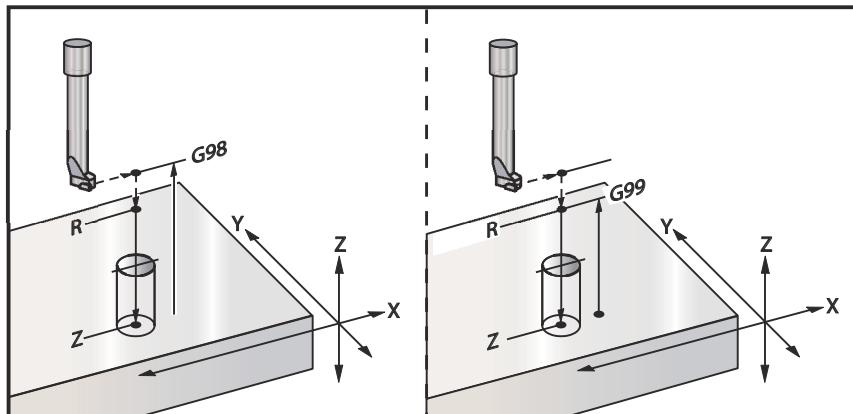
***X** - X-Axis location of holes

***Y** - Y-Axis location of holes

***Z** - Position of the Z Axis at the bottom of hole

* indicates optional

F7.33: G85 Boring Canned Cycle



G86 Bore and Stop Canned Cycle (Group 09)

F - Feedrate

*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

* indicates optional

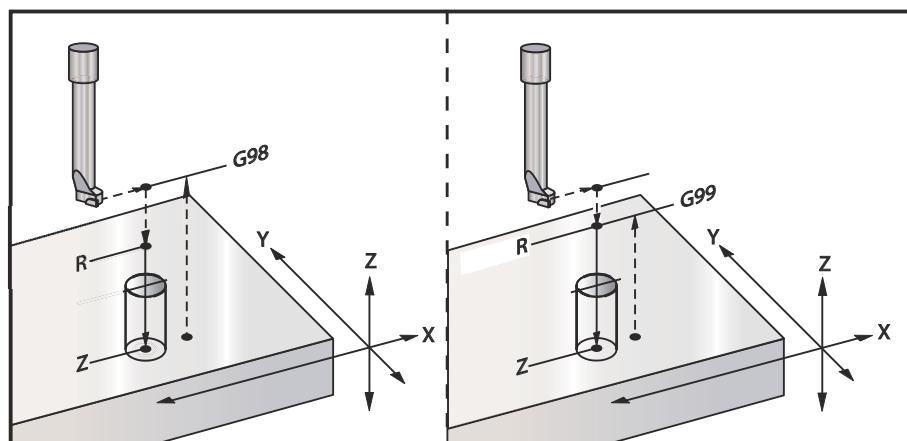


CAUTION:

Unless you specify otherwise, this canned cycle uses the most recently commanded spindle direction (M03, M04, or M05). If the program did not specify a spindle direction before it commands this canned cycle, the default is M03 (clockwise). If you command M05, the canned cycle will run as a "no-spin" cycle. This lets you run applications with self-driven tools, but it can also cause a crash. Be sure of the spindle direction command when you use this canned cycle.

This G code will stop the spindle once the tool reaches the bottom of the hole. The tool is retracted once the spindle has stopped.

F7.34: G86 Bore and Stop Canned Cycles



G87 Bore In and Manual Retract Canned Cycle (Group 09)

F - Feedrate

***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

* indicates optional

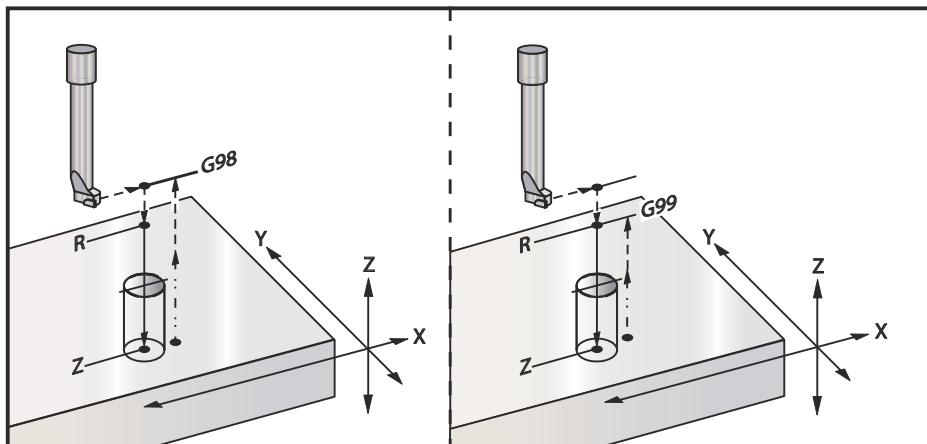


CAUTION:

Unless you specify otherwise, this canned cycle uses the most recently commanded spindle direction (M03, M04, or M05). If the program did not specify a spindle direction before it commands this canned cycle, the default is M03 (clockwise). If you command M05, the canned cycle will run as a “no-spin” cycle. This lets you run applications with self-driven tools, but it can also cause a crash. Be sure of the spindle direction command when you use this canned cycle.

This G code will stop the spindle at the bottom of the hole. You then manually jog the tool out. The program continues after you press [CYCLE START].

F7.35: G87 Bore and Stop and Manual Retract



G88 Bore In, Dwell, Manual Retract Canned Cycle (Group 09)

F - Feedrate

*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

* indicates optional

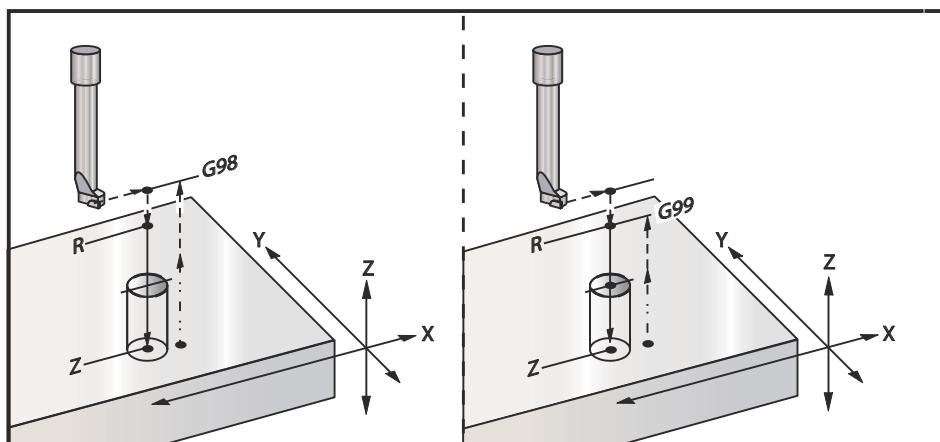


CAUTION:

Unless you specify otherwise, this canned cycle uses the most recently commanded spindle direction (M03, M04, or M05). If the program did not specify a spindle direction before it commands this canned cycle, the default is M03 (clockwise). If you command M05, the canned cycle will run as a “no-spin” cycle. This lets you run applications with self-driven tools, but it can also cause a crash. Be sure of the spindle direction command when you use this canned cycle.

This G code stops the tool at the bottom of the hole, and dwells with the tool turning for the time designated with the P value. At this point the tool is manually jogged out of the hole. The program will continue when [CYCLE START] is pressed.

F7.36: G88 Bore and Dwell and Manual Retract



G89 Bore In, Dwell, Bore Out Canned Cycle (Group 09)

F - Feedrate

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

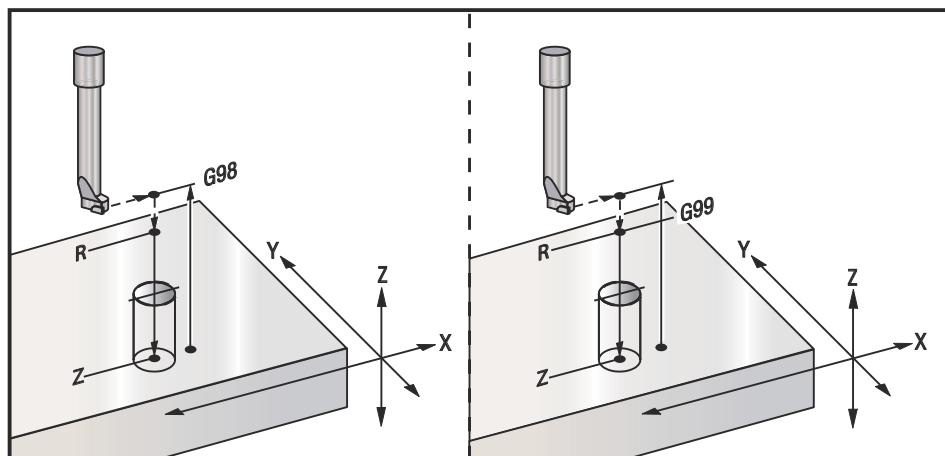
* indicates optional



CAUTION:

Unless you specify otherwise, this canned cycle uses the most recently commanded spindle direction (M03, M04, or M05). If the program did not specify a spindle direction before it commands this canned cycle, the default is M03 (clockwise). If you command M05, the canned cycle will run as a "no-spin" cycle. This lets you run applications with self-driven tools, but it can also cause a crash. Be sure of the spindle direction command when you use this canned cycle.

F7.37: G89 Bore and Dwell and Canned Cycle



G90 Absolute - 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).

The Basic Programming section of this manual, beginning on page 140, includes a discussion of absolute versus incremental programming.

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-G59, G110-G129) 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 RETURN]** and **[ALL]** or **[ZERO RETURN]** and **[SINGLE]** are used.

G92 Clear Shift Value From Within a Program

G92 shifts may be canceled by programming another G92 shift to change the current work offset back to the original value.

```
%  
O60921 (G92 SHIFT WORK OFFSETS) ;  
(G54 X0 Y0 Z0 is at the center of mill travel) ;  
G00 G90 G54 X0 Y0 (Rapid to G54 origin) ;  
G92 X2. Y2. (Shifts current G54) ;  
G00 G90 G54 X0 Y0 (Rapid to G54 origin) ;  
G92 X-2. Y-2. (Shifts current G54 back to original) ;  
G00 G90 G54 X0 Y0 (Rapid to G54 origin) ;  
M30 (End program) ;  
%
```

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.



NOTE:

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 G93.

G94 Feed Per Minute Mode (Group 05)

This code deactivates G93 (Inverse Time Feed Mode) and returns the control to Feed Per Minute mode.

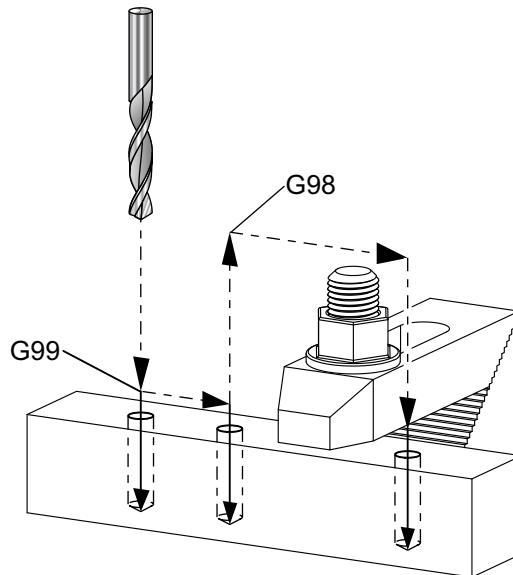
G95 Feed per Revolution (Group 05)

When G95 is active, a spindle revolution will result in a travel distance specified by the Feed value. If Setting 9 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) between each X/Y position. This lets you program up and around areas of the part, clamps, and fixtures.

- F7.38:** G98 Initial Point Return. After the second hole, the Z Axis returns to the starting position [G98] to move over the toe clamp to the next hole position.



```

%
O69899 (G98/G99 INITIAL POINT & R PLANE RETURN) ;
(G54 X0 Y0 is top right corner of part) ;
(Z0 is on top of the part) ;
(T1 is a drill) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G17 G40 G49 G54 (Safe startup) ;
G00 G54 X1. Y-0.5 (Rapid to 1st position) ;
S1000 M03 (Spindle on CW) ;
G43 H01 Z2. (Tool offset 1 on) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G81 G99 X1. Z-0.5 F10. R0.1 (Begin G81 using G99) ;
G98 X2. (2nd hole and then clear clamp with G98) ;
X4. (Drill 3rd hole) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z2. M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;

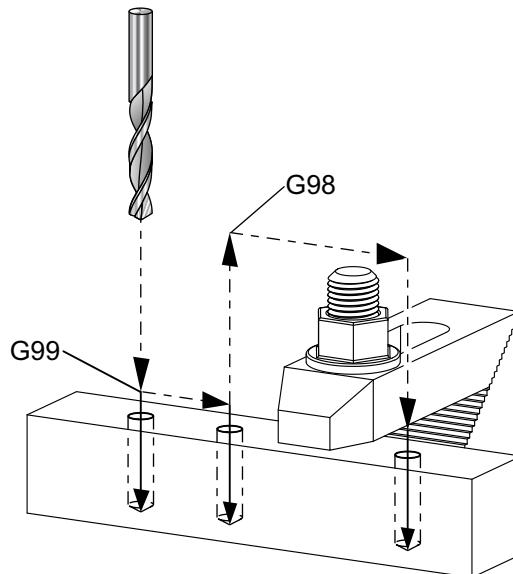
```

%

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.

- F7.39:** G99R Plane Return. After the first hole, the Z Axis returns to the R plane position [G99] and moves to the second hole position. This is a safe move in this case because there are no obstacles.



%

```

O69899 (G98/G99 INITIAL POINT & R PLANE RETURN) ;
(G54 X0 Y0 is top right corner of part) ;
(Z0 is on top of the part) ;
(T1 is a drill) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G17 G40 G49 G54 (Safe startup) ;
G00 G54 X1. Y-0.5 (Rapid to 1st position) ;
S1000 M03 (Spindle on CW) ;
G43 H01 Z2. (Tool offset 1 on) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G81 G99 X1. Z-0.5 F10. R0.1 (Begin G81 using G99) ;
G98 X2. (2nd hole and then clear clamp with G98) ;
X4. (Drill 3rd hole) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z2. M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;

```

```
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

G100/G101 Disable/Enable Mirror Image (Group 00)

- *X - X-Axis command
- *Y - Y-Axis command
- *Z - Z-Axis command
- *A - A-Axis command
- *B - B-Axis command
- *C - C-Axis command

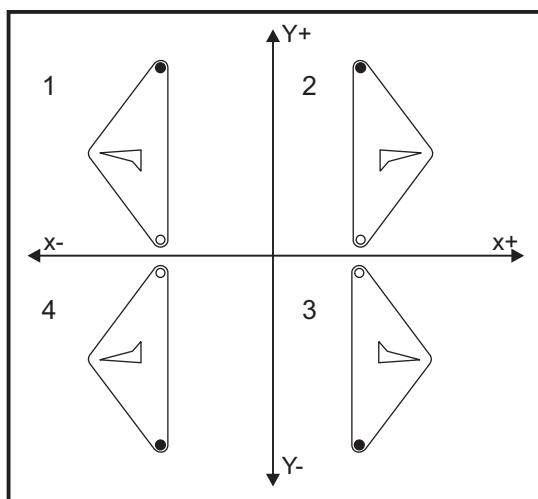
* indicates optional

Programmable mirror imaging is used to turn on or off any of 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 indicates when an axis is mirrored. Also see Settings 45, 46, 47, 48, 80, and 250 for mirror imaging.

The format for turning Mirror Image on and off is:

```
G101 X0. (turns on mirror imaging for the X-Axis) ;  
G100 X0. (turns off mirror imaging for the X-Axis) ;
```

F7.40: X-Y Mirror Image

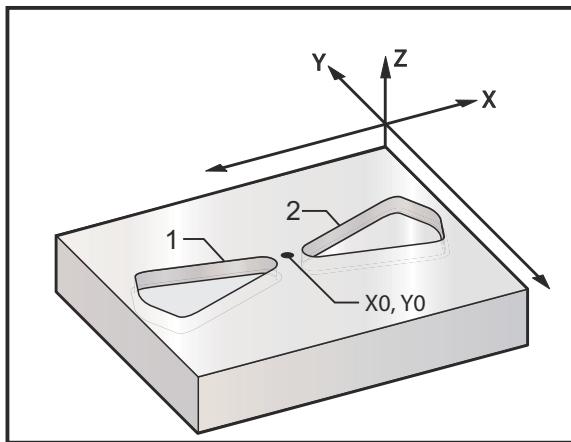


Mirror Image and Cutter Compensation

Turning on Mirror Image for only one of the X or Y axes causes the cutter to move along the opposite side of a cut. The control automatically switches the cutter compensation direction (G41, G42) and reverses the circular motion commands (G02, G03) as needed.

When milling a shape with XY motions, turning on Mirror Image for just one of the X or Y axes changes climb milling (G41) to conventional milling (G42) 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 eliminates this problem.

F7.41: Mirror Image and Pocket Milling



Program Code for Mirror Imaging in the X-Axis:

```
%  
O61011 (G101 MIRROR IMAGE ABOUT X AXIS) ;  
(G54 X0 Y0 is at the center of part) ;  
(Z0 is on top of the part) ;  
(T1 is a 0.250 in. diameter endmill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X-.4653 Y.052 (Rapid to 1st position) ;  
S5000 M03 (Spindle on CW) ;  
G43 H01 Z.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 Z-.25 F5. (Feed to depth of cut) ;  
M98 P61012 F20. (Call contour subprogram) ;  
G00 Z.1 (Rapid retract above part) ;  
G101 X0. (Mirror imaging on for X Axis) ;  
X-.4653 Y.052 (Rapid to 1st position) ;  
G01 Z-.25 F5. (Feed to depth of cut) ;
```

```
M98 P61012 F20. (Call contour subprogram) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G100 X0. (Mirror imaging off for X Axis) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%
%
O61012 (G101 CONTOUR SUBPROGRAM) ;
(Subprogram for pocket in O61011) ;
(Must have a feedrate in M98) ;
G01 X-1.2153 Y.552 (Linear move) ;
G03 X-1.3059 Y.528 R.0625 (CCW arc) ;
G01 X-1.5559 Y.028 (Linear move) ;
G03 X-1.5559 Y.-.028 R.0625 (CCW arc) ;
G01 X-1.3059 Y.-.528 (Linear move) ;
G03 X-1.2153 Y.-.552 R.0625 (CCW arc) ;
G01 X-.4653 Y.-.052 (Linear move) ;
G03 X-.4653 Y.052 R.0625 (CCW arc) ;
M99 (Exit to Main Program) ;
%
```

G102 Programmable Output to RS-232 (Group 00)

***X** - X-Axis command

***Y** - Y-Axis command

***Z** - Z-Axis command

***A** - A-Axis command

* indicates optional

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 Look-Ahead (Group 00)

G103 specifies the maximum number of blocks the control looks ahead (Range 0-15), for example:

```
G103 [P..] ;
```

During machine motions, the control prepares future blocks (lines of code) ahead of time. This is commonly called “Block Look-ahead.” While the control executes the current block, it has already interpreted and prepared the next block for continuous motion.

A program command of G103 P0, or simply G103, disables block limiting. A program command of G103 Pn limits look-ahead to n blocks.

G103 is useful for debugging macro programs. The control interprets Macro expressions during look-ahead time. If you insert a G103 P1 into the program, the control interprets macro expressions (1) block ahead of the currently executing block.

It is best to add several empty lines after a G103 P1 is called. This ensures that no lines of code after the G103 P1 are interpreted until they are reached.

G103 affects cutter compensation and High Speed Machining.

G107 Cylindrical Mapping (Group 00)

- ***X** - X-Axis command
- ***Y** - Y-Axis command
- ***Z** - Z-Axis command
- ***A** - A-Axis command
- ***B** - B-Axis command
- C** - C-Axis command
- ***Q** - Diameter of the cylindrical surface
- ***R** - Radius of the rotary axis

* indicates optional

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.

Introduction

- 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, B, or C; and Q or R.

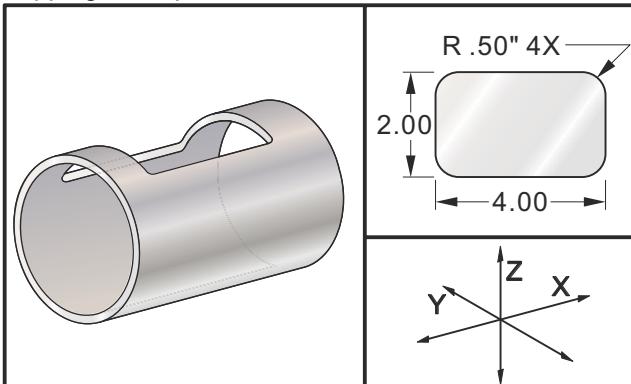
X, Y, or Z: An X, Y, or Z address specifies the linear axis that will be mapped to the specified rotary axis (A or B). When one of these linear axes is specified, a rotary axis must also be specified.

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 [RESET] turns off any cylindrical mapping that is currently in effect, regardless of the status of Setting 56.

F7.42: Cylindrical Mapping Example



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.

```
%  
O61071 (G107 CYLINDRICAL MAPPING) ;  
(G54 X0 Y0 is in center of rectangular slot) ;  
(Z0 is on highest point of cylindrical surface) ;  
(T1 is a .625 in. dia endmill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;
```

```

G00 G90 G40 G49 G54 (Safe startup) ;
G28 G91 A0 (Home A axis) ;
G00 G90 G54 X1.5 Y0 (Rapid to 1st position) ;
S5000 M03 (Spindle on CW) ;
G107 A0 Y0 R2. (Cylindrical mapping on) ;
(Move to A0 Y0, Part has radius of 2 inches) ;
G43 H01 Z0.1 (Activate tool offset 1) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G01 Z-0.25 F25. (Feed to depth of cut) ;
G41 D01 X2. Y0.5 (Cutter comp on) ;
G03 X1.5 Y1. R0.5 (CCW cutting move) ;
G01 X-1.5 (Linear cutting move) ;
G03 X-2. Y0.5 R0.5 (CCW cutting move) ;
G01 Y-0.5 (Linear cutting move) ;
G03 X-1.5 Y-1. R0.5 (CCW cutting move) ;
G01 X1.5 (Linear cutting move) ;
G03 X2. Y-0.5 R0.5 (CCW cutting move) ;
G01 Y0. (Linear cutting move) ;
G40 X1.5 (Cutter comp off) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G91 G28 A0. (Home A axis) ;
G107 (Cylindrical mapping off) ;
G90 G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%

```

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. Use it to set work offsets to the center of a work piece with a work probe.

F - Feedrate

***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

* indicates optional

Automatic Work Offset Center Measurement (G136) is used to command a spindle probe to set work offsets. A G136 will feed the axes of the machine in an effort to probe the work piece with a spindle mounted probe. The axis (axes) will move until a signal (skip signal) from the probe is received or the end of the programmed move is reached. Tool compensation (G41, G42, G43, or G44) must not be active when this function is performed. 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 the measured center of the two probed points.

Notes:

This code is non-modal and only applies to the block of code in which G136 is specified.

The points probed are offset by the values in Settings 59 through 62. See the Settings section of this manual for more information.

Do not use Cutter Compensation (G41, G42) with a G136.

Do not use tool length Compensation (G43, G44) with G136

To avoid damaging the probe, use a feed rate below F100. (inch) or F2500. (metric).

Turn on the spindle probe before using G136.

If your mill has the standard Renishaw probing system, use the following commands to turn on the spindle probe:

M59 P1134 ;

Use the following commands to turn off the spindle probe:

M69 P1134 ;

Also see M75, M78, and M79.

Also see G31.

This sample program measures the center of a part in the Y Axis and records the measured value to the G58 Y Axis work offset. To use this program, the G58 work offset location must be set at or close to the center of the part to be measured.

```
%  
O61361 (G136 AUTO WORK OFFSET - CENTER OF PART) ;  
(G58 X0 Y0 is at the center of part) ;  
(Z0 is on top of the part) ;  
(T1 is a spindle probe) ;  
(BEGIN PREPARATION BLOCKS) ;
```

```

T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54 (Safe startup) ;
G00 G58 X0. Y1. (Rapid to 1st position) ;
(BEGIN PROBING BLOCKS) ;
M59 P1134 (Spindle probe on) ;
Z-10. (Rapid spindle down to position) ;
G91 G01 Z-1. F20. (Incremental feed by Z-1.) ;
G31 Y-1. F10. M75 (Measure & record Y reference) ;
G01 Y0.25 F20. (Feed away from surface) ;
G00 Z2. (Rapid retract) ;
Y-2. (Move to opposite side of part) ;
G01 Z-2. F20. (Feed by Z-2.) ;
G136 Y1. F10. ;
(Measure and record center in the Y axis) ;
G01 Y-0.25 (Feed away from surface) ;
G00 Z1. (Rapid retract) ;
M69 P1134 (Spindle probe off) ;
(BEGIN COMPLETION BLOCKS) ;
G00 G90 G53 Z0. (Rapid retract to Z home) ;
M30 (End program) ;
%

```

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 - Feedrate

* indicates optional

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. Similar to other uses of I, J, and K, these are incremental distances from the X, Y, and Z point called.

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 toolpath 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 tool wear diameter offset to use. A feedrate must be programmed on each line if in G93 Inverse Time Feed mode.

With a unit vector, the length of the vector line must always equal 1. In the same way that a unit circle in mathematics is a circle with a radius of 1, a unit vector is a line that indicates a direction with a length of 1. Remember, the vector line does not tell the control how far to move the tool when a wear value is entered, just the direction in which to go.

Only the endpoint of the commanded block is compensated in the direction of I, J, and K. For this reason, this compensation is recommended only for surface toolpaths having a tight tolerance (small motion between blocks of code). G141 compensation does not prohibit the toolpath from crossing over itself when excessive cutter compensation is entered. The tool will be offset, in the direction of the vector line, by the combined values of the tool offset geometry plus the tool offset wear. If compensation values are in diameter mode (Setting 40), the move will be half the amount entered in these fields.

For best results, program from the tool center using a ball nose endmill.

```
%  
O61411 (G141 3D CUTTER COMPENSATION) ;  
(G54 X0 Y0 is at the bottom-left) ;  
(Z0 is on top of the part) ;  
(T1 is a ball nose endmill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X0 Y0 Z0 A0 B0 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G141 D01 X0. Y0. Z0. ;  
(Rapid to position with 3D+ cutter comp) ;  
G01 G93 X.01 Y.01 Z.01 I.1 J.2 K.9747 F300. ;  
(Inverse time feed on, 1st linear motion) ;  
N1 X.02 Y.03 Z.04 I.15 J.25 K.9566 F300. (2nd motion) ;
```

```

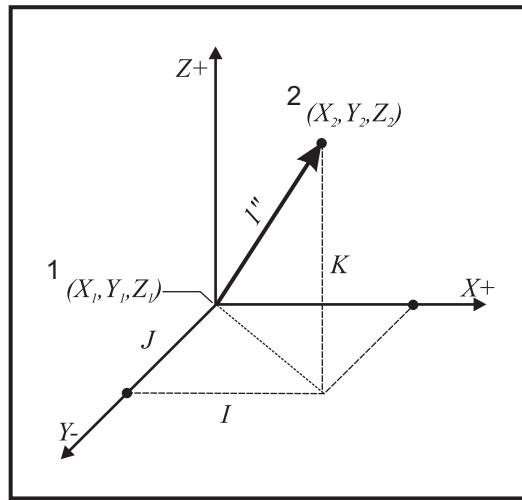
X.02 Y.055 Z.064 I.2 J.3 K.9327 F300. (3rd motion) ;
X2.345 Y.1234 Z-1.234 I.25 J.35 K.9028 F200. ;
(Last motion) ;
(BEGIN COMPLETION BLOCKS) ;
G94 F50. (Inverse time feed off) ;
G00 G90 G40 Z0.1 M09 (Cutter comp off) ;
(Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%

```

In the above example, we can see where the *I*, *J*, and *K* were derived by plugging the points into the following formula:

$AB = [(x_2 - x_1)^2 + (y_2 - y_1)^2 + (z_2 - z_1)^2]$, a 3D version of the distance formula. Looking at line N1, we use 0.15 for x_2 , 0.25 for y_2 , and 0.9566 for Z_2 . Because *I*, *J*, and *K* are incremental, we will use 0 for x_1 , y_1 , and z_1 .

- F7.43:** Unit Vector Example: The commanded line endpoint [1] is compensated in the direction of the vector line [2](*I,J,K*), by the amount of the Tool Offset Wear.



$$\begin{aligned}
 & \% \\
 & AB = [(.15)^2 + (.25)^2 + (.9566)^2] \\
 & AB = [.0225 + .0625 + .9150] \\
 & AB = 1 \\
 & %
 \end{aligned}$$

A simplified example is listed below:

```

%
061412 (G141 SIMPLE 3D CUTTER COMPENSATION) ;

```

```
(G54 X0 Y0 is at the bottom-left) ;
(Z0 is on top of the part) ;
(T1 is a ball nose endmill) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54 (Safe startup) ;
G00 G54 X0 Y0 (Rapid to 1st position) ;
S1000 M03 (Spindle on CW) ;
G43 H01 Z0.1 (Activate tool offset 1) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G141 D01 X0. Y0. Z0. ;
(Rapid to position with 3D+ cutter compensation) ;
N1 G01 G93 X5. Y0. I0. J-1. K0. F300. ;
(Inverse time feed on & linear motion) ;
(BEGIN COMPLETION BLOCKS) ;
G94 F50. (Inverse time feed off) ;
G00 G90 G40 Z0.1 M09 (Cutter compensation off) ;
(Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%
```

In this case, the wear value (DIA) for T01 is set to -.02. Line N1 moves the tool from (X0., Y0., Z0.) to (X5., Y0., Z0.). The J value tells the control to compensate the endpoint of the programmed line only in the Y Axis.

Line N1 could have been written using only the J-1. (not using I0. or K0.), but a Y value must be entered if a compensation is to be made in this axis (J value used).

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, such as VR-series mills)

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.

```
%  
O61431 (G143 5-AXIS TOOL LENGTH) ;  
(G54 X0 Y0 is at the top-right) ;  
(Z0 is on top of the part) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X0 Y0 Z0 A0 B0 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G143 H01 X0. Y0. Z0. A-20. B-20. ;  
(Rapid to position w/ 5 Axis tool length comp) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 G93 X.01 Y.01 Z.01 A-19.9 B-19.9 F300. ;  
(Inverse time feed on , 1st linear motion) ;  
X0.02 Y0.03 Z0.04 A-19.7 B-19.7 F300. ( 2nd motion) ;  
X0.02 Y0.055 Z0.064 A-19.5 B-19.6 F300. (3rd motion) ;  
X2.345 Y.1234 Z-1.234 A-4.127 B-12.32 F200. ;  
(Last motion) ;  
(BEGIN COMPLETION BLOCKS) ;  
G94 F50. (Inverse time feed off) ;  
G00 G90 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Tool length comp off) ;  
(Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

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** - Spindle speed
X - X start position
Y - Y start position
Z - Final depth of pocket

* indicates optional

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, which 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.

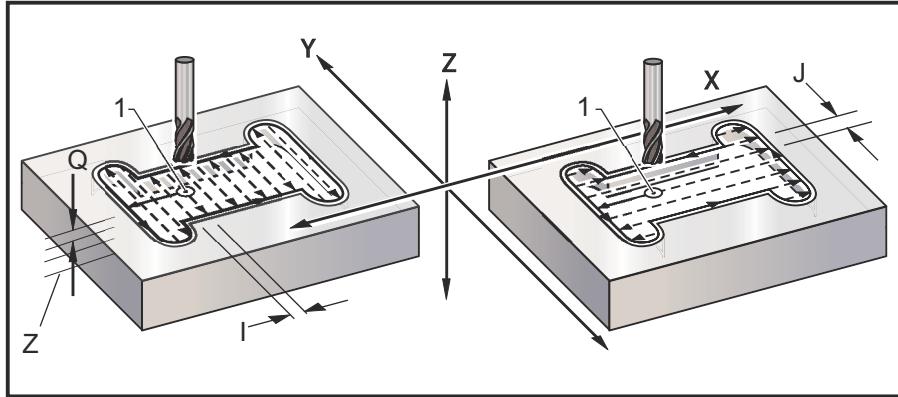
Multiple passes in the pocket area are done, starting from the R plane, with each Q (Z-Axis depth) pass to the final depth. The G150 command will first make a pass around pocket geometry, leaving stock with K, then doing passes of I or J roughing out inside of pocket after feeding down by the value in 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.

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.

F7.44: G150 General Pocket Milling: [1] Start Point, [Z] Final depth.

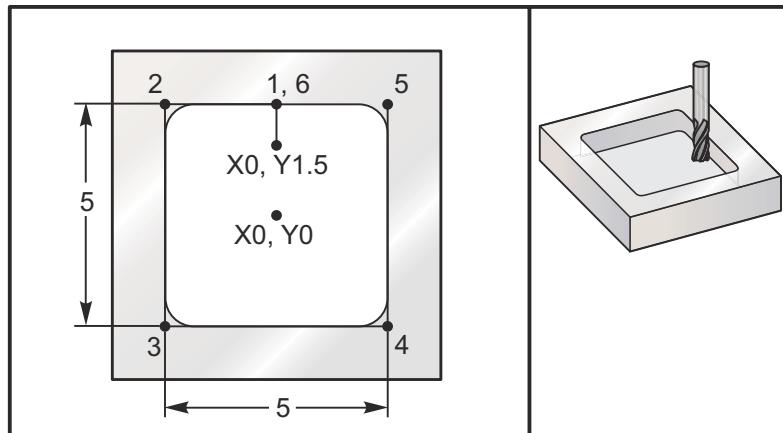


```
%  
O61501 (G150 GENERAL POCKET MILLING) ;  
(G54 X0 Y0 is at the bottom-left) ;  
(Z0 is on top of the part) ;  
(T1 is a .5" endmill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X3.25 Y4.5 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z1.0 (Activate tool offset 1) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G150 X3.25 Y4.5 Z-1.5 G41 J0.35 K.01 Q0.25 R.1 P61502 D01 F15.  
;  
(Pocket mill sequence, call pocket subprogram) ;  
(Cutter comp on) ;  
(0.01" finish pass (K) on sides) ;  
G40 X3.25 Y4.5 (Cutter comp off) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%  
%  
O61502 (G150 GENERAL POCKET MILL SUBPROGRAM) ;
```

```
(Subprogram for pocket in O61501) ;
(Must have a feedrate in G150) ;
G01 Y7. (First linear move onto pocket geometry) ;
X1.5 (Linear move) ;
G03 Y5.25 R0.875 (CCW arc) ;
G01 Y2.25 (Linear move) ;
G03 Y0.5 R0.875 (CCW arc) ;
G01 X5. (Linear move) ;
G03 Y2.25 R0.875 (CCW arc) ;
G01 Y5.25 (Linear move) ;
G03 Y7. R0.875 (CCW arc) ;
G01 X3.25 (Close pocket geometry) ;
M99 (Exit to Main Program) ;
%
```

Square Pocket

F7.45: G150 General Purpose Pocket Milling: 0.500 diameter endmill.



5.0 x 5.0 x 0.500 DP. Square Pocket

Main Program

```
%  
O61503 (G150 SQUARE POCKET MILLING) ;
(G54 X0 Y0 is at the center of the part) ;
(Z0 is on top of the part) ;
(T1 is a .5" endmill) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54 (Safe startup) ;
G00 G54 X0 Y1.5 (Rapid to 1st position) ;
S1000 M03 (Spindle on CW) ;
G43 H01 Z1.0 (Activate tool offset 1) ;
```

```

M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G01 Z0.1 F10. (Feed right above the surface) ;
G150 P61504 Z-0.5 Q0.25 R0.01 J0.3 K0.01 G41 D01 F10. ;
(Pocket mill sequence, call pocket subprogram) ;
(Cutter comp on) ;
(0.01" finish pass (K) on sides) ;
G40 G01 X0. Y1.5 (Cutter comp off) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract,Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%

```

Subprogram

```

%
O61505 (G150 INCREMENTAL SQUARE POCKET MILLING SUBPROGRAM) ;
(Subprogram for pocket in O61503) ;
(Must have a feedrate in G150) ;
G91 G01 Y0.5 (Linear move to position 1) ;
X-2.5 (Linear move to position 2) ;
Y-5. (Linear move to position 3) ;
X5. (Linear move to position 4) ;
Y5. (Linear move to position 5) ;
X-2.5 (Linear move to position 6, Close Pocket Loop) ;
G90 (Turn off incremental mode, Turn on absolute) ;
M99 (Exit to Main Program) ;
%
```

Absolute and Incremental examples of a subprogram called up by the P#### command in the G150 line:

Absolute Subprogram

```

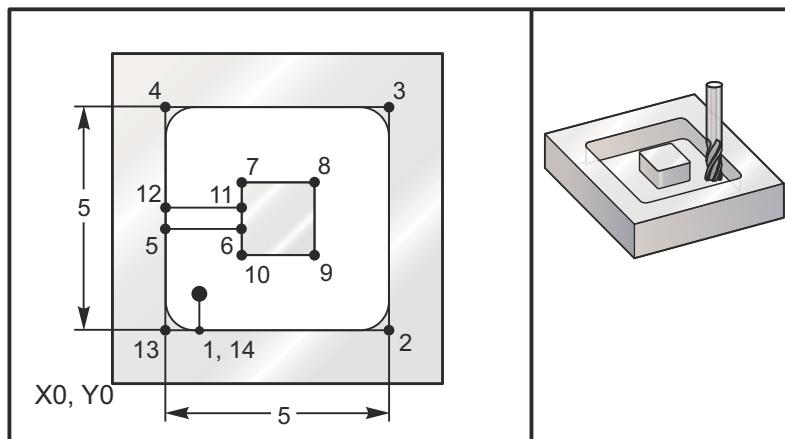
%
O61504 (G150 ABSOLUTE SQUARE POCKET MILLING SUBPROGRAM) ;
(Subprogram for pocket in O61503) ;
(Must have a feedrate in G150) ;
G90 G01 Y2.5 (Linear move to position 1) ;
X-2.5 (Linear move to position 2) ;
Y-2.5 (Linear move to position 3) ;
X2.5 (Linear move to position 4) ;
Y2.5 (Linear move to position 5) ;
X0. (Linear move to position 6, Close Pocket Loop) ;
M99 (Exit to Main Program) ;
%
```

Incremental Subprogram

```
%  
O61505 (G150 INCREMENTAL SQUARE POCKET MILLING SUBPROGRAM) ;  
(Subprogram for pocket in O61503) ;  
(Must have a feedrate in G150) ;  
G91 G01 Y0.5 (Linear move to position 1) ;  
X-2.5 (Linear move to position 2) ;  
Y-5. (Linear move to position 3) ;  
X5. (Linear move to position 4) ;  
Y5. (Linear move to position 5) ;  
X-2.5 (Linear move to position 6, Close Pocket Loop) ;  
G90 (Turn off incremental mode, Turn on absolute) ;  
M99 (Exit to Main Program) ;  
%
```

Square Island

F7.46: G150 Pocket Milling Square Island: 0.500 diameter endmill.



5.0 x 5.0 x 0.500 DP. Square Pocket with Square Island

Main Program

```
%  
O61506 (G150 SQUARE ISLAND POCKET MILLING) ;  
(G54 X0 Y0 is at the bottom-left) ;  
(Z0 is on top of the part) ;  
(T1 is a .5" endmill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X2. Y2. (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;
```

```

G43 H01 Z1.0 (Activate tool offset 1) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G01 Z0.01 F30. (Feed right above the surface) ;
G150 P61507 X2. Y2. Z-0.5 Q0.5 R0.01 I0.3 K0.01 G41 D01 F10. ;
(Pocket mill sequence, call pocket subprogram) ;
(Cutter comp off) ;
(0.01" finish pass (K) on sides) ;
G40 G01 X2.Y2. (Cutter comp off) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%

```

Subprogram

```

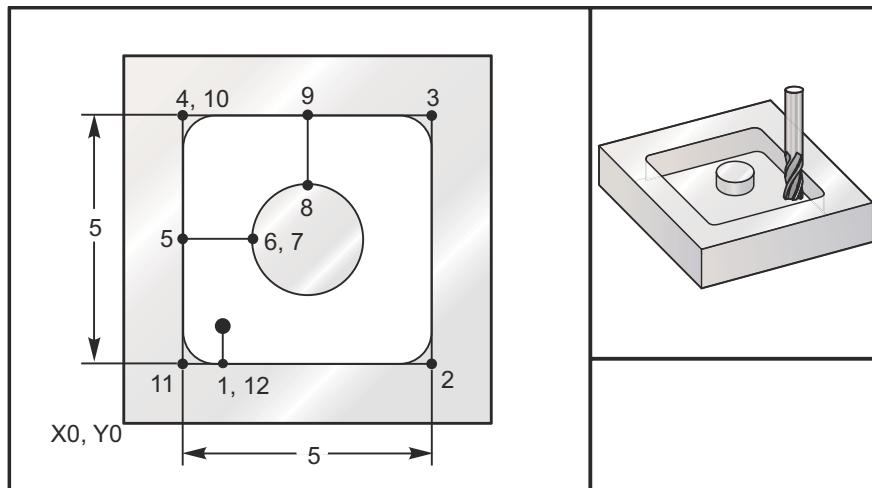
%
O61507 (G150 SQUARE ISLAND POCKET MILLING SUBPROGRAM) ;
(Subprogram for pocket in O61503) ;
(Must have a feedrate in G150) ;
G01 Y1. (Linear move to position 1) ;
X6. (Linear move to position 2) ;
Y6. (Linear move to position 3) ;
X1. (Linear move to position 4) ;
Y3.2 (Linear move to position 5) ;
X2.75 (Linear move to position 6) ;
Y4.25 (Linear move to position 7) ;
X4.25 (Linear move to position 8) ;
Y2.75 (Linear move to position 9) ;
X2.75 (Linear move to position 10) ;
Y3.8 (Linear move to position 11) ;
X1. (Linear move to position 12) ;
Y1. (Linear move to position 13) ;
X2. (Linear move to position 14, Close Pocket Loop) ;
M99 (Exit to Main Program) ;
%

```

Round Island

Introduction

F7.47: G150 Pocket Milling Round Island: 0.500 diameter endmill.



5.0 x 5.0 x 0.500 DP. Square Pocket with Round Island

Main Program

```
%  
O61508 (G150 SQ POCKET W/ ROUND ISLAND MILLING) ;  
(G54 X0 Y0 is at the bottom-left) ;  
(Z0 is on top of the part) ;  
(T1 is a .5" endmill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X2. Y2. (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H01 Z1.0 M08 (Activate tool offset 1) ;  
(Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 Z0.01 F30. (Feed right above the surface) ;  
G150 P61509 X2. Y2. Z-0.5 Q0.5 R0.01 J0.3 K0.01 G41 D01 F10. ;  
(Pocket mill sequence, call pocket subprogram) ;  
(Cutter comp on) ;  
(0.01" finish pass (K) on sides) ;  
G40 G01 X2.Y2. (Cutter comp off) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

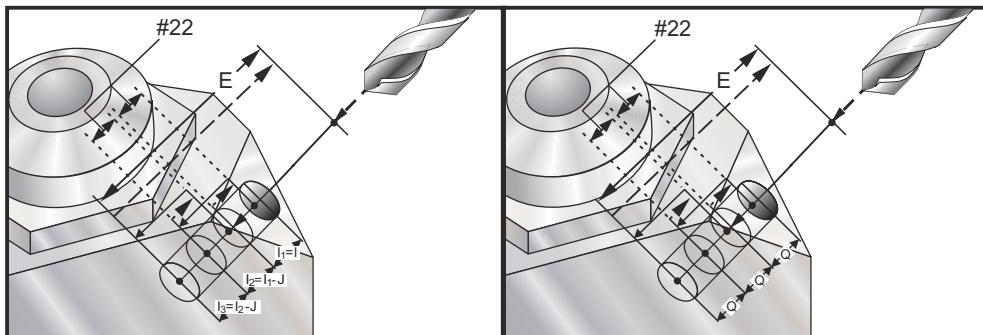
Subprogram

```
%  
O61509 (G150 SQ POCKET W/ ROUND ISLAND MILLING SUBPROGRAM) ;  
(Subprogram for pocket in O61503) ;  
(Must have a feedrate in G150) ;  
G01 Y1. (Linear move to position 1) ;  
X6. (Linear move to position 2) ;  
Y6. (Linear move to position 3) ;  
X1. (Linear move to position 4) ;  
Y3.5 (Linear move to position 5) ;  
X2.5 (Linear move to position 6) ;  
G02 I1. (CW circle along X axis at position 7) ;  
G02 X3.5 Y4.5 R1. (CW arc to position 8) ;  
G01 Y6. (Linear move to position 9) ;  
X1. (Linear move to position 10) ;  
Y1. (Linear move to position 11) ;  
X2. (Linear move to position 12, Close Pocket Loop) ;  
M99 (Exit 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
- 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

F7.48: G153 5-Axis High Speed Peck Drilling: [#22] Setting 22.



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:

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:

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)
```

Introduction

#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 performs only 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

L - Number of repeats

A - A-Axis tool starting position

B - B-Axis tool starting position

X - X-Axis tool starting position

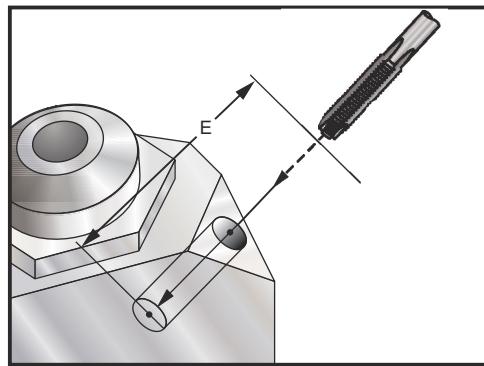
Y - Y-Axis tool starting position

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 automatically starts the spindle counterclockwise before this canned cycle.

F7.49: G155 5-Axis Reverse Tap Canned Cycle



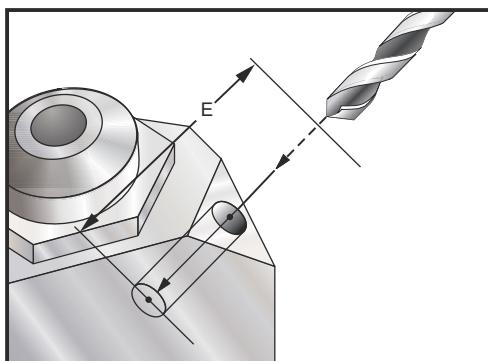
G161 5-Axis Drill Canned Cycle (Group 09)

- E** - Specifies the distance from the start position to the bottom of the hole
- F** - Feedrate
- 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



CAUTION: Unless you specify otherwise, this canned cycle uses the most recently commanded spindle direction (M03, M04, or M05). If the program did not specify a spindle direction before it commands this canned cycle, the default is M03 (clockwise). If you command M05, the canned cycle will run as a "no-spin" cycle. This lets you run applications with self-driven tools, but it can also cause a crash. Be sure of the spindle direction command when you use this canned cycle.

F7.50: G161 5-Axis Drill Canned Cycle



A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded.

```
%  
(G54 X0 Y0 is) ;  
(Z0 is on the top of the part) ;  
(T1 - n/a ) ;  
;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
G00 G54 X0 Y0 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;
```

```

G43 H01 Z0.1 M08 (Activate tool offset 1, Coolant on) ;
;
(BEGIN CUTTING BLOCKS) ;
(DRILL RIGHT, FRONT) ;
G01 G54 G90 X8. Y-8. B23. A22. F360. (Clearance Position) ;
G143 H01 Z15. M8 ;
G01 X7. Y-7. Z11. F360. (Initial Start position) ;
G161 E.52 F7. (Begin G161) ;
G80 ;
X8. Y-8. B23. A22. Z15. (Clearance Position) ;
;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home and Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%

```

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
- 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



CAUTION:

Unless you specify otherwise, this canned cycle uses the most recently commanded spindle direction (M03, M04, or M05). If the program did not specify a spindle direction before it commands this canned cycle, the default is M03 (clockwise). If you command M05, the canned cycle will run as a "no-spin" cycle. This lets you run applications with self-driven tools, but it can also cause a crash. Be sure of the spindle direction command when you use this canned cycle.

A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded.

```

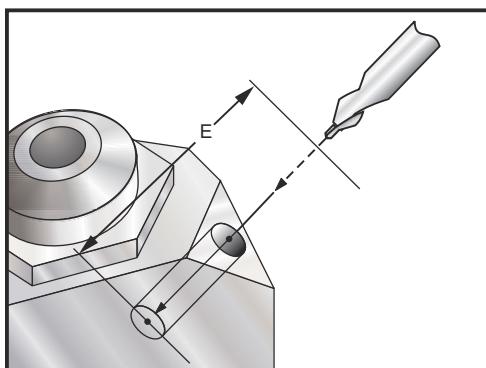
%
(COUNTER DRILL RIGHT, FRONT) ;
T2 M6 ;
G01 G54 G90 X8.4221 Y-8.4221 B23. A21.342 S2200 M3 F360.
(Clearance Position) ;

```

Introduction

```
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 ;
%
```

F7.51: G162 Spot Drill Canned Cycle



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

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:

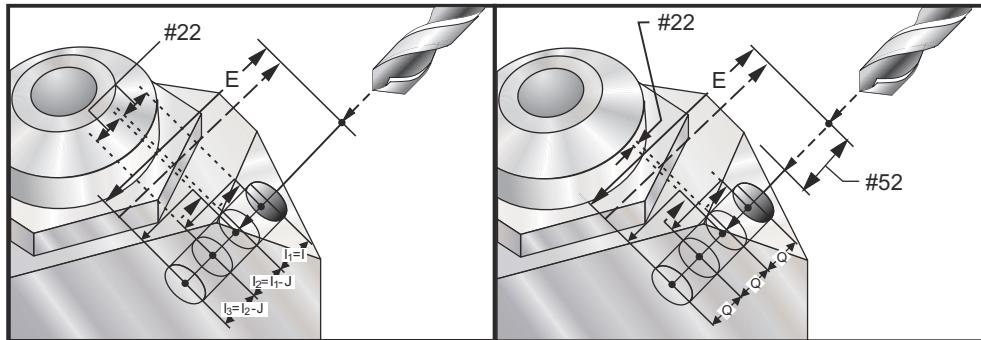
G163 E0.62 F15. Q0.175 P1.5.



NOTE:

The same dwell time applies to all subsequent blocks that do not specify a dwell time.

F7.52: G163 5-Axis Normal Peck Drilling Canned Cycle: [#22] Setting 22, [#52] Setting 52.



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 moves above the start position by the amount given in this setting.

```
%  
(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 performs only floating taps. G174/G184 is available for 5-axis rigid tapping.

E - Specifies the distance from the start position to the bottom of the hole

F - Feedrate

A - A-Axis tool starting position

B - B-Axis tool starting position

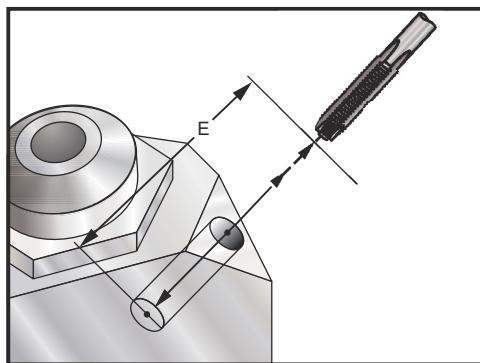
X - X-Axis tool starting position

Y - Y-Axis tool starting position

Z - Z-Axis tool starting position

S - Spindle Speed

F7.53: G164 5-Axis Tapping Canned Cycle



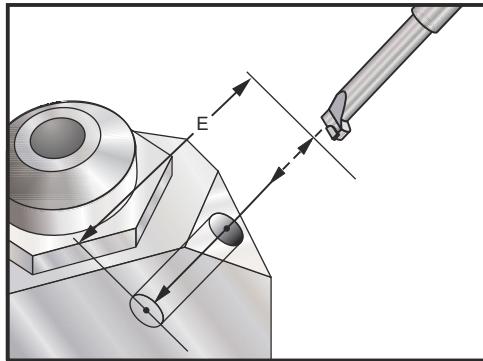
A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded. The control automatically starts the spindle CW before this canned cycle.

```
%  
(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
- 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

F7.54: G165 5-Axis Boring Canned Cycle



A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded.

```
%  
(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 ;  
G00 G28 G91 Z0. ;  
G91 G28 B0. A0. ;  
M01 ;  
%
```

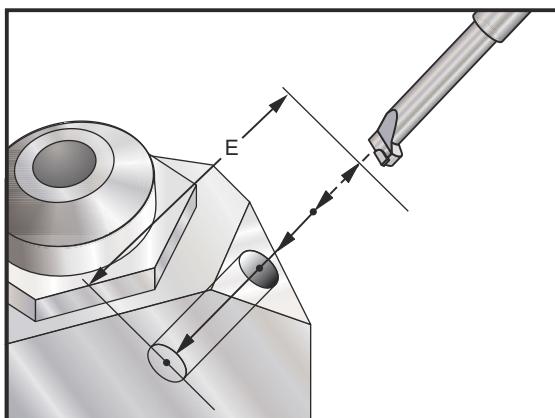
G166 5-Axis Bore and Stop Canned Cycle (Group 09)

- E** - Specifies the distance from the start position to the bottom of the hole
- F** - Feedrate
- 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



CAUTION: Unless you specify otherwise, this canned cycle uses the most recently commanded spindle direction (M03, M04, or M05). If the program did not specify a spindle direction before it commands this canned cycle, the default is M03 (clockwise). If you command M05, the canned cycle will run as a "no-spin" cycle. This lets you run applications with self-driven tools, but it can also cause a crash. Be sure of the spindle direction command when you use this canned cycle.

F7.55: G166 5-Axis Bore and Stop Canned Cycle



A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded.

```
%  
(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 ;
G00 G28 G91 Z0. ;
G91 G28 B0. A0. ;
M01 ;
%

```

G169 5-Axis Bore and Dwell Canned Cycle (Group 09)

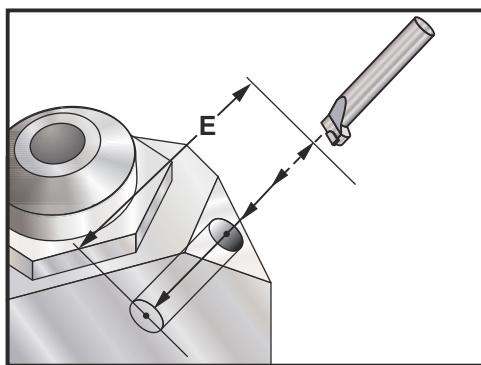
- E** - Specifies the distance from the start position to the bottom of the hole
- F** - Feedrate
- P** - The dwell time at the bottom of the hole
- A** - A-Axis tool starting position
- B** - B-Axis tool starting position
- X** - X-Axis tool starting position
- Y** - Y-Axis tool starting position
- Z** - Z-Axis tool starting position



CAUTION:

Unless you specify otherwise, this canned cycle uses the most recently commanded spindle direction (M03, M04, or M05). If the program did not specify a spindle direction before it commands this canned cycle, the default is M03 (clockwise). If you command M05, the canned cycle will run as a “no-spin” cycle. This lets you run applications with self-driven tools, but it can also cause a crash. Be sure of the spindle direction command when you use this canned cycle.

F7.56: 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.

%

```
(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 P0.5 F12. (Canned Cycle) ;
G80 ;
X8.4221 Y-8.4221 B23. A21.342 Z14.6228 (Clearance Position) ;
M5 ;
G00 G28 G91 Z0. ;
G91 G28 B0. A0. ;
M01 ;
%
```

G174 CCW - G184 CW Non-Vertical Rigid Tap (Group 00)

F - Feedrate

X - X position at bottom of hole

Y - Y position at bottom of hole

Z - Z position at bottom of hole

***S** - Spindle Speed

* indicates optional

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)

G187 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). Temporarily overrides Setting 191.

E - Sets the max corner rounding value. Temporarily overrides 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 85 to a low value may make the machine operate as if it is in exact stop mode.

**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 canceled whenever [**RESET**] is pressed, M30 or M02 is executed, the end of program is reached, or [**EMERGENCY 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.

G234 Tool Center Point Control (TCPC) (Group 08)

G234 Tool Center Point Control (TCPC) lets a machine correctly run a contouring 4- or 5-axis program when the workpiece is not located in the exact location specified by the CAM-generated program. This eliminates the need to repost a program from the CAM system when the programmed and the actual workpiece locations are different.

For more information refer to the UMC-750 Operator's Manual Supplement.

G254 Dynamic Work Offset (DWO) (Group 23)

G254 Dynamic Work Offset (DWO) is similar to TCPC, except that it is designed for use with 3+1 or 3+2 positioning, not for simultaneous 4- or 5-axis machining. If the program does not use the B and C Axes, there is no need to use DWO.

For more information refer to the UMC-750 Operator's Manual Supplement.

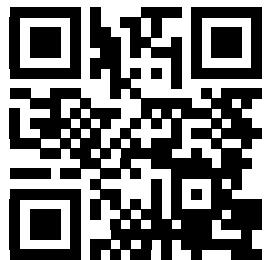
G255 Cancel Dynamic Work Offset (DWO) (Group 23)

G255 cancels G254 Dynamic Work Offset (DWO).

More Information Online

7.2 More Information Online

For updated and supplemental information, including tips, tricks, maintenance procedures, and more, visit the Haas Resource Center at diy.HaasCNC.com. You can also scan the code below with your mobile device to go directly to the Resource Center:



Chapter 8: M-codes

8.1 Introduction

This chapter gives detailed descriptions of the M-codes that you use to program your machine.


CAUTION:

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


NOTE:

The sample programs in this manual represent a very conservative programming style. The samples are intended to demonstrate safe and reliable programs, and they are not necessarily the fastest or most efficient way to operate a machine. The sample programs use G-codes that you might choose not to use in more efficient programs.

8.1.1 List of M-codes

Code	Description	Page
M00	Stop Program	335
M01	Optional Program Stop	335
M02	Program End	335
M03	Spindle Commands	335
M04	Spindle Commands	335
M05	Spindle Commands	335
M06	Tool Change	335
M07	Shower Coolant On	336

Introduction

Code	Description	Page
M08	Coolant On	336
M09	Coolant Off	336
M10	Engage 4th Axis Brake	337
M11	Release 4th Axis Brake	337
M12	Engage 5th Axis Brake	337
M13	Release 5th Axis Brake	337
M16	Tool Change	337
M17	Unclamp APC Pallet and Open APC Door	337
M18	Clamp APC Pallet and Close Door	337
M19	Orient Spindle	337
M21	Optional User M Function with M-Fin	338
M22	Optional User M Function with M-Fin	338
M23	Optional User M Function with M-Fin	338
M24	Optional User M Function with M-Fin	338
M25	Optional User M Function with M-Fin	338
M26	Optional User M Function with M-Fin	338
M27	Optional User M Function with M-Fin	338
M28	Optional User M Function with M-Fin	338
M30	Program End and Reset	339
M31	Chip Conveyor Forward	339
M33	Chip Conveyor Stop	339
M34	Coolant Increment	339
M35	Coolant Decrement	339

Code	Description	Page
M36	Pallet Part Ready	340
M39	Rotate Tool Turret	340
M41	Low Gear Override	340
M42	High Gear Override	340
M46	Jump if Pallet Loaded	340
M48	Check Validity of Current Program	341
M49	Set Status of Pallet	341
M50	Execute Pallet Change	341
M51	Set Optional User M-codes	341
M52	Set Optional User M-codes	341
M53	Set Optional User M-codes	341
M54	Set Optional User M-codes	341
M55	Set Optional User M-codes	341
M56	Set Optional User M-codes	341
M57	Set Optional User M-codes	341
M58	Set Optional User M-codes	341
M59	Set Output Relay	341
M61	Clear Optional User M-codes	342
M62	Clear Optional User M-codes	342
M63	Clear Optional User M-codes	342
M64	Clear Optional User M-codes	342
M65	Clear Optional User M-codes	342
M66	Clear Optional User M-codes	342

Introduction

Code	Description	Page
M67	Clear Optional User M-codes	342
M68	Clear Optional User M-codes	342
M69	Clear Output Relay	342
M75	Set G35 or G136 Reference Point	342
M76	Control Display Inactive	342
M77	Control Display Active	342
M78	Alarm if Skip Signal Found	342
M79	Alarm if Skip Signal Not Found	343
M80	Auto Door Open	343
M81	Auto Door Close	343
M82	Tool Unclamp	343
M83	Auto Air Gun On	343
M84	Auto Air Gun Off	343
M86	Tool Clamp	343
M88	Through-Spindle Coolant On	343
M89	Through-Spindle Coolant Off	343
M95	Sleep Mode	344
M96	Jump If No Input	344
M97	Local Sub-Program Call	345
M98	Sub-Program Call	345
M99	Sub-Program Return or Loop	346
M109	Interactive User Input	347

About M-codes

M-codes are miscellaneous machine commands that do not command axis motion. The format for an M-code is the letter M followed by two to three digits; for example M03.

Only one M-code is allowed per line of code. All M-codes take effect at the end of the block.

M00 Stop Program

The M00 code stops a program. It stops the axes, spindle, and turns off the coolant (including auxiliary coolant). The next block after the M00 is highlighted when viewed in the program editor. Press [CYCLE START] to 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. Press [OPTION STOP] to toggle the feature on and off.

M02 Program End

M02 ends a program.



NOTE:

The most common way of ending a program is with an M30.

M03 / M04 / M05 Spindle CW / CCW / Stop

M03 turns the spindle on in the clockwise (CW) direction.

M04 turns the spindle on in the counter-clockwise (CCW) direction.

M05 stops the spindle, and waits for it to stop.

Spindle speed is controlled with an S address code; for example, S5000 commands a spindle speed of 5000 RPM.

If your machine has a gearbox, the spindle speed you program determines the gear that the machine uses, unless you use M41 or M42 to override gear selection. Refer to page 340 for more information on the gear select override M-codes.

M06 Tool Change

T - Tool number

The M06 code is used to change tools. For example, M06 T12 puts tool 12 into the spindle. If the spindle is running, the spindle and coolant (including TSC) is stopped by the M06 command.



NOTE:

The M06 command automatically stops the spindle, stops coolant, moves the Z Axis to the tool change position, and orients the spindle for the tool change. You do not need to include these commands for a tool change in your program.



NOTE:

M00, M01, any work offset G-code (G54, etc.), and block delete slashes before a tool change stop look-ahead, and the control does not pre-call the next tool to the change position (for a side-mount tool changer only). This can cause significant delays to program execution, because the control must wait for the tool to arrive at the change position before it can execute the tool change. You can command the carousel to the tool position with a T code after a tool change; for example:

```
M06 T1 (FIRST TOOL CHANGE) ;  
T2 (PRE-CALL THE NEXT TOOL) ;
```

Refer to page 99 for more information on side-mount tool changer programming.

M07 Shower Coolant On

M07 starts the optional shower coolant. M09 stops the shower coolant and also stops the standard coolant. The optional shower coolant stops automatically before a tool change or a pallet change. It automatically starts again after a tool change if it was ON before a tool change command.



NOTE:

Some machines use optional relays and optional M-codes to command shower coolant, such as M51 on and M61 off. Check your machine configuration for the correct M-code programming.

M08 Coolant On / M09 Coolant Off

M08 starts the optional coolant supply and M09 stops it. Use M34/M35 to start and stop the optional Programmable Coolant (P-Cool). Use M88/M89 to start and stop the optional Through-Spindle Coolant.



NOTE:

The control checks the coolant level only at the start of a program, so a low coolant condition will not stop a running program.



CAUTION: *Do not use straight or “neat” mineral cutting oils. They cause damage to rubber components in the machine.*

M10 Engage 4th Axis Brake / M11 Release 4th Axis Brake

M10 applies the brake to the optional 4th axis and M11 releases the brake. The optional 4th axis brake is normally engaged, so the M10 command is only required when an M11 has released the brake.

M12 Engage 5th Axis Brake / M13 Release 5th Axis Brake

M12 applies the brake to the optional 5th axis and M13 releases the brake. The optional 5th axis brake is normally engaged, so the M12 command is only required when an M13 has released the brake.

M16 Tool Change

T - Tool number

This M16 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 APC Pallet and Close APC Door

M17 unclamps the APC pallet and opens the APC door on vertical machining centers with pallet changers. M18 clamps the APC pallet and closes the APC door. M17 / M18 are used for maintenance and test only. Use M50 for pallet changes.

M19 Orient Spindle (Optional P and R Values)

P - Number of degrees (0 - 360)

R - Number of degrees with two decimal places (0.00 - 360.00).

M19 adjusts the spindle to a fixed position. The spindle only orients to the zero position without the optional M19 orient spindle feature. The orient spindle function allows P and R address codes. For example:

```
M19 P270. (orients the spindle to 270 degrees) ;
```

The R-value allows the programmer to specify up to two decimal places; for example:

```
M19 R123.45 (orients the spindle to 123.45 degrees) ;
```

M21-M28 Optional User M Function with M-Fin

M21 through M28 are optional for user-defined relays. Each M-code closes one of the optional relays. [RESET] stops any operation waiting for a relay-activated accessory to finish. Also, see M51 through M58 and M61 through M68.

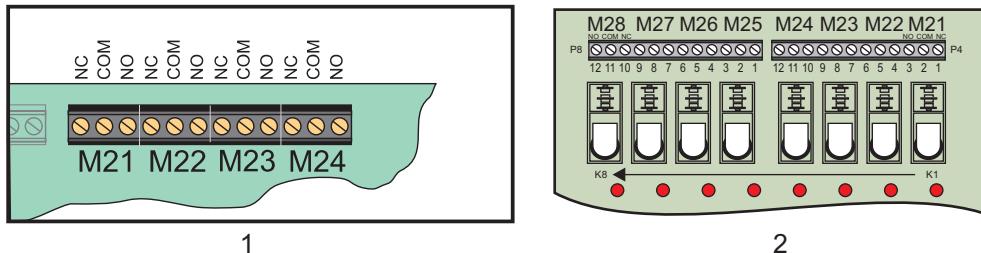
Some or all of the M21 through M25 (M21 through M22 on Toolroom, Office, and Mini mills) on the I/O PCB may be used for factory-installed options. Inspect the relays for existing wires to determine which have been used. Contact your dealer for more details.

Only one relay is switched 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 are 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 positions for, Normally Open (NO), Normally Closed (NC), and Common (COM).

- F8.1:** Main I/O PCB M-code Relays: [1] Main I/O PCB M-code relays, [2] 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 IO PCB. 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/M69. Parameter 352 is shipped set to 1 as standard.

M30 Program End and Reset

M30 stops a program. It also stops the spindle, turns off the coolant (including TSC), and returns the program cursor to the start of the program.


NOTE:

M30 cancels tool length offsets.

M31 Chip Conveyor Forward / M33 Chip Conveyor Stop

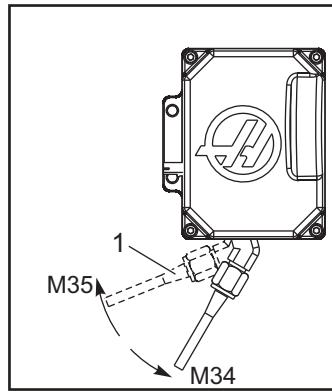
M31 starts the optional chip removal system (auger, multi-auger, or belt-style conveyor) in the forward direction; the direction that moves the chips out of the machine. You should run the chip conveyor intermittently, as this allows piles of larger chips to collect smaller chips and carry them out of the machine. You can set the chip conveyor duty cycle and run time with Settings 114 and 115.

The optional conveyor coolant washdown runs while the chip conveyor is on.

M33 stops conveyor motion.

M34 Coolant Increment / M35 Coolant Decrement

F8.2: P-Cool Spigot



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.


CAUTION:

Do not rotate the coolant spigot by hand. Serious motor damage will occur.

M36 Pallet Part Ready

Used on machines with pallet changers. M36 delays the pallet change until **[PART READY]** is pushed. A pallet change occurs after **[PART READY]** is pushed and the doors are closed. For example:

```
%  
Onnnnn (program number) ;  
M36 (Flash "Part Ready" light, wait until the button is  
pressed) ;  
M01 ;  
M50 (Perform pallet change after [PART READY] is pushed) ;  
(Part Program) ;  
M30 ;  
%
```

M39 Rotate Tool Turret

M39 is used to rotate the side mount tool changer without a tool change. Program the tool pocket number (Tn) before M39.

M06 is the command to change tools. M39 is normally useful for diagnostic purposes, or to recover from a tool changer crash.

M41 / M42 Low / High Gear Override

On machines with a transmission, M41 holds the machine in low gear and M42 holds the machine in high gear. Normally, the spindle speed (Snnnn) determines which gear the transmission should be in.

Command M41 or M42 with the spindle speed before the spindle start command, M03. For example:

```
%  
S1200 M41 ;  
M03 ;  
%
```

The gear state reverts to default at the next spindle speed (Snnnn) command. The spindle does not have to stop.

M46 Jump if Pallet Loaded

P - Program line number to go to when conditional test is met

Q - Pallet number.

M46 causes the program to jump to the line number specified by the P-code if the pallet specified by the Q-code is currently loaded.

Example:

M46 Qm Pnn (Jump to line nn in the current program if pallet m is loaded, otherwise go to the next block) ;

M48 Check Validity of Current Program

M48 is a safeguard for pallet changing machines. Alarm 909 (910) displays if the current pallet program is not listed in the Pallet Schedule Table.

M49 Set Status of Pallet

M49 sets the status of the pallet specified by the P-code to the value specified by the Q-code. The possible Q-codes are: 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, [PALLET READY], or the Pallet Schedule Table to perform a pallet change.

M51-M58 Set Optional User M-codes

M51 through M58 are optional for user interfaces. They turn on one of the optional M-code relays on relay board 1. M61 through M68 turns the relay off. [RESET] turns off all of these relays.

Refer to M21 through M28 on page 338 for details on the M-code relays.

M59 Set Output Relay

P - Discrete output relay from 1100 to 1155.

M59 turns on a relay. An example of its usage is M59 P11nn, where nn is the number of the relay being turned on. M59 can turn on any of the discrete output relays in the range from 1100 to 1155 in the same order as axes motion. 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:

The 8 spare M functions on relay board 1 use addresses 1140 - 1147

M61-M68 Clear Optional User M-codes

M61 through M68 are optional and turn off one of the relays. The M number corresponds to M51 through M58 that turned on the relay. [RESET] turns off all of these relays. Refer to M21-M28 on page 338 for details on the M-code relays.

M69 Clear Output Relay

M69 turns off a relay. An example of its usage is M69 P11nn, where nn is the number of the relay being turned off. An M69 command can 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 in the same order as axes motion.

M73 Tool Air Blast (TAB) On / M74 TAB Off

These M-codes control the Tool Air Blast (TAB) option. M73 turns on TAB, and M74 turns it off.

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.

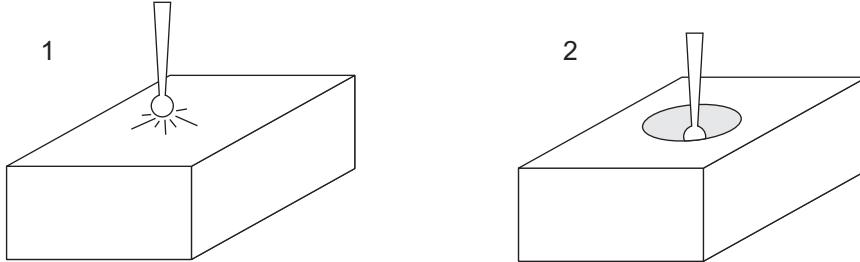
M78 Alarm if Skip Signal Found

M78 is used with a probe. An M78 generates 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

M79 is used with a probe. An M79 generates 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.

- F8.3:** Probe Positioning Error: [1] Signal Found. [2] Signal not Found.



M80 Auto Door Open / M81 Auto Door Close

M80 opens the Auto Door and M81 closes it. The control pendant beeps while the door is in motion.

M82 Tool Unclamp

M82 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 / MQL On / M84 Auto Air Gun / MQL Off

M83 turns the Auto Air Gun (AAG) or Minimum Quantity Lubrication (MQL) option on, and M84 turns it off. M83 with a Pnnn argument (where nnn is in milliseconds) turns the AAG or MQL on for the specified time, then turns it off. You can also press [SHIFT] and then [COOLANT] to turn on the AAG or MQL manually.

M86 Tool Clamp

M86 clamps 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

M88 turns on through-spindle coolant (TSC), and M89 turns off TSC.

The control automatically stops the spindle before it executes M88 or M89. The control does not automatically start the spindle again after M89. If your program continues with the same tool after an M89 command, be sure to add a spindle speed command before further motion.



CAUTION: You must use proper tooling, with a through-hole, when you use the TSC system. Failure to use proper tooling can flood the spindle head with coolant and void the warranty.

Sample Program



NOTE: The M88 command should be before the spindle speed command. If you command M88 after the spindle speed command, the spindle starts, then stops, turns on TSC, and then starts again.

```
%  
T1 M6 (TSC Coolant Through Drill) ;  
G90 G54 G00 X0 Y0 ;  
G43 H01 Z.5 ;  
M88 (Turn TSC on) ;  
S4400 M3 ;  
G81 Z-2.25 F44. R.1 ;  
M89 G80 (Turn TSC off) ;  
G91 G28 Z0 ;  
G90 ;  
M30 ;  
%
```

M95 Sleep Mode

Sleep mode is a long dwell. The format of the M95 command is: M95 (hh:mm).

The comment immediately following M95 must contain the duration, in hours and minutes, that you want the machine to sleep. For example, if the current time were 6 p.m. and you want the machine to sleep until 6:30 a.m. the next day, command M95 (12:30). The line(s) after 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)

M96 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 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 Nnnnn that matches 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 Subprogram Call

P - Program line number to go to when conditional test is met

L - Repeats subprogram call (1-99) times.

M97 is used to call a subprogram 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 subprograms within a program; does not require a separate program. The subprogram must end with an M99. Lnn code in the M97 block repeats the subprogram call nn times.

**NOTE:**

The subprogram is within the body of the main program, placed after the M30.

M97 Example:

```
%  
000001 ;  
M97 P100 L4 (CALLS N100 SUBPROGRAM) ;  
M30 ;  
N100 (SUBPROGRAM) ; ;  
M00 ;  
M99 (RETURNS TO MAIN PROGRAM) ;  
%
```

M98 Subprogram Call

P - Subprogram number to go to when conditional test is met

L - Repeats subprogram call (1-99) times.

M98 is used to call a subprogram, 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 M98 and causes the sub-program to be called nn times before continuing to the next block.

Introduction

When an M98 subprogram is called, the control looks for the subprogram on the active drive, and then in memory if the subprogram cannot be located. The active drive may be memory, USB drive, or hard drive. An alarm occurs if the control does not find the subprogram on either the active drive or in memory.

M98 Example:

The subprogram is a separate program (000100) from the main program (000002).

```
%  
000002 ;  
M98 P100 L4 (CALLS 000100 SUB 4 TIMES) ;  
M30 ;  
%  
%  
000100 (SUBPROGRAM) ;  
M00 ;  
M99 (RETURN TO MAIN PROGRAM) ;  
%
```

M99 Subprogram Return or Loop

P - Program line number to go to when conditional test is met

M99 has three main uses:

- M99 is used at the end of a subprogram, local subprogram, or macro to return back to the main program.
- An M99 Pnn jumps the program to the corresponding Nnn in the program.
- An M99 in the main program will cause the program to loop back to the beginning and execute until [RESET] is pressed.



NOTE:

Fanuc behavior is simulated by using the following code:

	Haas	Fanuc
calling program:	00001 ;	00001 ;

	N50 M98 P2 ;	N50 M98 P2 ;
	N51 M99 P100 ;	...

	Haas	Fanuc
	...	N100 (continue here) ;
	N100 (continue here) ;	...
	...	M30 ;
	M30 ;	
subprogram:	00002 ;	00002 ;
	M99 ;	M99 P100 ;

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=dddd in the sub-program and then using M99 P#nnn after the sub-program call.

M109 Interactive User Input

P - A number in the range (500-599) representing the macro variable of the same name.

M109 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 asks the user a Y or N question, then waits for either a Y or an N to be entered. All other characters are ignored.

```
%  
o61091 (M109 INTERACTIVE USER INPUT) ;  
(This program has no axis movement) ;  
N1 #501= 0. (Clear the variable) ;  
M109 P501 (Sleep 1 min?) ;  
N5 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) ;
```

Introduction

```
M30 ;  
%
```

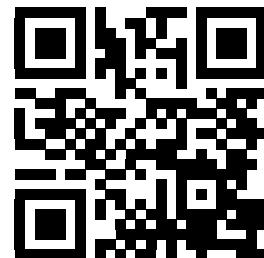
The following sample program asks the user to select a number, then wait for a 1, 2, 3, 4 or a 5 to be entered; all other characters are ignored.

```
%  
O00065 (M109 INTERACTIVE USER INPUT 2) ;  
(This program has no axis movement) ;  
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 (End Program);  
%
```

8.2 More Information Online

For updated and supplemental information, including tips, tricks, maintenance procedures, and more, visit the Haas Resource Center at diy.HaasCNC.com. You can also scan the code below with your mobile device to go directly to the Resource Center:



More Information Online

Chapter 9: Settings

9.1 Introduction

This chapter gives detailed descriptions of the settings that control the way that your machine works.

9.1.1 List of Settings

Setting	Description
1	Auto Power Off Timer
2	Power Off at M30
4	Graphics Rapid Path
5	Graphics Drill Point
6	Front Panel Lock
7	Parameter Lock
8	Prog Memory Lock
9	Dimensioning
10	Limit Rapid at 50%
11	Baud Rate Select
12	Parity Select
13	Stop Bit
14	Synchronization
15	H & T Code Agreement
16	Dry Run Lock Out
17	Opt Stop Lock Out
18	Block Delete Lock Out

Introduction

Setting	Description
19	Feedrate Override Lock
20	Spindle Override Lock
21	Rapid Override Lock
22	Can Cycle Delta Z
23	9xxx Progs Edit Lock
24	Leader To Punch
25	EOB Pattern
26	Serial Number
27	G76/G77 Shift Dir.
28	Can Cycle Act w/o X/Y
29	G91 Non-modal
30	4th Axis Enable
31	Reset Program Pointer
32	Coolant Override
33	Coordinate System
34	4th Axis Diameter
35	G60 Offset
36	Program Restart
37	RS-232 Data Bits
39	Beep @ M00, M01, M02, M30
40	Tool Offset Measure
41	Add Spaces RS-232 Out
42	M00 After Tool Change

Setting	Description
43	Cutter Comp Type
44	Min F in Radius CC %
45	Mirror Image X-Axis
46	Mirror Image Y-Axis
47	Mirror Image Z-Axis
48	Mirror Image A-Axis
49	Skip Same Tool Change
52	G83 Retract Above R
53	Jog w/o Zero Return
55	Enable DNC from MDI
56	M30 Restore Default G
57	Exact Stop Canned X-Y
58	Cutter Compensation
59	Probe Offset X+
60	Probe Offset X,
61	Probe Offset Z+
62	Probe Offset Z
63	Tool Probe Width
64	Tool Offset Measure Uses
65	Graph Scale (Height)
66	Graphics X Offset
67	Graphics Y Offset
68	Graphics Z Offset

Introduction

Setting	Description
69	DPRNT Leading Spaces
70	DPRNT Open/CLOS DCode
71	Default G51 Scaling
72	Default G68 Rotation
73	G68 Incremental Angle
74	9xxx Progs Trace
75	9xxxx Progs Singls BLK
76	Tool Release Lock Out
77	Scale Integer F
78	5th axis Enable
79	5th-axis Diameter
80	Mirror Image B-Axis
81	Tool At Power Up
82	Language
83	M30/Resets Overrides
84	Tool Overload Action
85	Maximum Corner Rounding
86	M39 Lockout
87	M06 Resets Override
88	Reset Resets Overrides
90	Max Tools To Display
100	Screen Saver Delay
101	Feed Overide- > Rapid

Setting	Description
103	CYC START/FH Same Key
104	Jog Handle to SNGL BLK
108	Quick Rotary G28
109	Warm-Up Time in MIN.
110	Warmup X Distance
111	Warmup Y Distance
112	Warmup Z Distance
114	Conveyor Cycle Time (minutes)
115	Conveyor On-Time (minutes)
116	Pivot Length
117	G143 Global Offset
118	M99 Bumps M30 CNTRS
119	Offset Lock
120	Macro Var Lock
130	Tap Retract Speed
131	Auto Door
133	REPT Rigid Tap
142	Offset Chng Tolerance
143	Machine Data Collect
144	Feed Override->Spindles
155	Load Pocket Tables
156	Save Offset with PROG
157	Offset Format Type

Introduction

Setting	Description
158	X Screw Thermal COMP%
159	Y Screw Thermal COMP%
160	Z Screw Thermal COMP%
162	Default To Float
163	Disable .1 Jog Rate
164	Rotary Increment
167-186	Periodic Maintenance
187	Machine Data Echo
188	G51 X SCALE
189	G51 Y SCALE
190	G51 Z SCALE
191	Default Smoothness
196	Conveyor Shutoff
197	Coolant Shutoff
198	Background Color
199	Display Off Timer (Minutes)
201	Show Only Work and Tool Offsets In Use
216	Servo and Hydraulic Shutoff
238	High Intensity Light Timer (minutes)
239	Worklight Off Timer (minutes)
240	Tool Life Warning
242	Air Water Purge Interval (minutes)
243	Air Water Purge On-Time (seconds)

Setting	Description
244	Master Gage Tool Length (inches)
245	Hazardous Vibration Sensitivity
247	Simultaneous XYZ Motion Tool Change
249	Enable Haas Startup Screen
900	CNC Network Name
901	Obtain Address Automatically
902	IP Address
903	Subnet Mask
904	Default Gateway
905	DNS Server
906	Domain/Workgroup Name
907	Remote Server Name
908	Remote Share Path
909	User Name
910	Password
911	Access to CNC Share (Off, Read, Full)
912	Floppy Tab Enabled
913	Hard Drive Tab Enabled
914	USB Tab Enabled
915	Net Share
916	Second USB Tab Enabled

Introduction to Settings

The setting pages contain values that control machine operation and that you may need to change.

Settings are presented in tabbed menus. For information on navigating tabbed menus in the Haas control, refer to page 53. The on-screen settings are organized into groups.

Use the **[UP]** and **[DOWN]** cursor arrow keys to highlight a setting. To quickly access a setting, with the Settings display active on the screen, type the setting number and press the **[DOWN]** cursor arrow.

Some settings have numerical values that fit in a given range. To change the value of these settings, type the new value and press **[ENTER]**. Other settings have specific available values that you select from a list. For these settings, use the **[LEFT]** and **[RIGHT]** cursor arrow keys to display the choices. Press **[ENTER]** to change the value. The message near the top of the screen tells you how to change the selected setting.

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 remains idle until it is powered down. The machine does not power down while a program is running, and the time (number of minutes) starts back at zero anytime a button is pressed or the **[HANDLE JOG]** control is used. The auto-off sequence gives the operator a 15-second warning before power down, at which time pressing any button stops the power down.

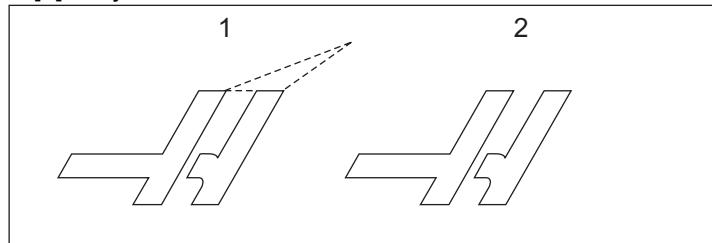
2 - Power Off at M30

If this setting is set to **ON**, the machine powers down at the end of a program (**M30**). The machine gives the operator a 15-second warning once an **M30** is reached. Press any key to interrupt the power-off sequence.

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.

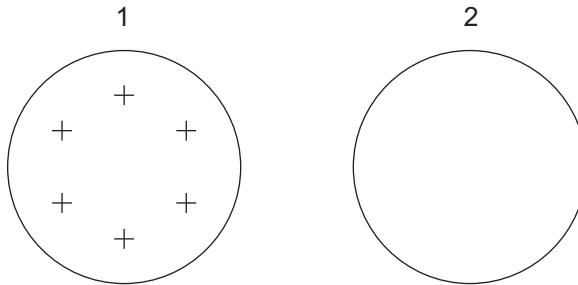
- F9.1:** Setting 4 -Graphics Rapid Path:[1] All Rapid Tool Motions Shown with a Dashed Line When **ON**. [2] Only Cut Lines Shown When **OFF**.



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.

- F9.2:** Setting 5, Graphics Drill Point: [1] X Mark Displays When **ON**. [2] No X Marks Shown When **OFF**.



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** stops the parameters from being changed, except for parameters 81-100.



NOTE:

*Each time the control is powered up, this setting is set to **ON**.*

8 - Prog Memory Lock

This setting locks out the memory editing functions (**[ALTER]**, **[INSERT]**, etc.) when it is set to **ON**. This also locks out MDI. Editing functions are not restricted by this setting.

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 **MM**, programmed units are millimeters, to 0.001 mm. All offset values are converted when this setting is changed from inches to metric, or vice versa. However, changing this setting does 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 **MM**, the default G-code is **G21**.

	Inch	Metric
Feed	in/min	mm/min
Max Travel	Varies by axis and model	
Minimum programmable dimension	.0001	.001

Axis jog key	Inch	Metric
.0001	.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	.1 in/jog click	1 mm/jog click

10 - Limit Rapid at 50%

Turning this setting **ON** limits 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 is limited to 350 ipm when this setting is **ON**. The control displays 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 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 RS-232 Serial Port. 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 you know what your system needs, for example, **XMODEM** must use 8 data bits and no parity (set to **NONE**). This setting must match the parity from the personal computer.

13 - Stop Bit

This setting designates the number of stop bits for the RS-232 Serial Port. It can be 1 or 2. This setting must match the number of stop bits from the personal computer.

14 - Synchronization

This setting changes the synchronization protocol between sender and receiver for the RS-232 Serial Port. This setting must match the synchronization protocol 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 and T Code Agreement

Turning this setting **ON** has the machine check to ensure that the **H** offset code matches the tool in the spindle. This check can help to prevent crashes.



NOTE:

*This setting does not generate an alarm with an **H00**. **H00** is used to cancel the tool length offset.*

16 - Dry Run Lock Out

The Dry Run feature is not available when this setting is turned **ON**.

17 - Opt Stop Lock Out

The Optional Stop feature is not available when this setting is **ON**.

18 - Block Delete Lock Out

The Block Delete feature is not available when this setting is **ON**.

19 - Feedrate Override Lock

The feedrate override buttons are disabled when this setting is turned **ON**.

20 - Spindle Override Lock

The spindle speed override keys are disabled when this setting is turned ON.

21 - Rapid Override Lock

The axis rapid override keys are disabled when this setting is turned ON.

22 - Can Cycle Delta Z

This setting specifies the distance to retract the Z Axis to clear chips during a G73 canned cycle. The range is 0.0000 to 29.9999 inches (0-760 mm).

23 - 9xxx Progs Edit Lock

Turning this setting ON prevents the 9000 series of programs from being viewed in memory, edited, or deleted. 9000 series programs cannot be uploaded or downloaded while this setting is ON.



NOTE:

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 RS-232 Serial Port.

25 - EOB Pattern

This setting controls the EOB (End of Block) pattern when data is sent and received to/from the serial port (RS-232). This setting must match the EOB pattern from the personal computer. The choices are CR LF, LF ONLY, LF CR, and CR ONLY.

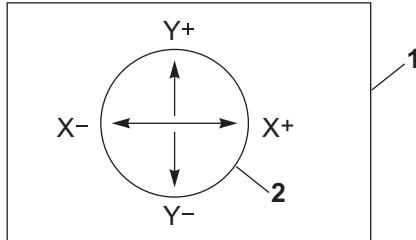
26 - Serial Number

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 page [277](#).

- F9.3:** Setting 27, Direction the Tool is Shifted to Clear Boring Tool: [1] Part, [2] Bored hole.



28 - Can Cycle Act w/o X/Y

This is an **ON/OFF** setting. The preferred setting is **ON**.

When it is **OFF**, the initial canned cycle definition block requires an X or Y code for the canned cycle to be executed.

When it is **ON**, the initial canned cycle definition block causes one cycle to be executed even when there is no X or Y code in the block.



NOTE:

When an I0 is in that block, it will not execute the canned cycle on the definition line. This setting has no effect on G72 cycles.

29 - G91 Non-modal

Turning this setting **ON** uses the G91 command only in the program block it is in (non-modal). When it is **OFF**, and a G91 is commanded, the machine uses incremental moves for all axis positions.



NOTE:

This setting must be OFF for G47 engraving cycles.

30 - 4th Axis Enable

This setting initializes the control for a specific 4th axis. For details on changing this Setting, see the 4th and 5th Axis Programming section of this manual. 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:

*Selections: **USER1** and **USER2** can be used to set-up a unique rotary table.*

31 - Reset Program Pointer

When this setting is **OFF**, **[RESET]** does not change the position of the program pointer. When it is **ON**, pressing **[RESET]** moves 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 gives the message **FUNCTION LOCKED** if an attempt is made to turn the coolant on manually or from a program. The **IGNORE** selection ignores 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 are 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 clears out the value in G52.

Set to **HAAS** with G52:

Any values in the G52 register are 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 enters the current machine location as a new zero point (Work Zero Offset), and that location is entered into and viewed in the G52 list.

Set to **FANUC** or **HAAS** with G92:

Selecting **FANUC** or **HAAS** with a G92, works like the **YASNAC** setting, except that the new Work Zero location value is loaded as a new G92. This new value in the G92 list is 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 (0.0000 to 50.0000 inches), which the control uses to determine the angular feedrate. The feedrate in a program is always inches or millimeters per minute (G94); therefore, the control must know the diameter of the part being machined in the A Axis in order to compute angular feedrate. Refer to Setting 79 on page 373 for information on the 5th axis diameter setting.

35 - G60 Offset

This is a numeric entry in the range 0.0000 to 0.9999 inches. It is used to specify the distance an axis travels 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 directs the control to scan the entire program to make sure 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.



NOTE:

The machine goes to the position and changes to the tool specified in the block before the cursor position first. For example, if the cursor is on a tool change block in the program, the machine changes to the tool loaded before that block, then it changes to the tool specified in the block at the cursor location.

The control processes these M-codes when Setting 36 is enabled:

M08 Coolant On

M09 Coolant Off

M41 Low Gear

M42 High Gear

M51-M58 Set User M

M61-M68 Clear User M

When Setting 36 is **OFF** the control starts the program, but it does not check 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 the Serial Port (RS-232). This setting must match the data bits 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.

39 - Beep @ M00, M01, M02, M30

Turning this setting **ON** causes the keyboard beeper to sound when an **M00**, **M01** (with Optional Stop active), **M02**, or an **M30** is found. The beeper continues 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**. The selection also effects the Tool Diameter geometry and wear values displayed on the **TOOL OFFSETS** table. If setting 40 is changed from **RADIUS** to **DIAMETER**, the displayed value is twice the value entered before.

41 - Add Spaces RS-232 Out

When this setting is **ON**, spaces are added between address codes when a program is sent out via the RS-232 serial port. 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** stops the program after a tool change and a message is displayed stating this. **[CYCLE START]** 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 on page **149**.

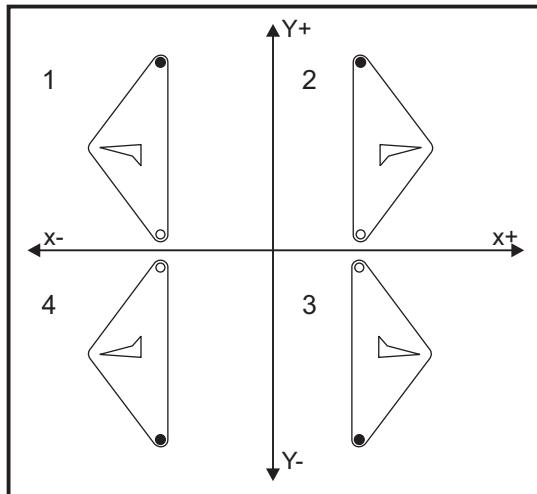
44 - Min F in Radius CC %

Minimum feedrate in radius cutter compensation percent setting affects the feed rate when cutter compensation moves the tool towards the inside of a circular cut. This type of cut slows 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, 46, 47 - Mirror Image X, Y, Z Axis

When one or more of these settings is **ON**, axis motion is mirrored (reversed) around the work zero point. See also G101, Enable Mirror Image.

- F9.4:** No Mirror Image [1], Setting 45 **ON** - X Mirror [2], Setting 46 **ON** - Y Mirror [4], Setting 45 and Setting 46 **ON** - XY Mirror [3]



48 - Mirror Image A Axis

This is an **ON/OFF** setting. When it is **OFF**, axis motions occur normally. When it is **ON**, A-Axis motion may be mirrored (or reversed) around the work zero point. Also, see G101 and Settings 45, 46, 47, 80, and 250.

49 - Skip Same Tool Change

In a program, the same tool may be called in the next section of a program or a subprogram. The control does two tool changes and finishes with the same tool in the spindle. Turning this setting **ON** skips same-tool changes; a tool change only occurs if a different tool is placed in the spindle.



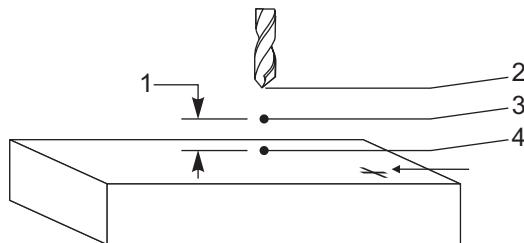
NOTE:

This setting only affects machines with carousel (umbrella) tool changers.

52 - G83 Retract Above R

Range is 0.0000 to 30.0000 inches (0-761mm). This setting changes the way G83 (peck drilling cycle) behaves. 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 drills through this empty distance. If Setting 52 is set to the distance required to clear chips, the R plane can be set much closer to the part being drilled.

- F9.5:** Setting 52, Drill Retract Distance: [1] Setting 52, [2] Start Position, [3] Retract Distance Set by Setting 52, [4] R Plane



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 possibly damage the machine. When the control is powered up, this setting automatically returns to **OFF**.

55 - Enable DNC from MDI

Turning this setting **ON** makes the DNC feature available. DNC is selected in the control by pressing **[MDI/DNC]** twice.

The DNC Direct Numerical Control feature is not available when Setting 55 is 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-Y

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** makes the mill 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 YASNAC). See the Cutter Compensation section on page 149.

59, 60, 61, 62 - Probe Offset X+, X-, Y+, Y-

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, equal to the radius of the probe stylus tip.

You can use macros to access these settings; for more information, refer to the Macro section of this manual (starting on page 178).



NOTE:

These settings are not used with the Renishaw WIPS option.

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. This value is equal to the diameter of the tool probe stylus.

64 - T. Ofs Meas Uses Work

The (Tool Offset Measure Uses Work) setting changes the way the [TOOL OFFSET MEASURE] key works. When this is ON, the entered tool offset is 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. Use this formula to set a specific scale:

$$\text{Total Y travel} = \text{Parameter 20}/\text{Parameter 19}$$

$$\text{Scale} = \text{Total Y travel}/\text{Setting 65}$$

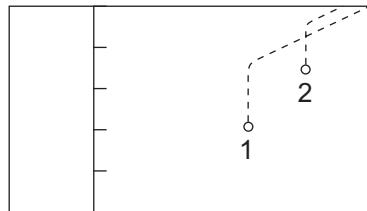
66 - Graphics X Offset

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.

- F9.6:** Setting 67, Graphics Y Offset: [1] Setting 66 and 67 set to 0, [2] Setting 66 and 67 set to 2.0



68 - Graphics Z Offset

Reserved for future use.

69 - DPRNT Leading Spaces

This is an **ON/OFF** setting. When set to **OFF**, the control does not use leading spaces generated by a macro DPRNT format statement. Conversely, when set to **ON**, the control uses leading spaces. This example illustrates control behavior when this setting is **OFF** or **ON**.

```
%  
#1 = 3.0 ;  
G0 G90 X#1 ;  
DPRNT[X#1[44]] ;  
%
```

OUTPUT

OFF	ON
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 **POOPEN** and **PCLOS** statements in macros send DC control codes to the serial port. When the setting is **ON**, these statements send DC control codes. When it is **OFF**, the control codes are suppressed. The 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), the value specified in the R address is added to the previous rotation angle. For example, an R value of 10 causes the feature rotation to be 10 degrees the first time commanded, 20 degrees the next time, etc.

**NOTE:**

This setting must be OFF when you command an engraving cycle (G47).

74 - 9xxx Progs Trace

This setting, along with Setting 75, is useful for debugging CNC programs. When Setting 74 is ON, the control displays the code in the macro programs (09xxxx). When the setting is OFF, the control does not display the 9000 series code.

75 - 9xxxx Progs Single BLK

When Setting 75 is ON and the control is operating in Single Block mode, then the control stops at each block of code in a macro program (09xxxx) and waits for the operator to press [CYCLE START]. When Setting 75 is OFF the macro program is run continuously, the control does 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 executes 9000 series programs without displaying the program code. If the control is in Single-Block mode, no single-block pause occurs during the running of the 9000 series program.

When Setting 75 is ON and Setting 74 is OFF, 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 you always use a decimal point.) This setting helps operators run programs developed on a control other than Haas. For example **F12** becomes:

- 0.0012 units/minute with Setting 77 **OFF**
- 12.0 units/minute with Setting 77 **ON**

There are 5 feedrate settings. This chart shows the effect of each setting on a given F10 address.

INCH		MILLIMETER	
DEFAULT	(.0001)	DEFAULT	(.001)
INTEGER	F1 = F1	INTEGER	F1 = F1
.1	F10 = F1.	.1	F10 = F1.
.01	F10 = F.1	.01	F10 = F.1
.001	F10 = F.01	.001	F10 = F.01
.0001	F10 = F.001	.0001	F10 = F.001

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:

*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 5th axis (0.0 to 50 inches), that the control uses to determine the angular feedrate. The feedrate in a program is always inches or millimeters per minute; therefore, the control must know the diameter of the part being machined in the 5th-axis in order to compute angular feedrate. Refer to Setting 34 on page 365 for more information on the 4th axis diameter setting.

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 G101 and Settings 45, 46, 47, 48, and 250.

81 - Tool At Power Up

When **[POWER UP/RESTART]** is pressed, the control changes 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, causes one of these actions to occur after you press **[POWER UP/RESTART]**:

- If Setting 81 is set to zero, the carousel rotates to pocket #1. No tool change is performed.
- If Setting 81 contains the tool #1, and the tool currently in the spindle is tool #1, and **[ZERO RETURN]** then **[ALL]** are pressed, the carousel remains at the same pocket and no tool change is performed.
- If Setting 81 contains the tool number of a tool not currently in the spindle, the carousel rotates to pocket #1 and then to the pocket containing the tool specified by Setting 81. A tool change is 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 with the **[LEFT]** and **[RIGHT]** cursor arrows, then 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

When a tool becomes overloaded, Setting 84 designates the control response. These settings cause specified actions:

- **ALARM** causes the machine to stop.
- **FEEDHOLD** displays the message *Tool Overload* and the machine stops in a feedhold situation. Press any key to clear the message.

- **BEEP** causes an audible noise (beep) from the control.
- **AUTOFEED** causes the control to automatically limit the feedrate based on the tool load.



NOTE:

When tapping (rigid or floating), the feed and spindle overrides are locked out, so the AUTOFEED setting is ineffective (the control appears to respond to the override buttons, by displaying the override messages).



CAUTION:

Do not use the AUTOFEED setting when thread milling or auto reversing tapping heads, as it can cause unpredictable results or even a crash.

The last commanded feedrate is restored at the end of program execution, or when the operator presses **[RESET]** or turns **OFF** the **AUTOFEED** setting. The operator can use **[FEEDRATE OVERRIDE]** while the **AUTOFEED** setting is selected. These keys are recognized by the **AUTOFEED** setting 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 ignores **[FEEDRATE OVERRIDE]**.

85 - Maximum Corner Rounding

This setting defines the machining accuracy tolerance around corners. The initial default value is 0.0250". This means that the control keeps the radii of corners no bigger than 0.0250".

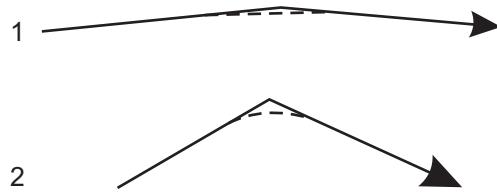
Setting 85 causes the control to adjust feeds around corners in all 3 axes to meet the tolerance value. The lower the value of Setting 85, the slower the control feeds around corners to meet the tolerance. The higher the value of Setting 85, the faster the control feeds around corners, up to the commanded feedrate, but it could round the corner off to a radius up to the tolerance value.



NOTE:

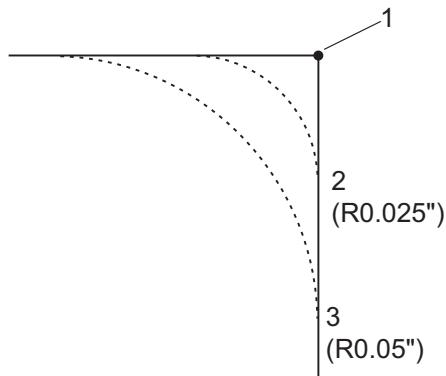
The angle of the corner also affects the change to the feedrate. The control can cut shallow corners within tolerance at a higher feedrate than it can with tighter corners.

- F9.7:** The control can cut corner [1] within tolerance at a higher feedrate than it can cut corner [2].



If Setting 85 has a value of zero, the control acts as if exact stop is active in each motion block.

- F9.8:** Assume that the commanded feedrate is too high to achieve corner [1]. If Setting 85 has a value of 0.025, then the control slows the feedrate enough to achieve corner [2] (with a radius of 0.025"). If Setting 85 has a value of 0.05, then the control slows the feedrate enough to achieve corner [3]. The feedrate to achieve corner [3] is faster than the feedrate to achieve corner [2].



86 - M39 (Rotate Tool Turret) Lockout

When this setting is **ON**, the control ignores M39 commands.

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 **[RESET]** is pressed, any overrides are canceled and set to their programmed values or defaults (100%).

90 - Max Tools To Display

This setting limits the number of tools displayed on the Tool Offsets screen. The range of this setting is 6 to 200.

100 - Screen Saver Delay

When this setting has a value of zero, the screen saver is disabled. A nonzero value specifies the number of minutes before the screen saver starts. Press [CANCEL] to exit the screen saver. The screen saver does not start if the control is in Sleep, Jog, Edit, or Graphics mode.

101 - Feed Override -> Rapid

Pressing [HANDLE CONTROL FEED], with this setting **ON**, will cause the jog handle to affect both the feedrate and the rapid rate overrides. Setting 10 affects the maximum rapid rate. The rapid rate cannot exceed 100%. Also, [+10% FEEDRATE], [-10% FEEDRATE], and [100% FEEDRATE] change the rapid and feed rate together.

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 automatically turns off.

104 - Jog Handle to SNGL BLK

The [HANDLE JOG] control can single-step through a program when this setting is **ON**. Reversing the [HANDLE JOG] control direction generates a feed hold.

This setting cannot be turned on while Setting 103 is **ON**. When one of them is set to **ON**, the other automatically turns off.

108 - Quick Rotary G28

If this setting is **ON**, the control returns the rotary axes to zero in +/-359.99 degrees or less.

For example, if the rotary unit is at +/-950.000 degrees and a zero return is commanded, the rotary table rotates +/-230.000 degrees to the home position if this setting is **ON**.



NOTE:

The rotary axis returns to the machine home position, not the active work coordinate position.

To use Setting 108, Parameter 43:10 (for the A Axis) and Parameter 151:10 (for the B Axis) must be set to 1. If these parameter bits are not set to 1 the control ignores 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 control gives this warning:

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 is 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 warm-up time is displayed on the bottom right hand corner of the Diagnostics Inputs 2 screen using the standard hh:mm:ss format.

110, 111, 112 - Warmup X, Y, 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 effect.

114 - Conveyor Cycle (minutes)

Setting 114 Conveyor Cycle Time is the interval that the conveyor turns on automatically. For example, if setting 114 is set to 30, the chip conveyor turns on every half an hour.

On-time should be set no greater than 80% of cycle time. Refer to Setting 115 on page 371.

NOTE:

The [CHIP FWD] button (or M31) starts the conveyor in the forward direction and starts the cycle.

The [CHIP STOP] button (or M33) stops the conveyor and cancels the cycle.

115 - Conveyor On-time (minutes)

Setting 115 Conveyor On-Time is the amount of time the conveyor runs. For example, if setting 115 is set to 2, the chip conveyor runs for 2 minutes, then turns off.

On-time should be set no greater than 80% of cycle time. Refer to Setting 114 Cycle Time on page **377**.

NOTE: *The [CHIP FWD] button (or M31) starts the conveyor in the forward direction and starts the cycle.*

The [CHIP STOP] button (or M33) stops the conveyor and cancels the cycle.

116 - Pivot Length (VR Models Only)

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 (VR Models Only)

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 is entered into this setting, and it is applied to the G143 tool length compensation.

118 - M99 Bumps M30 CNTRS

When this setting is **ON**, an M99 adds one to the M30 counters (these are visible after pressing **[CURRENT COMMANDS]**).



NOTE: *M99 only increases the counters as it occurs in a main program, not in a sub-program.*

119 - Offset Lock

Turning the setting **ON** does not allow the values in the Offset display to be altered. However, programs that alter offsets with macros or G10 are permitted to do so.

120 - Macro Var Lock

Turning this setting **ON** does not allow the macro variables to be altered. However, programs that alter macro variables can do so.

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, commands the mill to retract the tap twice as fast as it went in. If the value is 3, it retracts three times as fast. A value of 0 or 1 has no effect on the retract speed (Range 0-9, but the recommended range is 0-4).

Entering a value of 2 is the equivalent of using a J address code value of 2 for G84 (tapping canned cycle). However, specifying a J code for a rigid tap overrides Setting 130.

131 - Auto Door

This setting supports the Auto Door option. Set it **ON** for machines with an autodoor. Refer to M80 / M81 (Auto Door Open / close M-codes) on page 343.

**NOTE:**

The M-codes work only while the machine receives a cell-safe signal from a robot. For more information, contact a robot integrator.

The door closes when [CYCLE START] is pressed and opens 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 (Repeat Rigid Tap) ensures that the spindle is oriented during tapping so that the threads line up when a second tapping pass is programmed in the same hole.

**NOTE:**

*This setting must be **ON** when a program commands peck tapping.*

142 - Offset Chng Tolerance

This setting generates a warning message if an offset is changed by more than the amount entered for this setting. If an attempt is made to change an offset by more than the entered amount (either positive or negative), the control gives this prompt: *XX changes the offset by more than Setting 142! Accept (Y/N)?*

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. For detailed information, refer to the Machine Data Collection section on page 82.

144 - Feed Override->Spindle

This setting is intended to keep the chip load constant when an override is applied. When this setting is **ON**, any feedrate override is also applied to the spindle speed, and the spindle overrides are disabled.

155 - Load Pocket Tables

This setting is 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 hardware device, the contents of the **Pocket Tool** table is unaltered. Setting 155 automatically defaults to **OFF** when the machine is turned on.

156 - Save Offset with PROG

When this setting is **ON**, the control includes the offsets in the program file when you save it. The offsets appear in the file before the final % sign, under the heading 0999999.

When you load the program back into memory, the control prompts *Load Offsets (Y/N?)*. Press **Y** if you want to load the saved offsets. Press **N** if you do not want to load them.

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 edited on a PC and later reloaded.

When it is set to **B**, each offset is saved on a separate line with an **x** value and a **v** value.

158,159,160 - X, Y, Z Screw Thermal COMP%

These settings can be set from -30 to +30 and adjust the existing screw thermal compensation by -30% to +30% accordingly.

162 - Default To Float

When this setting is **ON**, the control adds a decimal point to values entered without a decimal point for certain address codes. When the setting is **OFF**, values given after address codes that do not include decimal points are taken as machinist's notation; for example, thousandths or ten-thousandths. The feature applies to these address codes: X, Y, Z, A, B, C, E, F, I, J, K, U, and W.

	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:

This setting affects the interpretation of all programs entered either manually or from a hardware device. 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 **[PALLET ROTATE]** button on the EC-300 and EC-1600. It specifies the rotation for the rotary table in the load station. It should be set to a value from 0 to 360. The default value is 90. For example, entering 90 rotates the pallet 90 degrees each time the rotary index button is pressed. If it is set to zero, the rotary table does not rotate.

187 - Machine Data Echo

When this setting is **ON**, the data collection Q commands issued from the user's PC are shown on the PC screen. When this setting is **OFF**, the PC screen does not show these commands.

188, 189, 190 - G51 X, Y, Z SCALE

You can scale the axes individually with these settings (the value must be a positive number).

Setting 188 = G51 X SCALE

Setting 189 = G51 Y SCALE

Setting 190 = G51 Z SCALE

If setting 71 has a value, then the control ignores Settings 188 - 190, and it uses the value in setting 71 for scaling. If the value for setting 71 is zero, then the control uses Settings 188 - 190.



NOTE:

When settings 188-190 are in effect, only linear interpolation, G01, is allowed. If G02 or G03 is used, alarm 467 is generated.

191 - Default Smoothness

This setting is 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 Shutoff

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 Shutoff

This setting is the amount of time to wait without activity before Coolant flow stops. Units are minutes.

198 - Background Color

Specifies the background color for inactive display panes. Range is 0 to 254. The default value is 235.

199 - Backlight Timer

This setting is the time in minutes after which the machine display backlight turns 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 ([CANCEL] is preferred).

201 - Show Only Work and Tool Offsets In Use

When this setting is on only the Work and Tool offsets used by the running program display. The program must run in the graphics mode first to turn on this feature.

216 - Servo and Hydraulic Shutoff

This setting turns off the servomotors and hydraulic pump, if equipped, after the specified number of seconds 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 when activated. The light turns on when the door is opened and the work light switch is on. If this value is zero, then the light will remain turned on while the doors are open.

239 - Worklight Off Timer (minutes)

Specifies the amount of time in minutes after which the work light will turn off automatically if there are no key presses or [HANDLE JOG] changes. If a program is running when the light turns off, the program will continue running.

240 - Tool Life Warning

The percentage of remaining tool life at which to trigger a tool life warning. Tools with remaining life below Setting 240 appear with an orange background in the ATM screen, and the beacon light flashes yellow.

242 - Air Water Purge Interval (minutes)

This setting specifies the interval for the purge of condensates in the system air reservoir. When the time specified by setting 242 lapses, starting from midnight, the purge begins.

243 - Air Water Purge On-Time (seconds)

This setting specifies the duration of the purge of condensates in the system air reservoir. The units are seconds. When the time specified by Setting 242 lapses, starting from midnight, the purge is begun for the number of seconds specified by Setting 243.

244 - Master Gage Tool Length (inches)

This setting specifies the length of the master gage used to locate the tool touch-off surface during setup. It is the length from the base to the tip of the master gage. It can generally be measured on a tool pre-setter gage.

245 - Hazardous Vibration Sensitivity

This setting selects from three sensitivity levels (**LOW**, **MEDIUM**, or **HIGH**) for the hazardous vibration sensor (if installed). This setting defaults to **HIGH** each time the machine is powered up.

247 - Simultaneous XYZ Motion in Tool Change

Setting 247 defines how the axes move during a tool change. If Setting 247 is **OFF**, the Z Axis retracts first, followed by X- and Y-Axis motion. This feature can be useful in avoiding tool collisions for some fixture configurations. If Setting 247 is **ON**, the axes move simultaneously. This may cause collisions between the tool and the workpiece, due to B- and C-Axis rotations. It is strongly recommended that this setting remain **OFF** on the UMC-750, due to the high potential for collisions.

249 - Enable Haas Startup Screen

If this setting is **ON**, the screen shows startup instructions each time the machine is powered on. You can turn Setting 249 **ON** or **OFF** through the settings page, or you can press **[F1]** at the startup screen to turn it off.

250 - Mirror Image C-Axis

This is an **ON/OFF** setting. When it is **OFF**, axis motions occur normally. When it is **ON**, C-Axis motion may be mirrored (or reversed) around the work zero point. Also, see G101 and Settings 45, 46, 47, 48, and 80.

900 - CNC Network Name

This setting contains 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 has *** entered.



NOTE:

*The ADMIN section at the end provides 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:

1. At the control, press **[LIST PROGRAM]**.
2. Press **[CANCEL]**.
3. Press the right arrow for the Hard Drive directory and press **[ENTER]**.
4. Type in **ADMIN** and press **[INSERT]**.
5. Select **ADMIN** folder and press **[ENTER]**.

6. Copy the ipconfig.txt file to disk or USB and read it on a Windows computer.

902 - IP Address

This setting is needed on a network with static TCP/IP addresses (DHCP Off). The network administrator assigns 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

This setting is needed on a network with static TCP/IP addresses. The network administrator assigns a mask value. The machine must be turned off and back on for changes to this setting to take effect.

904 - Default Gateway

This setting is needed to gain access through routers. The network administrator assigns an address. The machine must be turned off and back on for changes to this setting to take effect.

905 - DNS Server

This setting contains 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

This setting is the CNC control workgroup or domain. 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, this setting contains the NETBIOS name from the computer where the share folder resides. IP address is not supported.

908 - Remote Share Path

This setting contains the name of the shared network folder. To rename the shared folder after a host name is selected, enter the new shared folder name and press [ENTER].



NOTE:

Do not use spaces in the shared folder name.

909 - User Name

This setting 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 setting 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

This setting is used for the CNC hard drive read/write privileges. **OFF** stops the hard drive from being networked. **FULL** allows read/write access to the drive from the network. Turning off this setting and Setting 913 disables network card communication.

912 - Floppy Tab Enabled

Refer to Setting 914 USB Tab Enabled for this functionality. (Older software used this setting to turn off/on access to the USB floppy drive. When set to **OFF**, the USB floppy drive is not accessible.)

913 - Hard Drive Tab Enabled

This setting turns off/on access to the hard drive. When set to **OFF**, the hard drive is not accessible. Turning off this setting and CNC Share (Setting 911) disables network card communication.

914 - USB Tab Enabled

This setting turns off/on access to the USB port. When set to **OFF**, the USB port is not accessible.

915 - Net Share

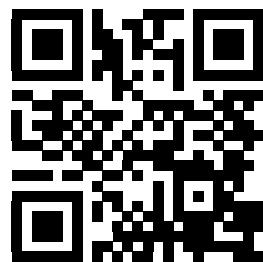
This setting turns off/on access to the server drive. When set to **OFF** access to the sever from the CNC control is not possible.

916 - Second USB Tab Enabled

This setting turns off/on access to the secondary USB port. When set to **OFF** the USB port will not be accessible.

9.2 More Information Online

For updated and supplemental information, including tips, tricks, maintenance procedures, and more, visit the Haas Resource Center at diy.HaasCNC.com. You can also scan the code below with your mobile device to go directly to the Resource Center:



More Information Online

Chapter 10: Maintenance

10.1 Introduction

Regular maintenance is important to make sure that your machine has a long and productive life with minimal downtime. The most common maintenance tasks are simple and you can do them yourself. You can also ask your HFO about their comprehensive preventive maintenance program for complex maintenance tasks.

10.2 Maintenance Monitor

The Haas control features a maintenance monitor to tell you when you need to do certain maintenance tasks. There are (14) included maintenance items, and (6) spare items that you can designate yourself.

10.2.1 Maintenance Settings

Settings 167-186 control the default maintenance interval for each maintenance item. The Maintenance Monitor page shows only maintenance items that have a default interval (non-zero).

Maintenance intervals have (3) possible unit values:

- On-time (hours): The control counts down this interval while the machine power is on.
- Motion time (hours): The control counts down this interval only while the specified component is in motion.
- Tool changes (each): The control counts this interval down by (1) after each tool change.

You can change each setting to increase or decrease the default interval. At the end of each maintenance interval, the control will show a *MAINTENANCE DUE* message and icon. Go to the maintenance monitor page to see the maintenance required.

Maintenance Monitor

F10.1: Maintenance Settings Tab

MAINT DEFALTS	
167 Coolant Replacement default in power-on hours	1000
168 Control Air Filter Replacement default in power-on hours	0
169 Oil Filter Replacement default in power-on hours	2500
170 Gearbox Oil Replacement default in power-on hours	5000
171 Coolant Tank Level Check default in power-on hours	20
172 Way Lube Level Check default in motion-time hours	250
173 Gearbox Oil Level Check default in power-on hours	250
174 Seals/Wipers Inspection default in motion-time hours	250
175 Air Supply Filter Check default in power-on hours	40
176 Hydraulic Oil Level Check default in power-on hours	100
177 Hydraulic Filter Replacement default in motion_time hours	150
178 Grease Fittings default in motion_time hours	250
179 Grease Chuck default in motion_time hours	0
180 Grease Tool Changer Cams default in tool-changes	1000
181 Spare Maintenance Setting #1 default in power-on hours	0
182 Spare Maintenance Setting #2 default in power-on hours	0
183 Spare Maintenance Setting #3 default in motion-time hours	0
184 Spare Maintenance Setting #4 default in motion-time hours	0
185 Spare Maintenance Setting #5 default in tool-changes	0
186 Spare Maintenance Setting #6 default in tool-changes	0

10.2.2 The Maintenance Monitor Page

To find the Maintenance Monitor Page:

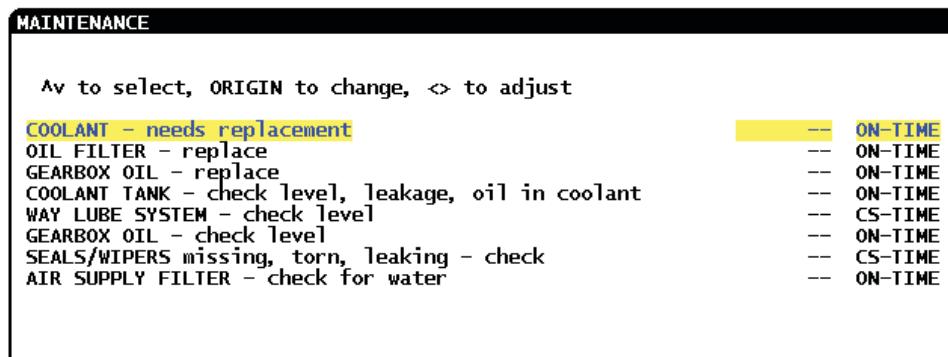
1. Press [CURRENT COMMANDS].
2. Press [PAGE UP] or [PAGE DOWN] until you see the Maintenance page.

F10.2: The Maintenance Page

MAINTENANCE	
Av to select, ORIGIN to change, <> to adjust	
COOLANT - needs replacement	-- ON-TIME
OIL FILTER - replace	-- ON-TIME
GEARBOX OIL - replace	-- ON-TIME
COOLANT TANK - check level, leakage, oil in coolant	-- ON-TIME
WAY LUBE SYSTEM - check level	-- CS-TIME
GEARBOX OIL - check level	-- ON-TIME
SEALS/WIPERS missing, torn, leaking - check	-- CS-TIME
AIR SUPPLY FILTER - check for water	-- ON-TIME

10.2.3 Start, Stop, or Adjust Maintenance Monitoring

To start or stop monitoring on the maintenance page:



1. Use the [UP] or [DOWN] cursor arrow keys to highlight a maintenance item.
Maintenance items that show -- instead of a number are not currently monitored.
2. Press [ORIGIN] to start monitoring the item. The -- changes into the default maintenance interval.
3. To adjust the current interval count, use the [RIGHT] or [LEFT] cursor arrow key.
On-time and motion-time intervals increase or decrease by (1) when you press the [RIGHT] or [LEFT] cursor arrow key. Tool-change intervals increase or decrease by (25).
4. Press [ORIGIN] again to stop monitoring the item. The maintenance interval changes into --.

More Information Online

10.3 More Information Online

For detailed maintenance procedures, drawings of machine components, and other useful information, visit the Haas Automation Resource Center at diy.HaasCNC.com. You can also scan this code with your mobile device to go directly to the maintenance information in the Resource Center.



Chapter 11: Other Equipment

11.1 Introduction

Some Haas machines have unique characteristics which are beyond the scope of this manual to describe. These machines come with a printed manual addendum, but you can also download them at www.haascnc.com.

11.2 Mini Mills

Mini Mills are versatile and compact vertical mills.

11.3 VF-Trunnion Series

These vertical mills come standard with a TR-series rotary unit pre-installed for five-axis applications.

11.4 Gantry Routers

Gantry Routers are large-capacity open-frame vertical mills, suitable for milling and routing applications.

11.5 Office Mill

The Office Mill series are compact small-scale vertical mills that can fit through a standard door frame and run on single-phase power.

11.6 EC-400 Pallet Pool

The EC-400 Pallet Pool increases productivity with a multi-station pallet pool and innovative scheduling software.

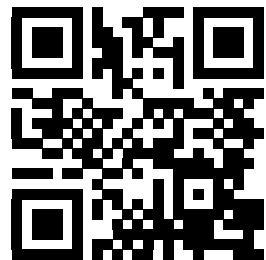
11.7 UMC-750

The UMC-750 is a versatile five-axis mill that features an integrated dual-axis trunnion table.

More Information Online

11.8 More Information Online

For updated and supplemental information, including tips, tricks, maintenance procedures, and more, visit the Haas Resource Center at diy.HaasCNC.com. You can also scan the code below with your mobile device to go directly to the Resource Center:



Index

#

3D cutter compensation (G141)..... 303

A

absolute positioning (G90)
 versus incremental 140

active codes 42

active codes display
 current commands 40

active program 76

active tool display 43

advanced editor 112
 edit menu 115
 modify menu 119
 pop-up menu 113
 program menu 114
 search menu 118
 text selection 116

Advanced Tool Management 41

Advanced Tool Management (ATM) 88
 macros and 92
 tool group setup 91
 tool group usage 91

auto door (option)
 override 25

axis motion
 absolute versus incremental 140
 circular 147
 linear 147

axis overload timer 106

B

background edit 111

beacon light
 status 26

boring and reaming canned cycles 159

BT tooling 87

C

calculator
 circle 58
 circle-circle tangent 60
 circle-line tangent 59
 triangle 57

canned cycles
 boring and reaming 159
 drilling 158
 general information 227
 r plane and 159
 tapping 158

circular interpolation 147

clipboard
 copy to 117
 cut to 117
 paste from 117

communications
 RS-232 82

control display
 active codes 42
 active pane 39
 active tool 43
 basic layout 38
 offsets 40

control pendant 24, 25
 USB port 25

coolant
 operator override 37
 setting 32 and 364

coolant level gauge 43

copying files 76

CT tooling 87

current commands	40
additional setup.....	104
cutter compensation	
circular interpolation and	155
entry and exit.....	152
feed adjustments	153
general description	149
improper application example.....	153
Setting 58 and	149
D	
data collection	82
spare M-codes	84
with RS-232	82
deleting programs	77
device manager.....	74
program selection	76
DIR FULL message.....	77
direct numerical control (DNC).....	85
operating notes	87
display	
graphics	42
settings	42
distance to go position	48
DNC.....	85
DPRNT	
DNC and	87
drilling canned cycles.....	158
drip mode.....	87
dry run.....	106
duplicating a program	78
dxf importer	134
chain and group	135
part origin	135
toolpath selection	136
dynamic work offset (G254).....	329
E	
edit keys	
ALTER	110
DELETE	110
INSERT.....	110
UNDO	110
editing	
highlight code	110

F	
Fanuc	150
Features	
axis overload timer	104
background edit	104
dry run	104
Graphics	104
feed adjustments	
in cutter compensation	153
feed hold	
as override.....	37
file directory system	75
directory creation	75
navigation.....	75
file numerical control (FNC)	85
display footer.....	123
display line numbers	124
display modes	122
FNC editor	121
loading a program	121
menus	122
opening multiple programs	123
file numerical control (FNC) editor	
text selection.....	127
files	
copying	76
folder, See directory structure	
G	
gauges display	
coolant.....	43
G-codes	221
canned cycles	227
cutting.....	147
graphics mode	104
H	
help	
calculator.....	55
drill table	55
keyword search	55
tabbed menu.....	54
help function	53
high-speed SMTC	
heavy tools and	96

I	
icon bar	64
incremental positioning (G91)	
versus absolute.....	140
input bar	49
interpolation motion	
circular.....	147
linear	147
Intuitive Programming System (IPS)	
dxf importer and	135
J	
jog mode.....	101
K	
keyboard	
alpha keys	34
cursor keys	29
display keys	29
function keys	28
jog keys.....	35
key groups	27
mode keys	30
numeric keys	33
override keys.....	36
L	
linear interpolation.....	147
local subprograms (M97)	164
M	
M30 counters	44
machine data	
back up and recover.....	79
backup	79
restore	80
machine position.....	48
machine power-up	73
macro variables	
#3006 programmable stop.....	197
#5021-#5026 current machine coordinate po-	
sition.....	198
#6996-#6999 parameter access	199
#8550-#8567 tooling	202
axis position	198
current commands display	40
macros	
1-bit discrete outputs	194
g- and m-codes	179
look ahead.....	180
M30 counters and	44
round off	179
settings	179
variables	184
main spindle display.....	52
maintenance.....	389
current commands	41
manual data input (MDI).....	111
material	
fire risk.....	3
M-codes	331
coolant commands	146
program stop.....	146
spindle commands	146
memory lock	25
mode display	39
O	
O09xxx program numbers	109
offset	
tool	144
work	144
offsets	
displays	40
operating modes	39
operation	
device manager	74
dry run	106
unattended	3
operator position	48
optional stop.....	335
overrides	37
disabling	37

P		R	
part setup.....	101	r plane.....	159
offsets.....	101	rotary offset	
tool offset.....	103	tilt center	175
work offset	102	RS-232.....	82
position display.....	47	cable length	82
axis selection.....	48	data collection	82
current commands.....	40	DNC and	85
positioning		DNC settings.....	86
absolute vs. incremental	140	running programs	106
positions		run-stop-jog-continue	106
distance to go	48		
machine	48		
operator.....	48		
work (G54).....	48		
program			
active.....	76	safety	
basic search.....	81	decals.....	6
line numbers removal	119	during operation.....	2
program number		electrical	2
change.....	78	introduction.....	1
program numbers		keyswitch operation	4
change in memory	78	maintenance	3
O09xxx.....	109	part loading/unloading	2
Onnnnn format.....	76	robot cells.....	5
program optimizer.....	133	safety decals	
screen.....	134	standard layout.....	6
program selection	76	symbol reference	7
Programming		second home.....	25
canned cycles.....	158	Setting 28	227
programming		Settings.....	351
basic example.....	137	settings	
safe startup line.....	138	list.....	351
subprograms	161	setup mode	3
programs		keyswitch	25
.nc file extension	76	side-mount tool changer (SMTC)	
basic editing	109	door panel	100
deleting	77	extra-large tools	98
duplication	78	moving tools	97
file naming	76	recovery	99
maximum number of	77	zero pocket designation.....	97
running.....	106	special G-codes	
transfer.....	76	engraving	160
		mirror image	160
		pocket milling	160
		rotation and scaling	160
		spindle load meter	52

spindle warm-up	73
subprograms.....	161
external	161
local.....	164
subprograms, See subprograms	
T	
tabbed menus	
basic navigation	53
tapping canned cycles	158
text selection	
advanced editor and.....	116
FNC editor and	127
Through-Spindle Coolant.....	35, 71
drilling cycle and.....	158
M-code.....	344
tilt axis	
center of rotation offset	175
timers and counters display.....	44
tool center point control	329
tool changer.....	93
safety.....	100
tool life display	
current commands	41
tool load limits	104
tool loading	
large / heavy tools	94
tool management tables	
save and restore	93
tool offset	144
tool offsets.....	103
tooling	
pull studs	88
Tnn code	145
tool holder care	87
tool holders	87
U	
umbrella tool changer	
loading	98
recovery	98
unattended operation.....	3
USB device	74

