



Haas Automation, Inc.

Lathe Operator's Manual

96-8900
Revision B
October 2014
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

© 2014 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



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: CNC Lathes (Turning Centers)*

*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 2004/108/EC
- Low Voltage Directive 2006/95/EC
- Additional Standards:
 - EN 60204-1:2006/A1:2009
 - EN 614-1:2006+A1:2009
 - EN 894-1:1997+A1:2008
 - EN 13849-1:2008/AC:2009
 - EN 14121-1:2007

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

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

Person authorized to compile technical file:

Patrick Goris

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 B11.5-1984 (R1994) Lathes*
- *ANSI B11.19-2003 Performance Criteria for Safeguarding*
- *ANSI B11.22-2002 Safety Requirements for Turning Centers and Automatic Numerically Controlled Turning 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.	<i>Service > 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	Introduction	1
1.1.1	Read Before Operating	1
1.2	Environmental and Noise Limits	4
1.3	Unattended Operation	5
1.4	Setup Mode	5
1.4.1	Robot Cells	6
1.4.2	Machine Behavior with the Door Open	6
1.5	Modifications to the Machine	9
1.6	Safety Decals	9
1.6.1	Lathe Warning Decals	10
1.6.2	Other Safety Decals	11
1.7	More Information Online	12
Chapter 2	Introduction	13
2.1	Lathe Orientation	13
2.2	Control Pendant	21
2.2.1	Control Display	22
2.2.2	Keyboard	48
2.2.3	Pendant Front Panel	61
2.2.4	Screen Capture	62
2.2.5	Pendant Right Side, Top, and Bottom Panels	63
2.3	Tabbed Menu Basic Navigation	64
2.4	Help	65
2.4.1	The Help Tabbed Menu	66
2.4.2	Search Tab	66
2.4.3	Help Index	66
2.4.4	Drill Table Tab	67
2.4.5	Calculator Tab	67
2.5	More Information Online	72
Chapter 3	Operation	73
3.1	Machine Power-On	73
3.2	Spindle Warm-Up Program	74
3.3	Device Manager	74
3.3.1	File Directory Systems	75
3.3.2	Program Selection	76
3.3.3	Program Transfer	76
3.3.4	Deleting Programs	77

3.3.5	Maximum Number of Programs	78
3.3.6	File Duplication	78
3.3.7	Changing Program Numbers	79
3.4	Backing Up Your Machine.	79
3.4.1	Making a Backup	80
3.4.2	Restoring From a Backup	81
3.5	Basic Program Search.	82
3.6	RS-232	82
3.6.1	Cable Length	83
3.6.2	Machine Data Collection.	83
3.7	File Numeric Control (FNC)	85
3.8	Direct Numeric Control (DNC).	86
3.8.1	DNC Notes.	87
3.9	Part Setup	87
3.9.1	Chuck Foot Pedal	88
3.9.2	Chuck/Drawtube Warnings	88
3.9.3	Drawtube Operation	90
3.9.4	Chuck and Collet Replacement	91
3.9.5	Steady Rest Foot Pedal	94
3.10	Tailstock Setup and Operation	94
3.10.1	Tailstock Types	95
3.10.2	ST-20/30/40 Tailstock Operation	99
3.10.3	Tailstock Restricted Zone	101
3.10.4	Jogging the Tailstock	103
3.11	Tooling.	103
3.11.1	Jog Mode	103
3.11.2	Setting the Tool Offset.	104
3.11.3	Manually Set the Tool Offset	105
3.11.4	Hybrid Turret, VDI, and BOT Centerline Offset	105
3.11.5	Additional Tooling Set-up	106
3.12	Setting Part (Work Piece) Zero for Z-axis (Part Face).	106
3.13	Features	106
3.13.1	Graphics Mode	107
3.13.2	Dry Run Operation.	107
3.13.3	Running Programs.	108
3.13.4	Background Edit	108
3.13.5	Axis Overload Timer.	108
3.13.6	Screen Capture	109
3.14	Run-Stop-Jog-Continue	109
3.15	Program Optimizer	110
3.15.1	Program Optimizer Operation	110
3.16	Advanced Tool Management	111
3.16.1	Navigation	112

3.16.2	Tool Group Setup	112
3.16.3	Operation	113
3.16.4	Macros	113
3.16.5	Tips & Tricks.	113
3.17	Tool Turret Operations	114
3.17.1	Air Pressure	114
3.17.2	Eccentric Locating Cam Buttons.	114
3.17.3	Protective Cap	115
3.17.4	Tool Load or Tool Change.	116
3.18	Tool Nose Compensation	116
3.18.1	Programming	117
3.18.2	Tool Nose Compensation Concept	118
3.18.3	Using Tool Nose Compensation.	119
3.18.4	Approach and Departure Moves For TNC.	120
3.18.5	Tool Nose Radius and Wear Offset	121
3.18.6	Tool Nose Compensation and Tool Length Geometry. .	123
3.18.7	Tool Nose Compensation in Canned Cycles	123
3.18.8	Example Programs Using Tool Nose Compensation .	123
3.18.9	Imaginary Tool Tip and Direction	131
3.18.10	Programming Without Tool Nose Compensation . .	133
3.18.11	Manually Calculating Compensation.	133
3.18.12	Tool Nose Compensation Geometry.	134
3.19	More Information Online.	146
Chapter 4	Programming	147
4.1	Numbered Programs	147
4.2	Program Editors	148
4.2.1	Basic Program Editing	148
4.2.2	The FNC Editor	149
4.2.3	The Advanced Editor Pop-up Menu	162
4.2.4	Manual Data Input (MDI).	170
4.2.5	Advanced Editor	170
4.2.6	Background Edit	171
4.3	Tips and Tricks	172
4.3.1	Programming	172
4.3.2	Offsets	174
4.3.3	Settings and Parameters	174
4.3.4	Operation	175
4.3.5	Calculator	176
4.4	DXF File Importer	176
4.5	Basic Programming	179
4.5.1	Preparation	180
4.5.2	Cutting	181

4.5.3	Completion	181
4.5.4	Absolute vs. Incremental (XYZ vs. UVW)	181
4.6	Tool Functions	182
4.6.1	FANUC Coordinate System	182
4.6.2	YASNAC Coordinate System	182
4.6.3	Tool Offsets Applied by T101, FANUC vs YASNAC	183
4.7	Coordinate Systems	183
4.7.1	Effective Coordinate System	184
4.7.2	Automatic Setting of Tool Offsets	185
4.7.3	Global Coordinate System (G50)	185
4.8	Live Image	186
4.8.1	Live Image Stock Setup	186
4.8.2	Program Example	187
4.8.3	Live Image Tool Setup	188
4.8.4	Tailstock Setup (Live Image)	191
4.8.5	Operation	193
4.8.6	Run Part	194
4.8.7	Flipping a Part	196
4.9	Tailstock Setup and Operation	197
4.9.1	M-code Programming	197
4.10	Visual Quick Code	197
4.10.1	Selecting a Category	198
4.10.2	Selecting a Part Template	198
4.10.3	Entering the Data	198
4.11	Subroutines	199
4.12	More Information Online	199
Chapter 5	Options Programming	201
5.1	Options Programming	201
5.2	Macros (Optional)	201
5.2.1	Introduction	201
5.2.2	Operation Notes	204
5.2.3	System Variables In-Depth	215
5.2.4	Address Substitution	224
5.2.5	FANUC-Style Macro Features not Included	240
5.2.6	Example Program Using Macros	241
5.3	Live Tooling and C Axis	242
5.3.1	Live Tooling Introduction	242
5.3.2	Live Tooling Cutting Tool Installation	243
5.3.3	Live Tool Mounting in Turret	244
5.3.4	Live Tooling M-codes	246
5.3.5	C Axis	246
5.3.6	Cartesian to Polar Transformation (G112)	246

5.3.7	Cartesian Interpolation	247
5.3.8	Tool Radius Cutter Comp with G112 and G17	250
5.4	Y Axis	255
5.4.1	Y-Axis Travel Envelopes	256
5.4.2	Y-Axis Lathe with VDI Turret	256
5.4.3	Operation and Programming	256
5.5	Parts Catcher	259
5.5.1	Operation	259
5.5.2	Chuck Interference	260
5.6	Dual-Spindle Lathes (DS-Series)	261
5.6.1	Synchronized Spindle Control	261
5.6.2	Secondary Spindle Programming	264
5.7	Automatic Tool Setting Probe	265
5.7.1	Operation	265
5.7.2	Manual Mode	266
5.7.3	Automatic Mode	267
5.7.4	Break Detect Mode	268
5.7.5	Tool Tip Direction	268
5.7.6	Automatic Tool Probe Calibration	269
5.7.7	Tool Probe Alarms	270
5.8	Servo Auto Door	271
5.9	More Information Online	272
Chapter 6	G-codes, M-codes, Settings	273
6.1	G-codes	273
6.2	M-codes	370
6.3	Settings	388
6.4	More Information Online	427
Chapter 7	Maintenance	429
7.1	Introduction	429
7.2	Daily Maintenance	429
7.3	Weekly Maintenance	429
7.4	Monthly Maintenance	430
7.5	Every (6) Months	430
7.6	Annual Maintenance	430
Chapter 8	Other Equipment	431
8.1	Introduction	431
8.2	Office Lathe	431
8.3	Toolroom Lathe	431
8.4	More Information Online	431

Index.	433
-----------------------	------------

Chapter 1: Safety

1.1 Introduction



CAUTION: *This equipment is to be operated only by authorized and trained personnel in accordance with the Operator's manual, safety decals, safety procedures, and instructions for safe machine operation.*

IMPORTANT: *Read all appropriate warnings, cautions, and instructions before operating this machine.*

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

1.1.1 Read Before Operating



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

Basic safety:

- Consult your local safety codes and regulations before operating the machine. Contact your dealer anytime safety issues need to be addressed.
- It is the shop owner's responsibility to make sure that everyone who is involved in installing and operating the machine is thoroughly acquainted with the installation, operation, and safety instructions provided with the machine BEFORE they perform any actual work. The ultimate responsibility for safety rests with the shop owner and the individuals who work with the machine.
- Use appropriate eye and ear protection while operating the machine. ANSI-approved impact safety goggles and OSHA-approved ear protection are recommended to reduce the risks of sight damage and hearing loss.
- This machine is automatically controlled and may start at any time.
- This machine can cause severe bodily injury.
- Windows must be replaced if damaged or severely scratched. Replace damaged windows immediately.

- As sold, your machine is not equipped to process toxic or flammable material; this can create deadly fumes or suspended particles in the air. Consult with the material manufacturer for safe handling of material by-products, and implement all precautions before you work with such materials

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 and 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 the equipment.
- Never service the machine with the power connected.
- 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 properly. When a program runs, the tool turret can move rapidly at any time, and in any direction.
- **[EMERGENCY STOP]** is the large, circular red button located on the control pendant. Some machines may also have buttons in other locations. When you press **[EMERGENCY STOP]**, the axis motors, spindle motor, pumps, tool changer, and gear motors all stop. While **[EMERGENCY STOP]** is active, both automatic and manual motion is disabled. Use **[EMERGENCY STOP]** in case of emergency, and also to disable the machine for safety when you need to access motion areas.
- Check for damaged parts and tools before operating the machine. Any part or tool that is damaged should be properly repaired or replaced by authorized personnel. Do not operate the machine if any component does not appear to be functioning correctly.
- 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.

Chuck safety:

- Do not exceed the chuck's rated speed. Higher speeds reduce chuck clamping force.
- Unsupported barstock must not extend outside the drawtube.

- Chucks must be greased weekly and regularly serviced.
- Chuck jaws must not protrude beyond the diameter of the chuck.
- Do not machine parts larger than the chuck.
- Follow all of the chuck manufacturer's warnings regarding the chuck and workholding procedures.
- Hydraulic pressure must be set correctly to securely hold the work piece without distortion.
- Improperly clamped parts at high velocity may puncture the safety door. You must reduce the spindle speed to protect the operator when performing dangerous operations (e.g. turning oversized or marginally clamped parts).



DANGER: *Improperly clamped parts or oversized parts may be ejected with deadly force.*

Follow these guidelines while performing jobs on the machine:

- Normal operation - Keep the door closed and guards in place while machine is operating.
- Part loading and unloading – An operator opens the door or guard, completes task, closes the door or guard before pressing **[CYCLE START]** (starting automatic motion).
- Tool loading or unloading – A machinist enters the machining area to load or unload tools. Exit the area completely before automatic movement is commanded (for example, **[NEXT TOOL]**, **[TURRET FWD]**, **[TURRET REV]**).
- Machining job set-up – Press **[EMERGENCY STOP]** before adding or removing machine fixtures.
- Maintenance / Machine Cleaner– Press **[EMERGENCY STOP]** or **[POWER OFF]** on the machine before entering enclosure.

1.2 Environmental and Noise Limits

The following table lists the environmental and noise limits for safe operation:

T1.1: Environmental and Noise Limits

	Minimum	Maximum
Environmental (Indoor Use Only)*		
Operating Temperature	41 °F (5 °C)	122 °F (50 °C)
Storage Temperature	-4 °F (-20 °C)	158 °F (70 °C)
Ambient Humidity	20% relative, non-condensing	90% relative, non-condensing
Altitude	Sea level	6,000 ft. (1,829 m)
Noise		
Emitted from all areas of machine during use at a typical operator's position	70 dB	Greater than 85 dB

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

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

1.3 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 their responsibility to manage the progress of these methods. The machining process must be monitored to prevent damage if a hazardous condition occurs.

For example, if there is the risk of fire due to the material machined, then 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 perform an appropriate action without human intervention to prevent an accident, should a problem be detected.

1.4 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 will make the machine unsafe and void the warranty.

1.4.1 Robot Cells

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

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

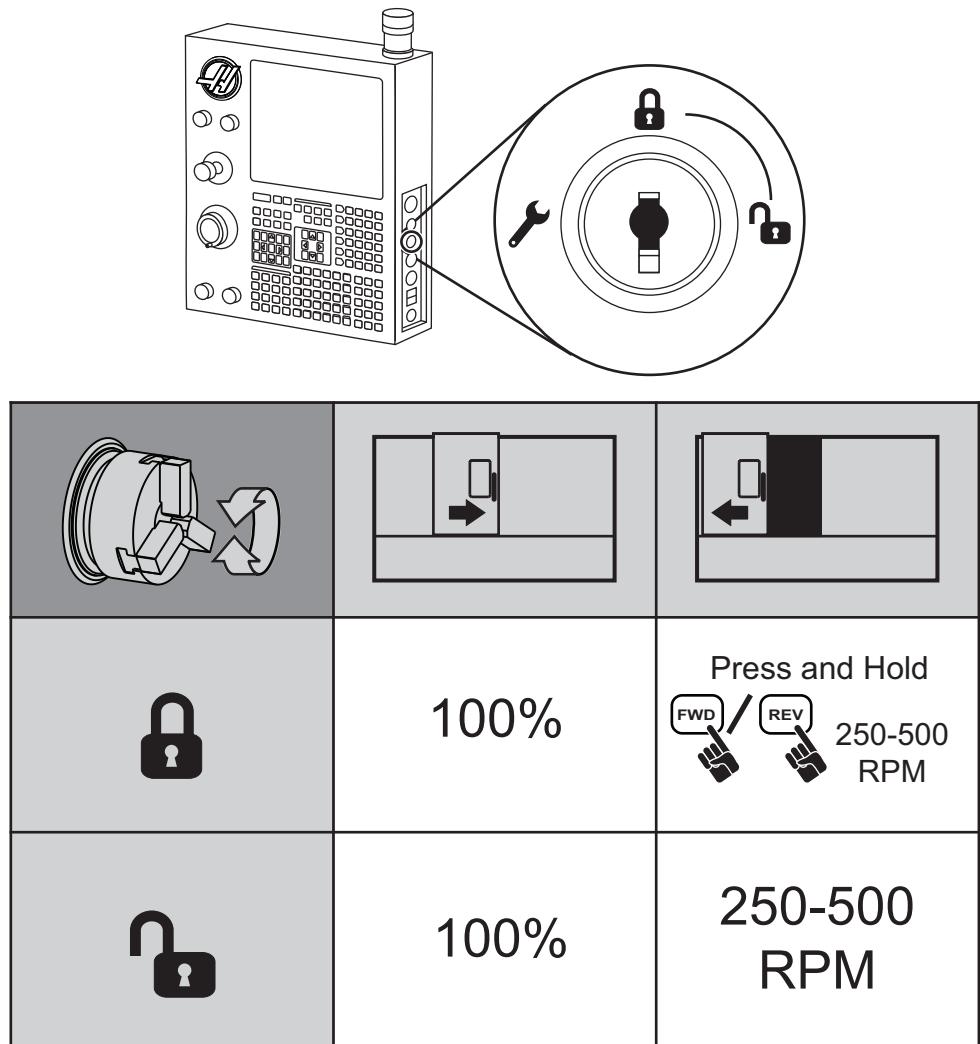
Robot cell setup is beyond the scope of this manual. Work with a robot-cell integrator and your HFO to correctly set up a safe robot cell.

1.4.2 Machine Behavior with the Door Open

For safety, machine operations are stopped when the door is open and the setup keyswitch is locked. The unlock position permits limited machine functions.

T1.2: Setup / Run Mode Limited Overrides with the Machine Doors Open

Machine Function	Locked (Run Mode)	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 [FWD] / [REV]	Allowed, but you must press and hold [FWD] or [REV]. Maximum 250-500 RPM, depending on the lathe model.	Allowed, but maximum 250-500 RPM, depending on the lathe model.
Tool Change	Not allowed.	Not allowed.
Next Tool feature	Not allowed.	Not allowed.
Opening door while a program is running	Not allowed. The door is locked.	Allowed, but axis motion stops and the spindle slows to a maximum of 250-500 RPM.
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.

F1.1: Spindle Control, Setup and Run Mode

F1.2: Axis Motion Rates, Setup and Run Mode

G00 G01 		
	100%	0%
	100%	0%

F1.3: Setup Mode, Tool Change and Conveyor Control with the Door Open.

	100% 100%	X 100%
	100% 100%	X 100%

1.5 Modifications to the Machine

DO NOT modify or alter this equipment in any way. Your Haas Factory Outlet (HFO) must handle all modification requests. Modification or alteration of any Haas machine without factory authorization could lead to personal injury and mechanical damage, and will void your warranty.

1.6 Safety Decals

To help ensure that CNC machine dangers are quickly communicated and understood, hazard symbol decals are placed on Haas Machines in locations where hazards exist. If decals become damaged or worn, or if additional decals are needed to emphasize a particular safety point, contact your dealer or the Haas factory.



NOTE:

Never alter or remove any safety decal or symbol.

Each hazard is defined and explained on the general safety decal, located at the front of the machine. Review and understand the four parts of each safety warning, explained below, and familiarize yourself with the symbols in this section.

F1.4: Standard Warning Layout



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

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

A: Hazard.

B: Consequence if warning is ignored.

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

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

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

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

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

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

Green + "INFORMATION" = Details about machine components.

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

1.6.1 Lathe Warning Decals

This is an example of a general lathe warning decal in English. You can contact your Haas Factory Outlet (HFO) to get these decals in other languages.

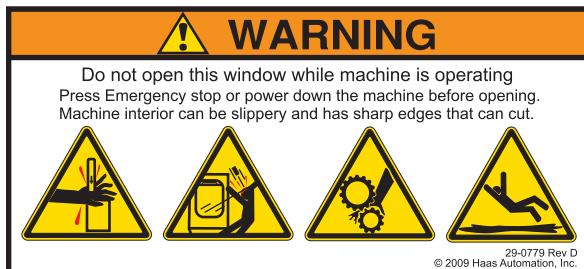
F1.5: Lathe General Warning Decal



1.6.2 Other Safety Decals

You may find other decals on your machine, depending on the model and options installed. Be sure to read and understand these decals. These are examples of other safety decals in English. You can contact your Haas Factory Outlet (HFO) to get these decals in other languages.

F1.6: Other Safety Decal Examples



1.7 More Information Online

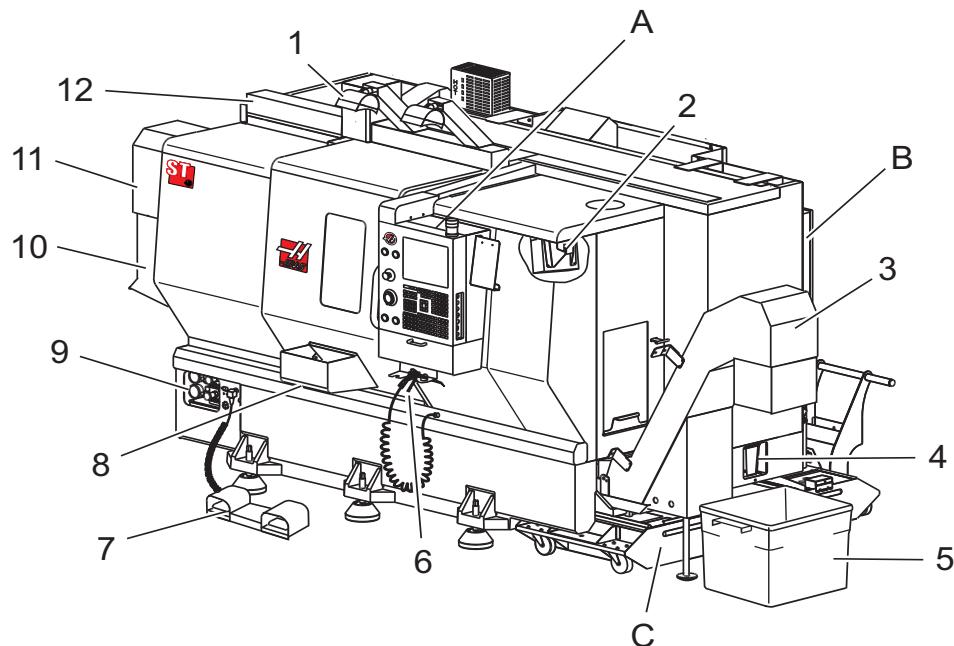
For updated and supplemental information, including tips, tricks, maintenance procedures, and more, go to www.HaasCNC.com and select the **Resource Center**.

Chapter 2: Introduction

2.1 Lathe Orientation

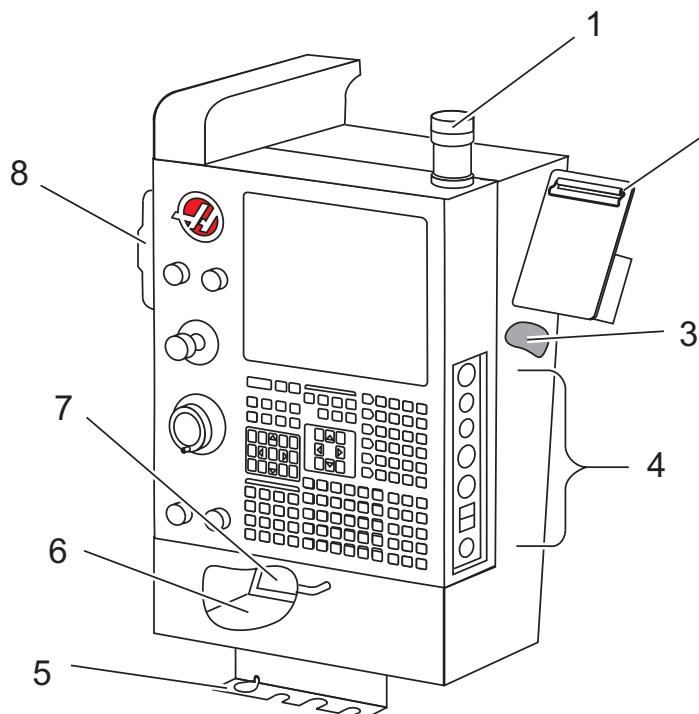
The following figures show some of the standard and optional features of your Haas Turning Center. Some of the features shown are highlighted in their appropriate sections. Note that these figures are representative only; your machine's appearance may vary depending on the model and installed options.

F2.1: Lathe Features (front view)

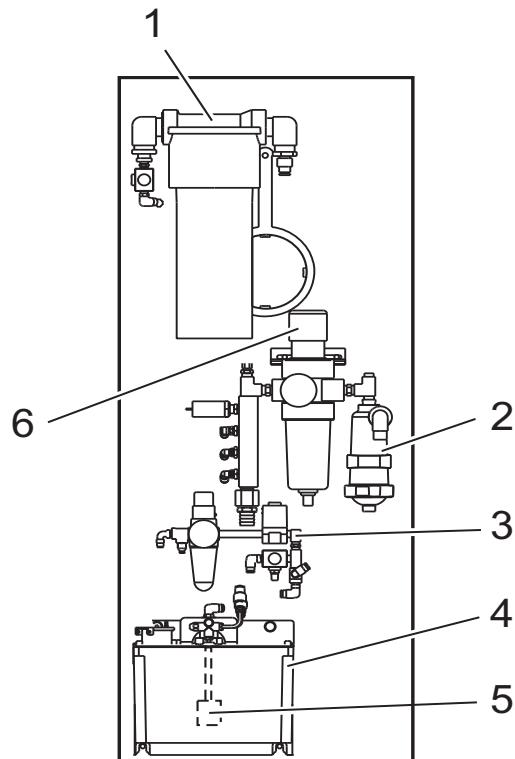


1. 2X High Intensity Lights (Optional)
2. Work Light (2X)
3. Chip Conveyor (Optional)
4. Oil Drain Container
5. Chip Container
6. Air Gun
7. Foot Pedal
8. Parts Catcher (Optional)
9. Hydraulic Power Unit (HPU)
10. Coolant Collector
11. Spindle Motor
12. Servo Auto Door (Optional)
- A. Control Pendant
- B. Minimal Lube Panel Assembly
- C. Coolant Tank

F2.2: Lathe Features (front view) Detail A - Control Pendant

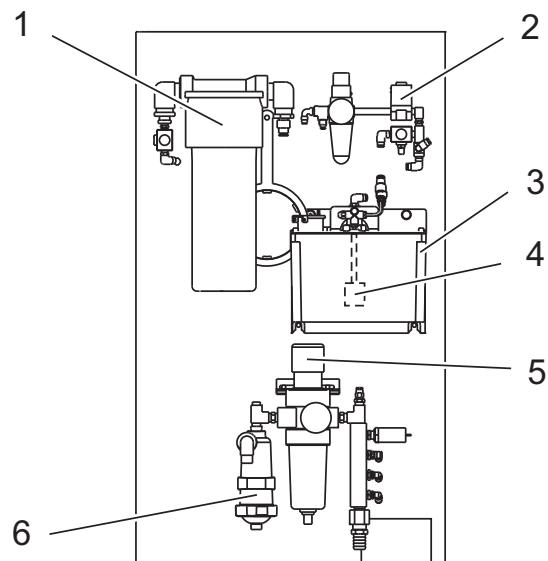


1. Work Beacon
2. Clipboard
3. Tool Tray
4. Side panel Controls
5. Vise Handle Holder
6. G- and M-code Reference List
7. Operator's Manual and Assembly Data (stored inside)
8. Remote Jog Handle

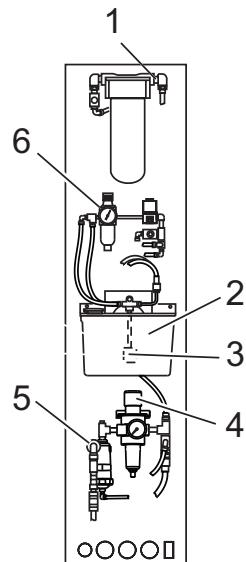
F2.3: Lathe Features (front view) Detail B - ST-10 minimal Lube Panel Assembly

1. Grease Reservoir Assembly
2. Water Separator Assembly
3. Spindle Air and Pump Control
4. Spindle Oil Tank Pump Assembly
5. Spindle Pump Assembly
6. Main Regulator Air Manifold Assembly

F2.4: Lathe Features (front view) Detail B - ST-20 Minimal Lube Panel Assembly

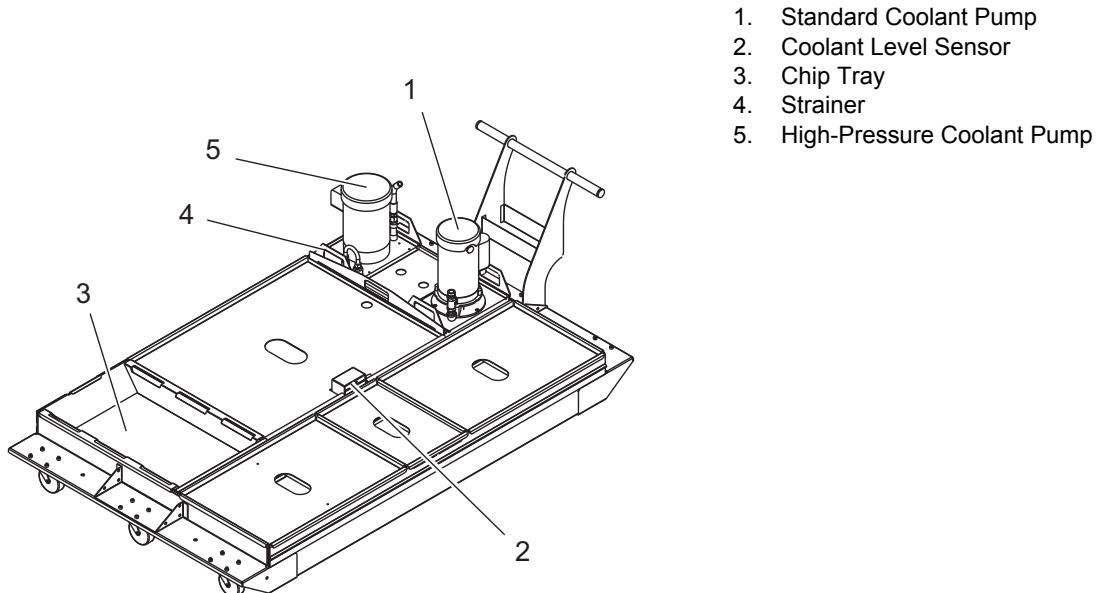


1. Grease Reservoir Assembly
2. Spindle Air and Pump Control
3. Spindle Oil Tank Pump Assembly
4. Spindle Pump Assembly
5. Main Regulator Air Manifold Assembly
6. Water Separator Assembly

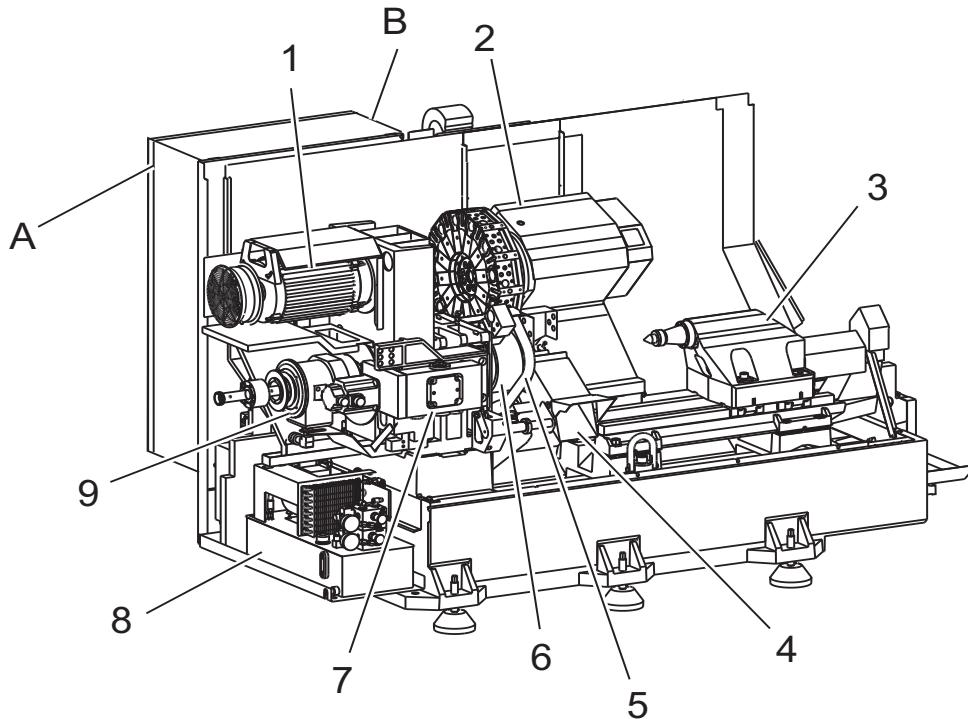
F2.5: Lathe Features (front view) Detail B - ST/DS-30 Minimal Lube Panel Assembly

1. Grease Reservoir Assembly
2. Spindle Oil Tank Pump Assembly
3. Spindle Pump Assembly
4. Main Regulator Air Manifold Assembly
5. Water Separator Assembly
6. Spindle Air and Pump Control

F2.6: Lathe Features (front view) Detail C - Coolant Tank Assembly

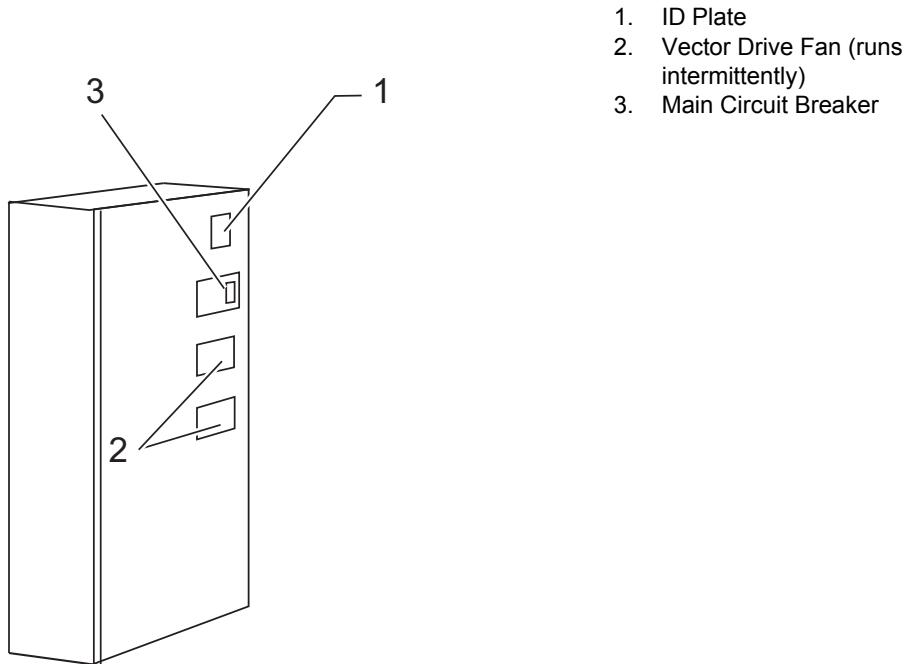


F2.7: Lathe Features (front view with covers removed)

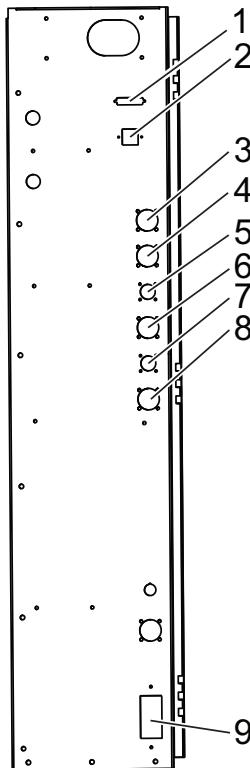


- 1. Spindle Motor
- 2. Tool Turret Assembly
- 3. Tailstock (Optional)
- 4. Parts Catcher (Optional)
- 5. LTP Arm (Optional)
- 6. Chuck
- 7. C-Axis Drive Assembly (Optional)
- 8. Hydraulic Power Unit (HPU)
- 9. Spindle Head Assembly
- A Control Cabinet
- B Control Cabinet Side Panel

F2.8: Lathe Features (front view with covers removed) Detail A - Control Cabinet



F2.9: Lathe Features (rear view) Detail B - Control Cabinet Side Panel



1. RS-232 (Optional)
2. Enet (Optional)
3. Scale A-Axis (Optional)
4. Scale B-Axis (Optional)
5. A-Axis Power (Optional)
6. A-Axis Encoder (Optional)
7. B-Axis Power (Optional)
8. B-Axis Encoder (Optional)
9. 115 VAC @ 5A

2.2 Control Pendant

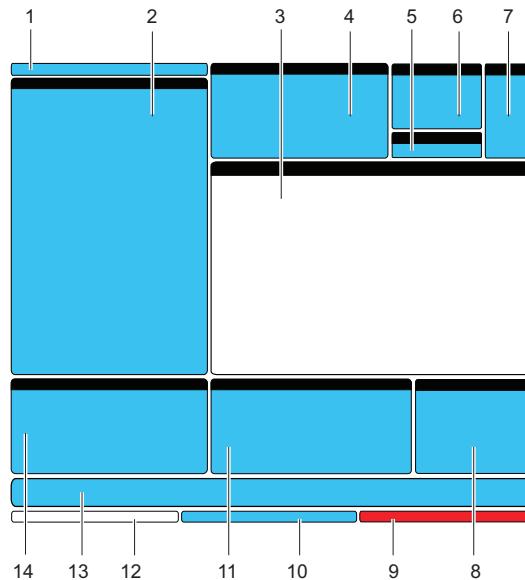
The control pendant is the main interface to your Haas machine. This is where you program and run your CNC machining projects. This control pendant orientation section describes the different pendant sections:

- Pendant front panel
- Pendant right side, top, and bottom
- Keyboard
- Screen displays

2.2.1 Control Display

The control display is organized into panes that vary depending on the current mode and what display keys are used.

F2.10: Lathe Basic Control Display Layout



1. Mode and Active Display Bar
2. Program Display
3. Main Display
4. Active Codes
5. Tailstock
6. Active Tool
7. Coolant
8. Timers Counters/Tool Management
9. Alarm Status
10. System Status Bar
11. Position Display/Axes Load Meters/Clipboard
12. Input Bar
13. Icon Bar
14. Main Spindle/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.

- F2.11:** The Mode and Display bar shows [1] the current mode and [2] the current display function.



T2.1: 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

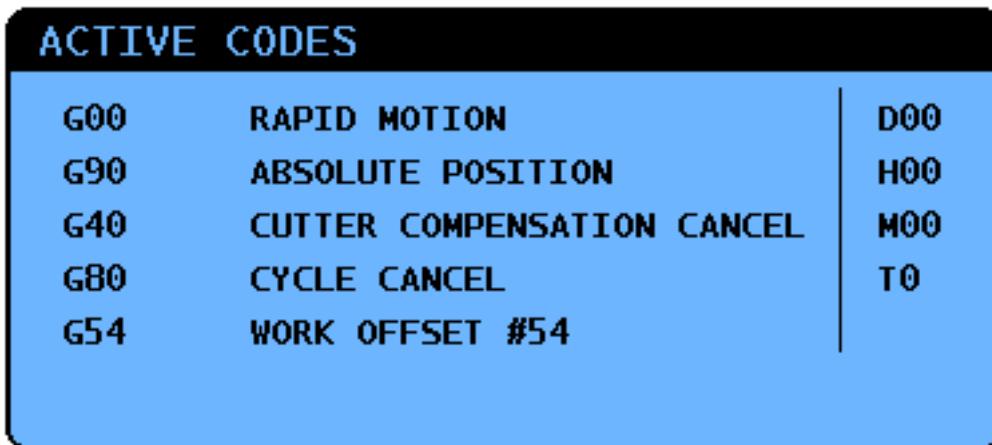
There are two offset tables, the Program Tool Offsets table and the Active Work Offset table. Depending on the mode, these tables may appear in two separate display panes, or they may share a pane; press [OFFSET] to toggle between tables.

T2.2: Offset Tables

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

Active Codes

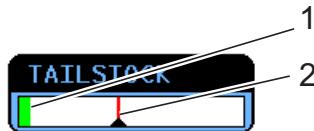
F2.12: Active Codes Display Example



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.

Tailstock Display

F2.13: Tailstock Display Example



This display gives information on the tailstock [1] current pressure and [2] maximum pressure.

Active Tool

F2.14: 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 Level Gauge

The coolant level is displayed in the upper-right of the screen in **OPERATION : MEM** mode. A vertical bar shows the coolant level. The vertical bar flashes when the coolant reaches a level that could cause coolant flow problems. 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 **26** 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.

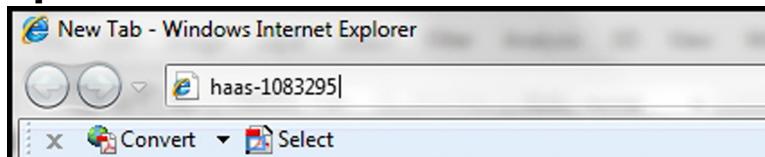
Press **[CANCEL]** or **[SPACE]** to delete existing characters. Press **[DELETE]** to delete an entire line. Your message data is automatically stored and maintained even in a power-off state.

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.

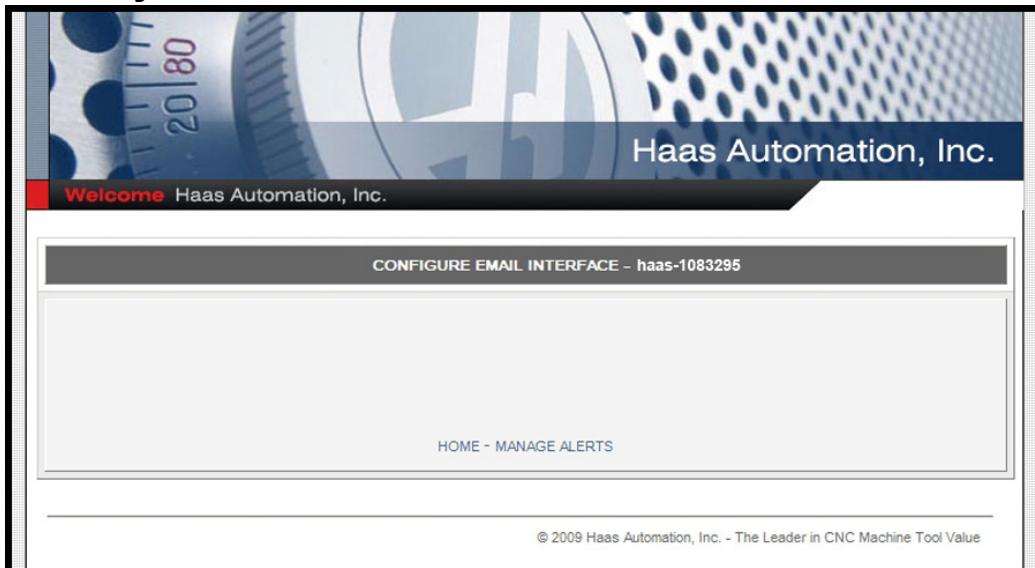
Before you set up alerts, make sure the machine has an established connection to your Local Area Network, and that Setting 900 defines a unique network name for the machine. This feature requires the Ethernet option and software version 18.01 or later.

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]**.

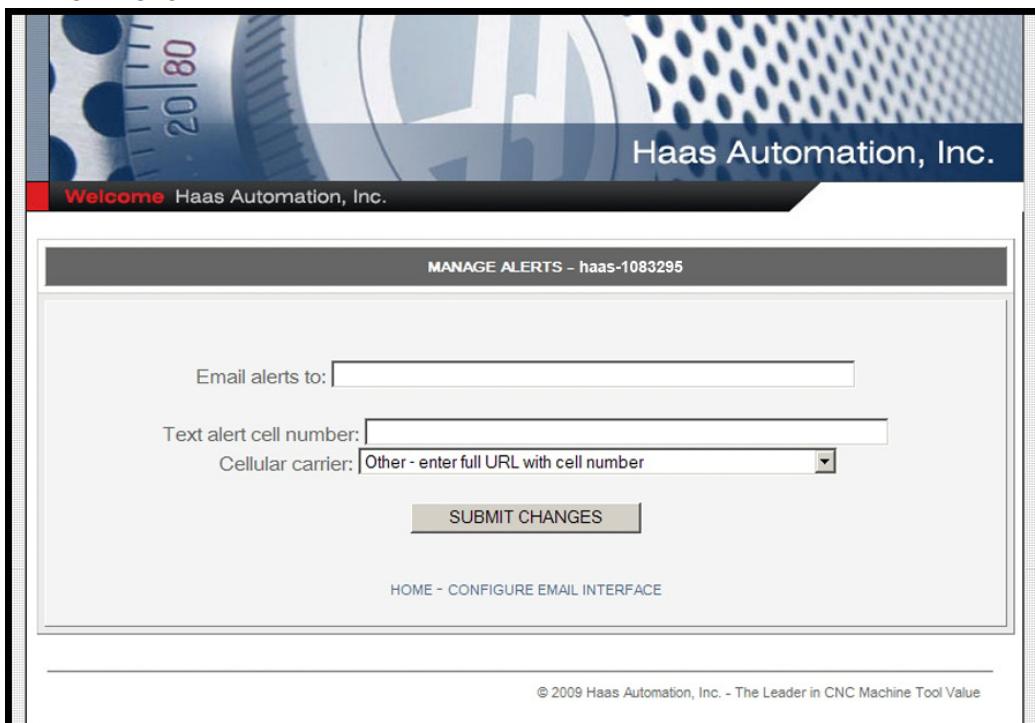


2. A message may appear with a request to set a cookie in your browser. This will happen each time you access the machine using a different computer or browser, or after an existing cookie has expired. Click **OK**.

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



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**.



**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 a web-based configuration interface for an email account. At the top, there is a banner with a blue and white abstract background featuring gears and numbers like 20 and 80. The banner contains the text "Haas Automation, Inc." and "Welcome Haas Automation, Inc.". Below the banner is a dark header bar with the text "CONFIGURE EMAIL INTERFACE - haas-1083295". The main form area has four input fields: "DNS IP address:", "SMTP server name:", "SMTP server port" (set to 25), and "Authorized EMAIL account:". Below these fields is a "SUBMIT CHANGES" button. At the bottom of the form is a link "HOME - MANAGE ALERTS". At the very bottom of the page, a copyright notice reads "© 2009 Haas Automation, Inc. - The Leader in CNC Machine Tool Value".

**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.

T2.3: Axis Position Reference Points

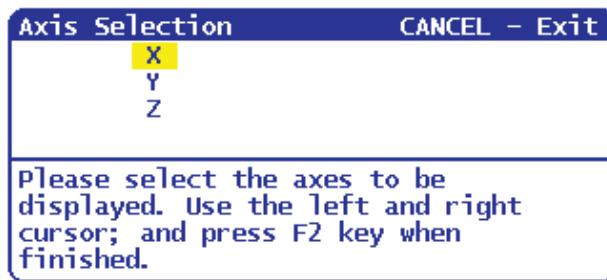
Coordinate Display	Function
OPERATOR	This position shows the distance you have jogged the axes. This does not necessarily represent the actual distance the axis is from machine zero, except when the machine is first powered on. Type the axis letter and press [ORIGIN] to zero the position value for that axis.
WORK (G 54)	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.

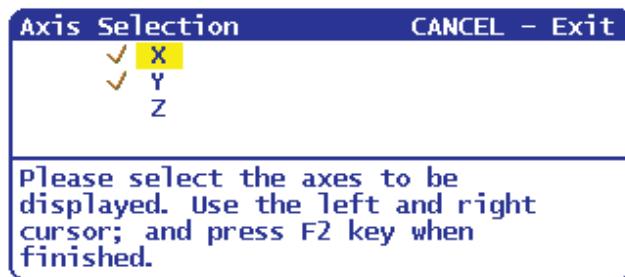
- With a position display active, press [F2]. The **Axis Selection** pop-up menu appears.

F2.15: The Axis Selection Pop-up Menu

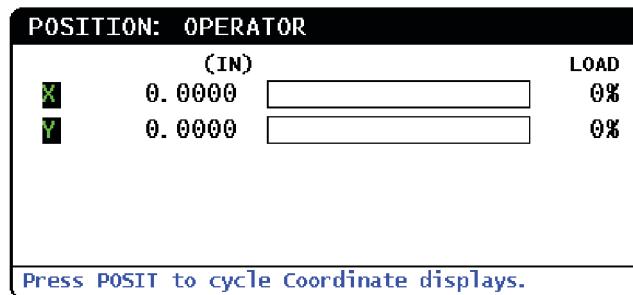


- Press the [LEFT] and [RIGHT] cursor arrow keys to highlight an axis letter.
- Press [ENTER] to place a check mark next to the highlighted axis letter. This mark means that you want to include that axis letter in the position display.

F2.16: The X and Y Axes Selected in the Axis Selection Menu



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

F2.17: The Updated Position Display

Setting/Graphic Display Function

The Settings are selected by pressing **[SETTING/GRAFIC]**. There are some special functions in the settings which change the way the lathe behaves; refer to the “Settings” section starting on page **388** for a more detailed description.

The Graphics function is selected by pressing **[SETTING/GRAFIC]** twice. Graphics is 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 may be considered more useful than the Dry Run mode, because all of your work offsets, tool offsets, and travel limits can be checked before running the machine. The risk of a crash during setup is greatly reduced.

Graphics Mode Operation

To run a program in Graphics, a program must be loaded and the control must be in either **MEM**, **MDI** or **Edit** mode. From **MEM** or **MDI**, press **[SETTING/GRAFIC]** twice to select the **Graphics** mode. From **Edit** mode, press **[CYCLE START]** while the active program edit pane is selected to start a simulation.

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.
- **Locator 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 top view of the X and Z axes. It displays tool paths during a graphics simulation of the program. Rapid moves are displayed as dotted lines, while feed motion is displayed as fine continuous lines.

**NOTE:**

Setting 4 disables the rapid path.

The places where a drilling canned cycle is used are marked with an X.

**NOTE:**

Setting 5 disables the drill mark.

- **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** This feature consists of a horizontal line displayed on the Z-axis bar at the top-right corner of the graphics screen to indicate 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.
- **[F3] / [F4]** Use these keys to control simulation speed. **[F3]** decrements speed, **[F4]** increments speed.

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.

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, starting on page **201** for more information.

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. Refer to page **30** for more information on position displays.

**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.

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.

T2.4: Offset Tables

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

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.

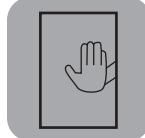
Icon Bar

The Icon Bar is divided into 18 image display fields. A machine condition icon will appear in one or more of the fields.

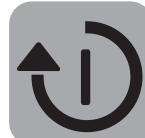
T2.5: Field 1

Name	Icon	Meaning
SETUP LOCKED		Setup mode is locked. Refer to page 5 for more information.
SETUP UNLOCKED		Setup mode is unlocked. Refer to page 5 for more information

T2.6: Field 2

Name	Icon	Meaning
DOOR HOLD		Machine motion has stopped because of door rules.
RUNNING		The machine is running a program.

T2.7: Field 3

Name	Icon	Meaning
RESTART		The control is scanning the program before a program restart. Refer to Setting 36 on page 399.
SINGB STOP		SINGLE BLOCK mode is active, and the control is waiting for a command to continue. Refer to page 52 for more information.
DNC RS232		DNC RS-232 mode is active.

T2.8: Field 4

Name	Icon	Meaning
FEED HOLD		The machine is in feed hold. Axis motion has stopped, but the spindle continues to turn.
FEED		The machine is executing a cutting move.
M FIN		The control is waiting for M-finish signal from an optional user interface (M121-M128).
M FIN*		The control is waiting for the M-finish signal from an optional user interface (M121-M128) to stop.
RAPID		The machine is executing a non-cutting axis move at the fastest possible rate.
DWELL		The machine is executing a dwell (G04) command.

T2.9: Field 5

Name	Icon	Meaning
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.
JOGGING, YZ MANUAL JOG, VECTOR JOG		An axis is jogging at the current jog rate.
REMOTE JOG		The optional remote jog handle is active.
RESTRICTED ZONE		A current axis position is in the restricted zone. (Lathe only)

T2.10: Field 6

Name	Icon	Meaning
G14		Mirroring mode is active.
X MIRROR, Y MIRROR, XY MIRROR		Mirroring mode is active in the positive direction.
X -MIRROR, Y -MIRROR, XY -MIRROR		Mirroring mode is active in the negative direction.

T2.11: Field 7

Name	Icon	Meaning
A/B/C/AB/CB/CA AXIS UNCLAMPED		A rotary axis, or a combination of rotary axes, is unclamped.
SPINDLE BRAKE ON		The lathe spindle brake is on.

T2.12: Field 8

Name	Icon	Meaning
TOOL UNCLAMPED		The tool in the spindle is unclamped. (Mill only)
CHECK LUBE, LOW SS LUBE		The control has detected a low lubrication state.
LOW AIR PRESSURE		Air pressure to the machine is insufficient.
LOW ROTARY BRAKE OIL		The rotary brake oil level is low.
MAINTENANCE DUE		A maintenance procedure is due, based on information in the MAINTENANCE page. Refer to page 30 for more information.

T2.13: Field 9

Name	Icon	Meaning
EMERGENCY STOP, PENDANT	 A red square icon containing a white hand with a red cross over it, with the number '1' in the top left corner.	[EMERGENCY STOP] on the pendant has been pressed. This icon disappears when [EMERGENCY STOP] is released.
Mill: EMERGENCY STOP, PALLET Lathe: EMERGENCY STOP, BARFEED	 A red square icon containing a white hand with a red cross over it, with the number '2' in the top left corner.	[EMERGENCY STOP] on the pallet changer (mill) or the bar feeder (lathe) has been pressed. This icon disappears when [EMERGENCY STOP] is released.
Mill: EMERGENCY STOP, TC CAGE Lathe: EMERGENCY STOP, AUXILIARY 1	 A red square icon containing a white hand with a red cross over it, with the number '3' in the top left corner.	[EMERGENCY STOP] on the tool changer cage (mill) or auxiliary device (lathe) has been pressed. This icon disappears when [EMERGENCY STOP] is released..
Mill: EMERGENCY STOP, AUXILIARY Lathe: EMERGENCY STOP, AUXILIARY 2	 A red square icon containing a white hand with a red cross over it, with the number '4' in the top left corner.	[EMERGENCY STOP] on an auxiliary device has been pressed. This icon disappears when [EMERGENCY STOP] is released.

T2.14: Field 10

Name	Icon	Meaning
SINGLE BLK	 A grey square icon showing a dashed L-shaped line.	SINGLE BLOCK mode is active. Refer to page 52 for more information.

T2.15: Field 11

Name	Icon	Meaning
DRY RUN		DRY RUN mode is active. Refer to page 107 for more information.

T2.16: Field 12

Name	Icon	Meaning
OPTIONAL STOP		OPTIONAL STOP is active. The control stops the program at each M01 command.

T2.17: Field 13

Name	Icon	Meaning
BLOCK DELETE		BLOCK DELETE is active. The control skips program blocks that begin with a slash (/).

T2.18: Field 14

Name	Icon	Meaning
CAGE OPEN		The side-mount tool changer door is open.
TC MANUAL CCW		The side-mount tool changer carousel is rotating counter-clockwise as commanded by a manual carousel rotation button.
TC MANUAL CW		The side-mount tool changer carousel is rotating clockwise as commanded by a manual carousel rotation button.
TC MOTION		A tool change is in progress.

T2.19: Field 15

Name	Icon	Meaning
PROBE DOWN		The probe arm is down for a probing operation.
PART CATCHER ON		The parts catcher is activated. (Lathe only)
TS PART HOLDING		The tailstock is engaged with the part. (Lathe only)
TS PART NOT HOLDING		The tailstock is not engaged with the part. (Lathe only)
CHUCK CLAMPING		The collet closer-type chuck is clamping. (Lathe only)

Control Display

T2.20: Field 16

Name	Icon	Meaning
TOOL CHANGE		A tool change is in progress.

T2.21: Field 17

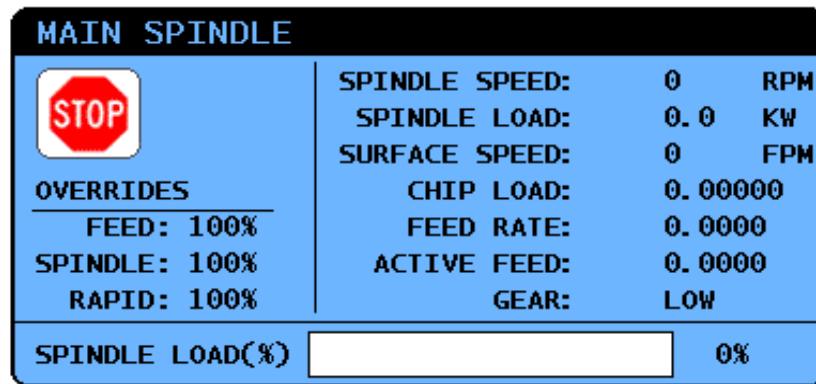
Name	Icon	Meaning
AIR BLAST ON		The Auto Air Gun (mill) or Auto Jet Air Blast (lathe) is active.
CONVEYOR FORWARD		The conveyor is active and currently moving forward.
CONVEYOR REVERSE		The conveyor is active and currently moving in reverse.

T2.22: Field 18

Name	Icon	Meaning
COOLANT ON		The main coolant system is active.
THROUGH-SPINDLE COOLANT (TSC) ON		The Through-Spindle Coolant (TSC) system is active. (Mill only)
HIGH PRESSURE COOLANT		The High-Pressure Coolant system is active. (Lathe only)

Main Spindle Display

F2.18: 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.

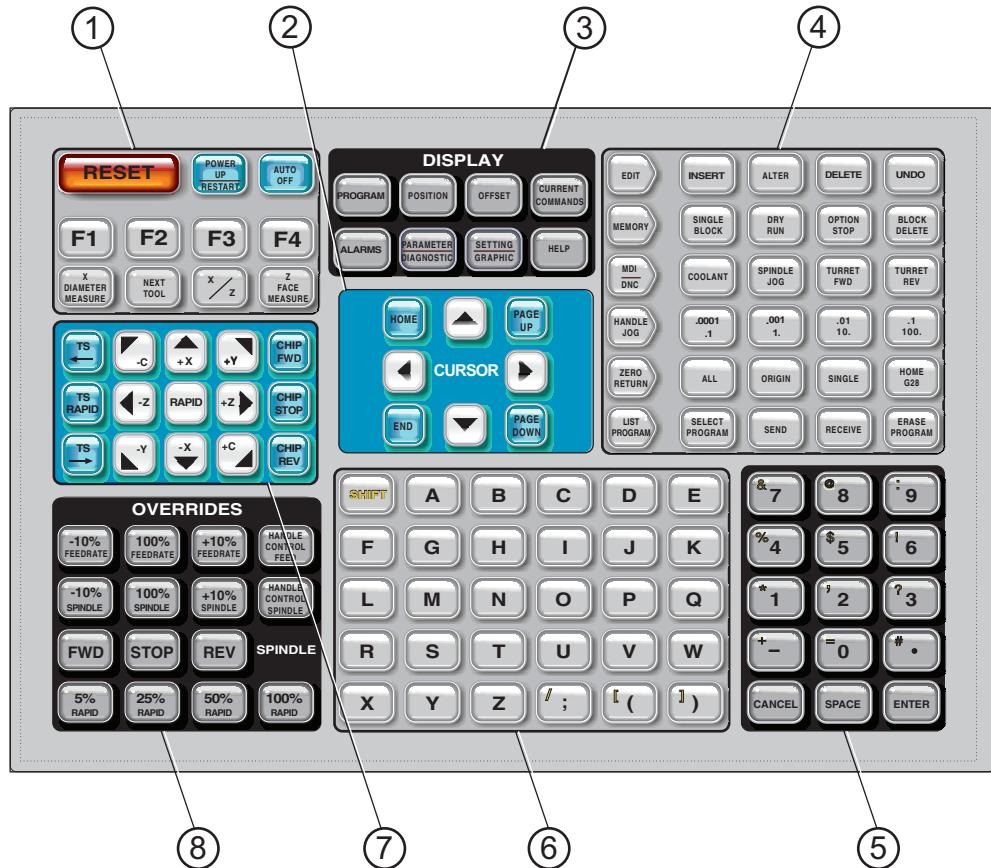
2.2.2 Keyboard

The keyboard on the control pendant operates with single or multiple key presses. The keys are grouped into the following functional areas:

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

Refer to the figure for key group location.

- F2.19:** Lathe Pendant Keypad: Function Keys [1], Cursor Keys [2], Display Keys [3], Mode Keys [4], Numeric Keys [5], Alpha Keys [6], Jog Keys [7], Overrides Keys [8]



Function Keys

Name	Key	Function
Reset	[RESET]	Clears alarms. Sets overrides to default values.
Power up/Restart	[POWER UP/RESTART]	Homes the machine. Clears alarm 102. Displays Current Commands page.

Name	Key	Function
Auto Off	[AUTO OFF]	Makes a tool change and shuts down the lathe after a specified time.
F1- F4	[F1 - F4]	These buttons have different functions depending on mode of operation. See the specific mode section for further descriptions and examples.
X Diameter Measure	[X DIAMETER MEASURE]	Used to record X-Axis tool shift offsets on the offset page during part setup.
Next Tool	[NEXT TOOL]	Used to select the next tool from the turret (usually used during part setup).
X/Z	[X/Z]	Used to toggle between X-axis and Z-Axis jog modes during part setup.
Z Face Measure	[Z FACE MEASURE]	Used to record Z-Axis tool shift offsets on the offset page during part setup.

Cursor Keys

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.  NOTE: <i>This manual refers to these keys by their spelled-out names.</i>
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

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

Name	Key	Function
Program	[PROGRAM]	Selects the active program pane in most modes. In MDI mode, press this key to access VQC and IPS/WIPS (if installed).
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.
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 CNC machine tool. Once a mode key is pressed, the keys in the same row are made available to the user. The current mode is always displayed on the top line just to the right of the current display.

T2.23: Edit Mode Keys

Name	Key	Function
Edit	[EDIT]	Selects edit mode. This mode is used to edit programs in the control's memory. Edit mode provides two editing panes: one for the currently active program, and another for background editing. Switch between the two panes by pressing the [EDIT] key.  NOTE: <i>While using this mode in an active program, press F1 to access help pop-up menus.</i>
Insert	[INSERT]	Pressing this key will enter commands into the program at the cursor. This key will also insert the text from the clipboard to the current cursor location, and is also used to copy blocks of code in a program.
Alter	[ALTER]	Pressing this key will change the highlighted command or text to the newly entered commands or text. This key will also change the highlighted variables to the text stored in the clipboard, or move a selected block to another location.
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.

T2.24: Memory Mode Keys

Name	Key	Function
Memory	[MEMORY]	Selects memory mode. This page displays the current active program. You run programs from this mode, and the [MEMORY] row contains keys that control the manner in which a program is executed.
Single Block	[SINGLE BLOCK]	Turns single block on or off. When single block is on, only one block of the program is executed for every press of [CYCLE START] .
Dry Run	[DRY RUN]	This is used to check actual machine movement without cutting a part (see the Dry Run section in the Operation Chapter).
Optional Stop	[OPTION STOP]	Turns optional stops on and off. When this feature is on and an M01 (optional stop) code is programmed, the machine will stop when it reaches the M01. The machine will continue once [CYCLE START] is pressed. If you press [OPTION STOP] during a program, it will take effect on the line after the highlighted line when [OPTION STOP] was pressed.
Block Delete	[BLOCK DELETE]	Turns the block delete function on and off. Blocks with a slash ("/") as the first item are ignored (not executed) when this option is enabled. If a slash is within a line of code, the commands after the slash will be ignored if this feature is enabled. Block delete takes effect two lines after [BLOCK DELETE] is pressed, except when cutter compensation is used; in this case, block delete will not take effect until at least four lines after the highlighted line. Processing will slow down for paths containing block deletes during high-speed machining. Block delete stays active after power is cycled.

T2.25: MDI/DNC Mode Keys

Name	Key	Function
Manual Data Input/Direct Numeric Control	[MDI/DNC]	MDI mode is where a program can be written but it is not entered into memory. DNC mode allows large programs to be "drip fed" into the control so it can be executed (See DNC mode section).
Coolant	[COOLANT]	Turns the optional coolant on and off. The optional HPC (High Pressure Coolant) is activated by pressing [SHIFT] followed by [COOLANT] . Note that as HPC and regular coolant share a common orifice, they cannot both be on at the same time.
Spindle Jog	[SPINDLE JOG]	Rotates the spindle at the speed selected in Setting 98 (Spindle Jog RPM).
Turret Forward	[TURRET FWD]	Rotates the tool turret forward to the next sequential tool. If Tnn is entered on the input line, the turret will advance in the forward direction to tool nn.
Turret Reverse	[TURRET REV]	Rotates the tool turret backward to the previous tool. If Tnn is entered on the input line, the turret will advance in the reverse direction to tool nn.

T2.26: Jog Mode Keys

Name	Key	Function
Handle Jog	[HANDLE JOG]	Selects axis jogging mode .0001, .1 - 0.0001 inches (metric 0.001mm) for each division on the jog handle. For dry run, .1 inches/min.
.0001/.1	[.0001 .1], [.001 1], [.01 10], [.1 100]	The first number (top number), when in inch mode, selects that amount to be jogged for each click of the jog handle. When the lathe is in MM mode the first number is multiplied by ten when jogging the axis (e.g. .0001 becomes 0.001mm). The second number (bottom number) is used for dry run mode selection of the speed, feedrate, and axis motions. These keys can also control feedrate when you hold an axis button down.

T2.27: Zero Return Mode Keys

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.
All	[ALL]	Returns all axes to machine zero. This is similar to [POWER UP/RESTART] except a tool change will not occur. This can be used to establish the initial axes zero position. This will not work on toolroom lathes, secondary spindle lathes, or automatic parts loader (APL).
Origin	[ORIGIN]	Resets selected displays and timers.
Single	[SINGLE]	Returns one axis to machine zero. Press desired axis letter on the Alpha keyboard and then press [SINGLE]. This moves a single axis to the initial axis zero position.
Home G28	[HOME G28]	Returns all axes to zero in rapid motion. If you enter an axis letter on the Alpha keyboard and press [HOME G28], the single axis returns to zero.  CAUTION: <i>There is no warning message to alert the operator of any possible collision.</i>

T2.28: List Programs Mode Keys

Name	Key	Function
List Programs	[LIST PROGRAM]	Controls all loading and saving of data in the control.
Select Programs	[SELECT PROGRAM]	Makes the highlighted program in the program list the active program.  NOTE: <i>The active program is marked with an "A" in the program list.</i>
Send	[SEND]	Transmits programs through the optional RS-232 serial port.

Name	Key	Function
Receive	[RECEIVE]	Receives programs through the optional RS-232 serial port.
Erase Program	[ERASE PROGRAM]	Erases the cursor selected programs in List Program mode or the entire program when in MDI mode.

Numeric Keys

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

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

Alpha Keys

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

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

Lathe Jog Keys

Name	Key	Function
Tailstock towards spindle	[TS <—]	Press and hold this key to move the tailstock towards the spindle.
Tailstock rapid	[TS RAPID]	Increases the speed of the tailstock when pressed simultaneously with one of the other tailstock keys.
Tailstock away from spindle	[TS —>]	Press and hold this key to move the tailstock away from the spindle.
Axis keys	[+X/-X, +Z/-Z, +Y/-Y, +C/-C]	Press and hold an individual key or press the desired axes and use the jog handle.

Name	Key	Function
Rapid	[RAPID]	Press and hold this key simultaneously with one of the above keys (X+, X-, Z+, Z-) to move that axis in the selected direction at maximum jog speed.
Chip Conveyor Forward	[CHIP FWD]	Starts optional chip conveyor in the "Forward" direction, moving chips out of the machine.
Chip Conveyor Stop	[CHIP STOP]	Stops the chip conveyor.
Chip Conveyor Reverse	[CHIP REV]	Starts the optional chip conveyor in the "Reverse" direction, which is useful in clearing jams and debris.

Y-Axis Lathes

To jog the Y Axis:

1. Press **[Y]**.
2. Press **[HANDLE JOG]**.
3. Turn the jog handle to jog the Y Axis.

XZ (Two-Axis) Jogging

The lathe X and Z Axes can be jogged simultaneously using the **[+X]/[-X]** and **[+Z]/[-Z]** jog keys.



NOTE:

Normal tailstock restricted zone rules are active while engaged in XZ jogging.

1. Hold any combination of **[+X]/[-X]** and **[+Z]/[-Z]** to jog the X and Z Axes simultaneously.
2. If only a single key is released, the control will continue jogging the single axis of the key still held.

C-Axis Lathes

To jog the C Axis:

1. Press [C].
2. Press [HANDLE JOG].
3. Turn the [HANDLE JOG] control to jog the C-Axis.

Override Keys

The override keys allow you to override the speed of rapid (non-cutting) axis motion, programmed feeds, and spindle speeds. These keys are listed in the following table.

Name	Key	Function
-10% Feedrate	[-10% FEEDRATE]	Decreases the current feedrate by 10%, down to 0%.
100% Feedrate	[100% FEEDRATE]	Sets the overridden feedrate to the programmed feed rate.
+10% Feedrate	[+10% FEEDRATE]	Increases the current feedrate by 10%, up to 990%.
Handle Control Feedrate	[HANDLE CONTROL FEED]	Allows you to use the jog handle to control the feedrate in $\pm 1\%$ increments, from 0% to 999%.
-10% Spindle	[-10% SPINDLE]	Decreases the current spindle speed by 10%, down to 0%.
100% Spindle	[100% SPINDLE]	Sets the overridden spindle speed to the programmed speed.
+10% Spindle	[+10% SPINDLE]	Increases the current spindle speed by 10%, up to 990%.
Handle Control Spindle RPM	[HANDLE CONTROL SPINDLE]	Allows you to use the [HANDLE JOG] control to change spindle speed in $\pm 1\%$ increments, from 0% to 999%.

Name	Key	Function
Forward	[FWD]	Starts the spindle in the clockwise direction. The spindle can be started or stopped with the [FWD] or [REV] buttons any time the machine is at a Single Block stop or [FEED HOLD] has been pressed. When the program is restarted with [CYCLE START] , the spindle will be turned back on to the previously defined speed.
Stop	[STOP]	Stops the spindle.
Reverse	[REV]	Starts the spindle in the Reverse (counterclockwise) direction. The spindle can be started or stopped by pressing [FWD] or [REV] any time the machine is at a Single Block stop or [FEED HOLD] has been pressed. When the program is restarted with [CYCLE START] , the spindle is turned back on to the previously defined speed.
Rapids	[5% RAPID] / [25% RAPID] / [50% RAPID] / [100% RAPID]	Limits machine rapids to the value on the key. [100% RAPID] allows maximum rapid.
You can also type an RPM value and press [FWD] or [REV] to command the spindle to that speed and direction.		

Override Usage

Overrides allow you to 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, stopping rapid and feed moves when it is pressed. It also stops tool changes and part timers, but will not stop a threading cycle or a dwell timer.

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.

2.2.3 Pendant Front Panel

T2.29: Front Panel Controls

Name	Image	Function
[POWER ON]		Powers the machine on
[POWER OFF]	O	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.

Name	Image	Function
[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.

2.2.4 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.

2.2.5 Pendant Right Side, Top, and Bottom Panels

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

T2.30: 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).

T2.31: 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.

Beacon Light	
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.

T2.32: Pendant Bottom Panel

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

2.3 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.

2.4 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 66 for information on that menu. Press [HELP] again to exit the help function.

F2.20: 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 66.
- **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.

2.4.1 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 **64**.

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 **65**.
- **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 **67** for more information.

2.4.2 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.

2.4.3 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.

2.4.4 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.

2.4.5 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 **/**).

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 when one of the other functions (**+**, **-**, *****, **/**) is selected, that calculation will be performed with the number just entered and any number that was already in the calculator window (like RPN).
3. The calculator will also accept a mathematical expression such as $23*4 - 5.2 + 6/2$, evaluating it (doing multiplication and division first) and placing the result, 89.8 in this case, in the window. 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]** will copy the highlighted triangle/circular milling/tapping value into the data entry line at the bottom of the screen. This is useful when the calculated solution will be used in a program.

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.

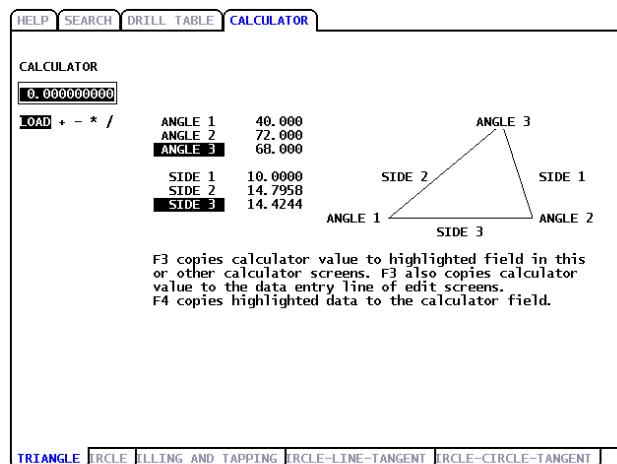
Triangle Sub-tab

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.

F2.21: Calculator Triangle Example



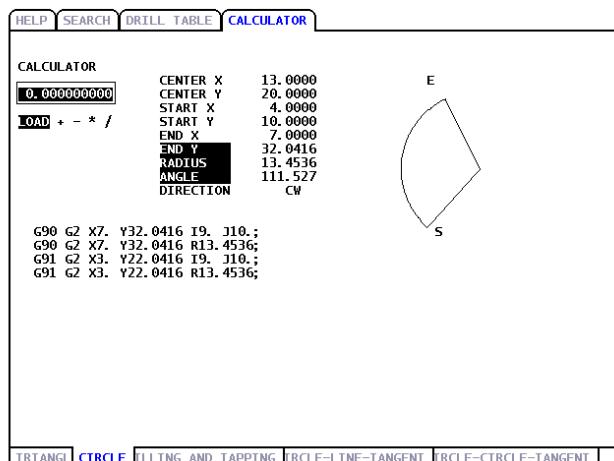
Circle Sub-tab

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.

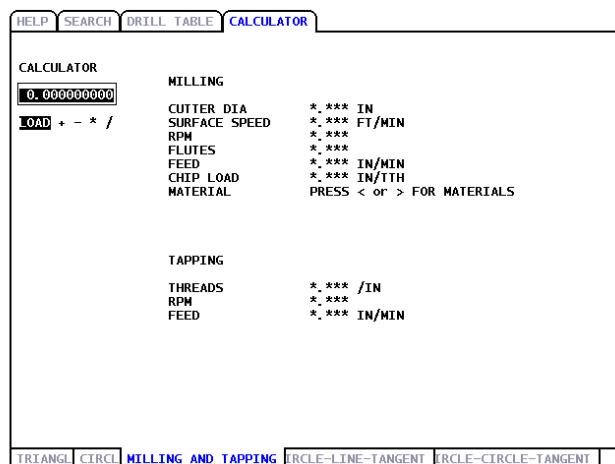
F2.22: Calculator Circle Example



Milling and Tapping Sub-tab

This calculator helps you determine the correct speeds and feeds for your application. Enter all of the available information about your tooling, material, and planned program, and the calculator fills in recommended feedrates when it has enough information.

F2.23: Calculator Milling and Tapping Example



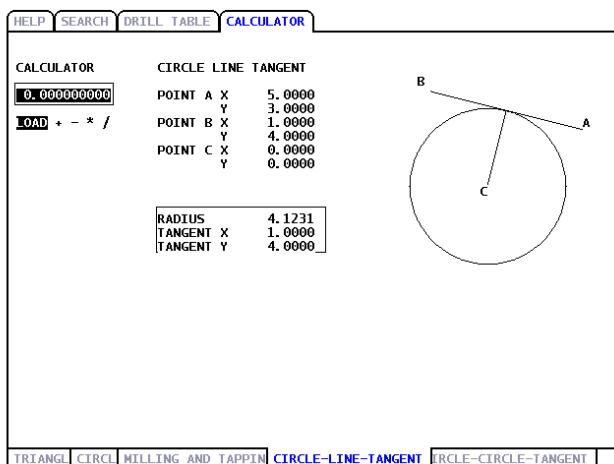
Circle-Line-Tangent Sub-tab

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.

F2.24: 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:

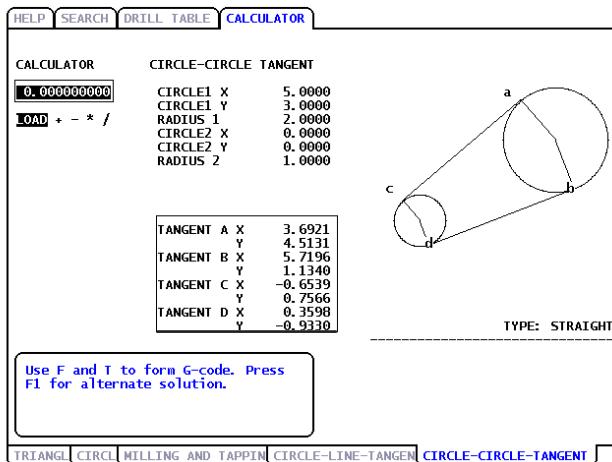
For every input condition (two disjointed circles), there are up to eight intersection points. Four points are from drawing straight tangents and four points by forming cross tangents.

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.

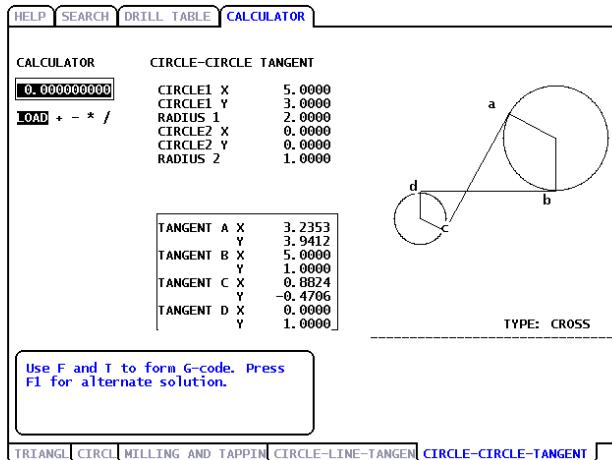
The Input Bar displays the G code for the segment. Solution is in G90 mode. Press M to toggle to G91 mode.

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

F2.25: Calculator Circle-Circle-Tangent Type: Straight Example



F2.26: Calculator Circle-Circle-Tangent Type: Cross Example



2.5 More Information Online

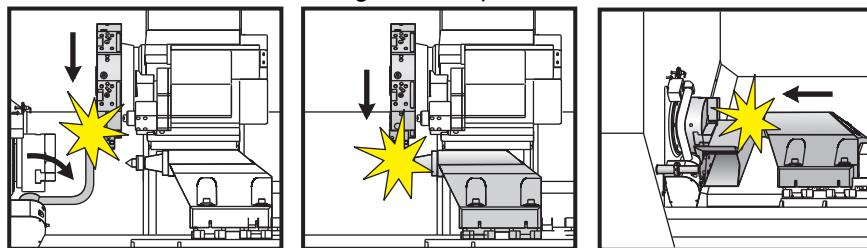
For updated and supplemental information, including tips, tricks, maintenance procedures, and more, go to www.HaasCNC.com and select the **Resource Center**.

Chapter 3: Operation

3.1 Machine Power-On

Before following this procedure to power on your turning center, clear potential crash areas, such as the tool probe, parts catcher, tailstock, tool turret, and secondary spindle.

F3.1: Possible Crash Areas During Power-Up



To turn the lathe on:

1. On the pendant, press and hold **[POWER ON]** until the Haas logo appears. The machine performs a self test and then displays either the Haas Start Up screen, the Messages screen (if a message was left), or the Alarms screen. In any case, the control has one or more alarms present (102 SERVOS OFF, the tool probe, parts catcher, tailstock, tool turret, and secondary spindle, etc.).
2. Follow the directions in the System Status bar on the bottom center of the display. Generally, the doors need to be cycled and **[EMERGENCY STOP]** cleared before the Power Up or Auto All Axes operations become available. For more information on safety lock features, refer to **5**.
3. Press **[RESET]** to clear each alarm. If an alarm cannot be cleared, the machine may need servicing; if this is the case, call your dealer.
4. Once the alarms are cleared the machine needs to start all operations from a reference point; this point is called Home. To home the machine, press **[POWER UP/RESTART]**.



NOTE:

[POWER UP/RESTART] does not work on *TL* lathes and Dual Spindle machines. These machine's axes need to be ZERO returned individually.



WARNING: *Automatic motion begins when you press [POWER UP/RESTART]. There is no further prompt or warning.*

5. Watch the tool probe, parts catcher, tailstock, tool turret, and secondary spindle for proper position during start-up and machining cycles.



NOTE: *Pressing [POWER UP/RESTART] automatically clears Alarm 102 if it was present.*

6. **Y-Axis Lathes:** Always command the Y Axis home before the X Axis. If the Y Axis is not at the zero position (spindle centerline) the X Axis may not be able to return home. The machine may give an alarm or message like *Y Axis is not at home*.

When this power-on procedure is complete, the control displays the **OPERATION:MEM** mode. The lathe is ready to run.

3.2 Spindle Warm-Up Program

If your machine's spindle has been idle for more than 4 days, you must run the spindle warm-up program before you use the machine. This program brings the spindle up to speed slowly, which distributes the lubrication and allows the spindle to thermally stabilize.

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

3.3 Device Manager

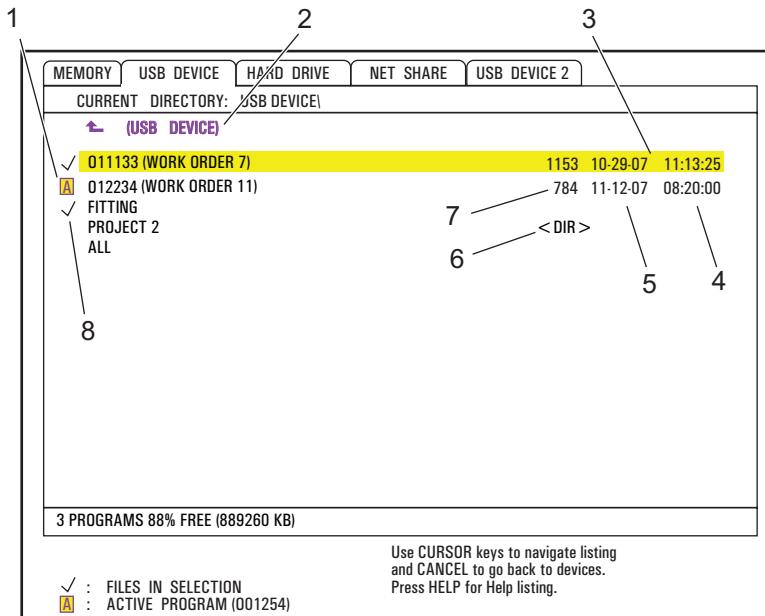
The Device Manager shows you the available memory devices and their contents in a tabbed menu. For information on navigating tabbed menus in the Haas control, refer to page **64**.



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.

F3.2: USB Device Menu



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

3.3.1 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 which may contain further 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

1. Highlight the directory you want to open. Directories have a <DIR> designation in the file list, then press [ENTER].
2. To return to the previous directory level, highlight the directory name at the top of the file list (it also has an arrow icon). 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.

3.3.2 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 64 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 name and press [SELECT PROGRAM].

The program becomes the active program.

If the active program is in **MEMORY**, it is designated with the letter **A**. If the program is on a USB memory device, the hard drive, or net share, it is designated with **FNC**.

3. In **OPERATION:MEM** mode, you can type an existing program name and press the [UP] or [DOWN] cursor arrow to quickly change programs.

3.3.3 Program Transfer

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

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 files with the program number and no extension, but some PC applications may not recognize the file without the extension.

Files developed in the control will be 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.
2. Once all programs are selected, press [**F2**]. This will open the **Copy To** window. Use the cursor arrows to select the destination and press [**ENTER**] to copy the program. Files copied from the control's memory to a device will have the extension *.NC* appended to the file name. However the name can be changed by navigating to the destination directory, entering a new name, and then pressing [**F2**].

3.3.4 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 number.
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.

3.3.5 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.

3.3.6 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 number (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, cursor to the program without typing a new program number and press [**F2**].
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.

3.3.7 Changing Program Numbers

You can change a program number

1. Highlight the file.
2. Type a new name.
3. Press **[ALTER]**.

3

Program Number Change (in Memory)

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

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

The program number changes to the number you specified.

If the new program name already exists in **MEMORY**, the control returns the message *Prog exists*, and the program name does not change.

3.4 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.

F3.3: Save and Load Popup



3.4.1 Making a Backup

The backup function saves your files with a filename that you designate. Each data type gets an associated extension:

Save File Type	File Extension
Offsets	.OFS
Settings	.SET
Macros - Variables	.VAR
Parameters	.PAR
Parameters - Pallet Positions (Mill)	.PAL
Parameters - Linear Screw Compensation	.LSC
A Axis Rotary Parameters (Mill)	.ROT
B Axis Rotary Parameters (Mill)	.ROT
History	.HIS
Program	.PGM

Save File Type	File Extension
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 **76** for instructions.
 4. Press **[F4]**.
The **Save and Load** popup menu appears.
 5. Highlight the option you want.
 6. Type a filename, then press **[ENTER]**.
- The control saves the data you chose, under the filename you typed (plus extensions), in the current directory on the USB memory device.

3.4.2 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. Select the file type to load, and then press **[ENTER]**.
- The

7. To load all file types (settings, parameters, programs, macros, tool offsets, variables, etc.) with the same name, select **Load All - Restore**.
8. Type a file name with no extension (e.g., 28012014), and press **[ENTER]**. All the files are loaded on the machine.

3.5 Basic Program Search

You can search a program for specific codes or text in **MDI**, **EDIT** or **MEMORY** mode.



NOTE:

This is a quick-search function that will find the first match in the search direction that you specify. You can use the Advanced Editor for a more full-featured search. Refer to page 166 for more information on the Advanced Editor search function.

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

The **[UP]** cursor arrow searches toward the start of the program from the current cursor position. The **[DOWN]** cursor arrow searches toward the end of the program. The first match found appears highlighted.

3.6 RS-232

RS-232 is one way of connecting the Haas CNC control to a computer. This feature enables the programmer to 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.

3.6.1 Cable Length

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

T3.1: Cable Length

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

3.6.2 Machine Data Collection

Machine Data Collection is enabled by Setting 143, which allows the user to extract data from the control using a Q command sent through the RS-232 port (or by using an optional hardware package). This feature is software-based and requires an additional computer to request, interpret, and store data from the control. 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 following output format is used:

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

- *STX* (0x02) marks the start of data. This control character is for the remote computer.
- *CSV 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 >. The following commands are available:

T3.2: 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

The user has the ability to request the contents of any macro or system variable by using the **Q600** command, for example, **Q600 xxxx**. This will display the contents of macro variable **xxxx** on the remote computer. In addition, macro variables #1-33, 100-199, 500-699 (note that variables #550-580 are unavailable if the mill is equipped with a probing system), 800-999 and #2001 thru #2800 can be written to using an **E** command, for example, **Exxxx yyyyyy.yyyyyy** where **xxxx** is the macro variable and **yyyyyy.yyyyyy** is the new value.

**NOTE:**

This command should be used only when there are no alarms present.

Data Collection Using Optional Hardware

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

Once installed, output relays 40 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.

The following machine statuses will be 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

3.7 File Numeric 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.

3.8 Direct Numeric Control (DNC)

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

F3.4: DNC Waiting and Received Program

PROGRAM (DNC)	N00000000
WAITING FOR DNC . . .	
DNC RS232	
	<pre> PROGRAM (DNC) N00000000 ;I ;001000 ; ;(G-CODE FINAL QC TEST CUT) ; ;(MATERIAL IS 2x8x8 6061 ALUMINUM) ; ; ; ;(MAIN) ; ; M00 ; (READ DIRECTIONS FOR PARAMETERS AND SETTINGS) ; (FOR VF-SERIES MACHINES WITH 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>

T3.3: Recommended RS-232 Settings for DNC

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

1. DNC is enabled using Parameter 57 bit 18 and Setting 55. Turn the parameter bit on (1) and change Setting 55 to **ON**.
2. It is recommended that DNC be run with XMODEM or parity selected because an error in transmission will then be detected and will stop the DNC program without crashing. The settings between the CNC control and the other computer must match. To change the setting in the CNC control, press [**SETTING/GRAFIC**] and scroll to the RS-232 settings (or enter 11 and press the up or down arrow).
3. Use the [**UP**] and [**DOWN**] cursor arrows to highlight the variables and the left and right arrows to change the values.
4. Press [**ENTER**] when the proper selection is highlighted.
5. 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.
6. 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.
7. Start sending the program to the control before [**CYCLE START**] is pushed. Once the message *DNC Prog Found* is displayed, Press [**CYCLE START**].

3.8.1 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 will perform one block (command) at a time. Each block will be performed immediately with no block look-ahead. The exception is when Cutter Compensation is commanded. Cutter Compensation requires three blocks of motion commands to be read prior to a compensated block being performed.

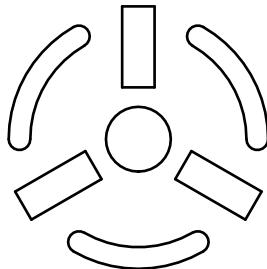
Full duplex communication during DNC is possible by using the **G102** command or **DPRNT** to output axes coordinates back to the controlling computer.

3.9 Part Setup

It is necessary to properly secure the part. Refer to the workholding manufacturer's manual for the proper procedure for fixturing a workpiece.

3.9.1 Chuck Foot Pedal

F3.5: Chuck Foot Pedal Icon



NOTE:

Dual-spindle lathes have a pedal for each chuck. The relative positions of the pedals indicate the chuck that they control (i.e., the left-hand pedal controls the main spindle and the right-hand pedal controls the secondary spindle).

When you press this pedal, the automatic chuck clamps or unclamps, equivalent to an M10 / M11 command for the main spindle, or M110 / M111 command for the secondary spindle. This allows you to operate the spindle hands-free while you load or unload a workpiece.

The ID / OD clamp settings for the main and secondary spindles apply when you use this pedal (refer to Setting 92 on page **408** and Setting 122 on page **414** for more information).

Use Setting 76 to enable or disable all pedal controls. Refer to page **405** for more information.

3.9.2 Chuck/Drawtube Warnings



WARNING:

Check the workpiece in the chuck or collet after any power loss. A power outage reduces the clamping pressure on the workpiece which can shift in the chuck or collet. Setting 216 turns off the Hydraulic pump after the time specified for the setting.



Damage results if you attach dead length stops to the hydraulic cylinder.



Do not machine parts larger than the chuck.



Follow all of the warnings of the chuck manufacturer.

3



Hydraulic pressure must be set correctly.



See the Hydraulic System Information on the machine for safe operation. Setting a pressure beyond the recommendations damages the machine and/or inadequately holds the workpiece.



Chuck jaws must not protrude beyond the diameter of the chuck.



Improperly or inadequately clamped parts eject with deadly force.



Do not exceed rated chuck RPM.



Higher RPM reduces chuck clamping force. See the following chart.

Maximum force	Total gripping force of all three jaws at maximum pressure	Maximum operating pressures		
(kgf) lbs (18144) 40000 (15876) 35000 (13608) 30000 (11338) 25000 (9070) 20000 (6803) 15000 (4535) 10000 (2268) 5000 0	(kgf) lbs (18144) 40000 (15876) 35000 (13608) 30000 (11338) 25000 (9070) 20000 (6803) 15000 (4535) 10000 (2268) 5000 0	PSI (kgf/cm ²)	5" Chuck	330 (23)

5" Chuck	330	(23)
6" Chuck	330	(23)
8" Chuck	330	(23)
10" Chuck	330	(23)
12" Chuck	400	(28)
15" Chuck	300	(21)
18" Chuck	300	(21)
Tailstock	400	(28)



NOTE: Chucks must be greased weekly and kept free from debris.

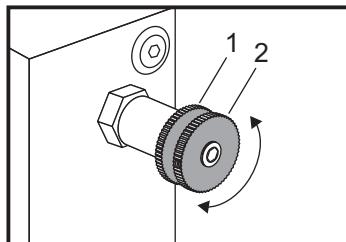
3.9.3 Drawtube Operation

The hydraulic unit provides the pressure necessary to clamp a part.

Clamping Force Adjustment Procedure

To adjust the clamping force on the drawtube:

F3.6: Draw tube Clamping Force Adjustment: [1] Locking knob, [2] Adjustment knob.

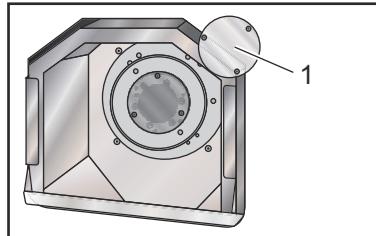


1. Go to Setting 92 on the **Settings** page and choose either I.D. clamping or O.D. clamping. Do not do this with a program running.
2. Turn the locking knob [1] counterclockwise to loosen.
3. Turn the adjustment knob [2] until the gauge reads the desired pressure. Turn clockwise to increase pressure. Turn counterclockwise to decrease the pressure.
4. Turn the locking knob [1] clockwise to tighten.

Drawtube Cover Plate

Before using the Bar Feeder,

F3.7: Draw Tube Cover Plate [1].



1. Remove the cover plate [1] at the far end of the drawtube.
2. Replace the cover plate anytime bar stock is not being fed automatically.

3.9.4 Chuck and Collet Replacement

These procedures describe how to remove and replace a chuck or collet.

For detailed instructions on the procedures listed in this section, go to www.HaasCNC.com and select the **Resource Center**.

Chuck Installation

To install a chuck:



NOTE:

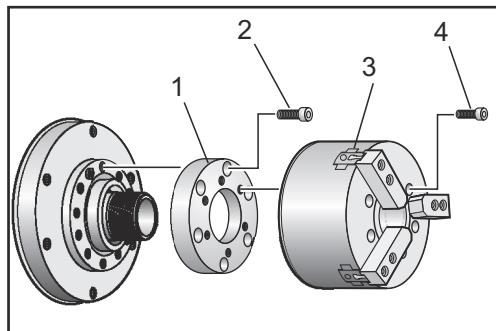
If necessary, install an adapter plate before installing the chuck.

1. Clean the face of the spindle and the back face of the chuck. Position the drive dog at the top of the spindle.
2. Remove the jaws from the chuck. Remove the center cup or coverplate from the front of the chuck. If available, install a mounting guide into the drawtube and slide the chuck over it.
3. Orient the chuck so that one of the guide holes are aligned with the drive dog. Use the chuck wrench to thread the chuck onto the drawtube.
4. Screw the chuck all the way onto the drawtube and back it off 1/4 turn. Align the drive dog with one of the holes in the chuck. Tighten the six (6) SHCS.
5. Install the center cup or plate with three (3) SHCS.
6. Install the jaws. If necessary replace the rear cover plate. This is located on the left side of the machine.

Chuck Removal

This is a summary of the chuck removal process.

F3.8: Chuck Removal Illustration: [1] Chuck Adapter Plate, [2] 6X SHCS, [3] Chuck, [4] 6X SHCS.



1. Move both axes to their zero positions. Remove the chuck jaws.
2. Remove the three (3) screws that mount the center cup (or plate) from the center of the chuck and remove the cup.



CAUTION: *You must clamp the chuck when you do this next step, or you will damage the drawtube threads.*

3. Clamp the chuck [3] and remove the six (6) SHCS [4] that mount the chuck to the spindle nose or adapter plate.

4. Unclamp the chuck. Place a chuck wrench inside the center bore of the chuck and unscrew the chuck from the drawtube. If equipped, remove adapter plate [1].



WARNING: *The chuck is heavy. Be prepared to use lifting equipment to support the chuck as it is removed.*

Collet Installation

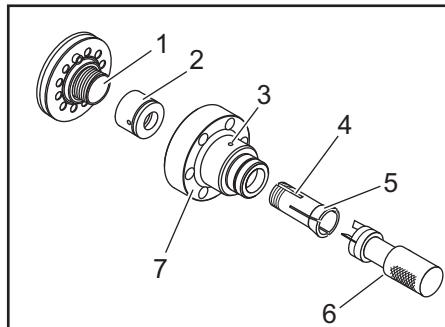
To install a collet:

1. Thread the collet adapter into the drawtube.
2. Place the spindle nose on the spindle and align one of the holes on the back of the spindle nose with the drive dog.
3. Fasten the spindle nose to the spindle with six (6) SHCS.
4. Thread the collet onto the spindle nose and align the slot on the collet with the set screw on the spindle nose. Tighten the setscrew on the side of the spindle nose.

Collet Removal

To remove the collet:

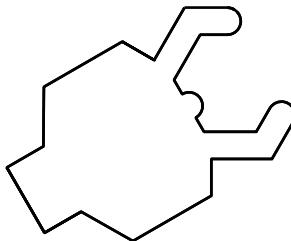
F3.9: Collet Removal Illustration: [1] Draw Tube, [2] Collet adapter, [3] Set Screw, [4] Set Screw Slot, [5] Collet, [6] Collet wrench, [7] Spindle Nose.



1. Loosen the set screw [3] on the side of the spindle nose [7]. Using the collet wrench [6], unscrew the collet [5] from the spindle nose [7].
2. Remove the six (6) SHCS from the spindle nose [7] and remove it.
3. Remove the collet adapter [2] from the drawtube [1].

3.9.5 Steady Rest Foot Pedal

F3.10: Steady Rest Foot Pedal Icon



When you press this pedal, the hydraulic steady rest clamps or unclamps, equivalent to the M-code commands that control the steady rest (M59 P1155 to clamp, M60 P1155 to unclamp). This allows you to operate the steady rest hands-free while you handle the workpiece.

Use Setting 76 to enable or disable all pedal controls. Refer to page **405** for more information.

3.10 Tailstock Setup and Operation

The tailstock is used to support the end of a turning workpiece. It runs along two linear guides. Tailstock motion is controlled through program code, in jog mode, or by a foot pedal.



NOTE:

The tailstock is not field-installable.

Tailstocks are controlled with hydraulic pressure in ST-10 (quill only), ST-20, and ST-30 models.

In ST-40 models, the tailstock is positioned and hold force applied by a servo motor.

The tailstock is engaged when the tailstock quill is against the workpiece, applying the specified force.

3.10.1 Tailstock Types

There are three basic types of tailstock: hydraulic quill, hydraulic positioned, and servo. The type of tailstock you have depends on the lathe model, and each type has different operation characteristics.

ST-10 Tailstock Operation

3

In the ST-10, you position the tailstock manually and activate a lock lever to hold it in place.

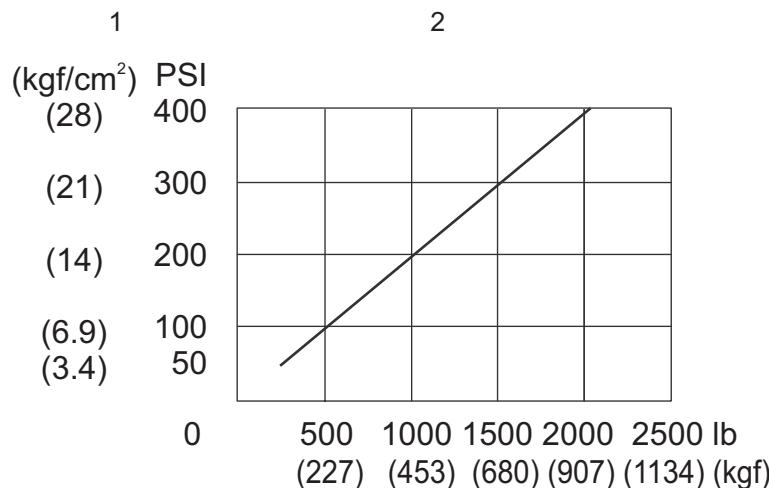


CAUTION: *Be sure to move the tailstock when necessary to avoid a collision.*

The ST-10 tailstock has a fixed head, and a quill with 4" (102 mm) of travel. Therefore, the only automatically-moving part is the quill. Adjust the hydraulic pressure at the HPU to control quill hold force. Refer to the chart in Figure F3.11.

You cannot move the tailstock quill with the **[HANDLE JOG]** control or the Remote Jog Handle. Also, **[POWER UP/RESTART]** or **[ZERO RETURN]** and **[ALL]** do not move the tailstock quill. The ST-10 tailstock does not have an axis assignment.

F3.11: ST-10 Hydraulic Quill Force: [1] Maximum pressure, [2] Hydraulic Quill Force.

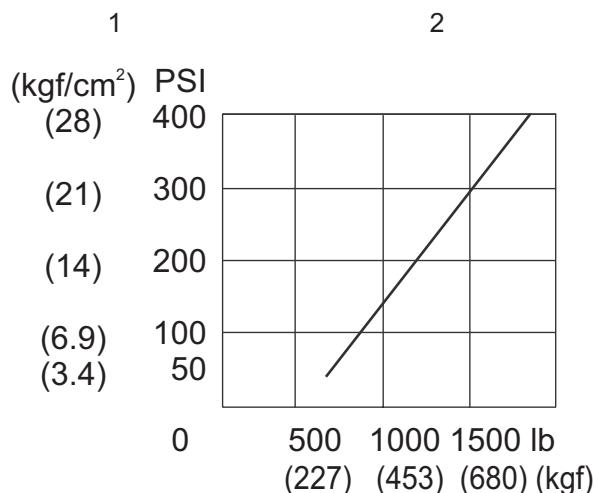


Hydraulic Tailstock (ST-20/30)

In ST-20 and ST-30 model lathes, a hydraulic cylinder positions the tailstock and applies hold force to the workpiece.

Adjust hydraulic pressure at the HPU to control tailstock hold force. Refer to the chart in Figure F3.12 to determine the pressure setting for the hold force you need.

F3.12: ST-20/30 Tailstock Pressure Chart: [1] Maximum pressure, [2] Tailstock Hold Force.



Recommended minimum hydraulic tailstock operating pressure is 120 psi. If hydraulic pressure is set lower than 120 psi, the tailstock may not function reliably.



NOTE:

During machine operation, [FEED HOLD] does not stop hydraulic tailstock motion. You must press [RESET] or [EMERGENCY STOP].

Startup Procedure

If power to the lathe is shut off or interrupted while the hydraulic tailstock is engaged with a workpiece, the hold force is lost. Support the workpiece and zero return the tailstock to resume operation when power is restored.

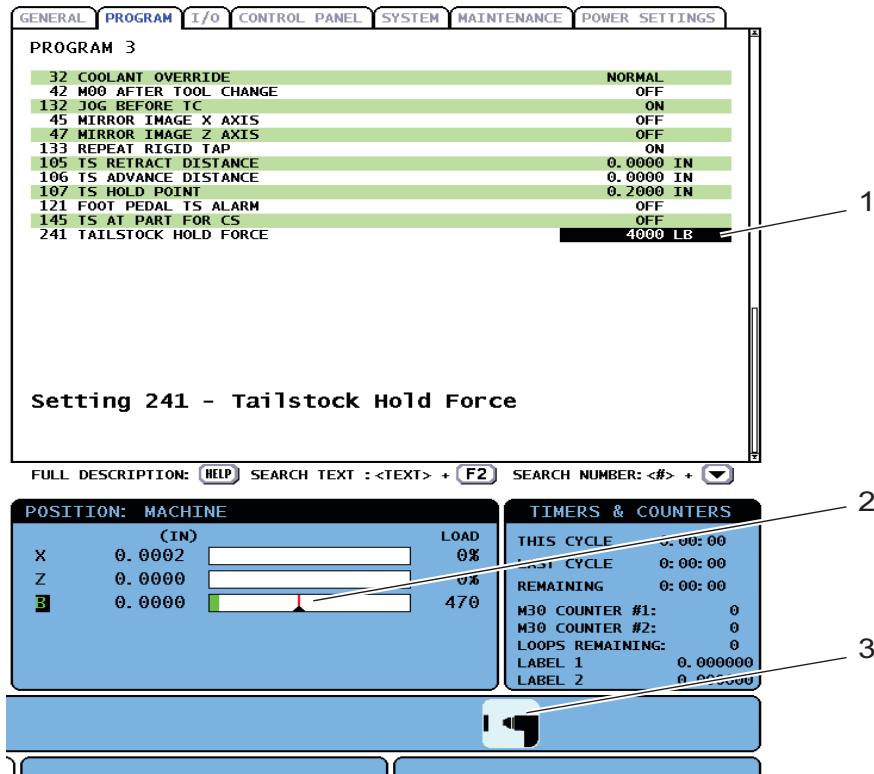
ST-40 Servo Tailstock Operation

In ST-40 model lathes, a servo motor positions the tailstock and applies hold force to the workpiece.

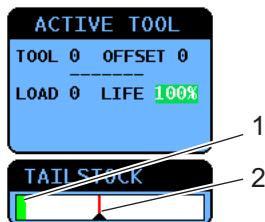
Change Setting 241 to control servo tailstock hold force. Use a value between 1000 and 4500 pounds-force (if Setting 9 is INCH) or 4450 and 20110 Newtons (if Setting 9 is MM).

Tailstock load and current hold force are displayed as the B Axis in the axis load display (in modes such as **MDI** and **MEM**). The bar graph indicates current load, and the red line indicates the maximum hold force value specified in Setting 241. Actual hold force is displayed next to the bar graph. In **Jog** mode, this display appears in the **Active Tool** pane.

F3.13: Maximum Hold Force [1], B-Axis Gauge [2], and Tailstock Hold Icon [3]



A hold icon [3] displays whether or not the tailstock is engaged. Refer to page 36 for more information on the tailstock hold icon.

F3.14: Force Gauge Actual Pressure [1] and Maximum Pressure [2] Indicators

Startup Procedure

If power to the lathe is shut off or interrupted while the servo tailstock is engaged with a workpiece, the servo brake engages to preserve hold force and keep the tailstock in place.

When power is restored, the control displays the message *Tailstock Force Restored*. You can resume operating the lathe without zero returning the tailstock, if there are no M22 commands in the program. These commands cause the tailstock to back away from the workpiece, which could then drop.

**CAUTION:**

Before you resume a program with an M22 command after a power interruption, edit the program to remove or block delete the tailstock motion commands. You can then resume the program and complete the part. Keep in mind that until you zero return the tailstock, the control does not know the tailstock's position; therefore, Settings 93 and 94 do not protect the tailstock restricted zone from a crash.

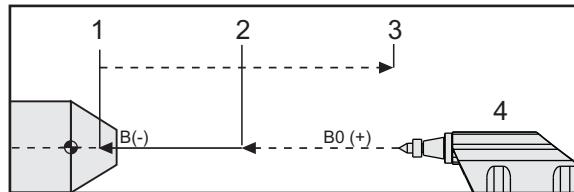
Zero return the tailstock before starting a new cycle on a new workpiece. You can then add the tailstock motion commands back into the program for future cycles.

The first use of the tailstock foot pedal after a power interruption zero returns the tailstock. Make sure the workpiece is supported before activating the tailstock foot pedal.

3.10.2 ST-20/30/40 Tailstock Operation

The ST-20/30/40 tailstock operation includes Settings, M-codes, Foot Pedal, and jogging functions.

F3.15: Setting 105 [3], 106 [2], 107 [1], and [4] Home position.



Setting 105 - Retract Point [3] and Setting 106 - Advance Point [2] are Relative to Setting 107 - Hold Point [1]. Setting 107 is absolute. Settings 105 and 106 are incremental from Setting 107.

Tailstock Settings

Tailstock motion is defined by three settings:

- **Hold Point (Setting 107):** The point at which hold force is applied. No default value. This setting has a negative value.
- **Advance Point (Setting 106):** The distance from the hold point through which the tailstock moves at feed speed. The value is relative to Setting 107, and contains a default value that varies by lathe model. This setting has a positive value.
- **Retract Point (Setting 105):** The distance from the advance point through which the tailstock moves at rapid speed. Value is relative to Setting 107, and contains a default value that varies by lathe model. This setting has a positive value.

Settings 105 and 106 have default values based on the lathe model. If desired, enter new values in inches (when Setting 9 is **INCH**) or millimeters (when Setting 9 is **MM**).



NOTE:

These settings are defined relative to Setting 107, and not absolute machine position.



NOTE:

Settings 105, 106, and 107 do not apply to the ST-10 tailstock, since it is positioned manually.

Tailstock Hold Point Creation (Setting 107)

To set the Tailstock Hold Point (Setting 107):

1. Select the B Axis in **Jog** mode.
2. Jog the tailstock to the workpiece, until the center contacts the workpiece surface.
3. Add 0.25" (6 mm) to the value in the **Machine Position** display for the B Axis and record this value.
4. Enter the value from step 3 in Setting 107.

Tailstock Advance/Retract Point (Setting 106/105)

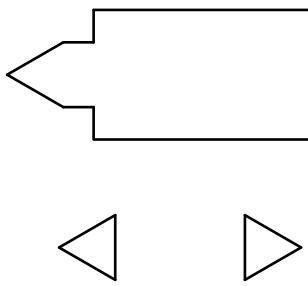
Settings 106 Advance Point and 105 Retract Point have default values based on the lathe model. You can enter new values in inches (when Setting 9 is **INCH**) or millimeters (when Setting 9 is **MM**).



REMEMBER: *These settings are defined relative to Setting 107, and not absolute machine position.*

Tailstock Foot Pedal Operation

F3.16: Tailstock Foot Pedal Icon



When you press this pedal, the tailstock (or the tailstock quill) moves toward or away from the spindle, equivalent to an M21 or M22 command, depending on the current position. If the tailstock is away from the retract point, the foot pedal moves the tailstock toward the retract point (M22). If the tailstock is at the retract point, the foot pedal moves the tailstock toward the hold point (M21).

If you press the foot pedal while the tailstock is in motion, the tailstock stops and a new sequence must begin.

Press and hold the pedal for 5 seconds to retract the tailstock quill the full distance and maintain retract pressure. This makes sure the tailstock quill does not creep forward. Use this method to stow the tailstock quill any time it is not in use.

**NOTE:**

The tailstock position can change over time if it is left at a position that is not fully retracted or not in contact with a workpiece. This is due to normal hydraulic system leakage.

Use Setting 76 to enable or disable all pedal controls. Refer to page **405** for more information.

3.10.3 Tailstock Restricted Zone

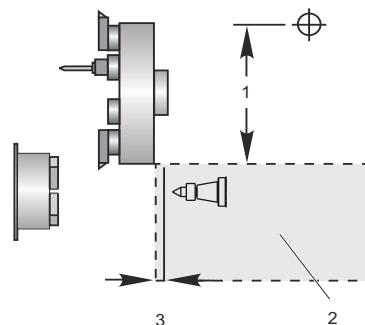
Setting up the tailstock involves setting a tailstock Restricted Zone.

Use Setting 93 and Setting 94 to make sure that the turret or any tools in the turret do not collide with the tailstock. Test the limits after you change these settings.

These settings make a restricted zone. The restricted zone is a protected rectangular area at the lower right of the lathe work space. The restricted zone changes so that the Z-Axis and tailstock maintain a safe distance from each other when below a specified X-Axis clearance plane.

Setting 93 specifies the X-Axis clearance plane and Setting 94 specifies the separation between the Z Axis and the B Axis (tailstock axis). If a programmed motion crosses the restricted zone, a warning message appears.

F3.17: [2] Tailstock Restricted Zone, [1]Setting 93, [3]Setting 94.



X Clearance Plane (Setting 93)

To set a value for the X clearance plane (Setting 93):

1. Place the control in **MDI** mode.
2. Select the longest tool that protrudes furthest on the X-axis plane in the turret.
3. Place the control in **Jog** mode.
4. Select the X-axis for jogging and move the X-axis clear of the tailstock.
5. Select the tailstock (B-axis) for jogging and move the tailstock beneath the selected tool.
6. Select the X-axis and approach the tailstock until the tool and tailstock are about 0.25" apart.
7. Enter this value for Setting 93 in the X-axis **Machine Position** on the display. Back the tool away in the X Axis a small amount before entering the value in Setting 93.

Z- and B-Axis below the X Clearance Plane (Setting 94)

To set a separation for Z- and B-Axis below the X Clearance Plane (Setting 94):

1. Press **[ZERO RETURN]** and **[HOME G28]**.
2. Select the X-axis and move the turret in front of the tailstock quill tip.
3. Move the Z-axis so that the rear of the tool turret is within about 0.25" of the tailstock quill tip.
4. Enter the value in the Z-Axis **Machine Position** display for Setting 94.

Canceling a Restricted Zone.

A restricted zone is not always desired (when setting up, for example). To cancel a restricted zone:

1. Enter a **0** in Setting 94.
2. Enter maximum X-Axis machine travel in Setting 93.

3.10.4 Jogging the Tailstock

**CAUTION:**

Do not use an M21 in the program if the tailstock is positioned manually. If this is done, the tailstock backs away from the workpiece and then repositions against the workpiece, which may cause the workpiece to drop. When a servo tailstock restores hold force after a power interruption, the tailstock is considered manually positioned since the control does not know the tailstock position until it is zero returned.

The ST-40 servo tailstock cannot be jogged while engaged with a workpiece or while the spindle is running.

To jog the tailstock:

1. Select **Jog** mode.
2. Press **[TS <—]** to jog the tailstock at feed speed toward the chuck, or press **[TS —>]** to jog the tailstock at feed speed away from the chuck.
3. Press **[TS RAPID]** and **[TS <—]** simultaneously to move the tailstock at rapid speed toward the chuck. Or, press **[TS RAPID]** and **[TS —>]** simultaneously to move the tailstock at rapid speed away from the chuck. The control reverts to the last jogged axis when the keys are released.

3.11 Tooling

The **Tnn** code is used to select the tool to be used in a program.

3.11.1 Jog Mode

Jog Mode allows you to jog each of the axes to a desired location. Before jogging the axes it is necessary to home (beginning axes reference point) the axes.

To enter jog mode:

1. Press **[HANDLE JOG]**.
2. Pick an increment speed to be used while in jog mode (**[.0001]**, **[.001]**, **[.01]** or **[.1]**).
3. Press the desired axis (**[+X]**, **[-X]**, **[+Z]**, or **[-Z]**) and either press and hold these axis jog keys or use the **[HANDLE JOG]** control to move the selected axis.

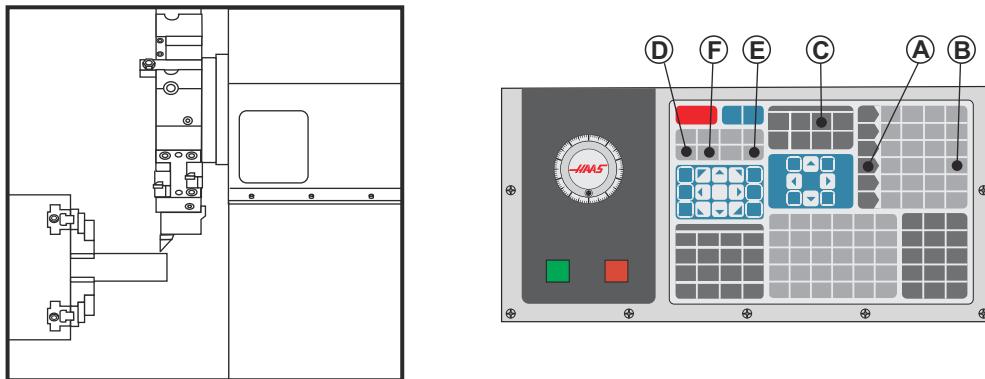
3.11.2 Setting the Tool Offset

The next step is to touch off the tools. Doing this defines the distance from the tip of the tool to the side of the part. This procedure requires the following:

- An O.D. Turning Tool
- A work piece that fits in the chuck jaws
- A measuring tool to inspect the workpiece diameter

For information on setting up Live tools, refer to page **243**.

F3.18: Lathe Tool Offset



1. Load an O.D. turning tool into the tool turret.
2. Clamp the workpiece in the spindle.
3. Press **[HANDLE JOG]** [A].
4. Press **[.1/100] [B]**. The axis selected moves at a fast rate when the handle is turned.
5. Close the lathe door. Type 50 and press **[FWD]** for the spindle to start.
6. Use the turning tool loaded in station 1 to make a small cut on the diameter of the material clamped in the spindle. Approach the part carefully and feed slowly during the cut.
7. After the small cut is done, jog away from the part using Z-axis. Move far enough away from the part so that you can take a measurement with your measuring tool.
8. Press Spindle **[STOP]** and open the door.

9. Use the measuring tool to measure the cut made on the workpiece
10. Press **[X DIAMETER MEASURE]** to record the X-axis position in the offset table.
11. Type the workpiece diameter and press **[ENTER]** to add it to the X-axis offset. The offset that corresponds to the tool and turret station is recorded.
12. Close the lathe door. Type 50 and press **[FWD]** for the spindle to start.
13. Use the turning tool loaded in station 1 to make a small cut on the face of the material clamped in the spindle. Approach the part carefully and feed slowly during the cut.
14. After the small cut is done, jog away from the part using X-axis. Move far enough away from the part so that you can take a measurement with your measuring tool.
15. Press **[Z FACE MEASURE]** (E) to record the current Z position in the offset table.
16. The cursor moves to the Z-axis location for the tool.
17. Repeat all of the previous steps for each tool in the program. Do tool changes at a safe location with no obstructions.

3.11.3 Manually Set the Tool Offset

Offsets can be entered manually by:

1. Choose one of the tool offsets pages.
2. Move the cursor to the desired column.
3. Type a number and press **[ENTER]** or **[F1]**.

Pressing **[F1]** enters the number in the selected column. Entering a value and pressing **[ENTER]** adds the amount entered to the number in the selected column.

3.11.4 Hybrid Turret, VDI, and BOT Centerline Offset

To set the X offset to centerline for tools:

1. Press **[HANDLE JOG]** and enter the **Tool Geometry** offset page.
2. Select the X Offset column and press **[F2]**.

For BOT (Bolt-On) turrets: Pressing **[F2]** sets an X-axis I.D. Tool Offset on center for a 1" (25 mm) I.D. BOT tool. Adjust offset manually for other size tooling or after-market toolholders.

For VDI (Verein Deutscher Ingenieure) turrets: Pressing **[F2]** sets an X-axis tool offset on center of the VDI40 stations.

For Hybrid (BOT and VDI40 combination) turrets: Pressing **[F2]** sets an X-axis tool offset on center of the VDI40 stations.

3.11.5 Additional Tooling Set-up

There are other tool set-up 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 5) 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.

3.12 Setting Part (Work Piece) Zero for Z-axis (Part Face)

The CNC control programs all move from Part Zero, a user defined reference point. To set Part Zero:

1. Select Tool #1 by pressing **[MDI/DNC]**.
2. Enter **T1** and press **[TURRET FWD]**.
3. Jog X and Z until the tool is just touching the face of the part.
4. Press **[OFFSET]** until **Work Zero Offset** display is active. Highlight the **Z Axis** column and G-code row desired (G54 recommended).
5. Press **[Z FACE MEASURE]** to set part zero.

3.13 Features

Some of the Haas Turning Center features include:

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

3.13.1 Graphics Mode

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

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

1. Press **[SETTING/GRAFIC]** until the **GRAPHICS** 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, press **[MDI/DNC]** until DNC mode is active, then go to graphics display and send the program to the machine's control (See the DNC section).
3. There are three helpful display features in Graphics mode that can be accessed by pressing **[F1] - [F4]**. **[F1]** is the help button, which will give a short description of each of the functions possible in graphics mode. **[F2]** is the zoom button, which highlights an area using the arrow buttons, **[PAGE UP]** and **[PAGE DOWN]** to control the zoom level, and pressing the **[ENTER]** button. **[F3]** and **[F4]** are used to control the simulation speed.



NOTE:

Not all machine functions or motions are simulated in graphics.

3.13.2 Dry Run Operation

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



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 (see the previous section on Graphics Mode).

1. Dry Run is selected by pressing **[DRY RUN]** while in **MEM** or **MDI** mode. When in Dry Run, all rapids and feeds are run at the speed selected with the jog speed keys. Dry Run makes all of the requested tool changes. Overrides keys adjust the Spindle speeds in Dry Run.
2. Dry Run is only turned on or off when a program is completely finished or **[RESET]** is pressed.

3.13.3 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.

3.13.4 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 M30), 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.*

3.13.5 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.

3.13.6 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.

3.14 Run-Stop-Jog-Continue

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

1. Press [FEED HOLD] to stop the running program.
2. Press [X] or [Z] followed by [HANDLE JOG]. The control stores the current X and Z positions.

**NOTE:**

Axes other than X and Z cannot be jogged.

3. The control displays the message *Jog Away*. Use [HANDLE JOG] control, remote jog handle, [+X]/[-X], [+Z]/[-Z], or [RAPID] to move the tool away from the part. The spindle is controlled by pressing [FWD], [REV], or [STOP]. If necessary, tool inserts can be changed.



CAUTION:

When the program is continued, the old offsets are used for the return position. Therefore, it is unsafe and not recommended to change tools and offsets when the program is interrupted.

4. Jog to a position as close as possible to the stored position, or to a position where there is an unobstructed rapid path back to the stored position.
5. Return to the previous mode by pressing [MEMORY], or [MDI/DNC]. The control only continues if the mode that was in effect when the machine stopped is re-entered.
6. Press [CYCLE START]. The control displays the message *Jog Return and rapid X and Z at 5% to the position where Feed Hold was pressed, then return the Z-axis.*



CAUTION:

*The control does not follow the path used to jog away. If [FEED HOLD] is pressed during this motion, the axes motion pauses and the control displays the message *Jog Return Hold*. Pressing [CYCLE START] causes the control to resume the *Jog Return* motion. When the motion is completed, the control again goes into a feed hold state.*

7. Press [CYCLE START] again and the program resumes normal operation. Also refer to Setting 36 on page 399.

3.15 Program Optimizer

This feature allows you to override spindle speed, axis feed, and coolant positions (for a mill) in a program, as the program runs. Once the program is finished, the Program Optimizer highlights the program blocks that you changed and allows you to make the change 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].

3.15.1 Program Optimizer Operation

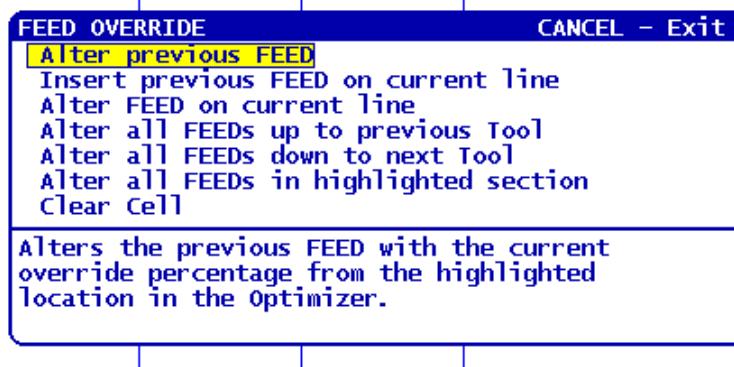
To go to the Program Optimizer screen:

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

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.

F3.19: Program Optimizer Screen: Feed Override Popup Example (Mill Screen Shown)



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

3.16 Advanced Tool Management

F3.20: Advanced Tool Management Display: [1] Tool group window, [2] Allowed limits window, [3] Tool data window, [4] Help text.

ADVANCED TOOL MANAGEMENT		CURRENT GROUP: 12345	TOOL 1 IN POSITION
		PRESS F4 TO CHANGE ACTIVE WINDOW	
GROUP ID:	12345	GROUPS	1 of 1
<PREVIOUS>	<NEXT>	<ADD>	<DELETE>
SEARCH			
GROUP USAGE :	IN ORDER	USAGE :	0
DESCRIPTION :		FEED TIME :	0
		TOTAL TIME :	0
		TOOL LOAD :	0
		TL ACTION :	ALARM
1		2	
TOOL# EXP LIFE		GEOMETRY X GEOMETRY Z RADIUS TIP	
0		WEAR X WEAR Z	
0			
0			
0			
0		FEED TIME TOTAL TIME USAGE LOAD	
3		4	
WRITE/ENTER to display the previous tool group's data.			

Advanced Tool Management (ATM) allows the user to set up and access duplicate tools for the same or a series of jobs.

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

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

The Advanced Tool Management page is located in the Current Commands mode.

1. Press **[CURRENT COMMANDS]**.
2. Press **[PAGE UP]** until you access the Advanced Tool Management page.

3.16.1 Navigation

The ATM interface uses three separate windows where data is entered: The tool group window, allowed limits window, and the tool data window (this window includes both the tool list on the left and tool data on the right).

The lower area of the screen displays help information for the item currently selected in the active window.

1. Press **[F4]** to switch between windows.
2. Use the Cursor Arrow Keys to move between fields in the active window.
3. Depending on the item selected press **[ENTER]** to modify or clear values.

3.16.2 Tool Group Setup

To add a tool group:

1. Press **[F4]** until the **Tool Group** window is active.
2. Use the cursor arrows to highlight **<ADD>**.
3. Enter a five-digit tool group ID number between 10000 and 30000.
4. Press **[F4]** again to add data for the tool group to the **Allowed Limits** window.
5. Add tools to the group in the **Tool Data** window.

3.16.3 Operation

To operate Advanced Tool Management, you need to set up your tools using the following five procedures:

- Tool Group Setup
- Tool Group
- Allowed Limits
- Tools Table
- Tool Data
- Tool Group Usage

3.16.4 Macros

Macro variables 8550-8567 enable a G code program to obtain individual tool information. When an individual tool ID number is specified using macro 8550, the control returns the individual tool information in macro variables 8551-8567. Additionally, a user can specify an ATM group number using macro 8550. In this circumstance, the control returns the individual tool information for the current tool in the specified ATM tool group using macro variables 8551-8567. Refer to page 224 in the Programming chapter for macro variable data information. The values in these macros provide data that is also accessible from macros 2001, 2101, 2201, 2301, 2701, 2801, 2901, 5401, 5501, 5601, 5701, 5801, and 5901. Macros 8551-8567 provide access to the same data, but for tools 1-50 for all data items. Any future increase in the total number of tools is accessible through 8551-8567.

3.16.5 Tips & Tricks

Comment out tool details to keep them in the program while using ATM groups. These tool details can include tool numbers in the group, tool type, operator instructions, etc. For example:

```
...
G00 G53 X0 Z#508 ;
(T100 PRIMARY TOOL ATM GROUP 10000) (Comment: tool and tool
group) ;
(T300 SECONDARY TOOL SAME GROUP) (Comment: secondary tool) ;
G50 S3500 T10000 (T101) (Comment out T call and replace with
tool group) ;
G97 S550 T10000 (T101) ;
G97 S1200 M08 ;
G00 Z1. ;
X2.85 ;
```

...

3.17 Tool Turret Operations

To operate the tool turret, refer to the following sections: Air Pressure, Eccentric Locating Cam Buttons, Protective Cap, and Tool Load or Tool Change.

3.17.1 Air Pressure

Low air pressure or insufficient air volume reduces the pressure applied to the turret clamp/unclamp piston. This can slow down the turret index time, or the turret may not unclamp.

3.17.2 Eccentric Locating Cam Buttons

Bolt-on turrets are equipped with eccentric locating cam buttons that allow for fine alignment of ID tool holders to spindle center line.

Mount the tool holder to the turret and align the tool holder to the spindle in the X-axis. Measure the alignment in the Y-axis. If necessary remove the tool holder and use a narrow tool in the cam button hole, to rotate the eccentric to correct misalignment.

The following table gives the result for specific positions of the cam button.

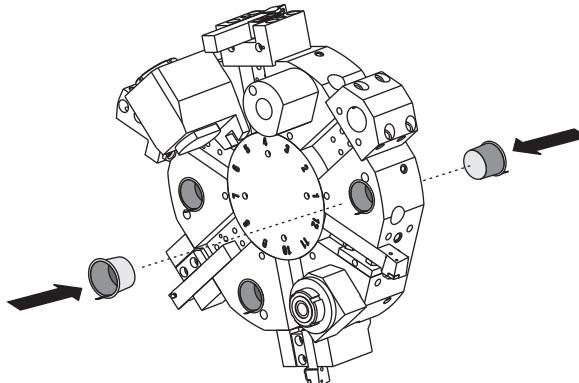
Rotation (degrees)	Result
0	no change
15	0.0018" (0.046 mm)
30	0.0035" (0.089 mm)
45	0.0050" (0.127 mm)
60	0.0060" (0.152 mm)
75	0.0067" (0.170 mm)
90	0.0070" (0.178 mm)

3.17.3 Protective Cap



IMPORTANT: Insert protective caps into empty turret pockets to protect them from accumulating debris.

F3.21: Turret Protective Caps in Empty Pockets



To load or change tools:

3.17.4 Tool Load or Tool Change

To load or change tools:



NOTE:

Y-Axis lathes return the turret to the zero position (spindle centerline) after a tool change.

1. Enter **MDI** mode.
2. Optional: Type the tool number that you want to change to in the format **Tnn**.
3. Press **[TURRET FWD]** or **[TURRET REV]**.

If you specified a tool number, the turret indexes to that turret position. Otherwise, the turret indexes to the next or the previous tool.

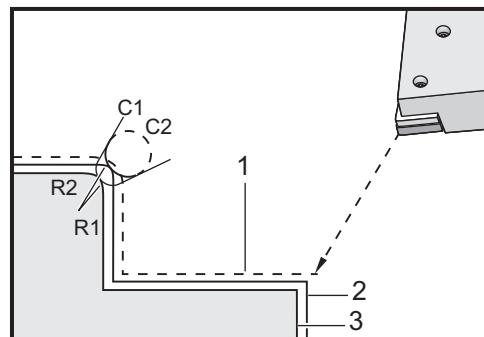
3.18 Tool Nose Compensation

Tool Nose Compensation (TNC) is a feature that allows the user to adjust a programmed tool path in response to different cutter sizes or for normal cutter wear. The user can do this by entering minimal offset data at run-time without any additional programming effort.

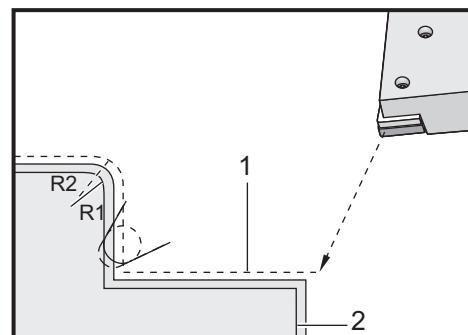
3.18.1 Programming

Tool Nose Compensation is used when the tool nose radius changes, and cutter wear is to be accounted for with curved surfaces or tapered cuts. Tool Nose Compensation generally does not need to be used when programmed cuts are solely along the X- or Z-axis. For taper and circular cuts, as the tool nose radius changes, under or overcutting can occur. In the figure, suppose that immediately after setup, C₁ is the radius of the cutter that cuts the programmed tool path. As the cutter wears to C₂, the operator might adjust the tool geometry offset to bring the part length and diameter to dimension. If this were done, a smaller radius would occur. If tool nose compensation is used, a correct cut is achieved. The control automatically adjusts the programmed path based on the offset for tool nose radius as set up in the control. The control alters or generates code to cut the proper part geometry.

F3.22: Cutting path without tool nose compensation: [1] Tool Path, [2] Cut after wear [3] Desired cut.



F3.23: Cutting path with tool nose compensation: [1] Compensated tool path, [2] Desired cut and programmed tool path.



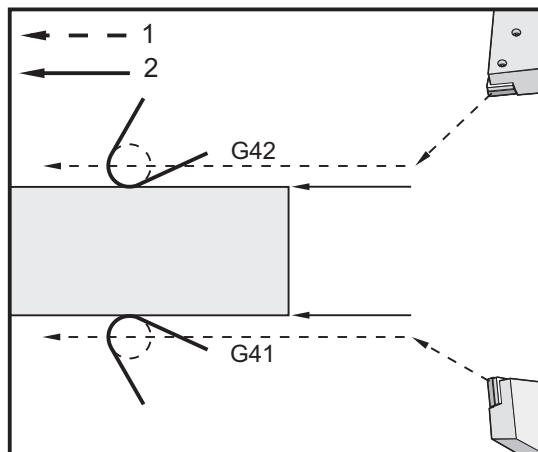
**NOTE:**

The second programmed path coincides with the final part dimension. Although parts do not have to be programmed using tool nose compensation, it is the preferred method because it makes program problems easier to detect and resolve.

3.18.2 Tool Nose Compensation Concept

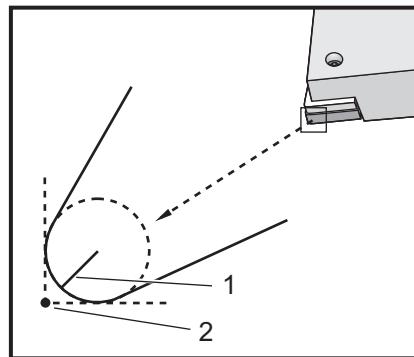
Tool nose compensation works by shifting the Programmed Tool Path to the right or to the left. The programmer usually programs the tool path to the finished size. When tool nose compensation is used, the control compensates for the radius of the tool based on special instructions written into the program. Two G-code commands are used to do this for compensation within a two-dimensional plane. G41 commands the control to shift to the left of the programmed tool path, and G42 commands the control to shift to the right of the programmed tool path. Another command, G40, is provided to cancel any shift made by tool nose compensation.

F3.24: TNC Shift Direction: [1] Toolpath relative to the workpiece, [2] Programmed toolpath.



The shift direction is based on the direction of the tool movement relative to the tool and which side of the part it is on. When thinking about which direction the compensated shift occurs in tool nose compensation, imagine looking down the tool tip and steering the tool. Commanding G41 moves the tool tip to the left and G42 moves the tool tip to the right. This means that normal O.D. turning requires G42 for correct tool compensation, while normal I.D. turning requires G41.

F3.25: Imaginary tool tip: [1] Tool nose radius, [2] Imaginary tool tip.



Tool nose compensation assumes that a compensated tool has a radius at the tool tip that it must compensate for. This is called the Tool Nose Radius. Since it is difficult to determine exactly where the center of this radius is, a tool is usually set up using what is called the Imaginary Tool Tip. The control also needs to know which direction the tool tip is relative to the center of the tool nose radius, or the Tip direction. The tip direction should be specified for each tool.

The first compensated move is generally a move from a non-compensated position to a compensated position and is therefore unusual. This first move is called the Approach move and is required when using tool nose compensation. Similarly, a Depart move is required. In a Depart move, the control moves from a compensated position to a non-compensated position. A Depart move occurs when tool nose compensation is canceled with a G40 command or Txx00 command. Although Approach and Depart moves can be precisely planned, they are generally uncontrolled moves and the tool should not be in contact with the part when they occur.

3.18.3 Using Tool Nose Compensation

The following steps are used to program a part using TNC:

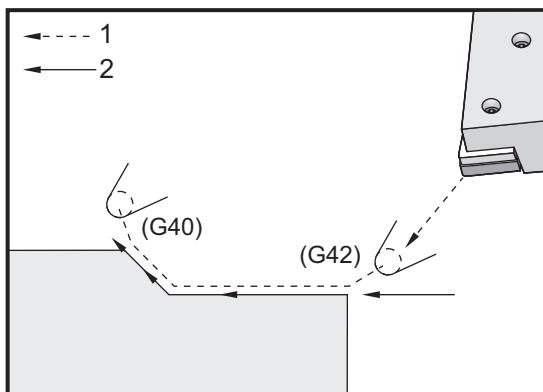
1. **Program** the part to finished dimensions.
2. **Approach and Departure** – Ensure that there is an approach move for each compensated path and determine which direction (G41 or G42) is used. Ensure that there is also a departure move for each compensated path.
3. **Tool Nose Radius and Wear** – Select a standard insert (tool with radius) to be used for each tool. Set the tool nose radius of each compensated tool. Clear the corresponding tool nose wear offset to zero for each tool.
4. **Tool Tip Direction** – Input the tool tip direction for each tool that is using compensation, G41 or G42.

5. **Tool Geometry Offset** – Set the tool length geometry and clear the length wear offsets of each tool.
6. **Check Compensation Geometry** – Debug the program in graphics mode and correct any tool nose compensation geometry problems that may occur. A problem can be detected in two ways: an alarm is generated indicating compensation interference, or the incorrect geometry is seen generated in graphics mode.
7. **Run and Inspect First Article** – Adjust compensated wear for the setup part.

3.18.4 Approach and Departure Moves For TNC

The first X or Z motion in the same line that contains a G41 or G42 is called the Approach move. The approach must be a linear move, that is a G01 or G00. The first move is not compensated, yet at the end of the approach move the machine position is fully compensated. See the following figure.

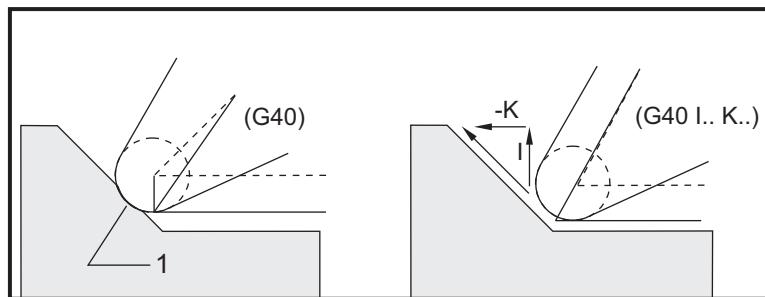
F3.26: TNC Approach and Depart Moves: [1] Compensated Path, [2] Programmed path.



Any line of code with a G40 cancels tool nose compensation and is called the Departure move. The departure must be a linear move, that is a G01 or G00. The start of a departure move is fully compensated; the position at this point is at a right angle to the last programmed block. At the end of the departure move the machine position is not compensated. See the previous figure.

The following figure shows the condition just prior to canceling tool nose compensation. Some geometries result in over or undercutting of the part. This is controlled by including an I and K address code in the G40 cancellation block. The I and K in a G40 block define a vector that is used to determine the compensated target position of the previous block. The vector is usually aligned with an edge or wall of the completed part. The following figure shows how I and K correct undesired cutting in a departure move.

F3.27: TNC Use of I and K in G40 Block: [1] Overcut.



3.18.5 Tool Nose Radius and Wear Offset

Each turning tool that uses tool nose compensation requires a Tool Nose Radius. The tool tip (tool nose radius) specifies how much the control is to compensate for a given tool. If standard inserts are being used for the tool, then the tool nose radius is simply the tool tip radius of the insert.

Associated with each tool on the geometry offsets page is a Tool Nose Radius Offset. The column labeled **Radius** contains the value for the tool nose radius of each tool. If the value of any tool nose radius offset is set to zero, no compensation is generated for that tool.

Associated with each radius offset is a Radius Wear Offset, located on the **Wear Offset** page. The control adds the wear offset to the radius offset to obtain an effective radius that is used for generating compensated values.

Small adjustments (positive values) to the radius offset during production runs should be placed in the wear offset page. This allows the operator to easily track the wear for a given tool. As a tool is used, the insert generally wears so that there is a larger radius at the end of the tool. When replacing a worn tool with a new one, clear the wear offset to zero.

It is important to remember that tool nose compensation values are in terms of radius rather than diameter. This is important when tool nose compensation is canceled. If the incremental distance of a compensated departure move is not twice the radius of the cutting tool, overcutting occurs. Always remember that programmed paths are in terms of diameter and allow for twice the tool radius on departure moves. The Q block of canned cycles that require a **PQ** sequence is often a departure move. The following example illustrates how incorrect programming results in overcutting.

Preparation:

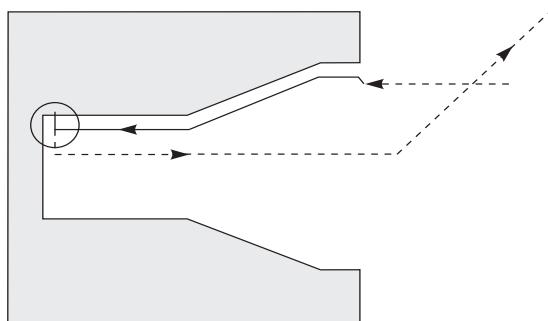
- Setting 33 is FANUC

Tool Geometry	X	Z	Radius	Tip
8	-8.0000	-8.00000	.0160	2

Example:

```
%  
O0010 ;  
G28 ;  
T808 ; (Boring bar)  
G97 S2400 M03 ;  
G54 G00 X.49 Z.05;  
G41 G01 X.5156 F.004 ;  
Z-.05 ;  
X.3438 Z-.25  
Z-.5 ;  
X.33; (Move less than .032. Required to avoid cut-in with a  
departure move before TNC is cancelled.)  
G40 G00 X.25 ;  
Z.05 ;  
G53 X0;  
G53 Z0;  
M30 ;  
%
```

F3.28: TNC Departure Cutting Error



3.18.6 Tool Nose Compensation and Tool Length Geometry

The length geometries of tools that use tool nose compensation are set up in the same manner as tools not using compensation. Refer to page 103 for details on touching off tools and recording tool length geometries. When a new tool is set up, the geometry wear should be cleared to zero.

Often a tool exhibits uneven wear. This occurs when particularly heavy cuts occur on one edge of the tool. In this case it is desirable to adjust the **X or Z Geometry Wear** rather than the **Radius Wear**. By adjusting X or Z length geometry wear, the operator can often compensate for uneven tool nose wear. Length geometry wear shifts all dimensions for a single axis.

The program design may not allow the operator to compensate for wear by using length geometry shift. Which wear to adjust can be determined by checking several X and Z dimensions on a finished part. Wear that is even results in similar dimensional changes on the X and Z axes, and suggests that the radius wear offset should be increased. Wear that affects the dimensions on one axis only suggests length geometry wear.

Good program design based on the geometry of the part being cut should eliminate uneven wear problems. Generally, rely on finishing tools that use the entire radius of the cutter for tool nose compensation.

3.18.7 Tool Nose Compensation in Canned Cycles

Some canned cycles ignore tool nose compensation, expect a specific coding structure, or perform their own specific canned cycle activity (also refer to page 279 for more information on using canned cycles).

The following canned cycles ignore tool nose radius compensation. Cancel tool nose compensation before any of these canned cycles:

- G74 End face grooving cycle, peck drilling
- G75 O.D./I.D. grooving cycle, peck drilling
- G76 Thread cutting cycle, multiple pass
- G92 Thread cutting cycle, modal

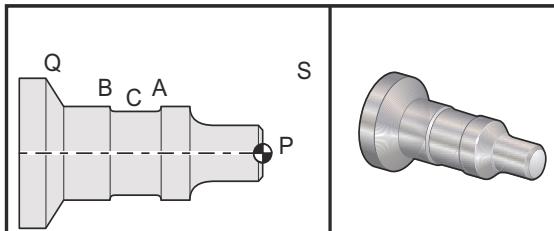
3.18.8 Example Programs Using Tool Nose Compensation

This section gives several examples of programs that use Tool Nose Compensation.

Example 1: TNC Standard Interpolation Modes G01/G02/G03

This example of general TNC uses standard interpolation modes G01/G02/G03.

F3.29: TNC Standard Interpolation G01, G02, and G03



Preparation

- Turn Setting 33 to FANUC.
- Set up the following tools:
 - T1 Insert with .0312 radius, roughing
 - T2 Insert with .0312 radius, finishing
 - T3 .250 wide grooving tool with .016 radius/same tool for offsets 3 and 13

Tool	Offset	X	Z	Radius	Tip
T1	01	-8.9650	-12.8470	.0312	3
T2	02	-8.9010	-12.8450	.0312	3
T3	03	-8.8400	-12.8380	.016	3
T3	13	"	-12.588	.016	4

Program Example:

```
%  
O0811 (G42 Test BCA Example 1) ;  
N1 G50 S1000 ;  
T101 (Tool 1, Offset 1. Tip direction for offset 1 is 3) ;  
G97 S500 M03 ;  
G54 G00 X2.1 Z0.1 (Move to point S) ;  
G96 S200 ;  
G71 P10 Q20 U0.02 W0.005 D.1 F0.015 (Rough P to Q with T1 using  
G71 and TNC. Define part path PQ sequence) ;  
N10 G42 G00 X0. Z0.1 F.01 (P)(G71 Type II, TNC right) ;
```

```
G01 Z0 F.005 ;
X0.65 ;
X0.75 Z-0.05 ;
Z-0.75 ;
G02 X1.25 Z-1. R0.25 ;
G01 Z-1.5 (A) ;
G02 X1. Z-1.625 R0.125 ;
G01 Z-2.5
G02 X1.25 Z-2.625 R0.125 (B) ;
G01 Z-3.5 ;
X2. Z-3.75 ;
N20 G00 G40 X2.1 (TNC Cancel) ;
G97 S500 ;
G53 X0 (Zero for tool change clearance) ;
G53 Z0 ;
M01 ;
N2 G50 S1000 ;
T202 ;
G97 S750 M03 (Tool 2, Offset 2. Tip direction is 3) ;
G00 X2.1 Z0.1 (move to point S) ;
G96 S400 G70 P10 Q20 (Finish P to Q with T2 using G70 and TNC)
;
G97 S750 ;
G53 X0 (Zero for tool change clearance) ;
G53 Z0 ;
M01 ;
N3 G50 S1000 ;
T303 (Tool 3, Offset 3. Tip direction is 3) ;
G97 S500 M03 (Groove to point B Using Offset 3) ;
G54 G42 X1.5 Z-2.0 (Move to point C TNC right) ;
G96 S200 ;
G01 X1. F0.003 ;
G01 Z-2.5 ;
G02 X1.25 Z-2.625 R0.125 (B) ;
G40 G01 X1.5 (TNC cancel - Groove to point A using offset 4) ;
T313 (Change offset to other side of tool) ;
G00 G41 X1.5 Z-2.125 (Move to point C - TNC approach) ;
G01 X1. F0.003 ;
G01 Z-1.625 ;
G03 X1.25 Z-1.5 R0.125 (A) ;
G40 G01 X1.6 (TNC cancel) ;
G97 S500 ;
G53 X0 ;
G53 Z0 ;
M30 ;
%
```

**NOTE:**

The suggested template of the previous section for G70 is used. Also note that compensation is enabled in the PQ sequence but is canceled after G70 is completed.

Example 2: TNC with a G71 Roughing Canned Cycle

This example is using TNC with a G71 roughing canned cycle.

Preparation:

- Turn Setting 33 to FANUC.
- Tools:
T1 Insert with .032 radius, roughing

Tool	Offset	Radius	Tip
T1	01	.032	3

Program Example:

```
%  
O0813 (Example 2) ;  
G50 S1000 ;  
T101 (Select tool 1) ;  
G00 X3.0 Z.1 (Rapid to start point) ;  
G96 S100 M03 ;  
G71 P80 Q180 U.01 W.005 D.08 F.012 (Rough P to Q with T1 using  
G71 and TNC. Define Part Path PQ sequence) ;  
N80 G42 G00 X0.6 (P) (G71 Type I, TNC right) ;  
G01 Z0 F0.01 (Start of finish part path) ;  
X0.8 Z-0.1 F0.005 ;  
Z-0.5 ;  
G02 X1.0 Z-0.6 I0.1 ;  
G01 X1.5 ;  
X2.0 Z-0.85 ;  
Z-1.6 ;  
X2.3 ;  
G03 X2.8 Z-1.85 K-0.25 ;  
G01 Z-2.1(Q) (End of part path) ;  
N180 G40 G00 X3.0 M05 (TNC cancel) ;  
G53 X0 (Zero X for tool change clearance) ;  
G53 Z0 ;
```

```
M30 ;
%
```

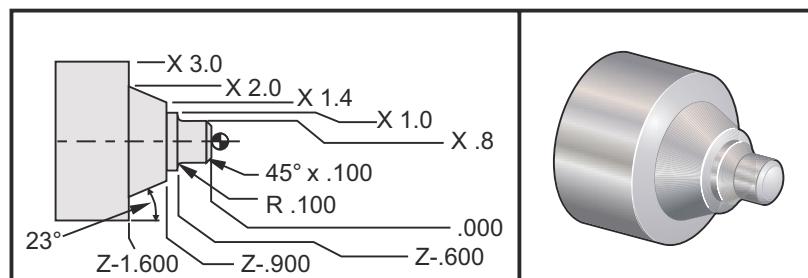
**NOTE:**

This part is a G71 Type I path. When using TNC it is very unusual to have a Type II path, as the compensation methods can only compensate the tool tip in one direction.

Example 3: TNC with a G72 Roughing Canned Cycle

This example is TNC with a G72 roughing canned cycle. G72 is used instead of G71 because the roughing strokes in X are longer than the Z roughing strokes of a G71. It is therefore more efficient to use G72.

F3.30: TNC G72 Roughing Canned Cycle



Operation	Tool	Offset	Tool Nose Radius	Tip
roughing	T1	01	0.032	3
finishing	T2	02	0.016	3

Setting 33: FANUC

Program Example:

```
%  
O0813 (Example 3) ;  
G50 S1000 ;  
T101 (Select tool 1) ;  
G00 X3.0 Z.1 (Rapid to start point) ;  
G96 S100 M03 ;  
G71 P80 Q180 U.01 W.005 D.08 F.012 (Rough P to Q with T1 using
```

```
G71 and TNC. Define Part Path PQ sequence) ;
N80 G42 G00 X0.6 (P) (G71 Type I, TNC right) ;
G01 Z0 F0.01 (Start of finish part path) ;
X0.8 Z-0.1 F0.005 ;
Z-0.5 ;
G02 X1.0 Z-0.6 I0.1 ;
G01 X1.5 ;
X2.0 Z-0.85 ;
Z-1.6 ;
X2.3 ;
G03 X2.8 Z-1.85 K-0.25 ;
G01 Z-2.1(Q) (End of part path) ;
N180 G40 G00 X3.0 M05 (TNC cancel) ;
G53 X0 (Zero X for tool change clearance) ;
G53 Z0 ;
M30 ;
%
```

Example 4: TNC with G73 Roughing Canned Cycle

This example is TNC with a G73 roughing canned cycle. G73 is best used when you want to remove a consistent amount of material in both the X and Z axes.

Preparation:

- Turn Setting 33 to FANUC
- Tools:
T1 Insert with .032 radius, roughing
T2 Insert with .016 radius, finishing

Tool	Offset	Radius	Tip
T1	01	.032	3
T2	02	.016	3

Program Example:

```
%  
00815 (Example 4) ;  
T101 (Select Tool 1) ;  
G50 S1000 ;  
G00 X3.5 Z.1 (Move to point S) ;  
G96 S100 M03 ;
```

```

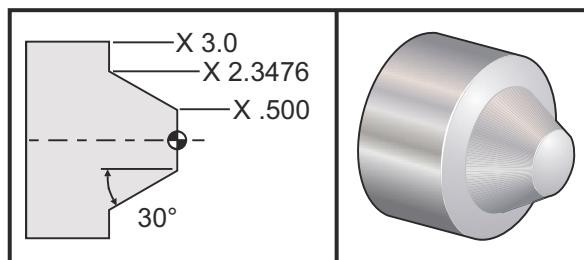
G73 P80 Q180 U.01 W0.005 I0.3 K0.15 D4 F.012 (Rough P to Q
with T1 using G73 and TNC) ;
N80 G42 G00 X0.6 (Part path PQ sequence, G72 Type I, TNC right)
;
G01 Z0 F0.1 ;
X0.8 Z-0.1 F.005 ;
Z-0.5 ;
G02 X1.0 Z-0.6 I0.1 ;
G01 X1.4 ;
X2.0 Z-0.9 ;
Z-1.6 ;
X2.3 ;
G03 X2.8 Z-1.85 K-0.25 ;
G01 Z-2.1 ;
N180 G40 X3.1 (Q) ;
G00 Z0.1 M05 (TNC Cancel) ;
(*****Optional Finishing Sequence*****);
G53 X0 (Zero for tool change clearance) ;
G53 Z0 ;
M01 ;
T202 (Select tool 2) ;
N2 G50 S1000 ;
G00 X3.0 Z0.1 (Move to start point) ;
G96 S100 M03 ;
G70 P80 Q180 (Finish P to Q with T2 using G70 and TNC) ;
G00 Z0.5 M05 ;
G28 (Zero for tool change clearance) ;
M30 ;
%

```

Example 5: TNC with G90 Modal Rough Turning Cycle

This example is TNC with a G90 modal rough turning cycle.

F3.31: TNC With G90 Rough Turning Cycle



Operation	Tool	Offset	Tool Nose Radius	Tip
roughing	T1	01	0.032	3

Setting 33: FANUC

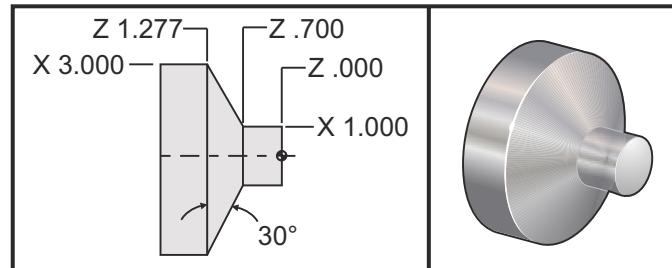
Program Example:

```
%  
O0816 (Example 5) ;  
T101 (Select tool 1) ;  
G50 S1000 ;  
G00 X4.0 Z0.1 (Move to start point) ;  
G96 S100 M03 ;  
(ROUGH 30 DEG. ANGLE TO X2. AND Z-1.5 USING G90 AND TNC) ;  
G90 G42 X2.55 Z-1.5 I-0.9238 F0.012 ;  
X2.45 (Optional Additional Passes) ;  
X2.3476 ;  
G00 G40 X3.0 Z0.1 M05 (TNC Cancel) ;  
G53 X0 (Zero for tool change clearance) ;  
G53 Z0 ;  
M30 ;  
%
```

Example 6: TNC with G94 Modal Rough Turning Cycle

This example is TNC with a G94 modal rough turning cycle.

F3.32: TNC G94 Rough Turning Cycle



Operation	Tool	Offset	Tool Nose Radius	Tip
roughing	T1	01	0.032	3

Setting 33: FANUC

Program Example:

```
%  
O0817 (Example 6) ;  
G50 S1000 ;  
T101 (Select tool 1) ;  
G00 X3.0 Z0.1 (Move to start point) ;  
G96 S100 M03 ;  
G94 G41 X1.0 Z-0.5 K-0.577 F.03 (Rough 30° angle to X1. and  
Z-0.7 using G94 and TNC) ;  
Z-0.6 (Optional additional passes) ;  
Z-0.7 ;  
G00 G40 X3. Z0.1 M05 (TNC Cancel) ;  
G53 X0 (Zero for tool change clearance) ;  
G53 Z0 ;  
M30 ;  
%
```

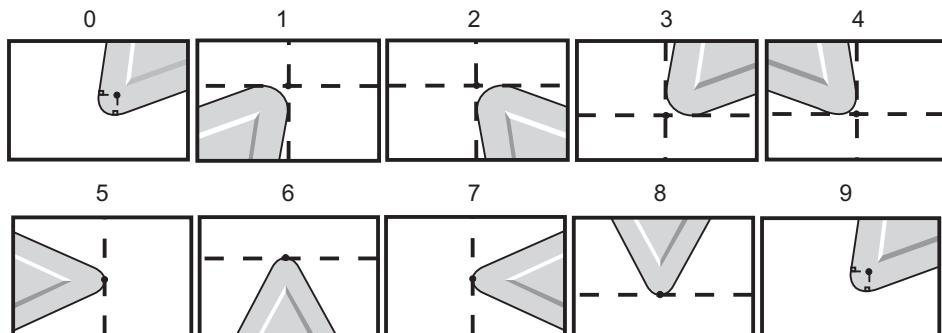
3.18.9 Imaginary Tool Tip and Direction

It is not easy to determine the center of a tool radius on a lathe. The cutting edges are set when a tool is touched off to record tool geometry. The control calculates where the center of the tool radius is by using the edge information, the tool radius, and the direction the cutter is expected to cut in. The X- and Z-axis geometry offsets intersect at a point, called the Imaginary Tool Tip, that aids in determining the tool tip direction. The Tool Tip Direction is determined by a vector originating from the center of the tool radius and extending to the imaginary tool tip, see the following figures.

The tool tip direction of each tool is coded as a single integer from 0 to 9. The tip direction code is found next to the radius offset on the geometry offsets page. It is recommended that a tip direction be specified for all tools using tool nose compensation. The following figure is a summary of the tip coding scheme along with cutter orientation examples.

**NOTE:**

The tip indicates to the setup person how the programmer intends to measure the tool offset geometry. For example, if the setup sheet shows tip direction 8, the programmer intends the tool geometry to be at the edge of and on the centerline of the tool insert.

F3.33: Tip Codes and Center Location

Tip Code	Tool Center Location
0	No specified direction. 0 is not usually used when Tool Nose Compensation is desired.
1	Direction X+, Z+: Off tool
2	Direction X+, Z-: Off tool
3	Direction X-, Z-: Off tool
4	Direction X-, Z+: Off tool
5	Direction Z+: Tool edge
6	Direction X+: Tool edge
7	Direction Z-: Tool edge
8	Direction X-: Tool edge
9	Same as Tip 0

3.18.10 Programming Without Tool Nose Compensation

Without TNC you can manually calculate the compensation and use various tool nose geometries described in the following sections.

3.18.11 Manually Calculating Compensation

When programming a straight line in either X or Z axes the tool tip touches the part at the same point where you touched your original tool offsets in X- and Z-axes. However, when you program a chamfer or an angle, the tip does not touch the part at those same points. Where the tip actually touches the part is dependent upon the degree of angle being cut and also the size of the tool insert. Overcutting or undercutting occurs when programming a part without any compensation.

The following pages contain tables and illustrations demonstrating how to calculate the compensation in order to program the part accurately.

Along with each chart are three examples of compensation using both types of inserts and cutting along three different angles. Next to each illustration is a sample program and explanation of how the compensation is calculated.

Refer to the illustrations on the following pages.

The tool tip is shown as a circle with X and Z points called out. These points designate where the X diameter and Z face offsets are touched off.

Each illustration is a 3" diameter part with lines extending from the part and intersecting at 30°, 45°, and 60° angles.

The point at which the tool tip intersects the lines is where the compensation value is measured.

The compensation value is the distance from the face of the tool tip to the corner of the part. Notice that the tool tip is slightly offset from the actual corner of the part; this is so the tool tip is in the correct position to make the next move and to avoid any overcutting or undercutting.

Use the values found on the charts (angle and radius size) to calculate the correct tool path position for the program.

3.18.12 Tool Nose Compensation Geometry

The following figure shows the various geometries of tool nose compensation. It is organized into four categories of intersection. The intersections can be:

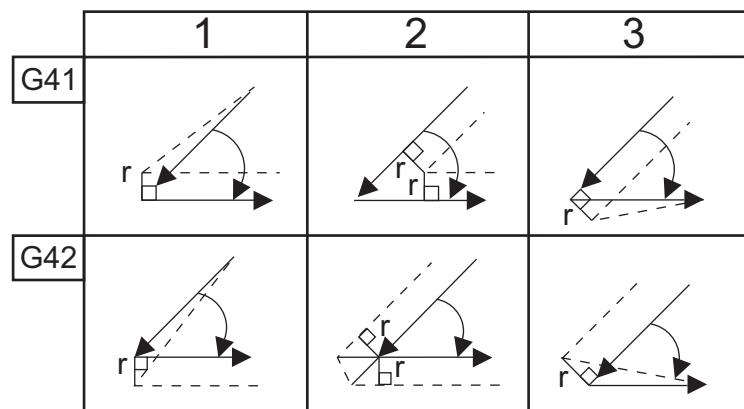
1. linear to linear
2. linear to circular
3. circular to linear
4. circular to circular

Beyond these categories the intersections are classified into angle of intersection and approach, mode to mode, or departure motions.

Two FANUC compensation types are supported, Type A and Type B. The default compensation is Type A.

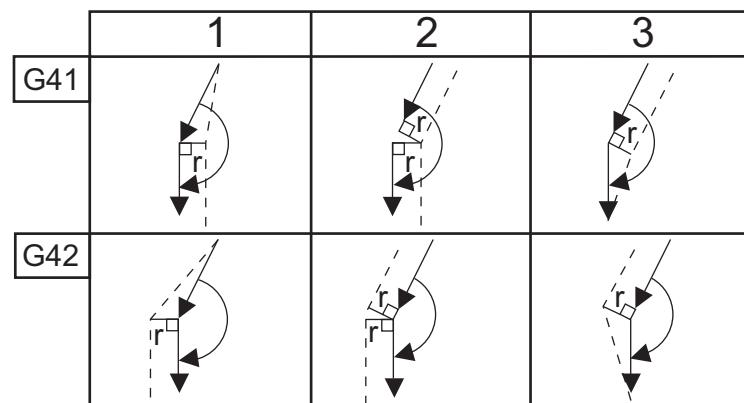
F3.34: TNC Linear-to-Linear (Type A): [1] Approach, [2], Mode to mode, [3] Departure.

<90

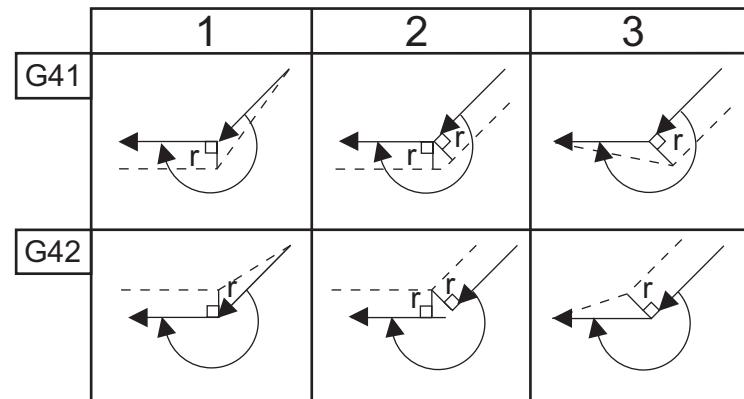


3

>=90, <180

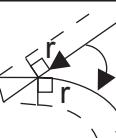


>180

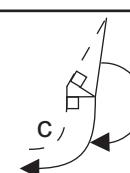


F3.35: TNC Linear-to-Circular (Type A): [1] Approach, [2], Mode to mode, [3] Departure.

<90

	1	2	3
G41			
G42			

>=90, <180

	1	2	3
G41			
G42			

>180

	1	2	3
G41			
G42			

F3.36: TNC Circular-to-Linear (Type A): [1] Approach, [2], Mode to mode, [3] Departure.

<90

	1	2	3
G41			
G42			

3

>=90, <180

	1	2	3
G41			
G42			

>180

	1	2	3
G41			
G42			

Tool Radius And Angle Chart (1/32 RADIUS)

The X measurement calculated is based on part diameter.

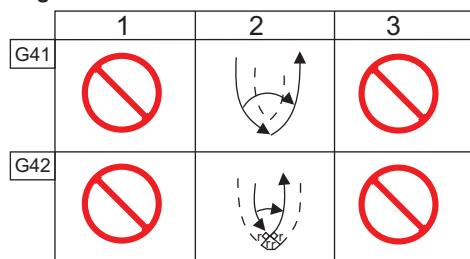
ANGLE	Xc CROSS	Zc LONGITUDINAL	ANGLE	Xc CROSS	Zc LONGITUDINAL
1.	.0010	0310	46.	.0372	.0180
2.	.0022	.0307	47.	.0378	.0177
3.	.0032	.0304	48.	.0386	.0173
4.	.0042	.0302	49.	.0392	.0170
5.	.0052	.0299	50.	.0398	.0167
6.	.0062	.0296	51.	.0404	.0163
7.	.0072	.0293	52.	.0410	.0160
8.	.0082	.0291	53.	.0416	.0157
9.	.0092	.0288	54.	.0422	.0153
10.	.01	.0285	55.	.0428	.0150
11.	.0011	.0282	56.	.0434	.0146
12.	.0118	.0280	57.	.0440	.0143
13.	.0128	.0277	58.	.0446	.0139
14.	.0136	.0274	59.	.0452	.0136
15.	.0146	.0271	60.	.0458	.0132
16.	.0154	.0269	61.	.0464	.0128
17.	.0162	.0266	62.	.047	.0125
18.	.017	.0263	63.	.0474	.0121
19.	.018	.0260	64.	.0480	.0117
20.	.0188	.0257	65.	.0486	.0113

ANGLE	Xc CROSS	Zc LONGITUDINAL	ANGLE	Xc CROSS	Zc LONGITUDINAL
21.	.0196	.0255	66.	.0492	.0110
22.	.0204	.0252	67.	.0498	.0106
23.	.0212	.0249	68.	.0504	.0102
24.	.022	.0246	69.	.051	.0098
25.	.0226	.0243	70.	.0514	.0094
26.	.0234	.0240	71.	.052	.0090
27.	.0242	.0237	72.	.0526	.0085
28.	.025	.0235	73.	.0532	.0081
29.	.0256	.0232	74.	.0538	.0077
30.	.0264	.0229	75.	.0542	.0073
31.	.0272	.0226	76.	.0548	.0068
32.	.0278	.0223	77.	.0554	.0064
33.	.0286	.0220	78.	.056	.0059
34.	.0252	.0217	79.	.0564	.0055
35.	.03	.0214	80.	.057	.0050
36.	.0306	.0211	81.	.0576	.0046
37.	.0314	.0208	82.	.0582	.0041
38.	.032	.0205	83.	.0586	.0036
39.	.0326	.0202	84.	.0592	.0031
40.	.0334	.0199	85.	.0598	.0026
41.	.034	.0196	86.	.0604	.0021
42.	.0346	.0193	87.	.0608	.0016
43.	.0354	.0189	88.	.0614	.0011

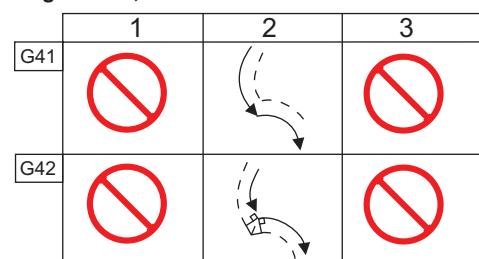
ANGLE	Xc CROSS	Zc LONGITUDINAL	ANGLE	Xc CROSS	Zc LONGITUDINAL
44.	.036	.0186	89.	.062	.0005
45.	.0366	.0183			

F3.37: TNC Circular-to-Circular (Type A): [1] Approach, [2], Mode to mode, [3] Departure.

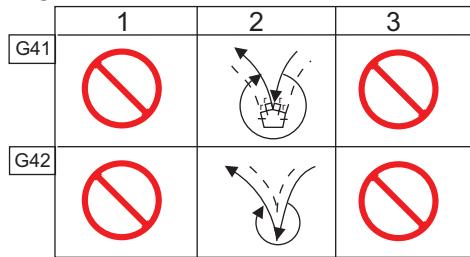
Angle: <90



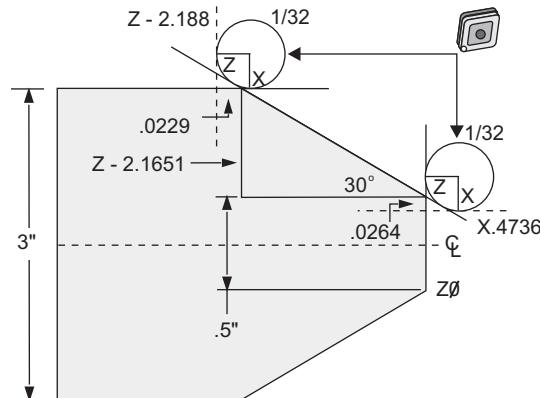
Angle: >=90, <180



Angle: >180

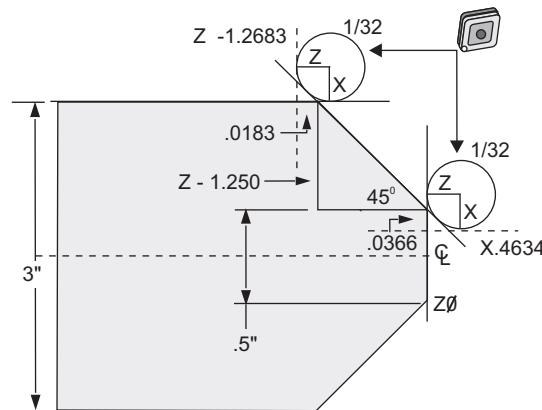


F3.38: Tool Nose Radius Calculation, 1/32, Compensation value for 30 degree angle.



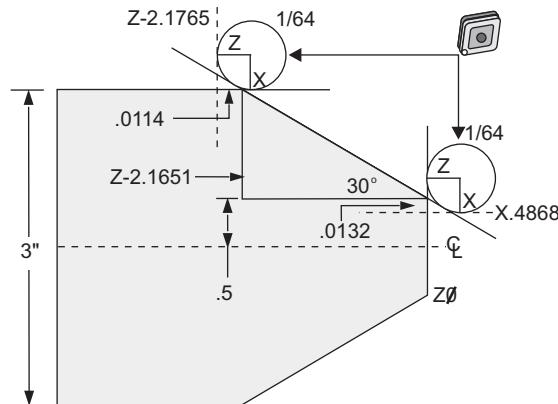
Code	Compensation (1/32 tool nose radius)
G0 X0 Z.1	
G1 Z0	
X.4736	(X.5-0.0264 compensation)
X 3.0 Z-2.188	(Z-2.1651+0.0229 compensation)

F3.39: Tool Nose Radius Calculation, 1/32, Compensation value for 45 degree angle.



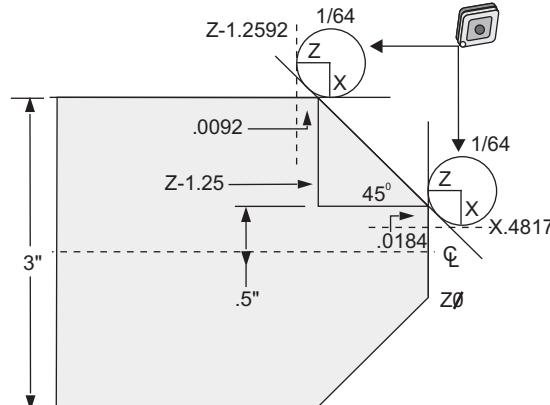
Code	Compensation (1/32 tool nose radius)
G0 X0 Z.1	
G1 Z0	
X.4634	(X.5-0.0366 compensation)
X 3.0 Z-1.2683	(Z-1.250+0.0183 compensation)

F3.40: Tool Nose Radius Calculation, 1/64, Compensation value for 30 degree angle.



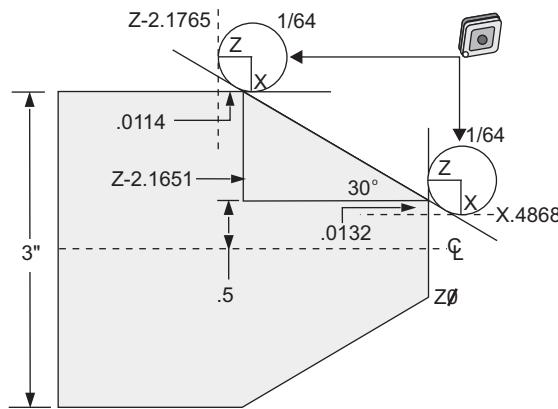
Code	Compensation (1/64 tool nose radius)
G0 X0 Z.1	
G1 Z0	
X.4868	(X.5-0.0132 compensation)
X 3.0 Z-2.1765	(Z-2.1651+0.0114 compensation)

F3.41: Tool Nose Radius Calculation, 1/64, Compensation value for 45 degree angle.



Code	Compensation (1/64 tool nose radius)
G0 X0 Z.1	
G1 Z0	
X.4816	(X.5-0.0184 compensation)
X 3.0 Z-1.2592	(Z-1.25+0.0092 compensation)

F3.42: Tool Nose Radius Calculation, 1/64, Compensation value for 60 degree angle.



Code	Compensation (1/64 tool nose radius)
G0 X0 Z.1	
G1 Z0	
X.4772	(X.5-0.0132 compensation)
X 3.0 Z-.467	(Z-0.7217+0.0066 compensation)

Tool Radius And Angle Chart (1/64 Radius)

The X measurement calculated is based on part diameter.

ANGLE	Xc CROSS	Zc LONGITUDINAL	ANGLE	Xc CROSS	Zc LONGITUDINAL
1.	.0006	.0155	46.	.00186	.0090
2.	.0001	.0154	47.	.0019	.0088
3.	.0016	.0152	48.	.0192	.0087
4.	.0022	.0151	49.	.0196	.0085
5.	.0026	.0149	50.	.0198	.0083
6.	.0032	.0148	51.	.0202	.0082
7.	.0036	.0147	52.	.0204	.0080
8.	.0040	.0145	53.	.0208	.0078
9.	.0046	.0144	54.	.021	.0077
10.	.0050	.0143	55.	.0214	.0075
11.	.0054	.0141	56.	.0216	.0073
12.	.0060	.0140	57.	.022	.0071
13.	.0064	.0138	58.	.0222	.0070
14.	.0068	.0137	59.	.0226	.0068
15.	.0072	.0136	60.	.0228	.0066
16.	.0078	.0134	61.	.0232	.0064
17.	.0082	.0133	62.	.0234	.0062
18.	.0086	.0132	63.	.0238	.0060
19.	.0090	.0130	64.	.024	.0059
20.	.0094	.0129	65.	.0244	.0057
21.	.0098	.0127	66.	.0246	.0055
22.	.0102	.0126	67.	.0248	.0053

ANGLE	Xc CROSS	Zc LONGITUDINAL	ANGLE	Xc CROSS	Zc LONGITUDINAL
23.	.0106	.0124	68.	.0252	.0051
24.	.011	.0123	69.	.0254	.0049
25.	.0014	.0122	70.	.0258	.0047
26.	.0118	.0120	71.	.0260	.0045
27.	.012	.0119	72.	.0264	.0043
28.	.0124	.0117	73.	.0266	.0041
29.	.0128	.0116	74.	.0268	.0039
30.	.0132	.0114	75.	.0272	.0036
31.	.0136	.0113	76.	.0274	.0034
32.	.014	.0111	77.	.0276	.0032
33.	.0142	.0110	78.	.0280	.0030
34.	.0146	.0108	79.	.0282	.0027
35.	.015	.0107	80.	.0286	.0025
36.	.0154	.0103	81.	.0288	.0023
37.	.0156	.0104	82.	.029	.0020
38.	.016	.0102	83.	.0294	.0018
39.	.0164	.0101	84.	.0296	.0016
40.	.0166	.0099	85.	.0298	.0013
41.	.017	.0098	86.	.0302	.0011
42.	.0174	.0096	87.	.0304	.0008
43.	.0176	.0095	88.	.0308	.0005
44.	.018	.0093	89.	.031	.0003
45.	.0184	.0092			

3.19 More Information Online

For updated and supplemental information, including tips, tricks, maintenance procedures, and more, go to www.HaasCNC.com and select the **Resource Center**.

Chapter 4: Programming

4.1 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 name and an end-of-block character (semicolon).

4.2 Program Editors

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

4.2.1 Basic Program Editing

This section describes the basic program editing controls. For information on more advanced program editing functions, refer to page 5.

F4.1: Edit Program Screen Example

```
EDIT: EDIT EDITOR
ACTIVE PROGRAM - 099997
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-H
ole) ;
```

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]**.
 - b. To edit a numbered program, select it, then press **[EDIT]**. Refer to page 76 to learn how to select a program.
2. To highlight code to edit:
 - 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.
3. To add code to the program:
 - 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, highlight the desired portion of the program using the arrow keys or the **[HANDLE JOG]** control, enter the replacement code, and press **[ALTER]**.
 - 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, highlight the text and press **[DELETE]**.
 - 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 149.*

4

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

4.2.2 The FNC Editor

The FNC Editor provides the same familiar 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** (**HDD**, **USB**, **Net Share**). Refer to the **Basic Editing** (page 148) and **Advanced Editor** (page 5) sections 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.

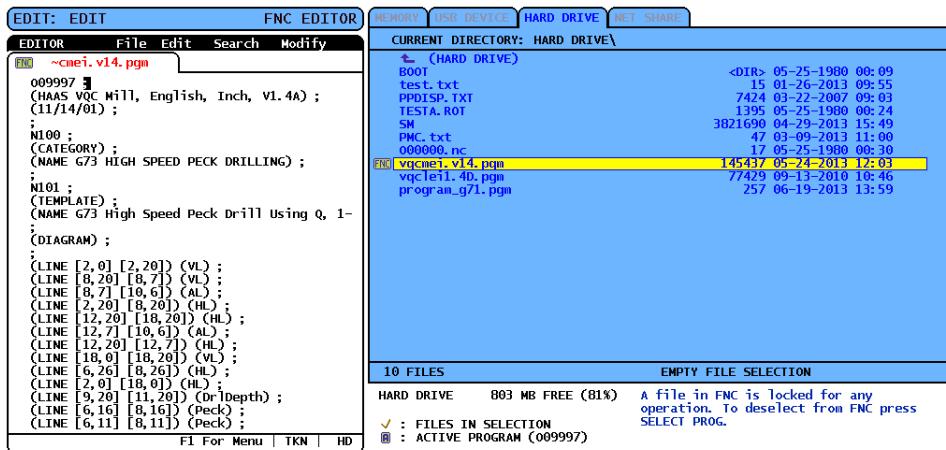
Loading a Program (FNC)

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.

F4.2: 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. 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.

F4.3: Footer Section of Program Display

```

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

F1 For Menu	TKN	USB
-------------	-----	-----

4

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:

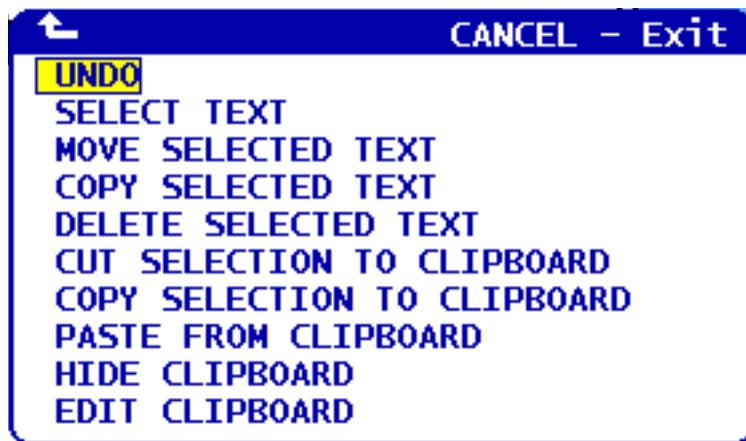
These are not the same as Nxx line numbers; they are only for reference when viewing the program.

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

Edit Menu (FNC)

To access the Edit menu:

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

F4.4: Edit Menu

Undo

To reverse 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

To highlight a block of text in FNC EDITOR mode:

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

Move/Copy/Delete Selected Text

To remove selected text from its current location and place it after the cursor position (Hot Key: **[ALTER]**), to place selected text after the cursor position without deleting it from its current location (Hot Key: **[INSERT]**), or to remove the selected text from the program (Hot Key: **[DELETE]**) in FNC EDITOR mode:

1. Before choosing this menu option or using Hot Keys: **[ALTER]**, **[INSERT]**, or **[DELETE]**, position the cursor at the line above where you wish to paste 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. Cursor to the Edit menu and select Move Selected Text, Copy Selected Text, or Delete Selected Text.

Cut/Copy Selection to Clipboard

To remove the selected text from the current program and move it to the clipboard or to place 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. Cursor to the Edit menu and select Cut Selection to Clipboard or Copy Selection to Clipboard.

4

Paste from Clipboard

To place the clipboard contents after the cursor location in FNC EDITOR mode:

**NOTE:**

Does not delete the clipboard contents.

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

Hide/Show Clipboard

To hide the clipboard to view the position and timers & counters displays in its place or to restore the clipboard display in FNC EDITOR mode:

1. Press [F1].
2. 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 Haas Editor cannot be pasted into Advanced Editor.

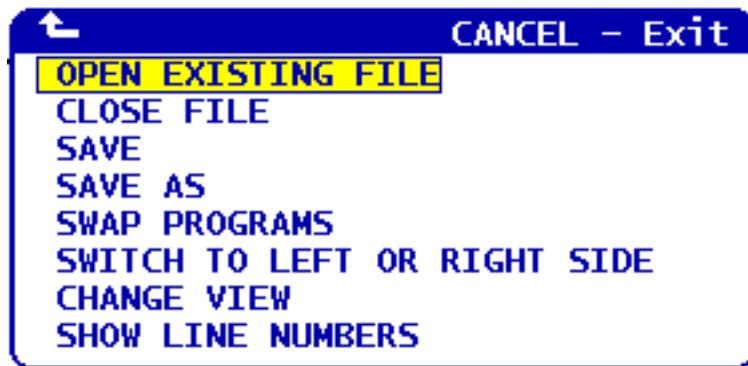
1. Press **[F1]**.
2. Cursor to the Edit menu and select Edit Clipboard.
3. When finished, press **[F1]**, cursor to the Edit menu, and select Close Clipboard.

File Menu (FNC)

To access the File menu:

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

F4.5: File Menu



Open Existing File

When in FNC EDITOR mode,

1. Press **[F1]**.
2. Cursor to the File menu and select Open Existing File.
3. Check mark a file to open and 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]**.
2. Cursor to the File menu and select Close File.

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

Save



NOTE:

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

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

1. Press **[F1]**.
2. Cursor to the File menu and select **Save**.

Saves the current active file under the same filename.

Save As

When in FNC EDITOR mode,

1. Press **[F1]**.
2. Cursor to the File menu and select Save As.

Saves the current active file under a new filename. Follow prompts for naming 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]**.
2. Cursor to the File menu and select Swap Programs.

Brings the next program in a tabbed pane to 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]**, 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]**.
2. Cursor to the File menu and select Change View.

Switches between List, Main, and Split view modes.

Show Line Numbers

When in FNC EDITOR mode,

1. Press [F1].
2. Cursor to the File menu and 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.

Search Menu (FNC)

To access the Search menu:

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

F4.6: Search Menu



Find Text

To define a search term, search direction, and to locate the first occurrence of the search term in the direction indicated in FNC EDITOR mode:

1. Press [F1].
2. Cursor to the Search menu and select Find Text.
3. Enter text item to locate.
4. Enter 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

To locate the next occurrence of the search term in FNC EDITOR mode:

1. Press **[F1]**.
2. 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

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

1. Press **[F1]**.
2. Cursor to the Search menu and select Find and Replace Text.
3. Enter text to locate.
4. Enter replacement text.
5. Enter 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.
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 your choice to continue. If you choose **Yes** or **No**, the editor will execute your choice and move to the next occurrence of the search term. Choose **All** to automatically replace all occurrences of the search term. Choose **Cancel** to back out of the function without making changes (text already replaced will remain so if you choose this option).

Find Tool

To search the program for tool numbers in FNC EDITOR mode:

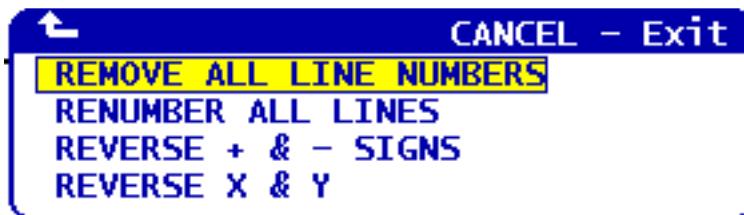
1. Press **[F1]**.
2. Cursor to the Search menu and select Find Tool.
3. Select it again to locate the next tool number.

Modify Menu (FNC)

To access the Modify menu:

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

F4.7: Modify Menu



Remove All Line Numbers

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

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

Renumber All Lines

To renumber all program lines with Nxx codes in FNC EDITOR mode:

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

Reverse + and - Signs

To change all positive values to negative and vice versa in FNC EDITOR mode:

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

4.2.3 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.

F4.8: The Advanced Editor Program Menu



Create New Program

1. Select the **CREATE NEW PROGRAM** command from the **PROGRAM** pop-up menu category.
2. Type a program name (Onnnnn) 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.

4

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.

Switch to Left or Right Side

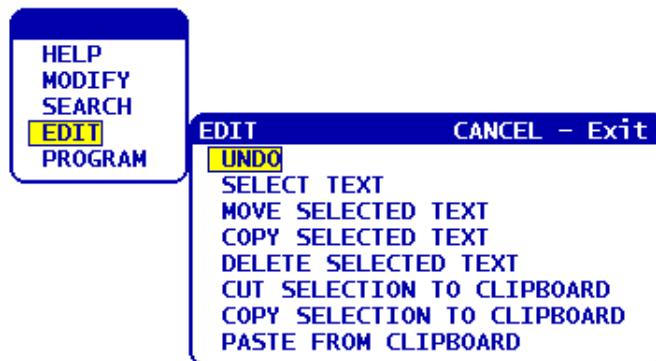
This switches edit control between the active and inactive program. Inactive and active programs remain in their respective panes.

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.

F4.9: 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

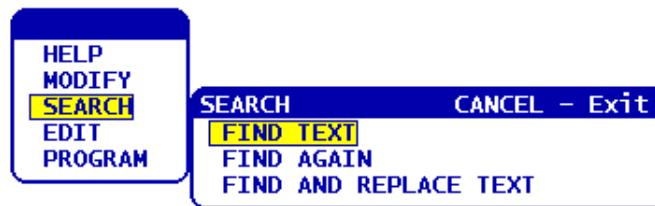
To copy the contents of the clipboard to the line after the cursor position:

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.

F4.10: 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.

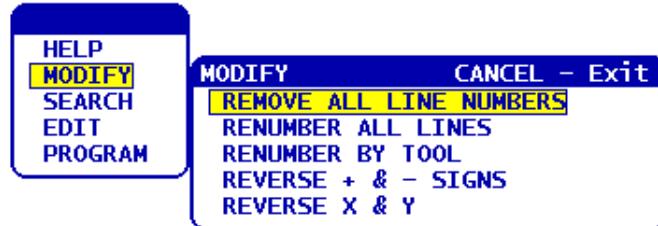
Choose **All** to automatically replace all occurrences of the search term.

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.

F4.11: Advanced Modify Popup



Remove All Line Numbers

This command automatically removes all unreferenced line numbers from the edited program. If you have selected a group of lines (refer to page 164), 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 164), 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 address code(s) you want to change.



NOTE:

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

3. Press **[ENTER]**.

4.2.4 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.

F4.12: MDI Input Page Example



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

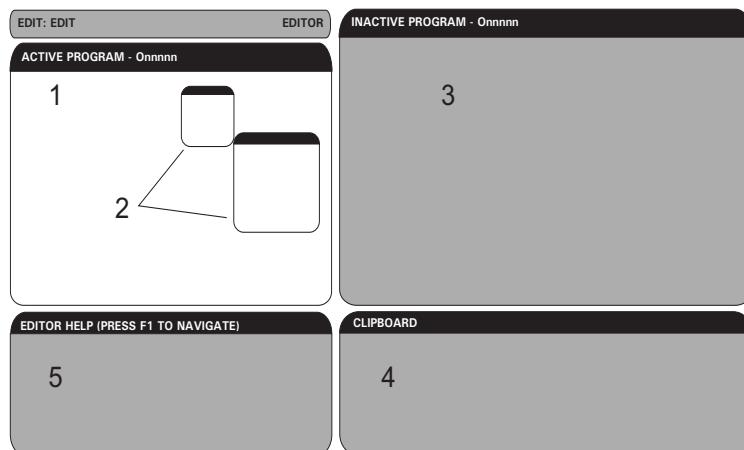
4.2.5 Advanced Editor

Advanced Editor lets you use popup menus to edit programs.

Press **[EDIT]** to enter edit mode. Two editing panes are available; an active program pane and an inactive program pane. Press **[EDIT]** to switch between the two.

To edit a program, enter the program name (Onnnnn) from the active program pane and press **[SELECT PROGRAM]**; the program will open in the active window. Pressing F4 will open another copy of that program in the inactive program pane if there is not a program there already. To select a different program in the inactive program pane, press **[SELECT PROGRAM]** from the inactive program pane and select the program from the list. Press **[F4]** to exchange the programs between the two panes (make the active program inactive and vice versa). Use the **[HANDLE JOG]** control or the **[DOWN]/[UP]** arrow keys to scroll through the program code.

F4.13: Basic Edit Mode Layout: [1] Active Program Pane, [2] Pop-up menus, [3] Inactive program pane, [4] Clipboard, [5] Context-sensitive help messages



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

4.2.6 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 M30), 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.*

4.3 Tips and Tricks

The following sections provide insight into efficiently programming your Haas Turning Center.

4.3.1 Programming

Short programs looped many times do not reset the chip conveyor if the intermittent feature is activated. The conveyor continues to start and stop at the commanded times. Refer to page **413** for information on the conveyor interval settings.

The screen displays the spindle and axis loads, the current feed and speed, positions, and the currently active codes while a program runs. Changing display modes changes the information displayed.

To clear all offsets and macro variables, press [**ORIGIN**] at the **Active Work Offset** screen. The control displays a popup menu. Pick **Clear Work Offsets** for the displayed message *Are you sure you want to Zero (Y/N)*. If **Y** is entered, all the work offsets (macros) in the area being displayed are set to zero. The values in the **Current Commands** display pages can be cleared as well. The Tool Life, Tool Load, and Timer registers are cleared by selecting the one to clear and pressing [**ORIGIN**]. To clear everything in a column, scroll to the top of the column onto the title and press [**ORIGIN**].

Selecting another program quickly can be accomplished simply by entering the program number (Onnnnn) and pressing arrow up or down. The machine must be in either **Memory** or **Edit** mode. Searching for a specific command in a program is done as well in either Memory or Edit mode. Enter the address code (A, B, C etc.), or the address code and the value. (A1 . 23), and press the up or down arrow key. If the address code is entered without a value, the search stops at the next use of that letter.

Transfer or save a program in MDI to the list of programs by positioning the cursor at the beginning of the MDI program, entering a program number (Onnnnn), and pressing **[ALTER]**.

Program Review - Program Review allows the operator to cursor through and review a copy of the active program on the right side of the display screen, while also viewing the same program as it is running on the left side of the screen. To display a copy of the active program in the **Inactive Program** display, press **[F4]** while the **Edit** pane containing the program is active.

Background Edit - This feature allows editing while a program is running. Press **[EDIT]** until the background **Edit** pane (on the right side of the screen) is active. Select a program to edit from the list and press **[ENTER]**. Press **[SELECT PROGRAM]** from this pane to select another program. Edits are possible as the program runs, however, edits to the running program will not take effect until the program ends with an **M30** or **[RESET]**.

Graphics Zoom Window - **[F2]** activates the zoom window when in **Graphics** mode. **[PAGE DOWN]** zooms in and page up expands the view. Use the arrow keys to move the window over the desired area of the part and press **[ENTER]**. Press **[F2]** and **[HOME]** to see full table view.

Copying Programs - In **Edit** mode, a program can be copied into another program, a line, or a block of lines in a program. Start defining a block with the **[F2]** key, then cursor to the last program line to define, press **[F2]** or **[ENTER]** to highlight the block. Select another program to copy the selection to. Cursor to the point where the copied block is to be placed and press **[INSERT]**.

Loading Files - Load multiple files by selecting them in the device manager, then press **[F2]** to select a destination.

Editing Programs - Press **[F4]** while in **Edit** mode to display another version of the current program in the right-hand pane. Different portions of the programs can be edited alternately by pressing **[EDIT]** to switch from one side to the other. The program is updated once switched to the other program.

Duplicating a Program - Using the **List Program** mode, an existing program can be duplicated. To do this select the program number you wish to duplicate, type in a new program number (Onnnnn) and press **[F2]**. This can also be done through the popup help menu. Press **[F1]**, then select the option from the list. Type the new program name and press **[ENTER]**.

Several programs can be sent to the serial port. Select the desired programs from the program list by highlighting them and pressing [ENTER]. Press [SEND] to transfer the files.

4.3.2 Offsets

Entering offsets:

1. To toggle back and forth between the **Tool Geometry** and **Work Zero Offset** panes, press [OFFSET].
2. To add the entered number to the cursor-selected value, press [ENTER].
3. To take the entered number and overwrite the cursor selected offset register, press [F1].
4. To enter the negative value into the offset, press [F2].

4.3.3 Settings and Parameters

The **[HANDLE JOG]** control is used to scroll through settings and parameters, when not in jog mode. Enter a known parameter or setting number and press the up or down arrow key to jump to the entered parameter.

The Haas control can power off the machine using settings. These settings are: Setting 1 turns off power after machine is idle for nn minutes, and Setting 2 turns off power when M30 is executed.

Memory Lock (Setting 8) when On, memory edit functions are locked out. When Off, memory can be modified.

Dimensioning (Setting 9) changes from **Inch** to **MM**. This changes all offset values too.

Reset Program Pointer (Setting 31) turns on and off the program pointer returning to the program beginning.

Scale Integer F (Setting 77) changes the interpretation of a feed rate. A feed rate can be misinterpreted if there is not a decimal point in the Fnn command. The selections for this setting are **Default**, to recognize a 4 place decimal. Another selection is **Integer** which recognizes a feed rate for a selected decimal position, for a feed rate that does not have a decimal.

Max Corner Rounding (Setting 85) is used to set the corner rounding accuracy required by the user. Any feed rate up to the maximum can be programmed without the errors getting above that setting. The control slows at corners only when needed.

Reset Resets Override (Setting 88) turns on and off the Reset key setting the overrides back to 100%.

Cycle Start/Feed hold (Setting 103) when on, **[CYCLE START]** must be pressed and held to run a program. Releasing **[CYCLE START]** generates a Feed Hold condition.

Jog Handle to Single Block (Setting 104) allows the **[HANDLE JOG]** control to be used to step through a program. Reversing the **[HANDLE JOG]** control generates a Feed Hold condition.

Offset Lock (Setting 119) prevents the operator from altering any of the offsets.

Macro Variable Lock (Setting 120) prevents the operator from altering any of the macro variables.

4.3.4 Operation

[MEMORY LOCK] key switch - prevents the operator from editing programs and from altering settings when in the locked position.

[HOME G28] - Returns all axes to machine zero. To send just one axis to machine home, enter the axis letter and press **[HOME G28]**. To zero out all axes on the Distance-To-Go display, while in **Jog** mode, press any other operation mode (**[EDIT]**, **[MEMORY]**, **[MDI/DNC]**, etc.) then press **[HANDLE JOG]**. Each axis can be zeroed independently to show a position relative to the selected zero. To do this go to the **Position Operator** page, press **[HANDLE JOG]**, position the axes to the desired position and press **[ORIGIN]** to zero that display. In addition a number can be entered for the axis position display. To do this, enter an axis and number, for example, **X2.125** then **[ORIGIN]**.

Tool Life - Within the **Current Commands** page there is a **Tool Life** window displaying tool usage. This register counts each time the tool is used. The tool life monitor stops the machine once the tool reaches the value in the alarms column.

Tool Overload - Tool load can be defined by the Tool Load monitor; this changes normal machine operation if it reaches the tool load defined for that tool. When a tool overload condition is encountered, one of four actions occurs depending on Setting 84:

- **Alarm** - Generate an alarm
- **Feedhold** - Stop the feed
- **Beep** - Sounds an audible alarm
- **Autofeed** - Automatically increase or decrease the feed rate

Spindle speed is verified by checking the **Current Commands All Active Codes** display (also displayed in the Main Spindle window). Live tooling spindle axis RPM is also displayed on this page.

To select an axis for jogging, enter the axis name on the input line and press **[HANDLE JOG]**.

The Help display has all the G and M codes listed. They are available within the first tab of the Help tabbed menu.

The jogging speeds of 100, 10, 1.0 and 0.1 inches per second can be adjusted by the Feed Rate Override keys. This gives an additional 10% to 200% control.

4.3.5 Calculator

The number in the calculator box can be transferred to the data entry line by pressing [F3] in **Edit** or **MDI** mode. This transfers the number from the calculator box to the **Edit** or **MDI** input buffer (enter a letter, X, Z, etc., for the command to use with the number from the calculator).

The highlighted **Trig**, **Circular**, or **Turning and Tapping** data can be transferred to load, add, subtract, multiply, or divide in the calculator by selecting the value and pressing [F4].

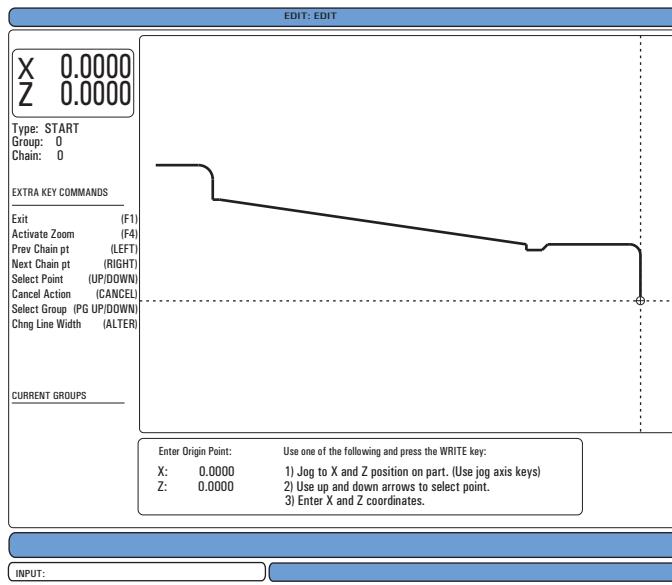
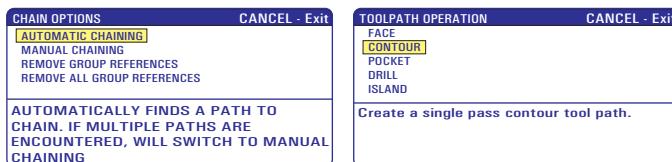
Simple expressions can be entered into the calculator. For example $23*4-5.2+6/2$, is evaluated when **ENTER** is pressed and the result (89.8 in this case) is displayed in the calculator box.

4.4 DXF File Importer

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

**NOTE:**

DXF importer is available only with the Intuitive Programming System (IPS) option.

F4.14: DXF Imported File**F4.15:** Chain Option Tool Path Menus

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

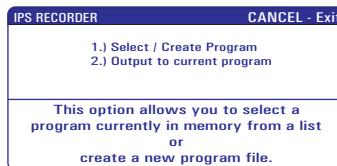
1. Start by setting up the cutting tools in IPS. Select a .dxf file and press F2. The control recognizes a DXF file and imports it into the editor. Set the origin of the part. This can be done by using one of three methods.
 - a. Point Selection
 - b. Jogging
 - c. Enter Coordinates
 - d. The **[HANDLE JOG]** control or arrow keys are used to highlight a point; press **[ENTER]** to accept the highlighted point as the origin. This is used to set the work coordinate information of the raw part.

2. Chain / Group. This step finds the geometry of the shape(s). The auto chaining function finds most part geometry. If the geometry is complex and branches off, a prompt displays so the operator can select one of the branches. The automatic chaining continues once a branch is selected.
 - a. This changes the color of that part feature and adds a group to the register under **Current Group** on the left hand side of the window.
 - b. Press **[F2]** to open the dialog box.
 - c. Use **[HANDLE JOG]** control or arrow keys to choose the starting point of the tool path.
 - d. Choose the option that best suits the desired application. The Automatic Chaining function is typically the best choice as it automatically plots the tool path for a part feature. Press **[ENTER]**.

**NOTE:**

The cutting tools should have been previously set up in IPS.

3. Select Tool Path. This step applies a tool-path operation to a particular chained group.
 - a. Select **Group** and press **[F3]** to choose a tool path.
 - b. Use the **[HANDLE JOG]** control to bisect an edge of the part feature; this is used as a entry point for the tool. Once a tool-path is selected, the IPS (Intuitive Programming System) template for that path displays. Most IPS templates are filled with reasonable defaults. They are derived from tools and materials that have been set up.
 - c. Press **[F4]** to save the tool path once the template is completed; either add the IPS G-code segment to an existing program or create a new program.
 - d. Press **[EDIT]** to return to the DXF import feature to create the next tool path.

F4.16: IPS Recorder Menu

4.5 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.

2. **Cutting:**

This portion of the program defines the tool path, spindle speed, 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.

The following program makes a 0.100" (2.54 mm) deep cut with Tool 1 in a piece of material along a straight line path from Z=0.0, X=2.0 to Z=-3.0, X=2.0.



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.

Also, line numbers given here are for reference; they should not be included in the actual program.

1. % (Preparation)
2. 000100 (Basic Program Number- Preparation) ;
3. T101 (Preparation) ;
4. G00 G18 G20 G40 G54 G80 G99 (Preparation) ;
5. S2000 G50 (Preparation) ;
6. S500 G97 M03 (Preparation) ;
7. G00 X2.0 Z0.1 M08 (Preparation) ;
8. S900 G96 (Preparation) ;
9. G01 Z-3.0 F.01 (Cutting) ;
10. G00 X2.1 M09 (Completion) ;
11. G53 X0 Z0 (Completion) ;
12. M30 (Completion) ;
13. % (Completion)

4.5.1 Preparation

These are the preparation code blocks in the sample program:

Preparation Code Block	Description
%	Denotes the beginning of a program written in a text editor.
O00100 (Basic Program)	O00100 is the name of the program. Program naming convention follows the Onnnnn format: The letter "O" followed by a 5-digit number.
T101 ;	Selects the tool, the offset, and commands the tool change to Tool 1.
G00 G18 G20 G40 G54 G80 G99 ;	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 in Rapid Motion mode. G18 defines the cutting plane as the XZ plane. G20 defines the coordinate positioning to be in Inches. G40 cancels Cutter Compensation. G54 defines the coordinate system to be centered on the Work Offset stored in G54 on the Offset display. G80 cancels any canned cycles. G99 puts the machine in Feed per Rev mode.
S2000 G50 ;	Limits the spindle to a max of 2000 RPM.
S500 G97 M03 ;	S500 is the spindle speed address. Using Snnnn address code, where nnnn is the desired spindle RPM value. G97 cancels constant surface speed (CSS) making the S value a direct RPM of 500. 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 379 for more information on these M-codes. M03 turns on the spindle.
G00 X2.0 Z0.1 M08 ;	G00 defines axis movement following it to be in Rapid Motion mode. X2.0 commands the X-Axis to X=2.0. Z0.1 commands the Z-Axis to Z=0.1. M08 turns on the coolant.
S900 G96 ;	G96 turns on CSS. S900 specifies the cutting speed to be used along with the current diameter to calculate the correct RPM.

4.5.2 Cutting

These are the cutting code blocks in the sample program:

Cutting Code Block	Description
G01 Z-3.0 F.01 ;	G01 defines axis movements following it to be in a straight line. G01 requires address code Fn.nnnn. F.01 specifies the feed rate for the motion is .01" (.254 mm)/Rev. Z-3.0 commands the Z-Axis to Z=-3.0.

4.5.3 Completion

These are the completion code blocks in the sample program:

Completion Code Block	Description
G00 X2.1 M09 ;	G00 commands the axis motion to be completed in Rapid Motion mode. X2.1 commands the X-Axis to X=2.1. M09 turns off the coolant.
G53 X0 Z0 ;	G53 defines axis movements following it to be with respect to the machine coordinate system. X0 Z0 commands the X-Axis and Z-Axis to move to X=0.0, Z=0.0.
M30 ;	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.

4.5.4 Absolute vs. Incremental (XYZ vs. UVW)

Absolute (XYZ) and incremental positioning (UVW) define how the control interprets axis motion commands.

When you command axis motion using X, Y, or Z, the axes move to that position relative to the origin of the coordinate system currently in use.

When you command axis motion using U(X), V(Y), or W(Z), 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.

4.6 Tool Functions

The `Tnnoo` code is used to select the next tool (`nn`) and offset (`oo`). The use of this code differs slightly depending on Setting 33 (FANUC or YASNAC coordinate system).

4.6.1 FANUC Coordinate System

T-codes have the format `Txxyy` where `xx` specifies the tool number from 1 to the maximum number of stations on the turret; and `yy` specifies the tool geometry and tool wear indices from 1 to 50. The tool geometry `x` and `z` values are added to the work offsets. If tool nose compensation is used, `yy` specifies the tool geometry index for radius, taper, and tip. If `yy` = 00 no tool geometry or wear is applied.

4.6.2 YASNAC Coordinate System

T-codes have the format `Tnnoo`, `nn` has different meanings depending on whether the T-code is inside or outside a G50 block. The `oo` value specifies the tool wear from 1 to 50. If tool nose compensation is used, `50+oo` specifies the tool shift index for radius, taper, and tip. If `oo+00`, no tool wear or tool nose compensations are applied.

Outside a G50 block, `nn` specifies the tool number from 1 to the maximum number of stations on the turret.

Inside a G50 block, `nn` specifies the tool shift index from 51 to 100. The tool shift X and Z values are subtracted from the work offsets and thus are of opposite sign than the tool geometries used in the FANUC coordinate system.

4.6.3 Tool Offsets Applied by T101, FANUC vs YASNAC

Setting a negative tool wear in the tool wear offsets moves the tool further in the negative direction of the axis. Thus, for O.D. turning and facing, setting a negative offset in the X-axis results in a smaller diameter part and setting a negative value in the Z-axis results in more material being taken off the face.

**NOTE:**

There is no X or Z motion required prior to performing a tool change and it wastes time in most cases to return X or Z to the home position. However, you must position X or Z to a safe location prior to a tool change in order to prevent a crash between the tools and the fixture or part.

Low air pressure or insufficient volume reduces the pressure applied to the turret clamp/unclamp piston and slows down the turret index time or does not unclamp the turret.

To load or change tools:

1. Press **[POWER UP/RESTART]** or **[ZERO RETURN]** and then **[ALL]**.
The control moves the tool turret to a normal position.
2. Press **[MDI/DNC]** to toggle to MDI mode.
3. Press **[TURRET FWD]** or **[TURRET REV]**.
The machine indexes the turret to the next tool position.
Shows the current tool in the **Active Tool** window in the lower right of the display.
4. Press **[CURRENT COMMANDS]**.
Shows the current tool in the **Active Tool** display in the upper right of the screen.

4

4.7 Coordinate Systems

CNC controls use a variety of coordinate systems and offsets that allow control of the location of the tooling point to the part. This section describes interaction between various coordinate systems and tooling offsets.

4.7.1 Effective Coordinate System

The effective coordinate system is the sum total of all coordinate systems and offsets in effect. It is the system that is displayed under the label **Work G54** on the **Position** display. It is also the same as the programmed values in a G code program assuming no Tool Nose Compensation is being performed. Effective Coordinate = global coordinate + common coordinate + work coordinate + child coordinate + tool offsets.

FANUC Work Coordinate Systems - Work coordinates are an additional optional coordinate shift relative to the global coordinate system. There are 105 work coordinate systems available on a Haas control, designated G54 through G59 and G154 P1 through G154 P99. G54 is the work coordinate in effect when the control is powered on. The last used work coordinate stays in effect until another work coordinate is used or the machine is powered off. G54 can be deselected by ensuring that the X and Z values on the work offset page for G54 are set to zero.

FANUC Child Coordinate System - A child coordinate is a coordinate system within a work coordinate. Only one child coordinate system is available and it is set through the G52 command. Any G52 set during the program is removed once the program finishes at an M30, pressing **[RESET]**, or pressing **[POWER OFF]**.

FANUC Common Coordinate System - The common (Comm) coordinate system is found on the second work coordinate offsets display page just below the global coordinate system (G50). The common coordinate system is retained in memory when power is turned off. The common coordinate system can be changed manually with G10 command or by using macro variables.

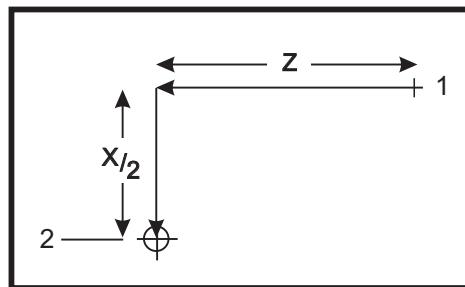
YASNAC Work Coordinate Shift - YASNAC controls discuss a work coordinate shift. It serves the same function as the common coordinate system. When Setting 33 is set to **YASNAC**, it is found on the **Work Offsets** display page as T00.

YASNAC Machine Coordinate System - The effective coordinates take the value from machine zero coordinates. Machine coordinates can be referenced by specifying G53 with X and Z in a motion block.

YASNAC Tool Offsets - There are two offsets available: **Tool Geometry** offsets and **Tool Wear** offsets. **Tool Geometry** offsets adjusts for different lengths and widths of tools, so that every tool comes to the same reference plane. **Tool Geometry** offsets are usually done at setup time and remain fixed. **Tool Wear** offsets allow the operator to make minor adjustments to the geometry offsets to compensate for normal tool wear. **Tool Wear** offsets are usually zero at the beginning of a production run and may change as time progresses. In a FANUC compatible system, both **Tool Geometry** and **Tool Wear** offsets are used in the calculation of the effective coordinate system.

In a YASNAC compatible system, **Tool Geometry** offsets are not available; they are replaced with tool shift offsets (50 tool shift offsets numbered 51 - 100). YASNAC tool shift offsets modify the global coordinate to allow for varying tool lengths. Tool shift offsets must be used prior to calling for the use of a tool with a G50 T_{xx}00 command. The tool shift offset replaces any previously calculated global shift offset and a G50 command overrides a previously selected tool shift.

F4.17: G50 YASNAC Tool Shift: [1] Machine (0,0), [2] Spindle centerline .



```

000101 ;
N1 G51 (Return to machine Zero) ;
N2 G50 T5100 (Offset for Tool 1) ;
.
.
.
%
```

4.7.2 Automatic Setting of Tool Offsets

Tool offsets are recorded automatically by pressing **[X DIAMETER MEASURE]** or **[Z FACE MEASURE]**. If the common, global, or currently selected work offset have values assigned to them, the recorded tool offset differs from actual machine coordinates by these values. After setting up tools for a job, all tools should be commanded to a safe X, Z coordinate reference point as a tool change location.

4.7.3 Global Coordinate System (G50)

The global coordinate system is a single coordinate system that shifts all work coordinates and tool offsets away from machine zero. The global coordinate system is calculated by the control so the current machine location becomes the effective coordinates specified by a G50 command. The calculated global coordinate system values can be seen on the **Active Work Offset** coordinates display just below auxiliary work offset G154 P99. The global coordinate system is cleared to zero automatically when the CNC control is powered on. The global coordinate is not changed when **[RESET]** is pressed.

4.8 Live Image

To bring up the Live Image window (either before or after **[CYCLE START]**):

1. Press **[CURRENT COMMANDS]**.
2. Press **[PAGE UP]** until the Live Image window appears.
3. Press **[F2]** to switch Zoom On/Off (Off shows *Currently Zoomed*).
4. Use **[PAGE UP]** to zoom out. Use **[PAGE DOWN]** to zoom in.
5. Use **[LEFT]/[RIGHT]** or **[UP]/[DOWN]** cursors to move the zoomed window over the area to be monitored.
6. Press **[ENTER]** to fix zoomed window position and clear the screen to start the graphic where the program is currently running or where you want to view once the program is started.
7. Screen Shows: Live Image Scale, Currently Running program, Current tool, and Current Offset

4.8.1 Live Image Stock Setup

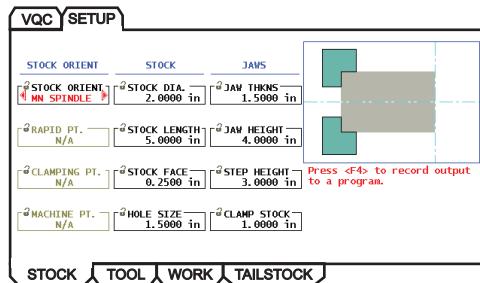
Data values for stock and jaw dimensions are stored in the Stock Setup screen. Live Image applies this stored data to each tool.



NOTE:

Turn Setting 217 ON (refer to page 420) to show the chuck jaws in the display.

F4.18: Tailstock Setup Screen



To enter stock and jaw values:

1. Press [MDI/DNC], then [PROGRAM] to enter **I PS JOG** mode.
2. Use the right/left arrow keys to select the **SETUP** tab and press [**ENTER**]. Use the right/left arrow keys to select the **STOCK** tab and press [**ENTER**] to display the **Stock Setup** screen. Screens are navigated by using the left/right/up/down arrow keys to navigate through variables. To enter the information requested by a parameter selection, use the number pad, then press [**ENTER**]. To exit a screen, press [**CANCEL**].

The Stock Setup screen displays stock and chuck jaw parameters that are changed to run a particular part.

3. Once the values are entered press [**F4**] to save the stock and jaw information to the program.
4. Select one of the choices and press [**ENTER**]. The control enters the new lines of code at the cursor. Ensure the new code is entered at the line after the program number.

4.8.2 Program Example

```
%  
O01000 ;  
;  
G20 (INCH MODE) (Start of Live Image information) ;  
(STOCK) ;  
([0.0000, 0.1000] [6.0000, 6.0000]) ([Hole Size, Face]  
[Diameter, Length]) ;  
(JAWS) ;  
([1.5000, 1.5000] [0.5000, 1.0000]) ([Height, Thickness]  
[Clamp, Step Height]) (End of Live Image Information) ;  
M01 ;  
;  
[Part Program]
```

The advantage of entering the Stock Settings into the program is that these settings are saved with the program, and the Stock Setup screen does not require further data entry when the program is run in the future.

Further settings for Live Image, such as **x** and **z Offset**, **Rapid Path** and **Feed Path** **Live Image** and **Show Chuck Jaws** are accessed by pressing [**SETTING/GRAPHIC**], typing in the first **LIVE IMAGE** setting (202) and pressing the [**UP**] cursor arrow. Refer to page **418** for more information.

F4.19: Control Panel Live Image Settings

LIVE IMAGE	
202	LIVE IMAGE SCALE (HEIGHT)
203	LIVE IMAGE X OFFSET
205	LIVE IMAGE Z OFFSET
206	STOCK HOLE SIZE
207	Z STOCK FACE
208	STOCK OD DIAMETER
209	LENGTH OF STOCK
210	JAW HEIGHT
211	JAW THICKNESS
212	CLAMP STOCK
213	JAW STEP HEIGHT
214	SHOW RAPID PATH LIVE IMAGE
215	SHOW FEED PATH LIVE IMAGE
217	SHOW CHUCK JAWS
218	SHOW FINAL PASS
219	AUTO ZOOM TO PART
220	TS LIVE CENTER ANGLE
221	TAILSTOCK DIAMETER
222	TAILSTOCK LENGTH

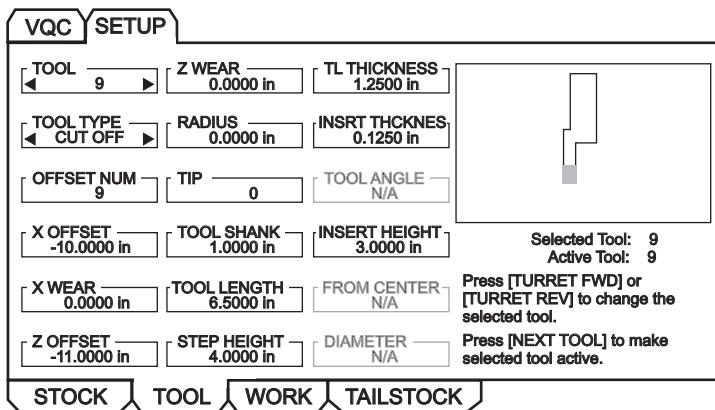
4.8.3 Live Image Tool Setup

Tool data is stored in offsets in the IPS tabs. Live Image uses this information to draw and simulate the tool in the cut. Required dimensions can be found in a tooling supplier's catalog or by measuring the tool.

**NOTE:**

Setup parameter entry boxes are grayed out if they do not apply to the selected tool.

F4.20: Tool Setup



NOTE:

Tool offset data may be entered for up to 50 tools.

4

The following section shows part of a lathe program that is cutting a piece of stock. The program and the appropriate tool setting illustrations follows:

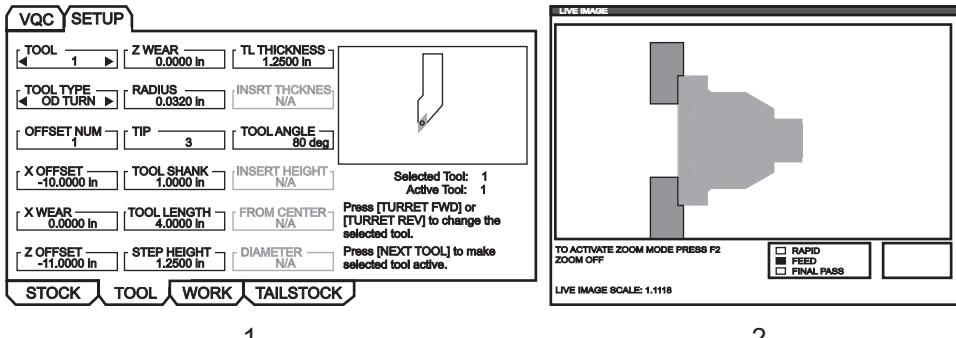
```

001000 ;
T101 ;
G54 ;
G50 S4000 ;
G96 S950 M03 ;
M08 ;
G00 X6.8 ;
Z0.15 ;
G71 P80103 Q80203 D0.25 U0.02 W0.005 F0.025 ;
N80103 ;
G00 G40 X2. ;
G01 X2.75 Z0. ;
G01 X3. Z-0.125 ;
G01 X3. Z-1.5 ;
G01 X4.5608 Z-2.0304 ;
G03 X5. Z-2.5606 R0.25 ;
G01 X5. Z-3.75 ;
G02 X5.5 Z-4. R0.25 ;
G01 X6.6 Z-4. ;
N80203 G01 G40 X6.8 Z-4. ;
G00 X6.8 Z0.15 ;
M09 ;
M01 ;
G53 X0;

```

```
G53 Z0;
M30 ;
```

F4.21: [1] T101 Settings , and [2] Part worked from T101 Settings.

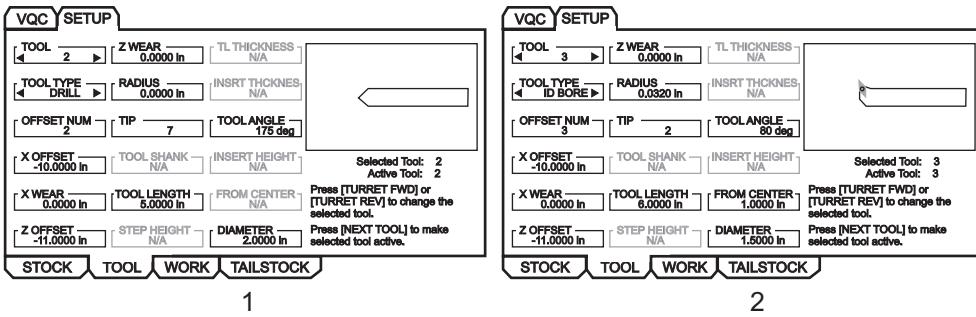


1

2

Sample Tool Setup Screens

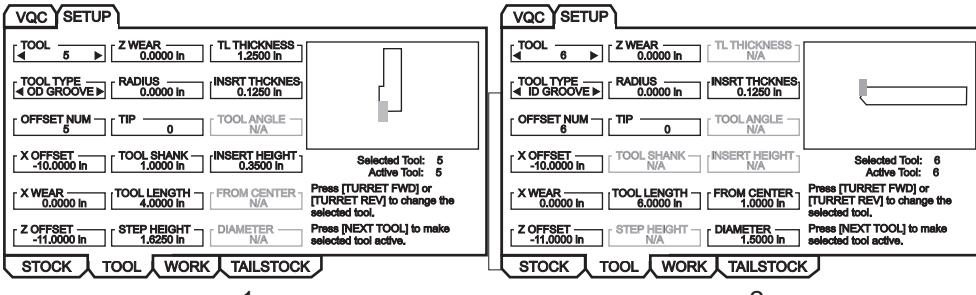
F4.22: Tool Setup: [1] Drill, [2] ID Bore



1

2

F4.23: Tool Setup: [1] OD Grove, [2] ID Groove



1

2

F4.24: Tool Setup: [1] OD Thread, [2] ID Thread

1	2

F4.25: Tool Setup: [1] Tap, [2] Face Groove

1	2

1. From the stock setup tab, press **[CANCEL]**, select the **TOOL** tab and press **[ENTER]**.
2. Select the tool number, type, and enter the specific parameters required for that tool (i.e., offset number, length, thickness, shank size, etc.).

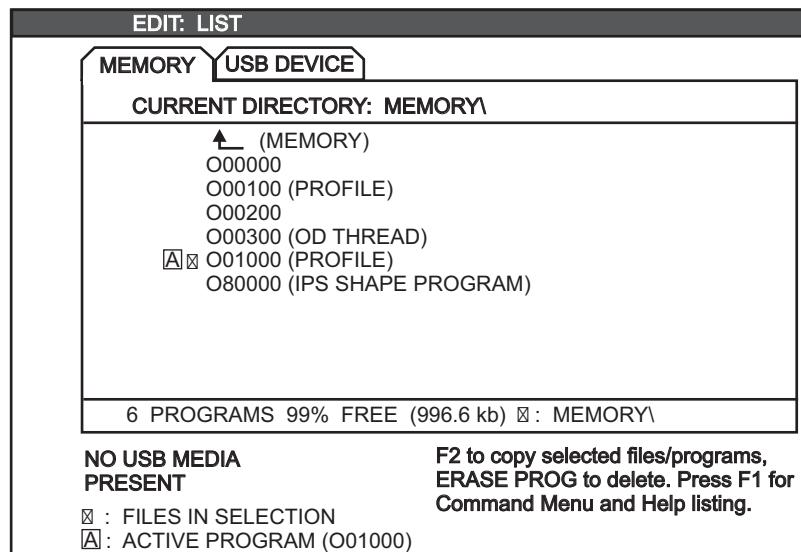
4.8.4 Tailstock Setup (Live Image)

Data values for tailstock parameters are stored in offsets in the Tailstock Setup screen.



NOTE:

Tailstock tab is only visible when the machine has a tailstock.

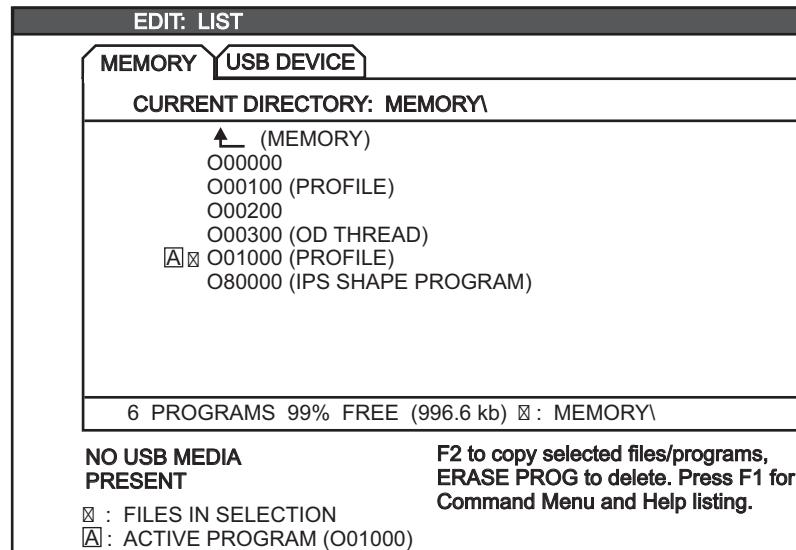
F4.26: Tailstock Setup Screen

1. Press **[MDI/DNC]**, then **[PROGRAM]** to enter **IPS JOG** mode.
2. Use the right/left arrow keys to select the **SETUP** tab and press **[ENTER]**. Use the right/left arrow keys to select the **TAILSTOCK** tab and press **[ENTER]** to display the **Tailstock Setup Screen**.
3. To change data, enter a value on the input line and press **[ENTER]** to add the entered value to the current value, or press **[F1]** to overwrite the current value with the entered value.
4. When highlighting **TS POSITION**, pressing **[Z FACE MEASURE]** takes the value of the B axis and places it in **TS POSITION**. When highlighting **X CLEARANCE**, pressing **[X DIAMETER MEASURE]** takes the value of the X Axis and places it in **X CLEARANCE**. When highlighting **Z CLEARANCE**, pressing **[Z FACE MEASURE]** takes the value of the Z Axis and places it in **Z CLEARANCE**.
5. Pressing **[ORIGIN]** when highlighting **X CLEARANCE** sets clearance to max travel. Pressing **[ORIGIN]** when highlighting **z CLEARANCE** sets clearance to zero.

4.8.5 Operation

Pick a program to run:

F4.27: Current Directory Memory Screen

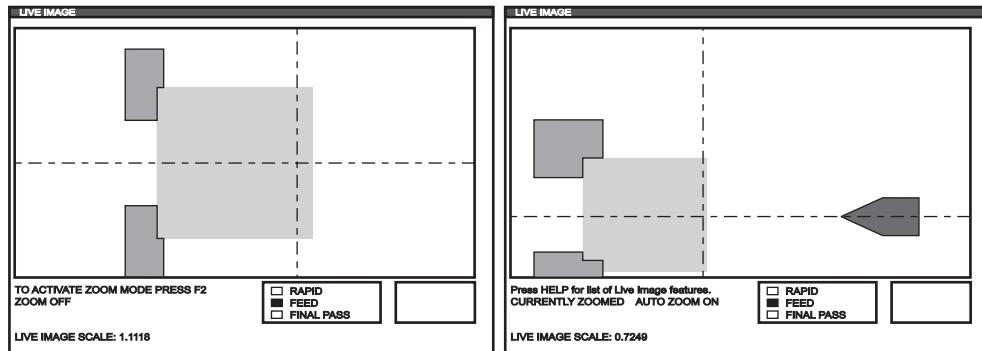


1. Select the desired program by pressing [**LIST PROGRAM**] to display the **EDIT: LIST** screen. Select the **MEMORY** tab and press [**ENTER**] to display **CURRENT DIRECTORY: MEMORY** screen.
2. Select a program (i.e., O01000) and press [**ENTER**] to choose it as the active program.

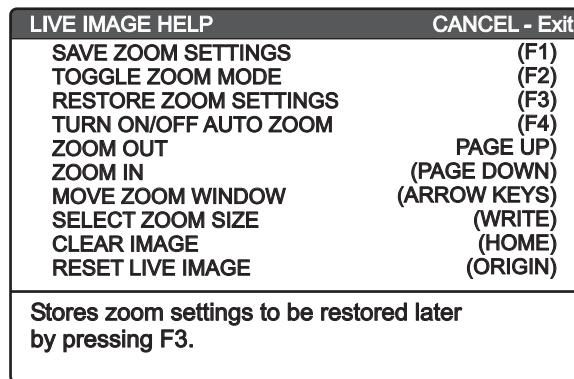
4.8.6 Run Part

To watch **Live Image** screen while a part is machined:

F4.28: Live Image Screen with Stock Drawn



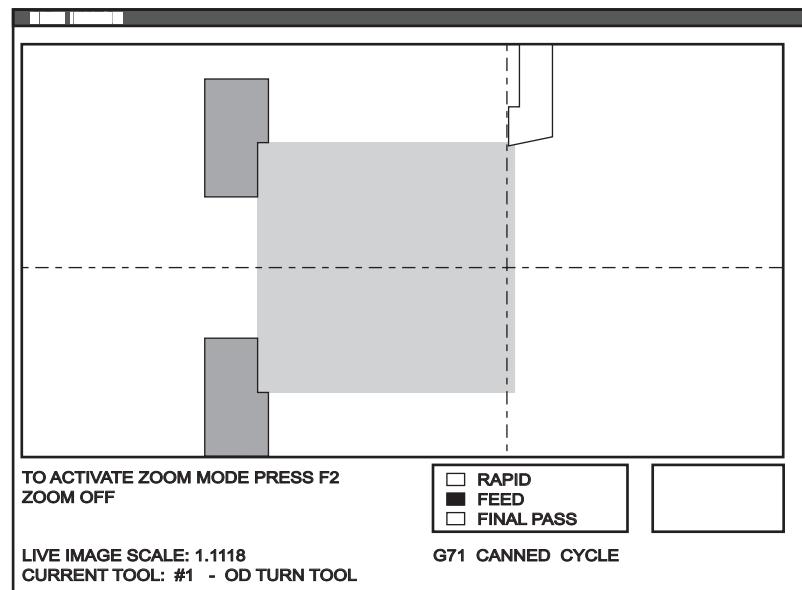
F4.29: Live Image Feature List



NOTE:

When the Bar Feeder reaches G105, the part is refreshed.

F4.30: Live Image Tool working the Part



NOTE:

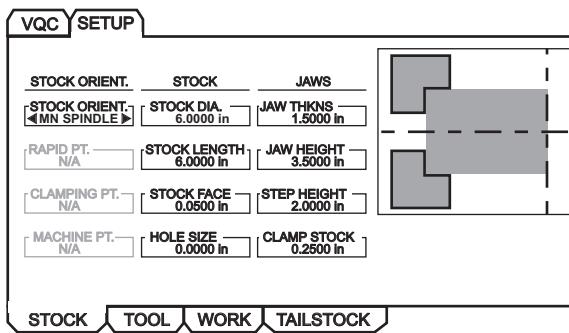
Data displayed on the screen while the program is running includes: program, main spindle, machine position, and timers and counters.

1. Press **[MEMORY]**, then **[CURRENT COMMANDS]**, and then **[PAGE UP]**. When the screen appears, press **[ORIGIN]** to display the **Live Image** screen with stock drawn.
 - a. Press **[F2]** to enter **zoom** mode. Use **[PAGE UP]** and **[PAGE DOWN]** to zoom the display and the direction keys to move the display. Press **[ENTER]** when the desired zoom is achieved. Press **[ORIGIN]** to return to zero zoom, or press **[F4]** to auto zoom to the part. Press **[F1]** to save a zoom and press **[F3]** to load a zoom setting.
 - b. Press **[HELP]** for a pop-up containing a list of **Live Image** features.
2. Press **[CYCLE START]**. A warning pops up on the screen. Press **[CYCLE START]** again to run the program. When a program is running and tool data has been set up, the **Live Image** screen shows the tool working the part in real time as the program runs.

4.8.7 Flipping a Part

A graphical representation of a part that has been flipped manually by the machinist is depicted by adding the following comments to the program following an M00.

F4.31: Flipped Part Setup Screen



```
000000 ;
[Code for first operation of Live Image] ;
[Code for first operation of machined part] ;
M00 ;
G20 (INCH MODE) (Start of Live Image Information for flipped
part) ;
(FLIP PART) ;
(CLAMP) ([2.000, 3.0000]) ([Diameter, Length]) (End of Live
Image Information flipped part) ;
;
M01 ;
;
[Part Program for the second operation] ;
```

1. Press **[F4]** to enter **Live Image** code to the program.
2. Live Image redraws the part with a flipped orientation, and with the chuck jaws clamped at a position specified by **x** and **y** within the comment **(CLAMP) (x y)** if the comments **(FLIP PART)** and **(CLAMP) (x y)** follow the **M00 (stop program)** instruction in the program.

4.9 Tailstock Setup and Operation

The tailstock is used to support the end of a turning workpiece. It runs along two linear guides. Tailstock motion is controlled through program code, in jog mode, or by a foot pedal.

**NOTE:**

The tailstock is not field-installable.

Tailstocks are controlled with hydraulic pressure in ST-10 (quill only), ST-20, and ST-30 models.

In ST-40 models, the tailstock is positioned and hold force applied by a servo motor.

The tailstock is engaged when the tailstock quill is against the workpiece, applying the specified force.

4

4.9.1 M-code Programming

The ST-10 tailstock is manually positioned, then the quill is hydraulically applied to the workpiece. Command hydraulic quill motion using the following M-codes:

M21: Tailstock Forward

M22: Tailstock Reverse

When an M21 is commanded, the tailstock quill moves forward and maintains continuous pressure. The tailstock body should be locked in place before commanding an M21.

When an M22 is commanded, the tailstock quill moves away from the workpiece. Continuous hydraulic pressure is applied to prevent the quill drifting forward.

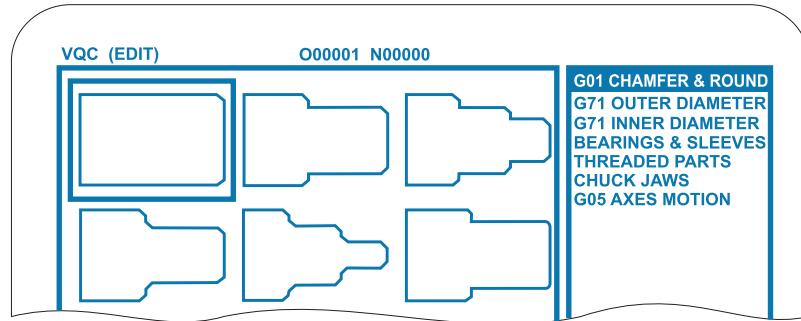
4.10 Visual Quick Code

To start Visual Quick Code (VQC), press [MDI/DNC], then [PROGRAM]. Select vqc from the tabbed menu.

4.10.1 Selecting a Category

To select a category:

F4.32: VQC Part Category Selection



1. Use the arrow keys to select the parts category whose description closely matches the desired part.
2. Press **[ENTER]**.

A set of illustrations of the parts in that category appear.

4.10.2 Selecting a Part Template

To select a part template:

1. Use the arrow keys to select a template on the page.
2. Press **[ENTER]**.

The console displays an outline of the part and waits for you to enter values to make the selected part.

4.10.3 Entering the Data

The control prompts the programmer for information about the selected part. Once the information is entered, the control asks where to place the G-code:



NOTE:

*The program is also available for editing in **Edit** mode. It is a good idea to check the program by running it in **Graphics** mode.*

1. **Select/Create a Program** – This adds the new lines of code to the selected program.
 - a. A window opens prompting selection of a program name.
 - b. Highlight the program and press **[ENTER]**.
If the program already contains code, VQC enters the new code at the beginning of the program before the existing code.
 - c. You have the option to create a new program by entering a program name and pressing **[ENTER]** to add the lines of code to the new program.
2. **Add to Current Program** – The code generated by VQC is added after the cursor.
3. **MDI** – The code outputs to **MDI** and anything in MDI is overwritten.
4. **Cancel** – The window closes and the program values are displayed.

4.11 Subroutines

Subroutines (subprograms) are usually a series of commands that are repeated several times in a program. Instead of repeating the commands many times in the main program, subroutines are written in a separate program. The main program has a single command that calls the subroutine program. A subroutine is called using M97 or M98 and a P code.

When using M97, the P code (nnnnn) is the same as the program location (Nnnnnn) of the subroutine. When using M98, the P code (nnnnn) is the same as the program number (Onnnnn) of the subroutine.

The subroutines can include an L for repeat count. If there is an L, the subroutine call is repeated that number of times before the main program continues with the next block.

When using M97 the subprogram must be within the main program and when using M98 the subprogram must reside in the control memory or hard drive (optional).

4.12 More Information Online

For updated and supplemental information, including tips, tricks, maintenance procedures, and more, go to www.HaasCNC.com and select the **Resource Center**.

Chapter 5: Options Programming

5.1 Options Programming

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.

5.2 Macros (Optional)

5.2.1 Introduction

**NOTE:**

This control feature is optional; contact your dealer for information.

5

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.

A Macro is any routine/subprogram that runs multiple times. A macro statement assigns a value to a variable or reads a value from a variable, evaluates an expression, conditionally or unconditionally branches to another point within a program, or conditionally repeats some section of program.

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

- **Simple Patterns that are Repeated**

Patterns that recur over and over again can be defined using macros and stored. For example:

- a) Family of parts
- b) Soft jaw machining
- c) User defined canned cycles (such as custom grooving cycles)

- **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-#2950).

- **Probing**

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

- Profiling of a part to determine unknown dimensions for later machining
- Tool calibration for offset and wear values
- Inspection prior to machining to determine material allowance on castings

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 (refer to page **385**)

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]... ;
```

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

Lookahead

Look ahead is of great importance to the macro programmer. 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 dwell is being processed. G103 P1 is used to limit lookahead to 1 block. To make this example work properly, it must be modified as follows:

```
G103 P1 (See the G-code section of the manual for a further
explanation of G103) ;
;
#1101=1 ;
G04 P1. ;
;
;
;
#1101=0 ;
```

Block Look Ahead and Block Delete

The Haas control uses the feature Block Look Ahead to read and prepare for blocks of code ahead of the current block of code being executed. This allows the control smoothly transition from one motion to the next. G103 Limit Block Buffering limits how far ahead the control will look at blocks of code. G103 takes the argument Pnn which specifies how far ahead the control is allowed to look ahead. For additional information, refer to the G and M-code section.

The Haas control also has the ability to skip blocks of code when the **[BLOCK DELETE]** button is pressed. To configure a block of code to be skipped in Block Delete mode, begin the line of code with a / character. Using a

```
/ M99 (Sub-Program Return) ;
```

before a block with

```
M30 (Program End and Rewind) ;
```

allows a program to be used as a program when Block Delete is on. The program is used as a sub-program when Block Delete is off.

5.2.2 Operation Notes

Macro variables can be saved or loaded through the RS-232 or USB ports much like settings and offsets. Refer to page 5.

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. The macro variable is set by entering a value and then pressing [**ENTER**]. Macro variables can be cleared by pressing [**ORIGIN**], which will clear all variables.
3. Entering the macro variable number and pressing up or down arrow will search for that variable.
4. The variables displayed represent values of the variables during running of the program. At times, this may be up to 15 blocks ahead of actual machine actions. Debugging programs is easier when inserting a G103 P1 at the beginning of a program to limit block buffering and then removing the G103 P1 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 of sending values to and setting the local variables of a macro subroutine.

The following two tables indicate the mapping of the alphabetic address variables to the numeric variables used in a macro subroutine.

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
G	-	T	20
H	11	U	21
I	4	V	22
J	5	W	23

Address	Variable	Address	Variable
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:

G65, P9910, A1, B2, C3

The values are passed to macro subroutines according to the following 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 one could send two sets of coordinate locations to a macro subroutine. Local variables #4 through #9 would be set to .0001 through .0006 respectively.

Example:

```
G65 P2000 I1 J2 K3 I4 J5 K6;
```

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

Macro Variables

There are three categories of macro variables: system variables, global variables, and local variables. Constants are floating point values placed in a macro expression. They can be combined with addresses A...Z or they can stand alone when used within an expression. Examples of constants are .0001, 5.3 or -10.

Local Variables

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



NOTE:

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

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

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

Global Variables

Global variables are variables that are accessible at all times. There is only one copy of each global variable. Global variables occur in three ranges: 100-199, 500-699 and 800-999. The global variables remain in memory when power is turned off.

Occasionally, there have been some macros written for factory installed options that use the global variables. For example, probing, pallet changers, etc. When using global variables, be sure they are not in use by another program on the machine.

System Variables

System variables give the programmer the ability to interact with a variety of control conditions. By setting a system variable, the function of the control can be modified. By reading a system variable, a program can modify its behavior based on the value in the variable. Some system variables have a Read Only status; this means that the programmer cannot modify them. A brief table of currently implemented system variables follows with an explanation of their use.

VARIABLES	USAGE
#0	Not a number (read only)
#1-#33	Macro call arguments
#100-#199	General-purpose variables saved on power off
#500-#549	General-purpose variables saved on power off
#550-#580	Probe calibration data (if equipped)
#581-#699	General-purpose variables saved on power off
#700-#749	Hidden variables for internal use only.
#800-#999	General-purpose variables saved on power off
#1000-#1063	64 discrete inputs (read only)
#1064-#1068	Maximum axis loads for X, Y, Z, A, and B Axes respectively
#1080-#1087	Raw analog to digital inputs (read only)
#1090-#1098	Filtered analog to digital inputs (read only)

VARIABLES	USAGE
#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 TT Axes respectively
#2001-#2050	X Axis tool shift offsets
#2051-#2100	Y Axis tool shift offsets
#2101-#2150	Z Axis tool shift offsets
#2201-#2250	Tool nose radius offsets
#2301-#2350	Tool tip direction
#2701-#2750	X Axis tool wear offsets
#2751-#2800	Y Axis tool wear offsets
#2801-#2850	Z Axis tool wear offsets
#2901-#2950	Tool nose radius wear offsets
#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)

VARIABLES	USAGE
#3021	Cycle start timer
#3022	Feed timer
#3023	Present cycle time
#3024	Last cycle time
#3025	Previous cycle time
#3026	Tool in spindle (read only)
#3027	Spindle RPM (read only)
#3030	Single block
#3031	Dry run
#3032	Block delete
#3033	Opt stop
#3901	M30 count 1
#3902	M30 count 2
#4001-#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-#5006	Previous block end position
#5021-#5026	Present machine coordinate position
#5041-#5046	Present work coordinate position

VARIABLES	USAGE
#5061-#5069	Present skip signal position - X, Z, Y, A, B, C, U, V, W
#5081-#5086	Present tool offset
#5201-#5206	Common offset
#5221-#5226	G54 work offsets
#5241-#5246	G55 work offsets
#5261-#5266	G56 work offsets
#5281-#5286	G57 work offsets
#5301-#5306	G58 work offsets
#5321-#5326	G59 work offsets
#5401-#5450	Tool feed timers (seconds)
#5501-#5550	Total tool timers (seconds)
#5601-#5650	Tool life monitor limit
#5701-#5750	Tool life monitor counter
#5801-#5850	Tool load monitor maximum load sensed so far
#5901-#6000	Tool load monitor limit
#6001-#6277	Settings (read only)  NOTE: <i>The low order bits of large values do not appear in the macro variables for settings.</i>
#6501-#6999	Parameters (read only)  NOTE: <i>The low order bits of large values do 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)	G114 (G154 P3) additional work offsets
#7061-#7066 (#14061-#14066)	G115 (G154 P4) additional work offsets
#7081-#7086 (#14081-#14086)	G116 (G154 P5) additional work offsets
#7101-#7106 (#14101-#14106)	G117 (G154 P6) additional work offsets
#7121-#7126 (#14121-#14126)	G118 (G154 P7) additional work offsets
#7141-#7146 (#14141-#14146)	G119 (G154 P8) additional work offsets
#7161-#7166 (#14161-#14166)	G120 (G154 P9) additional work offsets
#7181-#7186 (#14181-#14186)	G121 (G154 P10) additional work offsets
#7201-#7206 (#14201-#14206)	G122 (G154 P11) additional work offsets
#7221-#7226 (#14221-#14221)	G123 (G154 P12) additional work offsets
#7241-#7246 (#14241-#14246)	G124 (G154 P13) additional work offsets
#7261-#7266 (#14261-#14266)	G125 (G154 P14) additional work offsets
#7281-#7286 (#14281-#14286)	G126 (G154 P15) additional work offsets
#7301-#7306 (#14301-#14306)	G127 (G154 P16) additional work offsets
#7321-#7326 (#14321-#14326)	G128 (G154 P17) additional work offsets
#7341-#7346 (#14341-#14346)	G129 (G154 P18) additional work offsets
#7361-#7366 (#14361-#14366)	G154 P19 additional work offsets
#7381-#7386 (#14381-#14386)	G154 P20 additional work offsets
#8550	Tool/tool group id
#8552	Maximum recorded vibrations

VARIABLES	USAGE
#8553	X Axis tool shift offsets
#8554	Z Axis tool shift offsets
#8555	Tool nose radius offsets
#8556	Tool tip direction
#8559	X Axis tool wear offsets
#8560	Z Axis tool wear offsets
#8561	Tool nose radius wear offsets
#8562	Tool feed timers
#8563	Total tool timers
#8564	Tool life monitor limit
#8565	Tool life monitor counter
#8566	Tool load monitor maximum load sensed so far
#8567	Tool load monitor limit
#14401-#14406	G154 P21 additional work offsets
#14421-#14426	G154 P22 additional work offsets
#14441-#14446	G154 P23 additional work offsets
#14461-#14466	G154 P24 additional work offsets
#14481-#14486	G154 P25 additional work offsets
#14501-#14506	G154 P26 additional work offsets
#14521-#14526	G154 P27 additional work offsets
#14541-#14546	G154 P28 additional work offsets
#14561-#14566	G154 P29 additional work offsets
#14581-#14586	G154 P30 additional work offsets

VARIABLES	USAGE
#14581+ (20n) - #14586+ (20n)	G154 P (30+n)
#15961-#15966	G154 P99 additional work offsets

5.2.3 System Variables In-Depth

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

1-Bit Discrete Inputs

Inputs designated as spare can be connected to external devices and used by the programmer.

1-Bit Discrete Outputs

The Haas control is capable of controlling up to 56 discrete outputs. However, a number of these outputs are already reserved for use by the Haas controller.

Maximum Axis Loads

The following variables contain the maximum axis loads 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 the Macro Variable is read.

#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

Use the following macro variables to read or set the following geometry, shift, or wear offset values:

#2001-#2050	X-Axis geometry/shift offset
#2051-#2100	Y-Axis geometry/shift offset
#2101-#2150	Z-Axis geometry/shift offset
#2201-#2250	Tool nose radius geometry
#2301-#2350	Tool tip direction
#2701-#2750	X-Axis tool wear
#2751-#2800	Y-Axis tool wear
#2801-#2850	Z-Axis tool wear
#2901-#2950	Tool nose radius wear

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 is the Single Block Suppression parameter. It overrides the Single Block function in G-code. In the following example Single Block is ignored when #3003 is set equal to 1. After #3003 is set = 1, each G-code command (lines 2-4) are executed continuously even though the Single Block function is on. When #3003 is set equal to zero, Single Block operates as normal. That is, the user must press **[CYCLE START]** to start each line of code (lines 6-8).

```
#3003=1 ;
G54 G00 G90 X0 Z0 ;
G81 R0.2 Z-0.1 F20 L0 ;
S2000 M03 ;
#3003=0 ;
T02 M06 ;
G83 R0.2 Z-1. F10. L0 ;
X0. Z0. ;
```

Variable #3004

Variable #3004 overrides specific control features while running.

The first bit disables **[FEED HOLD]**. If **[FEED HOLD]** is not used during a section of code, set variable #3004 to 1 before the specific lines of code. After that section of code set #3004 to 0 to restore the function of **[FEED HOLD]**. 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) ;

The following is a map of variable #3004 bits and the associated overrides. E – Enabled D – Disabled

#3004	Feed Hold	Feed Rate Override	Exact Stop Check
0	E	E	E
1	D	E	E
2	E	D	E
3	D	D	E
4	E	E	D
5	D	E	D
6	E	D	D
7	D	D	D

#3006 Programmable Stop

Stops can be programmed which act like an M00. The control stops and waits until Cycle Start is pressed. Once Cycle Start is pressed, the program continues with the block after the #3006. In the following example, the first 15 characters of the comment are displayed on the lower left part of the screen.

IF [#1 EQ #0] THEN #3006=101(comment here);

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

The grouping of G codes permits more efficient processing. G codes with similar functions are usually under the same group. For example, G90 and G91 are under group 3. These variables store the last or default G code for any of 21 groups. By reading the group code, a macro program can change the behavior of the G-code. If #4003 contains 91, then a macro program could determine that all moves should be incremental rather than absolute. There is no associated variable for group zero; group zero G codes are Non-modal.

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

Address codes A-Z (excluding G) are maintained as modal values. The information represented by the last line of code interpreted by the lookahead process is contained in variables #4101 through #4126. The numeric mapping of variable numbers to alphabetic addresses corresponds to the mapping under alphabetic addresses. For example, the value of the previously interpreted D address is found in #4107 and the last interpreted I value is #4104. When aliasing a macro to an M-code, you may not pass variables to the macro using variables #1 - #33; instead, use the values from #4101 - #4126 in the macro.

#5001-#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

5

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

#5021-#5026 Current Machine Coordinate Position

The current position in machine coordinates can be obtained through #5021- #5025, X, Z, Y, A, and B, respectively.

**NOTE:**

The values cannot be read while the machine is in motion.

The value of #5022 (Z) has tool length compensation applied to it.

#5041-#5046 Current Work Coordinate Position

The current position in the current work coordinates can be obtained through #5041-#5046, X, Z, Y, A, B and C, respectively.

**NOTE:**

The values cannot be read while the machine is in motion.

#5061-#5069 Current Skip Signal Position

The position where the last skip signal was triggered can be obtained through #5061 - #5069, X, Z, Y, A, B, C, U, V, and W respectively. Values are given in the current work coordinate system and can be used while the machine is in motion.

#5081-#5086 Tool Length Compensation

The current total tool length compensation that is being applied to the tool is returned. This includes tool geometry referenced by the current modal value set in the T code plus the wear value.

#6996-#6999 Parameter Access Using Macro Variables

It is possible for a program to access parameters 1 to 1000 and any of the parameter bits, as follows:

#6996: Parameter Number

#6997: Bit Number (optional)

#6998: Contains value of parameter number in variable #6996

#6999: Contains bit value (0 or 1) of parameter bit specified in variable #6997.

**NOTE:**

Variables #6998 and #6999 are read-only.

Usage

To access the value of a parameter, the number of that parameter is copied into variable #6996, after which, the value of that parameter is available using macro variable #6998, as shown:

```
#6996=601 (Specify parameter 601) ;
#100=#6998 (Copy the value of parameter 601 to variable #100)
;
```

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

```
#6996=57 (Specify parameter 57) ;
#6997=0 (Specify bit zero) ;
#100=#6999 (Copy parameter 57 bit 0 to variable #100) ;
```



NOTE:

Parameter bits are numbered 0 through 31. 32-bit parameters are formatted, on-screen, with bit 0 at the top-left, and bit 31 at the bottom-right.

Work Offsets

All work offsets can be read and set within a macro expression. This allows the programmer to preset coordinates to approximate locations, or to set coordinates to values based upon the results of skip signal locations and calculations. When any of the offsets are read, the interpretation lookahead queue is stopped until that block is executed.

#5201- #5206	G52 X, Z, Y, A, B, C offset values
#5221- #5226	G54 X, Z, Y, A, B, C offset values
#5241- #5246	G55 X, Z, Y, A, B, C offset values
#5261- #5266	G56 X, Z, Y, A, B, C offset values
#5281- #5286	G57 X, Z, Y, A, B, C offset values
#5301- #5306	G58 X, Z, Y, A, B, C offset values
#5321- #5326	G59 X, Z, Y, A, B, C offset values

System Variables In-Depth

#7001- #7006	G110 (G154 P1) additional work offsets
#7021-#7026 (#14021-#14026)	G111 (G154 P2) additional work offsets
#7041-#7046 (#14041-#14046)	G114 (G154 P3) additional work offsets
#7061-#7066 (#14061-#14066)	G115 (G154 P4) additional work offsets
#7081-#7086 (#14081-#14086)	G116 (G154 P5) additional work offsets
#7101-#7106 (#14101-#14106)	G117 (G154 P6) additional work offsets
#7121-#7126 (#14121-#14126)	G118 (G154 P7) additional work offsets
#7141-#7146 (#14141-#14146)	G119 (G154 P8) additional work offsets
#7161-#7166 (#14161-#14166)	G120 (G154 P9) additional work offsets
#7181-#7186 (#14181-#14186)	G121 (G154 P10) additional work offsets
#7201-#7206 (#14201-#14206)	G122 (G154 P11) additional work offsets
#7221-#7226 (#14221-#14221)	G123 (G154 P12) additional work offsets
#7241-#7246 (#14241-#14246)	G124 (G154 P13) additional work offsets
#7261-#7266 (#14261-#14266)	G125 (G154 P14) additional work offsets
#7281-#7286 (#14281-#14286)	G126 (G154 P15) additional work offsets
#7301-#7306 (#14301-#14306)	G127 (G154 P16) additional work offsets
#7321-#7326 (#14321-#14326)	G128 (G154 P17) additional work offsets

#7341-#7346 (#14341-#14346)	G129 (G154 P18) additional work offsets
#7361-#7366 (#14361-#14366)	G154 P19 additional work offsets
#7381-#7386 (#14381-#14386)	G154 P20 additional work offsets

Variable Usage

All variables are referenced with a number sign (#) followed by a positive number; such as: #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.

Variables can be used in place of G-code address where address refers to the letters A - Z.

In the block:

```
N1 G0 X1.0 ;
```

the variables can be set to the following values:

```
#7 = 0 ;
#1 = 1.0 ;
```

and the block replaced by:

```
N1 G#7 X#1 ;
```

The values in the variables at runtime are used as the address values.

#8550-#8567 Tooling

These variables provide information on tooling. Set variable #8550 to the tool or tool group number, then access information for the selected tool/tool group using the read-only macros #8551-#8567. If specifying a tool group number, the selected tool is the next tool in that group.

5.2.4 Address Substitution

The usual method of setting control addresses A - Z is the address followed by a number. For example:

```
G01 X1.5 Z3.7 F.02 ;
```

sets addresses G, X, Z, and F to 1, 1.5, 3.7, and 0.02 respectively, and thus instructs the control to move linearly, G01, to position X = 1.5 and Z = 3.7 at a feed rate of 0.02 inches per revolution. Macro syntax allows the address value to be replaced with any variable or expression.

The previous statement can be replaced by the following code:

```
#1= 1 ;
#2= 0.5 ;
#3= 3.7 ;
#4= 0.02 ;
G#1 X[#1+#2] Z#3 F#4 ;
```

The permissible syntax on addresses A - Z (exclude N or O) is as follows:

<address><-><variable>	A-#101
<address>[<expression>]	Z [#5041+3.5]
<address><->[<expression>]	Z-[SIN[#1]]

If the value of the variable does not agree with the range of the address, then the usual control alarm results. For instance, the following code results in an invalid G code alarm because there is no G143 code:

```
#1= 143 ;
G#1 ;
```

When a variable or expression is used in place of an address value, the value is rounded to the least significant digit.

```
#1= .123456 ;
```

```
G1 X#1 ;
```

moves the machine tool to .1235 on the X-axis. If the control is in the metric mode, the tool 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:

```
(#1 is undefined) ;
G00 X1.0 Z#1 ;
```

becomes

```
G00 X1.0 (no Z 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

Function	Argument	Returns	Notes
ABS[]	Decimal	Decimal	Absolute value
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 argument `ROUND` is rounded to the addresses significant precision. For metric and angle dimensions, three-place precision is the default. For inch, four-place precision is the default. Integral addresses such as `T` are rounded normally.

```
#1= 1.00333 ;
G00 X [ #1 + #1 ] ;
(X moves to 2.0067) ;
G00 X [ ROUND[ #1 ] ] + ROUND[ #1 ] ;
(X moves to 2.0066) ;
G00 C [ #1 + #1 ] ;
(Axis moves to 2.007) ;
G00 C [ ROUND[ #1 ] ] + ROUND[ #1 ] ;
(Axis moves to 2.006) ;
```

Fix vs. Round

```
#1=3.54 ;
#2=ROUND [#1] ;
#3=FIX [#1].
```

#2 will be set to 4. #3 will be set to 3.

Operators

Operators can be classified into three categories: Arithmetic, Logical and Boolean.

Arithmetic Operators

Arithmetic operators consist of unary and binary operators. They are:

+	- Unary plus	+1.23
-	- Unary minus	-[COS[30]]
+	- Binary addition	#1=#1+5
-	- Binary subtraction	#1=#1-1
*	- Multiplication	#1=#2*#3
/	- Division	#1=#2/4
MOD	- Remainder	#1=27 MOD 20 (#1 contains 7)

Logical Operators

Logical operators are operators that work on binary bit values. Macro variables are floating point numbers. When logical operators are used on macro variables, only the integer portion of the floating point number is used. The logical operators are:

OR - logically OR two values together

XOR - Exclusively OR two values together

AND - Logically AND two values together

Examples:

```
#1=1.0 ;
#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:

Care must be taken when using logical operators so that the desired result is achieved.

Boolean Operators

Boolean operators always evaluate to 1.0 (TRUE) or 0.0 (FALSE). There are six Boolean operators. These operators are not restricted to conditional expressions, but they most often are used in conditional expressions. They are:

EQ - Equal To

NE - Not Equal To

GT - Greater Than

LT - Less Than

GE - Greater Than or Equal To

LE - Less Than or Equal To

The following are four examples of how Boolean and Logical operators can be used:

Example	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.

Example	Explanation
#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.

Expressions

Expressions are defined as any sequence of variables and operators surrounded by the square brackets [and]. There are two uses for expressions: conditional expressions or arithmetic expressions. Conditional expressions return FALSE (0.0) or TRUE (any non zero) values. Arithmetic expressions use arithmetic operators along with functions to determine a value.

Conditional Expressions

In the Haas control, all expressions set a conditional value. The value is either 0.0 (FALSE) or the value is nonzero (TRUE). The context in which the expression is used determines if the expression is a conditional expression. Conditional expressions are used in the IF and WHILE statements and in the M99 command. Conditional expressions can make use of Boolean operators to help evaluate a TRUE or FALSE condition.

The M99 conditional construct is unique to the Haas control. Without macros, M99 in the Haas control has the ability to branch unconditionally to any line in the current subroutine by placing a P code on the same line. For example:

```
N50 M99 P10 ;
```

branches to line N10. It does not return control to the calling subroutine. With macros enabled, M99 can be used with a conditional expression to branch conditionally. To branch when variable #100 is less than 10 we could code the above line as follows:

```
N50 [#100 LT 10] M99 P10 ;
```

In this case, the branch occurs only when #100 is less than 10, otherwise processing continues with the next program line in sequence. In the above, the conditional M99 can be replaced with

```
N50 IF [#100 LT 10] GOTO10 ;
```

Arithmetic Expressions

An arithmetic expression is any expression using variables, operators, or functions. An arithmetic expression returns a value. Arithmetic expressions are usually used in assignment statements, but are not restricted to them.

Examples of Arithmetic expressions:

```
#101=#145*#30 ;
#1=#1+1 ;
X[#105+COS[#101]] ;
#[#2000+#13]=0 ;
```

Assignment Statements

Assignment statements allow the programmer to modify variables. The format of the assignment statement is:

```
<expression>=<expression>
```

The expression on the left of the equal sign must always refer to a macro variable, whether directly or indirectly. The following macro initializes a sequence of variables to any value. Here both direct and indirect assignments are used.

```
O0300(Initialize an array 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 ;
```

The above macro could be used to initialize three sets of variables as follows:

```
G65 P300 B101. S20 (INIT 101..120 TO #0) ;
G65 P300 B501. S5 V1. (INIT 501..505 TO 1.0) ;
G65 P300 B550. S5 V0 (INIT 550..554 TO 0.0) ;
```

The decimal point in B101., etc. would be required.

Control Statements

Control statements allow the programmer to branch, both conditionally and unconditionally. They also provide the ability to iterate a section of code based on a condition.

Unconditional Branch (GOTOnnn and M99 Pnnnn)

In the Haas control, there are two methods of branching unconditionally. An unconditional branch will always branch to a specified block. M99 P15 will branch unconditionally to block number 15. The M99 can be used whether or not macros is installed and is the traditional method for branching unconditionally in the Haas control. GOTO15 does the same as M99 P15. In the Haas control, a GOTO command can be used on the same line as other G-codes. The GOTO is executed after any other commands like M codes.

Computed Branch (GOTO#n and GOTO [expression])

Computed branching allows the program to transfer control to another line of code in the same subprogram. The block can be computed as the program is running, using the GOTO [expression] form. Or the block can be passed in through a local variable, as in the GOTO#n form.

The GOTO will round the variable or expression result that is associated with the Computed branch. For instance, if #1 contains 4.49 and GOTO#1 is executed, the control will attempt to transfer to a block containing N4. If #1 contains 4.5, then execution will transfer to a block containing N5.

The following code skeleton could be developed to make a program that adds serial numbers to parts:

```

O9200 (Engrave digit at current location) ;
(D=Decimal digit to engrave) ;
;
IF [[#7 NE #0] AND [#7 GE 0] AND [#7 LE 9]] GOTO99 ;
#3000=1 (Invalid digit) ;
;
N99
#7=FIX[#7] (Truncate any fractional part) ;
;
GOTO#7 (Now engrave the digit) ;
;
N0 (Do digit zero) ;
M99 ;
;
N1 (Do digit one) ;

```

```
;  
M99 ;  
;  
N2 (Do digit two) ;  
;  
...  
;  
(etc.,....)
```

With the above subroutine, you would engrave digit five with the following call:

```
G65 P9200 D5 ;
```

Computed **GOTOS** using expression could be used to branch processing based on the results of reading hardware inputs. An example might look like the following:

```
GOTO [#1030*2]+#1031] ;  
NO(1030=0, 1031=0) ;  
...  
M99 ;  
N1(1030=0, 1031=1) ;  
...  
M99 ;  
N2(1030=1, 1031=0) ;  
...  
M99 ;  
N3(1030=1, 1031=1) ;  
...  
M99 ;
```

The discrete inputs always return either 0 or 1 when read. The **GOTO [expression]** will branch to the appropriate line of code based on the state of the two discrete inputs #1030 and #1031.

Conditional Branch (IF and M99 Pnnnn)

Conditional branching allows the program to transfer control to another section of code within the same subroutine. Conditional branching can only be used when macros are enabled. The Haas control allows two similar methods for accomplishing conditional branching:

```
IF [<conditional expression>] GOTO
```

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 GOTOn.

5

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 subroutine.

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 subroutine can be executed any number of times by using the **L** address.

```
M98 P2000 L5 ;
```

This is limited since you cannot terminate execution of the subroutine on condition. Macros allow flexibility with the WHILE-DO-END construct. For example:

```
WHILE [<conditional expression>] DOn ;
<statements> ;
ENDn ;
```

This executes the statements between **DOn** and **ENDn** as long as the conditional expression evaluates to True. The brackets in the expression are necessary. If the expression evaluates to False, then the block after **ENDn** is executed next. **WHILE** can be abbreviated to **WH**. The **DOn-ENDn** portion of the statement is a matched pair. The value of n is 1-3. This means that there can be no more than three nested loops per subroutine. A 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 subroutine can have up to three levels of nesting. If there is a need to nest to a level greater than 3, then the segment containing the three lowest levels of nesting can be made into a subroutine thus overcoming the limitation.

If two separate **WHILE** loops are in a subroutine, they can use the same nesting index. For example:

```
#3001=0 (WAIT 500 MILLISECONDS) ;
WH [#3001 LT 500] DO1 ;
END1 ;
<Other statements>
#3001=0 (WAIT 300 MILLISECONDS) ;
WH [#3001 LT 300] DO1 ;
END1 ;
```

You can use **GOTO** to jump out of a region encompassed by a **DO-END**, but you cannot use a **GOTO** to jump into it. Jumping around inside a **DO-END** region using a **GOTO** is allowed.

An infinite loop can be executed by eliminating the WHILE and expression. Thus,

```
DO1 ;
<statements>
END1 ;
```

executes until the RESET key is pressed.



CAUTION: *The following code can be confusing:*

```
WH [#1] D01 ;
END1 ;
```

In this example, an alarm results indicating no Then was found; Then refers to the D01. Change D01 (zero) to D01 (letter O).

G65 Macro Subroutine Call Option (Group 00)

G65 is the command that calls a subroutine 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, subroutine 1000 is called once without conditions passed to the subroutine. G65 calls are similar to, but not the same as, M98 calls. G65 calls can be nested up to 9 times, which means, program 1 can call program 2, program 2 can call program 3 and program 3 can call program 4.

Example 1:

```
G65 P1000 (Call subroutine 1000 as a macro) ;
M30 (Program stop) ;
O1000 (Macro Subroutine) ;
...
M99 (Return from Macro Subroutine) ;
```

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.

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 subroutine 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. The following table lists which Haas parameters are reserved for macro subroutine aliasing.

T5.1: G-Code Aliasing

Haas Parameter	O Code
91	9010
92	9011
93	9012
94	9013
95	9014
96	9015
97	9016
98	9017
99	9018
100	9019

T5.2: M-Code Aliasing

Haas Parameter	O Code
81	9000
82	9001
83	9002
84	9003
85	9004
86	9005
87	9006
88	9007
89	9008
90	9009

Setting an aliasing parameter to 0 disables aliasing for the associated subroutine. If an aliasing parameter is set to a G-code and the associated subroutine is not in memory, then an alarm is given.

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.

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. The commands provided for this are POPEN, DPRNT [] and PCLOS.

Communication Preparatory Commands

POPN and PCLOS are not required on the Haas machine. It has been included so that programs from different controls can be sent to the Haas control.

Formatted Output

The DPRNT statement allows the programmer to send formatted text to the serial port. Any text and any variable can be printed to the serial port. The form of the DPRNT statement is as follows:

```
DPRNT [<text> <#nnnn[wf]>... ] ;
```

DPRNT must be the only command in the block. In the previous example, <text> is any character from A to Z or the letters (+,-,/,* , and the space). When an asterisk is output, it is converted to a space. The <#nnnn[wf]> is a variable followed by a format. The variable number can be any macro variable. The format [wf] is required and consists of two digits within square brackets. Remember that macro variables are real numbers with a whole part and a fractional part. The first digit in the format designates the total places reserved in the output for the whole part. The second digit designates the total places reserved for the fractional part. The total places reserved for output cannot be equal to zero or greater than eight. Thus the following formats are illegal: [00] [54] [45] [36] /* not legal formats */

A decimal point is printed out between the whole part and the fractional part. The fractional part is rounded to the least significant place. When zero places are reserved for the fractional part, then no decimal point is printed out. Trailing zeros are printed if there is a fractional part. At least one place is reserved for the whole part, even when a zero is used. If the value of the whole part has fewer digits than have been reserved, then leading spaces are output. If the value of the whole part has more digits than has been reserved, then the field is expanded so that these numbers are printed.

A carriage return is sent out after every DPRNT block.

DPRNT[] Examples

Code	Output
N1 #1= 1.5436 ;	
N2 DPRNT [X#1[44]*Z#1[03]*T#1[40]] ;	X1.5436 Z 1.544 T 1
N3 DPRNT [***MEASURED*INSIDE*DIAMETER** *] ;	MEASURED INSIDE DIAMETER
N4 DPRNT [] ;	(no text, only a carriage return)
N5 #1=123.456789 ;	
N6 DPRNT [X-#1[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 lookahead interpretation to one block, you would include the following command at the beginning of your program: (This actually results in a two block lookahead.)

```
G103 P1 ;
```

To cancel the lookahead limit, change the command to G103 P0. G103 cannot be used when cutter compensation is active.

Editing

Improperly structured or improperly placed macro statements generate an alarm. Be careful when editing expressions; brackets must be balanced.

The DPRNT [] function can be edited much like a comment. It can be deleted, moved as a whole item, or individual items within the bracket can be edited. Variable references and format expressions must be altered as a whole entity. If you wanted to change [24] to [44], place the cursor so that [24] is highlighted, enter [44] and press the write key. Remember, you can use the [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, the following block contains an address expression in X:

```
G01 X [ COS[ 90 ] ] Z3.0 (CORRECT) ;
```

Here, the X and brackets stand alone and are individually editable items. It is possible, through editing, to delete the entire expression and replace it with a number:

```
G01 X 0 Z3.0 (WRONG) ;
```

This block results in an alarm at runtime. The correct form looks as follows:

```
G01 X0 Z3.0 (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.

5.2.5 FANUC-Style Macro Features 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 block
G67	Modal cancel
M98	Aliasing, T Code Prog 9000, Var#149, enable bit
M98	Aliasing, S Code Prog 9029, Var #147, enable bit
M98	Aliasing, B Code Prog 9028, Var #146, enable bit
SKIP/N	N=1..9
#3007	Mirror Image On flag each axis
#4201-#4320	Current block modal data
#5101-#5106	Current servo deviation

Names for Variables for Display Purposes

ATAN []/[]	Arctangent, FANUC version
BIN []	Conversion from BCD to BIN
BCD []	Conversion from BIN to BCD
FUP []	Truncate fraction to ceiling
LN []	Natural Logarithm
EXP []	Base E exponentiation
ADP []	Re-scale VAR to whole number

BPRNT []	
GOTO-nnnn	

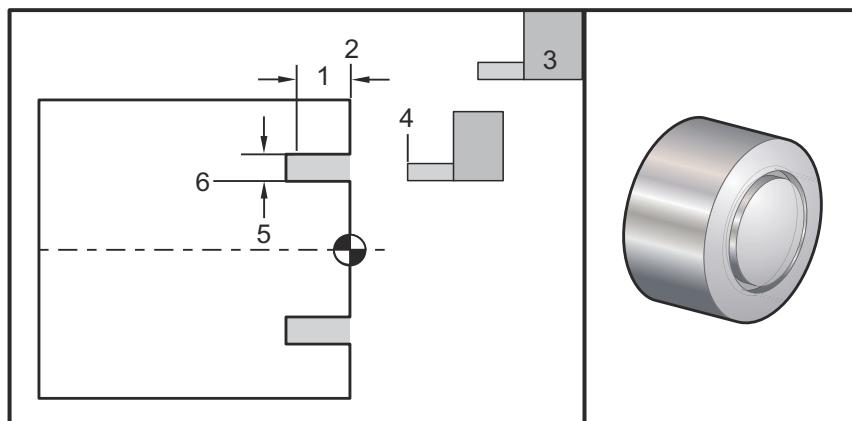
Searching for a block to jump to in the negative direction (i.e. backwards through a program) is not necessary if you use unique N address codes. A block search is made starting from the current block being interpreted. When the end of the program is reached, searching continues from the top of the program until the current block is encountered.

5.2.6 Example Program Using Macros

The following example cuts a face groove in a part using easily-edited variables.

```
%  
O0010 (MACRO G74) ;  
G50 S2000 ;  
G97 S1000 M03 T100 ;  
G00 T101 ;  
#24 = 1.3 (X MINOR DIAMETER) ;  
#26 = 0.14 (Z DEPTH) ;  
#23 = 0.275 (X GROOVE WIDTH) ;  
#20 = 0.125 (TOOL WIDTH) ;  
#22 = -0.95 (Z START POSITION) ;  
#6 = -1. (ACTUAL Z FACE) ;  
#9 = 0.003 (FEED RATE IPR) ;  
G00 X [ #24 + [ #23 * 2 ] - [ 20 * 2 ] ] Z#126 ;  
G74 U - [ [#23 - #20] * 2 ] W - [ #26 + ABS [ #6 - #22 ] ] K  
[ #20 * 0.75 ] I [ #20 * 0.9 ] F#9 ;  
G00 X0 Z0 T100 ;  
M30 ;  
%
```

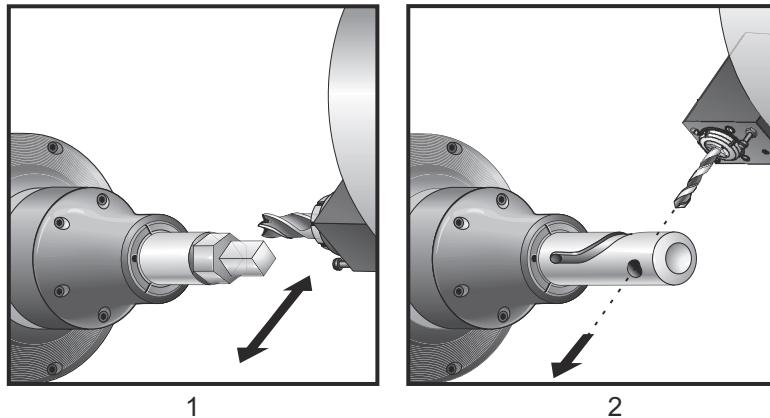
F5.1: Macro G74 Use: [1] Z Depth, [2] Z Face, [3] Groove_Tool, [4] Z Start Position, [5] X Width, [6] X Minor Diameter. Tool Width = 0.125"



5.3 Live Tooling and C Axis

This option is not field installable.

F5.2: Axial and Radial Live Tooling: [1] Axial Tool, [2] Radial Tool.



5.3.1 Live Tooling Introduction

The live tooling option allows the user to drive VDI axial or radial tools to perform such operations as milling, drilling, or slotting. Milling shapes is possible using the C-Axis and/or the Y Axis.

Programming Notes

The live tool drive automatically turns itself off when a tool change is commanded.

For the best milling accuracy, use the spindle clamp M Codes (M14 - Main Spindle / M114 - Secondary Spindle) before machining. The spindle automatically unclamps when a new main spindle speed is commanded or [RESET] is pressed.

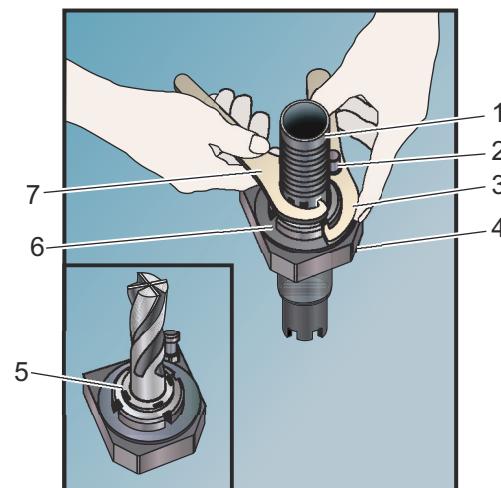
Maximum live tooling drive speed is 3000 RPM.

Haas live tooling is designed for medium duty milling, e.g.: 3/4" diameter end mill in mild steel max.

5.3.2 Live Tooling Cutting Tool Installation

To install tools for live tool cutting:

- F5.3:** ER-32-AN Tube Wrench and Spanner: [1] ER-32-AN Tube wrench, [2] Pin, [3] Spanner 1, [4] Tool holder, [5] ER-32-AN nut insert, [6] Collet housing nut, [7] Spanner 2.



1. Insert the tool bit into the ER-AN nut insert. Thread the nut insert into the collet housing nut.
2. Place the ER-32-AN tube wrench over the tool bit and engage the teeth of the ER-AN nut insert. Snug the ER-AN nut insert by hand using the tube wrench.
3. Place Spanner 1 [3] over the pin and lock it against the collet housing nut. It may be necessary to turn the collet housing nut to engage the spanner.
4. Engage the teeth of the tube wrench with Spanner 2 [7] and tighten.

5.3.3 Live Tool Mounting in Turret

Radial live tool holders can be adjusted for optimum performance during milling with the Y-Axis. The body of the tool holder can be rotated in the tool pocket relative to the X-axis. This allows for adjustment of the parallelism of the cutting tool with the X-axis.

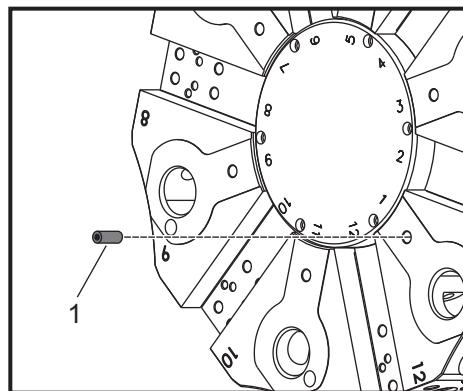
Adjustment set screws are standard on all radial live tool heads. An alignment dowel pin is included in Haas radial live tool kits.

Mounting and Alignment

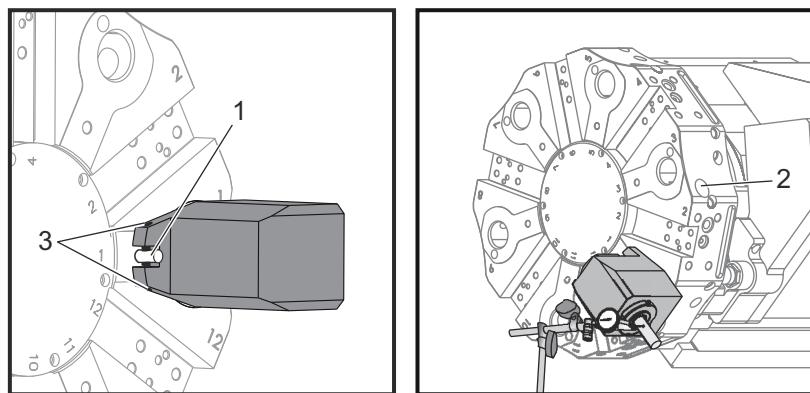
To mount and install live tools:

1. Install the alignment dowel pin that comes with the Haas-supplied live tooling holder on the turret.

F5.4: Install Alignment Pin [1]



2. Mount a radial live tool holder and snug adjustment set screws [3] against the dowel pin [1] at a visually even and centered position.
3. Snug the VDI allen bolt to allow for some movement and adjustment of the tool. Make sure the bottom face of the toolholder is clamped flush with the face of the turret.

F5.5: Set Screw Alignment

4. Position the Y-axis at zero.
5. Install a dowel pin, gauge pin, or cutting tool in the tool holder. Make sure the pin or tool sticks out at least 1.25" (32mm). This will be used to run the indicator across to check parallelism to the X-axis.
6. Set an indicator with a magnetic base on a rigid surface (for example, the tailstock base). Position the indicating tip on the end point of the pin and zero the indicator dial.
7. Sweep the indicator along the top of the pin or tool in the X-Axis.
8. Adjust the set screws [3] and keep indicating across the top of the pin or tool until the indicator reads zero along X-axis travel.
9. Tighten the VDI allen bolt to the recommended torque, and re-check parallelism. Adjust as necessary.
10. Repeat steps 1 through 8 for every radial tool used in set-up.
11. Thread an M10 bolt into the alignment dowel pin [1] and pull to remove the pin.

5.3.4 Live Tooling M-codes

The following M-Codes are used in Live Tooling. Also, refer to the M-codes section starting on page **370**.

M19 Orient Spindle (Optional)

An **M19** orients the spindle to the zero position. Use a **P** or an **R** value to orient the spindle to a specific position (in degrees). Degrees of accuracy - **P** rounds to the nearest whole degree, and **R** rounds to the nearest hundredth of a degree (**x.xx**). View the angle in the **Current Commands Tool Load Screen**.

M119 positions the secondary spindle (DS lathes) the same way.

M133/M134/M135 Live Tool Fwd/Rev/Stop (Optional)

Refer to page **387** for a complete description of these M-codes.

5.3.5 C Axis

The C Axis provides high-precision, bi-directional spindle motion that is fully interpolated with X and/or Z motion. You can command spindle speeds from 0.01 to 60 RPM.

C-Axis operation is dependent on the mass, diameter and length of the workpiece and/or the workholding (chuck). Contact the Haas Applications Department if any unusually heavy, large diameter, or long configuration is used.

5.3.6 Cartesian to Polar Transformation (G112)

Cartesian to Polar coordinate programming that converts X,Y position commands into rotary C-Axis and linear X-axis moves. Cartesian to Polar coordinate programming greatly reduces the amount of code required to command complex moves. Normally a straight line would require many points to define the path, however, in Cartesian, only end points are necessary. This feature allows face machining programming in the Cartesian coordinate system.

Programming Notes

Programmed moves should always position the tool centerline.

Tool paths should never cross the spindle centerline. If necessary re-orient the program so the cut does not go over the center of the part. Cuts that must cross spindle center can be accomplished with two parallel passes on either side of spindle center.

Cartesian to Polar conversion is a modal command. Refer to page 273 for more information on modal G-codes.

5.3.7 Cartesian Interpolation

Cartesian coordinate commands are interpreted into movements of the linear axis (turret movements) and spindle movements (rotation of the work piece).

Example Program

```
%  
O00069 ;  
N6 (Square) ;  
G59 T1111 ( Tool 11, .75 Dia. Endmill, cutting on center) ;  
M154 ;  
G00 C0. ;  
G97 M133 P1500 ;  
G00 Z1. ;  
G00 G98 X2.35 Z0.1 (Position) ;  
G01 Z-0.05 F25. ;  
G112  
G17 (Set to XY plane) ;  
G0 X-.75 Y.5 ;  
G01 X0.45 F10. (Point 1) ;  
G02 X0.5 Y0.45 R0.05 (Point 2) ;  
G01 Y-0.45 (Point 3) ;  
G02 X0.45 Y-0.5 R0.05 (Point 4) ;  
G01 X-0.45 (Point 5) ;  
G02 X-0.5 Y-0.45 R0.05 (Point 6) ;  
G01 Y0.45 (Point 7) ;  
G02 X-0.45 Y0.5 R0.05 (Point 8) ;  
G01 X0.45 Y.6 (Point 9) ;  
G113 ;  
G18 (Set to XZ plane) ;  
G00 Z3. ;  
M30 ;
```

%

Operation (M codes and Settings)

M154 engages the C-Axis and M155 disengages the C-Axis.

Setting 102 - Diameter is used to calculate the feed rate.

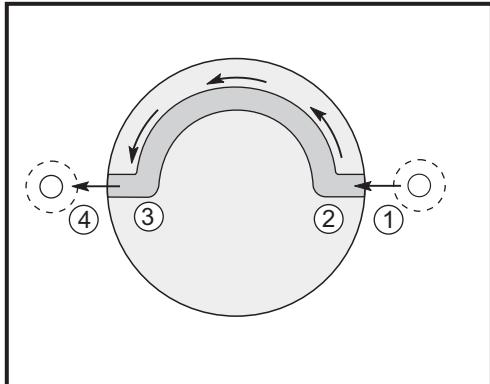
The lathe automatically disengages the spindle brake when you command the C-Axis to move and to reengage it afterwards if the M codes are still active.

C-Axis incremental moves are possible using the H address code as shown in the following example:

```
G0 C90. (C-Axis moves to 90. deg.) ;
H-10. (C-Axis moves to 80. deg. from the previous 90 deg
position) ;
```

Sample Programs

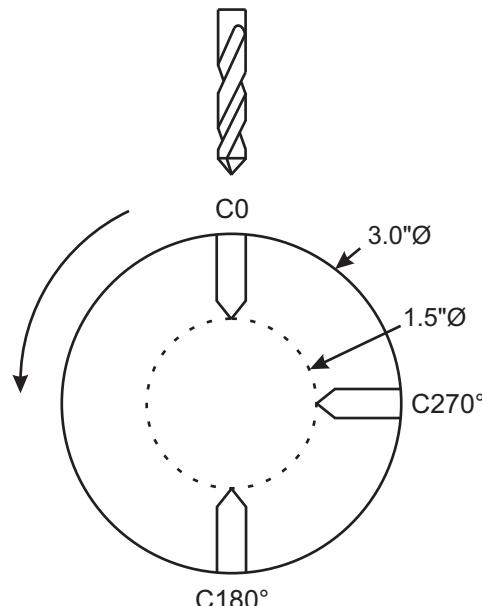
F5.6: Cartesian Interpolation Example 1



Example #1

```
%
O0054 ;
T101 ;
G54 ;
M133 P2000 (Live Tool On) ;
M154 (Engage C-axis) ;
G00 G98 (feed/min) X2.0 Z0 ;
C90 ;
G01 Z-0.1 F6.0 (position 1) ;
X1.0 (position 2) ;
C180. F10.0 (position 3) ;
X2.0 (position 1) ;
G00 Z0.5 ;
M155 ;
M135 ;
G53 X0 ;
G53 Z0 ;
M30 ;
%
```

F5.7: Cartesian Interpolation Example 2



5

```

%
o51122 (CARTESIAN INTERPOLATION EX 2);
(G54 X0 Y0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is a drill) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G19 (Call YZ plane) ;
G98 (Feed per min) ;
M154 (Engage C Axis) ;
G00 G54 X3.25 C0. Y0. Z0.25 ;
(Rapid to 1st position) ;
P1500 M133 (Live tool CW at 1500 RPM) ;
M08 (Coolant on) ;
G00 Z-0.75 (Rapid to Z position) ;
(BEGIN CUTTING BLOCKS) ;
G75 X1.5 I0.25 F6. (Begin G75 on 1st hole) ;
G00 C180. (Rotate C axis to new position) ;
G75 X1.5 I0.25 F6. (Begin G75 on 2nd hole) ;
G00 C270. (Rotate C axis to new position) ;
G75 X1.5 I0.25 F6. (Begin G75 on 3rd hole) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.25 M09 (Rapid retract, coolant off) ;

```

```
M155 (Disengage C Axis) ;  
M135 (Live tool off) ;  
G18 (Return to XZ plane) ;  
G53 X0 (X home) ;  
G53 Z0 (Z home) ;  
M30 (End program) ;  
%
```

5.3.8 Tool Radius Cutter Comp with G112 and G17

Tool Radius Cutter Compensation shifts the programmed tool path so that the centerline of the tool is moved to the left or right of the programmed path. The Offset page is used to enter the amount that the tool path is shifted in the radius column. The offset is entered as a radius value for both the geometry and the wear columns. The compensated value is calculated by the control from the values entered in the **Radius**. When using **G112**, Cutter Radius Compensation is only available in **G17 (XY)** plane. The tool tip does not need to be defined.

Tool Radius Cutter Compensation using Y-Axis in G18 (Z-X motion) and G19 (Z-Y motion) Planes.

Tool Radius Cutter Compensation shifts the programmed tool path so that the centerline of the tool is moved to the left or right of the programmed path. The Offset page is used to enter the amount that the tool path is shifted in the radius column. The offset is entered as a radius value for both the geometry and the wear columns. The compensated value is calculated by the control from the values entered in the **Radius**. Cutter radius compensation using Y-axis **MUST NOT** include C-Axis in any of the synchronized motion. Tool tip does not need to be defined.

- G41 selects cutter compensation left.
- G42 selects cutter compensation right.
- G40 cancels cutter compensation.

Offset values entered for the radius are in positive numbers. If the offset contains a negative value, cutter compensation operates as though the opposite G code is specified. For example, a negative value entered for a **G41** behaves as if a positive value is entered for **G42**.

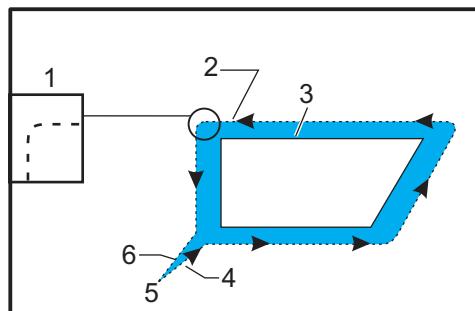
When selecting **YASNAC** for Setting 58, the control must be able to position the side of the tool along all of the edges of the programmed contour without over cutting the next two motions. A circular motion joins all of the outside angles.

When selecting **FANUC** for Setting 58, the control does not require that the tool cutting edge be placed along all edges of the programmed contour, preventing over-cutting. Outside angles less than or equal to 270° are joined by a sharp corner and outside angles of more than 270° are joined by an extra linear motion. The following diagrams show how cutter compensation works for the two values of Setting 58.

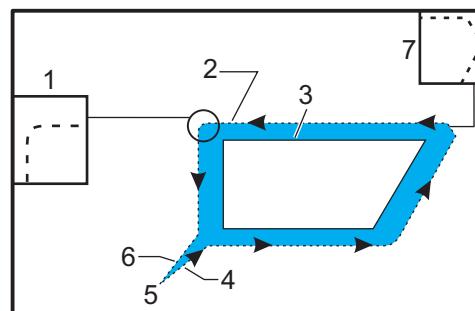
**NOTE:**

When canceled, the programmed path returns to being the same as the center of the cutter path. Cancel cutter comp (G40) before ending a program.

- F5.8:** G42 Cutter Compensation, YASNAC: [1] Radius, [2] Actual center of tool path, [3] Programmed path, [4] G42 [5] Start and end [6] G40.



- F5.9:** G42 Cutter Compensation, FANUC: [1] Radius, [2] Actual center of tool path, [3] Programmed path, [4] G42, [5] Start and end [6] G40, [7] Extra move.

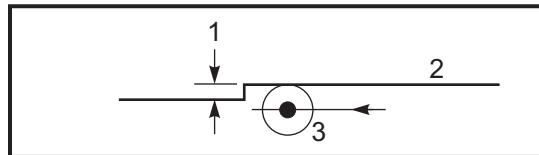


Entry and Exit

Cutting should not be performed when entering and exiting cutter compensation or when changing from left side to right side compensation. When cutter compensation is turned on, the starting position of the move is the same as the programmed position, but the ending position is offset, to either the left or right of the programmed path, by the amount entered in the radius offset column. In the block that turns off cutter compensation, compensation is turned off as the tool reaches the end of block position. 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 is offset to one side of the programmed path and ends 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 the tool position until the next X or Y move is encountered.

When turning on cutter compensation in a move that is followed by a second move at an angle of less than 90°, there are two ways of computing the first motion, type A or type B (Setting 43). The first, type A, moves the tool directly to the offset start point for the second cut. The diagrams on the following pages illustrate the differences between type A and type B for both **FANUC** and **YASNAC** settings (Setting 58).

- F5.10:** Improper Cutter Compensation. Move is Less Than Cutter Comp Radius [1].
Work Piece [2], Tool [3]



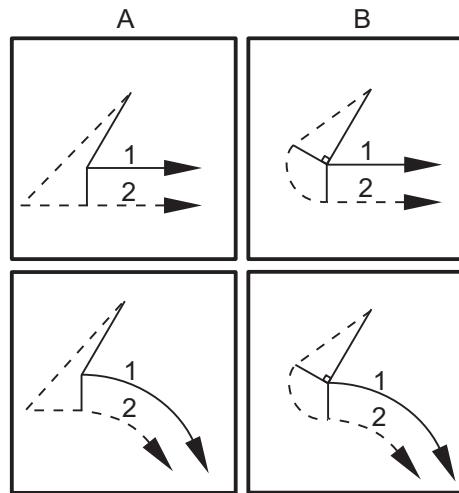
NOTE:

A small cut of less than tool radius and at a right angle to the previous motion only works with the **FANUC** setting. A cutter compensation alarm is generated if the machine is set to the **YASNAC** setting.

Feed Adjustments in Cutter Compensation

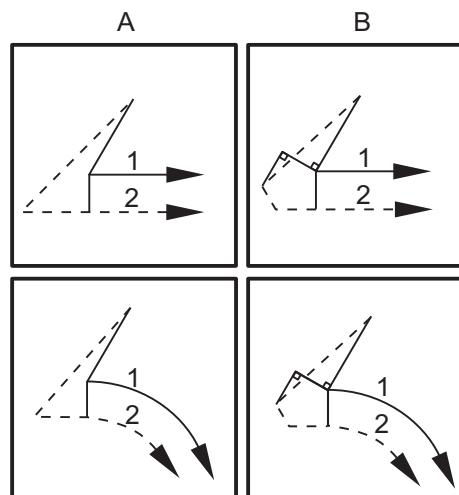
When using cutter compensation in circular moves, there is the possibility of speed adjustments to what has been programmed. If the intended finish cut is on the inside of a circular motion, the tool should be slowed down to ensure that the surface feed does not exceed what was intended.

F5.11: Cutter Compensation Entry, YASNAC: [A] Type A, [B] Type B, [1] Program path, [2] Tool center path.



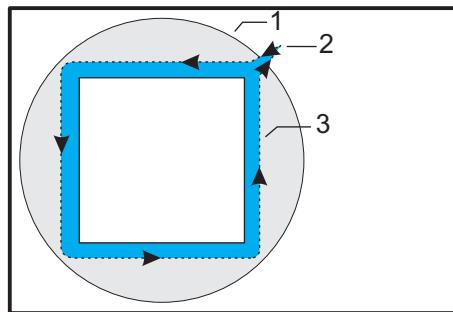
5

F5.12: Cutter Compensation Entry, FANUC: [A] Type A, [B] Type B, [1] Program path, [2] Tool center path.



Cutter Compensation Example

F5.13: Cutter Compensation 4-Flute Endmill: [1] 2" (50 mm) bar stock, [2] Starting point, [3] Program path and center of tool path.



```
T0101 (Tool .500" 4-Flute Endmill) ;
G54 ;
G17 ;
G112 ;
M154 ;
GO G98 Z.3 ;
GO X1.4571 Y1.4571 ;
M8 ;
G97 P3000 M133 ;
Z.15 ;
G01Z-.25F2 ;
G01 G42 X1.1036 Y1.1036 F10. ;
G01 X.75 Y.75 ;
G01 X-.5 ;
G03 X-.75 Y.5 R.25 ;
G01 Y-.5 ;
G03 X-.5 Y-.75 R.25 ;
G01 X.5 ;
G03 X.75 Y-.5 R.25 ;
G01 Y.75 ;
G01 X1.1036 Y1.1036 ;
GO G40 X1.4571 Y1.4571 ;
GO ZO. ;
G113 ;
G18 ;
M9 ;
M155 ;
M135 ;
GO G53 XO. ;
GO G53 ZO. ;
```

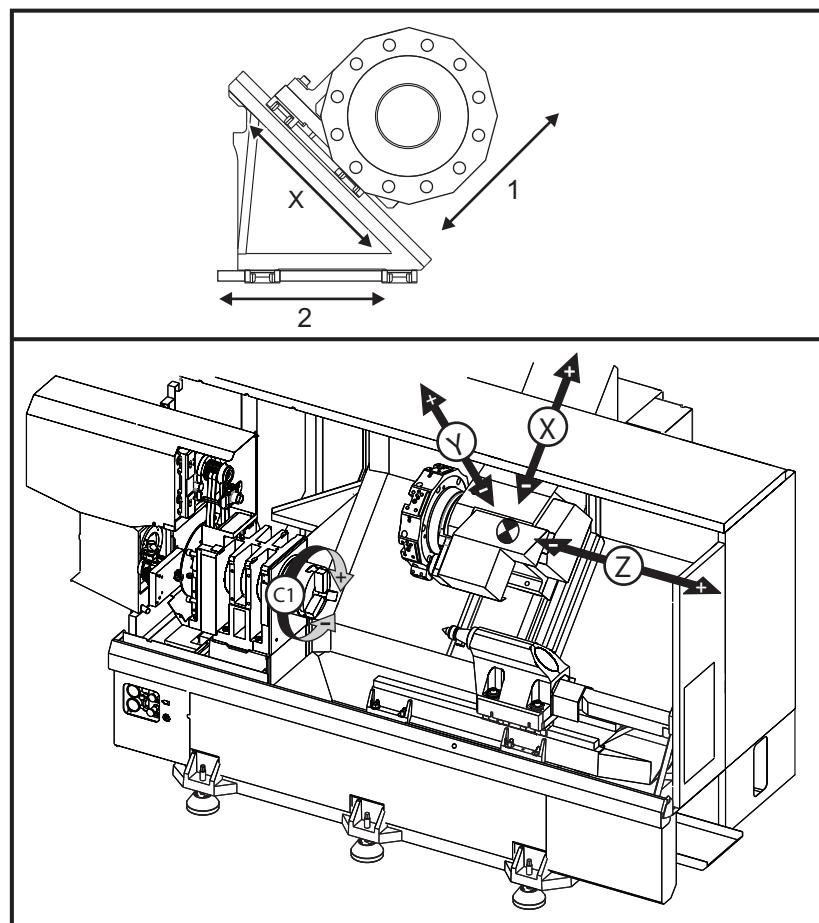
M30 ;

%

5.4 Y Axis

The Y Axis moves tools perpendicular to the spindle center line. This motion is achieved by compound motion of the X- and Y-Axis ballscrews. Refer to G17 and G18, starting on page 292, for programming information.

F5.14: Y-Axis Motion: [1] Y-Axis compound motion, [2] Horizontal plane.



5.4.1 Y-Axis Travel Envelopes

You can find detailed work and travel envelope information for your machine at www.HaasCNC.com. Select your machine model, and then choose the Dimensions option from the pull-down menu. The size and position of the available work envelope changes with the length of radial live tools.

When you set up tooling for the Y Axis, consider these factors:

- Work piece diameter
- Tool extension (radial tools)
- Required Y-Axis travel from the centerline

5.4.2 Y-Axis Lathe with VDI Turret

The position of the work envelope shifts when using radial live tools. The length the cutting tool extends from the centerline of the tool pocket is the distance the envelope shifts. You can find detailed work envelope information from your machine model dimensions page at www.HaasCNC.com.

5.4.3 Operation and Programming

The Y Axis is an additional axis on the lathes (if so equipped) that can be commanded and behaves in the same manner as the standard X and Z Axes. There is no activation command necessary for Y Axis.

The lathe automatically returns the Y Axis to spindle centerline after a tool change. Make sure the turret is correctly positioned before commanding rotation.

Standard Haas G and M codes are available when programming with the Y Axis.

Mill type cutter compensation can be applied in both G17 and G19 planes when performing live tool operations. Cutter compensation rules must be followed to avoid unpredictable motion when applying and canceling the compensation. The Radius value of the Tool being used must be entered in the **RADIUS** column of the tool geometry page for that tool. The tool tip is assumed as "0" and no value should be entered.

Programming recommendations:

- Command Axes home or to a safe tool change location in rapids using G53 which moves all axes at the same rate simultaneously. Regardless of the positions of Y Axis and X Axis in relation to each other both move at the MAX possible speed towards commanded position and usually do not finish at the same time. For example:

G53 X0 (command for home) ;

G53 X-2.0 (command for X to be 2" from home) ;

G53 X0 Y0 (command for home) ;

Refer to G53 on page 300.

If commanding the Y and X Axes home using G28 the following conditions must be met and the described behavior expected:

- Address identification for G28:

X = U

Y = Y

Z = W

B = B

C = H

Example:

G28 U0 (U Zero) ; sends the X Axis to home position.

G28 U0 ; is okay with Y Axis below spindle centerline.

G28 U0 ; produces a 560 alarm if Y Axis is above spindle centerline. However homing the Y Axis first or utilizing a G28 without a letter address does not generate the 560 alarm.

G28 ; sequence sends X, Y, and B home first then C and Z

G28 U0 Y0 ; produces no alarm regardless of the Y-Axis position.

G28 Y0 ; is okay with Y Axis above spindle centerline.

G28 Y0 ; is okay with Y Axis below spindle centerline

Pressing [POWER UP/RESTART] or [HOME G28] produces the message:

Function locked.

- If X Axis is commanded home while the Y Axis is above spindle centerline (positive Y-Axis coordinates), alarm 560 is generated. Command Y Axis home first, then X Axis.
- If X Axis is commanded home and the Y Axis is below spindle centerline (negative Y-Axis coordinates), the X Axis goes home and Y does not move.
- If both X Axis and Y Axis are commanded home using G28 U0 Y0, the X Axis and Y Axis go home at the same time regardless of Y being above or below the centerline.
- Clamp the main and/or secondary spindles (if so equipped) anytime live tooling operations are being performed and C Axis is not being interpolated.



NOTE:

The brake unclamps automatically any time C-Axis motion for positioning is commanded.

- These canned cycles can be used with the Y Axis. Refer to page 279 for more information.

Axial Only Cycles:

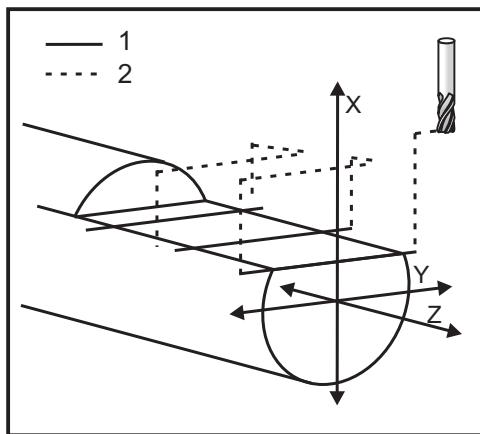
- Drilling: G74, G81, G82, G83,
- Boring: G85, G89,
- Tapping: G95, G186,

Radial Only Cycles:

- Drilling: G75 (a grooving cycle), G241, G242, G243,
- Boring: G245, G246, G247, G248
- Tapping: G195, G196

Program Example of Y-Axis Milling:

F5.15: Y-axis Milling Program Example: [1] Feed, [2] Rapid.



```
%  
o50004 (Y AXIS MILLING) ;  
(G54 X0 Y0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is an end mill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G19 (Call YZ plane) ;  
G98 (Feed per min) ;  
M154 (Engage C Axis) ;  
G00 G54 X4. C90. Y0. Z0.1 ;  
(Rapid to clear position) ;  
M14 (Spindle brake on) ;  
P1500 M133 (Live tool CW at 1500 RPM) ;  
M08 (Coolant on) ;
```

```
(BEGIN CUTTING BLOCKS) ;
G00 X3.25 Y-1.75 Z0. (Rapid move) ;
G00 X2.25 (Rapid approach) ;
G01 Y1.75 F22. (Linear feed) ;
G00 X3.25 (Rapid retract) ;
G00 Y-1.75 Z-0.375 (Rapid move) ;
G00 X2.25 (Rapid approach) ;
G01 Y1.75 F22. (Linear feed) ;
G00 X3.25 (Rapid retract) ;
G00 Y-1.75 Z-0.75 (Rapid move) ;
G00 X2.25 (Rapid approach) ;
G01 Y1.75 F22. (Linear feed) ;
(BEGIN COMPLETION BLOCKS) ;
G00 X3.25 M09 (Rapid retract, Coolant off) ;
M15 (Spindle brake off) ;
M155 (Disengage C Axis) ;
M135 (Live tool off) ;
G18 (Return to XZ plane) ;
G53 X0 Y0 (X & Y Home) ;
G53 Z0 (Z Home) ;
M30 (End program) ;
%
```

5.5 Parts Catcher

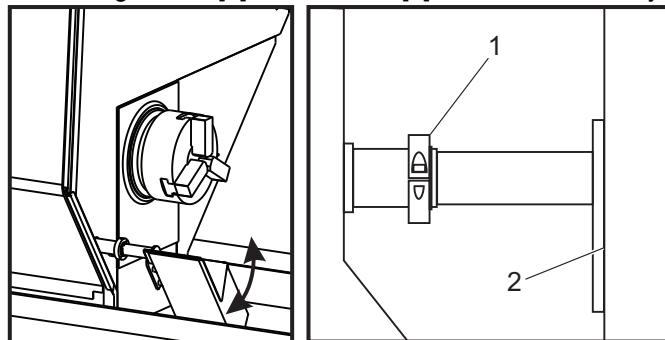
This option is an automatic part retrieval system designed to work with bar feed applications. It is commanded using M Codes (M36 to activate and M37 to deactivate). The parts catcher rotates to catch finished parts and direct them into the bin mounted on the front door.

5.5.1 Operation

The parts catcher must be properly aligned before operation.

1. Power on the machine. In MDI mode, activate the parts catcher (M36).
2. Loosen the screw in the shaft collar on the outer parts catcher shaft.

F5.16: Parts Catcher Alignment: [1] Shaft collar, [2] Parts Catcher Tray.



3. Slide the parts catcher tray into the shaft far enough to catch the part and clear the chuck. Rotate the tray to open the sliding cover of the parts collector mounted in the door and tighten the shaft collar on the part catcher shaft.



WARNING:

Check the Z-Axis, X-Axis, tool and turret positions during part catcher actuation to avoid possible collisions during operation.



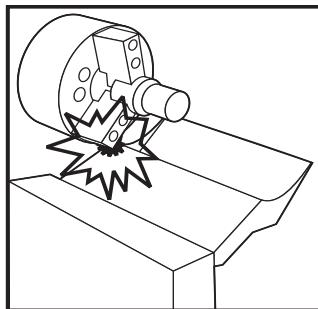
NOTE:

The operator door must be closed when actuating the part catcher.

5.5.2 Chuck Interference

Large chuck jaws may interfere with the operation of the parts catcher. Check the clearances before operating the parts catcher.

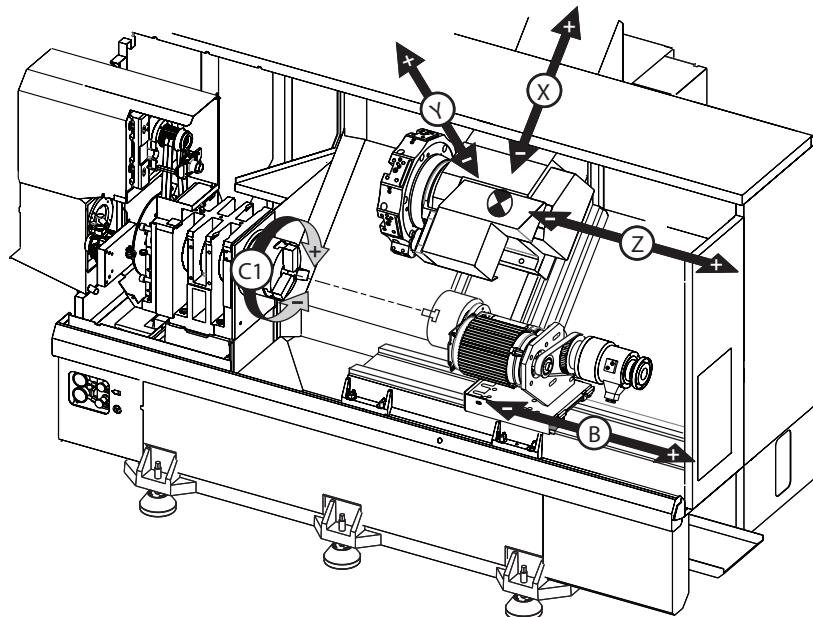
F5.17: Chuck Jaw Interferes With Parts Catcher



5.6 Dual-Spindle Lathes (DS-Series)

The DS-30 is a lathe with two spindles. The main spindle is in a stationary housing. The other spindle, the “secondary spindle”, has a housing that moves along a linear axis, designated “B”, and replaces the typical tailstock. You use a special set of M-codes to command the secondary spindle.

F5.18: Dual Spindle Lathe with an Optional Y Axis



5.6.1 Synchronized Spindle Control

Dual-spindle lathes can synchronize the main and secondary spindle. This means that when the main spindle receives a command to turn, the secondary spindle turns at the same speed, in the same direction. This is called Synchronous Spindle Control (SSC) mode. In SSC mode, both spindles accelerate, maintain speed, and decelerate together. You can then use both spindles to support a workpiece at both ends for maximum support and minimum vibration. You can also transfer the workpiece between the main and secondary spindle, effectively doing a “part flip” while the spindles continue to turn.

There are two G-codes associated with SSC:

G199 activates SSC.

G198 cancels SSC.

When you command G199, both spindles orient before they accelerate to the programmed speed.

**NOTE:**

When you program synchronized dual spindles, you should first bring both spindles up to speed with M03 (for the main spindle) and M144 (for the secondary spindle) before you command G199. If you command G199 before commanding spindle speed, the two spindles attempt to stay synchronized while they accelerate, causing acceleration to take much longer than normal.

If SSC mode is in effect, and you press [RESET] or [EMERGENCY STOP], SSC mode remains in effect until the spindles stop.

The Synchronized Spindle Control Display

F5.19: The Synchronized Spindle Control Display

SPINDLE SYNCHRONIZATION CONTROL			
	SPINDLE	SECONDARY SPINDLE	DIFFERENCE
G15/G14	G15		
SYNC (G199)			
POSITION (DEG)	0.0000	0.0000	0.0000
VELOCITY (RPM)	0	0	0
G199 R PHASE OFS		0.0000	
CHUCK			
LOAD %	0	0	
G-CODE INDICATES LEADING SPINDLE			

The spindle synchronization control display is available in the CURRENT COMMANDS display.

The SPINDLE column gives the main spindle status. The SECONDARY SPINDLE column gives the secondary spindle status. The third column shows miscellaneous status. On the left is a column of row titles. The following describes each row.

G15/G14 - If G15 appears in the SECONDARY SPINDLE column, the main spindle is the leading spindle. If G14 appears in the SECONDARY SPINDLE column, the secondary spindle is the leading spindle.

SYNC (G199) - When G199 appears in the row, spindle synchronization is active.

POSITION (DEG) - This row shows the current position, in degrees, of both the spindle and the secondary spindle. Values range from -180.0 degrees to 180.0 degrees. This is relative to the default orientation position of each spindle.

The third column indicates the current difference, in degrees, between the two spindles. When both spindles are at their respective zero marks, then this value is zero.

If the third column value is negative, it represents how much the secondary spindle currently lags the main spindle, in degrees.

If the third column value is positive, it represents how much the secondary spindle currently leads the main spindle, in degrees.

VELOCITY (RPM) - This row shows the actual RPM of the main spindle and the secondary spindle.

G199 R PHASE OFS. - This is the programmed **R** value for G199. This row is blank when G199 is not commanded; otherwise it contains the **R** value in the most recently executed G199 block. Refer to page **356** for more information on G199.

CHUCK - This column shows the clamped or unclamped status of the work holding (chuck or collet). This row is empty when clamped, or shows “UNCLAMPED” in red when the work holding is open.

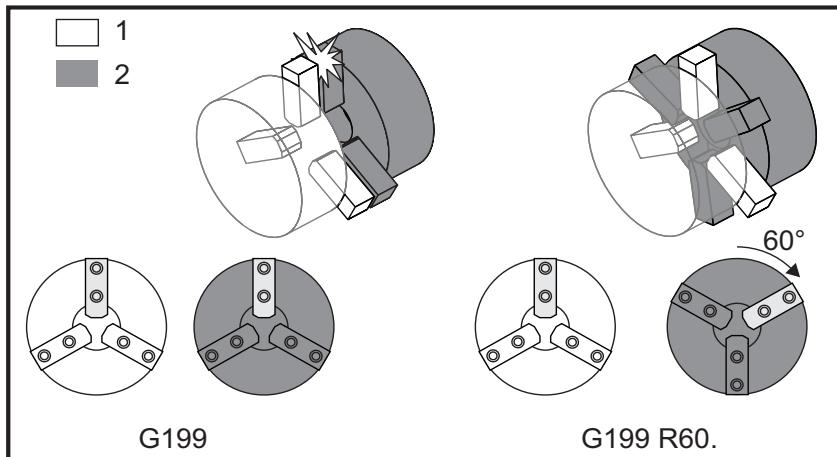
LOAD % - This shows the current load percent for each spindle.

R Phase Offset Explained

When dual lathe spindles are synchronized, they orient, then rotate at the same speed with their home positions stationary relative to each other. In other words, the relative orientation you see when both spindles are stopped at their home positions is preserved as synchronized spindles rotate.

You can use an R value with **G199**, **M19**, or **M119** to alter this relative orientation. The R value specifies an offset, in degrees, from the following spindle's home position. You can use this value to allow the chuck jaws to mesh during a workpiece hand-off operation. Refer to Figure **F5.20** for an example.

F5.20: G199 R Value Example: [1] Leading Spindle, [2] Following Spindle



Finding a G199 R Value

To find an appropriate G199 R value:

1. In **MDI** mode, command an M19 to orient the main spindle and an M119 to orient the secondary spindle.
This establishes the default orientation between the spindles' home positions.
2. Add an R value in degrees to the M119 to offset the secondary spindle's position.
3. Check the interaction between the chuck jaws. Change the M119 R value to adjust the secondary spindle position until the chuck jaws interact correctly.
4. Record the correct R value and use it in the G199 blocks in your program.

5.6.2 Secondary Spindle Programming

The program structure for the secondary spindle is the same as that for the main spindle. Use G14 to apply main spindle M-codes and canned cycles to the secondary spindle. Cancel G14 with G15. Refer to page 291 for more information on these G-codes.

Secondary Spindle Commands

Three M-Codes are used to start and stop the secondary spindle:

- M143 starts the spindle forward.
- M144 starts the spindle in reverse.

- M145 stops the spindle.

The P address code specifies the spindle speed, from 1 RPM to maximum speed.

Setting 122

Setting 122 selects between OD and ID clamping for the secondary spindle. Refer to page 414 for more information.

G14/G15 - Spindle Swap

These G-codes select which spindle leads during Synchronized Spindle Control (SSC) mode (G199).

G14 makes the secondary spindle the leading spindle, and G15 cancels G14.

The SPINDLE SYNCHRONIZATION CONTROL screen under current commands tells you which spindle currently leads. If the secondary spindle leads, G14 displays in the SECONDARY SPINDLE column. If the main spindle leads, G15 displays in the SPINDLE column.

5

5.7 Automatic Tool Setting Probe

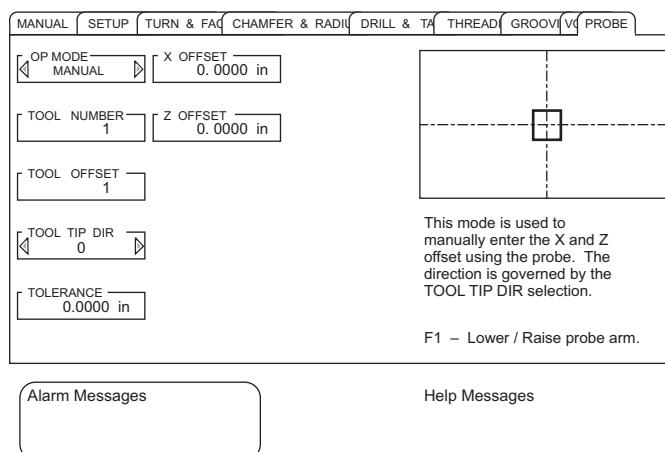
The tool setting system is used to set tool offsets by touching off tooling on a probe. The probe is first set up for tooling in manual mode, where initial tool measurements are made. After this setup, automatic mode is available using the Automatic Tool Setting Probe (ATP) to reset offsets when inserts are changed. Tool break detection is also available to monitor tool wear and breakage. The software generates G-code which can be inserted into lathe programs to enable probe use during automatic operation.

5.7.1 Operation

To access the tool probe menu:

1. Press [MDI/DNC] and then [PROGRAM].
Access the IPS tabbed menu.
2. Use the right cursor key to navigate to the PROBE tab and press [ENTER].
3. Use the up/down cursor arrow keys to move between menu options.

F5.21: Initial Probe Menu



Menu Item Explanation

OP MODE Use the left and right cursor arrow keys to choose between **MANUAL**, **AUTOMATIC**, and **BREAK DET.** modes.

TOOL NUMBER The tool number to be used. This value automatically sets to the current tool position in **MANUAL** mode. It can be changed in **AUTOMATIC** and **BREAK DET.** modes.

TOOL OFFSET Enter the tool offset number that is being measured.

TOOL TIP DIR Use the **[LEFT]** and **[RIGHT]** cursor arrows to choose the tool nose vector V1-V8. Refer to page 131 for more information.

TOLERANCE Sets the measurement difference tolerance for **BREAK DETECT** mode. Not available in other modes.

X OFFSET, Z OFFSET Displays the offset value for the specified axis. Read only.

5.7.2 Manual Mode

Tools must be touched off in manual mode before automatic mode can be used.

1. Enter the probe menu by pressing **[MDI/DNC]**, then **[PROGRAM]**, and selecting the **PROBE** tab. Press **[F1]** to lower the probe arm.
2. Select the tool to be touched off using **[TURRET FWD]** or **[TURRET REV]**.
3. Select Op Mode **MANUAL** using the left / right cursor arrow keys, then press **[ENTER]** or the down cursor arrow key.
4. The tool offset option is set according to the currently selected tool position. Press **[ENTER]** or the down cursor arrow key.

5. Type the tool offset number to be used, then press **[ENTER]**. The offset number is entered and the next menu option, **Tool Tip Dir**, is selected.
6. Use the **[LEFT]** and **[RIGHT]** cursor arrows to select a tool tip direction, then press **[ENTER]** or the **[DOWN]** cursor arrow. Refer to page 131 for more information on tool tip direction.
7. Use the **[HANDLE JOG]** control to move the tool tip to within approximately 0.25" (6 mm) of the tool probe in the direction indicated by the on-screen tool tip direction diagram.

**NOTE:**

If the tool tip is too far from the probe, the tool will not reach the probe and the operation alarms out.

8. Press **[CYCLE START]**. The tool tip is touched off and offsets are recorded and displayed. A G-Code program for the operation is generated in **MDI** and is used for tool movement.
9. Repeat steps 1-8 for each tool to be touched off. Be sure to jog the tool turret away from the probe before selecting the next tool position.
10. Press **[F1]** to raise the tool arm.

5.7.3 Automatic Mode

Once the initial tool measurement has been made in manual mode for a particular tool, automatic mode can be used to update that tool's offsets in the event of tool wear or a replacement insert.

1. Enter the probe menu by pressing **[MDI/DNC]** and then **[PROGRAM]**, and selecting the **PROBE** tab. Select Op Mode **Automatic** using the left/right cursor arrow keys, then press **[ENTER]** or the down cursor arrow key.
2. Type the tool number to be measured, then press **[ENTER]**.
3. Type the tool offset number to be used, then press **[ENTER]**.
4. The tool tip direction is preselected based on the direction set in manual mode for the tool offset.
5. Press **[CYCLE START]**. The tool tip is touched off and offsets are updated and displayed. A G-Code program for the operation is generated in **MDI** and is used for tool movement.
6. Repeat Steps 1 through 5 for each tool to be touched off.

5.7.4 Break Detect Mode

Break detect mode compares the current measurement of the tool against the recorded measurement, and applies a user-defined tolerance value. If the difference in measurements is greater than the defined tolerance, an alarm is generated and the operation stops.

1. Enter the probe menu by pressing **[MDI/DNC]** and then **[PROGRAM]**.
2. Select the **PROBE** tab and press **[ENTER]**.
3. Select **Op Mode Break Det.** using the left/right cursor arrow keys.
4. Type the tool number to be measured, then press **[ENTER]**.
5. Type the tool offset number to be used, then press **[ENTER]**.

The tool tip direction is automatically selected based on the direction set in manual mode for the tool offset.

6. Press the down cursor arrow key.
7. Type the desired tolerance value and press **[ENTER]**.
8. If you want to run this single tool test in MDI, go to step 12. If you want to copy the test to your program, continue to the next step.
9. To copy the resulting code, press **[F4]** from the **PROBE** tab screen to bring up the **IPS Recorder** popup.
10. Copy the generated code with the new tolerances to the selected destination for the program (a new program, or the current program in memory).
11. To check the code, press **[MEMORY]** and cursor down to the inserted code.
12. Press **[CYCLE START]**. The tool tip is touched off. If the tolerance value is exceeded, an alarm is generated.
13. Repeat steps 1 through 12 for each tool to be checked.

5.7.5 Tool Tip Direction

Refer to the illustration in the Imaginary Tool Tip and Direction (Tool Nose Compensation) section on page **131**.



NOTE:

Note that the automatic tool setting probe only uses codes 1-8.

5.7.6 Automatic Tool Probe Calibration

This ATP calibration procedure requires the following:

- An OD turning tool,
 - A workpiece that fits in the chuck jaws,
 - A 0-1.0" micrometer to measure the tool probe stylus,
 - A micrometer to inspect the workpiece diameter.
1. First, make sure that the Automatic Tool Probe (ATP) arm operates correctly by running the calibration check on page 269. If it does not operate correctly, contact Haas Service for help.
 2. If the probe arm operates as described, continue with the calibration procedure on page 269.

ATP Calibration - Operation Check

Check that the ATP arm operates correctly.

If the probe arm operates as described, continue with the calibration procedure. If it does not operate correctly, contact Haas Service for help.

1. Press **[MDI/DNC]**.
2. Type M104; M105; and press **[INSERT]**.
3. Press **[SINGLE BLOCK]**.
4. Press **[CYCLE START]**. The probe arm should move into ready position (down).
5. Press **[CYCLE START]**. The probe arm should move into stored position.

5

ATP Calibration Procedure

If the probe arm operates correctly, continue with the following procedure:

1. Install the OD turning tool in the tool 1 station of the tool turret.
2. Clamp the workpiece in the chuck.
3. Press **[OFFSET]** and clear the offset values for tool 1 on the **Tool Geometry** page.
4. Use the turning tool in station 1 to take a small cut on the diameter of the material clamped in the spindle.
5. In the Z-axis only, jog the tool away from the part—do not jog the X-axis away from the diameter.
6. Stop the spindle.

7. Use a micrometer to measure the diameter of the cut made on the workpiece.
8. Press [**X DIAMETER MEASURE**] to record the X-axis position in the offset table.
9. Type the workpiece diameter and press [**ENTER**] to add it to the X-axis offset. Record this value as a positive number. Call this Offset A.
10. Change Settings 59 through 63 to 0 (zero).
11. Jog the tool away to a safe position out of the ATP arm path.
12. Lower the ATP arm (**M104** in **MDI**).
13. Jog the Z-axis to approximately center the tool tip with the probe stylus.
14. Jog the X-axis to bring the tool tip to approximately 0.25" (6 mm) above the probe stylus.
15. Select the .001" jog increment by pressing [**.001 1.**] and hold down [**-X**] until the probe sounds off and stops the tool. Record the X-axis offset position as a positive number. Call this Offset B.
16. Subtract Offset B from Offset A. Enter the result as a positive value into Setting 59.
17. Measure the width of the probe stylus with the micrometer. Enter this value as a positive number for Settings 62 and 63. Once the tool probe is properly aligned, the values from [**X DIAMETER MEASURE**] and the value from the probe will be the same.
18. Multiply the probe stylus width by two. Subtract that value from Setting 59, and enter this new value as a positive number into Setting 60.

5.7.7 Tool Probe Alarms

The following alarms are generated by the tool probe system and are displayed in the alarm messages section of the display. They can only be cleared by resetting the control.

Probe Arm Not Down – The probe arm is not in position for the operation. Enter the probe menu by pressing [**MDI/DNC**] and then [**PROGRAM**], and selecting the **PROBE** tab. Press [**F1**] to lower the probe arm.

Probe Not Calibrated – The probe must be calibrated using the procedure described previously.

No Tool Offset – A tool offset must be defined.

Illegal Tool Offset Number – Tool offset “T0” is not allowed. If using the ‘T’ input on the cycle call line check that the value is not zero; otherwise, this alarm may occur if no tool or tool offset was selected in MDI before running the cycle.



CAUTION: *Make sure the turret is safely away from the probe before indexing the turret.*

Illegal Tool Nose Vector – Only vector numbers 1 through 8 are allowed. See the Tool Tip Direction diagram in the TNC section of this manual for tool nose vector definitions.

Tool Probe Open – This alarm occurs when the probe is in an unexpected open (triggered) condition. Make sure the tool is not in contact with the probe before beginning an operation.

Tool Probe Failure – This alarm occurs when the tool fails to contact the probe within the defined travel. Check that the probe has been calibrated. In manual probe mode, jog the tool tip to within 0.25" (6 mm) of the probe.

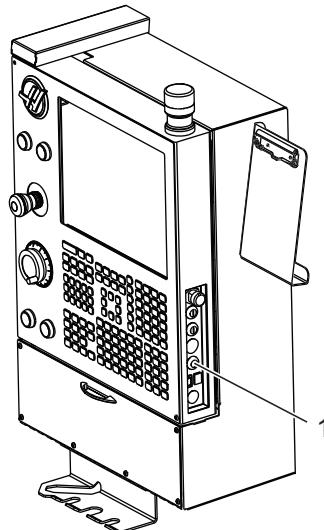
Broken Tool – This alarm is generated when the tool length error exceeds the defined tolerance.

5.8 Servo Auto Door

This option adds a rack-and-pinion assembly to the machine doors, which allows them to open automatically. There are (3) ways to activate the Servo Auto Door:

1. Press the **[SERVO AUTO DOOR OVERRIDE]** button on the side of the control pendant to toggle the door open or closed.

F5.22: **[SERVO AUTO DOOR OVERRIDE]** Button [1]



2. Use M-codes within a program that associate with the Servo Auto Door. For example, when Setting 131 is set to **ON**, the Servo Auto Door is paired with:
 - M00 (Stop program)
 - M01 (Optional Program Stop)
 - M30 (Program End and Reset)

When one of these M-codes is read, the door opens. Pressing [**CYCLE START**] automatically closes the door.

**NOTE:**

*Alarms do not result from a door fault if the door is operated with the [**SERVO AUTO DOOR OVERRIDE**] button. Alarms are generated for a door failure when operation is the result of the programming M-codes or the [**CYCLE START**] button operation.*

3. Command the Servo Auto Door in a program by inserting an M85 to open the door and an M86 to close it.
The Servo Auto Door M-codes, (M85/M86) are for robot tending. Additional set-up is necessary to safely use the Auto door. Contact your Haas Factory Outlet before using this style of Servo Auto Door programming.

5.9

More Information Online

For updated and supplemental information, including tips, tricks, maintenance procedures, and more, go to www.HaasCNC.com and select the **Resource Center**.

Chapter 6: G-codes, M-codes, Settings

6.1 G-codes

Code	Description	Group
G00	Rapid Motion Positioning	01
G01	Linear Interpolation Motion	01
G02	CW Circular Interpolation Motion	01
G03	CCW Circular Interpolation Motion	01
G04	Dwell	00
G09	Exact Stop	00
G10	Set Offsets	00
G14	Secondary Spindle Swap	17
G15	Secondary Spindle Cancel	17
G17	XY Plane	00
G18	Plane Selection	02
G19	YZ Plane	02
G20	Select Inches	06
G21	Select Metric	06
G28	Return To Machine Zero Point	00
G29	Return From Reference Point	00
G31	Skip Function	00
G32	Thread Cutting	01
G40	Tool Nose Compensation Cancel	07

Code	Description	Group
G41	Tool Nose Compensation (TNC) Left	07
G42	Tool Nose Compensation (TNC) Right	07
G50	Set Global coordinate Offset FANUC, YASNAC	00
G51	Cancel Offset (YASNAC)	00
G52	Set Local Coordinate System FANUC	00
G53	Machine Coordinate Selection	00
G54	Coordinate System #1 FANUC	12
G55	Coordinate System #2 FANUC	12
G56	Coordinate System #3 FANUC	12
G57	Coordinate System #4 FANUC	12
G58	Coordinate System #5 FANUC	12
G59	Coordinate System #6 FANUC	12
G61	Exact Stop Modal	15
G64	Exact Stop Cancel G61	15
G65	Macro Subroutine Call Option	00
G70	Finishing Cycle	00
G71	O.D./I.D. Stock Removal Cycle	00
G72	End Face Stock Removal Cycle	00
G73	Irregular Path Stock Removal Cycle	00
G74	End Face Grooving Cycle	00
G75	O.D./I.D. Grooving Cycle	00
G76	Threading Cycle, Multiple Pass	00
G80	Canned Cycle Cancel	09

Code	Description	Group
G81	Drill Canned Cycle	09
G82	Spot Drill Canned Cycle	09
G83	Normal Peck Drilling Canned Cycle	09
G84	Tapping Canned Cycle	09
G85	Boring Canned Cycle	09
G86	Bore and Stop Canned Cycle	09
G87	Bore and Manual Retract Canned Cycle	09
G88	Bore and Dwell and Manual Retract Canned Cycle	09
G89	Bore and Dwell Canned Cycle	09
G90	O.D./I.D. Turning Cycle	01
G92	Threading Cycle	01
G94	End Facing Cycle	01
G95	Live Tooling Rigid Tap (Face)	09
G96	Constant Surface Speed On	13
G97	Constant Surface Speed Off	13
G98	Feed Per Minute	10
G99	Feed Per Revolution	10
G100	Disable Mirror Image	00
G101	Enable Mirror Image	00
G102	Programmable Output to RS-232	00
G103	Limit Block Lookahead	00
G105	Servo Bar Command	09
G110	Coordinate System #7	12

Code	Description	Group
G111	Coordinate System #8	12
G112	XY to XC interpretation	04
G113	Cancel G112	04
G114	Coordinate System #9	12
G115	Coordinate System #10	12
G116	Coordinate System #11	12
G117	Coordinate System #12	12
G118	Coordinate System #13	12
G119	Coordinate System #14	12
G120	Coordinate System #15	12
G121	Coordinate System #16	12
G122	Coordinate System #17	12
G123	Coordinate System #18	12
G124	Coordinate System #19	12
G125	Coordinate System #20	12
G126	Coordinate System #21	12
G127	Coordinate System #22	12
G128	Coordinate System #23	12
G129	Coordinate System #24	12
G154	Select Work Coordinates P1-99	12
G159	Background Pickup / Part Return	
G160	APL Axis Command Mode Only	
G161	APL Axis Command Mode Off	

Code	Description	Group
G184	Reverse Tapping Canned Cycle For Left Hand Threads	09
G186	Reverse Live Tool Rigid Tap (For Left Hand Threads)	10
G187	Accuracy Control	00
G195	Forward Live Tool Radial Tapping (Diameter)	00
G196	Reverse Live Tool Radial Tapping (Diameter)	00
G198	Disengage Synchronous Spindle Control	00
G199	Engage Synchronous Spindle Control	00
G200	Index on the Fly	00
G211	Manual Tool Setting	
G212	Auto Tool Setting	
G241	Radial Drill Canned Cycle	09
G242	Radial Spot Drill Canned Cycle	09
G243	Radial Normal Peck Drilling Canned Cycle	09
G245	Radial Boring Canned Cycle	09
G246	Radial Bore and Stop Canned Cycle	09
G247	Radial Bore and Manual Retract Canned Cycle	09
G248	Radial Bore and Dwell and Manual Retract Canned Cycle	09
G249	Radial Bore and Dwell Canned Cycle	09

Introduction to G-codes

G-codes are used to command specific actions for the machine: such as simple machine moves or drilling functions. They also command more complex features which can involve optional live tooling and the C-Axis.

G-codes are divided into groups. Each group of codes is commands for a specific subject. For example, Group 1 G-codes command point-to point moves of the machine axes, Group 7 are specific to the Cutter Compensation feature.

Each group has a dominant G-code; referred to as the default G-code. A default G-code means they are the one in each group the machine uses unless another G-code from the group is specified. For example programming an X, Z move like this, `X-2. Z-4.` will position the machine using `G00`.



NOTE:

Proper programming technique is to preface all moves with a G-code.

Default G-codes for each group are shown on the **Current Commands** screen under **All Active Codes**. If another G-code from the group is commanded (active), that G-code is displayed on the **All Active Codes** screen.

G-code commands can be modal or non-modal. A modal G-code means that once commanded, the G-code will stay in effect until the end of the program or until another G-code from the same group is commanded. A non modal G-code only affects the line it is in; the program line after will not be affected by the previous line's non modal G-code. Group 00 codes are non-modal; the other groups are modal.

Most of the CNC programs require you to know the G-codes to build a program to complete a part. For a description on how to use the G-codes, refer to the Programming chapter.



NOTE:

The Haas Intuitive Programming System (IPS) is a programming mode that either hides G-codes or completely bypasses the use of G-codes.

Canned Cycles

A canned cycle is used to simplify the programming of a part. Canned cycles are defined for most common Z-axis repetitive operations, such as drilling, tapping, and boring. Once selected, a canned cycle is active until canceled with G80. When active, the canned cycle is executed every time an axis motion is programmed. Axis motions are executed as rapid commands (G00) and the canned cycle operation is performed after the axis motion. Applies to G17, G19 cycles, and Y-axis movements on Y-axis lathes.

Using Canned Cycles

Modal canned cycles remain in effect after they are defined and are executed in the Z-axis, for each position of the X, Y, or C-Axis.

**NOTE:**

X, Y, or C-Axis positioning moves, during a canned cycle will be rapid moves.

The operation of a canned cycle will vary according to whether incremental (U,W) or absolute (X, Y, or C) axis moves are used.

If a loop count (Lnn code number) is defined within the block, the canned cycle will repeat that many times with an incremental (U or W) move between each cycle. Enter the number of repeats (L) each time a repeated operation is needed; the number of repeats (L) is not remembered for the next canned cycle.

Spindle control M codes should not be used while a canned cycle is active.

Canned Cycles with Live Tooling

The canned cycles G81, G82, G83, G85, G86, G87, G88, G89, G95, and G186 can be used with axial live tooling, and G241, G242, G243, G245 and G249 can be used with radial live tooling. Some programs must be checked to be sure they turn on the main spindle before running the canned cycles.

**NOTE:**

G84 and G184 are not usable with live tooling.

Programming Notes

Group 01 G-codes cancel Group 09 (canned cycles) codes; for example, if a canned cycle (G73 through G89) is active, the use of G00 or G01 cancels the canned cycle.

G00 Rapid Motion Positioning (Group 01)

- ***B** - B-axis motion command
- ***C** - C-Axis motion command
- ***U** - X-axis incremental motion command
- ***W** - Z-axis incremental motion command
- ***X** - X-axis absolute motion command
- ***Y** - Y-axis absolute motion command
- ***Z** - Z-axis absolute motion command

* indicates optional

This G code is used to move the machine's 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 motions until another cutting move is specified.



NOTE:

Generally, rapid motion will not be in a straight line. Each axis specified is moved at the same speed, but all axes will not necessarily complete their motions at the same time. The machine will wait until all motions are complete before starting the next command.

G01 Linear Interpolation Motion (Group 01)

F - Feed rate
***B** - B-axis motion command
***C** - C-Axis motion command
***U** - X-axis incremental motion command
***W** - Z-axis incremental motion command
***X** - X-axis absolute motion command
***Y** - Y-axis absolute motion command
***Z** - Z-axis absolute motion command
A - Optional angle of movement (used with only one of X, Z, U, W)
,C - Distance from center of intersection where the chamfer begins
,R - Radius of the fillet or arc

This G code provides for straight line (linear) motion from point to point. Motion can occur in 1 or more axes. You can command a G01 with 3 or more axes All axes will start and finish motion at the same time. The speed of all axes is controlled so that the feed rate specified is achieved along the actual path. The C-Axis may also be commanded and this will provide a helical (spiral) motion. A C-Axis feed rate is dependent on the C-Axis diameter setting (Setting 102) to create a helical motion. The F address (feedrate) command is modal and may be specified in a previous block. Only the axes specified are moved.

Corner Rounding and Chamfering

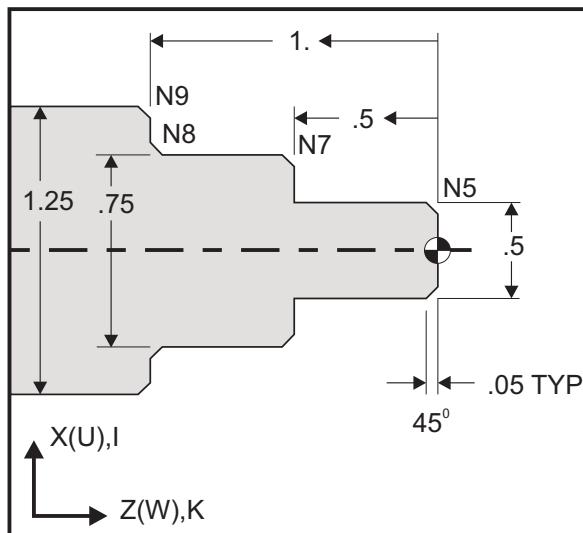
A chamfer block or a corner rounding block can be automatically inserted between two linear interpolation blocks by specifying ,C (chamfering) or ,R (corner rounding).



NOTE:

Both of these variables use a comma symbol (,) before the variable.

There must be a terminating linear interpolation block following the beginning block (a G04 pause may intervene). These two linear interpolation blocks specify a theoretical corner of intersection. If the beginning block specifies a ,C (comma C) the value following the C is the distance from the corner of intersection where the chamfer begins and also the distance from that same corner where the chamfer ends. If the beginning block specifies an ,R (comma R) the value following the R is the radius of a circle tangent to the corner at two points: the beginning of the corner rounding arc block that is inserted and the endpoint of that arc. There can be consecutive blocks with chamfer or corner rounding specified. There must be movement on the two axes specified by the selected plane (the active plane X-Y (G17), X-Z (G18) or Y-Z (G19)). For chamfering a 90° angle only, an I or K value can be substituted where ,C is used.

F6.1: Chamfering

The following G-code syntax automatically includes a 45° chamfer or corner radius between two blocks of linear interpolation which intersect a right (90 degree) angle.

Chamfering Syntax

```
G01 X(U) x Kk ;
G01 Z(W) z Ii ;
```

Corner Rounding Syntax

```
G01 X(U) x Rr ;
G01 Z(W) z Rr ;
```

Addresses:

I = chamfering, Z to X (X axis direction, +/-)

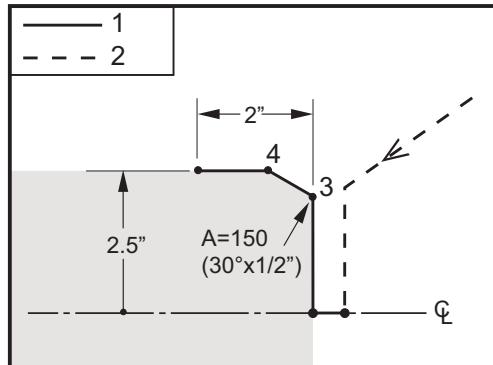
K = chamfering, X to Z (Z axis direction, +/-)

R = corner rounding (X or Z axis direction, +/-, Radius value)

G01 Chamfering with A

When specifying an angle (A), command motion in only one of the other axes (X or Z), the other axis is calculated based on the angle.

F6.2: G01 Chamfering with A: [1] Feed, [2] Rapid, [3] Start Point, [4] Finish Point.

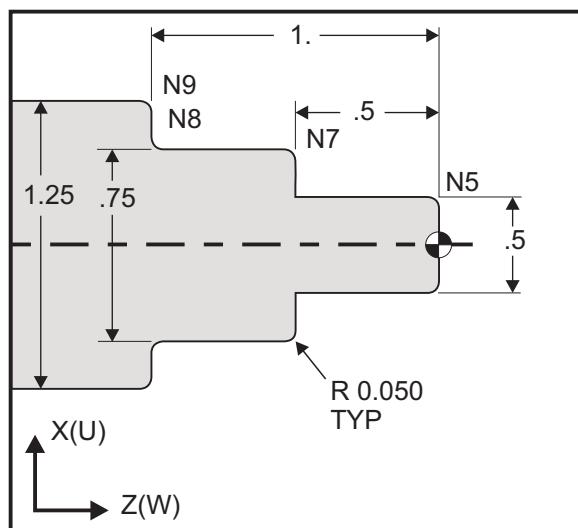


NOTE: $A -30 = A150; A -45 = A135$

Corner Rounding

F6.3: G01 Corner Rounding

6



Notes:

1. Incremental programming is possible if U or W is specified in place of X or Z , respectively. So its actions are as follows:
 $X(\text{current position} + i) = U_i$
 $Z(\text{current position} + k) = W_k$
 $X(\text{current position} + r) = U_r$
 $Z(\text{current position} + r) = W_r$
2. Current position of X or Z Axis is added to the increment.
3. I , K and R always specify a radius value (radius programming value).

F6.4: Chamfering Code Z to X: [1] Chamfering, [2] Code/Example, [3] Movement.

A	B	C	
1. Z+ to X+	X2.5 Z-2; G01 Z-0.5 I0.1; X3.5;	X2.5 Z-2; G01 Z-0.6; X2.7 Z-0.5; X3.5;	X3.5 Z-0.5 1
2. Z+ to X-	X2.5 Z-2.; G01 Z-0.5 I-0.1; X1.5;	X2.5 Z-2.; G01 Z-0.6; X2.3 Z-0.5; X1.5;	X1.5 Z-0.5 2
3. Z- to X+	X1.5 Z-0.5.; G01 Z-2. I0.1; X2.5;	X1.5 Z-0.5 G01 Z-1.9; X1.7 Z-2.; X2.5;	X2.5 Z-2. 3
4. Z- to X-	X1.5 Z-0.5.; G01 Z-2. I-0.1; X0.5;	X1.5 Z-0.5; G01 Z-1.9; X1.3 Z-2.; X0.5;	X1.5 Z-0.5 X0.5 Z-2. 4

F6.5: Chamfering Code X to Z: [1] Chamfering, [2] Code/Example, [3] Movement.

A

1. X- to Z-

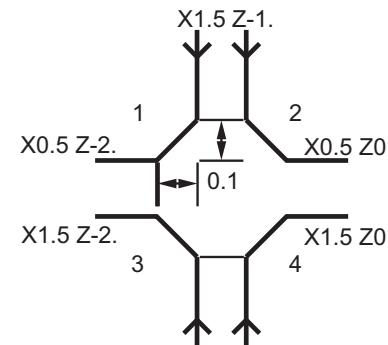
X1.5 Z-1.;
G01 X0.5 K-0.1;
Z-2.;

B

X1.5 Z-1.;
G01 X0.7;
X0.5 Z-1.1;
Z-2.

C

X1.5 Z-1.1;
G01 X0.7;
X0.5 Z-0.9;
Z0.;



2. X- to Z+

X1.5 Z-1.;
G01 X0.5 K0.1;
Z0.;

X1.5 Z-1.;
G01 X0.7;
X0.5 Z-0.9;
Z0.;

3. X+ to Z-

X0.5 Z-1.;
G01 X1.5 K-0.1;
Z-2.;

X0.5 Z-1.;
G01 X1.3;
X1.5 Z-1.1;
Z-2.

4. X+ to Z+

X0.5 Z-1.;
G01 X1.5 K0.1;
Z0.;

X0.5 Z-1.;
G01 X1.3;
X1.5 Z-0.9;
Z0.;

F6.6: Corner Rounding Code Z to X: [1] Corner rounding, [2] Code/Example, [3] Movement.

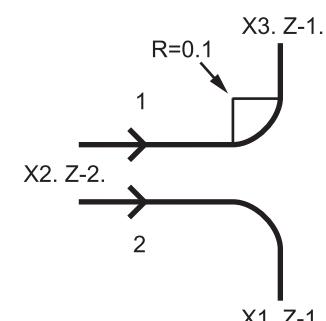
A

1. Z+ to X+

X2. Z-2.;
G01 Z-1 R.1;
X3.;

C

X2. Z-2.;
G01 Z-1.1;
G03 X2.2 Z-1. R0.1;
G01 X3.;



2. Z+ to X-

X2. Z-2.;
G01 Z-1. R-0.1;
X1.;

X2. Z-2.;
G01 Z-1.1;
G02 X1.8 Z-1 R0.1;
G01 X1.;

3. Z- to X+

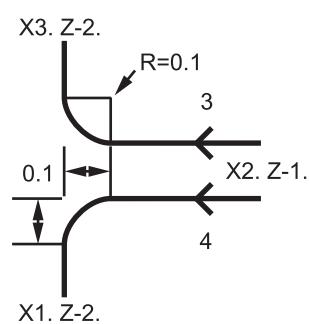
X2. Z-1.;
G01 Z-2. R0.1;
X3.;

X2. Z-1.;
G01 Z-1.9;
G02 X2.2 Z-2. R0.1;
G01 X3.;

4. Z- to X-

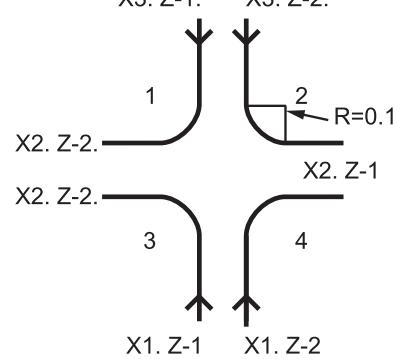
X2. Z-1.;
G01 Z-2. R-0.1;
X1.;

X2. Z-1.;
G01 Z-1.9. ;
G03 X1.8 Z-2.;
G01 X1.;



F6.7: Corner Rounding Code X to Z: [1] Corner rounding, [2] Code/Example, [3] Movement.

A	B	C	X3. Z-1.	X3. Z-2.
1. X- to Z-	X3. Z-1.; G01 X0.5 R-0.1; Z-2.;	X3. Z-1.; G01 X0.7; X0.5 Z-1.1; Z-2.;	X2. Z-2.	1
2. X- to Z+	X3. Z-2.; G01 X0.5 R0.1; Z0.;	X3. Z-2.; G01 X0.7; X0.5 Z-0.9; Z0.;	X2. Z-2.	X2. Z-1
3. X+ to Z-	X1. Z-1.; G01 X1.5 R-0.1; Z-2.;	X1. Z-1.; G01 X1.3; X1.5 Z-1.1; Z-2.;	X1. Z-1	3
4. X+ to Z+	X1. Z-2.; G01 X1.5 R0.1; Z0.;	X1. Z-21.; G01 X1.3; X1.5 Z-0.9; Z0.;	X1. Z-2	4



Rules:

1. Use **K** address only with **X (U)** address. Use **I** address only with **Z (W)** address.
2. Use **R** address with either **X (U)** or **Z (W)**, but not both in the same block.
3. Do not use **I** and **K** together on the same block. When using **R** address, do not use **I** or **K**.
4. The next block must be another single linear move that is perpendicular to the previous one.
5. Automatic chamfering or corner rounding cannot be used in a threading cycle or in a Canned cycle.
6. Chamfer or corner radius must be small enough to fit between the intersecting lines.
7. Use only a single X or Z-axis move in linear mode (G01) for chamfering or corner rounding.

G02 CW/G03 CCW Circular Interpolation Motion (Group 01)

F - Feed rate

***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 arc

***U** - X-axis incremental motion command

***W** - Z-axis incremental motion command

***X** - X-axis absolute motion command

***Y** - Y-axis absolute motion command

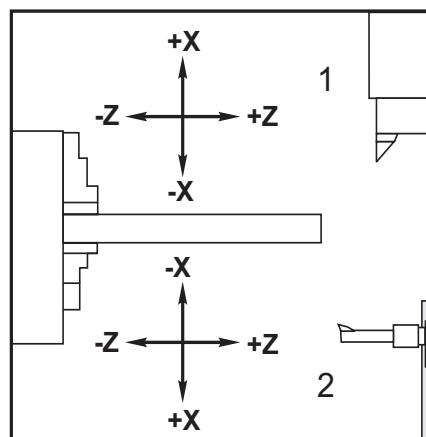
***Z** - Z-axis absolute motion command

* indicates optional

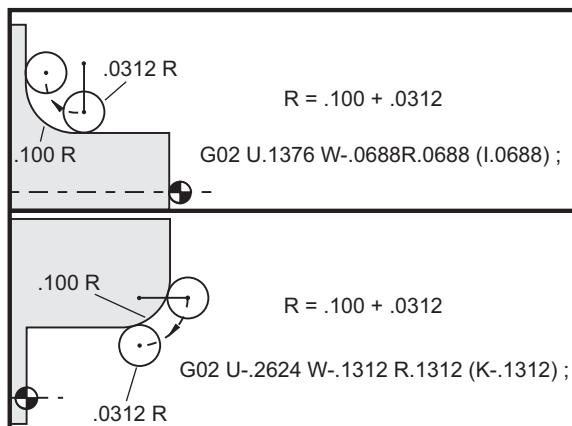
These G codes are used to specify a circular motion (CW or CCW) of the linear axes (Circular motion is possible in the X and Z axes as selected by G18). The **x** and **z** values are used to specify the end point of the motion and can use either absolute (**x** and **z**) or incremental motion (**u** and **w**). If either the **x** or **z** is not specified, the endpoint of the arc is the same as the starting point for that axis. There are two ways to specify the center of the circular motion; the first uses **i** or **k** to specify the distance from the starting point to the center of the arc; the second uses **r** to specify the radius of the arc.

For information on G17 and G19 Plane Milling, see the Live Tooling section.

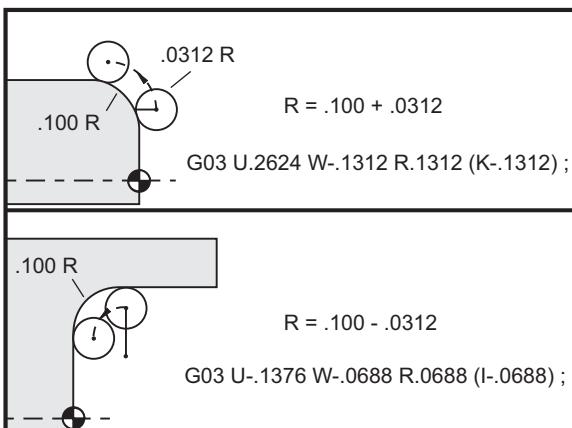
F6.8: G02 Axis Definitions: [1] Turret Lathes, [2] Table Lathes.



F6.9: G02 and G03 Programs



G02

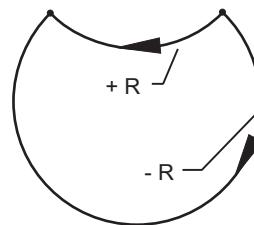


G03

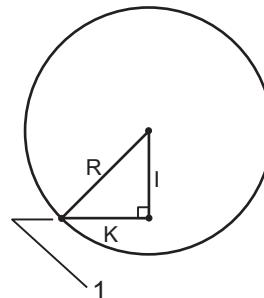
R is used to specify the radius of the arc. With a positive R , the control will generate a path of 180 degrees or less; to generate a radius of over 180 degrees, specify a negative R . X or Z is required to specify an endpoint if different from the starting point.

The following lines cut an arc of less than 180 degrees:

```
G01 X3.0 Z4.0 ;
G02 Z-3.0 R5.0 ;
```

F6.10: G02 Arc Using Radius

I and K are used to specify the center of the arc. When I and K are used, R may not be used. The I or K is the signed distance from the starting point to the center of the circle. If only one of I or K is specified, the other is assumed to be zero.

F6.11: G02 Defined X and Z: [1] Start.

6

G04 Dwell (Group 00)

P - The dwell time in seconds or milliseconds

G04 is used to cause a delay or dwell in the program. The block containing G04 will delay for the time specified by the P code. For example:

```
G04 P10.0 ;
```

Delays the program for 10 seconds.

**NOTE:**

The use of the decimal point G04 P10. is a dwell of 10 seconds; G04 P10 is a dwell of 10 milliseconds.

G09 Exact Stop (Group 00)

The G09 code is used to specify a controlled axes stop. It only affects the block in which it is commanded. It is non-modal and does not affect the following blocks. Machine moves will decelerate to the programmed point before another command is processed.

G10 Set Offsets (Group 00)

G10 allows the programmer to set offsets within the program. Using G10 replaces the manual entry of offsets (i.e. Tool length and diameter, and work coordinate offsets).

L - Selects offset category.

- L2 Work coordinate origin for COMMON and G54-G59
- L10 Geometry or shift offset
- L1 or L11 Tool wear
- L20 Auxiliary work coordinate origin for G110-G129

P - Selects a specific offset.

- P1-P50 - References geometry, wear or work offsets (L10-L11)
- P51-P100 - References shift offsets (YASNAC) (L10-L11)
- P0 - References COMMON work coordinate offset (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)

Q - Imaginary tool nose tip direction

R - Tool nose radius

***U** - Incremental amount to be added to X-axis offset

***W** - Incremental amount to be added to Z-axis offset

***X** - X-axis offset

***Z** - Z-axis offset

* indicates optional

Programming Examples

```
G10 L2 P1 W6.0 (Move coordinate G54 6.0 units to the right) ;
G10 L20 P2 X-10.Z-8. (Set work coordinate G111 to X-10.0,
Z-8.0) ;
G10 L10 P5 Z5.00 (Set geometry offset of Tool #5 to 5.00) ;
G10 L11 P5 R.0625 (Set offset of Tool #5 to 1/16") ;
```

G14 Secondary Spindle Swap / G15 Cancel (Group 17)

G14 causes the secondary spindle to become the primary spindle and will react to commands normally used for the main spindle. For example, M03, M04, M05 and M19 will affect the secondary spindle, and M143, M144, M145, and M119 will cause an alarm.

**NOTE:**

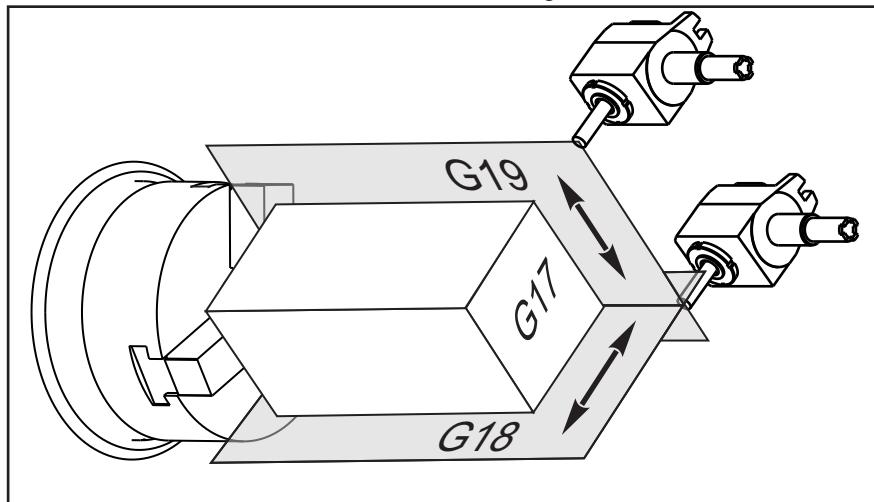
G50 will limit the secondary spindle speed, and G96 will set the secondary spindle surface feed value. These G-codes will adjust the secondary spindle speed when there is motion in the X-axis. G01 Feed Per Rev will feed based on the secondary spindle.

G14 automatically activates Z-axis mirroring. If the Z-axis is already mirrored (Setting 47 or G101), the mirror function will be canceled. G14 is canceled by a G15, an M30, reaching the end of a program, and by pressing [RESET].

G17 XY Plane (Group 02)

This code defines the plane in which tool path motion is performed. Programming tool nose radius compensation G41 or G42 applies Tool Radius cutter compensation in the G17 plane, regardless of whether G112 is active or not. For more information, refer to Cutter Compensation in the Programming section. Plane selection codes are modal and remain in effect until another plane is selected.

F6.12: G17, G18, and G19 Plane Selection Drawing



Program format with tool nose compensation:

```
G17 G01 X_ Y_ F_ ;
G40 G01 X_ Y_ I_ J_ F_ ;
```

G18 XZ Plane (Group 02)

This code defines the plane in which tool path motion is performed. Programming tool nose radius compensation G41 or G42 will apply compensation required for the nose radii of turning tools.

G19 YZ Plane (Group 02)

This code defines the plane in which tool path motion is performed. Programming tool nose radius compensation G41 or G42 will apply tool Radius cutter compensation in the G19 plane. For more information, refer to Cutter Compensation in the Programming section. Plane selection codes are modal and remain in effect until another plane is selected.

G20 Select Inches / G21 Select Metric (Group 06)

The G codes G20 (inch) and G21 (mm) codes are used to ensure that the inch/metric selection is set correctly for the program. Selection between inch and metric programming should be done using Setting 9. G20 in a program will cause the machine to alarm, if Setting 9 is not set to INCH. A G21 in a program will cause the machine to alarm if Setting 9 is not set to MM.

G28 Return to Machine Zero Point (Group 00)

The G28 code returns all axes (X, Y, Z, B and C) 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 offsets.

Programming Examples:

```
G28 X0 Z0 (moves to X0 Z0 in the current work coordinate system  
then to machine zero) ;  
G28 X1. Z1. (moves to X1. Z1. in the current work coordinate  
system then to machine zero) ;  
G28 U0 W0 (moves directly to machine zero because the initial  
incremental move is zero) ;  
G28 U-1. W-1 (moves incrementally -1. in each axis then to  
machine zero) ;
```

G29 Return From Reference Point (Group 00)

The G29 code is used to move the axes to a specific position. The axes selected in this block are moved to the G29 reference point saved in G28, and then moved to the location specified in the G29 command.

G31 Feed Until Skip (Group 00)

(This G-code is optional and requires a probe.)

This G-code is used to record a probed location to a macro variable.

**NOTE:**

Turn on the probe before using G31.

- F** - Feedrate in inches (mm) per minute
- ***U** - X-axis incremental motion command
- ***V** - Y-axis incremental motion command
- ***W** - Z-axis incremental motion command
- X** - X-axis absolute motion command
- Y** - Y-axis absolute motion command
- Z** - Z-axis absolute motion command
- C** - C-Axis absolute motion command

* 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 beeps and the skip signal position is recorded to macro variables. The program then executes the next line of code. If the probe does not receive a skip signal during the G31 move, the control does not beep, the skip signal position is recorded at the end of the programmed move, and the program continues.

Macro variables #5061 through #5066 are designated to store skip signal positions for each axis. For more information about these skip signal variables see Macros in the Programming section of this manual.

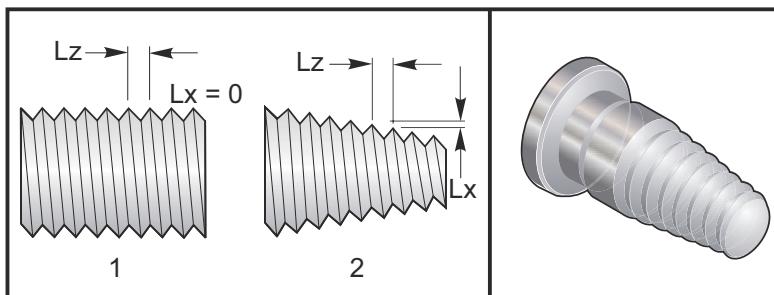
Do not use Cutter Compensation (G41 or G42) with a G31.

G32 Thread Cutting (Group 01)

**NOTE:**

Feedrate is equivalent to thread lead. Movement on at least one axis must be specified. Tapered threads have lead in both X and Z. In this case set the feedrate to the larger of the two leads. G99 (Feed per Revolution) must be active.

F6.13: G32 Definition of Lead (Feedrate): [1] Straight thread, [2] Tapered thread.



G32 differs from other thread cutting cycles in that taper and/or lead can vary continuously throughout the entire thread. In addition, no automatic position return is performed at the end of the threading operation.

At the first line of a G32 block of code, axis feed is synchronized with the rotation signal of the spindle encoder. This synchronization remains in effect for each line in a G32 sequence. It is possible to cancel G32 and recall it without losing the original synchronization. This means multiple passes will exactly follow the previous tool path. (The actual spindle RPM must be exactly the same between passes).



NOTE:

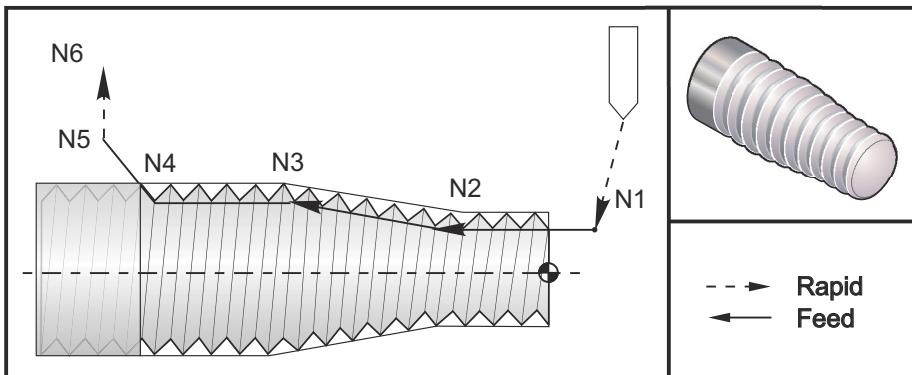
Single Block Stop and Feed Hold are deferred until last line of a G32 sequence. Feedrate Override is ignored while G32 is active, Actual Feedrate will always be 100% of programmed feedrate. M23 and M24 have no affect on a G32 operation, the user must program chamfering if needed. G32 must not be used with any G-code Canned Cycles (i.e.: G71). Do Not change spindle RPM during?? ?threading.



CAUTION:

G32 is Modal. Always cancel G32 with another Group 01 G-code at the end of a threading operation. (Group 01 G-codes: G00, G01, G02, G03, G32, G90, G92, and G9.

F6.14: Straight-to-Taper-to-Straight Thread Cutting Cycle

**NOTE:**

Example is for reference only, multiple passes are usually required to cut actual threads.

G32 Program Example:

```

...
G97 S400 M03 (Constant Surface Speed Cancel) ;
N1 G00 X0.25 Z0.1 (Rapid to Start Position) ;
N2 G32 Z-0.26 F0.065 (Straight thread, Lead(Lz) = 0.065) ;
N3 X0.455 Z-0.585 (Straight thread blends to tapered thread) ;
N4 Z-0.9425 (Taper thread blends back to straight thread) ;
N5 X0.655 Z-1.0425 (Escape at 45 degrees) ;
G00 X1.2 (Rapid to Finish Position, cancel G32) ;
G00 Z0.1 ;

```

Q Option Example:

```

G32 X-1.99 Z-2. Q60000 F0.2 (60 degree cut) ;
G32 X-1.99 Z-2. Q120000 F0.2 (120 degree cut) ;
G32 X-1.99 Z-2. Q270123 F0.2 (270.123 degree cut) ;

```

The following rules apply to the use of Q:

1. The start angle (Q) is not a modal value. It must be specified every time it is used. If no value is specified then a zero (0) angle is assumed.
2. The angle of threading increment is 0.001 degrees. Do not use a decimal point. A 180° angle must be specified as Q180000 and a 35° angle as Q35000.
3. The Q angle must be entered as a positive value from 0 to 360000.

G40 Tool Nose Compensation Cancel (Group 07)

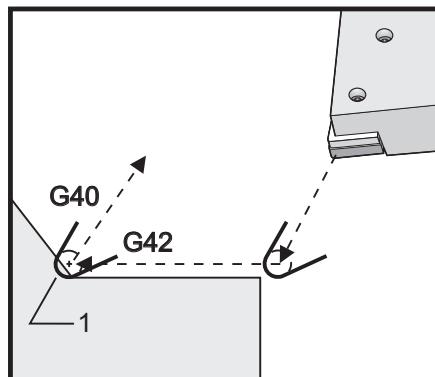
- ***X** - X Axis absolute location of departure target
- ***Z** - Z Axis absolute location of departure target
- ***U** - X Axis incremental distance to departure target
- ***W** - Z Axis incremental distance to departure target

* indicates optional

G40 cancels G41 or G42. Programming Txx00 will also cancel tool nose compensation. Cancel tool nose compensation before the end of a program.

The tool departure usually does not correspond with a point on the part. In many cases overcutting or undercutting can occur.

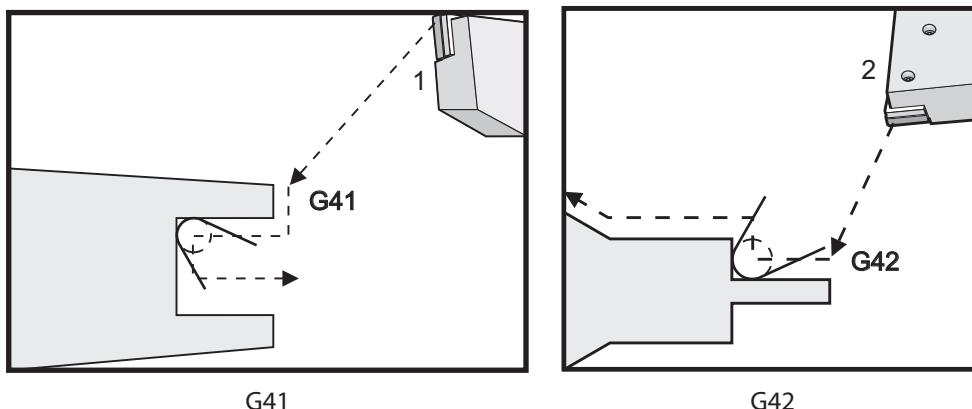
F6.15: G40 TNC Cancel: [1] Overcut.



G41 Tool Nose Compensation (TNC) Left / G42 TNC Right (Group 07)

G41 or G42 will select tool nose compensation. G41 moves the tool to the left of the programmed path to compensate for the size of a tool and vice versa for G42. A tool offset must be selected with a Tnxx code, where xx corresponds to the offsets that are to be used with the tool. For more information, see Tool Nose Compensation in the Operation section of this manual.

F6.16: G41 TNC Right and G42 TNC Left: [1] Tip = 2, [2] Tip = 3.



G50 Set Global coordinate Offset FANUC, YASNAC (Group 00)

U - Incremental amount and direction to shift global X coordinate.

X - Absolute global coordinate shift.

W - Incremental amount and direction to shift global Z coordinate.

Z - Absolute global coordinate shift.

S - Clamp spindle speed to specified value

T - Apply tool shift offset (YASNAC)

G50 can perform several functions. It can set the global coordinate, it can shift the global coordinate, and it can limit the spindle speed to a maximum value. Refer to the Global Coordinate System topic in the Programming section for a discussion of these.

To set the global coordinate, command G50 with an X or Z value. The effective coordinate will become the value specified in address code X or Z. Current machine location, work offsets and tool offsets are taken into account. The global coordinate is calculated and set.

Example:

```
G50 X0 Z0 (Effective coordinates are now zero) ;
```

To shift the global coordinate system, specify G50 with a U or W value. The global coordinate system will be shifted by the amount and direction specified in U or W. The current effective coordinate displayed will change by this amount in the opposite direction. This method is often used to place the part zero outside of the work cell.

Example:

```
G50 W-1.0 (Effective coordinates will be shifted left 1.0) ;
```

To set a YASNAC style work coordinate shift, specify G50 with a T value (Setting 33 must be set to YASNAC). The global coordinate is set to the X and Z values in the **Tool Shift offset** page. Values for the T address code are, T_{xx}yy where xx is between 51 and 100 and yy is between 00 and 50. For example, T5101 specifies tool shift index 51 and tool wear index 01; it does not cause tool number 1 to be selected. To select another T_{xx}yy code must be used outside the G50 block. The following two examples illustrate this method to select Tool 7 using Tool Shift 57 and Tool Wear 07.

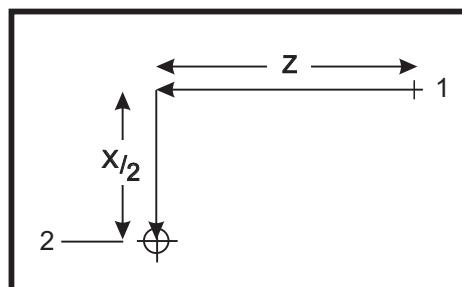
Example 1:

```
G51 (Cancel Offsets) ;
T700 M3 (Change to Tool 7, Turn on Spindle) ;
G50 T5707 (Apply Tool Shift 57 and Tool Wear 07 to Tool 7) ;
```

Example 2:

```
G51 (Cancel Offsets) ;
G50 T5700 (Apply Tool Shift 57) ;
T707 M3 (Change to Tool 7 and apply Tool Wear 07) ;
```

F6.17: G50 YASNAC Tool Shift: [1] Machine (0,0), [2] Spindle centerline.



G50 Spindle Speed Clamp

G50 can be used to limit the maximum spindle speed. The control will not allow the spindle to exceed the S address value specified in the G50 command. This is used in constant surface feed mode (G96).

This G code will also limit the secondary spindle on DS-Series machines.

```
N1G50 S3000 (Spindle rpm will not exceed 3000 rpm) ;  
N2G97 M3 (Enter constant surface speed cancel, spindle on) ;
```



NOTE:

To cancel this command, use another G50 and specify the maximum spindle RPM for the machine.

G51 Cancel Offset (YASNAC) (Group 00)

G51 is used to cancel any existing tool wear and work coordinate shift and return to the machine zero position.

G52 Set Local Coordinate System FANUC (Group 00)

This code selects the user coordinate system.

Work Coordinate Systems

The Haas CNC lathe control supports both YASNAC and FANUC coordinate systems. Work coordinates together with tool offsets can be used to position a part program anywhere within the work area. Also see the Tool Offsets section.

G53 Machine Coordinate Selection (Group 00)

This code temporarily cancels work coordinates offsets and uses the machine coordinate system.

G54 - G59 Coordinate System #1 - #6 FANUC (Group 12)

G54 - G59 codes are user-settable coordinate systems, #1 - #6, for work offsets. All subsequent references to axes' positions are interpreted in the new coordinate system. Work coordinate system offsets are entered from the **Active Work Offset** display page. For additional offsets, refer to G154 on page 351.

G61 Exact Stop Modal (Group 15)

The G61 code is used to specify exact stop. Rapid and interpolated moves will decelerate to an exact stop before another block is processed. In exact stop, moves will take a longer time and continuous cutter motion will not occur. This may cause deeper cutting where the tool stops.

G64 Exact Stop Cancel G61 (Group 15)

The G64 code is used to cancel exact stop. Selects normal cutting mode.

G65 Macro Subroutine Call Option (Group 00)

The G65 code is described in the Macros topic of the Programming section.

G70 Finishing Cycle (Group 00)

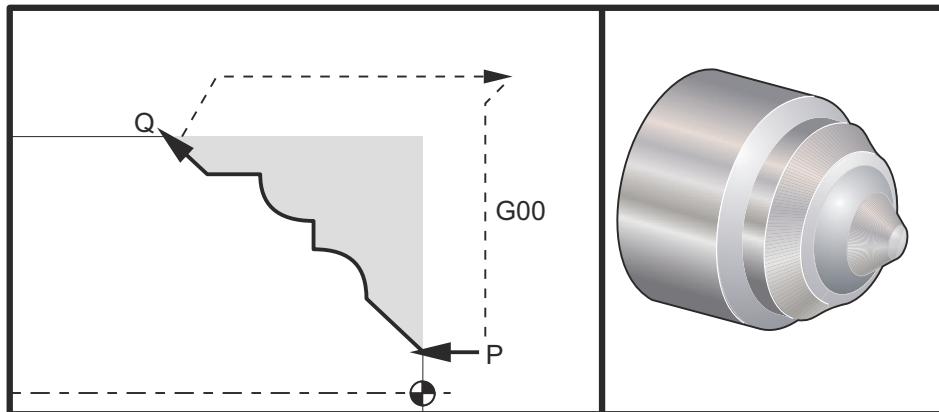
The G70 Finishing Cycle can be used to finish cut paths that are rough cut with stock removal cycles such as G71, G72 and G73.

P - Starting Block number of routine to execute

Q - Ending Block number of routine to execute

G18 Z-X plane must be active

F6.18: G70 Finishing Cycle: [P] Starting block, [Q] Ending Block.



Programming Example:

```
G71 P10 Q50 F.012 (rough out N10 to N50 the path) ;
N10 ;
F0.014 ;
...
N50 ;
...
G70 P10 Q50 (finish path defined by N10 to N50) ;
```

The G70 cycle is similar to a local subprogram call. However, the G70 requires that a beginning block number (P code) and an ending block number (Q code) be specified.

The G70 cycle is usually used after a G71, G72 or G73 has been performed using the blocks specified by P and Q. Any F, S, or T codes with the PQ block are effective. After execution of the Q block, a rapid (G00) is executed returning the machine to the start position that was saved before the starting of the G70. The program then returns to the block following the G70 call. A subroutine in the PQ sequence is acceptable providing that the subroutine does not contain a block with an N code matching the Q specified by the G70 call. This feature is not compatible with FANUC or YASNAC controls.

G71 O.D./I.D. Stock Removal Cycle (Group 00)

***D** - Depth of cut for each pass of stock removal, positive radius

***F** - Feedrate in inches (mm) per minute (G98) or per revolution (G99) to use throughout G71 PQ block

***I** - X-axis size and direction of G71 rough pass allowance, radius

***K** - Z-axis size and direction of G71 rough pass allowance

P - Starting Block number of path to rough

Q - Ending Block number of path to rough

***S** - Spindle speed to use throughout G71 PQ block

***T** - Tool and offset to use throughout G71 PQ block

***U** - X-axis size and direction of G71 finish allowance, diameter

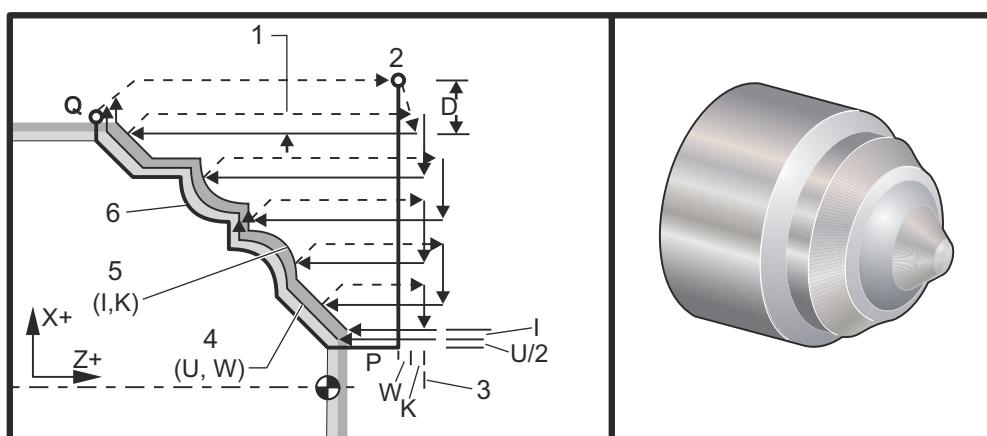
***W** - Z-axis size and direction of G71 finish allowance

***R1** - YASNAC select Type 2 roughing

* indicates optional

G18 Z-X plane must be active.

F6.19: G71 Stock Removal: [1] Setting 73, [2] Start position, [3] Z-Axis clearance plane, [4] Finishing allowance, [5] Roughing allowance, [6] Programmed path.



This canned cycle roughs material on a part given the finished part shape. Define the shape of a part by programming the finished tool path and then use the G71 PQ block. Any F,S or T commands on the G71 line or in effect at the time of the G71 is used throughout the G71 roughing cycle. Usually a G70 call to the same PQ block definition is used to finish the shape.

Two types of machining paths are addressed with a G71 command. The first type of path (Type 1) is when the X-Axis of the programmed path does not change direction. The second type of path (Type 2) allows the X-Axis to change direction. For both Type 1 and Type 2, the programmed path of the Z-axis cannot change direction. If the P block contains only an X-Axis position, then Type 1 roughing is assumed. If the P block contains both an X-Axis and Z-Axis position, then Type 2 roughing is assumed.

**NOTE:**

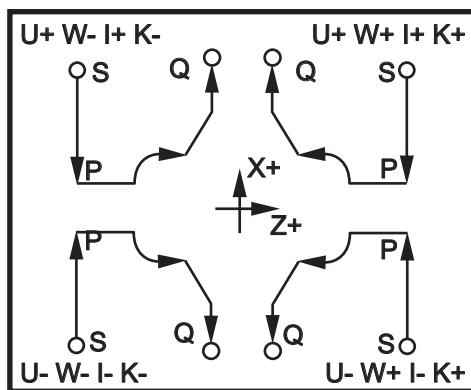
The Z-Axis position given in the P block to specify Type 2 roughing does not have to cause axis motion. You can use the current Z-Axis position. For example, In the program example on page 306, note that the P1 block (indicated by the comment in parentheses) contains the same Z-Axis position as the start position G00 block above.

When in YASNAC mode, include R1 on the G71 command block to select Type 2 roughing.

Any one of the four quadrants of the X-Z plane can be cut by specifying address codes D, I, K, U, and W properly.

In the figures, the start position S is the position of the tool at the time of the G71 call. The Z clearance plane [3] is derived from the Z-axis start position and the sum of W and optional K finish allowance.

F6.20: G71 Address Relationships



Type 1 Details

When Type 1 is specified by the programmer it is assumed that the X-axis tool path does not reverse during a cut. Each roughing pass X-axis location is determined by applying the value specified in `D` to the current X location. The nature of the movement along the Z clearance plane for each roughing pass is determined by the G code in block `P`. If block `P` contains a `G00` code, then movement along the Z clearance plane is a rapid mode. If block `P` contains a `G01` then movement will be at the `G71` feedrate.

Each roughing pass is stopped before it intersects the programmed tool path allowing for both roughing and finishing allowances. The tool is then retracted from the material, at a 45 degree angle by the distance specified in setting 73. The tool then moves in rapid mode to the Z-axis clearance plane.

When roughing is completed the tool is moved along the tool path to clean up the rough cut. If `I` and `K` are specified, an additional rough cut parallel to the tool path is performed.

Type 2 Details

When Type 2 is specified by the programmer the X Axis `PQ` path is allowed to vary (for example, the X-axis tool path can reverse direction).

The X Axis `PQ` path must not exceed the original starting location. The only exception is the ending `Q` block.

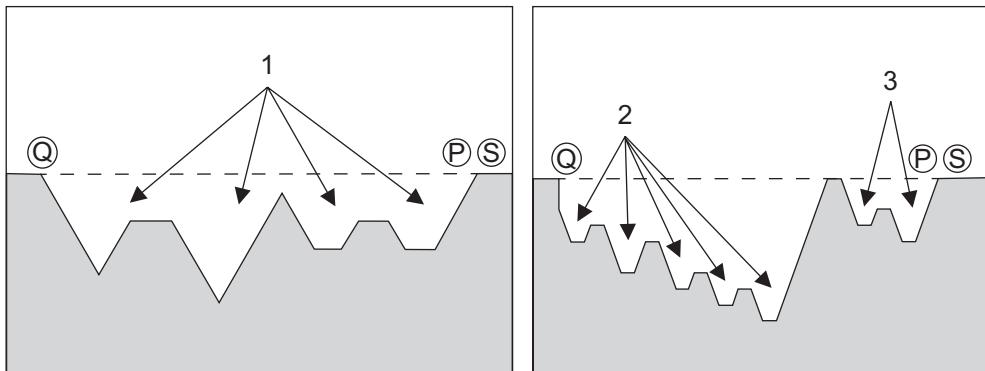
Type 2 roughing, when Setting 33 is set to `YASNAC`, must include `R1` (with no decimal) on the `G71` command block.

Type 2, when Setting 33 is set to `FANUC`, must have a reference move, in both the X and Z Axis, in the block specified by `P`.

Roughing is similar to Type 1 except after each pass along the Z Axis, the tool will follow the path defined by `PQ`. The tool will then retract parallel to the X Axis by a distance defined in Setting 73 (Can Cycle Retraction). The Type 2 roughing method does not leave steps in the part prior to finish cutting and typically results in a better finish.

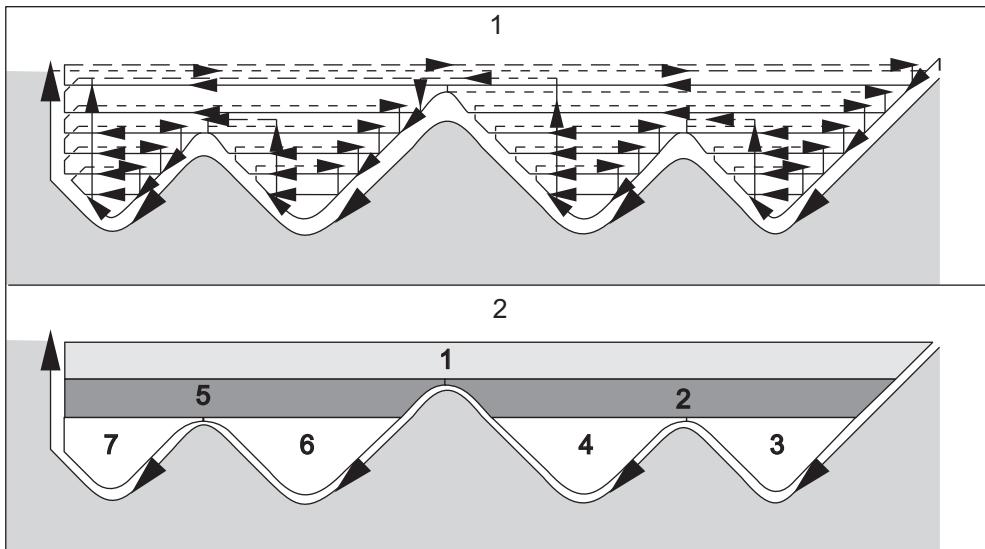
Troughs

F6.21: Single Nest with 4 Troughs [1] and Two Nests: one with 5 Troughs [2] and one with 2 Troughs [3].

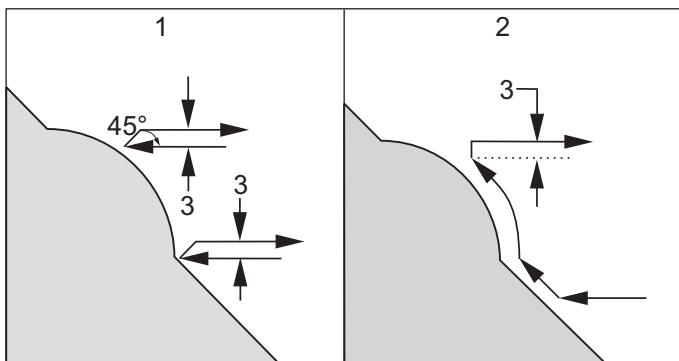


A trough can be defined as a change in direction which creates a concave surface in the material being cut. There can be no more than 10 troughs per cycle. If the part has more than 10 troughs, create another cycle. The following figures illustrate the sequence of roughing cuts (Type 1and 2) for PQ paths with multiple troughs. All material above the troughs is roughed first, followed by the troughs in the direction of Z.

F6.22: Path for Type 2 Roughing: [1] Cutter path, [2] Region Sequence.



F6.23: Type 1 and 2 Tool Retractions: [1] Type 1, [2] Type 2, [3] Setting 73.



NOTE:

An effect of using a Z finish or roughing allowance is the limit between the two cuts on one side of a trough and the corresponding point on the other side of the trough. This distance must be greater than double the sum of the roughing and finish allowances.

For example, if G71 Type 2 path contains the following:

```
...
X-5. Z-5. ;
X-5.1 Z-5.1 ;
X-3.1 Z-8.1 ;
...
```

The greatest allowance that can be specified is 0.999, since the horizontal distance from the start of cut 2 to the same point on cut 3 is 0.2. If a larger allowance is specified, over-cutting will occur.

Cutter compensation is approximated by adjusting the roughing allowance according to the radius and tip type of the tool. Therefore, the limitations that apply to the allowance also apply to the sum of the allowance and the tool radius.

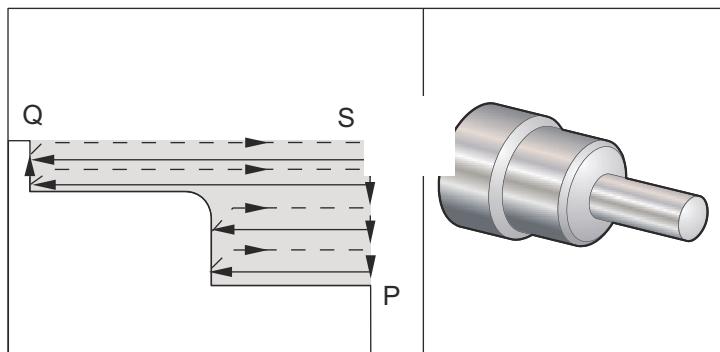


CAUTION:

If the last cut in the P-Q path is a non-monotonic curve (using a finish allowance), add a short retraction cut; do not use W.

Monotonic curves are curves that tend to move in only one direction as x increases. A monotonic increasing curve always increases as x increases, i.e. $f(a) > f(b)$ for all $a > b$. A monotonic decreasing curve always decreases as x increases, i.e. $f(a) < f(b)$ for all $a > b$. The same sort of restrictions are also made for the monotonic non-decreasing and monotonic non-increasing curves.

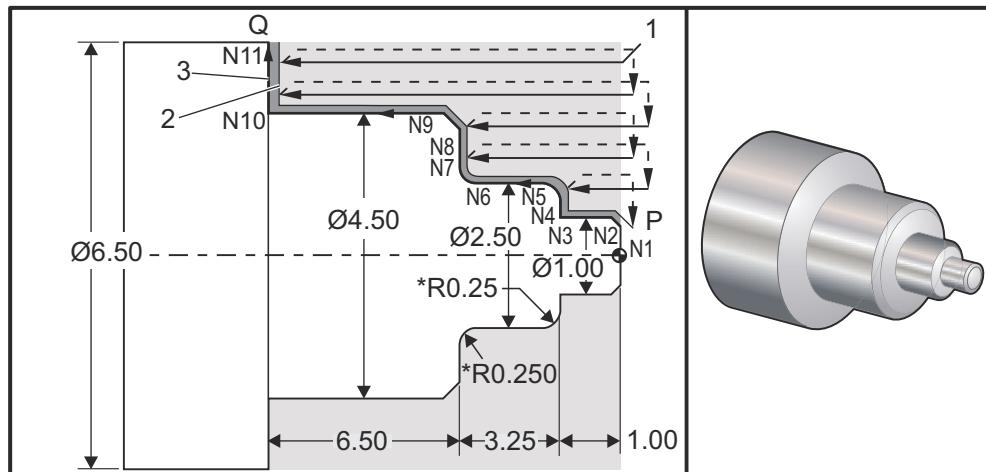
F6.24: G71 Basic G Code Example: [1] Start, [P] Starting block, [Q] Ending block.



Program Example:

```
%  
O0070(G71 Roughing Cycle) ;  
T101 ;  
G50 S2500 ;  
G97 S509 M03 ;  
G00 G54 X6. Z0.05 ;  
G96 S800 ;  
G71 P1 Q2 D0.15 U0.01 W0.005 F0.014 ;  
N1 G00 X2. ;  
G01 Z-3. F0.006 ;  
X3.5 ;  
G03 X4. Z-3.25 R0.25 ;  
G01 Z-6. ;  
N2 X6. ;  
G70 P1 Q2(FINISH PASS) ;  
M09 ;  
G53 X0 M05 ;  
G53 Z0 ;  
M30 ;  
%
```

F6.25: G71 Type 1 O.D./I.D. Stock Removal Example: [1] Start Position, [P] Starting block, [Q] Ending block, [R] Radius, [2] Finish allowance, [3] Programmed path.

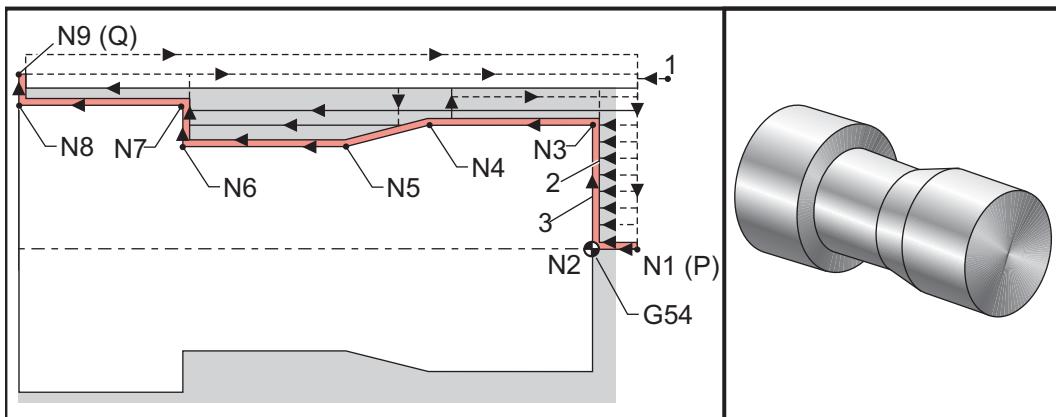


Program Example:

```
%  
O0071 (FANUC G71 TYPE 1 EXAMPLE) ;  
T101 (CNMG 432) (Tool change & apply offsets) ;  
G00 G54 X6.6 Z.05 M08 (Rapid to Home Position) ;  
G50 S2000 (Set Max RPM 2000) ;  
G97 S636 M03 (Spindle On) ;  
G96 S750 (Constant surface speed On) ;  
G71 P1 Q11 D0.15 U0.01 W0.005 F0.012 (Define rough cycle) ;  
N1 G00 X0.6634 (P Begin definition) ;  
N2 G01 X1. Z-0.1183 F0.004 (Finish pass .004" Feed) ;  
N3 Z-1. ;  
N4 X1.9376 ;  
N5 G03 X2.5 Z-1.2812 R0.2812 ;  
N6 G01 Z-3.0312 ;  
N7 G02 X2.9376 Z-3.25 R0.2188 ;  
N8 G01 X3.9634 ;  
N9 X4.5 Z-3.5183 ;  
N10 Z-6.5 ;  
N11 X6.0 (Q End definition) ;  
G00 X0 Z0 T100 (Rapid to tool change position) ;  
T202 (Finish tool) ;  
G50 S2500 ;  
G97 S955 M03 ;  
G00 X6. Z0.05 M08 ;  
G96 S1500 ;  
G70 P1 Q11 ;
```

```
G00 X0 Z0 T200 ;
M30 ;
%
```

F6.26: G71 Type 2 O.D./I.D. Stock Removal Example: [1] Start position, [P] Starting block, [Q] Ending block, [2] Finish allowance, [3] Programmed path.



Program Example:

```
%  
00125 (FANUC G71 TYPE 2 EXAMPLE)  
T101 (Tool change and apply tool offset) ;  
G54 (Select coordinate system) ;  
G50 S3000 (Spindle rpm will not exceed 3000 rpm);  
G96 S1500 M03 (Constant surface cutting speed) ;  
G00 X1. Z0.05 (Rapid move to approach starting position) ;  
G71 P1 Q9 D0.05 U0.015 W0.010 F0.01 (Define PQ block path) ;  
N1 G00 X0. Z0.05 (P1 block) ;  
N2 G01 Z0. ;  
N3 G01 X0.75 ;  
N4 G01 Z-0.5 ;  
N5 G01 X0.625 Z-0.75 ;  
N6 G01 Z-1.25 ;  
N7 G01 X0.875 ;  
N8 G01 Z-1.75 ;  
N9 G01 X1. (Q9 block) ;  
G53 G00 X0 (Rapid move to x machine home) ;  
G53 G00 Z0 (Rapid move to z machine home) ;  
T202 (Tool change and apply tool offset) ;  
G96 S1500 M03 (Constant surface cutting speed) ;  
G70 P1 Q9 F0.005(Finish path defined by PQ block) ;  
G53 G00 X0 (Rapid move to x machine home) ;  
G53 G00 Z0 (Rapid move to z machine home) ;
```

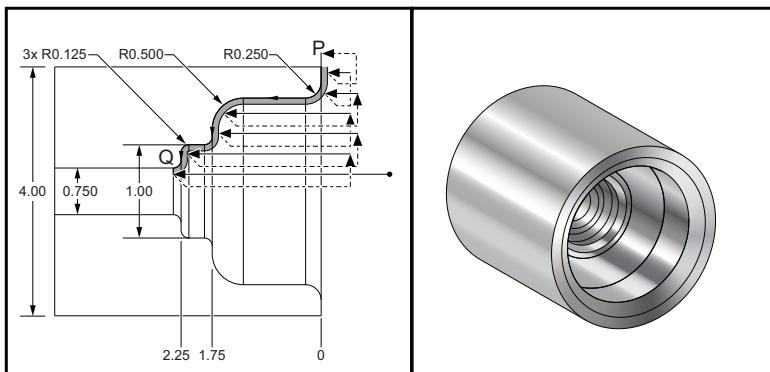
M30 ;
%

G71 I.D. Stock Removal Example


NOTE:

Be sure the start position of the tool is positioned below the diameter of the part you wish to start roughing out, before defining a G71 on an I.D. with this cycle.

F6.27: G71 I.D. Stock Removal Example



%
O1136 (Example of using a G71 on an I.D.) ;
N1 T101 (Tool 1 Offset 1) ;
N2 G97 S2000 M03 ;
N3 G54 G00 X0.7 Z0.1 M08 (Rapid to start position) ;
N4 G71 P5 Q12 U-0.01 W0.005 D0.08 F0.01 (U is a minus for G71 I.D. Roughing) ;
N5 G00 X4.5 (N5 is start of part path geometry defined by P6 in G71 line) ;
N6 G01 X3. ,R.25 F.005 ;
N7 Z-1.75 ,R.5 ;
N8 X1.5 ,R.125 ;
N9 Z-2.25 ,R.125 ;
N10 X.75 ,R.125 ;
N11 Z-3. ;
N12 X0.73 (N12 is end of part path geometry defined by Q12 in G71 line) ;
N13 G70 P5 Q12 (G70 Defines a finish pass for lines P5 through

```
Q12) ;
N14 M09 ;
N15 G53 X0 (To send machine home for a tool change) ;
G53 Z0 ;
M30 ;
%
```

G72 End Face Stock Removal Cycle (Group 00)

***D** - Depth of cut for each pass of stock removal, positive

***F** - Feedrate in inches (mm) per minute (G98) or per revolution (G99) to use throughout
G71 PQ block

***I** - X-axis size and direction of G72 rough pass allowance, radius

***K** - Z-axis size and direction of G72 rough pass allowance

P - Starting Block number of path to rough

Q - Ending Block number of path to rough

***S** - Spindle speed to use throughout G72 PQ block

***T** - Tool and offset to use throughout G72 PQ block

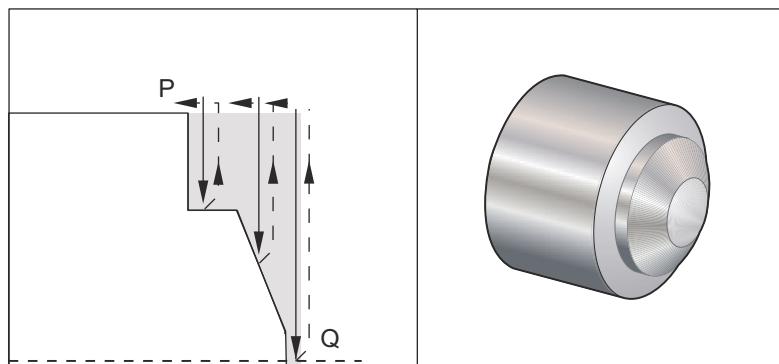
***U** - X-axis size and direction of G72 finish allowance, diameter

***W** - Z-axis size and direction of G72 finish allowance

*indicates optional

G18 Z-X plane must be active.

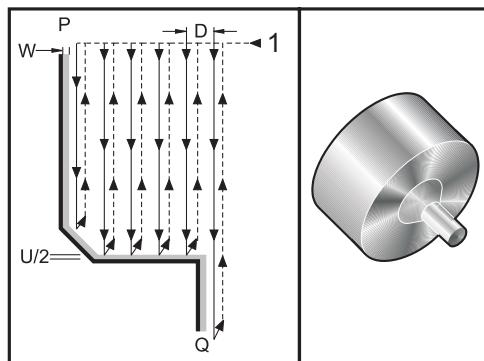
F6.28: G72 Basic G Code Example: [P] Starting block, [1] Start position,
[Q] Ending block.



Program Example:

```
%  
O0069 ;  
T101 ;  
G50 S2500 ;  
G97 S509 M03 ;  
G54 G00 X6. Z0.05 ;  
G96 S800  
G72 P1 Q2 D0.075 U0.01 W0.005 F0.012 ;  
N1 G00 Z-0.65 ;  
G01 X3. F0.006 ;  
Z-0.3633 ;  
X1.7544 Z0. ;  
X-0.0624 ;  
N2 G00 Z0.02 ;  
G70 P1 Q2(Finish Pass) ;  
M05 ;  
G53 X0 ;  
G53 Z0 ;  
M30 ;  
%
```

F6.29: G72 Tool Path: [P] Starting block, [1] Start position, [Q] Ending block.

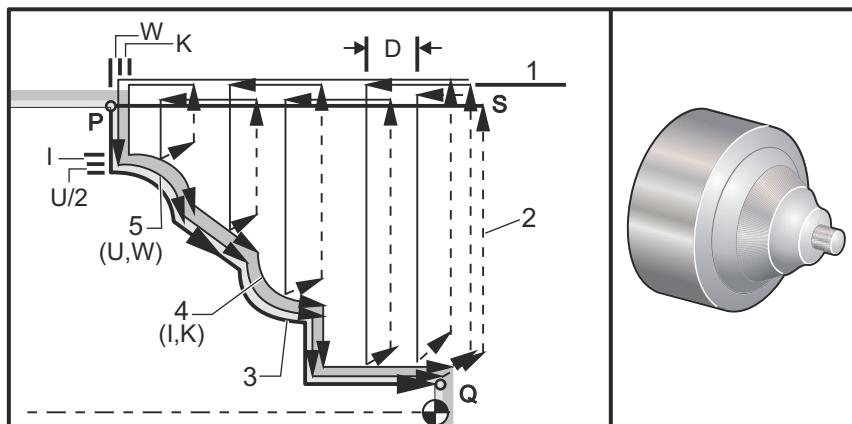


This canned cycle removes material on a part given the finished part shape. It is similar to G71 but removes material along the face of a part. Define the shape of a part by programming the finished tool path and then use the G72 PQ block. Any F,S or T commands on the G72 line or in effect at the time of the G72 is used throughout the G72 roughing cycle. Usually a G70 call to the same PQ block definition is used to finish the shape.

Two types of machining paths are addressed with a G72 command.

- The first type of path (Type 1) is when the Z Axis of the programmed path does not change direction. The second type of path (Type 2) allows the Z Axis to change direction. For both the first type and the second type of programmed path the X Axis cannot change direction. If Setting 33 is set to FANUC, Type 1 is selected by having only an X-axis motion in the block specified by P in the G72 call.
- When both an X-axis and Z-axis motion are in the P block then Type 2 roughing is assumed. If Setting 33 is set to YASNAC, Type 2 is specified by including R1 on the G72 command block (Refer to Type 2 details).

F6.30: G72 End Face Stock Removal Cycle: [P] Starting block,
 [1] X-Axis clearance plane, [2] G00 block in P, [3] Programmed path,
 [4] Roughing allowance, [5] Finishing allowance.

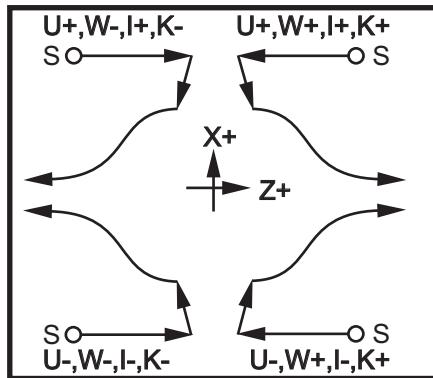


The G72 consists of a roughing phase and a finishing phase. The roughing and finishing phase are handled differently for Type 1 and Type 2. Generally the roughing phase consists of repeated passes along the X-axis at the specified feed rate. The finishing phase consists of a pass along the programmed tool path to remove excess material left by the roughing phase while leaving material for a G70 finishing cycle. The final motion in either type is a return to the starting position S.

In the previous figure the start position S is the position of the tool at the time of the G72 call. The X clearance plane is derived from the X-axis start position and the sum of U and optional I finish allowances.

Any one of the four quadrants of the X-Z plane can be cut by specifying address codes I, K, U, and W properly. The following figure indicates the proper signs for these address codes to obtain the desired performance in the associated quadrants.

F6.31: G72 Address Relationships



Type 1 Details

When Type 1 is specified by the programmer it is assumed that the Z-axis tool path does not reverse during a cut.

Each roughing pass Z-axis location is determined by applying the value specified in D to the current Z location. The nature of the movement along the X clearance plane for each roughing pass is determined by the G code in block P . If block P contains a G00 code, then movement along the X clearance plane is a rapid mode. If block P contains a G01 then movement will be at the G72 feed rate.

Each roughing pass is stopped before it intersects the programmed tool path allowing for both roughing and finishing allowances. The tool is then retracted from the material, at a 45 degree angle by the distance specified in Setting 73. The tool then moves in rapid mode to the X-axis clearance plane.

When roughing is completed the tool is moved parallel to the tool path to clean up the rough cut. If I and K are specified, an additional semi-finish cut parallel to the tool path is performed.

Type 2 Details

When Type 2 is specified by the programmer the Z Axis PQ path is allowed to vary (for example, the Z-axis tool path can reverse direction).

The Z Axis PQ path must not exceed the original starting location. The only exception is on the Q block.

Type 2 roughing when Setting 33 is set to YASNAC, must include R1 (with no decimal) on the G71 command block.

Type 2, when Setting 33 is set to **FANUC**, must have a reference move, in both the X and Z Axis, in the block specified by **P**.

Roughing is similar to Type 1 except after each pass, along the X Axis, the tool will follow the path defined by **PQ**. The tool will then retract parallel to the Z Axis by the distance defined in Setting 73 (Can Cycle Retraction). The Type 2 roughing method does not leave steps in the part prior to finish cutting and typically results in a better finish.

A side effect of using a X finish or roughing allowance is the limit between the two cuts on one side of a trough and the corresponding point on the other side of the trough. This distance must be greater than double the sum of the roughing and finish allowances.

For example, if G72 Type 2 path contains the following:

```
... ;  
X-5. Z-5. ;  
X-5.1 Z-5.1 ;  
X-8.1 Z-3.1 ;  
... ;
```

The greatest allowance that can be specified is 0.999, since the horizontal distance from the start of cut 2 to the start point on cut 3 is 0.2. If a larger allowance is specified, overcutting occurs.

Cutter compensation is approximated by adjusting the roughing allowance according to the radius and tip type of the tool. Thus, the limitations that apply to the allowance also apply to the sum of the allowance and the tool radius.

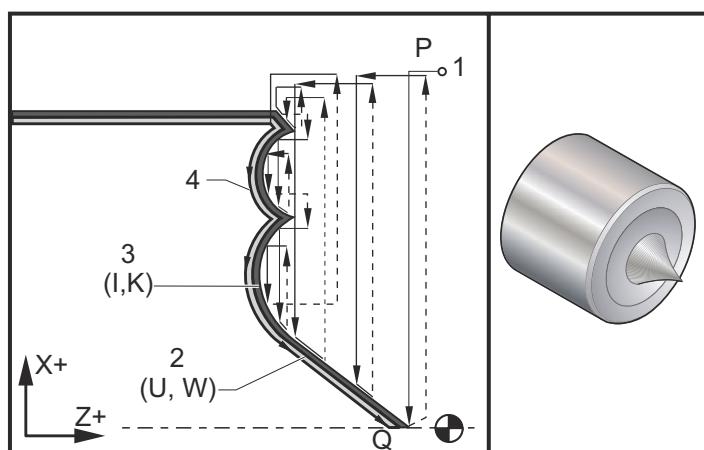


CAUTION:

If the last cut in the P-Q path is a non-monotonic curve using a finish allowance, add a short retraction cut (do not use U).

Monotonic curves are curves that tend to move in only one direction as x increases. A monotonic increasing curve always increases as x increases, i.e. $f(a) > f(b)$ for all $a > b$. A monotonic decreasing curve always decreases as x increases, i.e. $f(a) < f(b)$ for all $a > b$. The same sort of restrictions are also made for the monotonic non-decreasing and monotonic non-increasing curves. As shown in Figure F6.32, as X increases, Z decreases, then increases, then decreases, and finally increases. This X-Z curve is definitely non-monotonic. Thus, the need for a short retraction cut.

- F6.32:** G72 End Face Removal: [P] Starting block, [1] Start position, [Q] Ending block, [2] Finishing allowance, [3] Roughing allowance, [4] Programmed path.



Program Example:

```
%  
00722 (G72 Roughing Cycle) ;  
T101 ;  
S1000 M03 ;  
G00 G54 X2.1 Z0.1 ;  
G72 P1 Q2 D0.06 I0.02 K0.01 U0.0 W0.01 S1100 F0.015 ;  
N1 G01 Z-0.46 X2.1 F0.005 ;  
X2. ;  
G03 X1.9 Z-0.45 R0.2 ;  
G01 X1.75 Z-0.4 ;  
G02 X1.65 Z-.4 R0.06 ;  
G01 X1.5 Z-0.45 ;  
G03 X1.3 Z-0.45 R0.12 ;  
G01 X1.17 Z-0.41 ;  
G02 X1.03 Z-0.41 R0.1 ;  
G01 X0.9 Z-0.45 ;  
G03 X0.42 Z-0.45 R0.19 ;  
G03 X0.2 Z-0.3 R0.38 ;  
N2 G01 X0.01 Z0 ;  
G70 P1 Q2 (Finish Pass) ;  
M05 ;  
G53 X0 ;  
G53 Z0 ;  
M30 ;  
%
```

G73 Irregular Path Stock Removal Cycle (Group 00)

D - Number of cutting passes, positive number

"F - Feedrate in inches (mm) per minute (G98) or per revolution (G99) to use throughout G73 PQ block

I - X-axis distance and direction from first cut to last, radius

K - Z-axis distance and direction from first cut to last

P - Starting Block number of path to rough

Q - Ending Block number of path to rough

***S** - Spindle speed to use throughout G73 PQ block

***T** - Tool and offset to use throughout G73 PQ block

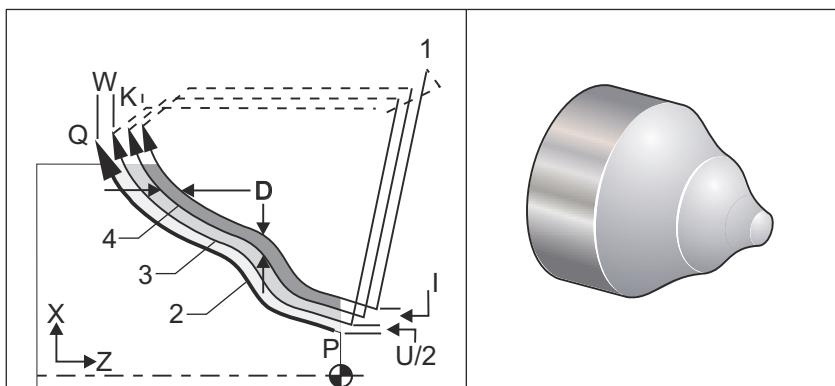
***U** - X-axis size and direction of G73 finish allowance, diameter

***W** - Z-axis size and direction of G73 finish allowance

* indicates optional

G18 Z-X plane must be active

F6.33: G73 Irregular Path Stock Removal: [P] Starting block, [Q] Ending block
 [1] Start position, [2] Programmed path, [3] Finish allowance,
 [4] Roughing allowance.



The G73 canned cycle can be used for rough cutting of preformed material such as castings. The canned cycle assumes that material has been relieved or is missing a certain known distance from the programmed tool path PQ.

Machining starts from the current position (S), and either rapids or feeds to the first rough cut. The nature of the approach move is based on whether a G00 or G01 is programmed in block P. Machining continues parallel to the programmed tool path. When block Q is reached a rapid departure move is executed to the Start position plus the offset for the second roughing pass. Roughing passes continue in this manner for the number of rough passes specified in D. After the last rough is completed, the tool returns to the starting position S.

Only F, S and T prior to or in the G73 block are in effect. Any feed (F), spindle speed (S) or tool change (T) codes on the lines from P to Q are ignored.

The offset of the first rough cut is determined by $(U/2 + I)$ for the X Axis, and by $(W + K)$ for the Z Axis. Each successive roughing pass moves incrementally closer to the final roughing finish pass by an amount of $(I/(D-1))$ in the X Axis, and by an amount of $(K/(D-1))$ in the Z Axis. The last rough cut always leaves finish material allowance specified by U/2 for the X Axis and W for the Z Axis. This canned cycle is intended for use with the G70 finishing canned cycle.

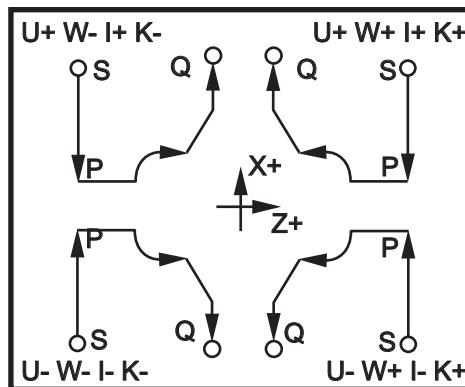
The programmed tool path PQ does not have to be monotonic in X or Z, but care has to be taken to insure that existing material does not interfere with tool movement during approach and departure moves.


NOTE:

Monotonic curves are curves that tend to move in only one direction as x increases. A monotonic increasing curve always increases as x increases, i.e. $f(a)>f(b)$ for all $a>b$. A monotonic decreasing curve always decreases as x increases, i.e. $f(a)<f(b)$ for all $a>b$. The same sort of restrictions are also made for the monotonic non-decreasing and monotonic non-increasing curves.

The value of D must be a positive integral number. If the D value includes a decimal, an alarm is generated. The four quadrants of the ZX plane can be machined if the following signs for U, I, W, and K are used.

F6.34: G71 Address Relationships



G74 End Face Grooving Cycle (Group 00)

***D** - Tool clearance when returning to starting plane, positive

***F** - Feed rate

***I** - X-axis size of increment between peck cycles, positive radius

K - Z-axis size of increment between pecks in a cycle

***U** - X-axis incremental distance to furthest peck (diameter)

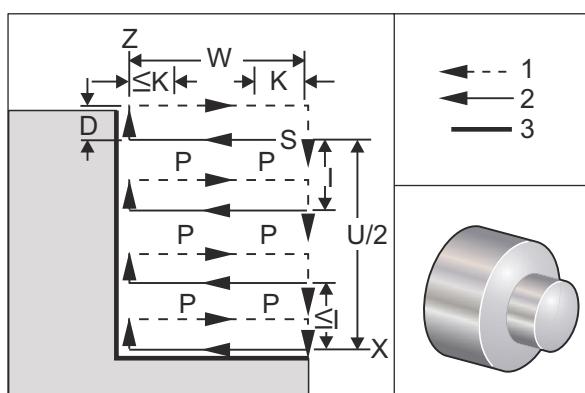
W - Z-axis incremental distance to total pecking depth

***X** - X-axis absolute location of furthest peck cycle (diameter)

Z - Z-axis absolute location total pecking depth

*indicates optional

F6.35: G74 End Face Grooving Cycle Peck Drilling: [1] Rapid, [2] Feed, [3] Programmed Path, [S] Start position, [P] Peck retraction (Setting 22).

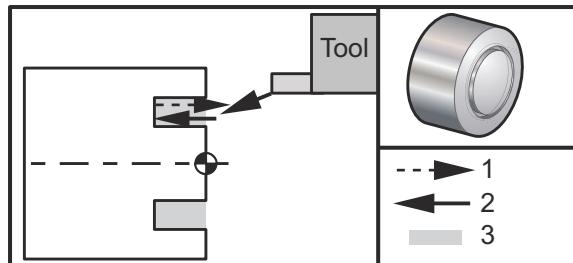


The G74 canned cycle is used for grooving on the face of a part, peck drilling, or turning.

A minimum of two pecking cycles occur, if an **X**, or **U**, code is added to a G74 block and **X** is not the current position. One at the current location and then at the **X** location. The **I** code is the incremental distance between X-Axis pecking cycles. Adding an **I** performs multiple pecking cycles between the starting position **S** and **X**. If the distance between **S** and **X** is not evenly divisible by **I** then the last interval is less than **I**.

When **K** is added to a G74 block, pecking is performed at each interval specified by **K**, the peck is a rapid move opposite the direction of feed with a distance defined by Setting 22. The **D** code can be used for grooving and turning to provide material clearance when returning to starting plane **S**.

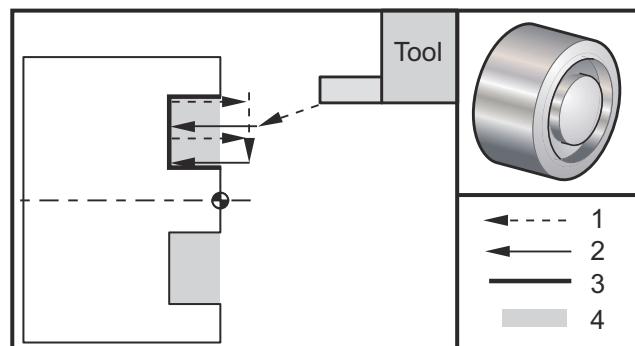
F6.36: G74 End Face Grooving Cycle: [1] Rapid, [2] Feed, [3] Groove.



Program Example:

```
%  
O0071 ;  
T101 ;  
G97 S750 M03 ;  
G00 X3. Z0.05 (Rapid to Start position) ;  
G74 Z-0.5 K0.1 F0.01 (Feed Z-.5 with a .100" peck) ;  
G53 X0 ;  
G53 Z0 ;  
M30 ;  
%
```

F6.37: G74 End Face Grooving Cycle (Multiple Pass): [1] Rapid, [2] Feed, [3] Programmed path, [4] Groove.



Program Example:

```
%  
O0074 ;  
T101 ;  
G97 S750 M03 ;  
G00 X3. Z0.05 (Rapid to Start position) ;  
G74 X1.75 Z-0.5 I0.2 K0.1 F0.01 (Face grooving cycle multiple  
pass) ;  
G53 X0 ;  
G53 Z0 ;  
M30 ;  
%
```

G75 O.D./I.D. Grooving Cycle (Group 00)

***D** - Tool clearance when returning to starting plane, positive

***F** - Feed rate

***I** - X-axis size of increment between pecks in a cycle (radius measure)

***K** - Z-axis size of increment between peck cycles

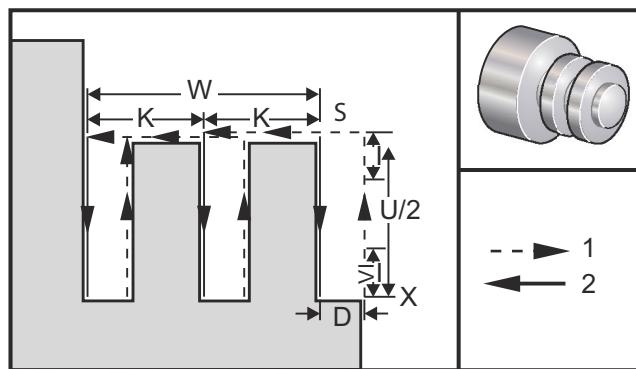
***U** - X-axis incremental distance to total pecking depth

W - Z-axis incremental distance to furthest peck cycle

***X** - X-axis absolute location total pecking depth (diameter)

Z - Z-axis absolute location to furthest peck cycle

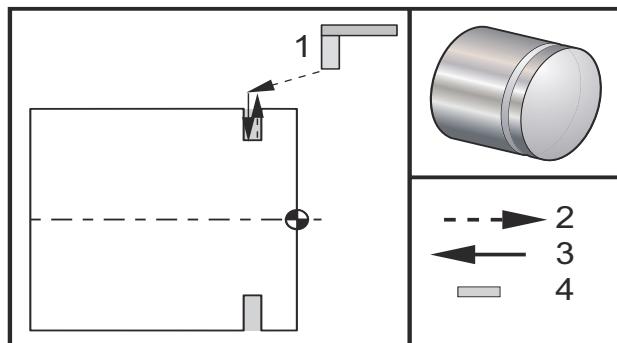
* indicates optional

F6.38: G75 O.D./I.D. Grooving Cycle: [1] Rapid, [2] Feed, [S] Start position.

The G75 canned cycle can be used for grooving an outside diameter. When a Z , or w , code is added to a G75 block and Z is not the current position, then a minimum of two pecking cycles occur. One at the current location and another at the Z location. The K code is the incremental distance between Z axis pecking cycles. Adding a K performs multiple, evenly spaced, grooves. If the distance between the starting position and the total depth (Z) is not evenly divisible by K then the last interval along Z is less than K .

**NOTE:**

Chip clearance is defined by Setting 22.

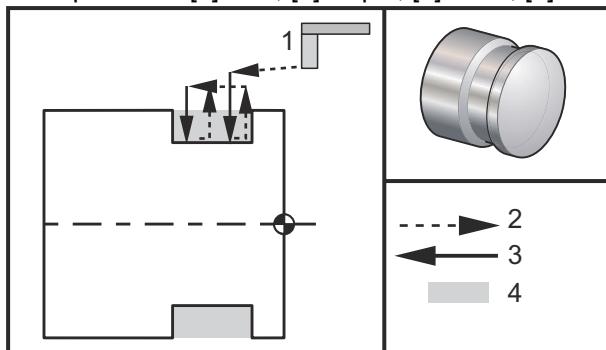
F6.39: G75 O.D. Single Pass

Program Example:

```
%  
O0075 ;  
T101 ;  
G97 S750 M03 ;  
G00 X4.1 Z0.05 (Rapid to Clear position) ;  
G01 Z-0.75 F0.05 (Feed to Groove location) ;  
G75 X3.25 I0.1 F0.01 (O.D./I.D. Peck grooving single pass) ;  
G00 X5. Z0.1 ;  
G53 X0 ;  
G53 Z0 ;  
M30 ;  
%
```

The following program is an example of a G75 program (Multiple Pass):

F6.40: G75 O.D. Multiple Pass: [1] Tool, [2] Rapid, [3] Feed, [4] Groove.



Program Example:

```
%  
O0075 ;  
T101 ;  
G97 S750 M03 ;  
G00 X4.1 Z0.05 (Rapid to Clear position) ;  
G01 Z-0.75 F0.05 (Feed to Groove location) ;  
G75 X3.25 Z-1.75 I0.1 K0.2 F0.01 (O.D./I.D. Peck grooving  
multiple pass) ;  
G00 X5. Z0.1 ;  
G28 ;  
M30 ;  
%
```

G76 Threading Cycle, Multiple Pass (Group 00)

***A** - Tool nose angle (value: 0 to 120 degrees) Do not use a decimal point

D - First pass cutting depth

F(E) - Feed rate, the lead of the thread

***I** - Thread taper amount, radius measure

K - Thread height, defines thread depth, radius measure

***P** - Single Edge Cutting (load constant)

***Q** - Thread Start Angle (Do not use a decimal point)

***U** - X-axis incremental distance, start to maximum thread Depth Diameter

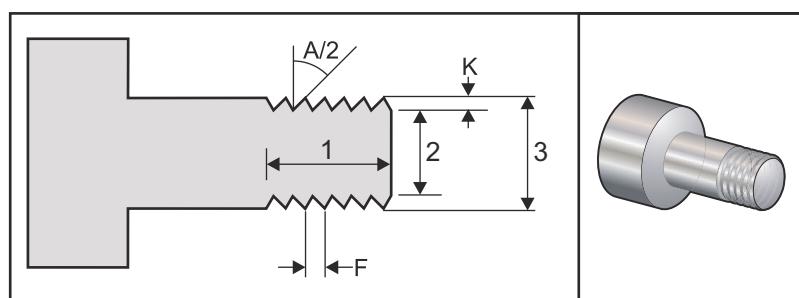
***W** - Z-axis incremental distance, start to maximum thread length

***X** - X-axis absolute location, maximum thread Depth Diameter

***Z** - Z-axis absolute location, maximum thread length

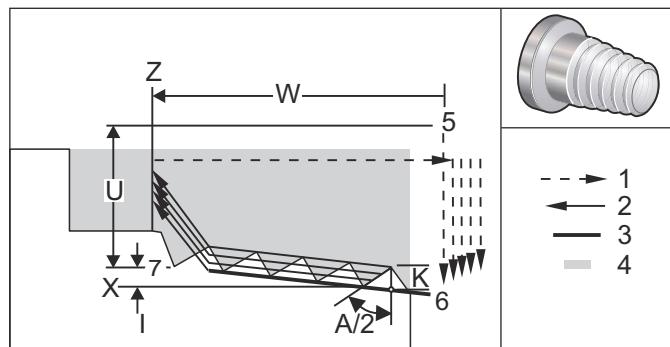
* indicates optional

F6.41: G76 Threading Cycle, Multiple Pass: [1] Z depth, [2] Minor diameter, [3] Major diameter.



Setting 95/Setting 96 determine chamfer size/angle; M23/M24 turn chamfering ON/OFF.

F6.42: G76 Threading Cycle, Multiple Pass Tapered: [1] Rapid, [2] Feed, [3] Programmed path, [4] Cut allowance, [5] Start position, [6] Finished diameter, [7] Target, [A] Angle.



The G76 canned cycle can be used for threading both straight or tapered (pipe) threads.

The height of the thread is defined as the distance from the crest of the thread to the root of the thread. The calculated depth of thread (K) is the value of K less the finish allowance (Setting 86, Thread Finish Allowance).

The thread taper amount is specified in I . Thread taper is measured from the target position x, z at point [7] to position [6]. The I value is the difference in radial distance from the start to the end of the thread, not an angle.



NOTE:

A conventional O.D. taper thread will have a negative I value.

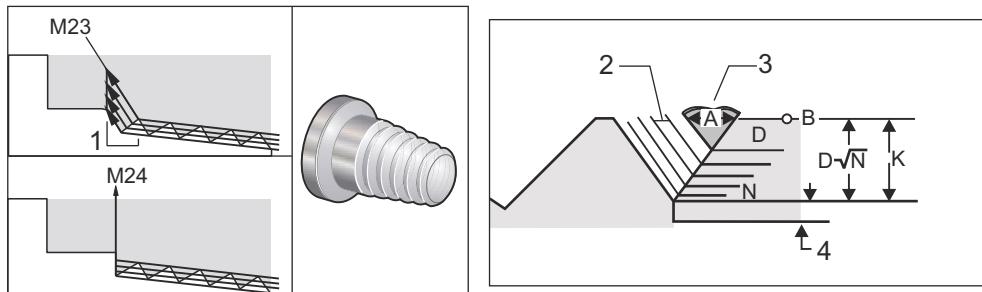
The depth of the first cut through the thread is specified in D . The depth of the last cut through the thread can be controlled with Setting 86.

The tool nose angle for the thread is specified in A . The value can range from 0 to 120 degrees. If A is not used, 0 degrees is assumed. To reduce chatter while threading use A59 when cutting a 60 degree included thread.

The F code specifies the feed rate for threading. It is always good programming practice to specify G99 (feed per revolution) prior to a threading canned cycle. The F code also indicates the thread pitch or lead.

At the end of the thread an optional chamfer is performed. The size and angle of the chamfer is controlled with Setting 95 (Thread Chamfer Size) and Setting 96 (Thread Chamfer Angle). The chamfer size is designated in number of threads, so that if 1.000 is recorded in Setting 95 and the feed rate is .05, then the chamfer will be .05. A chamfer can improve the appearance and functionality of threads that must be machined up to a shoulder. If relief is provided for at the end of the thread then the chamfer can be eliminated by specifying 0.000 for the chamfer size in Setting 95, or using M24. The default value for Setting 95 is 1.000 and the default angle for the thread (Setting 96) is 45 degrees.

- F6.43:** G76 Using an A Value: [1] Setting 95 and 96 (see Note),
[2] Setting_99 - Thread Minimum Cut, [3] Cutting Tip,
[4] Setting 86 - Finish Allowance.



**NOTE:**

Setting 95 and 96 will affect the final chamfer size and angle.

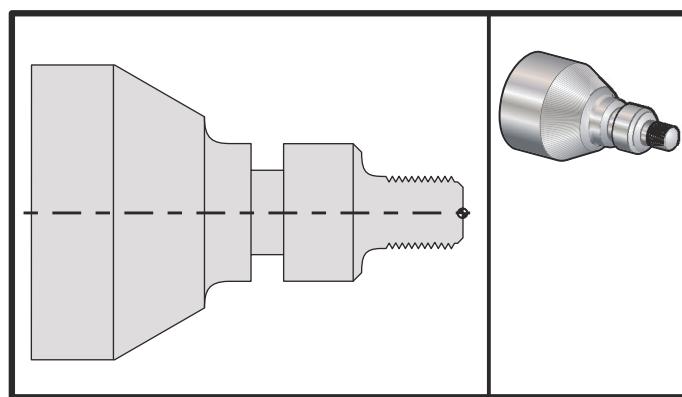
Four options for G76 Multiple Thread Cutting are available:

1. P1:Single edge cutting, cutting amount constant
2. P2:Double edge cutting, cutting amount constant
3. P3: Single edge cutting, cutting depth constant
4. P4: Double edge cutting, cutting depth constant

P1 and P3 both allow for single edge threading, but the difference is that with P3 a constant depth cut is done with every pass. Similarly, P2 and P4 options allow for double edge cutting with P4 giving constant depth cut with every pass. Based on industry experience, double edge cutting option P2 may give superior threading results.

D specifies the depth of the first cut. Each successive cut is determined by the equation $D * \sqrt{N}$ where N is the Nth pass along the thread. The leading edge of the cutter does all of the cutting. To calculate the X position of each pass you have to take the sum of all the previous passes, measured from the start point the X value of each pass

F6.44: G76 Thread Cutting Cycle, Multiple Pass



Program Example:

```
%  
T101 ;  
G50 S2500 (Set max RPM select tool geometry) ;  
G97 S1480 M03 (Spindle on select tool one offset one) ;  
G54 G00 X3.1 Z0.5 M08 (Select work coord. and rapid to  
reference point, coolant on) ;  
G96 S1200 (Constant surface speed ON) ;  
G01 Z0 F0.01 (Position to part Z0) ;  
X-0.04 ;
```

```
G00 X3.1 Z0.5 ;
G71P1 Q10 U0.035 W0.005 D0.125 F0.015 (Define roughing cycle)
;
N1 X0.875 Z0 (Begin tool path) ;
N2 G01 X1. Z-0.075 F0.006 ;
N3 Z-1.125 ;
N4 G02 X1.25 Z-1.25 R0.125 ;
N5 G01 X1.4 ;
N6 X1.5 Z-1.3 ;
N7 Z-2.25 ;
N8 G02 X1.9638 Z-2.4993 R0.25 ;
N9 G03X2.0172 Z-2.5172 R0.0325 ;
N10 G01 X3. Z-3.5 (End tool path) ;
G00 Z0.1 M09 ;
G53 X0 ;
G53 Z0 ;
N20 (Thread sample program FANUC System) ;
T505 ;
G50 S2000 ;
G97 S1200 M03 (Threading tool) ;
G00 X1.2 Z0.3 M08 (Rapid to position) ;
G76 X0.913 Z-0.85 K0.042 D0.0115 F0.0714 (Threading cycle) ;
G00X1.5 Z0.5 G28 M09 ;
N30 (HAAS SL-Series FANUC System) ;
T404 ;
G50 S2500 ;
G97 S1200 M03 (Groove tool) ;
G54 G00 X1.625 Z0.5 M08 ;
G96 S800 ;
G01 Z-1.906 F0.012 ;
X1.47 F0.006 ;
X1.51 ;
W0.035 ;
G01 W-0.035 U-0.07 ;
G00 X1.51 ;
W-0.035 ;
G01 W0.035 U-0.07 ;
X1.125 ;
G01 X1.51 ;
G00 X3. Z0.5 M09 ;
G53 X0 ;
G53 Z0 ;
M30 ;
%
```

Example Using Start Thread Angle (Q)

```
G76 X1.92 Z-2. Q60000 F0.2 D0.01 K0.04 (60 degree cut) ;
G76 X1.92 Z-2. Q120000 F0.2 D0.01 K0.04 (120 degree cut) ;
G76 X1.92 Z-2. Q270123 F0.2 D0.01 K0.04 (270.123 degree cut) ;
```

The following rules apply to the usage of Q:

1. The start angle, Q, must be specified every time it is used. If no value is specified then a zero (0) angle is assumed.
2. Do not use a decimal point. The angle of threading increment is 0.001 degrees. Therefore, a 180° angle must be specified as Q180000 and an angle of 35° as Q35000.
3. The Q angle must be entered as a positive value from 0 to 360000.

Multiple Start Threading Example

Multiple threads can be cut by changing the start point for each threading cycle.

The previous example has been modified to now create a multiple start thread.

To calculate the additional start points the feed F0.0714 (Pitch) is multiplied by the number of start points (3) to give $.0714 * 3 = .2142$. This is the new feed rate F0.2142 (lead).

The pitch (0.0714) is added to the initial Z-axis start point (N2) in order to calculate the next start point (N5).

Add the same amount again to the previous start point (N5) to calculate the next start point (N7).

Example #1

```
T101 (1.00-14 3 LEAD THREAD) ;
(1.00/14 = PITCH = 0.0714) ;
(PITCH = 0.0714 is the Z Axis shift for each lead) ;
(0.0714 * 3 = LEAD = .2142) ;
(LEAD = .2142 is the feed rate) ;
N1 M08 ;
N2 G00 G54 X1.100 Z.500 (Initial Start Point) ;
N3 G97 S400 M03 ;
N4 G76 X.913 Z-.850 K.042 D.0115 F.2142 (Thread Cycle) ;
N5 G00 X1.100 Z.5714 (.500 ORIGINAL START +.0714) ;
N6 G76 X.913 Z-.850 K.042 D.0115 F.2142 (Thread Cycle) ;
N7 G00 X1.100 Z.6428 (2ND START .5714 +.0714) ;
N8 G76 X.913 Z-.850 K.042 D.0115 F.2142 (Thread Cycle) ;
N9 G00 X6.00 Z6.00 ;
N10 M30 ;
```

G80 Canned Cycle Cancel (Group 09*)

This G code is modal in that it deactivates all canned cycles.



NOTE:

Use of G00 or G01 will also cancel a canned cycle.

G81 Drill Canned Cycle (Group 09)

***C** - C-Axis absolute motion command (optional)

F - Feed Rate

***L** - Number of repeats

R - Position of the R plane

***W** - Z-axis incremental distance

***X** - X-axis motion command

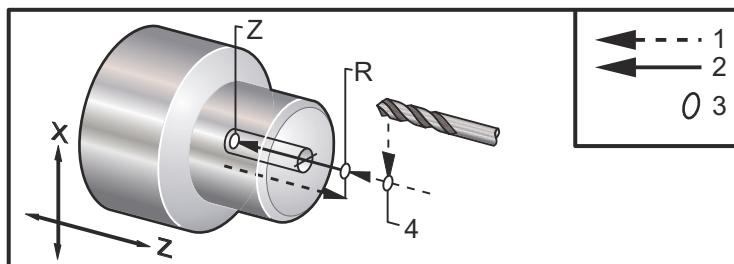
***Y** - Y-axis absolute motion command

***Z** - Position of bottom of hole

* indicated optional

Also see G241 for radial drilling and G195/G196 for radial tapping with live tooling.

F6.45: G81 Drill Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Starting plane, [R] R plane, [Z] Position at the bottom of the hole.



G82 Spot Drill Canned Cycle (Group 09)

*C - C-Axis absolute motion command (optional)

F - Feed Rate in inches (mm) per minute

*L - Number of repeats

P - The dwell time at the bottom of the hole

R - Position of the R plane

W - Z-axis incremental distance

*X - X-axis motion command

*Y - Y-axis motion command

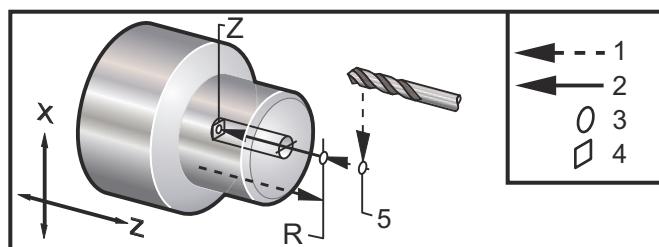
*Z - Position of bottom of hole

* indicates optional

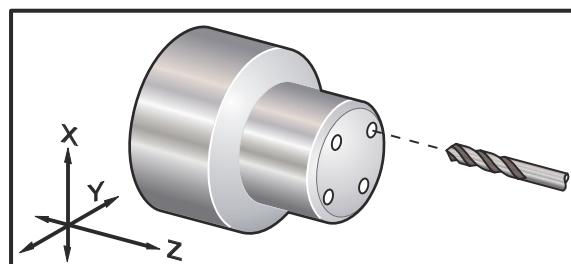
This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X will cause this canned cycle to be executed.

Also, see G242 for radial live tool spot drilling.

F6.46: G82 Spot Drill Canned Cycle:[1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Dwell, [5] Starting plane, [R] R plane, [Z] Position of the bottom of the hole.



F6.47: G82 Y-Axis Drill



```
(Live Spot Drill - Axial) ;
T1111 ;
G18 (Call reference plane) ;
```

```
G98 (Feed per minute) ;
M154 (Engage C-Axis) ;
G00 G54 X6. C0. Y0. Z1. ;
G00 X1.5 Z0.25 ;
G97 P1500 M133 ;
M08 ;
G82 G98 C45. Z-0.25 F10. P80;
C135. ;
C225. ;
C315. ;
G00 G80 Z0.25 M09 ;
M155 ;
M135 ;
M09 ;
G00 G28 H0. (Unwind C-Axis) ;
G00 X6. Y0. Z1. ;
G18 (Return to XZ plane) ;
G99 (Inches per minute) ;
M01 ;
M30 ;
%
```

To calculate how long you should dwell at the bottom of your spot drill cycle, use the following formula:

$$P = \text{Dwell Revolutions} \times 60000/\text{RPM}$$

If you want the tool to dwell for two full revolutions at its full Z depth in the program above (running at 1500 RPM), you would calculate:

$$2 \times 60000 / 1500 = 80$$

Enter P80 (80 milliseconds or P.08 (.08 seconds) on the G82 line, to dwell for 2 revolutions at 1500 RPM.

G83 Normal Peck Drilling Canned Cycle (Group 09)

***C** - C-Axis absolute motion command (optional)

F - Feed Rate in inches (mm) per minute

***I** - Size of first cutting depth

***J** - Amount to reduce cutting depth each pass

***K** - Minimum depth of cut

***L** - Number of repeats

***P** - The dwell time at the bottom of the hole

***Q** - The cut-in value, always incremental

***R** - Position of the R plane

***W** - Z-axis incremental distance

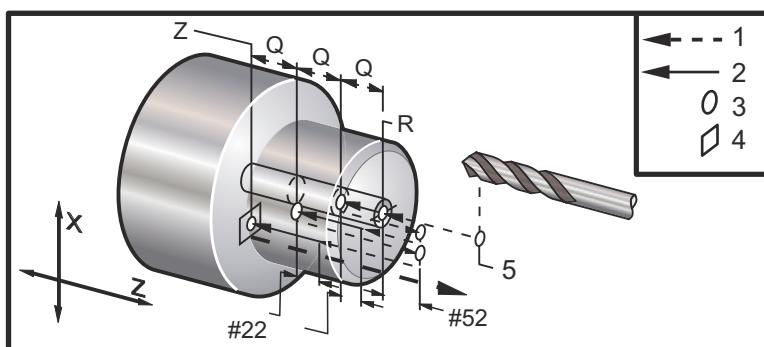
***X** - X-axis motion command

***Y** - Y-axis motion command

Z - Position of bottom of hole

* indicates optional

F6.48: G83 Peck Drilling Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Dwell, [#22] Setting 22, [#52] Setting 52.



NOTE:

If I , J , and K are specified, a different operating mode is selected. The first pass will cut in the value 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 .

Setting 52 changes the way G83 works when it returns to the R plane. Usually the R plane is set well outside the cut to insure that the chip clearing motion allows the chips to clear the hole. However, this is wasted motion when first drilling through this empty space. If Setting 52 is set to the distance required to clear chips, the R plane can be put much closer to the part being drilled. When the clear move to R occurs, the Z will be moved past R by this value in Setting 52. Setting 22 is the amount to feed in Z to get back to the same point at which the retraction occurred.

Program Example:

```
T101 ;
G97 S500 M03 ;
G00 X0 Z1. M08 ;
G99
G83 Z-1.5 F0.005 Q0.25 R0.1 ;
G80 ;
M09 ;
G53 X0 ;
G53 Z0 ;
M30 ;
%
```

Program Example (Live Tool):

```
(LIVE PECK DRILL - AXIAL) ;
T1111 ;
G98 ;
M154 (Engage C-Axis) ;
G00 G54 X6. C0. Y0. Z1. ;
G00 X1.5 Z0.25 ;
G97 P1500 M133 ;
M08 ;
G83 G98 C45. Z-0.8627 F10. Q0.125 ;
C135. ;
C225. ;
C315. ;
G00 G80 Z0.25 ;
M155 ;
M135 ;
M09 ;
G28 H0. (Unwind C-Axis) ;
G00 G54 X6. Y0. Z1. ;
G18 ;
G99 ;
M01 ;
M30 ;
%
```

G84 Tapping Canned Cycle (Group 09)

F - Feed Rate

***R** - Position of the R plane

S - RPM, called prior to G84

***W** - Z-axis incremental distance

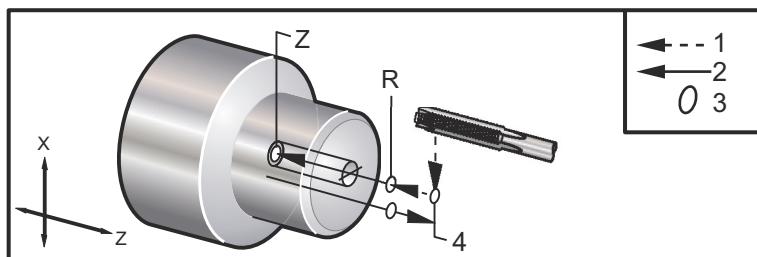
***X** - X-axis motion command

Z - Position of bottom of hole

* indicates optional

Programming Notes: It is not necessary to start the spindle CW before this canned cycle. The control does this automatically.

F6.49: G84 Tapping Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Starting plane, [R] R plane, [Z] Position at the bottom of the hole.



When G84 tapping on a lathe, it is simplest to use G99 Feed Per Revolution.

The feedrate, when using G99, is equal to the Lead of the tap.

The Lead is the distance traveled along a screw's axis, with each full revolution.

An S value must be called prior to the G84. The S value determines the RPM of the tapping cycle.

In Metric Mode (G99, with Setting 9 = MM), the feedrate is the metric equivalent of the lead, in MM.

In Inch Mode (G99, with Setting 9 = INCH), the feedrate is the Inch equivalent of the lead, in inches.

Examples:

The lead (and G99 feedrate) of an M10 x 1.0mm tap is 1.0mm, or .03937" (1.0/25.4=.03937).

The lead of a 5/16-18 tap is 1.411mm (1/18*25.4=1.411), or .0556" (1/18=.0556)

This canned cycle can be used on the secondary spindle of a Dual Spindle DS lathe, when prefaced by a G14. Refer to the G14 Secondary Spindle Swap on 291 for more information.

For Axial Live-Tool tapping, use a G95 or G186 command.

For Radial Live-Tool tapping, use a G195 or G196 command.

For Reverse Tapping (left hand thread) on the Main or Secondary Spindle, refer to 353.

More programming examples, in both Inch and Metric, are shown below:

Setting 9 Dimensioning = mm

Imperial tap, G99 Feed per revolution	Metric tap, G99 Feed per revolution
O00840 (G84 TAP, SET9=MM) ; G21 (ALARM IF SET9 NOT MM) ; T0101 (1/4-20 TAP) ; G54 G00 X0. Z12.7 ; G99 (FEED PER REV) ; S800 (RPM OF TAP CYCLE) ; G84 Z-12.7 R12.7 F1.27 (1/20*25.4=1.27) ; G00 G80 ; M30 ;	O00841 (G84 TAP, SET9=MM) ; G21 (ALARM IF SET9 NOT MM) ; T0202 (M8 x 1.25 TAP) ; G54 G00 X0. Z12.7 ; G99 (FEED PER REV) ; S800 (RPM OF TAP CYCLE) ; G84 Z-12.7 R12.7 F1.25 (LEAD=1.25) ; G00 G80 ; M30 ;

Setting 9 Dimensioning = inch

Imperial tap, G99 Feed per revolution	Metric tap, G99 Feed per revolution
O00842 (G84 TAP, SET9=IN) ; G20 (ALARM IF SET9 NOT INCH) ; T0101 (1/4-20 TAP) ; G54 G00 X0. Z.5 ; G99 (FEED PER REV) ; S800 (RPM OF TAP CYCLE) ; G84 Z-.5 R.5 F0.05 (1/20=.05) ; G00 G80 ; M30 ;	O00843 (G84 TAP, SET9=IN) ; G20 (ALARM IF SET9 NOT INCH) ; T0202 (M8 x 1.25 TAP) ; G54 G00 X0. Z.5 ; G99 (FEED PER REV) ; S800 (RPM OF TAP CYCLE) ; G84 Z-.5 R.5 F0.0492 (1.25/25.4=.0492) ; G00 G80 ; M30 ;

G85 Boring Canned Cycle (Group 09)


NOTE:

This cycle feeds in and feeds out.

F - Feed Rate

***L** - Number of repeats

***R** - Position of the R plane

***W** - Z-axis incremental distance

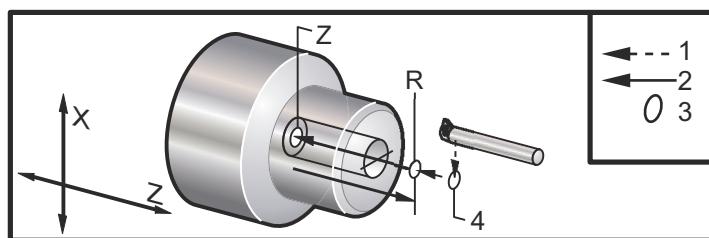
***X** - X-axis motion command

***Y** - Y-axis motion command

Z - Position of bottom of hole

* indicates optional

F6.50: G85 Boring Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Starting plane, [R] R plane, [Z] Position of the bottom of the hole.



G86 Bore and Stop Canned Cycle (Group 09)

6


NOTE:

The spindle stops and it rapids out of the hole.

F - Feed Rate

***L** - Number of repeats

***R** - Position of the R plane

***W** - Z-axis incremental distance

***X** - X-axis motion command

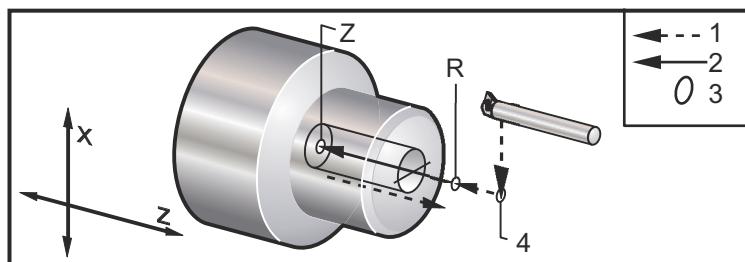
***Y** - Y-axis motion command

***Z** - Position of bottom of hole

* indicates optional

This G code stops the spindle once the tool reaches the bottom of the hole. The tool retracts once the spindle has stopped.

F6.51: G86 Bore and Stop Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Starting plane, [R] R plane, [Z] Position of the bottom of the hole.



G87 Bore and Manual Retract Canned Cycle (Group 09)

F - Feed Rate

***L** - Number of repeats

***R** - Position of the R plane

***W** - Z-axis incremental distance

***X** - X-axis motion command

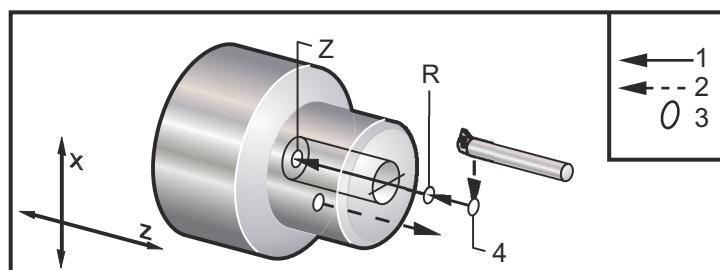
***Y** - Y-axis motion command

***Z** - Position of bottom of hole

* indicates optional

This G code stops the spindle at the bottom of the hole. At this point the tool is manually jogged out of the hole. The program continues when **[CYCLE START]** is pressed.

F6.52: G87 Bore and Manual Retract Canned: [1] Feed, [2] Manual Retract, [3] Start or end of stroke, [4] Starting plane, [R] R plane, [Z] Position of the bottom of the hole. Cycle.



G88 Bore and Dwell and Manual Retract Canned Cycle (Group 09)

F - Feed Rate

***L** - Number of repeats

***P** - The dwell time at the bottom of the hole

***R** - Position of the R plane

***W** - Z-axis incremental distance

***X** - X-axis motion command

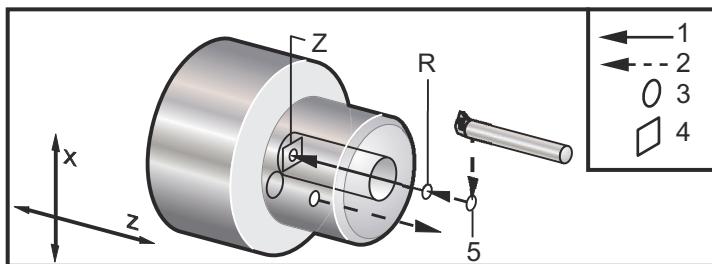
***Y** - Y-axis motion command

***Z** - Position of bottom of hole

* indicates optional

This G code stops the tool at the bottom of the hole and dwells with the spindle turning for the time designated with the **P** value. At this point the tool is manually jogged out of the hole. The program continues when **[CYCLE START]** is pressed.

F6.53: G88 Bore and Dwell and Manual Retract Canned Cycle: [1] Feed, [2] Manual Retract, [3] Start or end of stroke, [4] Dwell, [5] Starting plane, [R] R plane, [Z] Position of the bottom of the hole.

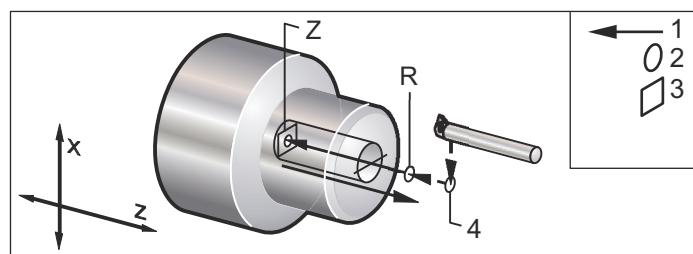


G89 Bore and Dwell Canned Cycle (Group 09)

**NOTE:***This cycle feeds in and feeds out.***F** - Feed Rate***L** - Number of repeats***P** - The dwell time at the bottom of the hole***R** - Position of the R plane***W** - Z-axis incremental distance***X** - X-axis motion command***Y** - Y-axis motion command***Z** - Position of bottom of hole

* indicates optional

F6.54: G89 Bore and Dwell Canned Cycle: [1] Feed, [2] Start or end of stroke, [3] Dwell, [4] Starting plane, [R] R plane, [Z] Position of the bottom of the hole.

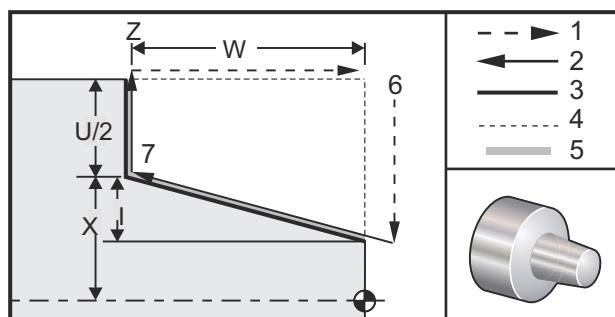


G90 O.D./I.D. Turning Cycle (Group 01)

F(E) - Feed rate***I** - Optional distance and direction of X Axis taper, radius***U** - X-axis incremental distance to target, diameter***W** - Z-axis incremental distance to target**X** - X-axis absolute location of target**Z** - Z-axis absolute location of target

*indicates optional

F6.55: G90 O.D./I.D. Turning Cycle: [1] Rapid, [2] Feed, [3] Programmed path, [4] Cut allowance, [5] Finish allowance, [6] Start position, [7] Target.

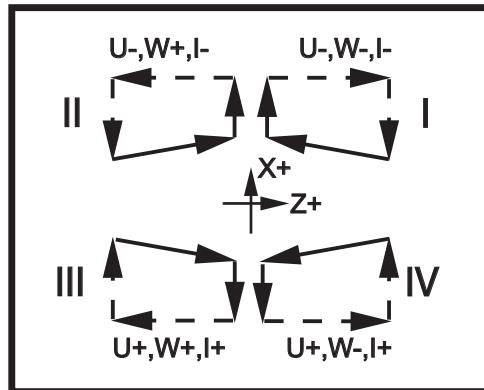


G90 is used for simple turning, however, multiple passes are possible by specifying the X locations of additional passes.

Straight turning cuts are made by specifying X , Z and F . By adding an I value, a taper cut is made. The amount of taper is referenced from the target. That is, I is added to the value of X at the target.

Any of the four ZX quadrants can be programmed using U , W , X , and Z ; the taper is positive or negative. The following figure gives a few examples of the values required for machining in each of the four quadrants.

F6.56: G90-G92 Address Relationships



G92 Threading Cycle (Group 01)

F(E) - Feed rate, the lead of the thread

***I** - Optional distance and direction of X Axis taper, radius

***Q** - Start Thread Angle

***U** - X-axis incremental distance to target, diameter

***W** - Z-axis incremental distance to target

X - X-axis absolute location of target

Z - Z-axis absolute location of target

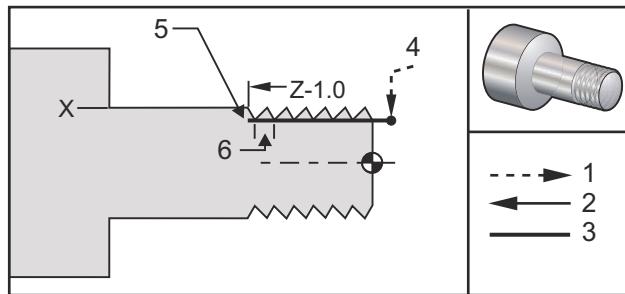
* indicates optional

Programming Notes: Setting 95/Setting 96 determine chamfer size/angle. M23/M24 turn chamfering on/off.

G92 is used for simple threading, however, multiple passes for threading are possible by specifying the **X** locations of additional passes. Straight threads are made by specifying **X**, **Z**, and **F**. By adding an **I** value, a pipe or taper thread is cut. The amount of taper is referenced from the target. That is, **I** is added to the value of **X** at the target. At the end of the thread, an automatic chamfer is cut before reaching the target; default for this chamfer is one thread at 45 degrees. These values can be changed with Setting 95 and Setting 96.

During incremental programming, the sign of the number following the **U** and **W** variables depends on the direction of the tool path. For example, if the direction of a path along the **X**-axis is negative, the value of **U** is negative.

F6.57: G92 Threading Cycle: [1] Rapid, [2] Feed, [3] Programmed path, [4] Start position, [5] Minor diameter, [6] 1/Threads per inch = Feed per revolution (Inch formula; **F** = lead of thread).



Program Example:

```
%  
O0156 (1"-12 THREAD CUTTING PROGRAM) ;  
T101 ;  
G54 ;  
G50 S3000 M3 ;  
G97 S1000 ;  
X1.2 Z.2 ( RAPID TO CLEAR POSITION) ;  
G92 X.980 Z-1.0 F0.0833 (SET UP THREAD CYCLE) ;  
X.965 (2ND PASS) (SUBSEQUENT CYCLES) ;  
X.955 (3RD PASS) ;  
X.945 (4TH PASS) ;  
X.935 (5TH PASS) ;  
X.925 (6TH PASS) ;  
X.917 (7TH PASS) ;  
X.910 (8TH PASS) ;  
X.905 (9TH PASS) ;  
X.901 (10TH PASS) ;  
X.899 (11TH PASS) ;  
G53 X0 ;  
G53 Z0 ;  
M30 ;  
%
```

Example Using Start Thread Angle Q

6

```
G92 X-1.99 Z-2. Q60000 F0.2 (60 degree cut) ;  
G92 X-1.99 Z-2. Q120000 F0.2 (120 degree cut) ;  
G92 X-1.99 Z-2. Q270123 F0.2 (270.123 degree cut) ;
```

The following rules apply to the usage of Q:

1. The start angle, Q, must be specified every time it is used. If no value is specified then a zero (0) angle is assumed.
2. The angle of threading increment is 0.001 degrees. Do not use a decimal point in the entry; for example, a 180° angle must be specified as Q180000 and an angle of 35° as Q35000.
3. The Q angle must be entered as a positive value from 0 to 360000.

In general, when multi-threads are being performed it is a good practice to achieve the depth of the threads at a uniform level across all the threading angles. One way to achieve this is to make a sub-program that only causes the Z-axis to move for the different angles of threading. After the sub-program has finished, change the X-axis depth and call the sub-program again.

G94 End Facing Cycle (Group 01)

F(E) - Feed rate

***K** - Optional distance and direction of Z Axis coning

***U** - X-axis incremental distance to target, diameter

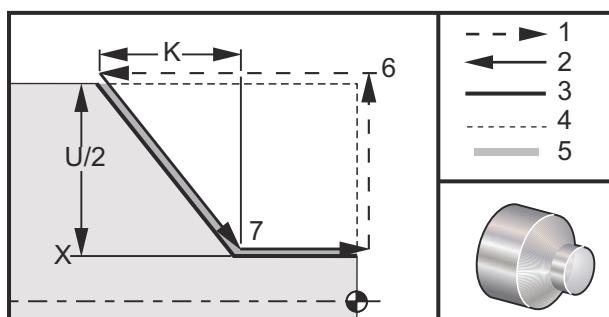
***W** - Z-axis incremental distance to target

X - X-axis absolute location of target

Z - Z-axis absolute location of target

*indicates optional

F6.58: G94 End Facing Cycle: [1] Rapid, [2] Feed, [3] Programmed path, [4] Cut allowance, [5] Finish allowance, [6] Start position, [7] Target.

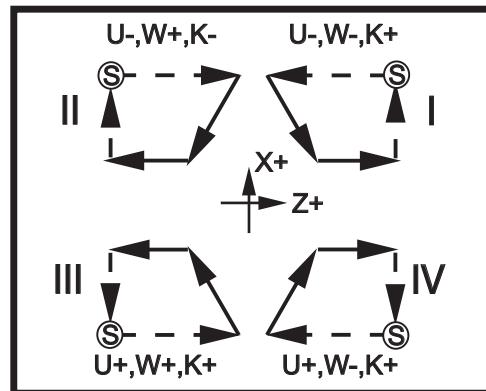


Straight end facing cuts can be made by specifying **X**, **Z** and **F**. By adding **K** a cone-shaped face is cut. The amount of coning is referenced from the target. That is **K** is added to the value of **X** at the target.

Any of the four ZX quadrants is programmed by varying **U**, **W**, **X**, and **Z**. The coning is positive or negative. The following figure gives a few examples of the values required for machining in each of the four quadrants.

During incremental programming, the sign of the number following the **U** and **W** variables depends on the direction of the tool path. If the direction of a path along the **X**-axis is negative, the value of **U** is negative.

F6.59: G94 Address Relationships: [S] Start position.



G95 Live Tooling Rigid Tap (Face) (Group 09)

*C - C-Axis absolute motion command (optional)

F - Feed Rate

R - Position of the R plane

S - RPM, called prior to G95

W - Z-axis incremental distance

X - Optional Part Diameter X-axis motion command

*Y - Y-axis motion command

Z - Position of bottom of hole

* indicates optional

G95 Live Tooling Rigid Tapping is an axial tapping cycle similar to G84 Rigid Tapping in that it uses the F, R, X and Z addresses, however, it has the following differences:

- The control must be in G99 Feed per Revolution mode in order for tapping to work properly.
- An S (spindle speed) command must have been issued prior to the G95.
- The X Axis must be positioned between machine zero and the center of the main spindle, do not position beyond spindle center.

Program Example:

```
T1111 (LIVE TAP - AXIAL 1/4 x 20 Tap) ;
G99 ;
M154 (ENGAGE C-Axis) ;
G00 G54 X6. C0. Y0. Z1. ;
G00 X1.5 Z0.5 ;
M08 ;
S500 ;
G95 C45. Z-0.5 R0.5 F0.05 ;
```

```
C135. ;
C225. ;
C315. ;
G00 G80 Z0.5 M09 ;
M135 ;
M155 ;
G28 H0. (Unwind C-Axis) ;
G00 G54 X6. Y0 Z1. ;
G99 (Inches per minute) ;
M01 ;
M30 ;
%
```

G96 Constant Surface Speed ON (Group 13)

G96 commands the control to maintain a constant cutting speed at the tip of the tool. The spindle RPM is based on the diameter of the part where the cut is taking place, and the commanded S value (RPM=3.82xSFM/DIA). This means the spindle speed increases as the tool gets closer to X0. When Setting 9 is set to **INCH**, the S value specifies Surface Feet Per Minute. When Setting 9 is set to **MM**, the S value specifies Surface Meters Per Minute.



WARNING: *It is safest to specify a maximum spindle speed for the Constant Surface Speed feature. Use G50 to set a maximum spindle RPM.*

Not setting a limit allows the spindle speed to increase as the tool reaches the center of the part. The excessive speed can throw parts and damage tooling.

G97 Constant Surface Speed OFF (Group 13)

This commands the control to NOT adjust the spindle speed based on the diameter of cut and cancels any G96 command. When G97 is in effect, any S command is revolutions per minute (RPM).

G98 Feed Per Minute (Group 10)

G98 changes how the F address code is interpreted. The value of F indicates inches per minute when Setting 9 is set to **INCH**, and F indicates millimeters per minute when Setting 9 is set to **MM**.

G99 Feed Per Revolution (Group 10)

This command changes how the **F** address is interpreted. The value of **F** indicates inches per revolution of the spindle when Setting 9 is set to **INCH**, while **F** indicates millimeters per revolution of the spindle when Setting 9 is set to **MM**.

G100/G101 Disable/Enable Mirror Image (Group 00)

***X** - X-axis command

***Z** - Z-axis command

* indicates optional. At least one is required.

Programmable mirror image can be turned on or off individually for the X and/or Z Axis. The bottom of the screen indicates when an axis is mirrored. These G codes are used in a command block without any other G codes and do not cause any Axis motion. G101 turns on mirror image for any Axis listed in that block. G100 turns off mirror image for any Axis listed in the block. The actual value given for the **X** or **Z** code has no effect; G100 or G101 by itself has no effect. For example, G101 X 0 turns on X-axis mirror.



NOTE:

Settings 45 through 48 may be used to manually select mirror image.

G102 Programmable Output to RS-232 (Group 00)

***X** - X-axis command

***Z** - Z-axis command

* indicates optional

Programmable output to the RS-232 port sends the current work coordinates of the axes to another computer. Use this G code in a command block without any other G codes; it does not cause any axis motion.

Programming note: Optional spaces (Setting 41) and EOB control (Setting 25) are applied.

Digitizing a part is possible using this G code and a program that steps over a part in X-Z and probes across in Z with a G31. When the probe hits, the next block could be a G102 to send the **X** and **Z** position out to a computer which stores the coordinates as a digitized part. Additional software for the personal computer is required to complete this function.

G103 Limit Block Lookahead (Group 00)

G103 contains the maximum number of blocks the control looks ahead (Range 0-15), for example:

```
G103 [P..] ;
```

This is commonly referred to, as “Block Look-ahead”, and describes what the control is doing in the background during machine motions. The control prepares future blocks (lines of code) ahead of time. While the current block is executing, the next block has already been interpreted and prepared for continuous motion.

When G103 P0 is programmed, block limiting is disabled. Block limiting is also disabled if G103 appears in a block without a P address code. When G103 Pn is programmed, look-ahead is limited to n blocks.

G103 is also useful for debugging macro programs. Macro expressions are done during look-ahead time. For example, by inserting a G103 P1 into the program, macro expressions are performed one 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.

G105 Servo Bar Command

This is the G-code used to command a Bar Feeder.

```
G105 [In.nnnn] [Jn.nnnn] [Kn.nnnn] [Pnnnnn] [Rn.nnnn]
```

I - Optional Initial Push Length (macro variable #3101) Override (variable #3101 if I is not commanded)

J - Optional Part Length + Cutoff (macro variable #3100) Override (variable #3100 if J is not commanded)

K - Optional Min Clamping Length (macro variable #3102) Override (variable #3102 if K is not commanded)

P - Optional subprogram

R - Optional spindle orientation for new bar

I, J, K are overrides to macro variable values listed on the Current Commands Page. The control applies the override values only to the command line in which they are located. The values stored in Current Commands are not modified.

You should normally put the G105 command at the end of your part program, to prevent a double push if you stop and then restart the program.

When you command G105, the Bar Feeder does one of these operations, based on the length of the current bar and the value of **MINIMUM CLAMPING LENGTH** (#3102 or **K**) added to **PART LENGTH + CUTOFF** (#3100 or **J**):

1. If the current bar is long enough to correctly clamp and machine a new part (the bar is longer than **MINIMUM CLAMPING LENGTH** plus **PART LENGTH + CUTOFF**):
 - a) If there is a **P** value in the G105 block, the control runs the subprogram.
 - b) The spindle stops.
 - c) The workholding unclamps.
 - d) The Bar Feeder pushes the bar the distance specified in **PART LENGTH + CUTOFF** (#3100) or, if the G105 block has a **K** value, the distance specified by **K**.
 - e) The workholding clamps and the program continues.
2. If the current bar is too short to correctly clamp and machine a new part (the bar is shorter than **MINIMUM CLAMPING LENGTH** plus **PART LENGTH + CUTOFF**):
 - a) If there is a **P** value in the G105 block, the control runs the subprogram.
 - b) The spindle stops.
 - c) The workholding unclamps, and the pushrod moves to the unloaded position.
 - d) If the G105 block has an **R** value, the spindle orients.
 - e) The Bar Feeder loads a new bar and pushes it the distance specified by **INITIAL PUSH LENGTH** (#3101) or, if the G105 block has an **I** value, the distance specified by **I**. If #3101 and **I** have values of zero, the Bar Feeder pushes the bar the distance specified by **REFERENCE POSITION** (#3112).
 - f) The workholding clamps.
 - g) If there is a **P** value in the G105 block, the control runs the subprogram.
 - h) The program continues.

Under some conditions, the system may stop at the end of the bar feed and display the message *Check Bar Position*. Make sure that the current bar position is correct, then press **[CYCLE START]** to start the program again.

G110/G111 Coordinate System #7/#8 (Group 12)

G110 selects #7 and G111 selects #8 additional work offset coordinates. All subsequent references to axes positions are interpreted in the new work offset coordinate system. Operation of G110 and G111 is the same as G154 P1 and G154 P2.

G112 XY to XC interpretation (Group 04)

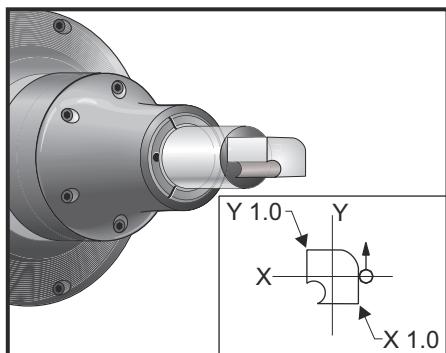
The G112 Cartesian to Polar coordinate transformation feature allows the user to program subsequent blocks in Cartesian XY coordinates, which the control automatically converts to polar XC coordinates. While it is active, the G17 XY plane is used for G01 linear XY strokes and G02 and G03 for circular motion. X, Y position commands are converted into rotary C-Axis and linear X-axis moves.

**NOTE:**

Mill-style Cutter Compensation becomes active when G112 is used. Cutter Compensation (G41, G42) must be canceled (G40) before exiting G112.

G112 Program Example

F6.60: G112 XY to XC Interpretation



G113 Cancel G112 (Group 04)

G113 cancels the Cartesian to Polar coordinate conversion.

G114 - G129 Coordinate System #9 - #24 (Group 12)

G114 - G129 codes are user-settable coordinate systems, #9 - #24, for work offsets. All subsequent references to axes' positions are interpreted in the new coordinate system. Work coordinate system offsets are entered from the **Active Work Offset** display page. Operation of G114 - G129 codes is the same as G154 P3 - G154 P18.

G154 Select Work Coordinates P1-99 (Group 12)

This feature provides 99 additional work offsets. G154 with a P value from 1 to 99 activates the 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 the 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 G114)
#14061- #14066 G154 P4 (also #7061-#7066 and G115)
#14081- #14086 G154 P5 (also #7081-#7086 and G116)
#14101- #14106 G154 P6 (also #7101-#7106 and G117)
#14121- #14126 G154 P7 (also #7121-#7126 and G118)
#14141- #14146 G154 P8 (also #7141-#7146 and G119)
#14161- #14166 G154 P9 (also #7161-#7166 and G120)
#14181- #14186 G154 P10 (also #7181-#7186 and G121)
#14201- #14206 G154 P11 (also #7201-#7206 and G122)
#14221- #14221 G154 P12 (also #7221-#7226 and G123)
#14241- #14246 G154 P13 (also #7241-#7246 and G124)
#14261- #14266 G154 P14 (also #7261-#7266 and G125)
#14281- #14286 G154 P15 (also #7281-#7286 and G126)
#14301- #14306 G154 P16 (also #7301-#7306 and G127)
#14321- #14326 G154 P17 (also #7321-#7326 and G128)
#14341- #14346 G154 P18 (also #7341-#7346 and G129)
#14361- #14366 G154 P19 (also #7361-#7366)
#14381- #14386 G154 P20 (also #7381-#7386)
#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
```

G159 Background Pickup / Part Return

Automatic Parts Loader (APL) Command. See the Haas APL manual.

G160 APL Axis Command Mode Only

Lathes with an Automatic Parts Loader use this command to inform the control that the subsequent axes commands are for the APL (not the lathe). See the Haas APL manual.

Lathes with Bar Feeders use this command to inform the control that the subsequent V-axis commands move the Bar Feeder V-axis, and are not interpreted as an incremental Y-axis move of the lathe turret. This command must be followed by a G161 command to cancel this mode.

Example:

```
G160 ;  
G00 V-10.0 ;  
G161 ;
```

The above example moves the Bar Feeder 10 units (in/mm) to the right of its home position. This command is sometimes used to position the Bar Feeder pushrod as a part stop.



NOTE:

Any Bar Feeder movements commanded this way are not used in bar length calculations by the control. If incremental bar feed movements are required, a G105 J1.0 command is more appropriate. See the Bar Feeder Manual for more information.

G161 APL Axis Command Mode Off

The G161 command turns off the G160 axis control mode and returns the lathe to normal operation. See the Haas APL manual.

G184 Reverse Tapping Canned Cycle For Left Hand Threads (Group 09)

F - Feed Rate in inches (mm) per minute

R - Position of the R plane

S - RPM, called prior to G184 is necessary

***W** - Z-axis incremental distance

***X** - X-axis motion command

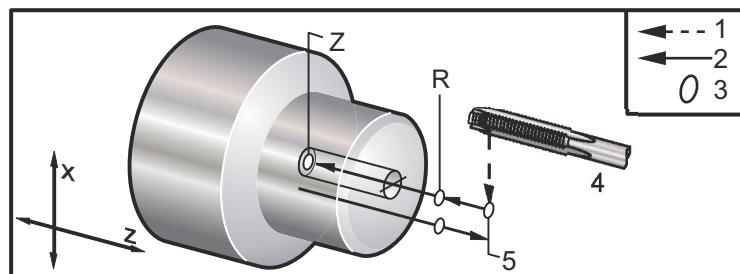
***Z** - Position of bottom of hole (optional)

* indicates optional

Programming Notes: When tapping, the feedrate is the lead of the thread. See example of G84, when programmed in G99 Feed per Revolution.

It is not necessary to start the spindle CCW before this canned cycle; the control does this automatically.

F6.61: G184 Reverse Tapping Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Left-hand tap, [5] Starting plane, [R] R plane, [Z] Position of the bottom of the hole.



G186 Reverse Live Tool Rigid Tap (For Left Hand Threads) (Group 09)

F - Feed Rate

C - C-Axis position

R - Position of the R plane

S - RPM, called prior to G186 is necessary

W - Z-axis incremental distance

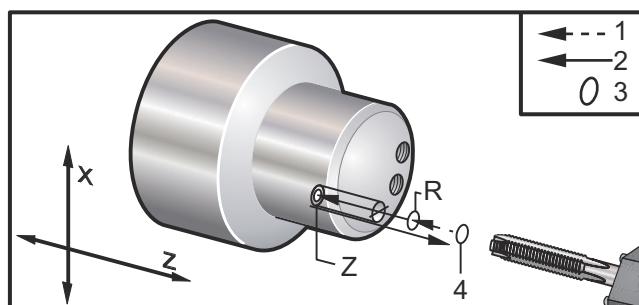
***X** - Part Diameter X-axis motion command

***Y** - Y-axis motion command

Z - Position of bottom of hole

* indicates optional

F6.62: G95, G186 Live Tooling Rigid Tapping: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Starting plane, [R] R plane, [Z] Position of the bottom of the hole.



It is not necessary to start the spindle CW before this canned cycle; the control does this automatically. See G84.

G187 Accuracy Control (Group 00)

Programming G187 is as follows:

```
G187 E0.01 (to set value) ;
G187 (to revert to setting 85 value) ;
```

G187 is used to select the accuracy with which corners are machined. The form for using G187 is G187 Ennnn, where nnnn is the desired accuracy.

G195/G196 Forward/Reverse Live Tool Radial Tapping (Diameter) (Group 00)

F - Feed Rate per revolution (G99)

U - X-Axis incremental distance

S - RPM, called prior to G195

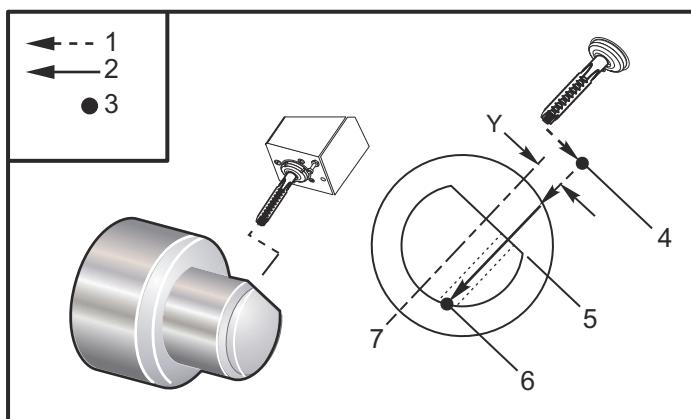
X - Position of the X Axis at the bottom of hole

Z - Z-Axis position before drilling

The tool must be positioned to the start point before commanding G195/G196. This G code is called for each hole being tapped. The cycle begins from the current position, tapping to the X-axis depth specified. An R plane is not used. Only X and F values should be used on G195/G196 lines. The tool must be positioned to the start point of any additional holes before commanding G195/G196 again.

S RPM should be called out as a positive number. It is not necessary to start the spindle in the correct direction; the control does this automatically.

F6.63: G195/G196 Live Tooling Rigid Tapping: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Starting point, [5] Part surface, [6] Bottom of the hole, [7] Centerline.



Program Example:

```
%  
O01950 (LIVE TAP - RADIAL) ;  
T101 ;  
M154 (Engage C-Axis) ;  
G00 G54 X6. C0. Y0. Z1. ;  
G00 X3.25 Z-0.75 C0. Y0. (Start Point) ;  
G99 (Must Set to Feed Per Rev. for this cycle) ;  
S500 ;  
G195 X2. F0.05 (Taps to X2., bottom of hole) ;
```

```
G00 C180. (Index C-Axis. New Start Point) ;
G195 X2. F0.05 ;
G00 C270. Y-1. Z-1. (Optional Y and Z-axis positioning, New
Start Point) ;
G195 X2. F0.05 ;
G00 G80 Z0.25 ;
M135 ;
M155 ;
G00 G28 H0. (Returns C-Axis to Home Position) ;
G00 X6. Y0. Z3. ;
G98 ;
M30 ;
%
```

G198 Disengage Synchronous Spindle Control (Group 00)

G198 disengages synchronous spindle control and allows independent control of the main spindle and the secondary spindle.

G199 Engage Synchronous Spindle Control (Group 00)

*R - Degrees, phase relationship of following spindle to commanded spindle

* indicates optional

This G code synchronizes the RPM of the two spindles. Position or speed commands to the following spindle, usually the secondary spindle, are ignored when spindles are in synchronous control. However, M codes on the two spindles are controlled independently.

The spindles remain synchronized until synchronous mode is disengaged using G198. This is the case even if power is cycled.

An R value on the G199 block positions the following spindle to a specified number of degrees, relative to the 0 mark on the commanded spindle. The following table includes examples of R values in G199 blocks:

```
G199 R0.0 (The following spindle's origin, 0-mark, matches the
commanded spindle's origin, 0-mark) ;
G199 R30.0 (The following spindle's origin, 0-mark, is
positioned +30 degrees from the commanded spindle's origin,
0-mark) ;
G199 R-30.0 (The following spindle's origin, 0-mark, is
positioned -30 degrees from the commanded spindle's origin,
0-mark) ;
```

When an R value is specified on the G199 block, the control first matches the velocity on the following spindle to that of the commanded spindle, then adjusts the orientation (R value in the G199 block). Once the specified R orientation is achieved the spindles are locked in synchronous mode until disengaged with a G198 command. This can also be achieved at zero RPM. Refer also to the G199 portion of the Synchronized Spindle Control Display on **262**.

Programming example of G199:

```
(Part cut off in synchronous spindle control) ;
G53 G00 X-1. Y0 Z-11. ;
T1010 ;
G54 ;
G00 X2.1 Z0.5 ;
G98 G01 Z-2.935 F60. (inches per minute) ;
M12 (Air blast on) ;
M110 (Secondary spindle chuck clamp) ;
M143 P500 (Secondary spindle to 500 RPM) ;
G97 M04 S500 (Main spindle to 500 RPM) ;
G99 ;
M111 (Secondary spindle chuck unclamp) ;
M13 (Air blast off) ;
M05 (main spindle off) ;
M145 (Secondary spindle off) ;
G199 (Synch spindles) ;
G00 B-28. (Rapid secondary spindle to face of part) ;
G04 P0.5 ;
G00 B-29.25 (Feed secondary spindle into part) ;
M110 (secondary spindle chuck clamp) ;
G04 P0.3 ;
M08 ;
G97 S500 M03 ;
G96 S400 ;
G01 X1.35 F0.0045 ;
X-.05 ;
G00 X2.1 M09 ;
G00 B-28.0 ;
G198 (Synch spindle off) ;
M05 ;
G00 G53 B-13.0 ;
G53 G00 X-1. Y0 Z-11. ;
M01 ;
(Secondary spindle) ;
(Finish face) ;
(G14 example) ;
N11 G55 G99 (G55 for secondary spindle work offset) ;
G00 G53 B-13.0 ;
G53 G00 X-1. Y0 Z-11. ;
```

```
G14 ;
T626 (Tool #6 Offset #26) ;
G50 S3000 ;
G97 S1300 M03 ;
G00 X2.1 Z0.5 ;
Z0.1 M08 ;
G96 S900 ;
G01 Z0 F0.01 ;
X-0.06 F0.005 ;
G00 X1.8 Z0.03 ;
G01 Z0.005 F0.01 ;
X1.8587 Z0 F0.005 ;
G03 X1.93 Z-0.0356 K-0.0356 ;
G01 X1.935 Z-0.35 ;
G00 X2.1 Z0.5 M09 ;
G97 S500 ;
G15 ;
G53 G00 X-1. Y0 Z-11. ;
M01 ;
```

G200 Index on the Fly (Group 00)

U - Optional relative move in X to tool change position

W - Optional relative move in Z to tool change position

X - Optional final X position

Z - Optional final Z position

T - Required tool number and offset number in standard form

G200 Index on the Fly causes the lathe to perform a move away, change tools, and move back to the part, to save time.



CAUTION:

The G200 does speed things up, but it also requires you to be more careful. Make sure you proof the program well, at 5% rapid, and be very cautious if you are starting from the middle of the program.

Normally, your tool change line consists of a few lines of code, like:

```
G53 G00 X0. (BRING TURRET TO SAFE X TC POS) ;
G53 G00 Z-10. (BRING TURRET TO SAFE Z TC POS) ;
T202 ;
```

Using G200, changes this code to:

```
G200 T202 U.5 W.5 X8. Z2. ;
```

If T101 just finished turning the O.D. of the part, you don't need to go back to a safe tool change position, when using a G200. Instead (as in the example) the moment the G200 line is called the turret:

1. Unclamps, in its current position.
2. Moves incrementally in the X and Z axes by the values stated in U and W (U .5 W .5)
3. Completes the tool change at this position.
4. Using the new tool and work offsets, it rapids to the XZ position called out on the G200 line (X8. Z2.).

This all happens very quickly, and nearly all at the same time, so try it out a few times, away from the chuck.

When the turret unclamps, it moves towards the spindle a tiny amount (perhaps .1-.2"), so you do not want the tool directly up against your jaws or collet when the G200 is commanded.

Because the U and W moves are incremental distances from where the tool is currently, if you hand jog away and start your program in a new position, the turret moves up and to the right of that new position. In other words, if you manually jogged back within .5" of your tailstock, and then commanded G200 T202 U.5 W1. X1. Z1., the turret would hit your tailstock - moving an incremental W1. (1" to right). For this reason, you may want to setup your Setting 93 and Setting 94, Tailstock Restricted Zone. Information on this can be found on **101**.

G211 Manual Tool Setting / G212 Auto Tool Setting

These two G codes are used in probing applications for both automatic and manual probes (SS and ST lathes only). For more information refer to "Automatic Tool Setting Probe" on **265**.

6

G241 Radial Drill Canned Cycle (Group 09)

C - C-Axis absolute motion command

F - Feed Rate

R - Position of the R plane (diameter)

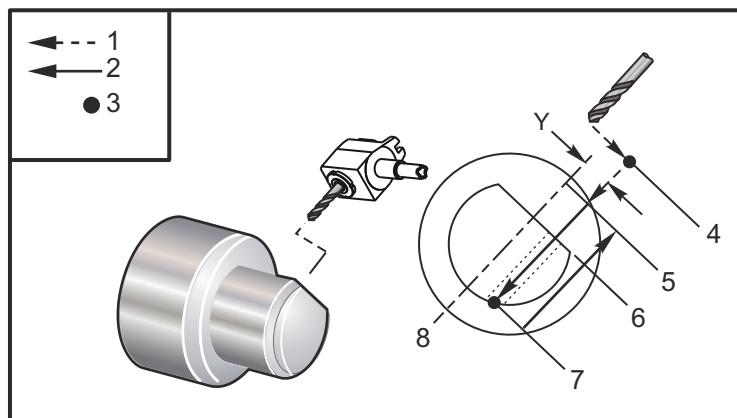
***X** - Position of bottom of hole (diameter)

***Y** - Y-axis absolute motion command

***Z** - Z-axis absolute motion command

* indicates optional

F6.64: G241 Radial Drill Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Starting point, [5] R plane, [6] Part surface, [Z] Bottom of the hole, [8] Centerline.



```
(G241 - RADIAL DRILLING) ;
G54 (Work Offset G54) ;
G00 G53 Y0 (Home Y-axis) ;
G00 G53 X0 Z-7. ;
T303 ;
M154 (Engage C Axis) ;
M133 P2500 (2500 RPM) ;
G98 (IPM) ;
G00 X5. Z-0.75 Y0 ;
G241 X2.1 Y0.125 Z-1.3 C35. R4. F20. (Drill to X 2.1) ;
X1.85 Y-0.255 Z-0.865 C-75. ;
G00 G80 Z1. ;
M135 (Stop live tool spindle) ;
G00 G53 X0. Y0. ;
G00 G53 X0 Z-7. ;
M00 ;
```

G242 Radial Spot Drill Canned Cycle (Group 09)

C - C-Axis absolute motion command

F - Feed Rate

P - The dwell time at the bottom of the hole

R - Position of the R plane (Diameter)

***X** - Position of bottom of hole (Diameter)

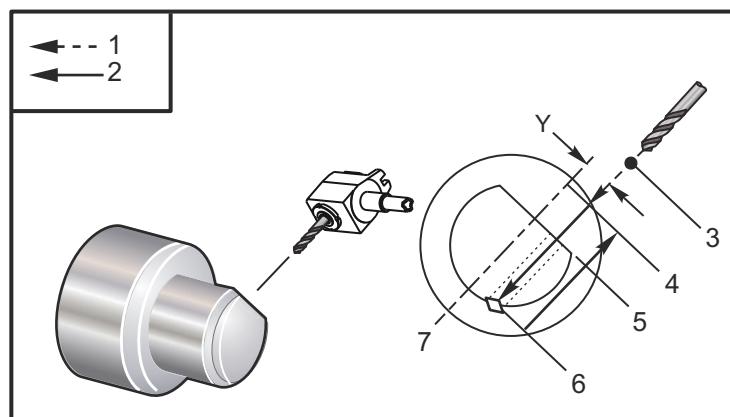
***Y** - Y-axis motion command

***Z** - Z-axis motion command

* indicates optional

This G code is modal. It remains active until it is canceled (G80) or another canned cycle is selected. Once activated, every motion of **Y** and/or **Z** executes this canned cycle.

F6.65: G242 Radial Spot Drill Canned Cycle: [1] Rapid, [2] Feed, [3] Starting point, [4] R plane, [5] Part surface, [6] Dwell at the bottom of the hole, [7] Centerline.



Program Example:

```
(G242 - RADIAL SPOT DRILLING) ;
G54 (Work offset G54) ;
G00 G53 Y0Home Y-axis) ;
G00 G53 X0 Z-7. ;
T303 ;
M154 (Engage C Axis) ;
M133 P2500(2500 RPM) ;
G98 (IPM) ;
G00 X5. Z-0.75 Y0 ;
G242 X2.1 Y0.125 Z-1.3 C35. R4. P0.5 F20. (Drill to X 2.1) ;
X1.85 Y-0.255 Z-0.865 C-75. P0.7 ;
G00 G80 Z1. ;
M135 (Stop live tool spindle) ;
G00 G53 X0. Y0. ;
G00 G53 X0 Z-7. ;
M00 ;
```

G243 Radial Normal Peck Drilling Canned Cycle (Group 09)

C - C-Axis absolute motion command

F - Feed Rate

***I** - Size of first cutting depth

***J** - Amount to reduce cutting depth each pass

***K** - Minimum depth of cut

***P** - The dwell time at the bottom of the hole

***Q** - The cut-in value, always incremental

R - Position of the R plane (Diameter)

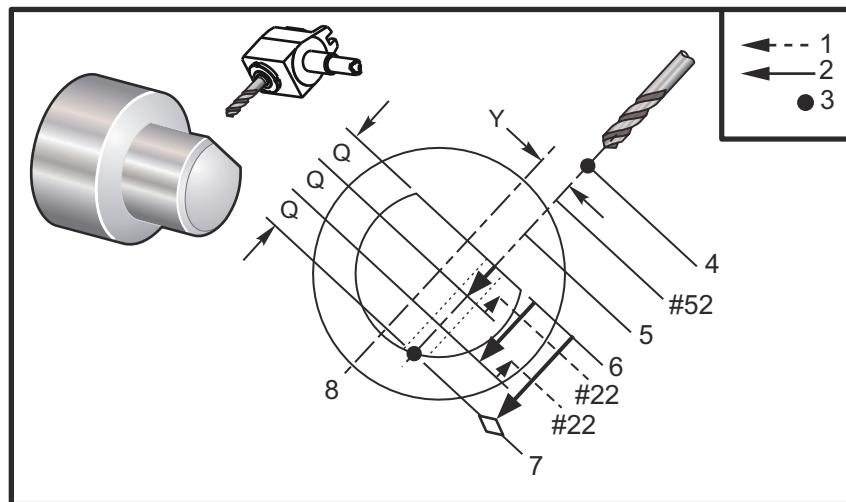
***X** - Position of bottom of hole (Diameter)

***Y** - Y-axis absolute motion command

***Z** - Z-axis absolute motion command

* indicates optional

F6.66: G243 Radial Normal Peck Drilling Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] R plane, [#52] Setting 52, [5] R plane, [6] Part surface, [#22] Setting 22, [7] Dwell at the bottom of the hole, [8] Centerline.



Programming Notes: If **I**, **J**, and **K** are specified, a different operating mode is selected. The first pass will cut in the value 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**.

Setting 52 changes the way G243 works when it returns to the R plane. Usually the R plane is set well outside the cut to insure that the chip clearing motion allows the chips to clear the hole. However, this is wasted motion when first drilling through this empty space. If Setting 52 is set to the distance required to clear chips, the R plane can be put much closer to the part being drilled. When the clear move to R occurs, the Z will be moved past R by this value in setting 52. Setting 22 is the amount to feed in X to get back the same point at which the retraction occurred.

Program Example:

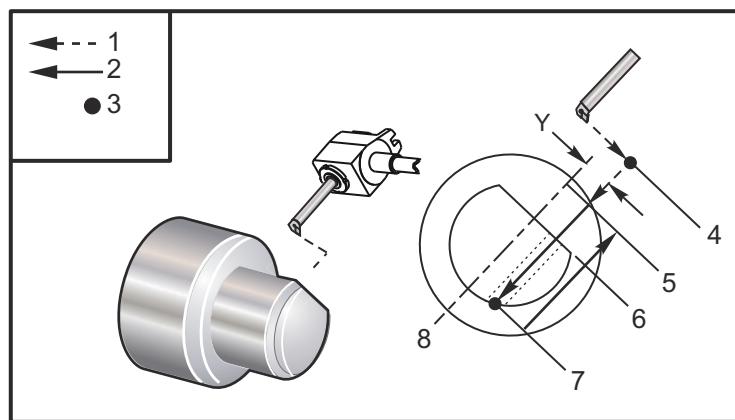
```
(G243 - RADIAL PECK DRILLING USING Q) ;
G54 (Work offset G54) ;
G00 G53 Y0 (Home Y-axis) ;
G00 G53 X0 Z-7. ;
T303 ;
M154 (Engage C Axis) ;
M133 P2500 (2500 RPM) ;
G98 (IPM) ;
G00 X5. Z-0.75 Y0 ;
G243 X2.1 Y0.125 Z-1.3 C35. R4. Q0.25 F20. (Drill to X 2.1) ;
X1.85 Y-0.255 Z-0.865 C-75. Q0.25 ;
G00 G80 Z1. ;
M135 (Stop live tool spindle) ;
G00 G53 X0. Y0. ;
G00 G53 X0 Z-7. ;
M00 ;
(G243 - RADIAL WITH I,J,K PECK DRILLING) ;
G54 (Work offset G54) ;
G00 G53 Y0 (Home Y-axis) ;
G00 G53 X0 Z-7 ;
T303 ;
M154 (Engage C Axis) ;
M133 P2500 (2500 RPM) ;
G98 (IPM) ;
G00 X5. Z-0.75 Y0 ;
G243 X2.1 Y0.125 Z-1.3 I0.25 J0.05 K0.1 C35. R4. F5. (Drill
to X 2.1) ;
X1.85 Y-0.255 Z-0.865 I0.25 J0.05 K0.1 C-75. ;
G00 G80 Z1. ;
M135 ;
G00 G53 X0. Y0. ;
G00 G53 Z-7. ;
M00 ;
```

G245 Radial Boring Canned Cycle (Group 09)

C - C-Axis absolute motion command
F - Feed Rate
R - Position of the **R** plane (Diameter)
***X** - Position of bottom of hole (Diameter)
***Y** - Y-axis absolute motion command
***Z** - Z-axis absolute motion command

* indicates optional

F6.67: G245 Radial Boring Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Starting point, [5] R plane, [6] Part surface, [7] Bottom of the hole, [8] Centerline.



6

```

(G245 - RADIAL BORING) ;
G54 (Work offset G54) ;
G00 G53 Y0 (Home Y-axis) ;
G00 G53 X0 Z-7. ;
T303 ;
M154 (Engage C Axis) ;
M133 P2500 (2500 RPM) ;
G98 (IPM) ;
G00 X5. Z-0.75 Y0 ;
G245 X2.1 Y0.125 Z-1.3 C35. R4. F20. (Drill to X 2.1) ;
X1.85 Y-0.255 Z-0.865 C-75. ;
G00 G80 Z1. ;
M135 (Stop live tool spindle) ;
G00 G53 X0. Y0. ;
G00 G53 X0 Z-7. ;
M30 ;
  
```

G246 Radial Bore and Stop Canned Cycle (Group 09)

C - C-Axis absolute motion command

F - Feed Rate

R - Position of the R plane (Diameter)

***X** - Position of bottom of hole (Diameter)

***Y** - Y-axis absolute motion command

***Z** - Z-axis absolute motion command

*indicates optional

This G code stops the spindle once the tool reaches the bottom of the hole. The tool is retracted once the spindle has stopped.

Example:

```
(G246 - RADIAL BORING) ;
G54 (Work offset G54) ;
G00 G53 Y0 (Home Y-axis) ;
G00 G53 X0 (Home X-axis) ;
G00 G53 X0 Z-7. ;
T303 ;
M154 (Engage C Axis) ;
M133 P2500 (2500 RPM) ;
G98 (IPM) ;
G00 X5. Z-0.75 Y0 ;
G246 X2.1 Y0.125 Z-1.3 C35. R4. F20. (Bore to X 2.1) ;
X1.85 Y-0.255 Z-0.865 C-75. ;
G00 G80 Z1. ;
M135 (Stop live tool spindle) ;
G00 G53 X0. Y0. ;
G00 G53 X0 Z-7. ;
M30 ;
```

G247 Radial Bore and Manual Retract Canned Cycle (Group 09)

C - C-Axis absolute motion command

F - Feed Rate

R - Position of the R plane (Diameter)

***X** - Position of bottom of hole (Diameter)

***Y** - Y-axis absolute motion command

***Z** - Z-axis absolute motion command

* indicates optional

This G code stops the spindle at the bottom of the hole. At this point the tool is manually jogged out of the hole. The program continues when **[CYCLE START]** is pressed.

Example:

```
(G247 - RADIAL BORING) ;
G54 (Work offset G54) ;
G00 G53 Y0 (Home Y-axis) ;
G00 G53 X0 (Home X-axis) ;
G00 G53 X0 Z-7. ;
T303 ;
M154 (Engage C Axis) ;
M133 P2500 (2500 RPM) ;
G98 (IPM) ;
G00 X5. Z-0.75 Y0 ;
G247 X2.1 Y0.125 Z-1.3 C35. R4. F20. (Bore to X 2.1) ;
X1.85 Y-0.255 Z-0.865 C-75. ;
G00 G80 Z1. ;
M135 (Stop live tool spindle) ;
G00 G53 X0. Y0. ;
G00 G53 X0 Z-7. ;
M30 ;
```

G248 Radial Bore and Dwell and Manual Retract Canned Cycle (Group 09)

C - C-Axis absolute motion command

F - Feed Rate

P - The dwell time at the bottom of the hole

R - Position of the R plane (Diameter)

***X** - Position of bottom of hole (Diameter)

***Y** - Y-axis absolute motion command

***Z** - Z-axis absolute motion command

* indicates optional

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 continues when **[CYCLE START]** is pressed.

Example:

```
(G248 - RADIAL BORING) ;
G54 (Work offset G54) ;
G00 G53 Y0 (Home Y-axis) ;
G00 G53 X0 (Home X-axis) ;
G00 G53 X0 Z-7. ;
```

```
T303 ;  
M154 (Engage C Axis) ;  
M133 P2500 (2500 RPM) ;  
G98 (IPM) ;  
G00 X5. Z-0.75 Y0 ;  
G248 X2.1 Y0.125 Z-1.3 C35. R4. P1. F20. (Bore to X 2.1) ;  
X1.85 Y-0.255 Z-0.865 C-75. ;  
G00 G80 Z1. ;  
M135 (Stop live tool spindle) ;  
G00 G53 X0. Y0. ;  
G00 G53 X0 Z-7. ;  
M30 ;
```

G249 Radial Bore and Dwell Canned Cycle (Group 09)

C - C-Axis absolute motion command

F - Feed Rate

P - The dwell time at the bottom of the hole

R - Position of the R plane

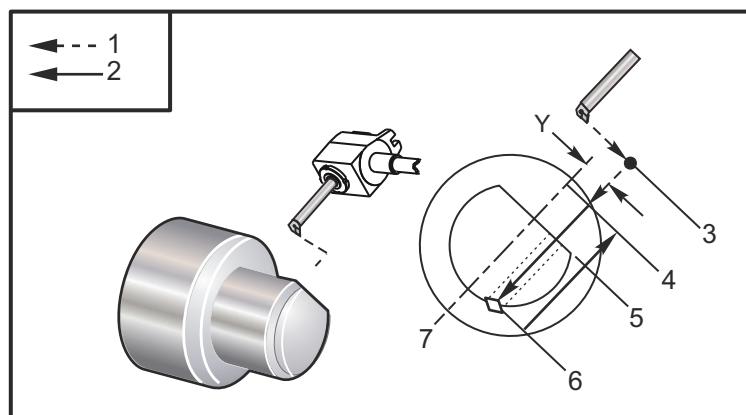
***X** - Position of bottom of hole

***Y** - Y-axis motion command

***Z** - Z-axis motion command

* indicates optional

F6.68: G249 Radial Bore and Dwell Canned Cycle: [1] Rapid, [2] Feed, [3] Starting point, [4] R plane, [5] Part surface, [6] Dwell at the bottom of the hole, [7] Centerline.



```
(G249 - RADIAL BORING & DWELL) ;  
G54 ;  
G00 G53 Y0 ;
```

```
G00 G53 X0 Z-7. ;
T303 ;
M154 (Engage C Axis) ;
M133 P2500 ;
G98 ;
G00 X5. Z-0.75 Y0 ;
G249 X2.1 Y0.125 Z-1.3 C35. P1.35 R4. F20. ;
X1.85 Y-0.255 Z-0.865 C-75. P1.65 ;
G00 G80 Z1. ;
M135 ;
G00 G53 X0. Y0. ;
G00 G53 X0 Z-7. ;
M30 ;
```

6.2 M-codes

Code	Description
M00	Stop Program
M01	Stop Program
M02	Program End
M03	Spindle On Fwd
M04	Spindle On Rev
M05	Spindle Stop
M08	Coolant On
M09	Coolant Off
M10	Chuck Clamp
M11	Chuck Unclamp
M12	Auto Jet Air Blast On (Optional)
M13	Auto Jet Air Blast Off (Optional)
M14	Main Spindle Brake On (Optional C-Axis)
M15	Main Spindle Brake Off (Optional C-Axis)
M17	Turret Rotation Fwd
M18	Turret Rotation Rev
M19	Orient Spindle (Optional)
M21	Tailstock Advance (Optional)
M22	Tailstock Retract (Optional)
M23	Chamfer Out of Thread On
M24	Chamfer Out of Thread Off

Code	Description
M30	End of Program and Reset
M31	Chip Auger Forward (Optional)
M33	Chip Auger Stop (Optional)
M36	Parts Catcher On (Optional)
M37	Parts Catcher Off (Optional)
M38	Spindle Speed Variation On
M39	Spindle Speed Variation Off
M41	Low Gear (Optional)
M42	High Gear (Optional)
M43	Turret Unlock (Service Use Only)
M44	Turret Lock (Service Use Only)
M51	User M Turn On (Optional)
M52	User M Turn On (Optional)
M53	User M Turn On (Optional)
M54	User M Turn On (Optional)
M55	User M Turn On (Optional)
M56	User M Turn On (Optional)
M57	User M Turn On (Optional)
M58	User M Turn On (Optional)
M59	Set Output Relay
M61	User M Turn Off (Optional)
M62	User M Turn Off (Optional)
M63	User M Turn Off (Optional)

Code	Description
M64	User M Turn Off (Optional)
M65	User M Turn Off (Optional)
M66	User M Turn Off (Optional)
M67	User M Turn Off (Optional)
M68	User M Turn Off (Optional)
M69	Clear Output Relay
M76	Display Disable
M77	Display Enable
M78	Alarm if Skip Signal Found
M79	Alarm if Skip Signal Not Found
M85	Automatic Door Open (Optional)
M86	Automatic Door Close (Optional)
M88	High Pressure Coolant On (Optional)
M89	High Pressure Coolant Off (Optional)
M95	Sleep Mode
M96	Jump If No Signal
M97	Local Subprogram Call
M98	Subprogram Call
M99	Subprogram Return Or Loop
M104	Probe Arm Extend (Optional)
M105	Probe Arm Retract (Optional)
M109	Interactive User Input
M110	Secondary Spindle Chuck Clamp (Optional)

Code	Description
M111	Secondary Spindle Chuck Unclamp (Optional)
M112	Secondary Spindle Air Blast On (Optional)
M113	Secondary Spindle Air Blast Off (Optional)
M114	Secondary Spindle Brake On (Optional)
M115	Secondary Spindle Brake Off (Optional)
M119	Secondary Spindle Orient (Optional)
M121	User M-codes (Optional)
M122	User M-codes (Optional)
M123	User M-codes (Optional)
M124	User M-codes (Optional)
M125	User M-codes (Optional)
M126	User M-codes (Optional)
M127	User M-codes (Optional)
M128	User M-codes (Optional)
M133	Live Tool Fwd (Optional)
M134	Live Tool Rev (Optional)
M135	Live Tool Stop (Optional)
M143	Secondary Spindle Forward (Optional)
M144	Secondary Spindle Reverse (Optional)
M145	Secondary Spindle Stop (Optional)
M154	C-Axis Engage (Optional)
M155	C-Axis Disengage (Optional)

Introduction to M-codes

M-codes are non-axis moving commands for the machine. The format for an M-code is the letter **M** followed by two to three numbers, for example **M03**.

Only one M-code may be programmed per line of code. All M-codes take effect at the end of the block.

M00 Stop Program

M00 stops a program. It stops the axes, spindle, turns off the coolant (including optional High Pressure Coolant). The next block (after **M00**) is highlighted when viewed in the program editor. Pressing **[CYCLE START]** continues program operation from the highlighted block.

M01 Program Stop

M01 works the same as **M00**, except the Optional Stop feature must be **ON**.

M02 Program End

M02 ends a program.



NOTE:

*The most common way of ending a program is with an **M30**.*

M03/M04/M05 Spindle On Fwd/On Rev/Stop

M03 turns spindle on in the forward direction. **M04** turns spindle on in the reverse direction. **M05** stops the spindle. For spindle speed, refer to G96/G97/G50.

M08/M09 Coolant On/Off

M08 turns on the optional coolant supply and **M09** turns it off. For High Pressure Coolant, see **M88/M89**.

M10/M11 Chuck Clamp/Unclamp

M10 clamps the chuck and M11 unclamps it. The direction of clamping is controlled by Setting 92 (refer to page 408 for more information).

M12/M13 Auto Jet Air Blast On/Off (Optional)

M12 and M13 activate the optional Auto Air Jet. M12 turns the air blast on and M13 turns the air blast off. Additionally, M12 Pnnn (nnn is in milliseconds) turns it on for the specified time, then turns off automatically. For sub spindle refer to M112/M113.

M14/M15 Main Spindle Brake On/Off (Optional C-Axis)

These M Codes are used for machines equipped with the optional C-Axis. M14 applies a caliper-style brake to hold the main spindle, while M15 releases the brake.

M17/M18 Turret Rotation Fwd/Rev

M17 and M18 rotate the turret in the forward (M17) or reverse (M18) direction when a tool change is made. The following M17 program code causes the tool turret to move forward to tool 1 or reverse to tool 1 if an M18 is commanded.

```
N1 T0101 M17 (Forward) ;
N1 T0101 M18 (Reverse) ;
```

An M17 or M18 stays in effect for the remainder of the program.



NOTE:

Setting 97, Tool Change Direction, must be set to M17/M18.

6

M19 Orient Spindle (Optional)

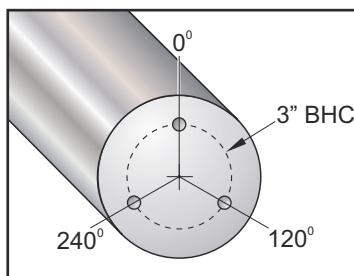
M19 adjusts the spindle to a fixed position. The spindle only orients to the zero position without the optional M19 orient spindle feature.

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.

Spindle orientation is dependent on the mass, diameter, and length of the workpiece and/or the workholding (chuck). Contact the Haas Applications Department if any unusually heavy, large diameter, or long configuration is used.

M19 Programming Example

F6.69: M19 Orient Spindle Bolt Hole Circle Example: 3 holes at 120 degrees on 3" BHC.



```
%  
O0050 ;  
T101 ;  
G54 ;  
G00 X3.0 Z0.1 ;  
G98 (Feed per minute) ;  
M19 P0 (Orient spindle) ;  
M14 (Turn on main spindle brake) ;  
M133 P2000 (Turn on live tool forward) ;  
G01 Z-0.5 F40.0 ;  
G00 Z0.1 ;  
M19 P120 (Orient spindle) ;  
M14 (Turn on main spindle brake) ;  
G01 Z-0.5 ;  
G00 Z0.1 ;  
M19 P240 (Orient spindle) ;  
M14 (Turn on main spindle brake) ;  
G01 Z-0.5 ;  
G00 Z0.1 ;  
M15 (Turn off main spindle brake) ;
```

M21/M22 Tailstock Advance/Retract (Optional)

M21 and M22 position the tailstock. M21 uses Settings 106 and 107 to move to the Tailstock Hold Point. M22 uses Setting 105 to move the tailstock to the Retract Point.



NOTE:

ST10 does not use any settings (105, 106, 107).

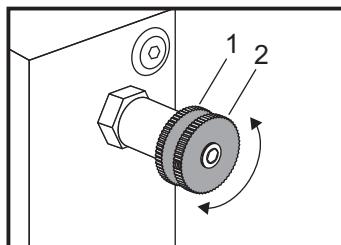
Adjust pressure using the valves on the HPU (except ST-40, which uses Setting 241 to define hold pressure). For the ST Pressure Charts, refer to pages **95** and **96**.



CAUTION:

Do not use an M21 in the program if the tailstock is positioned manually. If this is done, the tailstock backs away from the workpiece and then repositions against the workpiece, which may cause the workpiece to drop.

F6.70: Set Screw Hold Pressure Valve: [1] Locking knob, [2] Adjustment knob.



6

M23/M24 Chamfer Out of Thread On/Off

M23 commands the control to execute a chamfer at the end of a thread executed by G76 or G92. M24 commands the control not to perform chamfering at the end of the threading cycles (G76 or G92). An M23 remains in effect until changed by M24, likewise for M24. Refer to Settings 95 and 96 to control the chamfer size and angle. M23 is the default at power-up and when the control is reset.

M30 End of Program and Reset

M30 stops a program. It stops the spindle and turns off the coolant and the program cursor returns to the start of the program. M30 cancels tool offsets.

M31/M33 Chip Auger Forward/Stop (Optional)

M31 starts the optional chip auger motor in the forward direction (the direction that moves the chips out of the machine). The auger does not turn if the door is open. It is recommended that the chip auger be used intermittently. Continuous operation causes the motor to overheat. Settings 114 and 115 control the auger duty cycle times.

M33 stops auger motion.

M36/M37 Parts Catcher On/Off (Optional)

M36 rotates the parts catcher into position to catch a part. M37 rotates the parts catcher out of the work envelope.

M38/M39 Spindle Speed Variation On/Off

Spindle Speed Variation (SSV) allows the operator to specify a range within which the spindle speed continuously varies. This is helpful in suppressing tool chatter, which can lead to an undesirable part finish and/or damage to the cutting tool. The control varies the spindle speed based on Settings 165 and 166. For example, in order to vary spindle speed +/- 50 RPM from its current commanded speed with a duty cycle of 3 seconds, set Setting 165 to 50 and Setting 166 to 30. Using these settings, the following program varies the spindle speed between 950 and 1050 RPM after the M38 command.

M38/39 Program Example

```
00010 ;
S1000 M3 ;
G4 P3. ;
M38 (SSV ON) ;
G4 P60. ;
M39 (SSV OFF) ;
G4 P5. ;
M30 ;
```

The spindle speed continuously varies with a duty cycle of 3 seconds until an M39 command is found. At that point the machine comes back to its commanded speed and the SSV mode is turned off.

A program stop command such as M30 or pressing [RESET] also turns SSV Off. If the RPM swing is larger than the commanded speed value, any negative RPM values (below zero) translates into an equivalent positive value. The spindle, however, is not allowed to go below 10 RPM when SSV mode is active.

Constant Surface Speed: When Constant Surface Speed (G96) is activated (which calculates spindle speed) the M38 command alters that value using Settings 165 and 166.

Threading Operations: G92, G76 and G32 allow the spindle speed to vary in SSV mode. This is not recommended due to possible thread lead errors caused by mismatched acceleration of the spindle and the Z-axis.

Tapping cycles: G84, G184, G194, G195, and G196 are executed at their commanded speed and SSV is not applied.

M41/M42 Low/High Gear (Optional)

On machines with a transmission, M41 selects low gear and M42 selects high gear.

M43/M44 Turret Unlock/Lock (Service Use Only)

For Service use only.

M51-M58 User M Turn On (Optional)

The M51 through M58 codes are optional for user interfaces. They activate one of the relays and leave it active. Use M61-M68 to turn these off. [RESET] turns off all of these relays. See M121-M128 for details on the M-code relays.

M59 Set Output Relay

This M code turns on a relay. An example of its use is M59 Pnn, where nn is the number of the relay being turned on. An M59 command is used to turn on any of the discrete output relays in the range from 1100 to 1155. When using Macros, M59 P1103 does the same thing as using the optional macro command #1103 = 1, except that it is processed in the same order as axis motion.



NOTE:

The 8 spare M functions use addresses 1140-1147.

6

M61-M68 User M Turn Off (Optional)

The M61 through M68 codes are optional for user interfaces. They turn off one of the relays. Use M51-M58 to turn these on. [RESET] turns off all of these relays. See M121-M128 for details on the M-code relays.

M69 Clear Output Relay

This M code turns off a relay. An example of its usage is M69 Pnn, where nn is the number of the relay being turned off. An M69 command can 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 axis motion lines.

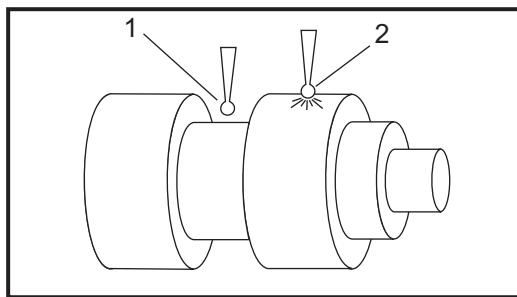
M76/M77 Display Disable/Enable

M76 and M77 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/M79 Alarm if Skip Signal Found/Not Found

This M-code is used with a probe. M78 generates an alarm if a programmed skip function (G31) receives a signal from the probe. This is used when a skip signal is not expected, and may indicate a probe crash. M79 generates an alarm if a programmed skip function (G31) did not receive a signal from the probe. This is used when the lack of the skip signal means a probe positioning error. These codes can be placed on the same line as the skip G-code or in any block after.

F6.71: M78/M79 Alarm if Skip Signal Found/Not Found: [1] Signal not found, [2] Signal found.



M85/M86 Automatic Door Open/Close (Optional)

M85 opens the Auto Door and M86 closes it. The control pendant beeps when the door is in motion.

M88/M89 High Pressure Coolant On/Off (Optional)

M88 turns on the high pressure coolant option, and M89 turns the coolant off. Use M89 to turn off High Pressure coolant during program execution before rotating the tool turret.



WARNING: *Turn off High Pressure Coolant before performing a tool change.*

M93/M94 Start/Stop Axis Pos Capture

These M codes permit the control to capture the position of an auxiliary Axis when a discrete input changes to a 1. The format is M93 Pnn Qmm. nn is the axis number. mm is a discrete input number from 0 to 63.

M93 causes the control to watch the discrete input specified by the Q value, and when it goes to a 1, it captures the position of the axis specified by the P value. The position is then copied to hidden macro variable 749. M94 stops the capture. M93 and M94 were introduced to support the Haas Bar Feeder, which uses a single axis controller to the V auxiliary Axis. P5 (V axis) and Q2 must be used for the bar feeder.

M95 Sleep Mode

Sleep mode is a long dwell. Sleep mode can be used when the user wants the machine to begin warming itself. So it is ready to use when the operators arrives. The format of the M95 command is: M95 (hh:mm).

The comment immediately following M95 must contain the hours and minutes for which the machine sleeps. For example, if the current time were 6pm and the user wanted the machine to sleep until 6:30am the next day, the following command would be used; M95 (12:30). The line(s) following M95 should be axis moves and spindle warm-up commands.

M96 Jump If No Signal

P - Program block to go to when conditional test is met

Q - Discrete input variable to test (0 to 63)

This code tests 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 on the diagnostic display (The upper left input is 0 and the lower right is input 63. When this program block is executed and the input signal specified by **Q** has a value of 0, the program block **Pnnnn** is performed (the **Pnnnn** line must be in the same program)).

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

This code calls a subprogram (subroutine) referenced by a line number (**N**) within the same program. A **Pnn** code is required and must match a line number within the same program. This is useful for subroutines within a program as it does not require a separate program. The subroutine must end with an **M99**. An **Lnn** code in the **M97** block will repeat the subroutine call **nn** times.

Example:

```
%  
000055 (M97 LOCAL SUBPROGRAM CALL);  
M97 P1000 L2 (L2 COMMAND WILL RUN THE N1000 LINE TWICE);  
;  
M30;  
N1000 T101 (N LINE THAT WILL RUN AFTER M97 P1000 IS RUN);  
G00 G18 G20 G40 G54 G80 G99;  
S2000 G50;  
S500 G97 M03;  
G00 X2. Z-2. M08 S900 G96;  
G01 X0 F0.01;  
Z-1.9;  
G00 X2.1 M09;
```

```
G53 X0 Z0;
M99;
%
```

M98 Subprogram Call

This code 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 contain `M99` to return to the main program. An `Lnn` count can be put on the line containing `M98` causing the subprogram to be called `nn` times before continuing to the next block.

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.

Example: This is the program that calls the subprogram to loop (4) times.

```
%  
O00056 (MAIN PROGRAM NUMBER);  
(M98 LOCAL SUBPROGRAM CALL);  
M98 P1000 L4 (CALL SUBPROGRAM, (NUMBER 1000), LOOP 4 TIMES);  
M30 (END PROGRAM);  
%
```

This is the subprogram itself.

```
%  
O1000 (SUBPROGRAM NUMBER)  
T101;  
G00 G18 G20 G40 G54 G80 G99;  
S2000 G50;  
S500 G97 M03;  
G00 X2. Z-2. M08 S900 G96;  
G01 X0 F0.01;  
Z-1.9;  
G00 X2.1 M09;  
G53 X0 Z0;  
M99;  
%
```

M99 Subprogram Return or Loop

This code has three main uses:

1. An M99 is used at the end of a subprogram, local subprogram, or macro to return back to the main program.
2. An M99 Pnn jumps the program to the corresponding Nnn in the program.
3. An M99 in the main program causes the program to loop back to the beginning and run until [RESET] is pressed.

Programming Notes - You can simulate Fanuc behavior by using the following code:

	Haas	Fanuc
Calling program:	O0001	O0001

	N50 M98 P2	N50 M98 P2
	N51 M99 P100	...
	...	N100 (continue here)
	N100 (continue here)	...
	...	M30
	M30	
Subprogram:	O0002	O0002
	M99	M99 P100

M99 With Macros - If the machine is equipped with the optional macros, you can use a global variable and specify a block to jump to by adding #nnn = dddd in the subroutine and then using M99 P#nnn after the subroutine call.

M104/M105 Probe Arm Extend/Retract (Optional)

The optional tool setting probe arm is extended and retracted using these M-codes.

M109 Interactive User Input

This M code allows a G-code program to place a short prompt (message) on the screen. A macro variable in the range 500 through 599 must be specified by a P code. The program can check for any character that can be entered from the keyboard by comparing with the decimal equivalent of the ASCII character (G47, Text Engraving, has a list of ASCII characters).

The following sample program asks the user a Yes or No question, then wait for either a Y or an N to be entered. All other characters are ignored.

```
N1 #501= 0. (Clear the variable) ;
N5 M109 P501 (Sleep 1 min?) ;
IF [ #501 EQ 0. ] GOTO5 (Wait for a key) ;
IF [ #501 EQ 89. ] GOTO10 (Y) ;
IF [ #501 EQ 78. ] GOTO20 (N) ;
GOTO1 (Keep checking) ;
N10 (A Y was entered) ;
M95 (00:01) ;
GOTO30 ;
N20 (An N was entered) ;
G04 P1. (Do nothing for 1 second) ;
N30 (Stop) ;
M30 ;
```

The following sample program 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.

```
% 
O01234 (M109 Program) ;
N1 #501= 0 (Clear Variable #501) ;
(Variable #501 will be checked) ;
(Operator enters one of the following selections) ;
N5 M109 P501 (1,2,3,4,5) ;
IF [ #501 EQ 0 ] GOTO5 ;
(Wait for keyboard entry loop until entry) ;
(Decimal equivalent from 49-53 represent 1-5) ;
IF [ #501 EQ 49 ] GOTO10 (1 was entered go to N10) ;
IF [ #501 EQ 50 ] GOTO20 (2 was entered go to N20) ;
IF [ #501 EQ 51 ] GOTO30 (3 was entered go to N30) ;
IF [ #501 EQ 52 ] GOTO40 (4 was entered go to N40) ;
IF [ #501 EQ 53 ] GOTO50 (5 was entered go to N50) ;
GOTO1 (Keep checking for user input loop until found) ;
N10 ;
(If 1 was entered run this sub-routine) ;
(Go to sleep for 10 minutes) ;
#3006= 25 (Cycle start sleeps for 10 minutes) ;
```

```
M95 (00:10) ;
GOTO100 ;
N20 ;
(If 2 was entered run this sub routine) ;
(Programmed message) ;
#3006= 25 (Programmed message cycle start) ;
GOTO100 ;
N30 ;
(If 3 was entered run this sub routine) ;
(Run sub program 20) ;
#3006= 25 (Cycle start program 20 will run) ;
G65 P20 (Call sub-program 20) ;
GOTO100 ;
N40 ;
(If 4 was entered run this sub routine) ;
(Run sub program 22) ;
#3006= 25 (Cycle start program 22 will be run) ;
M98 P22 (Call sub program 22) ;
GOTO100 ;
N50 ;
(If 5 was entered run this sub-routine) ;
(Programmed message) ;
#3006= 25 (Reset or cycle start will turn power off) ;
#1106= 1 ;
N100 ;
M30 ;
%
```

M110/M111 Secondary Spindle Chuck Clamp/Unclamp (Optional)

These M codes will clamp and unclamp the secondary spindle chuck. OD / ID clamping is set with Setting 122.

M112/M113 Secondary Spindle Air Blast On/Off (Optional)

M112 turns on the secondary spindle air blast. M113 turns the secondary spindle air blast off.

M114/M115 Secondary Spindle Brake On/Off (Optional)

M114 applies a caliper-style brake to hold the secondary spindle, while M115 releases the brake.

M119 Secondary Spindle Orient (Optional)

This command orients the secondary spindle (DS lathes) to the zero position. A P or R value is added to position the spindle to a specific position. A P value positions the spindle to that whole degree (e.g. P120 is 120°). An R value positions the spindle to a fraction of a degree (e.g. R12.25 is 12.25°). The format is: M119 Pxxx/M119 Rxx.x. The spindle angle is viewed in the Current Commands Tool Load screen.

M121-M128 Optional User M-codes (Optional)

The M121 through M128 codes are optional for user interfaces. They activate relays 1132 through 1139, wait for the M-fin signal, release the relay, and wait for the M-fin signal to cease. [RESET] terminates any operation that is hung-up waiting for M-fin.

M133/M134/M135 Live Tool Fwd/Rev/Stop (Optional)

M133 turns the live tool spindle in the forward direction. M134 turns the live tool spindle in the reverse direction. M135 stops the live tool spindle.

Spindle speed is controlled with a P address code. For example, P1200 would command a spindle speed of 1200 RPM.

M143/M144/M145 Secondary Spindle Fwd/Rev/Stop (Optional)

M143 turns the secondary spindle in the forward direction. M144 turns the secondary spindle in the reverse direction. M145 stops the secondary spindle

The subspindle speed is controlled with an P address code, for example, P1200 commands a spindle speed of 1200 RPM.

M154/M155 C-Axis Engage/Disengage (Optional)

This M code is used to engage or disengage the optional C-Axis motor.

6.3 Settings

Code	Name	Code	Name
1	Auto Power Off Timer	118	M99 Bumps M30 CNTRS
2	Power Off at M30	119	Offset Lock
3	3D Graphics	120	Macro Var Lock
4	Graphics Rapid Path	121	Foot Pedal TS Alarm
5	Graphics Drill Point	122	Secondary Spindle Chuck Clamping
6	Front Panel Lock	131	Auto Door
7	Parameter Lock	132	Jog Before TC
8	Prog Memory Lock	133	Repeat Rigid Tap
9	Dimensioning	142	Offset Chng Tolerance
10	Limit Rapid at 50%	143	Machine Data Collect
11	Baud Rate Select	144	Feed Override->Spindle
12	Parity Select	145	TS at Part for CS
13	Stop Bit	156	Save Offset with PROG
14	Synchronization	157	Offset Format Type
16	Dry Run Lock Out	158,159,160	XYZ Screw Thermal COMP%
17	Opt Stop Lock Out	162	Default To Float
18	Block Delete Lock Out	163	Disable .1 Jog Rate
19	Feedrate Override Lock	164	Powerup SP Max RPM
20	Spindle Override Lock	165	SSV Variation (RPM)
21	Rapid Override Lock	166	SSV CYCLE (0.1) SECS
22	Can Cycle Delta Z	167-186	Periodic Maintenance

Code	Name	Code	Name
23	9xxx Progs Edit Lock	187	Machine Data Echo
24	Leader To Punch	196	Conveyor Shutoff
25	EOB Pattern	197	Coolant Shutoff
26	Serial Number	198	Background Color
28	Can Cycle Act w/o X/Z	199	Display Off Timer
31	Reset Program Pointer	201	Show Only Work and Tool Offsets In Use
32	Coolant Override	202	Live Image Scale
33	Coordinate System	203	Live Image X Offset
36	Program Restart	205	Live Image Z Offset
37	RS-232 Data Bits	206	Stock Hole Size
39	Beep @ M00, M01, M02, M30	207	Z Stock Face
41	Add Spaces RS-232 Out	208	Stock OD Diameter
42	M00 After Tool Change	209	Length of Stock
43	Cutter Comp Type	210	Jaw Height
44	Min F in Radius TNC %	211	Jaw Thickness
45/47	Mirror Image X-axis/Z-axis	212	Clamp Stock
52	G83 Retract Above R	213	Jaw Step Height
53	Jog w/o Zero Return	214	Show Rapid Path Live Image
55	Enable DNC from MDI	215	Show Feed Path Live Image
56	M30 Restore Default G	216	Servo and Hydraulic Shutoff
57	Exact Stop Canned X-Z	217	Show Chuck Jaws
58	Cutter Compensation	218	Show Final Pass
59/60/61/62	Probe Offset X+/X-/Z+/Z-	219	Auto Zoom to Part

Code	Name	Code	Name
63	Tool Probe Width	220	TS Live Center Angle
64	T. Ofs Meas Uses Work	221	Tailstock Diameter
65	Graph Scale (Height)	222	Tailstock Length
66	Graphics X Offset	224	Flip Part Stock Diameter
68	Graphics Z Offset	225	Flip Part Stock Length
69	DPRNT Leading Spaces	226	SS Stock Diameter
70	DPRNT Open/CLOS DCode	227	SS Stock Length
72	Can Cycle Cut Depth	228	SS Jaw Thickness
73	Can Cycle Retraction	229	SS Clamp Stock
74	9xxx Progs Trace	230	SS Jaw Height
75	9xxx Progs Single BLK	231	SS Jaw Step Height
76	Foot Pedal Lock Out	232	G76 Default P Code
77	Scale Integer F	233	SS Clamping Point
81	Tool at Auto Off	234	SS Rapid Point
82	Language	235	SS Machine Point
83	M30/Resets Overrides	236	FP Z Stock Face
84	Tool Overload Action	237	SS Z Stock Face
85	Maximum Corner Rounding	238	High Intensity Light Timer (minutes)
86	Thread Finish Allowance	239	Worklight Off Timer (minutes)
87	TNN Resets Override	240	Tool Life Warning
88	Reset Resets Overrides	241	Tailstock Hold Force
90	Graph Z Zero Location	242	Air Water Purge Interval (minutes)

Code	Name	Code	Name
91	Graph X Zero Location	243	Air Water Purge On-Time (seconds)
92	Chuck Clamping	245	Hazardous Vibration Sensitivity
93	Tailstock X Clearance	249	Enable Haas Startup Screen
94	Tailstock Z Clearance	900	CNC Network Name
95	Thread Chamfer Size	901	Obtain Address Automatically
96	Thread Chamfer Angle	902	IP Address
97	Tool Change Direction	903	Subnet Mask
98	Spindle Jog RPM	904	Default Gateway
99	Thread Minimum Cut	905	DNS Server
100	Screen Saver Delay	906	Domain/Workgroup Name
101	Feed Override -> Rapid	907	Remote Server Name
102	C Axis Diameter	908	Remote Share Path
103	CYC START/FH Same Key	909	User Name
104	Jog Handle to SNGL BLK	910	Password
105	TS Retract Distance	911	Access To CNC Share (Off, Read, Full)
106	TS Advance Distance	912	Floppy Tab Enabled
107	TS Hold Point	913	Hard Drive Tab Enabled
109	Warm-Up Time in MIN.	914	USB Tab Enabled
110/111/112	Warmup X/Y/Z Distance	915	Net Share
113	Tool Change Method	916	Second USB Tab Enabled
114/115	Conveyor Cycle/On Time (minutes)		

Introduction to Settings

The setting pages contain values that control machine operation and that the user may need to change. Most settings can be changed by the operator. They are preceded by a short description on the left and the value on the right. In general, settings allow the operator or setup person to lock out or turn on specific functions.

Settings are presented in tabbed menus. For information on navigating tabbed menus in the Haas control, refer to the Introduction section of this manual. The on-screen settings are organized into pages of functionally similar groupings. The following list is separated into page groups with the page title as the heading.

Use the vertical cursor keys to move to the desired setting. Depending on the setting, you may change it by entering a new number or, if the setting has specific values, press the horizontal cursor keys to display the choices. Press **[ENTER]** to enter or change the value. The message near the top of the screen tells you how to change the selected setting.

The serial number is Setting 26 on this page and is protected from user change. If you need to change this setting, contact Haas or your dealer. The following sections describe each of the settings in detail.

1 - Auto Power Off Timer

This setting is used to power-down the machine when it has not been used for some time. The value entered in this setting is the number of minutes the machine remains idle until it is powered down. The machine is not automatically powered down while a program is running and the time (number of minutes) starts back at zero anytime a key is pressed or the jog handle is used. The auto-off sequence gives the operator a 15-second warning before power down, at which time pressing any key stops the power down.

2 - Power Off at M30

Powers down the machine at the end of a program (**M30**) if this setting is set to **ON**. The machine gives the operator a 15-second warning once an **M30** is reached; pressing any key interrupts the sequence.

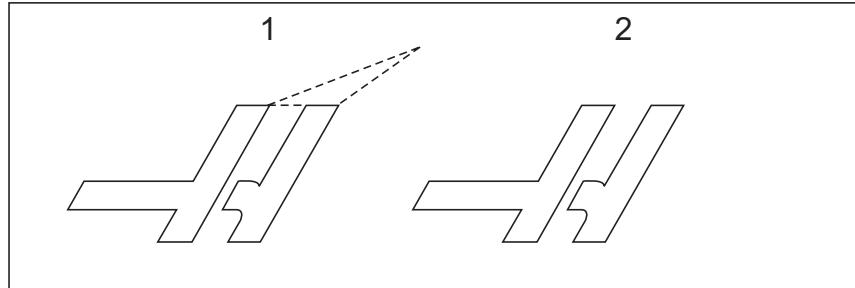
3 - 3D Graphics

3D Graphics.

4 - Graphics Rapid Path

This setting changes the way a program is viewed in the Graphics mode. When it is **OFF**, rapid (non-cutting) tool motions do not leave a path. When it is **ON**, rapid tool motions leave a dashed line on the screen.

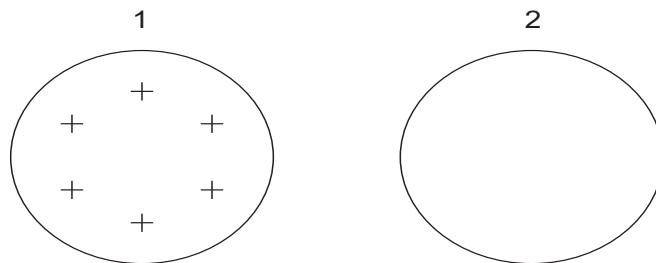
F6.72: Setting 4 - Graphics Rapid Path **ON** and **OFF**



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 leaves an X mark on the screen. When it is **OFF**, no additional marks are shown on the graphics display.

F6.73: Setting 5 - Graphics Drill Point **ON** and **OFF**



6

6 - Front Panel Lock

When set to **ON**, this Setting disables the Spindle **[FWD]/[REV]** keys and **[TURRET FWD]/[TURRET REV]** keys.

7 - Parameter Lock

Turning this setting **ON** stops the parameters from being changed, except for parameters 81-100.


NOTE:

When the control is powered up, this setting is ON.

8 - Prog Memory Lock

This setting locks out the memory editing functions (**ALTER**, **INSERT**, etc.) when it is set to **ON**. This also locks out MDI. Editing functions in FNC 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 0.001mm. All offset values are converted when this setting is changed from inches to millimeters, or vice versa. However, changing this setting does not automatically translate a program stored in memory; you must change the programmed axis values 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	mm
Feed	in/min and in/rev	mm/min and mm/rev
Max Travel	Varies by axis and model	
Minimum programmable dimension	.0001	.001
Feed range	.0001 to 500.00 in/min	.001 to 1000.000 mm/min

Axis jog key		
.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.

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 keys 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 the Z-axis is retracted to clear chips during a G73 canned cycle. The range is 0.0 to 29.9999 inches (0-760 mm).

23 - 9xxx Progs Edit Lock

Turning this setting **ON** 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.

6

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.

26 - Serial Number

This is the serial number of your machine. It cannot be changed.

28 - Can Cycle Act w/o X/Z

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 **Z** 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 **Z** code in the block.

**NOTE:**

*When an **L0** is in that block, it does not execute the canned cycle on the definition line.*

31 - Reset Program Pointer

When this setting is **OFF**, **[RESET]** does not change the position of the program pointer. When it is **ON**, **[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 generates an alarm 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 tool shift offsets work. It can be set to either **YASNAC** or **FANUC**. This setting changes the way a **Txxxx** command is interpreted and the way the coordinate system is specified. If it is **YASNAC**, tool shifts 51 to 100 are available on the offsets display and **G50 T5100** is allowed. If it is **FANUC**, tool geometry for tools 1 to 50 is available on the offsets display and **G54** style work coordinates are available.

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 ensure that the tools, offsets, G and M codes, and axis positions are set correctly before the program starts at the block where the cursor is positioned. The following M codes are processed when Setting 36 is enabled:

M08 Coolant On	M37 Parts Catcher Off
M09 Coolant Off	M41 Low Gear
M14 Clmp Main Spndl	M42 High Gear
M15 Unclmp Main Spndl	M51-M58 Set User M
M36 Parts Catcher On	M61-M68 Clear User M

When it is **OFF** the program starts without checking the conditions of the machine. Having this setting **OFF** may save time when running a proven program.

37 - RS-232 Data Bits

This setting is used to change the number of data bits for 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.

6

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 key is pressed.

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 makes a program much easier to read/edit on a personal computer (PC). When it is set to **OFF**, programs sent out the serial port have no spaces and are more difficult to read.

42 - M00 After Tool Change

Turning this setting **ON** will stop the program after a tool change and a message displays stating this. The **[CYCLE START]** button must be pressed to continue the program.

43 - Cutter Comp Type

This setting controls how the first stroke of a compensated cut begins and the way the tool is cleared from the part being cut. The selections can be **A** or **B**; see the cutter compensation section for examples.

44 - Min F in Radius TNC %

(Minimum feedrate in radius tool nose compensation percent) This setting affects the feedrate when cutter compensation moves the tool towards the inside of a circular cut. This type of cut will slow down to maintain a constant surface feedrate. This setting specifies the slowest feedrate as a percentage of the programmed feedrate (range 1-100).

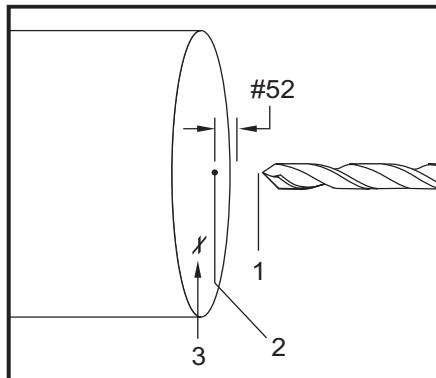
45/47 - Mirror Image X-axis/Z-axis

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**, in G-codes section.

52 - G83 Retract Above R

The Range is 0.0 to 30.00 inches or 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 empty distance. If Setting 52 is set to the distance required to clear chips, the R plane can be put closer to the part being drilled.

F6.74: Setting 52 - G83 Retract Above R: [#52] Setting 52, [1] Start position, [2] R plane, [3] Face of the part.



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**.

6

55 - Enable DNC from MDI

Turning this setting **ON** makes the DNC feature available. DNC is selected, in the control by pressing the **[MDI/DNC]** key twice. The DNC Direct Numeric Control feature is not available when set to **OFF**.

56 - M30 Restore Default G

When this setting is **ON**, ending a program with **M30** or pressing **[RESET]** returns all modal G codes to their defaults.

57 - Exact Stop Canned X-Z

The rapid XZ motion associated with a canned cycle may not achieve an exact stop when this setting is **OFF**. Turning this setting **ON** will ensure that the XZ motion comes to an exact stop.

58 - Cutter Compensation

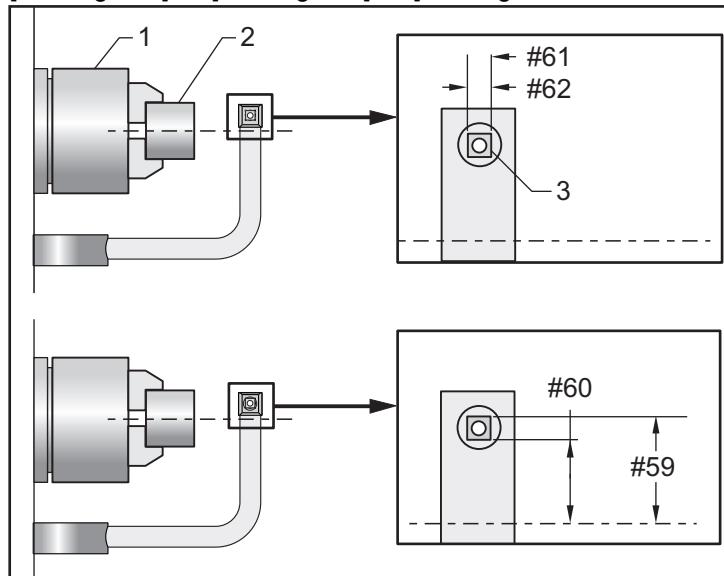
This setting selects the type of cutter compensation used (**FANUC** or **YASNAC**). See the cutter compensation section.

59/60/61/62 - Probe Offset X+/X-/Z+/Z-

These settings are used to define the displacement and size of the ATP. These four settings 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 the **G31** code. The values entered for each setting must be positive numbers.

Macros can be used to access these settings, see the Macro section for more information.

F6.75: 59/60/61/62 Tool Probe Offset:[1] Chuck, [2] Part, [3] Probe, [#59] Setting 59, [#60] Setting 60, [#61] Setting 61, [#62] Setting 62,



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.

64 - Tool Offset Measure Uses Work

This setting changes the way the **[Z FACE MEASURE]** keys 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 total X travel.

Total X travel = Parameter 6/Parameter 5
 Scale = Total X travel/Setting 65

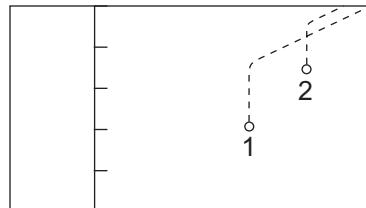
66 - Graphics X Offset

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.

68 - Graphics Z Offset

This setting locates the top of the zoom window relative to the machine Z zero position (see the Graphics section). Its default is zero.

F6.76: Setting 68 - Graphics Z Offset: [1] Setting 66 and 68 set 0, [2] Setting 66 and 68 set to 2.0.



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 does use leading spaces. The following example illustrates control behavior when this setting is **OFF** or **ON**.

	OUTPUT (Setting 69 - OFF)	OUTPUT (Setting 69 - ON)
#1 = 3.0 ;		
G0 G90 X#1 ;		
DPRNT [X#1[44]] ;	X3.0000	X 3.0000

Notice the space between the **X** and the **3** when the setting is **ON**. Information can be easier to read when this setting is **ON**.

70 - DPRNT Open/CLOS DCode

This setting controls whether the **POOPEN** and **PCLCLOS** statements in macros send **DC** control codes to the serial port. When the setting is **ON**, these statements will send **DC** control codes. When it is **OFF**, the control codes are suppressed. Its default value is **ON**.

72 - Can Cycle Cut Depth

Used with canned cycles **G71** and **G72**, this setting specifies the incremental depth for each pass during rough cutting. It is used if the programmer does not specify a **D** code. Valid values range from 0 to 29.9999 inches or 299.999 mm. The default value is .1000 inches.

73 - Can Cycle Retraction

Used with canned cycles **G71** and **G72**, this setting specifies the retraction amount after a roughing cut. It represents the tool to material clearance as the tool returns for another pass. Valid values range from 0 to 29.9999 inches or 299.999 mm. The default value is .0500 inches.

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, 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**, then 9000 series programs are displayed as they are executed.

76 - Foot Pedal Lock Out

This is an **on/off** setting. When it is **off**, the foot pedal operates normally. When it is **on**, any action at the foot pedal is ignored by the control.

77 - Scale Integer F

This setting allows the operator to select how the control interprets an **F** value (feedrate) that does not contain a decimal point. (It is recommended that programmers always use a decimal point.) This setting helps operators run programs developed on a control other than Haas. For Example **F12**:

Setting 77 **OFF** - 0.0012units/minute
Setting 77 **ON** - 12.0 units/minute

There are 5 feedrate settings:

INCH		MILLIMETER	
DEFAULT	(.0001)	DEFAULT	(.001)
INTEGER	F1 = F1	INTEGER	F1 = F1
.1	F1 = F.0001	.1	F1 = F.001
.01	F10 = F.001	.01	F10 = F.01
.001	F100 = F.01	.001	F100 = F.1
.0001	F1000 = F.1	.0001	F1000 = F1

81 - Tool at Auto Off

When **[AUTO OFF]** is pressed, the control performs a tool change to the tool specified in this setting. If zero (0) is specified, no tool change occurs before shutting down the lathe. The default setting is 1 for tool 1.

82 - Language

Languages other than English are available in the Haas control. To change to another language, choose a language and press **[ENTER]**.

83 - M30/Resets Overrides

When this setting is **ON**, an M30 restores any overrides (feedrate, spindle, rapid) to their default values (100%).

84 - Tool Overload Action

This setting causes the specified action (Alarm, Feedhold, Beep, Autofeed) to occur anytime a tool becomes overloaded (see the Tooling section).

Choosing **ALARM** causes the machine to stop when the tool is overloaded.

When set to **FEEDHOLD**, the message *Tool Overload* is displayed and the machine stops in a feedhold situation when this condition occurs. Pressing any key will clear the message.

Selecting **BEEP** causes an audible noise from the control when the tool is overloaded.

When set to **AUTOFEED**, the lathe automatically limits the feedrate based on the tool load.



NOTE:

When tapping (rigid or floating), the feed and spindle overrides are locked out, so the Autofeed feature is ineffective (the control appears to respond to the override keys, by displaying the override messages). The Autofeed feature should not be used when thread milling or auto reversing tapping heads, as it may cause unpredictable results or even a crash.



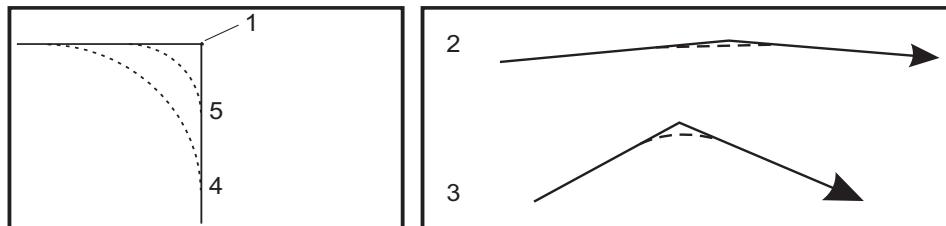
NOTE:

The last commanded feedrate is restored at the end of the program execution, or when the operator presses [RESET] or turns off the Autofeed feature. The operator may use the keyboard feedrate override keys while the Autofeed feature is selected. These keys are recognized by the Autofeed feature as the new commanded feedrate as long as the tool load limit is not exceeded. However, if the tool load limit is exceeded, the control ignores the feedrate override keys.

85 - Maximum Corner Rounding

Defines the machining accuracy of rounded corners within a selected tolerance. The initial default value is 0.05 inch. If this setting is zero (0), the control acts as if an exact stop is commanded in each motion block.

F6.77: Setting 85 - Maximum Corner Rounding: [1] Program point, [2] No slowdown required to meet accuracy setting, [3] Much lower speed needed to machine into the corner, [4] Setting 85 = 0.050, [5] Setting 85 = 0.025.



86 - Thread Finish Allowance

Used in G76 canned threading cycle, this setting specifies how much material is left on the thread for the final pass of the cycle. Values range from 0 to .9999 inches. The default value is 0.

87 - Tnn Resets Override

This is an **ON/OFF** setting. When a tool change is executed and this setting is **ON**, any overrides are canceled and set to their programmed values.

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 default values (100%).

90 - Graph Z Zero Location

This setting adjusts for extreme values in tool geometry or shift values. In graphics, tool offsets are ignored so that the cutting paths of different tools are displayed in the same location. Setting this to an approximate value of machine coordinates for the programmed part zero will void any Z Over Travel Range alarms that you may encounter in graphics. The default is -8.000.

91 - Graph X Zero Location

This setting adjusts for extreme values in tool geometry or shift values. In graphics, tool offsets are ignored so that the cutting paths of different tools are displayed in the same location. Setting this to an approximate value of machine coordinates for the programmed part zero will void any x Over Travel Range alarms that you may encounter in graphics. The default is -6.000.

92 - Chuck Clamping

This setting determines chuck clamping direction. Set to O.D., the chuck is considered clamped when the jaws are moved to the spindle center. Set to I.D., the chuck is considered clamped when the jaws are moved away from the spindle center.

93 - Tailstock X Clearance

This setting works with Setting 94 to define a tail stock travel restriction zone that limits interaction between the tailstock and the tool turret. This setting determines the X-axis travel limit when the difference between the Z-axis location and the tailstock location falls below the value in Setting 94. If this condition occurs and a program is running then an alarm is generated. When jogging, no alarm is generated, but travel will be limited.

94 - Tailstock Z Clearance

This setting is the minimum allowable difference between the Z-axis and the tailstock (see Setting 93). If units are in inches, a value of -1.0000 means that when the X-axis is below the X clearance plane (Setting 93), the Z-axis must be more than 1 inch away from the tailstock position in the Z-axis negative direction.

95 - Thread Chamfer Size

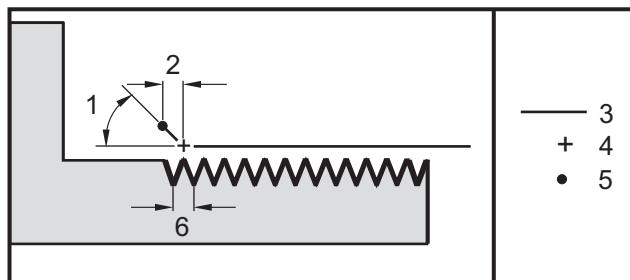
This setting is used in G76 and G92 threading cycles when an M23 is commanded. When command M23 is active, threading strokes end with an angled retraction, as opposed to pulling straight out. The value in Setting 95 is equal to the number of turns (chamfered threads) desired.


NOTE:

*Settings 95 and 96 interact with each other. Valid range: 0 to 29.999
(Multiple of current thread lead, F or E).*

6

- F6.78:** Setting 95 - Thread Chamfer Size, G76 or G92 threading stroke with M23 active:
[1] Setting 96 = 45, [2] Setting 95 x Lead, [3] Tool path, [4] Programmed thread endpoint, [5] Actual stroke endpoint, [6] Lead.



96 - Thread Chamfer Angle

See Setting 95. Valid range: 0 to 89 degrees (No decimal point allowed)

97 - Tool Change Direction

This setting determines the default tool change direction. It may be set to either **SHORTEST** or M17/M18.

When **SHORTEST** is selected, the control turns in the direction necessary to reach the next tool with the least movement. The program can still use M17 and M18 to fix the tool change direction, but once this is done it is not possible to revert back to the shortest tool direction other than **[RESET]** or M30/M02.

Selecting M17/M18, the control moves the tool turret either always forward or always reverse based on the most recent M17 or M18. When **[RESET]**, **[POWER ON]**, or M30/M02 is executed, the control assumes M17 as the tool turret direction during tool changes, always forward. This option is useful when a program must avoid certain areas of the tool turret due to odd-sized tools.

98 - Spindle Jog RPM

This setting determines the spindle rpm for the **[SPINDLE JOG]** key. The default value is 100 RPM.

99 - Thread Minimum Cut

Used in G76 canned threading cycle, this setting sets a minimum amount of successive passes of the thread cut. Succeeding passes cannot be less than the value in this setting. Values can range from 0 through .9999 inch. The default value is .0010 inches.

100 - Screen Saver Delay

When the setting is zero, the screen saver is disabled. If setting is set to some number of minutes, then after that time with no keyboard activity, the Haas logo displays and changes position every 2 seconds (deactivate with any key press, handle jog or alarm). The screen saver will not activate if the control is in Sleep, Jog, Edit, or Graphics mode.

101 - Feed 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.

102 - C Axis Diameter

This setting supports the C-Axis. See the C-Axis Section. The default value is 1.0 inches and the maximum allowable value is 29.999 inches.

103 - CYC START/FH Same Key

The **[CYCLE START]** button must be pressed and held to run a program when this setting is **ON**. When **[CYCLE START]** is released, a feed hold is generated. This setting cannot be turned **ON** while Setting 104 is **ON**. When one of them is set to **ON**, the other will automatically turn **OFF**.

104 - Jog Handle to SNGL BLK

The jog handle can be used to single-step through a program when this setting is **ON**. Reversing the jog handle direction will generate a feed hold. This setting cannot be turned **ON** while Setting 103 is **ON**. When one of them is set to **ON**, the other will automatically turn **OFF**.

105 - TS Retract Distance

The distance from the Hold Point (Setting 107) the tailstock will retract when commanded. This setting should be a positive value.

106 - TS Advance Distance

When the tailstock is moving toward the Hold Point (Setting 107), this is the point where it will stop its rapid movement and begin a feed. This setting should be a positive value.

107 - TS Hold Point

This setting is in absolute machine coordinates and should be a negative value. It is the point to advance to for holding when M21 is commanded. Usually this is inside of a part being held. It is determined by jogging to the part and adding .375 - .500" (9.5 - 12.7 mm) to the absolute position.

109 - Warm-Up Time in MIN.

This is the number of minutes (up to 300 minutes from power-up) during which the compensations specified in Settings 110-112 are applied.

Overview – When the machine is powered on, if Setting 109, and at least one of Settings 110, 111, or 112, are set to a nonzero value, the following warning will be displayed:

CAUTION! Warm up Compensation is specified!

Do you wish to activate

Warm up Compensation (Y/N) ?

If a Y is entered, the control immediately applies the total compensation (Setting 110, 111, 112), and the compensation begins to decrease as the time elapses. For instance, after 50% of the time in Setting 109 has elapsed, the compensation distance 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 warmup time is displayed on the bottom right hand corner of the Diagnostics Inputs 2 screen using the standard hh:mm:ss format.

110/112 - Warmup X/Z Distance

Settings 110 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 for settings 110 and 112 to have an affect.

113 - Tool Change Method

This setting is used for the TL-1 and TL-2 lathes. See the Toolroom Lathe manual.

114/115 - Conveyor Cycle/On Time (minutes)

Settings 114 and 115 control the optional chip conveyor. Setting 114 (Conveyor Cycle Time) is the interval that the conveyor turns on automatically. Setting 115 (Conveyor On-Time) is the amount of time the conveyor runs. For example, if setting 114 is set to 30 and setting 115 is set to 2, the chip conveyor turns on every half an hour, runs for 2 minutes, then turns off.

On-time should be set no greater than 80% of cycle time.



NOTE:

Pressing [CHIP FWD] (or M31) starts the conveyor in the forward direction and activates the cycle. The [CHIP STOP] key (or M33) stops the conveyor and cancels the cycle.

118 - M99 Bumps M30 CNTRS

When this setting is ON, an M99 adds one to the M30 counters (these are visible on the [CURRENT COMMANDS] displays).



NOTE:

An M99 only increases the counters as it occurs in a main program, not a sub-program.

6

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 macro variables to be altered. However, programs that alter macro variables are still able to do so.

121 - Foot Pedal TS Alarm

When M21 is used to move the tailstock to the hold point and hold a part, the control generates an alarm if a part is not found and the hold point is reached. Setting 121 can be switched to ON and an alarm is generated when the foot pedal is used to move the tailstock to the hold point and no part is found.

122 - Secondary Spindle Chuck Clamping

This feature supports Secondary-spindle lathes. Its value can be either O.D. or I.D.; similar to Setting 92 for the main spindle.

131 - Auto Door

This setting supports the Auto Door option. It should be set to ON for machines with an autodoor. Also see M85/M86 (Autodoor Open/Close M-codes).

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.

132 - Jog Before TC

This is a safety setting to help prevent crashing the turret when using [TURRET FWD], [TURRET REV], or [NEXT TOOL] keys. When this setting is ON, the control generates a message when one of these keys are pressed and not allow the turret to rotate unless all axes are at home position or one or more of the axis were moved in Handle Jog mode.

When this setting is OFF, no assumptions are made and the lathe performs tool changes without displaying a message.

133 - Repeat Rigid Tap

This setting ensures that the spindle is oriented during tapping so that the threads line up when a second tapping pass, in the same hole, is programmed.

142 - Offset Chng Tolerance

This setting generates a warning message if an offset is changed by more than the amount entered for this setting. If an attempt is made to change an offset by more than the entered amount (either positive or negative), the following prompt is displayed: *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 a Q command 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, see CNC Data Transfer in the Operation Programming section.

144 - Feed Override->Spindle

When this setting is ON, any feedrate override will also be applied to the spindle speed, and the spindle overrides will be disabled.

145 - TS at Part for CS

When Setting 145, Tail Stock at Part for [CYCLE START] is OFF, the machine behaves as before. When this setting is ON, the tail stock must be pressing against the part at the moment [CYCLE START] is pressed or a message is displayed and the program will not start.

6

156 - Save Offset with PROG

When this setting is ON, the control includes the offsets in the program file when you save it to USB, HD, or NetShare. The offsets appear in the file before the final % sign, under the heading O999999.

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 more easily edited on a PC and later reloaded.

When it is set to **B**, each offset is saved on a separate line with an **X** value and a **V** value.

158,159,160 - XYZ Screw Thermal COMP%

These settings can be set from -30 to +30 and will adjust the existing screw thermal compensation by -30% to +30% accordingly.

162 - Default To Float

When this setting is **ON**, the control adds a decimal point to values entered without a decimal point for certain address codes. When the setting is **OFF**, values following address codes that do not include decimal points are taken as machinists notation (i.e., thousandths or ten-thousandths). This setting excludes the **A** value (tool angle) in a **G76** block. Thus, the feature applies to the following address codes:

	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.

This feature applies to the following address codes:

X, Y, Z, A, B, C, E, F, I, J, K, U, W

A (except with G76) If a G76 A value containing a decimal point is found during program execution, Alarm 605 Invalid Tool Nose Angle is generated.

D (except with G73)

R (except with G71 in YASNAC mode)

**NOTE:**

This setting affects the interpretation of all programs entered either manually or from disk or via RS-232. It does not alter the effect of Setting 77 Scale Integer F.

163 - Disable .1 Jog Rate

This setting disables the highest jog rate. If the highest jog rate is selected, the next lower rate is automatically selected instead.

164 - Powerup SP Max RPM

This setting is used to set the maximum spindle RPM each time the machine is powered on. It will essentially cause a G50 Snnn command to be executed at power on time, where nnn is the value from the setting. If the setting contains zero, or a value equal to or greater than parameter 131 MAX SPINDLE RPM, Setting 164 will have no effect.

165 - SSV Variation (RPM)

Specifies the amount by which to allow the RPM to vary above and below its commanded value during the use of the Spindle Speed Variation feature. Positive value only.

166 - SSV CYCLE (0.1) SECS

Specifies the duty cycle, or the rate of change of Spindle Speed. Positive value only.

167-186 - Periodic Maintenance

There are 14 items that can be monitored, as well as six spare items, in the Periodic Maintenance Settings. These settings will allow the user to change the default number of hours for each item when it is initialized during use. If the number of hours is set to zero, the item will not appear in the list of items shown in the maintenance page of current commands.

187 - Machine Data Echo

Turning this setting ON displays the data collection Q commands on the PC screen.

196 - Conveyor Shutoff

This specifies the amount of time to wait without activity prior to turning off the chip conveyor. Units are minutes.

197 - Coolant Shutoff

Specifies the amount of time to wait without activity prior to Flood, Shower, and High Pressure Coolant turn off. Units are minutes.

198 - Background Color

Specifies the background color for inactive display panes. Range is 0 to 254.

199 - Display Off Timer

Specifies the time in minutes after which the machine display 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

Turning this setting **ON** displays only the Work and Tool offsets used by the running program. The program must be run in the Graphics mode first to activate this feature.

202 - Live Image Scale (Height)

Specifies the height of the work area that is displayed in Live Image. The maximum size is automatically limited to the default height. The default shows the machine's entire work area.

203 - Live Image X Offset

Locates the top of the scaling window relative to the machine X zero position. The default is zero.

205 - Live Image Z Offset

Locates the right side of the scaling window relative to the machine X zero position. The default is zero.

206 - Stock Hole Size

Demonstrates the I.D. of the part. This setting is adjusted also by entering a value in HOLE SIZE in the STOCK SETUP tab in IPS.

207 - Z Stock Face

Controls the Z stock face of the raw part displayed in Live Image. This setting is adjusted also by entering a value in STOCK FACE in the STOCK SETUP tab in IPS.

208 - Stock OD Diameter

This setting controls the diameter of the raw part that will be displayed in Live Image. This setting can also be adjusted from IPS.

209 - Length of Stock

Controls the length of the raw part displayed in Live Image. This setting is adjusted also by entering a value in STOCK LENGTH in the STOCK SETUP tab in IPS.

210 - Jaw Height

This setting controls the height of the chuck jaws that will be displayed in Live Image. This setting can also be adjusted from IPS.

211 - Jaw Thickness

Controls the thickness of the chuck jaws displayed in Live Image. This setting is adjusted also by entering a value in JAW THICKNESS in the STOCK SETUP tab in IPS.

212 - Clamp Stock

Controls the clamp stock size of the chuck jaws that is displayed in Live Image. This setting is adjusted also by entering a value in CLAMP STOCK in the STOCK SETUP tab in IPS.

213 - Jaw Step Height

Controls the height of the chuck jaws step that is displayed in Live Image. This setting is adjusted also by entering a value in JAW STEP HEIGHT in the STOCK SETUP tab in IPS.

214 - Show Rapid Path Live Image

Controls the visibility of a red dashed line that represents a rapid path in Live Image.

215 - Show Feed Path Live Image

Controls the visibility of a solid blue line that represents a feed path in Live Image.

216 - Servo and Hydraulic Shutoff

This setting turns the servomotors and hydraulic pump, if equipped, off after the specified number of minutes has elapsed without activity, such as running a program, jogging, key presses, etc. The default is 0.

217 - Show Chuck Jaws

Controls the display of the chuck jaws in Live Image.

218 - Show Final Pass

Controls the visibility of a solid green line that represents a final pass in Live Image. This is shown if the program has been previously run or simulated.

219 - Auto Zoom to Part

Controls whether or not Live Image auto zooms the part to the bottom left corner. Turn on or off by pressing **[F4]** on the Live Image page.

220 - TS Live Center Angle

Angle of the tailstock's live center measured in degrees (0 to 180). Used for Live Image only. Initialize with a value of 60.

221 - Tailstock Diameter

Diameter of the tailstock's live center measured in inch or metric (depending on Setting 9), times 10,000. Used for the Live Image only. Default value is 12500 (1.25"). Use positive value only.

222 - Tailstock Length

Length of the tailstock's live center measured in inch or metric (depending on Setting 9) times 10,000. Used for Live Image only. Default value is 20000 (2.0000"). Use positive value only.

224 - Flip Part Stock Diameter

Controls the new diameter location of the jaws after flipping the part

225 - Flip Part Stock Length

Controls the new length location of the jaws after flipping the part.

6

226 - SS Stock Diameter

Controls the diameter of the part where the secondary spindle clamps it.

227 - SS Stock Length

Controls the length of the secondary spindle from the left of the part.

228 - SS Jaw Thickness

Controls the secondary spindle jaw thickness.

229 - SS Clamp Stock

Controls the secondary spindle clamp stock value.

230 - SS Jaw Height

Controls the secondary spindle jaw height.

231 - SS Jaw Step Height

Controls the secondary spindle jaw step height.

232 - G76 Default P Code

The default P code value to use when a P code does not exist in a G76 line, or when the P code used has a value less than 1 or greater than 4. Possible values are P1, P2, P3, or P4.

233 - SS Clamping Point

Controls the clamping point (the location on the part where the secondary spindle clamps it) for display purposes in Live Image. This value is also used to create a G code program that will perform the desired secondary spindle operation.

234 - SS Rapid Point

Controls the rapid point (the location to which the secondary spindle rapids before clamping a part) for display purposes in Live Image. This value is also used to create a G code program that will perform the desired secondary spindle operation.

235 - SS Machine Point

Controls the machining point (the location where the secondary spindle machines a part) for display purposes in Live Image. This value is also used to create a G code program that will perform the desired secondary spindle operation.

236 - FP Z Stock Face

Controls the flip part stock face for display purposes in Live Image. This value is also used to create a G code program that will perform the desired secondary spindle operation.

237 - SS Z Stock Face

Controls the secondary spindle stock face for display purposes in Live Image. This value is also used to create a G code program that will perform the desired secondary spindle operation.

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 are highlighted in orange and the beacon light will flash yellow.

241 - Tailstock Hold Force

Force to apply to a part by the servo tailstock (ST-40 and ST-40L only). Unit is pounds-force in standard mode and Newton in metric mode, as per Setting 9. Valid range is 1000 (4448 in metric mode) to 4500 (20017 in metric mode).

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 is begun.

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.

900 - CNC Network Name

The control name you would like to show up on the network.

901 - Obtain Address Automatically

Retrieves a TCP/IP address and subnet mask from a DHCP server on a network (Requires a DHCP server). When DHCP is on, TCP/IP, SUBNET MASK and GATEWAY entries are no longer required and will have “***” entered. Also note the ADMIN section at the end to get the IP address from DHCP. The machine must be turned off and back on for changes to this setting to take effect.



NOTE:

To get IP settings from DHCP: At the control, press [LIST PROGRAM]. Arrow down to the Hard Drive. Press the right arrow key for the Hard Drive directory. Type in ADMIN and press [INSERT]. Select ADMIN folder and press [ENTER]. Copy the IPCConfig.txt file to disk or USB and read it on a Windows computer.

902 - IP Address

Used on a network with static TCP/IP addresses (DHCP off). The network administrator will assign an address (example 192.168.1.1). The machine must be turned off and back on for changes to this setting to take effect.

**NOTE:**

The address format for Subnet Mask, Gateway and DNS is XXX.XXX.XXX.XXX (example 255.255.255.255) do not end the address with a period. The max address is 255.255.255.255; no negative numbers.

903 - Subnet Mask

Used on a network with static TCP/IP addresses. The network administrator will assign a mask value. The machine must be turned off and back on for changes to this setting to take effect.

904 - Gateway

Used to gain access through routers. The network administrator will assign an address. The machine must be turned off and back on for changes to this setting to take effect.

905 - DNS Server

The Domain Name Server or Domain Host Control Protocol IP address on the network. The machine must be turned off and back on for changes to this setting to take effect.

6

906 - Domain/Workgroup Name

Tells the network which workgroup or domain the CNC control belongs to. The machine must be turned off and back on for changes to this setting to take effect.

907 - Remote Server Name

For Haas machines with WINCE FV 12.001 or higher, enter the NETBIOS name from the computer where the share folder resides. IP address is not supported.

908 - Remote Share Path

The name of the shared network folder. After a host name is selected, to rename the path, enter the new path and press [ENTER].



NOTE:

Do not use spaces in the PATH field.

909 - User Name

This is the name used to log on to the server or domain (using a user domain account). The machine must be turned off and back on for changes to this setting to take effect. User Names are case sensitive and cannot contain spaces.

910 - Password

This is the password used to log on to the server. The machine must be turned off and back on for changes to this setting to take effect. Passwords are case sensitive and cannot contain spaces.

911 - Access To CNC Share (Off, Read, Full)

Used for the CNC hard drive read/write privileges. **OFF** stops the hard drive from being networked. **READ** allows read-only access to the hard drive. **FULL** allows read/write access to the drive from the network. Turning **OFF** both this setting and Setting 913 will disable network card communication.

912 - Floppy Tab Enabled

This turns access to the USB floppy drive **OFF/ON**. When set to **OFF**, the USB floppy drive will not be accessible.

913 - Hard Drive Tab Enabled

Turns access to the hard drive **OFF/ON**. When set to **OFF**, hard drive will not be accessible. Turning **OFF** both this setting and CNC Share (Setting 911) will disable network card communication.

914 - USB Tab Enabled

Turns access to the USB port OFF/ON. When set to OFF the USB port will not be accessible.

915 - Net Share

Turns access to the server drive OFF/ON. When set to OFF access to the sever from the CNC control is not possible.

916 - Second USB Tab Enabled

Turns access to the secondary USB port OFF/ON. When set to OFF the USB port will not be accessible.

6.4 More Information Online

For updated and supplemental information, including tips, tricks, maintenance procedures, and more, go to www.HaasCNC.com and select the **Resource Center**.

Chapter 7: Maintenance

7.1 Introduction

Regular maintenance is important to make sure that your machine has a long and productive life with minimal downtime. This section gives you a list of maintenance tasks that you can do yourself at the intervals listed to keep your machine running. Your dealer also offers a comprehensive preventive maintenance program that you can take advantage of for more complex maintenance tasks.

For detailed instructions on the procedures listed in this section, go to www.HaasCNC.com and select the **Resource Center**.

7.2 Daily Maintenance

- Check the coolant level each eight-hour shift (especially during heavy HPC usage).



NOTE:

If your coolant system includes an auxiliary filter, do not completely fill the coolant tank at the end of the operating day. The auxiliary filter drains approximately (5) gallons (19 liters) of coolant back into the coolant tank overnight.

- Check the HPC pump oil level.
- Check the lubrication tank level.
- Clean the chips from the way covers and bottom pan.
- Clean chips from turret, housing, rotating union and extension tube. Make sure the drawtube cover plate is installed either on the rotating union or on the chuck opening.
- Check hydraulic unit oil level (DTE-25 only). Capacity: 8 gallons (10 gallons for SL-30B and up).

7.3 Weekly Maintenance

- Check the High-Pressure Coolant (HPC) filters. Clean or replace them if needed.
- Check for proper operation of auto drain on the filter regulator.
- On machines with the HPC option, clean the chip basket on the coolant tank. Do this monthly for machines without the HPC option.
- Check the air gauge/regulator for 85 psi.
- Clean all of the exterior surfaces with mild cleaner. DO NOT use solvents.

**CAUTION:**

Do not use a wash-down hose on the Haas lathe; doing so may cause damage to the spindle.

7.4 Monthly Maintenance

- Dump the oil drain bucket. Check the oil level in the gear box (if equipped).

**CAUTION:**

Before working on the coolant tank press [POWER OFF] at the control. Disconnect the coolant pump from the controller.

- Remove the pump from the coolant tank. Clean sediment from inside the tank. Reinstall the pump.
- Inspect the grease and oil reservoirs and add grease or oil as needed.
- Inspect way covers for proper operation and lubricate them with light oil, if necessary.
- Check for dust buildup on the electrical cabinet vector drive vents (beneath the power switch). If buildup exists, open the cabinet and wipe vents with a clean cloth. Apply compressed air as necessary to remove the dust accumulation.

7.5 Every (6) Months

- Replace coolant and thoroughly clean the coolant tank.
- Replace the hydraulic unit oil filter.
- Check all hoses and lubrication lines for cracking.

7.6 Annual Maintenance

- Replace the gearbox oil (if equipped).
- Clean the oil filter inside the lubrication panel oil reservoir and clean out the residue at the bottom of the filter.

Chapter 8: Other Equipment

8.1 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.

8.2 Office Lathe

The Office Lathe series are compact small-scale lathes that can fit through a standard door frame and run on single-phase power.

8.3 Toolroom Lathe

The Toolroom Lathe includes features aimed at a machinist used to a manually positioned lathe. The lathe uses familiar manual handles, while giving full CNC capabilities.

8.4 More Information Online

For updated and supplemental information, including tips, tricks, maintenance procedures, and more, go to www.HaasCNC.com and select the **Resource Center**.

Index

A

absolute positioning	181
active codes	24
active codes display	
current commands	34
active program.....	76
active tool display.....	25
Address	
substitution	224
advanced editor	170
edit menu	164
modify menu	168
pop-up menu	162
program menu.....	162
search menu.....	166
text selection.....	164
Advanced Tool Management	35
Advanced Tool Management, See ATM	
Aliased G and M codes	236
ATM	
macros and	113
navigation	112
operation.....	113
tips & tricks.....	113
tool group setup.....	112
ATP.....	265
alarms	270
automatic mode.....	267
break detect.....	268
calibration	269
calibration operation check	269
calibration procedure	269
manual mode	266
operation.....	265
tool tip direction	268

auto door (option)

override	63
automatic tool offset setting	185
automatic tool setting probe, See ATP	
axis overload timer	108

B

background edit.....	108, 171
barstock	
safety and	2
basic program example	
completion code blocks.....	181
cutting code blocks	181
preparation block.....	180
Basic Programming	179
completion code blocks.....	181
cutting code blocks	181
preparation.....	180
basic programming	
absolute vs. incremental.....	181
beacon light	
status	63

C

C- Axis	
jog the	59
c axis.....	246
calculator	
circle	69
circle-circle tangent	71
circle-line tangent	70
triangle	68
chuck	
installation of	91
removal of	92
safety and	2

chuck foot pedal	88
clipboard	
copy to	166
cut to	165
paste from.....	166
Collet installation	93
communications	
RS-232.....	82
control cabinet.....	20
secure latches	2
side panel	21
control display	
active codes	24
active tool	25
basic layout.....	22
offsets	24, 35
tailstock	25
control pendant	21–63
detail.....	14
front panel controls	61
USB port.....	63
coolant	
operator override	61
coolant level gauge	25
coolant tank assembly	
detail.....	18
coordinate system	
automatic tool offset setting	185
effective.....	184
FANUC	184
FANUC child coordinate.....	184
FANUC common coordinate	184
FANUC work coordinate	184
global.....	185
YASNAC machine coordinate	184
YASNAC work coordinate	184
coordinate systems	183
copying files.....	77
current commands	34
additional setup	106
D	
data collection	83
spare M-codes	85
with RS-232	83
deleting programs.....	77
Departure move	120
detail.....	20
device manager	74
program selection.....	76
DIR FULL message	78
direct numeric control (DNC).....	86
operating notes.....	87
display	
graphics	32
settings	32
distance to go position	30
door	
safety	6
Drawtube	
clamping force adjustment	90
cover plate.....	91
warnings	88
drip mode	87
Dual spindle	261
finding R value.....	264
R phase offset	263
secondary spindle	261
synchronization control display.....	262
synchronized spindle control.....	261
duplicating a program	78
E	
edit keys	
ALTER.....	149
DELETE.....	149
INSERT	148
UNDO.....	149
editing	
highlight code	148
F	
FANUC macro features	
not included	240
Features	
axis overload timer	106
background edit	106
dry run	106, 107
Graphics	106
running programs	106

feed hold	
as override	61
file directory system	75
directory creation.....	76
navigation	76
file numeric control (FNC)	85
display footer.....	151
display modes.....	150
FNC editor	149
loading a program.....	149
menus.....	150
opening multiple programs	151
file numeric control (FNC) editor	
text selection	153
files	
copying	77
folder , See directory structure	
foot pedals	
chuck.....	88
steady rest.....	94
tailstock.....	100
Functions	226
G	
G65 macro subroutine call	235
gauges display	
coolant	25
Global variables	209
Graphics Mode	
program run	32
graphics mode	107
H	
hazards	
environmental	4
rotating parts	1
help	
calculator	67
drill table.....	67
keyword search.....	66
tabbed menu	66
help function	65
High-Pressure Coolant	
HPC	18, 54, 429
I	
icon bar	36
incremental positioning	181
input bar	33
J	
Jog mode	103
entry.....	103
K	
keyboard	48
alpha keys	57
cursor keys	50
display keys	51
jog keys	57
mode keys	52
numeric keys.....	56
override	59
L	
lathe feature location	13
Live Imaging	
flipped manually.....	196
machining.....	194
operation.....	193
program example	187
stock setup	186
tailstock setup	191
tool setup	188
Live tooling	242
cartesian coordinate commands	247
cartesian coordinate programming	247
cartesian interpolation example	248
cartesian m-codes	248
cartesian to polar	246
cartesian to polar programming	247
c-axis	242
cutting tool installation	243
m133/m134/m135 fwd/rev/stop	246
m19 orient spindle	246
mounting and alignment	244
mounting in turret	244
programming notes	243
Local variables	208

M	
M30 counters	26
machine	
operating limits.....	4
machine data	
back up and recover.....	79
backup	80
restore	81
machine position.....	30
macro variables	
#3006 programmable stop	218
#4001-#4021 last block group codes ...	218
#5001-#5006 last target position	219
#5021-#5026 current machine coordinate position	219
#5041-#5046 current work coordinate position	220
#5061-#5069 current skip signal position	220
#5081-#5086 tool length compensation	220
#6996-#6999 parameter access	220
#8550-#8567 tooling	224
axis position	219
current commands display.....	34
macros	201
1-bit discrete outputs	215
example program	241
g- and m-codes	202
look ahead	203
M30 counters and	26
operation notes	204
round off	202
settings	202
variables.....	207
main spindle display	48
maintenance	429
annual.....	430
current commands	35
daily.....	429
monthly	430
weekly.....	429
maintence	
every (6) months	430
manual data input (MDI)	170
material	
fire risk	5
m-codes	
about.....	374
memory lock.....	63
mode display	23
O	
O09xxx program numbers	147
offsets	
displays.....	24, 35
operating modes	23
operation	
device manager	74
unattended	5
operator position	30
overrides	61
disabling.....	61
system.....	217
P	
part zero	106
setting for z axis.....	106
parts catcher	259
chuck interference	260
operation	259
position display	30
axis selection	31
current commands.....	34
positions	
distance to go.....	30
machine	30
operator	30
work (G54).....	30
power on	73
program	
active.....	76
line numbers	
removal	168
program names	
Onnnnn format	77
program number change	79
program numbers	
change in memory.....	79
O09xxx	147
program optimizer.....	110
screen	111

program selection	76	Secondary spindle	
programs		m-codes	264
.nc file extension	77	OD and ID clamping	265
basic editing	148	programming	264
basic search	82	spindle swap	265
changing a program number	79	Servo Auto Door.....	271
deleting	77	Servo tailstock	
duplication	78	power failure	98
file naming	77	start up	98
maximum number of	78	setup mode	
running	108	keyswitch	63
transfer.....	76	shop roles	
R		machine cleaner	3
robot cell		owner	1
integration.....	6	spindle load meter	48
RS-232	82	spindle warm-up.....	74
cable length.....	83	ST/DS-30 minimal lube panel	
data collection.....	83	detail	17
DNC and	86	ST-10 minimal lube panel	
DNC settings	86	detail	15
running programs.....	108	ST-20 minimal lube panel	
Run-Stop-Jog-Continue.....	109	detail	16
S		steady rest foot pedal	94
safety		Subroutines	199
decals	9	Synchronized Spindle Control (SSC)	265
electrical panel.....	2	System variables	209
eye and ear protection	1		
hazardous material	2		
introduction	1		
keyswitch operation	6		
part loading/unloading	3		
robot cells	6		
tool loading/unloading.....	3		
safety decals			
other	11		
standard layout	9		
safety modes			
setup	5		
second home	63		

tailstock	
advance point	100
cancel restricted zone.....	102
foot pedal.....	100
hold force.....	97
hold point.....	100
jogging	103
motion.....	99
programming	197
restricted zone	101
resume operation	96
retract point	100
Setting 94 and	102
settings	99
ST-40 servo brake engage	98
ST-40 servo operation	97
X-axis clearance plane	102
tailstock display	25
text selection	
advanced editor and.....	164
FNC editor and	153
Through-Spindle Coolant	
TSC.....	47
timers and counters display	26
tips and tricks	
calculator	176
operation	175
programming	172
offsets.....	174
settings and parameters.....	174
TNC	
approach and departure	120
approach move.....	120
canned cycles	123
concept.....	118
Ex1-standard interpolation	124
Ex2-G71 roughing canned cycle.....	126
Ex3-G72 roughing canned cycle.....	127
Ex4-G73 roughing canned cycle.....	128
Ex5-G90 modal rough turning cycle	129
Ex6-G94 modal rough turning cycle	130
general	116
geometry	134
Imaginary Tool Tip.....	131
manually calculating	133
programming.....	117
radius and radius wear offset.....	121
tool length geometries	123
using	119
without.....	133
Tnn code	103
Tool	
offsets.....	216
tool functions	182
FANUC coordinate system.....	182
load or change tools	183
YASNAC coordinate system.....	182
tool life display	
current commands.....	35
tool load limits	106
Tool Nose Compensation	120
tool nose compensation, See TNC	
tool offset.....	105
manual entry	105
manually set.....	105
setting.....	104
Tool offsets. See Tool offset	
tool radius cutter compensation	250
entry and exit	252
example	254
feed adjustments.....	253

tool turret	
air pressure	114
eccentric locating cam buttons	114
load or change tools	116
operations	114
protective caps	115
touch off the tools	104
U	
unattended operation	
fire risk and	5
USB device	74
V	
Variable	
usage	223
variables	
global	209
local	208
system	209
visual quick code, See VQC	
VQC	197
entering data	198
select a category	198
select a part template	198
W	
windows	
damaged, safety and	1
work (G54) position	30
work offsets	221
workholding	87
safety and	2
workpiece	
safety	3
X	
x and z axes	
jogging	58
x offset to centerline	
Hybrid BOT and VDI	105
setting	105

