



Haas Automation, Inc.

Lathe Operator's Manual

96-8900
Revision C
July 2015
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

© 2015 Haas Automation, Inc.

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

LIMITED WARRANTY CERTIFICATE

Haas Automation, Inc.

Covering Haas Automation, Inc. CNC Equipment

Effective September 1, 2010

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

Limited Warranty Coverage

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

Repair or Replacement Only

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

Disclaimer of Warranty

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

Limits and Exclusions of Warranty

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

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

Limitation of Liability and Damages

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

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

Entire Agreement

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

Transferability

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

Miscellaneous

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

Customer Feedback

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

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



diy.haascnc.com

The Haas Resource Center: Documentation and Procedures



atyourservice.haascnc.com

At Your Service: The Official Haas Answer and Information Blog



haasparts.com

Your Source for Genuine Haas Parts



www.facebook.com/HaasAutomationInc

Haas Automation on Facebook



www.twitter.com/Haas_Automation

Follow us on Twitter



www.linkedin.com/company/haas-automation

Haas Automation on LinkedIn



www.youtube.com/user/haasautomation

Product videos and information



www.flickr.com/photos/haasautomation

Product photos and information

Customer Satisfaction Policy

Dear Haas Customer,

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

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

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

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

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

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

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

International:

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

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

Declaration of Conformity

Product: 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 2014 / 30 / EU
- Low Voltage Directive 2014 / 35 / EU
- Additional Standards:
 - EN 60204-1:2006 / A1:2009
 - EN 614-1:2006+A1:2009
 - EN 894-1:1997+A1:2008
 - CEN 13849-1:2015

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

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

Person authorized to compile technical file:

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.	Service > <i>Documents and Software</i> >...
A Mode Reference describes a machine mode.	MDI
A Screen Element describes an object on the machine's display that you interact with.	Select the SYSTEM tab.
System Output describes text that the machine control displays in response to your actions.	PROGRAM END
User Input describes text that you should enter into the machine control.	G04 P1. ;
Variable n indicates a range of non-negative integers from 0 to 9.	Dnn represents D00 through D99.

Contents

Chapter 1	Safety	1
1.1	General Safety Notes	1
1.1.1	Read Before Operating	1
1.1.2	Machine Environmental Limits	4
1.1.3	Machine Noise Limits	4
1.2	Unattended Operation	4
1.3	Setup Mode	5
1.3.1	Machine Behavior with the Door Open	5
1.3.2	Robot Cells	6
1.4	Modifications to the Machine	6
1.5	Improper Coolants	7
1.6	Safety Decals	8
1.6.1	Warning Decals	9
1.6.2	Other Safety Decals	10
1.7	More Information Online	11
Chapter 2	Introduction	13
2.1	Lathe Orientation	13
2.2	Control Pendant	19
2.2.1	Pendant Front Panel	19
2.2.2	Pendant Right Side, Top, and Bottom Panels	20
2.2.3	Keyboard	22
2.2.4	Control Display	34
2.2.5	Screen Capture	47
2.3	Tabbed Menu Basic Navigation	48
2.4	Help	48
2.4.1	The Help Tabbed Menu	49
2.4.2	Search Tab	49
2.4.3	Help Index	50
2.4.4	Drill Table Tab	50
2.4.5	Calculator Tab	50
2.5	More Information Online	55
Chapter 3	Control Icons	57
3.1	Introduction	57
3.2	Control Icon Guide	58
3.3	More Information Online	67

Chapter 4 Operation	69
4.1 Machine Power-On	69
4.2 Device Manager	70
4.2.1 File Directory Systems	71
4.2.2 Program Selection	72
4.2.3 Program Transfer	72
4.2.4 Deleting Programs	73
4.2.5 Maximum Number of Programs	74
4.2.6 File Duplication	74
4.2.7 Changing Program Numbers	75
4.3 Backing Up Your Machine	75
4.3.1 Making a Backup	76
4.3.2 Restoring From a Backup	77
4.4 Basic Program Search	77
4.5 RS-232	78
4.5.1 Cable Length	78
4.5.2 Machine Data Collection	78
4.6 File Numerical Control (FNC)	81
4.7 Direct Numerical Control (DNC)	81
4.7.1 DNC Notes	83
4.8 Jog Mode	83
4.9 Setting the Tool Offset	83
4.10 Manually Set the Tool Offset	84
4.11 Hybrid Turret, VDI, and BOT Centerline Offset	85
4.12 Additional Tooling Setup	85
4.13 Part Setup	85
4.13.1 Chuck Foot Pedal	86
4.13.2 Chuck/Drawtube Warnings	86
4.13.3 Drawtube Operation	88
4.13.4 Chuck and Collet Replacement	89
4.13.5 Steady Rest Foot Pedal	91
4.14 Tailstock Setup and Operation	92
4.14.1 Tailstock Types	92
4.14.2 ST-20/30/40 Tailstock Operation	95
4.14.3 Tailstock Restricted Zone	97
4.14.4 Jogging the Tailstock	99
4.15 Tool Turret Operations	99
4.15.1 Air Pressure	99
4.15.2 Eccentric Locating Cam Buttons	100
4.15.3 Protective Cap	100
4.15.4 Tool Load or Tool Change	101
4.16 Setting Part Zero for the Z-axis (Part Face)	101
4.17 Features	102

4.17.1	Graphics Mode	102
4.17.2	Dry Run Operation	103
4.17.3	Axis Overload Timer	103
4.18	Running Programs	104
4.19	Run-Stop-Jog-Continue	104
4.20	More Information Online	105
Chapter 5	Programming	107
5.1	Numbered Programs	107
5.2	Program Editors	107
5.2.1	Basic Program Editing	107
5.2.2	Background Edit	108
5.2.3	Manual Data Input (MDI)	109
5.2.4	Advanced Editor	110
5.2.5	The File Numerical Control (FNC) Editor	118
5.3	Tips and Tricks	128
5.3.1	Programming	128
5.3.2	Offsets	129
5.3.3	Settings and Parameters	130
5.3.4	Operation	130
5.3.5	Calculator	131
5.4	Program Optimizer	131
5.4.1	Program Optimizer Operation	132
5.5	DXF File Importer	133
5.5.1	Part Origin	133
5.5.2	Part Geometry Chain and Group	134
5.5.3	Toolpath Selection	134
5.6	Basic Programming	135
5.6.1	Preparation	136
5.6.2	Cutting	137
5.6.3	Completion	137
5.6.4	Absolute vs. Incremental (XYZ vs. UVW)	138
5.7	Miscellaneous Codes	138
5.7.1	Tool Functions	138
5.7.2	Spindle Commands	140
5.7.3	Program Stop Commands	140
5.7.4	Coolant Commands	140
5.8	Cutting G-codes	141
5.8.1	Linear Interpolation Motion	141
5.8.2	Circular Interpolation Motion	141
5.9	Tool Nose Compensation	143
5.9.1	Programming	143
5.9.2	Tool Nose Compensation Concept	145

5.9.3	Using Tool Nose Compensation	146
5.9.4	Approach and Departure Moves For TNC	147
5.9.5	Tool Nose Radius and Wear Offset	148
5.9.6	Tool Nose Compensation and Tool Length Geometry	149
5.9.7	Tool Nose Compensation in Canned Cycles	150
5.9.8	Example Programs Using Tool Nose Compensation	150
5.9.9	Imaginary Tool Tip and Direction	159
5.9.10	Programming Without Tool Nose Compensation	160
5.9.11	Manually Calculating Compensation	160
5.9.12	Tool Nose Compensation Geometry	161
5.10	Coordinate Systems	173
5.10.1	Effective Coordinate System	173
5.10.2	Automatic Setting of Tool Offsets	174
5.10.3	Global Coordinate System (G50)	175
5.11	Live Image	175
5.11.1	Live Image Stock Setup	175
5.11.2	Program Example	176
5.11.3	Live Image Tool Setup	177
5.11.4	Tailstock Setup (Live Image)	180
5.11.5	Operation	181
5.11.6	Run Part	182
5.11.7	Flipping a Part	184
5.12	Tailstock Setup and Operation	184
5.12.1	M-code Programming	185
5.13	Subprograms	185
5.14	More Information Online	186
Chapter 6	Options Programming	187
6.1	Introduction	187
6.2	Macros (Optional)	187
6.2.1	Macros Introduction	187
6.2.2	Operation Notes	189
6.2.3	System Variables In-Depth	201
6.2.4	Variable Usage	209
6.2.5	Address Substitution	210
6.2.6	G65 Macro Subprogram Call Option (Group 00)	220
6.2.7	Communication With External Devices - DPRNT[]	221
6.2.8	Fanuc-Style Macros Not Included	223
6.3	Y Axis	224
6.3.1	Y-Axis Travel Envelopes	225
6.3.2	Y-Axis Lathe with VDI Turret	226
6.3.3	Operation and Programming	226
6.4	Live Tooling	229

6.4.1	Live Tooling Introduction	229
6.4.2	Live Tooling Cutting Tool Installation	230
6.4.3	Live Tool Mounting in Turret	231
6.4.4	Live Tooling M-codes	232
6.5	C Axis	233
6.5.1	Cartesian to Polar Transformation (G112)	233
6.5.2	Cartesian Interpolation	233
6.6	Dual-Spindle Lathes (DS-Series)	237
6.6.1	Synchronized Spindle Control	237
6.6.2	Secondary Spindle Programming	240
6.7	More Information Online	241
Chapter 7	G-codes	243
7.1	Introduction	243
7.1.1	List of G-codes	243
7.2	More Information Online	335
Chapter 8	M-codes	337
8.1	Introduction	337
8.1.1	List of M-codes	337
8.2	More Information Online	355
Chapter 9	Settings	357
9.1	Introduction	357
9.1.1	List of Settings	357
9.2	More Information Online	395
Chapter 10	Maintenance	397
10.1	Introduction	397
10.2	Maintenance Monitor	397
10.2.1	Maintenance Settings	397
10.2.2	The Maintenance Monitor Page	398
10.2.3	Start, Stop, or Adjust Maintenance Monitoring	399
10.3	More Information Online	400
Chapter 11	Other Equipment	401
11.1	Introduction	401
11.2	Office Lathe	401
11.3	Toolroom Lathe	401
11.4	More Information Online	401
Index		403

Chapter 1: Safety

1.1 General Safety Notes



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

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



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

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

1.1.1 Read Before Operating



DANGER: *Do not enter the machining area any time the machine is in motion. Severe injury or death may result.*

Basic safety:

- Consult your local safety codes and regulations before operating the machine. Contact your dealer any time 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 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.
- 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.
- Replace windows immediately if they are damaged or severely scratched.

Electrical safety:

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

- Improperly clamped parts machined at high speeds/feeds may be ejected and puncture the enclosure. It is not safe to machine oversized or marginally clamped parts.

Follow these guidelines 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).
- 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.
- 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]**).

Chuck safety:



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

- 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).

1.1.2 Machine Environmental Limits

This table lists the environmental limits for safe operation:

T1.1: Environmental Limits (Indoor Use Only*)

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

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

1.1.3 Machine Noise Limits



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

A person at a typical operator's position is subject to noise levels from 70 dB to 85 dB or more during machine operation.

1.2 Unattended Operation

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

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

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

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

1.3 Setup Mode

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

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

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



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

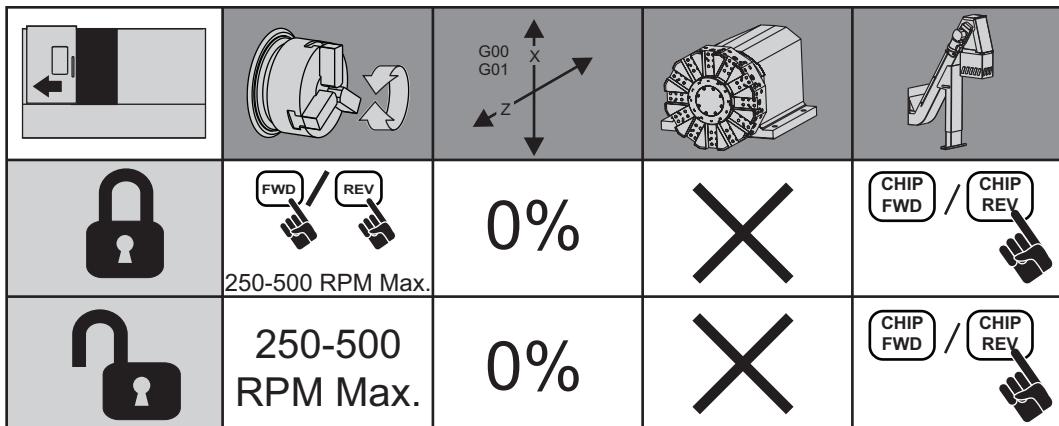
1.3.1 Machine Behavior with the Door Open

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

T1.2: 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	Not allowed.	Not allowed.

Machine Function	Locked (Run Mode)	Unlocked (Setup Mode)
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.



1.3.2 Robot Cells

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

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

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

1.4 Modifications to the Machine

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.

1.5 Improper Coolants

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

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

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

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

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

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



1.6 Safety Decals

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

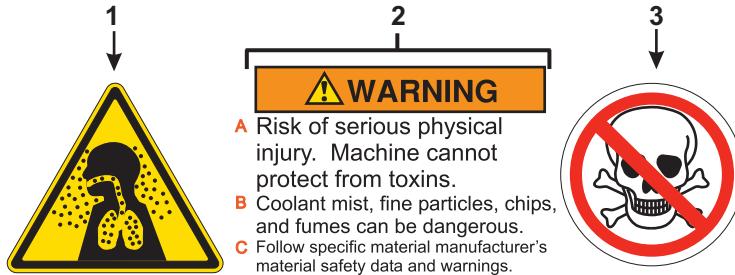


NOTE:

Never alter or remove any safety decal or symbol.

Each hazard is defined and explained on the general safety decal at the front of the machine. Review and understand each safety warning, and familiarize yourself with the symbols.

F1.1: Standard Warning Layout. [1] Warning Symbol, [2] Severity and Word Message, [3] Action Symbol. [A] Hazard Description, [B] Consequence of Ignoring the Warning, [C] Action to Prevent Injury.



1.6.1 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.

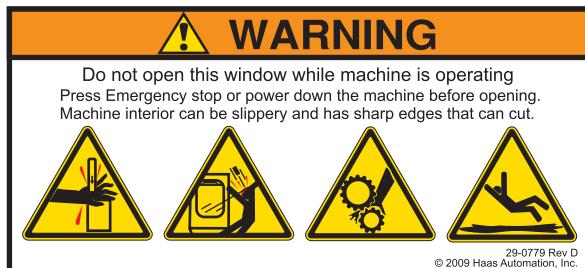
F1.2: Lathe General Warning Decal



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.

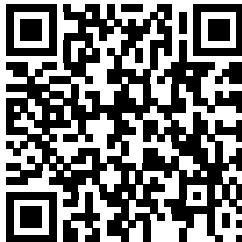
F1.3: Other Safety Decal Examples



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**.

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

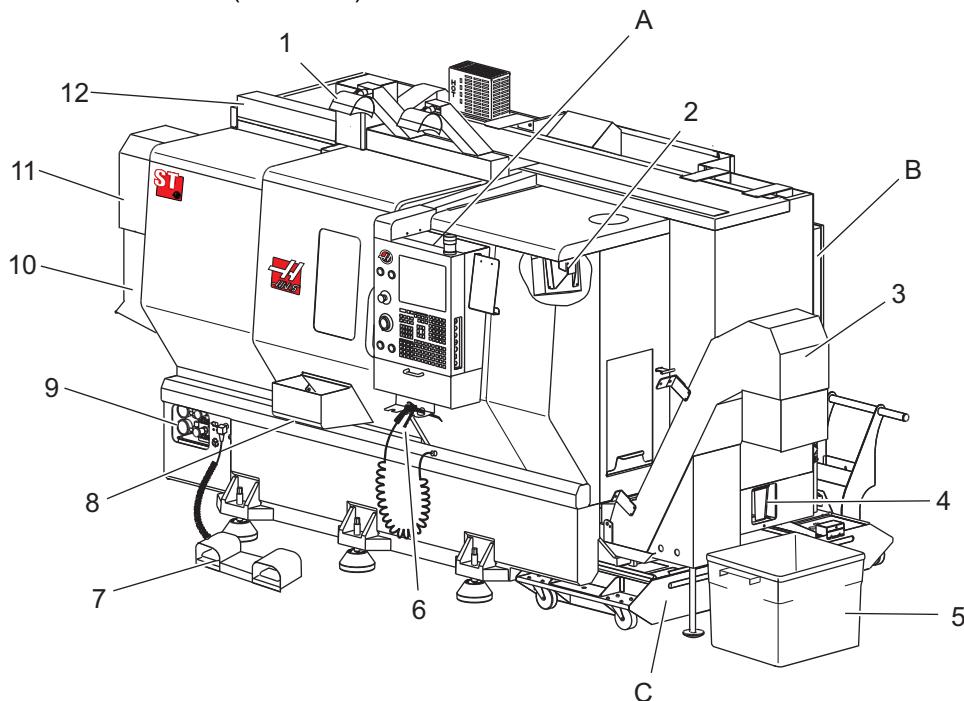


Chapter 2: Introduction

2.1 Lathe Orientation

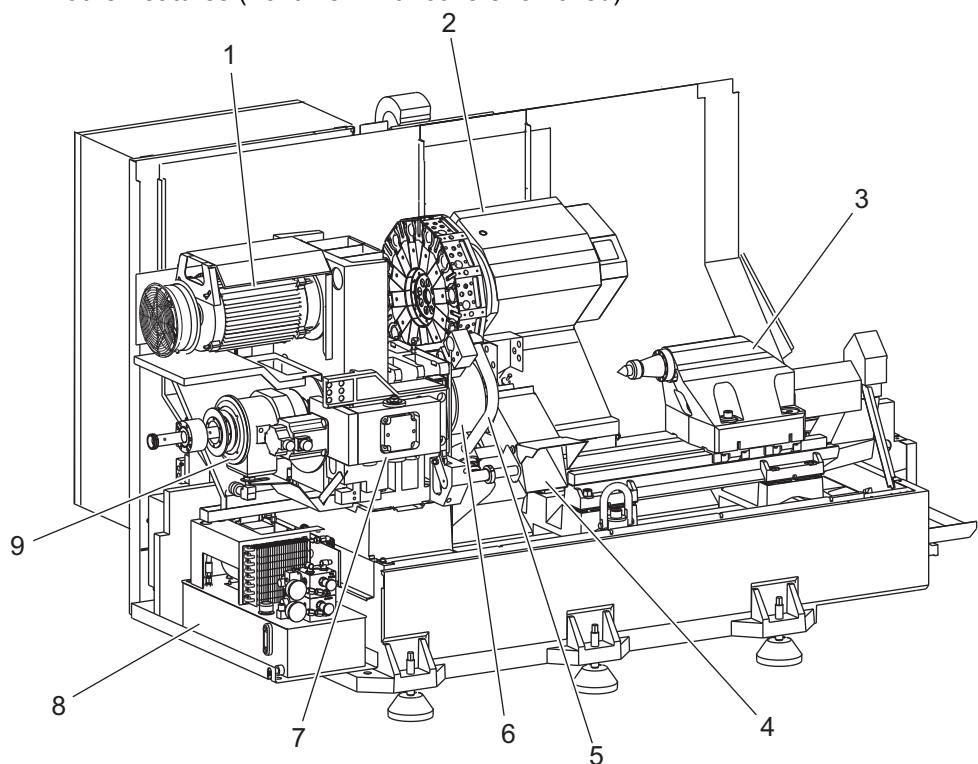
These figures show some of the standard and optional features of your Haas lathe. Some of the features shown are highlighted in their appropriate sections. Note that these figures are representative only; your machine's appearance may vary depending on the model and installed options.

F2.1: Lathe Features (front view)



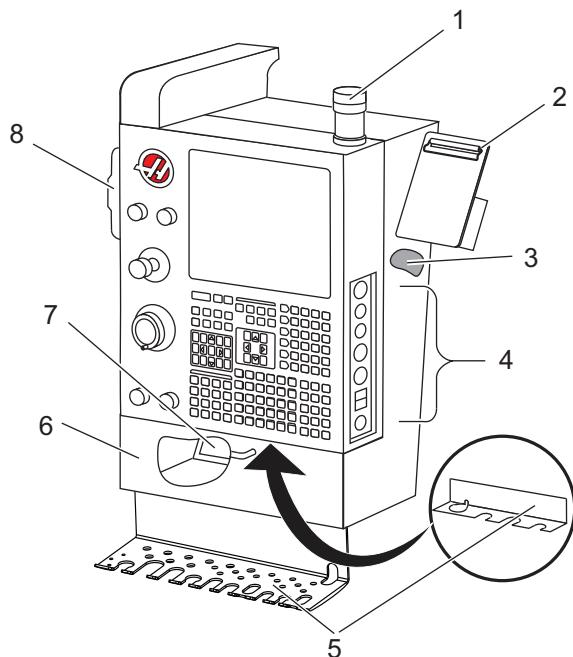
- | | |
|--|--------------------------------|
| 1. 2X High Intensity Lights (Optional) | 9. Hydraulic Power Unit (HPU) |
| 2. Work Light (2X) | 10. Coolant Collector |
| 3. Chip Conveyor (Optional) | 11. Spindle Motor |
| 4. Oil Drain Container | 12. Servo Auto Door (Optional) |
| 5. Chip Container |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

F2.2: Lathe Features (front view with covers removed)



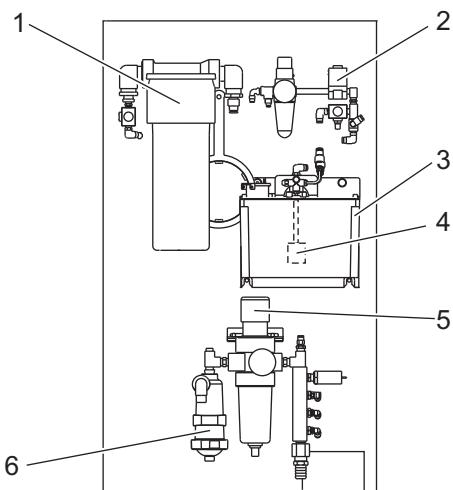
- 1. Spindle Motor
- 2. Tool Turret Assembly
- 3. Tailstock (Optional)
- 4. Parts Catcher (Optional)
- 5. LTP Arm (Optional)
- 6. Chuck
- 7. C-Axis Drive Assembly (Optional)
- 8. Hydraulic Power Unit (HPU)
- 9. Spindle Head Assembly
 - A Control Cabinet
 - B Control Cabinet Side Panel

F2.3: Lathe Features (front view) Detail A - Control Pendant with Cabinet



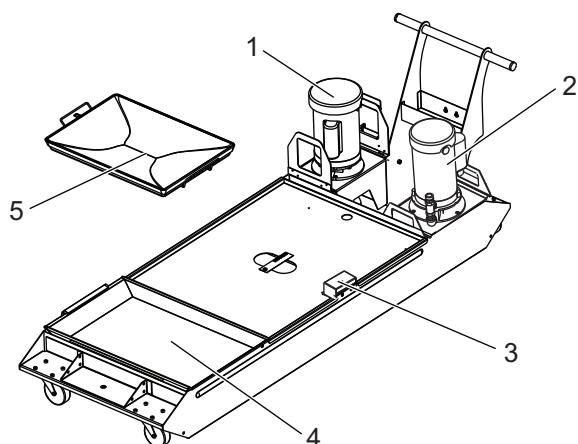
1. Work Beacon
2. Clipboard
3. Operator's Manual and Assembly Data (stored behind pendant)
4. Side panel Controls
5. Tool Holder (also shown, tool holder for thin pendant)
6. Storage Tray
7. G- and M-code Reference List
8. Remote Jog Handle

F2.4: Lubrication Panel Example

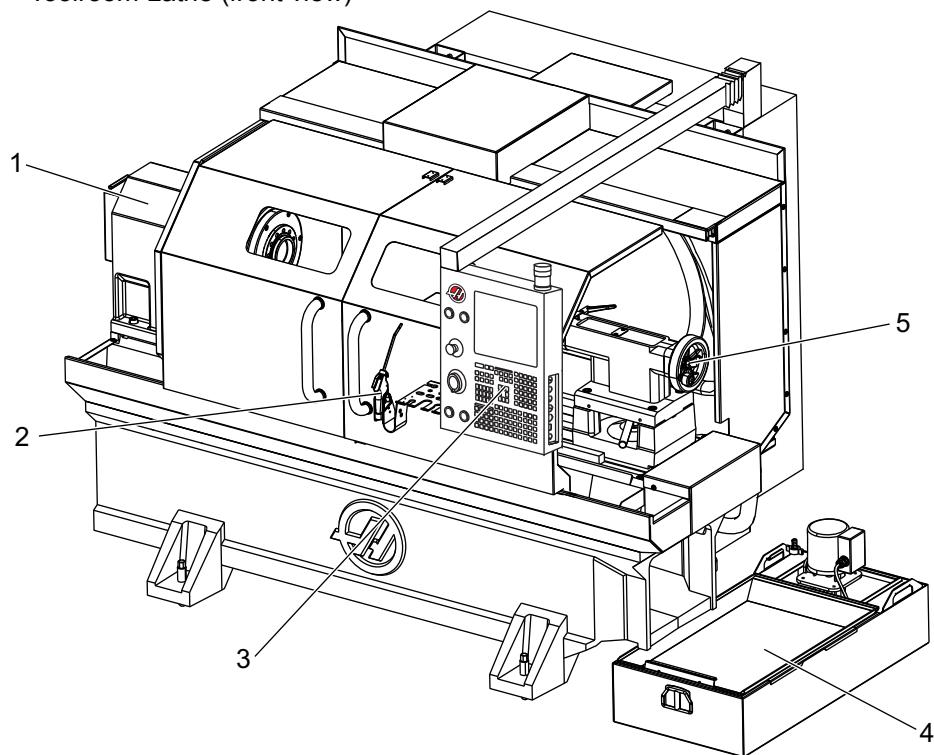


1. Grease Reservoir Assembly
2. Spindle Air and Pump Control
3. Spindle Oil Tank Pump Assembly
4. Spindle Pump Assembly
5. Main Regulator Air Manifold Assembly
6. Water Separator Assembly

F2.5: Lathe Features (3/4 side view) Detail C - Coolant Tank Assembly

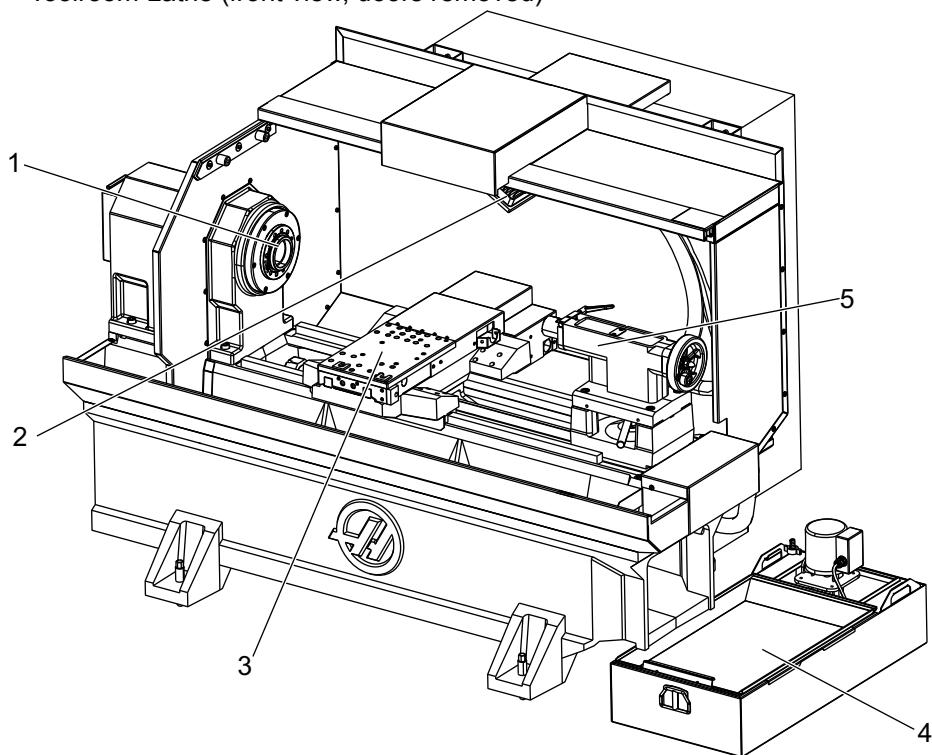


1. Standard Coolant Pump
2. High-Pressure Coolant Pump
(Optional)
3. Coolant Level Sensor
4. Chip Strainer
5. Strainer Basket

F2.6: Toolroom Lathe (front view)

1. Spindle assembly
2. Air Gun
3. Control Pendant
4. Coolant Tank
5. Tailstock

F2.7: Toolroom Lathe (front view, doors removed)



1. Spindle Nose
2. Worklight
3. Cross Slide (tool post / turret not shown)
4. Coolant Tank
5. Tailstock

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 Pendant Front Panel

T2.1: 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.
[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.

2.2.2 Pendant Right Side, Top, and Bottom Panels

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

T2.2: Right Side Panel Controls

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

T2.3: Pendant Top Panel

Beacon Light	
Provides quick visual confirmation of the machine's current status. There are five different beacon states:	
Light Status	Meaning
Off	The machine is idle.
Solid Green	The machine is running.

Beacon Light	
Flashing Green	The machine is stopped, but is in a ready state. Operator input is required to continue.
Flashing Red	A fault has occurred, or the machine is in Emergency Stop.
Flashing Yellow	A tool has expired, and the tool life screen automatically displays.

T2.4: Pendant Bottom Panel

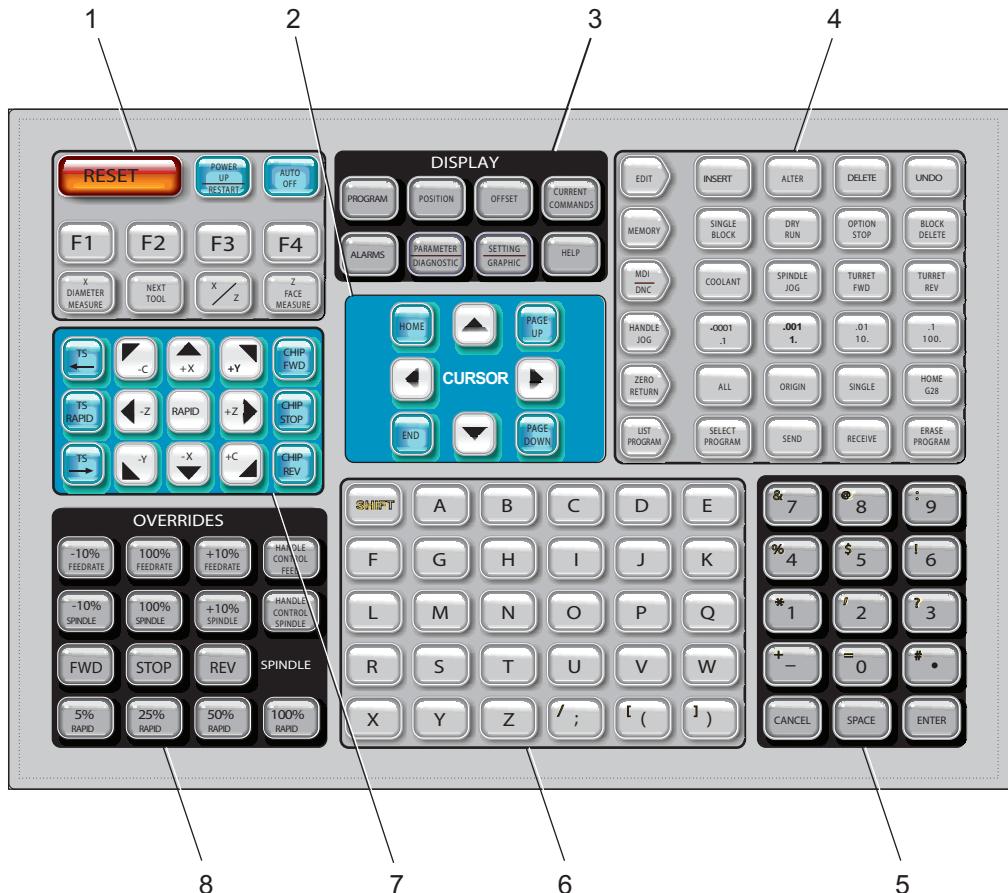
Name	Function
Keyboard Beeper	Located on the bottom of the control pendant. Turn the cover to adjust the volume.

2.2.3 Keyboard

Keyboard keys are grouped into these functional areas:

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

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



Function Keys

Name	Key	Function
Reset	[RESET]	Clears alarms. Sets overrides to default values.
Power up/Restart	[POWER UP/RESTART]	Sends the axes to their home positions. Clears alarm 102. Displays Current Commands page.
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]	Records X-Axis tool shift offsets on the offset page during part setup.
Next Tool	[NEXT TOOL]	Selects the next tool from the turret (usually used during part setup).
X/Z	[X/Z]	Toggles 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

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

T2.5: Cursor Key List

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

Display Keys

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

T2.6: List of Display Keys and How They Operate

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

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

Mode Keys

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

T2.7: List of [EDIT] Mode Keys and How They Operate

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

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

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

T2.9: List of [MDI/DNC] Mode Keys and How They Operate

Name	Key	Function
Manual Data Input/Direct Numeric Control	[MDI/DNC]	In MDI mode, you can write a program, but it is not entered into memory. DNC mode lets you "drip feed" large programs into the control (refer to the DNC mode section).
Coolant	[COOLANT]	Turns the optional coolant on and off. Press [SHIFT] and then [COOLANT] to turn on the optional High-Pressure Coolant (HPC). Because HPC and regular coolant share a common orifice, you cannot activate both 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.

T2.10: List of **[HAND JOG]** Mode Keys and How They Operate

Name	Key	Function
.0001/.1	[.0001 /.1], [.001 / 1], [.01 / 10], [.1 / 100]	Selects that amount to be jogged for each click of the jog handle. When the mill is in MM mode the first number is multiplied by ten when jogging the axis (e.g., .0001 becomes 0.001mm). The bottom number is used for dry run mode. Shows <i>SETUP: JOG</i> in upper left display.

T2.11: List of **[ZERO RETURN]** Mode Keys and How They Operate

Name	Key	Function
Zero Return	[ZERO RETURN]	Selects Zero Return mode, which displays axis location in four different categories, they are; Operator, Work G54, Machine, and Dist (distance) To Go. Press [POSITION] or [PAGE UP]/[PAGE DOWN] to switch between the categories. Shows <i>SETUP: ZERO</i> in upper left display.
All	[ALL]	Returns all axes to machine zero. This is similar to [POWER UP/RESTART] , except a tool change does not occur.
Origin	[ORIGIN]	Sets selected values to zero.
Single	[SINGLE]	Returns one axis to machine zero. Press the desired axis letter on the Alpha keyboard and then press [SINGLE] .
Home G28	[HOME G28]	Returns all axes to zero in rapid motion. [HOME G28] will also home a single axis in the same manner as [SINGLE] .



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

T2.12: List of [LIST PROGRAM] Mode Keys and How They Operate

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

Numeric Keys

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

T2.13: List of Numeric Keys and How They Operate

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

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

Alpha Keys

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

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

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

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

Jog Keys

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.
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

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%.

Name	Key	Function
Handle Control Feedrate	[HANDLE CONTROL FEED]	Lets you 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]	Lets you use the jog handle to change spindle speed in $\pm 1\%$ increments, from 0% to 999%.
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 let you temporarily adjust the speeds and feeds in your program. For example, you can slow down rapids while you prove out a program, or adjust the feedrate to experiment with its effect on part finish, etc.

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

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

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

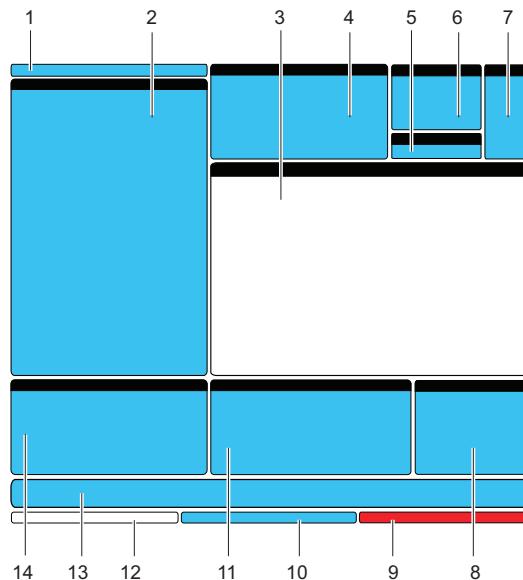
You can override the standard coolant setting by pressing [**COOLANT**]. The coolant pump remains either on or off until the next M-code or operator action (see Setting 32).

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

2.2.4 Control Display

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

F2.9: 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.

- F2.10:** The Mode and Display bar shows [1] the current mode and [2] the current display function.



- T2.15:** 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.

T2.16: Offset Tables

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

Current Commands

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

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

Operation Timers and Setup Display - This page shows:

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

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

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

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

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

**NOTE:**

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

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

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

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

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

Timer and Counter Reset

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

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

:

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

Date and Time Adjustment

To adjust the Date and Time:

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

**NOTE:**

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

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

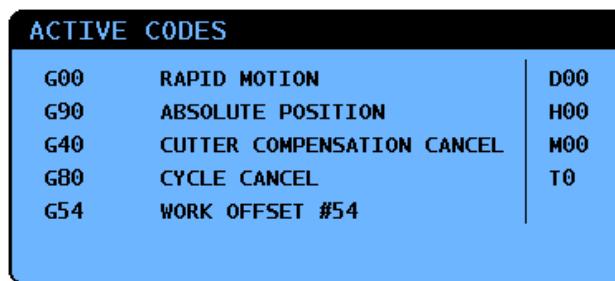
Setting/Graphic Display Function

Press **[SETTING/GRAFIC]** until you see the Setting screen. Settings change the way the lathe behaves; refer to the “Settings” section starting on page **357** for a more detailed description.

To use Graphics mode, press **[SETTING/GRAFIC]** until you see the Graphics screen. Graphics shows a visual dry run of your part program without the need to move the axes and risk tool or part damage from programming errors. This function is more useful than the Dry Run mode, because you can check all of your work offsets, tool offsets, and travel limits before running the machine. The risk of a crash during setup is greatly reduced. Refer to Graphics Mode on page **102** for a more detailed description.

Active Codes

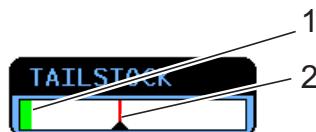
F2.11: Active Codes Display Example



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

F2.12: Tailstock Display Example



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

Active Tool

F2.13: Active Tool Display Example



Coolant Display

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

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

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

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

Timers & Counters Display

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

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

- M30 Counter #1: and M30 Counter #2: every time a program reaches an **M30** command, the counters increase by one. If Setting 118 is on, the counters also increment every time a program reaches an **M99** command.
- If you have macros, you can clear or change M30 Counter #1 with #3901 and M30 Counter #2 with #3902 (#3901=0).
- Refer to page **37** 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.

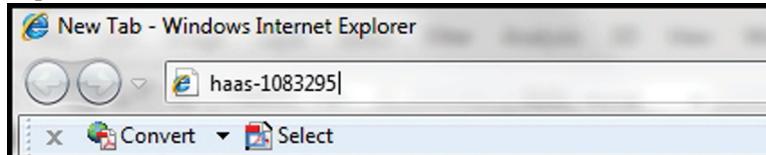
Press **[CANCEL]** or **[SPACE]** to delete existing characters. Press **[DELETE]** to delete an entire line. Your message data is automatically stored and maintained even in a power-off state.

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.

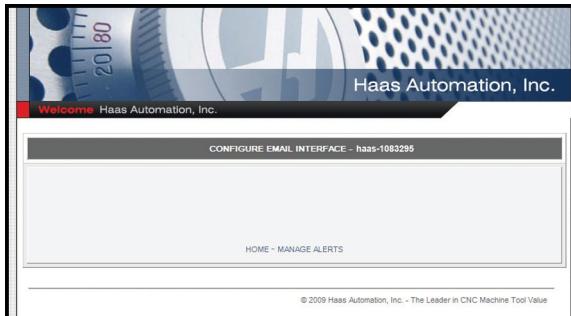
Before you set up alerts, make sure the machine has an established connection to your Local Area Network, and that Setting 900 defines a unique network name for the machine. This feature requires the Ethernet option and software version 18.01 or later.

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]**.

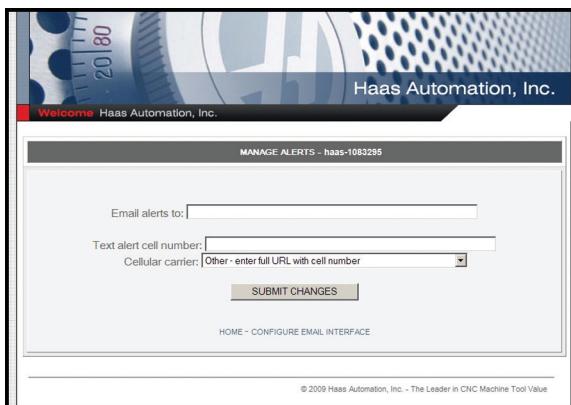


2. A message may appear with a request to set a cookie in your browser. This will happen each time you access the machine using a different computer or browser, or after an existing cookie has expired. Click **OK**.

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



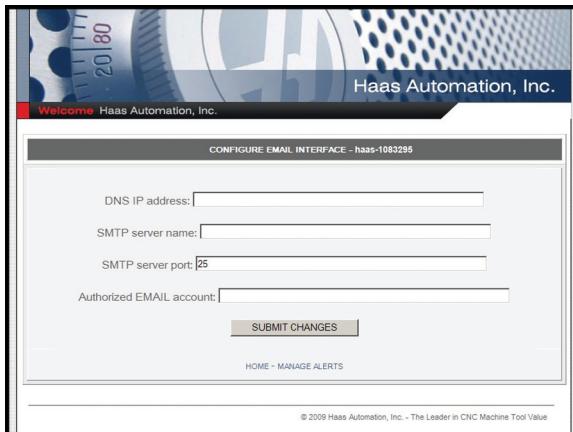
4. At the Manage Alerts screen, enter the email address and/or cellular phone number where you want to receive alerts. If you enter a cellular phone number, select your carrier from the pull-down menu under the cell number field. Click **SUBMIT CHANGES**.



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.



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.

T2.17: Axis Position Reference Points

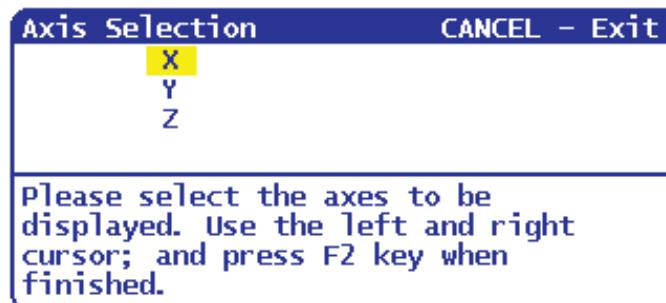
Coordinate Display	Function
OPERATOR	This position shows the distance you have jogged the axes. This does not necessarily represent the actual distance the axis is from machine zero, except when the machine is first powered on.
WORK (G54)	This displays the axis positions relative to part zero. On power-up,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

Position Display Axis Selection

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

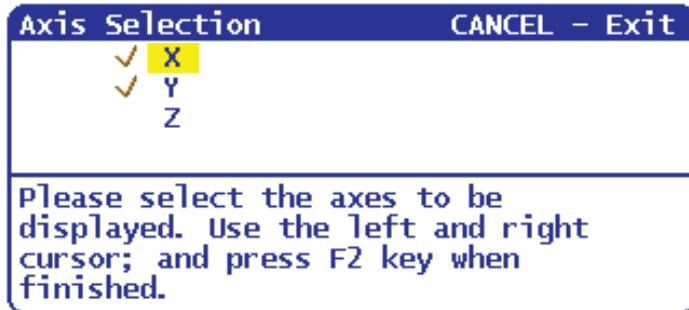
- With a position display active, press **[F2]**. The **Axis Selection** pop-up menu appears.

F2.14: The Axis Selection Pop-up Menu



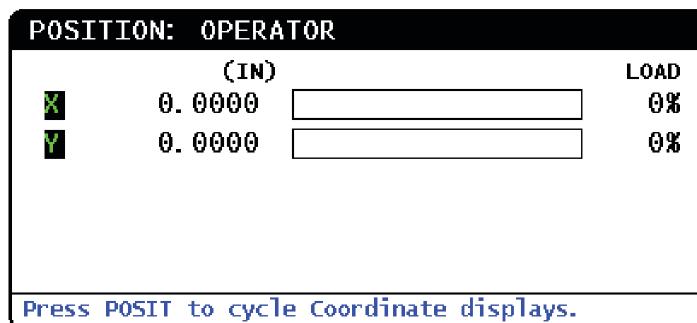
2. Press the [LEFT], [RIGHT], [UP], or [DOWN] cursor arrow keys to highlight an axis letter.
3. Press [ENTER] to place a check mark next to the highlighted axis letter. This mark means that you want to include that axis letter in the position display.

F2.15: The X and Y Axes Selected in the Axis Selection Menu



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

F2.16: The Updated Position Display



Input Bar

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

F2.17: Input Bar



Special Symbol Input

Some special symbols are not on the keypad.

T2.18: Special Symbols

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

Do these steps to enter special symbols:

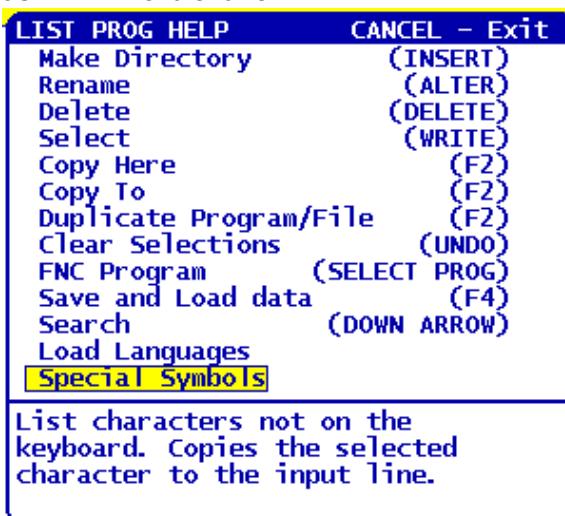


NOTE:

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

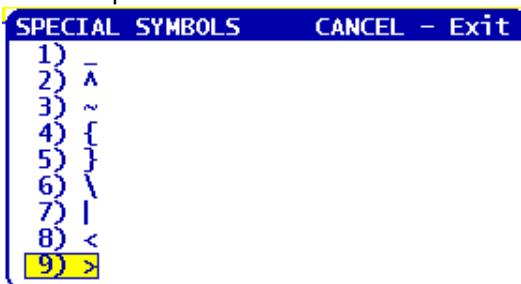
1. Press [LIST PROGRAMS] and select USB DEVICE or optional HARD DRIVE.
2. Press [F1].

The LIST PROG HELP menu shows:



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

The SPECIAL SYMBOLS pick list shows:



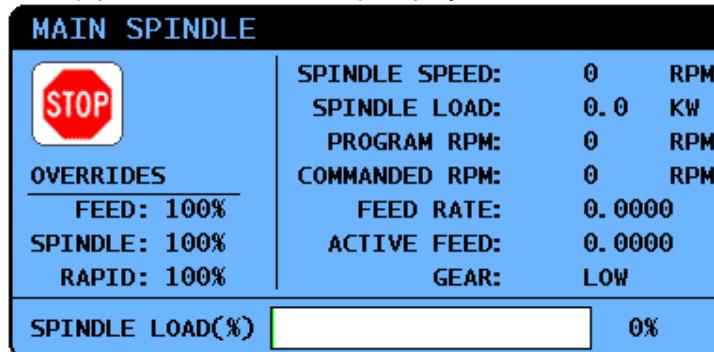
4. Select the symbol and press **[ENTER]** to copy the symbol to the INPUT: bar.

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

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

Main Spindle Display

- F2.18: 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 work piece. It also displays current programmed and commanded 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.

2.2.5 Screen Capture

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

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



NOTE:

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

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

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

2.3 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:**

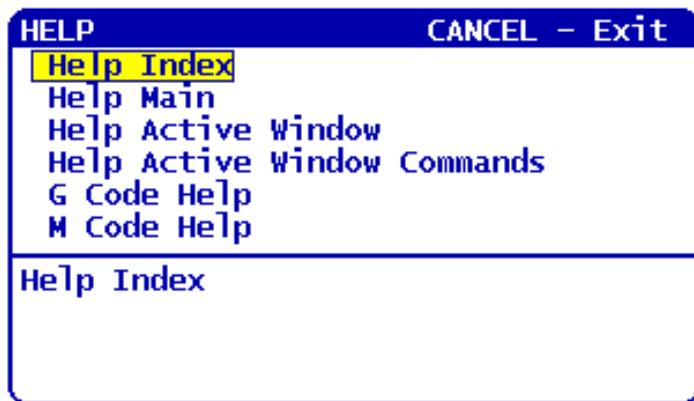
In the tabbed menus for parameters and settings, and in the ALARM VIEWER section of the Alarm / Messages display, you can type the number of a parameter, setting, or alarm that you want to view, then press the [UP] or [DOWN] cursor arrow to view it.

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

F2.19: 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 50.

- **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 **48**.

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 **50**.
- **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 **50** 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 /). Numbers are entered for calculation from the input bar after pressing [ENTER].

1. LOAD and the calculator window are initially highlighted. The other options can be selected with [LEFT]/[RIGHT] cursors. Numbers are entered by typing them and pressing [ENTER]. When a number is entered and LOAD and the calculator window are highlighted, that number is entered into the calculator window.
2. When a number is entered after one of the other functions (+, -, *, /) is selected, the calculation is performed with the number just entered and any number that was already in the calculator window.
3. The calculator also accepts a mathematical expression in the input bar. For example, type $23*4-5.2+6/2$ and press [ENTER]. The control evaluates this expression by doing multiplication and division first and then subtraction and addition. The result, 89.8, displays in the window. No exponents are allowed.

**NOTE:**

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

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

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

5. **[F3]:** In EDIT and MDI modes, **[F3]** copies the highlighted triangle/circular milling/tapping value into the data entry line at the bottom of the screen. This is useful when the calculated solution is used in a program.
6. In the Calculator function, pressing **[F3]** copies the value in the calculator window to the highlighted data entry for Trig, Circular or Milling/Tapping calculations.
7. **[F4]:** In the Calculator function, this button uses the highlighted Trig, Circular or Milling/Tapping data value to load, add, subtract, multiply, or divide with the calculator.

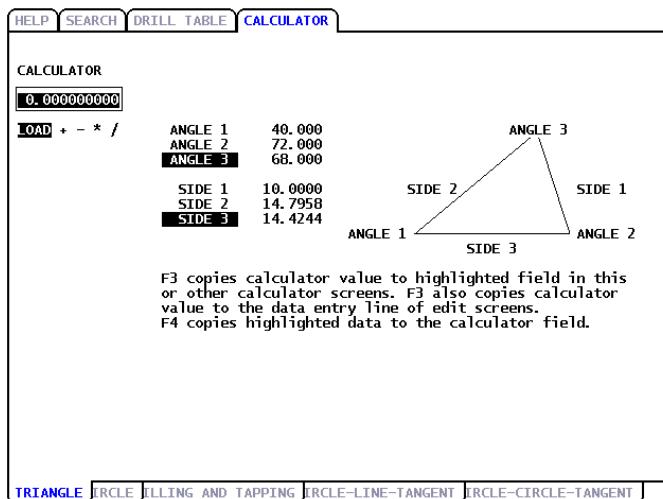
Triangle Sub-tab

The triangle calculator page takes a few triangle measurements and solves for the rest of the values. For inputs that have more than one solution, entering the last data value a second time will cause the next possible solution to be displayed.

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

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

F2.20: Calculator Triangle Example



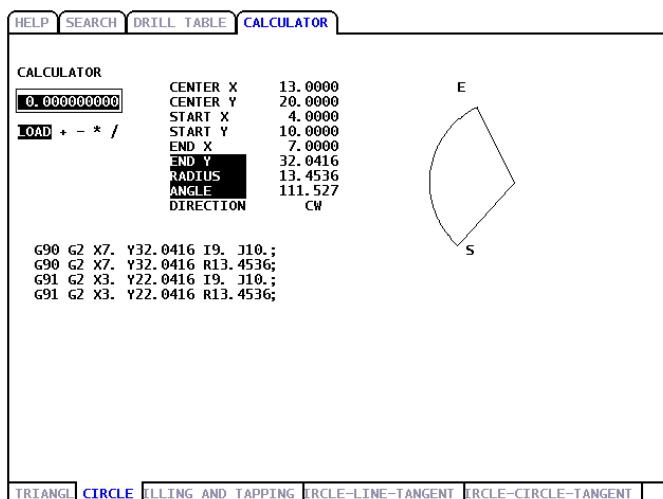
Circle Sub-tab

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.

F2.21: Calculator Circle Example



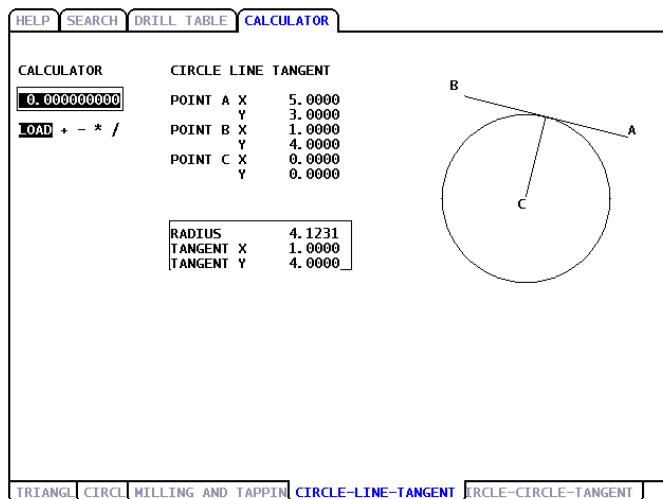
Circle-Line-Tangent Sub-tab

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

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

The control calculates the point of intersection. The point is where a normal line from point C will intersect with the line AB, as well as the perpendicular distance to that line.

F2.22: 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:

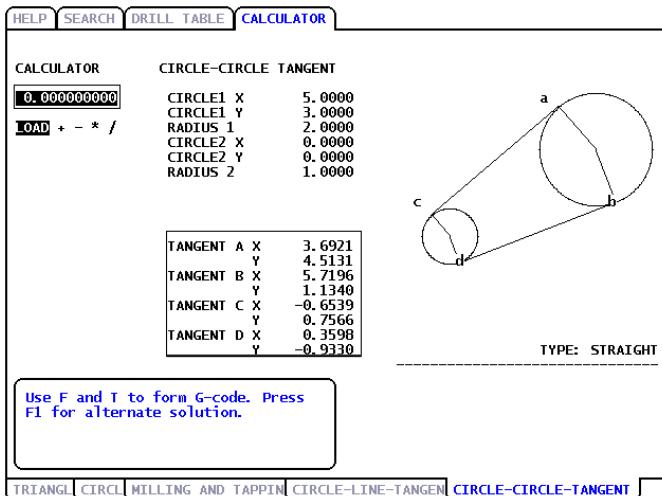
For every input condition (two disjointed circles), there are up to eight intersection points. Four points are from drawing straight tangents and four points by forming cross tangents.

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.

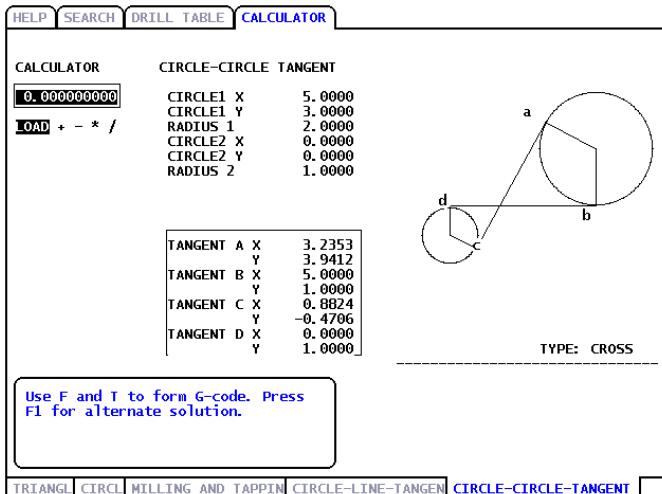
F2.23: Calculator Circle-Circle-Tangent Type: Straight Example



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

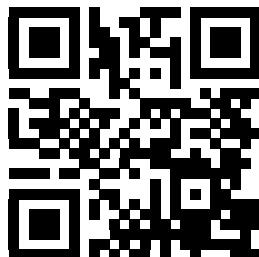
G01 X-4.346 Y-3.7565

F2.24: Calculator Circle-Circle-Tangent Type: Cross Example



2.5 More Information Online

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



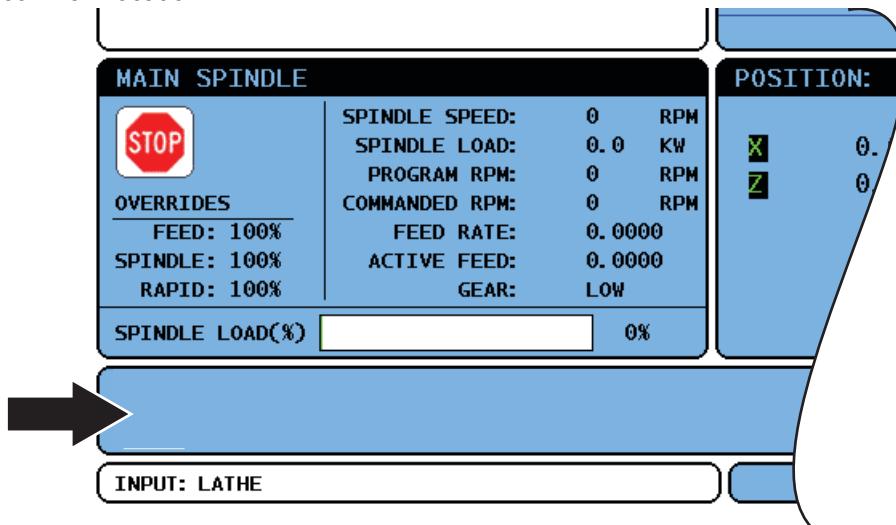
Chapter 3: Control Icons

3.1 Introduction

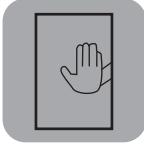
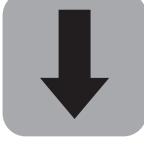
The control screen shows icons to quickly give you information about machine status. Icons tell you about current machine modes, your program as it runs, and machine maintenance status.

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

F3.1: Icon Bar Location



3.2 Control Icon Guide

Name	Icon	Meaning
SETUP KEY LOCKED		Setup mode is locked; the control is in “Run” mode. Most machine functions are disabled or limited while the machine doors are open.
SETUP KEY UNLOCKED		Setup mode is unlocked; the control is in “Setup” mode. Most machine functions are available, but may be limited, while the machine doors are open.
DOOR HOLD		Machine motion has stopped because of door rules.
RUNNING		The machine is running a program.
POWER SAVING SERVOS OFF		The power-saving servos off feature is active. Servos are turned off. The HPU pump is turned off. Press a key to activate the servos and the HPU pump.
JOG RETURN		This icon appears while the control returns to the workpiece during a run-stop-jog-continue operation.

Name	Icon	Meaning
JOG HOLD		You have pressed [FEED HOLD] during the return portion of a run-stop-jog-continue operation.
JOG AWAY		This icon prompts you to jog away during a run-stop-jog-continue operation.
RESTART		The control scans the program before a restart if Setting 36 is ON.
SINGBK STOP		SINGLE BLOCK mode is active, and the control needs a command to continue.
FEEDHOLD		The machine is in feed hold. Axis motion has stopped, but the spindle continues to turn.
FEED		The machine is executing a cutting move.
RAPID		The machine is executing a non-cutting axis move (G00) at the fastest possible rate.

Name	Icon	Meaning
DWELL		The machine is executing a dwell (G04) command.
RESTRICTED ZONE		A current axis position is in the restricted zone.
REMOTE JOG		The optional remote jog handle is active.
VECTOR JOG		An axis is jogging at the current jog rate.
G14		Secondary spindle swap with Z-Axis mirroring active.
X MIRROR		Mirroring mode is active in the negative direction. The icon message includes the currently mirrored axis.
MAIN SPINDLE UNCLAMPED		The lathe spindle brake is off. With C-Axis option, M15 or with a secondary spindle, M115 turns off spindle brake.

Name	Icon	Meaning
SPINDLE CLAMPED		The lathe spindle brake is on. With C-Axis option, M14 or with a secondary spindle, M114 turns on spindle brake.
WARNING LOW VOLTAGE		Power Fault Detect Module (PFDM) Incoming Voltage is under the nominal operating level.
WARNING HIGH VOLTAGE		PFDM Incoming Voltage is above the nominal operating level.
ALARM HIGH VOLTAGE		PFDM Incoming Voltage is above the nominal operating level.
ALARM LOW AIR PRESSURE		System air pressure is critically low.
WARNING LOW AIR PRESSURE		System air pressure is low.
WARNING HIGH AIR PRESSURE		System air pressure is high.

Name	Icon	Meaning
ALARM HIGH AIR PRESSURE		System air pressure is critically high
LOW GEAR BOX OIL FLOW LOW GEAR BOX OIL LEVEL		The spindle gear box oil level is low.
LOW COOLANT CONCENTRATE		The concentrate reservoir for the Coolant Refill system needs service.
LOW SPINDLE OIL LEVEL LOW SECOND SPINDLE OIL LOW GREASE LEVEL		The spindle lubrication oil system detected a low oil condition, or the axis ball screw lubrication system detected a low grease or low pressure condition. Refer to the note after this table.
MAINTENANCE DUE		A maintenance procedure is due, based on information in the MAINTENANCE page. The maintenance page is part of Current Commands.
WARNING LOW HPU OIL LEVEL		The Hydraulic Power Unit (HPU) oil level needs service
WARNING HIGH HPU OIL TEMPERATURE		The HPU oil temperature has reached the caution range.

Name	Icon	Meaning
ALARM HIGH HPU OIL TEMPERATURE		The HPU oil temperature has reached the alarm level.
BARFEED OUT OF POSITION		The Haas Bar Feeder is not correctly oriented or aligned with the lathe.
BARFEED SAFETY COVER OPEN		The Haas Bar Feeder cover is open. Bar loading will proceed at reduced rates and some operations will be prohibited.
EMERGENCY STOP, PENDANT		[EMERGENCY STOP] on the pendant has been pressed. This icon disappears when [EMERGENCY STOP] is released.
EMERGENCY STOP, BARFEED		[EMERGENCY STOP] on the bar feeder has been pressed. This icon disappears when [EMERGENCY STOP] is released.
EMERGENCY STOP, AUXILIARY 1		[EMERGENCY STOP] on an auxiliary device has been pressed. This icon disappears when [EMERGENCY STOP] is released.
EMERGENCY STOP, AUXILIARY 2		[EMERGENCY STOP] on an auxiliary device has been pressed. This icon disappears when [EMERGENCY STOP] is released.

Name	Icon	Meaning
SINGLE BLOCK		SINGLE BLOCK mode is active. The control executes programs (1) block at a time, and you need to press [CYCLE START] to execute the next block.
DRY RUN		DRY RUN mode is active.
OPTION STOP		OPTIONAL STOP is active. The control stops the program at each M01 command.
BLOCK DELETE		BLOCK DELETE is active. The control skips program blocks that begin with a slash (/).
TOOL CHANGE		A tool change is in progress.
PROBE DOWN		The probe arm is down for a probing operation.
PART CATCHER ON		The parts catcher is activated.

Name	Icon	Meaning
TS PART HOLDING		The tailstock is engaged with the part.
TS PART NOT HOLDING		The tailstock is not engaged with the part.
CONVEYOR FORWARD		The conveyor is active and currently moving forward.
CONVEYOR REVERSE		The conveyor is active and currently moving in reverse.
HIGH PRESSURE COOLANT		The High-Pressure Coolant system is active.
AIR BLAST ON		The Auto Jet Air Blast is active.

Name	Icon	Meaning
COOLANT ON		The main coolant system is active.
COOLANT REFILL ON		Coolant Refill feature is mixing and adding coolant to the tank.

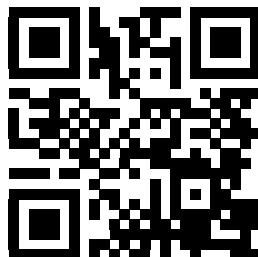
**NOTE:**

** - Axis grease message, for type 3, is Low Grease Level?. Axis grease messages, for type 5, depend on the state that is detected:*

- **The last lubrication cycle completed normally.**
- **The air pressure was low during the previous axis lubrication cycle.** Check that sufficient air pressure and volume is supplied to the machine whenever it is operating.
- **Axis lubrication pressure was not detected.** Refill the lubricant reservoir. If the reservoir has recently been refilled, this warning may appear for several lubrication cycles until the air has been purged from the system.
- **The lubrication pressure dropped faster than normal.** Refill the lubricant reservoir. If the reservoir has recently been refilled, this warning may appear for several lubrication cycles until the air has been purged from the system.?

3.3 More Information Online

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

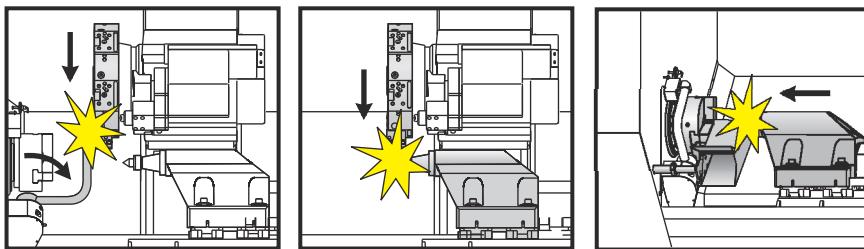


Chapter 4: Operation

4.1 Machine Power-On

Before you do this procedure, make sure that possible crash areas, such as the tool probe, parts catcher, tailstock, tool turret, and secondary spindle, are clear.

F4.1: Possible Crash Areas During Power-Up



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

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

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



WARNING:

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

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



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

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

4.2

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 **48**.

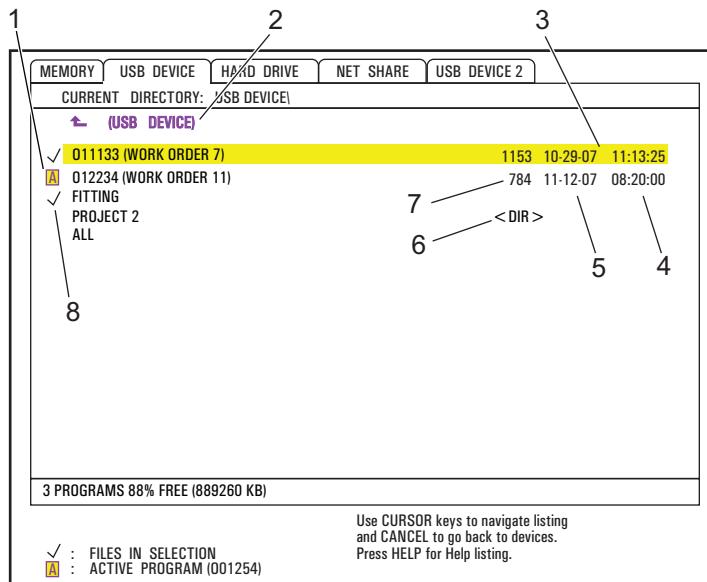


NOTE:

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

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

F4.2: USB Device Menu



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

4.2.1 File Directory Systems

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



NOTE:

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

Navigating Directories

To navigate directories:

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

Directory Creation

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

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

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

4.2.2 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 48 for more information on tabbed menu navigation.
2. Highlight the program you want to select and press [SELECT PROGRAM]. You can also type an existing program number and press [SELECT PROGRAM].
The program becomes the active program.
3. In OPERATION:MEM mode, you can type an existing program number and press the [UP] or [DOWN] cursor arrow to quickly change programs.

4.2.3 Program Transfer

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

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

File Name Convention

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

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

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

Copying Files

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

4.2.4 Deleting Programs

**NOTE:**

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

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

**NOTE:**

You cannot delete the active program.

4. Press [**Y**] at the prompt to delete the program, or [**N**] to cancel the process.

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

**NOTE:**

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

4.2.5 Maximum Number of Programs

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

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

4.2.6 File Duplication

To duplicate a file:

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

4.2.7 Changing Program Numbers

To change a program number:

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

Program Number Change (in Memory)

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

1. Make the program the active program. Refer to page 72 for more information on the active program.
2. Type the new program number in **EDIT** mode.
3. Press [ALTER].

The program number changes to the name you specified.

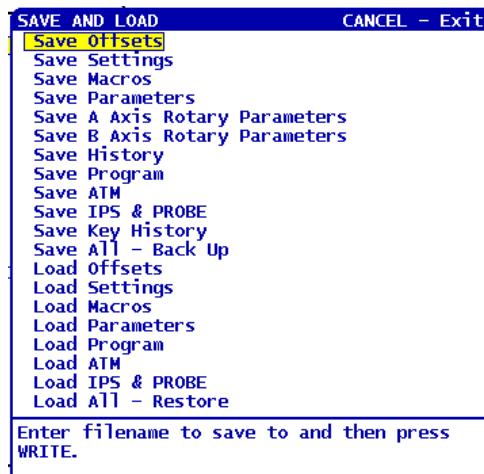
If a program in memory already has the new program number, the control returns the message *Prog exists*. The program number does not change.

4.3 Backing Up Your Machine

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

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

F4.3: Save and Load Popup



4.3.1 Making a Backup

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

Save File Type	File Extension
Offsets	.OFS
Settings	.SET
Macros - Variables	.VAR
Parameters	.PAR
Parameters - Pallet Positions (Mill)	.PAL
Parameters - Linear Screw Compensation	.LSC
A Axis Rotary Parameters (Mill)	.ROT
B Axis Rotary Parameters (Mill)	.ROT
History	.HIS
Program	.PGM
ATM - Advanced Tool Management	.ATM
IPS & Probe	.IPS
Key History	.KEY
All - Backup	

To back up the information from your machine:

1. Insert a USB memory device into the USB port on the right side of the control pendant.
2. Select the **USB** tab in the Device Manager.
3. Open the destination directory. If you want to create a new directory for your backup data, refer to page **72** for instructions.
4. Open the destination directory. If you want to create a new directory for your backup data, refer to Directory Creation for instructions.
5. Press **[F4]**.

6. Highlight the option you want.
7. Type a name for the backup. This name is attached to a unique extension for each backup option you picked. Press [ENTER].

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

4.3.2 Restoring From a Backup

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

1. Insert the USB memory device with the backup files into the USB port on the right side of the control pendant.
2. Select the **USB** tab in the Device Manager.
3. Press [**EMERGENCY STOP**].
4. Open the directory that contains the files you want to restore.
5. Press [**F4**].

The **Save and Load** pop-up menu appears.

6. Highlight **Load All - Restore** to load all file types (settings, parameters, programs, macros, tool offsets, variables, etc.)
7. Type the backup name with no extension (e.g., 28012014) that you want to restore, and press [**ENTER**].

All the files with the typed backup name are loaded on the machine. The message “Disk Done” displays after loading is complete.

8. To load a specific file type (like **name.PAR** for parameters), press [**F4**], highlight the file type (in this case, **Load Parameters**), type the backup name with no extension, and then press [**ENTER**].

The file with the typed backup name (in this case **name.PAR**) is loaded on the machine. The message “Disk Done” displays after loading is complete.

4.4 Basic Program Search

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



NOTE:

This is a quick-search function that finds the first match in the search direction that you specify. You can use the Advanced Editor for a more full-featured search. Refer to page 115 for more information on the Advanced Editor search function.

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

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

4.5 RS-232

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

You need a 9-pin to 25-pin null modem cable (not included) or a 9-pin to 25-pin straight Through cable with a null modem adapter to link the CNC control with the PC. There are two styles of RS-232 connections: the 25-pin connector and the 9-pin connector. The 9-pin connector is more commonly used on PCs. Plug the 25-pin connector end into the connector on the Haas machine located on the side panel of the control cabinet at the back of the machine.



NOTE:

Haas Automation does not supply null modem cables.

4.5.1 Cable Length

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

T4.1: Cable Length

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

4.5.2 Machine Data Collection

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

Data Collection Using the RS-232 Port

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

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

- *STX* (0x02) marks the start of data. This control character is for the remote computer.
- *CSV response* is Comma Separated Variables, one or more data variables separated by commas.
- *ETB* (0x17) marks the end of the data. This control character is for the remote computer.
- *CR/LF* tells the remote computer the data segment is complete and to move to the next line.
- *0x3E* Displays the > prompt.

If the control is busy, it outputs *Status, Busy*. If a request is not recognized, the control outputs *Unknown* and a new prompt >. These commands are available:

T4.2: Remote Q Commands

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

Command	Definition	Example
Q500	Three-in-one (PROGRAM, Oxxxxx, STATUS, PARTS, xxxx)	>Q500 STATUS, BUSY
Q600	Macro or system variable	>Q600 801 MACRO, 801, 333.339996

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

**NOTE:**

Use this command only when there are no alarms.

Data Collection Using Optional Hardware

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

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

These machine statuses are available:

- E-STOP contacts. This will be closed when the [EMERGENCY STOP] is pushed.
- Power ON - 115 VAC. Indicates the control is turned ON. It should be wired to a 115 VAC coil relay for interface.
- Spare Output Relay 40. Indicates that the control is In-Cycle (running.)
- Spare Output Relay 41 and 42:
 - 11 = MEM mode & no alarms (AUTO mode.)
 - 10 = MDI mode & no alarms (Manual mode.)
 - 01 = Single Block mode (Single mode)
 - 00 = Other modes (zero, DNC, jog, list program, etc.)

- Spare Output Relay 43 and 44:
 - 11 = Feed Hold stop (Feed Hold.)
 - 10 = M00 or M01 stop
 - 01 = M02 or M30 stop (Program Stop)
 - 00 = None of the above (could be single block stop or RESET.)
- Spare Output Relay 45 Feed Rate Override is active (Feed Rate is NOT 100%)
- Spare Output Relay 46 Spindle Speed Override active (Spindle Speed is NOT 100%)
- Spare Output Relay 47 Control is in EDIT mode

4.6 File Numerical Control (FNC)

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

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

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



CAUTION:

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

4.7 Direct Numerical Control (DNC)

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

F4.4: DNC Waiting and Received Program

PROGRAM (DNC)	N00000000	PROGRAM (DNC)	N00000000
WAITING FOR DNC . . .		O01000 ; (G-CODE FINAL QC TEST CUT) ; (MATERIAL IS 2x8x8 6061 ALUMINUM) ; (MAIN) ; M00 ; (READ DIRECTIONS FOR PARAMETERS AND SETTINGS) ; (FOR VF - SERIES MACHINES W/4TH AXIS CARDS) ; (USE / FOR 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	
DNC RS232			

T4.3: Recommended RS-232 Settings for DNC

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



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

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

1. [SETTING/GRAFIC] and scroll to the RS-232 settings (or enter 11 and press the up or down arrow).
2. Use the [UP] and [DOWN] cursor arrows to highlight the variables and the left and right arrows to change the values.
3. Press [ENTER] to confirm a selection.
4. DNC is selected by pressing [MDI/DNC] twice. DNC needs a minimum of 8k bytes of user memory available. This can be done by going to the List Programs page and checking the amount of free memory on the bottom of the page.
5. The program sent to the control must begin and end with a %. The data rate selected (Setting 11) for the RS-232 port must be fast enough to keep up with the rate of block execution of the program. If the data rate is too slow, the tool may stop in a cut.
6. Start sending the program to the control before [CYCLE START] is pushed. After the message *DNC Prog Found* shows, Press [CYCLE START].

4.7.1 DNC Notes

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

DNC supports drip mode. The control does (1) block (command) at a time. Each block is performed immediately with no block look-ahead. The exception is when Cutter Compensation is commanded. Cutter Compensation requires three blocks of motion commands to be read prior to a compensated block being performed.

Full duplex communication during DNC is possible with the G102 command or DPPRNT to output axis coordinates back to the controlling computer. Refer to page 313.

4.8 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.

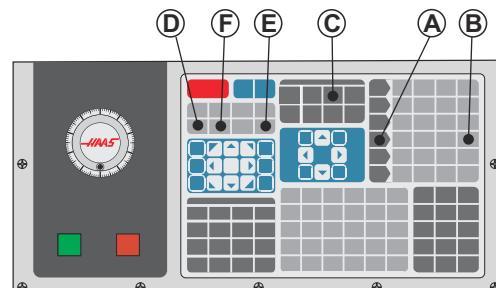
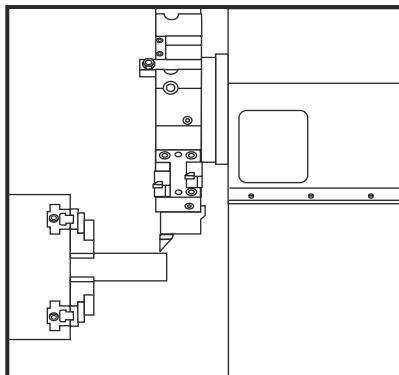
4.9 Setting the Tool Offset

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 229.

F4.5: Lathe Tool Offset



1. Load an O.D. turning tool into the tool turret. Press **[NEXT TOOL]** [F] until it is the current tool.
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]** [D] 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.

4.10 Manually Set the Tool Offset

To manually set tool offsets:

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.

4.11 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.

4.12 Additional Tooling Setup

There are other tool setup pages within the Current Commands.

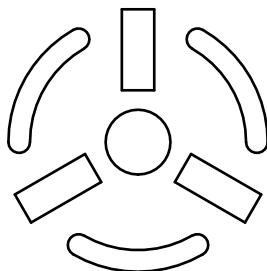
1. Press **[CURRENT COMMANDS]** and then use **[PAGE UP]/[PAGE DOWN]** to scroll to these pages.
2. The first is the page with Tool Load at the top of the page. You can add a tool load limit. The control references these values and can be set to do a specific action should the limitations be reached. Refer to Setting 84 (page 377) for more information on tool limit actions.
3. The second page is the Tool Life page. On this page there is a column called "Alarm". The programmer can put a value in this column, causing the machine to stop once the tool has been used that amount of times.

4.13 Part Setup

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

4.13.1 Chuck Foot Pedal

F4.6: 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 379 and Setting 122 on page 384 for more information).

Use Setting 76 to enable or disable all pedal controls. Refer to page 376 for more information.

4.13.2 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.



WARNING:

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



WARNING: *Do not machine parts larger than the chuck.*



WARNING: *Follow all of the warnings of the chuck manufacturer.*



WARNING: *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.*



WARNING: *Chuck jaws must not protrude beyond the diameter of the chuck.*



WARNING: *Improperly or inadequately clamped parts eject with deadly force.*



WARNING: *Do not exceed rated chuck RPM.*



WARNING: *Higher RPM reduces chuck clamping force. Refer to the chart.*



NOTE: *Grease your chuck weekly, and keep it clean.*

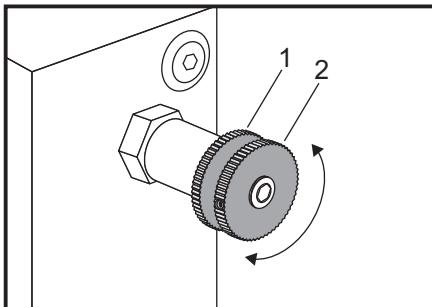
4.13.3 Drawtube Operation

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

Clamping Force Adjustment Procedure

To adjust the clamping force on the drawtube:

- F4.7: Draw tube Clamping Force Adjustment: [1] Locking knob, [2] Adjustment knob.

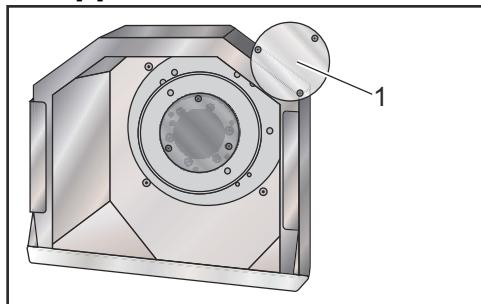


1. Go to Setting 92 on the **Settings** page and choose either I.D. clamping or O.D. clamping. Do not do this with a program running.
2. Turn the locking knob [1] counterclockwise to loosen.
3. Turn the adjustment knob [2] until the gauge reads the desired pressure. Turn clockwise to increase pressure.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

Drawtube Cover Plate

Before using the Bar Feeder,

- F4.8: 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.

4.13.4 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.

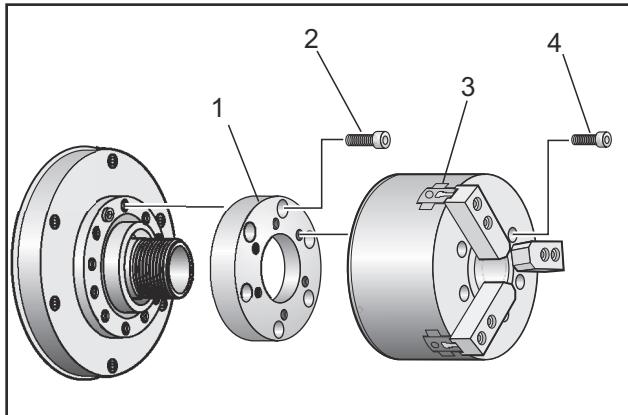
4

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.

F4.9: Chuck Removal Illustration: [1] Chuck Adapter Plate, [2] 6X Socket Head Cap Screws (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 (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 while you remove it.

Collet Installation

To install a collet:

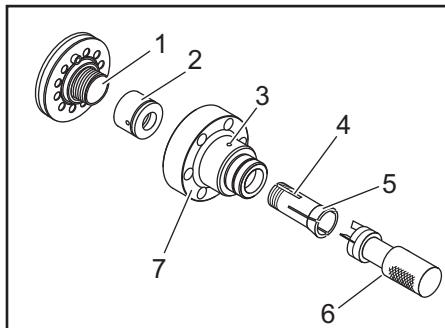
1. Thread the collet adapter into the drawtube.
2. Place the spindle nose on the spindle and align one of the holes on the back of the spindle nose with the drive dog.

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:

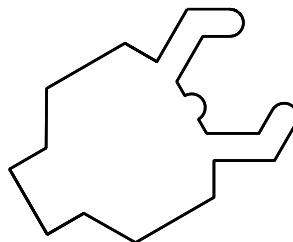
- F4.10:** Collet Removal Illustration: [1] Draw Tube, [2] Collet adapter, [3] Set Screw, [4] Set Screw Slot, [5] Collet, [6] Collet wrench, [7] Spindle Nose.



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].

4.13.5 Steady Rest Foot Pedal

- F4.11:** 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, M69 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 **376** for more information.

4.14 Tailstock Setup and Operation

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.

4.14.1 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

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. The only automatically moving part is the quill. Adjust the hydraulic pressure at the HPU to control quill hold force. Refer to the decal attached to the machine for information on quill hold force and hydraulic pressure.

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.

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 decal attached to your machine to determine the pressure setting for the hold force you need.

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.

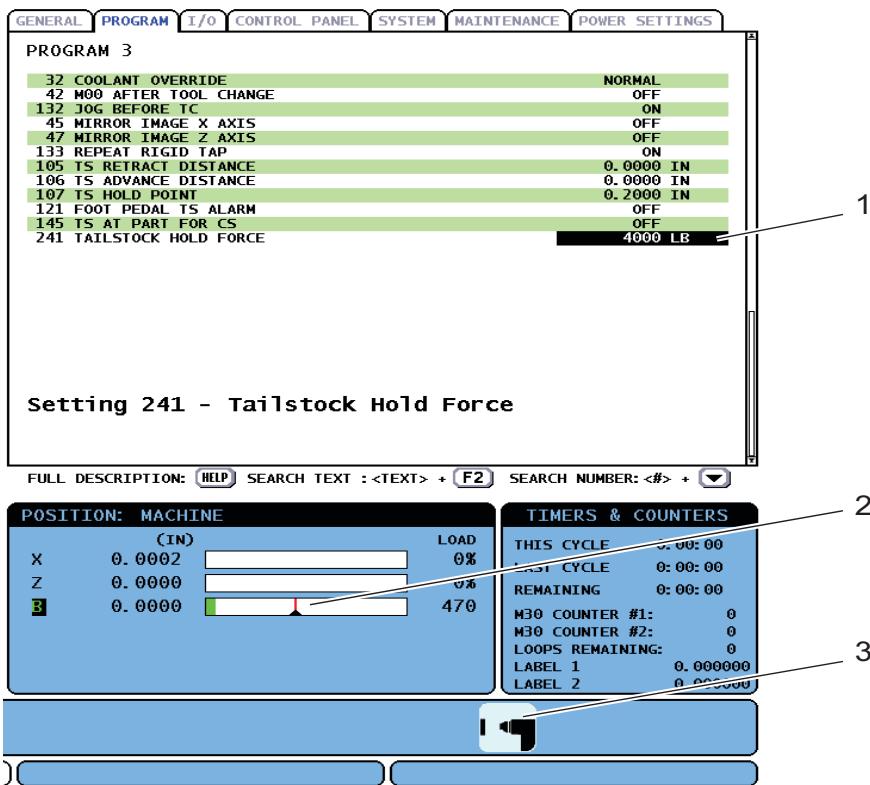
ST-40 Servo Tailstock Operation

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).

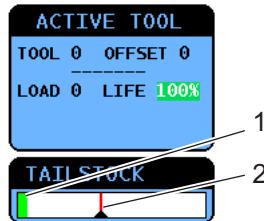
Tailstock load and current hold force are displayed as the B Axis in the axis load display (in modes such as **MDI** and **MEM**). The bar graph indicates current load, and the red line indicates the maximum hold force value specified in Setting 241. Actual hold force is displayed next to the bar graph.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

F4.12: 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 58 for more information on the tailstock hold icon.

F4.13: 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.

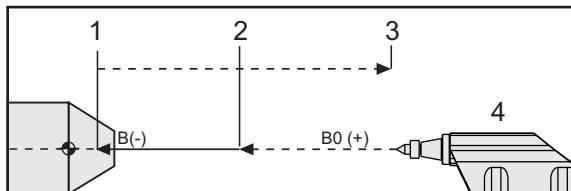
Zero return the tailstock before starting a new cycle on a new workpiece. You can then add the tailstock motion commands back into the program for future cycles.

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.

4.14.2 ST-20/30/40 Tailstock Operation

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

F4.14: 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**).

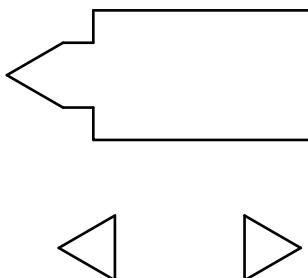
REMEMBER:

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

Tailstock Foot Pedal Operation

F4.15:

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 376 for more information.

4.14.3 Tailstock Restricted Zone

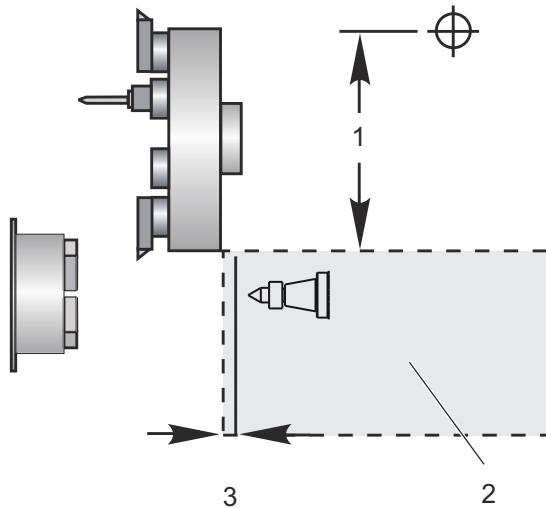
Setting up the tailstock involves setting a tailstock Restricted Zone.

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.

F4.16: [1] Setting 93, [2] Tailstock Restricted Zone, [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. Back the tool away in the X Axis a small amount then enter the value for 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

You may not always want to use a tailstock restricted zone (during setup, for example). To cancel a restricted zone:

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

4.14.4 Jogging the Tailstock

**CAUTION:**

If you position the tailstock manually, do not use an M21 in your program. This makes the tailstock move away from the workpiece and then 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 you zero return it.

You cannot jog the ST-40 servo tailstock while it is 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.

4.15 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.

4.15.1 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.

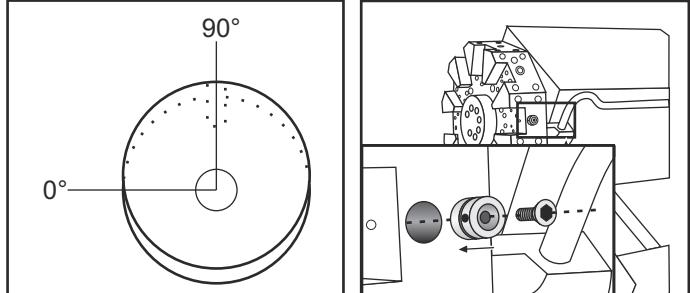
4.15.2 Eccentric Locating Cam Buttons

Bolt-on turrets have eccentric locating cam buttons that let you finely align your ID tool holders to the spindle centerline.

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.

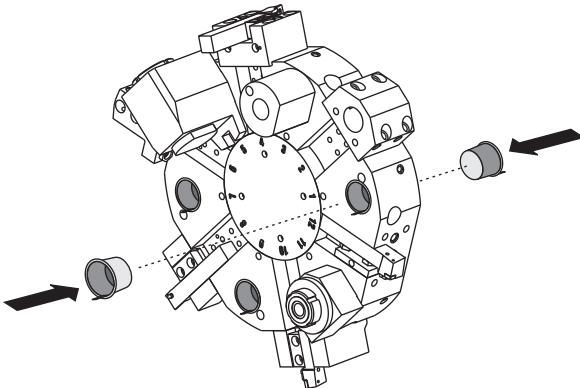
This table gives the result for specific positions of the cam button.

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)



4.15.3 Protective Cap

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

F4.17: Turret Protective Caps in Empty Pockets

4.15.4 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.

4.16 Setting Part Zero for the Z-axis (Part Face)

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

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

4.17 Features

Haas operation features:

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

4.17.1 Graphics Mode

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

The Graphics display has a number of available features:

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



NOTE:

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

- **Adjusting Zoom** Press **[F2]** to display a rectangle (zoom window) indicating the area to be magnified. Use **[PAGE DOWN]** to decrease the size of the zoom window (zooming in), and use **[PAGE UP]** to increase the size of the zoom window (zooming out). Use the Cursor Arrow keys to move the zoom window to the desired location and press **[ENTER]** to complete the zoom and rescale the tool path window. The locater 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.
- **Control Status** The lower left portion of the screen displays control status. It is the same as the last four lines of all other displays.
- **Position Pane** The position pane displays axes locations just as it would during a live part run.
- **Simulation Speed** **[F3]** decreases simulation speed and **[F4]** increases simulation speed.

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

1. Press **[SETTING/GRAFIC]** until the **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 mode, press **[MDI/DNC]** until DNC mode is active, then go to **GRAPHICS** page and send the program to the machine's control (See the DNC section).
3. Press **[CYCLE START]**.

**NOTE:**

Not all machine functions or motions are simulated in graphics.

4.17.2 Dry Run Operation

**CAUTION:**

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

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

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

**NOTE:**

Graphics mode is just as useful and may be safer as it does not move the axes of the machine before the program is checked.

4.17.3 Axis Overload Timer

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

4.18 Running Programs

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

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

4.19 Run-Stop-Jog-Continue

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

1. Press **[FEED HOLD]**.
Axis motion stops. The spindle continues to turn.
2. Press **[X]**, **[Y]** or **[Z]**, then press **[HANDLE JOG]**. The control stores the current X, Y, and Z positions.



NOTE:

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

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



CAUTION:

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

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



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

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

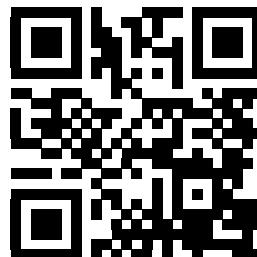


CAUTION: *If Setting 36 is ON, the control scans the program to make sure the machine is in the correct state (tools, offsets, G- and M-codes, etc.) to safely continue the program. If Setting 36 is OFF, the control does not scan the program. This can save time, but it could cause a crash in an unproven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

4

4.20 More Information Online

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



Chapter 5: Programming

5.1 Numbered Programs

To create a new program:

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



NOTE:

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

If the program exists, the control sets it as the active program (refer to page 72 for more information on the active program). If it does not yet exist, the control creates it and sets it as the active program.

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

5.2 Program Editors

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

5.2.1 Basic Program Editing

This section describes the basic program editing controls. For information on more advanced program editing functions, refer to page 110.

1. You write or make changes to programs in an active EDIT:EDIT or EDIT:MDI window.
 - a. To edit a program in MDI, press [MDI/DNC]. This is EDIT:MDI mode.
 - b. To edit a numbered program, select it, then press [EDIT]. This is EDIT:EDIT mode. Refer to page 72 to learn how to select a program.

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



NOTE:

*The control saves programs in **MEMORY** as you enter each line. To save programs in **USB**, **HD**, or **Net Share**, refer to the **Haas Editor (FNC)** section on page **118**.*

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

5.2.2 Background Edit

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

1. Press **[EDIT]** until the background edit pane (Inactive Program) on the right side of the screen is active.
2. Press **[SELECT PROGRAM]** to select a program to background edit (the program must be in Memory) from the list.
3. Press **[ENTER]** to begin background editing.
4. To select a different program to background edit, press **[SELECT PROGRAM]** from the background edit pane and choose a new program from the list.

5. All of the changes made during Background Edit will not affect the running program, or its subprograms. The changes will go into effect the next time the program is run. To exit background edit and return to the running program, press **[PROGRAM]**.
6. **[CYCLE START]** may not be used while in Background Edit. If the program contains a programmed stop (M00 or M01), exit Background Edit (press **[PROGRAM]**) and then press **[CYCLE START]** to resume the program.

**NOTE:**

All keyboard data is diverted to the Background Editor, when a M109 command is active and Background Edit is entered. Once an edit is complete (by pressing [PROGRAM] keyboard input will return to the M109 in the running program.

5.2.3 Manual Data Input (MDI)

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

F5.1: MDI Input Page Example

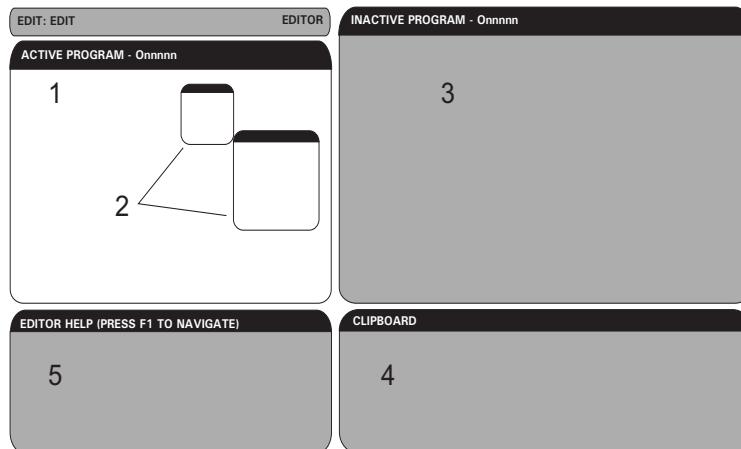
The screenshot shows a software interface titled "MDI". Inside the window, there is a text area containing the following G-code commands:
G97 S1000 M03 ;
G00 X2. Z0.1 ;
X1.78 ;
X1.76 ;
X1.75 ;

1. Press **[MDI/DNC]** to enter **MDI** mode.
2. Type program commands in the window. Press **[CYCLE START]** to execute the commands.
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 (**O**nnnnn).
 - c. Press **[ALTER]**.
The control saves your program to memory and clears the MDI input page. You can find the new program in the **MEMORY** tab in the Device Manager menu (press **[LIST PROGRAM]**).
4. Press **[ERASE PROGRAM]** to delete everything from the MDI input page.

5.2.4 Advanced Editor

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

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



1. Press **[EDIT]** to enter edit mode.
2. Two editing panes are available; an active program pane and an inactive program pane. Press **[EDIT]** to switch between the two panes.
3. Press **[SELECT PROGRAM]**.
The active window lists programs in memory with the active program marked with an asterisk (*) before the name.
4. To edit a program, type the program number (Onnnnn) or select it from the program list, and press **[SELECT PROGRAM]**.
The program opens in the active window.
5. Press **[F4]** to open another copy of that program in the inactive program pane if there is not a program there already.
6. You can also select a different program for the inactive program pane. Press **[SELECT PROGRAM]** from the inactive program pane and select the program from the list.
7. Press **[F4]** to exchange the programs between the two panes (make the active program inactive and vice versa).
8. Use the jog handle or cursor keys to scroll through the program code.
9. Press **[F1]** to access the pop-up menu.
10. Use the **[LEFT]** and **[RIGHT]** cursor arrows to select from the topic menu (HELP, MODIFY, SEARCH, EDIT, PROGRAM), and use the **[UP]** and **[DOWN]** cursor arrows or the jog handle to select a function.

11. Press [ENTER] to execute a command from the menu.

**NOTE:**

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

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

The Advanced Editor Pop-up Menu

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

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

The Program Menu

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

F5.3: The Advanced Editor Program Menu


Create New Program

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

Select Program From List

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

Duplicate Active Program

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

Delete Program From List

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

Swap Editor Programs

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

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

Switch to Left or Right Side

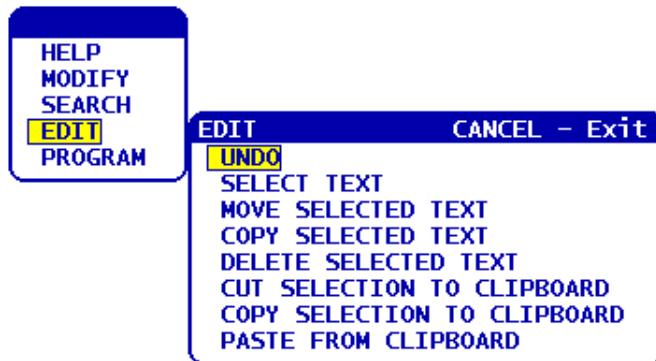
This switches edit control between the active and inactive program. Inactive and active programs remain in their respective panes.

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

The Edit Menu

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

F5.4: Advanced Edit Pop-up Menu



Undo

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

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

Select Text

>

This menu item will select lines of program code:

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

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

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

Move Selected Text

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

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

Copy Selected Text

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

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

Delete Selected Text

To delete selected text:

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

Cut Selection to Clipboard

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

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

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

Copy Selection To Clipboard

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

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

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

Paste From Clipboard

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

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

The Search Menu

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

F5.5: Advanced Search Popup



Find Text

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

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

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

Find Again

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

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

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

Find And Replace Text

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

1. Press **[F1]**. Select the **FIND AND REPLACE TEXT** command in the **SEARCH** pop-up menu category.
2. Type your search term.
3. Press **[ENTER]**.
4. Type the text with which you want to replace the search term.
5. Press **[ENTER]**.
6. Press **[F]** to search for the text below the cursor position. Press **[B]** to search above the cursor position.
7. When the control finds each occurrence of the search term, it gives the prompt *Replace (Yes/No/All/Cancel) ?*. Type the first letter of your choice to continue.

If you choose **Yes** or **No**, the editor will execute your choice and move to the next occurrence of the search term.

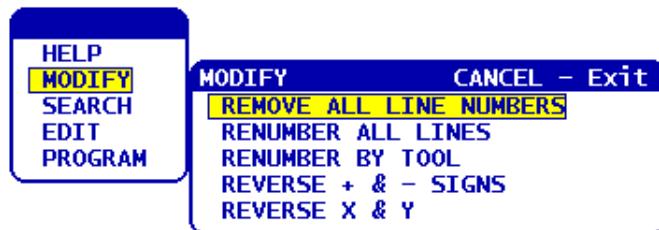
Choose **All** to automatically replace all occurrences of the search term.

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

The Modify Menu

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

F5.6: Advanced Modify Popup



Remove All Line Numbers

This command automatically removes all unreferenced N-code line numbers from the edited program. If you have selected a group of lines (refer to page 113), 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 113), this command affects only those lines.

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

Renumber By Tool

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

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

Reverse + and - Signs

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

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



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

3. Press [ENTER].

Reverse X and Y

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

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

5.2.5 The File Numerical Control (FNC) Editor

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

In general, you use the Advanced Editor with programs in MEM, while you use the FNC Editor with programs on drives other than MEM (ie., HDD, USB, and Net Share). Refer to Basic Program Editing on page 107 and Advanced Editor on page 110 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.

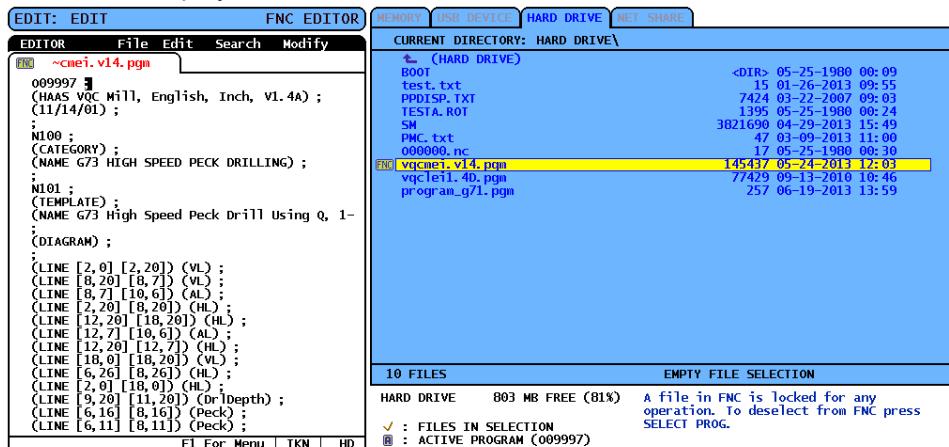
Loading a Program (FNC)

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.

F5.7: Edit: Edit Display



5

Menu Navigation (FNC)

To access the menu.

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

Display Modes (FNC)

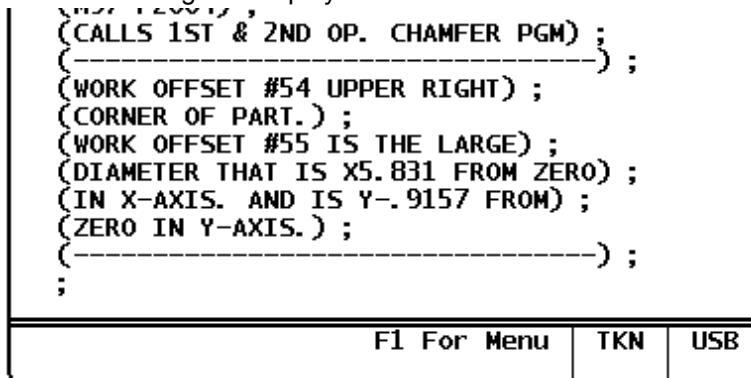
Three display modes are available. To switch between display modes:

1. Press **[F1]** for the popup File menu.
2. Use the Change View command.
3. Press **[ENTER]**.
4. List displays the current FNC program alongside the tabbed LIST PROG menu.
5. Main displays one program at a time in a tabbed pane (switch between tabs using the Swap Programs command in the File menu or by pressing **[F4]**).
6. Split displays the current FNC program on the left and the currently open programs in a tabbed pane on the right. Toggle the active pane using the Switch to Left or Right Side in the File menu or by pressing **[EDIT]**. When the tabbed pane is active, switch between tabs using the Swap Programs command in the **[F1]** popup File menu or by pressing **[F4]**.

Display Footer (FNC)

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

F5.8: Footer Section of Program Display



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

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

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

Opening Multiple Programs (FNC)

You can open up to three programs simultaneously in FNC Editor. To open an existing program while another program is open in FNC Editor:

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

Display Line Numbers (FNC)

To display line numbers independent of the program text:

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

**NOTE:**

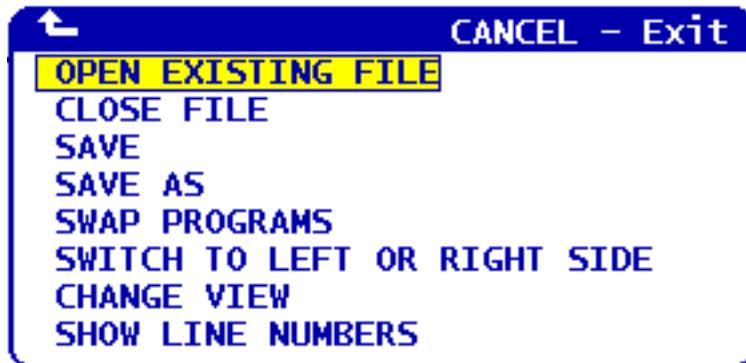
These are not the same as Nxx line numbers; they are only for reference when viewing the program.

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

File Menu (FNC)

To access the File menu:

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

F5.9: File Menu

Open Existing File

When in FNC EDITOR mode,

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

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

Close File

When in FNC EDITOR mode,

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

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

Save



NOTE:

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

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

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

Saves the current active file under the same filename.

Save As

When in FNC EDITOR mode,

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

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

Swap Programs

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

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

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

Switch to Left or Right Side

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

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

Change View

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

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

Switches between List, Main, and Split view modes.

Show Line Numbers

When in FNC EDITOR mode,

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

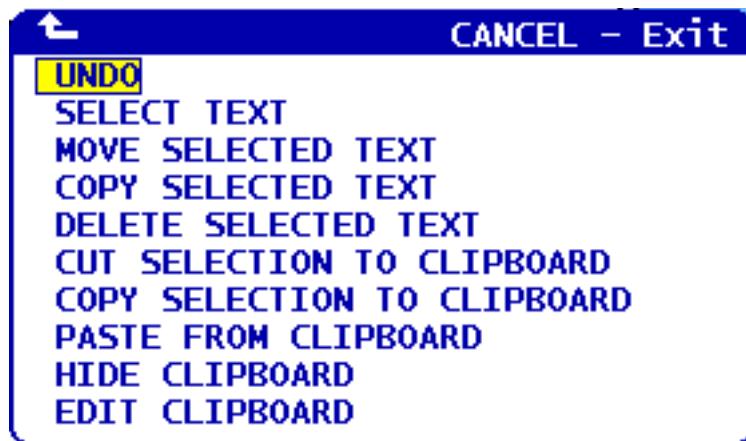
Displays reference-only line numbers independent of program text. They are never saved as part of the program like Nxx numbers would be. Select the option again to hide the line numbers.

Edit Menu (FNC)

To access the Edit menu:

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

F5.10: Edit Menu



Undo

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



NOTE:

Block and global functions cannot be undone.

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

Select Text

Highlights a block of text in FNC EDITOR mode:

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

Move/Copy/Delete Selected Text

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

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

Cut/Copy Selection to Clipboard

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

**NOTE:**

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

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

Paste from Clipboard

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

**NOTE:**

Does not delete the clipboard contents.

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

Hide/Show Clipboard

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

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

Edit Clipboard

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

**NOTE:**

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

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

Search Menu (FNC)

Accesses the Search menu:

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

F5.11: Search Menu



Find Text

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

1. Press **[F1]**.
2. Move the cursor to the Search menu and select Find Text.
3. Enter the search text.
4. Enter the search direction. When choosing a search direction, press F to search for the term below the cursor position, and press B to search above the cursor position.

Find Again

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

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

Find and Replace Text

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

1. Press **[F1]**.
2. Move the cursor to the Search menu and select Find and Replace Text.
3. Enter text to locate.
4. Enter replacement text.
5. Enter the search direction. When you choose a search direction, press F to search for the term below the cursor position, and press B to search above the cursor position.
6. When the first occurrence of the search term is found, the control will prompt *Replace (Yes/No/All/Cancel) ?*. Type the first letter of the selection to continue. If **Yes** or **No** is chosen, the editor will execute the choice and move to the next occurrence of the search term. Choose **All** to automatically replace all occurrences of the search term. Choose **Cancel** to back out of the function without changes (text that is already replaced will remain if this option is chosen).

Find Tool

Searches the program for tool numbers in FNC EDITOR mode:

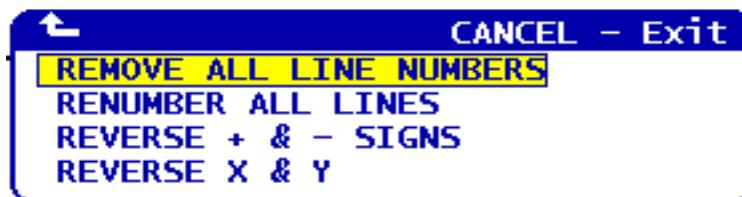
1. Press **[F1]**.
2. Move the cursor to the Search menu and select Find Tool.
3. Select it again to locate the next tool number.

Modify Menu (FNC)

Accesses the Modify menu:

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

F5.12: Modify Menu



Remove All Line Numbers

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

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

Renumber All Lines

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

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

Reverse + and - Signs

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

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

5.3 Tips and Tricks

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

5.3.1 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 383 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. Different display modes change the information that is 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]**.

To select another program enter the program number (**Onnnnn**) and press the arrow up or down. The machine must be in either **Memory** or **Edit** mode. To search for a specific command in a program, use 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.

To transfer or save a program in MDI to the list of programs position the cursor at the beginning of the MDI program, enter a program number (**Onnnnn**), and press **[ALTER]**.

Program Review - Program Review allows the operator to move the cursor and review a copy of the active program on the right side of the display screen, and view the same program as it runs 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 contains the active program.

Background Edit - This feature edits while a program runs. 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.

To Copy Programs - In **Edit** mode, a program can be copied into another program, a line, or a block of lines in a program. Define a block with the **[F2]** key, then move the cursor to the last program line to define, press **[F2]** or **[ENTER]** to highlight the block. Select another program to copy the selection to. Move the cursor to the point where the copied block is moved and press **[INSERT]**.

To Load Files - Select multiple files in the device manager, then press **[F2]** to select a destination.

To Edit 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.

To Duplicate a Program - An existing program can be duplicated in List Program mode. To do this, select the program number 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. Highlight the desired programs from the program list to select them and press **[ENTER]**. Press **[SEND]** to transfer the files.

5.3.2 Offsets

To enter

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

5.3.3 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. Reverse the **[HANDLE JOG]** control to generate 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.

5.3.4 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.

5.3.5 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 **Triangle**, **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.

5.4 Program Optimizer

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

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

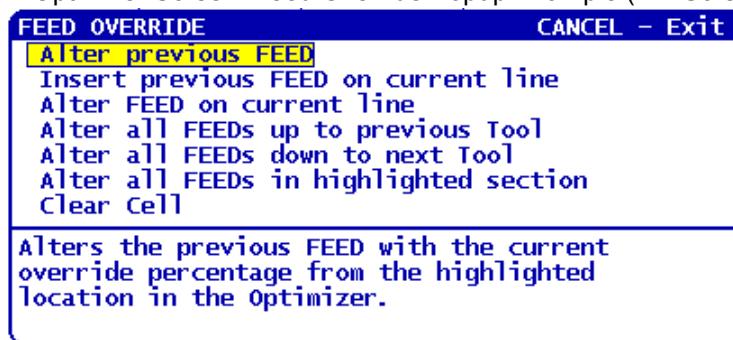
5.4.1 Program Optimizer Operation

To go to the Program Optimizer screen:

1. At the end of a program run, press [MEMORY].
2. Press [F4].
3. Use the right/left and up/down arrows, [PAGE UP]/[PAGE DOWN] and [HOME]/[END] to scroll through **Overrides** and **Notes** columns.
4. On the column topic to edit, press [ENTER].

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

F5.13: Program Optimizer Screen: Feed Override Popup Example (Mill Screen Shown)

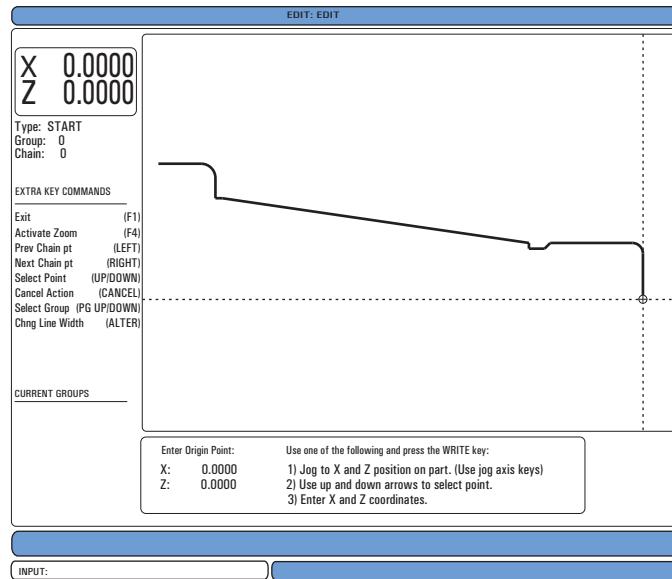


5. In addition, you can highlight a section of code (place the cursor at the start of your selection, press [F2], scroll to the end of your selection and press [F2]). Go back to the Program Optimizer (press [EDIT]) and press [ENTER] to alter all the feeds or speeds in the highlighted section.

5.5 DXF File Importer

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

F5.14: DXF File Import



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



NOTE:

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

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

5.5.1 Part Origin

Use one of these methods to set the part origin.

- Point Selection
- Jogging
- Enter Coordinates

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

5.5.2 Part Geometry Chain and Group

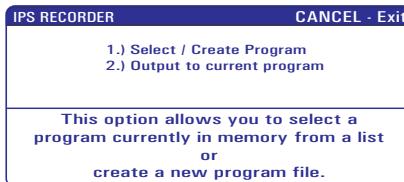
This step finds the geometry of the shape(s). The auto chaining function finds most part geometry. If the geometry is complex and branches off, a prompt displays so you can select one of the branches. The automatic chaining continues after you select a branch. DXF importer groups together holes for drilling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

1. Use the jog handle or cursor arrows to choose the toolpath starting location.
2. Press [F2] to open the dialog box.
3. Choose the option that best suits the desired application. Most times, the Automatic Chaining function is the best choice, because it automatically plots the toolpath for a part feature.
4. Press [ENTER]. This changes the color of that part feature and adds a group to the register under **Current group** on the left side of the window.

5.5.3 Toolpath Selection

This step applies a toolpath to a particular chained group.

F5.15: DXF IPS Recorder Menu



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

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

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

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

5.6 Basic Programming

A typical CNC program has (3) parts:

1. **Preparation:** This portion of the program selects the work and tool offsets, spindle speed, selects the cutting tool, and turns on the coolant.
2. **Cutting:** This portion of the program defines the tool path, and feed rate for the cutting operation.
3. **Completion:** This portion of the program turns off the coolant, moves the tool to Z-Axis home, moves the tool to X-Axis home, turns off the spindle, and allows the part to be unloaded from the chuck and inspected.

This program makes a 0.100" (2.54 mm) deep face cut in a piece of material with Tool 1 along the X Axis from X = 2.1 to X = - 0.02 (negative 0.02 X-Axis over-travel makes sure the uncompensated tool cuts the whole face).



NOTE:

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

```
%  
o40001 (BASIC PROGRAM) ;  
(G54 X0 is at the center of rotation) ;  
(Z0 is on face of the part) ;  
(T1 is an end face cutting tool) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G50 S1000 (Limit spindle to 1000 RPM) ;  
G97 S500 M03 (CSS off, Spindle on CW) ;  
G00 G54 X2.1 Z0.1 (Rapid to 1st position) ;  
M08 (Coolant on) ;  
G96 S200 (CSS on) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 Z-0.1 F.01 (Linear feed) ;  
X-0.02 (Linear feed) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, coolant off) ;  
G97 S500 (CSS off) ;  
G53 X0 (X home) ;  
G53 Z0 M05 (Z home, spindle off) ;  
M30 (End program) ;  
%
```

5.6.1 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.
O40001 (BASIC PROGRAM) ;	O40001 is the name of the program. Program naming convention follows the Onnnnn format: The letter “O”, or “o” followed by a 5-digit number.
(G54 X0 is at the center of rotation) ;	Comment
(Z0 is on face of the part) ;	Comment
(T1 is an end face cutting tool) ;	Comment
T101 (Select tool and offset 1) ;	T101 selects the tool, the offset 1, and commands the tool change to Tool 1.
G00 G18 G20 G40 G80 G99 (Safe startup) ;	This is referred to as a safe startup line. It is good machining practice to place this block of code after every tool change. G00 defines axis movement following it to be 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. G80 cancels any canned cycles. G99 puts the machine in Feed per Rev mode.
G50 S1000 (Limit spindle to 1000 RPM) ;	G50 limits the spindle to a max of 1000 RPM. S1000 is the spindle speed address. Using Snnnn address code, where nnnn is the desired spindle RPM value.
G97 S500 M03 (CSS off, Spindle on CW) ;	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 346 for more information on these M-codes. S500 is the spindle speed address. Using Snnnn address code, where nnnn is the desired spindle RPM value. M03 turns on the spindle.

Preparation Code Block	Description
G00 G54 X2.1 Z0.1 (Rapid to 1st position) ;	G00 defines axis movement following it to be in Rapid Motion mode. G54 defines the coordinate system to be centered on the Work Offset stored in G54 on the Offset display. X2.0 commands the X Axis to X = 2.0. Z0.1 commands the Z Axis to Z = 0.1.
M08 (Coolant on) ;	M08 turns on the coolant.
G96 S200 (CSS on) ;	G96 turns on CSS. S200 specifies a cutting speed of 200 ipm to be used along with the current diameter to calculate the correct RPM.

5.6.2 Cutting

These are the cutting code blocks in the sample program:

Cutting Code Block	Description
G01 Z-0.1 F.01 (Linear feed) ;	G01 defines axis movements after it to be in a straight line. Z-0.1 commands the Z Axis to Z = -0.1. G01 requires address code Fnnn.nnnn. F.01 specifies the feedrate for the motion is .0100" (.254 mm)/Rev.
X-0.02 (Linear feed) ;	X-0.02 commands the X Axis to X = -0.02.

5.6.3 Completion

These are the completion code blocks in the sample program:

Completion Code Block	Description
G00 Z0.1 M09 (Rapid retract, coolant off) ;	G00 commands the axis motion to be completed in Rapid Motion mode. Z0.1 commands the Z Axis to Z = 0.1. M09 turns off the coolant.
G97 S500 (CSS off) ;	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. S500 is the spindle speed address. Using Snnnn address code, where nnnn is the desired spindle RPM value.

Completion Code Block	Description
G53 X0 (X home) ;	G53 defines axis movements after it to be with respect to the machine coordinate system. X0 commands the X Axis to move to X = 0.0 (X home).
G53 Z0 M05 (Z home, spindle off) ;	G53 defines axis movements after it to be with respect to the machine coordinate system. Z0 commands the Z Axis to move to Z = 0.0 (Z home). M05 turns off the spindle.
M30 (End program) ;	M30 ends the program and moves the cursor on the control to the top of the program.
%	Denotes the end of a program written in a text editor.

5.6.4 Absolute vs. Incremental (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.

5.7 Miscellaneous Codes

This section lists frequently used M-codes. Most programs have at least one M-code from each of the following families. Refer to the M-code section of this manual, starting on page 341, for a listing of all M-codes with descriptions.

5.7.1 Tool Functions

The T_{nn}o_o code selects the next tool (nn) and offset (oo). The use of this code differs slightly depending on Setting 33 (FANUC or YASNAC coordinate system).

FANUC Coordinate System

T-codes have the format T_{xx}y_y 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.

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.

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.

5.7.2 Spindle Commands

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

- M03 commands the spindle to turn in the forward direction.
- M04 commands the spindle to turn in the reverse direction.



NOTE:

You can command the spindle speed with an `Snnnn` address code, where `nnnn` specifies the speed in rpm, but overrides from G50, G96, or G97 may apply to the actual spindle speed.

- M05 commands the spindle to stop.



NOTE:

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

5.7.3 Program Stop Commands

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

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



NOTE:

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

5.7.4 Coolant Commands

Use M08 to command standard coolant on. Use M09 to command standard coolant off. Refer to page 342 for more information on these M-codes.

If your machine has High-Pressure Coolant (HPC), use M88 to command it on, and M89 to command it off.

5.8 Cutting G-codes

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

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

5.8.1 Linear Interpolation Motion

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

Corners can be chamfered using the optional argument Cnn.nnnn to define the chamfer. Corners can be rounded using the optional address code Rnn.nnnn to define the radius of the arc. Refer to page 250 for more information on G01.

5.8.2 Circular Interpolation Motion

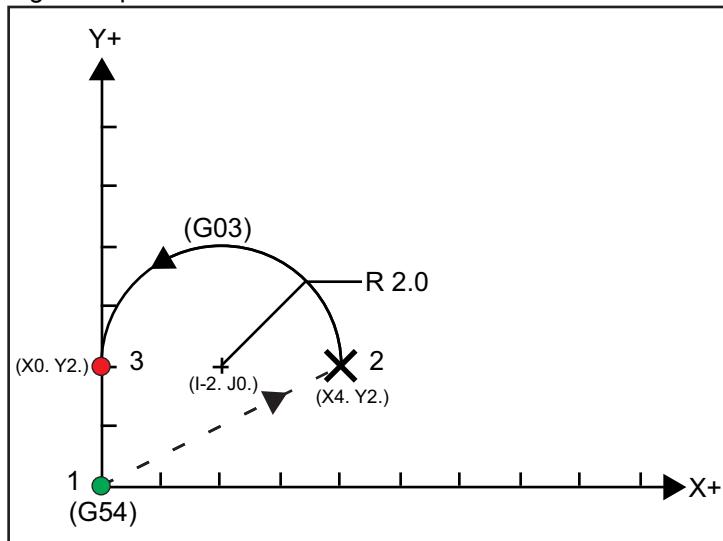
G02 and G03 are the G-codes for circular cutting motions. Circular Interpolation Motion has several optional address codes to define the arc or circle. The arc or circle begins cutting from the current cutter position [1] to the geometry specified within the G02/G03 command.

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

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

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

F5.16: Arc Cutting Example

**Method 1:**

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

Method 2:

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

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

```
%  
T01 M06 ;  
...  
G00 X4. Y2. ;  
G01 F20.0 Z-0.1 ;  
G02 F20.0 I2.0 J0. ;  
...  
M30 ;  
%
```

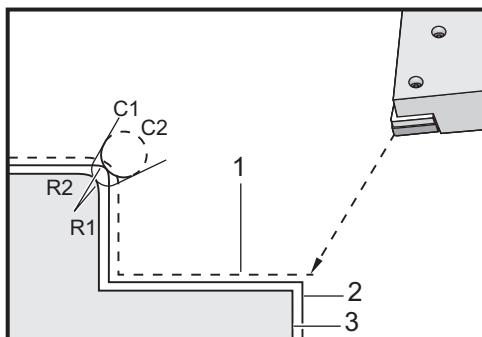
5.9 Tool Nose Compensation

Tool Nose Compensation (TNC) is a feature that lets you adjust a programmed tool path in for different cutter sizes, or for normal cutter wear. With TNC, you only need to enter minimal offset data when you run a program. You do not need to do additional programming.

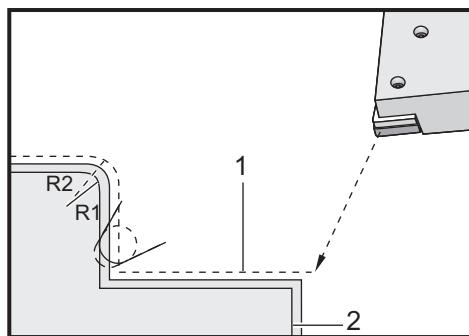
5.9.1 Programming

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.

- F5.17:** Cutting path without tool nose compensation: [1] Tool Path, [2] Cut after wear [3] Desired cut.



- F5.18:** Cutting path with tool nose compensation: [1] Compensated tool path, [2] Desired cut and programmed tool path.



NOTE:

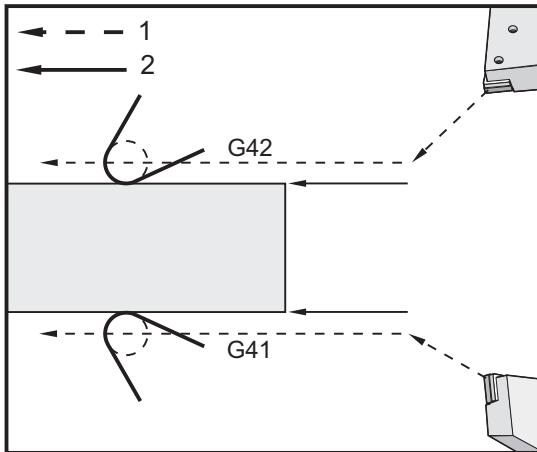
The second programmed path coincides with the final part dimension.

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.

5.9.2 Tool Nose Compensation Concept

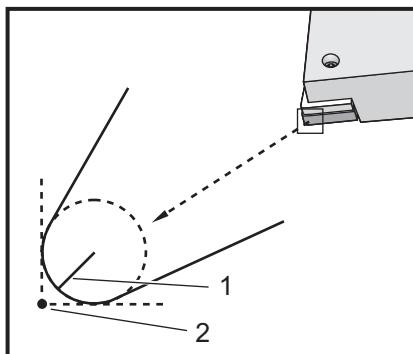
Tool nose compensation works by shifting the Programmed Tool Path to the right or to the left. The programmer usually programs the tool path to the finished size. When tool nose compensation is used, the control compensates for the radius of the tool based on special instructions written into the program. 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.

- F5.19:** 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.

- F5.20:** 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.

5.9.3 Using Tool Nose Compensation

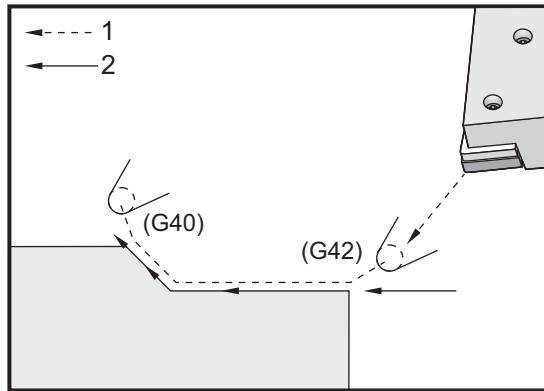
The following steps are used to program a part using TNC:

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.

5.9.4 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.

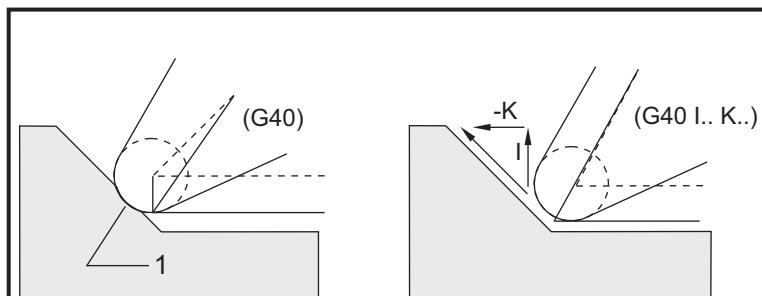
F5.21: TNC Approach and Depart Moves: [1] Compensated Path, [2] Programmed path.



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.

F5.22: TNC Use of I and K in G40 Block: [1] Overcut.



5.9.5 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:

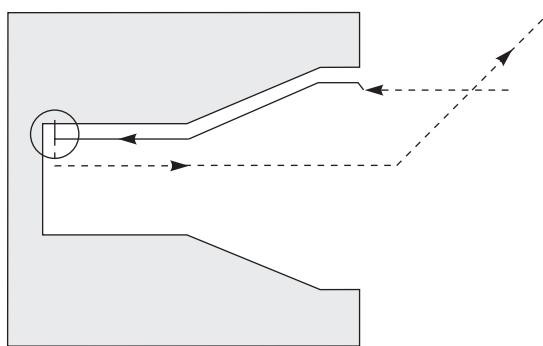
```
%  
o30411 (TOOL NOSE RADIUS AND WEAR OFFSET) ;  
(G54 X0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is a boring bar) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G50 S1000 (Limit spindle to 1000 RPM) ;  
G97 S500 M03 (CSS off, Spindle on CW) ;  
G00 G54 X0.49 Z0.05 (Rapid to 1st position) ;  
M08 (Coolant on) ;
```

```

(BEGIN CUTTING BLOCKS) ;
G96 S750 (CSS on) ;
G41 G01 X.5156 F.004 (TNC left on) ;
Z-.05 (Linear feed) ;
X.3438 Z-.25 (Linear feed) ;
Z-.5 (Linear feed) ;
X.33 (Linear feed) ;
G40 G00 X0.25 (TNC off, exit line) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, coolant off) ;
G53 X0 (X home) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%

```

F5.23: TNC Departure Cutting Error



5.9.6 Tool Nose Compensation and Tool Length Geometry

You set up the length geometries for tools that use tool nose compensation in the same way that you set up tools that do not use compensation.

Refer to page 83 for details on touching off tools and recording tool length geometries. When you set up a new tool, be sure to clear the geometry wear to zero.

If you command particularly heavy cuts on one edge of a tool, the tool can wear unevenly. In this case, adjust the **X or Z Geometry Wear** instead of the **Radius Wear**. You can often adjust X or Z length geometry wear to compensate for uneven tool nose wear. Length geometry wear shifts all dimensions for a single axis.

The program design may not let you use length geometry shift to compensate for wear. To determine which wear to adjust, check 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 you should increase the radius wear offset. Wear that affects the dimensions on one axis only suggests length geometry wear.

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

5.9.7 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 249 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

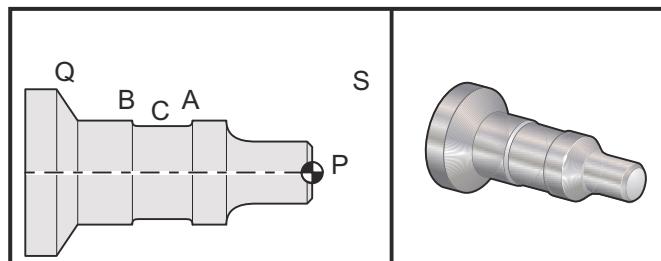
5.9.8 Example Programs Using Tool Nose Compensation

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.

F5.24: TNC Standard Interpolation G01, G02, and G03



Preparation

- Turn Setting 33 to FANUC.
- Set up these 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

Tool	Offset	X	Z	Radius	Tip
T3	03	-8.8400	-12.8380	.016	3
T3	13	-8.8400	-12.588	.016	4

```
%  
O30421 (TNC STANDARD INTERPOLATION G01/G02/G03) ;  
(G54 X0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is an rough OD tool) ;  
(T2 is a finish OD tool) ;  
(T3 is a groove tool) ;  
(T1 PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G50 S1000 (Limit spindle to 1000 RPM) ;  
G97 S500 M03 (CSS off, Spindle on CW) ;  
G00 G54 X2.1 Z0.1 (Rapid to position S) ;  
M08 (Coolant on) ;  
G96 S200 (CSS on) ;  
(T1 CUTTING BLOCKS) ;  
G71 P1 Q2 U0.02 W0.005 D.1 F0.015 (Begin G71) ;  
N1 G42 G00 X0. Z0.1 F.01 (P1 - TNC on) ;  
G01 Z0 F.005 (Begin toolpath) ;  
X0.65 (Linear feed) ;  
X0.75 Z-0.05 (Linear feed) ;  
Z-0.75 (Linear feed) ;  
G02 X1.25 Z-1. R0.25 (Feed CW) ;  
G01 Z-1.5 (Linear feed to position A) ;  
G02 X1. Z-1.625 R0.125 (Feed CW) ;  
G01 Z-2.5 (Linear feed) ;  
G02 X1.25 Z-2.625 R0.125 (Feed CW to position B) ;  
G01 Z-3.5 (Linear feed) ;  
X2. Z-3.75 (End of toolpath) ;  
N2 G00 G40 X2.1 (Q2 - TNC off) ;  
(T1 COMPLETION BLOCKS) ;  
G97 S500 (CSS off) ;  
G53 X0 M09 (X home, coolant off) ;  
G53 Z0 (Z home, clear for tool change) ;  
M01 (Optional program stop) ;  
(T2 PREPARATION BLOCKS) ;  
T202 (T2 is a finish OD tool) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G50 S1000 (Limit spindle to 1000 RPM) ;  
G97 S500 M03 (CSS off, Spindle on CW) ;
```

```
G00 G54 X2.1 Z0.1 (Rapid to position S) ;
M08 (Coolant on) ;
G96 S200 (CSS on) ;
(T2 CUTTING BLOCKS) ;
G70 P1 Q2 (Finish P1 - Q2 using T2, G70 and TNC) ;
(T2 COMPLETION BLOCKS) ;
G97 S500 (CSS off) ;
G53 X0 M09 (X home, coolant off) ;
G53 Z0 (Z home, clear for tool change) ;
M01 (Optional program stop) ;
(T3 PREPARATION BLOCKS) ;
T303 (T3 is a groove tool) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G97 S500 M03 (CSS off, Spindle on CW) ;
G54 G42 X1.5 Z-2.0 (TNC on, rapid to point C) ;
M08 (Coolant on) ;
G96 S200 (CSS on) ;
(T3 CUTTING BLOCKS) ;
G01 X1. F0.003 (Linear feed) ;
G01 Z-2.5 (Linear feed) ;
G02 X1.25 Z-2.625 R0.125 (Feed CW to position B) ;
G01 G40 X1.5 (TNC off) ;
T313 (Change offset to other side of insert) ;
G00 G41 X1.5 Z-2.125 (TNC left on) ;
G01 X1. F0.003 (Linear feed) ;
G01 Z-1.625 (Linear feed) ;
G03 X1.25 Z-1.5 R0.125 (Feed CCW to position A) ;
(T3 COMPLETION BLOCKS) ;
G00 G40 X1.6 M09 (TNC off, coolant off) ;
G97 S500 (CSS off) ;
G53 X0 (X home) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 ;
%
```



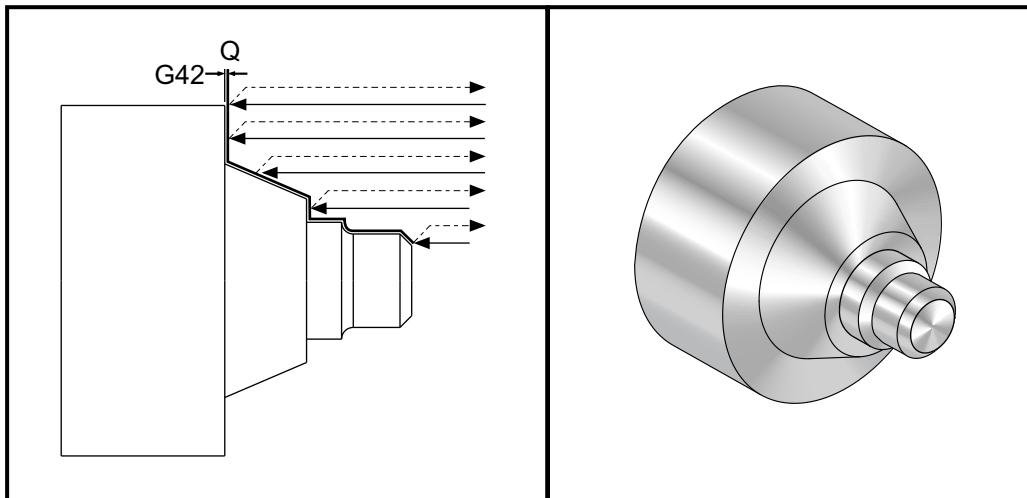
NOTE:

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 uses TNC with a G71 roughing canned cycle.

F5.25: TNC G71 Roughing Canned Cycle



Preparation:

- Setting 33 is **FANUC**.
- Tools:
T1 Insert with 0.032 radius, roughing

Tool	Offset	Radius	Tip
T1	01	.032	3

```
%  
o30711 (TNC WITH A G71 ROUGHING CYCLE) ;  
(G54 X0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is an OD cutting tool) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G50 S1000 (Limit spindle to 1000 RPM) ;  
G97 S500 M03 (CSS off, Spindle on CW) ;  
G00 G54 X3.0 Z0.1 (Rapid to 1st position) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G96 S200 (CSS on) ;
```

```

G71 P1 Q2 U.01 W.005 D.08 F.012 (Begin G71) ;
N1 G42 G00 X0.6 (P1 - TNC on) ;
G01 Z0 F0.01 (Begin toolpath) ;
X0.8 Z-0.1 F0.005 (45 deg. Chamfer) ; Z-0.5 (Linear feed) ;
G02 X1.0 Z-0.6 I0.1 (Feed CW) ;
G01 Z-0.9 (Linear feed) ;
X1.4 (Linear feed) ;
X2.0 Z-1.6 (23 deg. Taper) ;
G01 X3. (End of toolpath) ;
N2 G00 G40 X4. (Q2 - TNC off) ;
(BEGIN COMPLETION BLOCKS) ;
G97 S500 (CSS off) ;
G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%

```

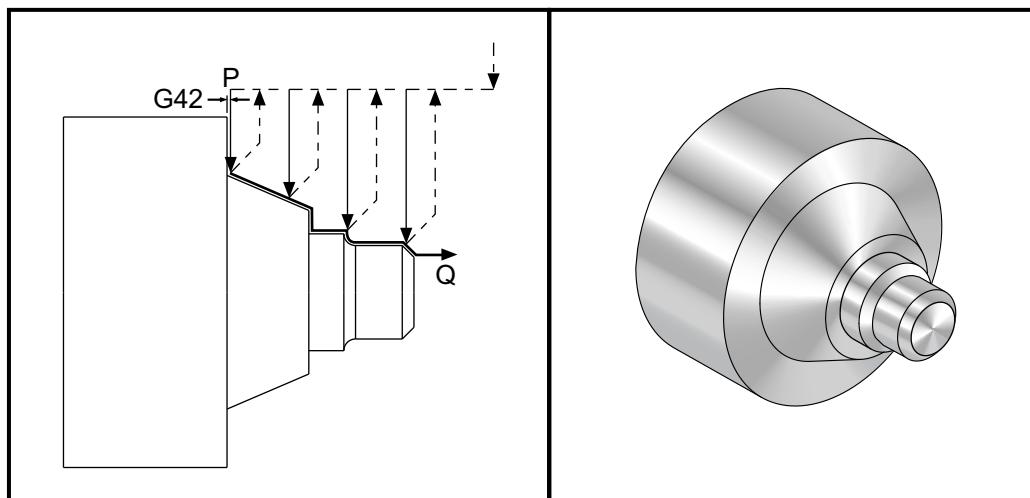
**NOTE:**

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.

F5.26: TNC G72 Roughing Canned Cycle



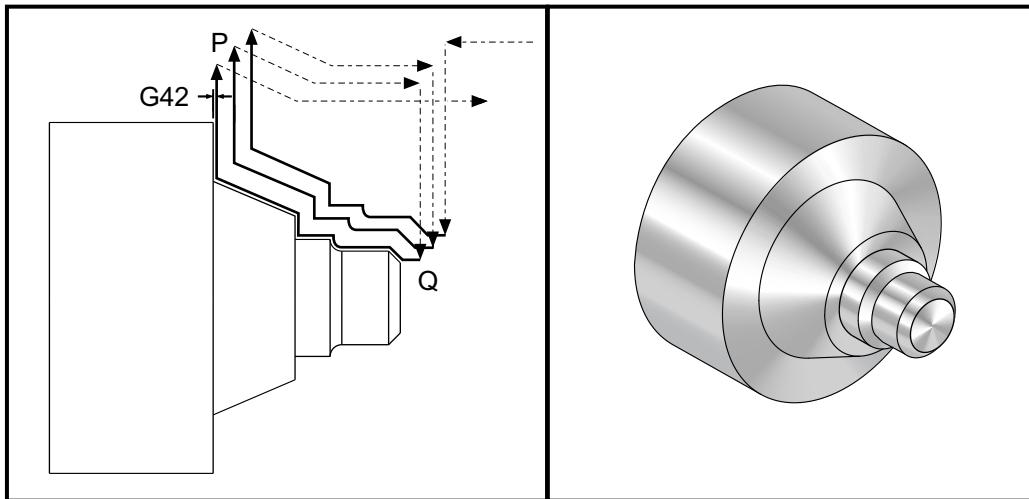
Setting 33 is **FANUC**.

```
%  
o30721 (TNC WITH A G72 ROUGHING CYCLE) ;  
(G54 X0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is an OD cutting tool) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G50 S1000 (Limit spindle to 1000 RPM) ;  
G97 S500 M03 (CSS off, Spindle on CW) ;  
G00 G54 X3.1 Z0 (Rapid to 1st position) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G96 S200 (CSS on) ;  
G72 P1 Q2 U.01 W.005 D.08 F.012 (Begin G72) ;  
N1 G41 G00 Z-1.6 (P1 - TNC on) ;  
G01 X2. F0.01 (Begin toolpath) ;  
X1.4 Z-0.9 (Taper) ;  
X1. (Linear feed) ;  
Z-0.6 (Linear feed) ;  
G03 X0.8 Z-0.5 R0.1 (Feed CCW) ;  
G01 Z-0.1 (Linear feed) ;  
X0.7 Z0 (Chamfer, End of toolpath) ;  
N2 G00 G40 Z0.1 (Q2 - TNC off) ;  
(BEGIN COMPLETION BLOCKS) ;  
G97 S500 (CSS off) ;  
G53 X0 M09 (X home, coolant off) ;  
G53 Z0 M05 (Z home, spindle off) ;  
M30 (End program) ;  
%
```

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.

F5.27: TNC G73 Roughing Canned Cycle



Setting 33 is FANUC

```
%  
o30731 (TNC WITH A G73 ROUGHING CYCLE) ;  
(G54 X0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is an OD cutting tool) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G50 S1000 (Limit spindle to 1000 RPM) ;  
G97 S500 M03 (CSS off, Spindle on CW) ;  
G00 G54 X3.0 Z0.1 (Rapid to 1st position) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G96 S200 (CSS on) ;  
G73 P1 Q2 U.01 W.005 I0.3 K0.15 D3 F.012 (Begin G73) ;  
N1 G42 G00 X0.6 (P1- TNC on) ;  
G01 Z0 F0.01 (Begin toolpath) ;  
X0.8 Z-0.1 F0.005 (Chamfer) ;  
Z-0.5 (Linear feed) ;  
G02 X1.0 Z-0.6 I0.1 (Feed CW) ;  
G01 Z-0.9 (Linear feed) ;  
X1.4 (Linear feed) ;  
X2.0 Z-1.6 (Taper) ;  
G01 X3. (End of toolpath) ;
```

```

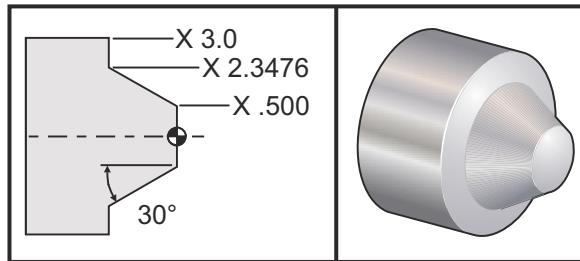
N2 G00 G40 X4. (Q2 - TNC off) ;
(BEGIN COMPLETION BLOCKS) ;
G97 S500 (CSS off) ;
G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%

```

Example 5: TNC with G90 Modal Rough Turning Cycle

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

F5.28: TNC With G90 Rough Turning Cycle



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

Setting 33: FANUC

```

%
o30901 (TNC WITH A G90 ROUGHING CYCLE) ;
(G54 X0 is at the center of rotation) ;
(Z0 is on face of the part) ;
(T1 is an OD cutting tool) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G50 S1000 (Limit spindle to 1000 RPM) ;
G97 S500 M03 (CSS off, Spindle on CW) ;
G00 G54 X4.0 Z0.1 (Rapid to 1st position) ;
M08 (Coolant on) ;
G96 S200 (CSS on) ;
(BEGIN CUTTING BLOCKS) ;
G90 G42 X2.55 Z-1.5 I-0.9238 F0.012 (Begin G90) ;
X2.45 (Optional additional pass) ;

```

```

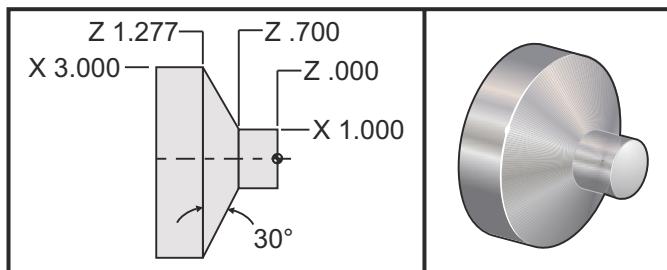
X2.3476 (Optional additional pass) ;
(BEGIN COMPLETION BLOCKS) ;
G00 G40 X3.0 Z0.1 M09 (TNC off, coolant off) ;
G97 S500 (CSS off) ;
G53 X0 (X home) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%

```

Example 6: TNC with G94 Modal Rough Turning Cycle

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

F5.29: TNC G94 Rough Turning Cycle



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

Setting 33: FANUC

```

%
○30941 (TNC WITH G94 MODAL TURNING CYCLE) ;
(G54 X0 is at the center of rotation) ;
(Z0 is on face of the part) ;
(T1 is an OD cutting tool) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G50 S1000 (Limit spindle to 1000 RPM) ;
G97 S500 M03 (CSS off, Spindle on CW) ;
G00 G54 X3.1 Z0.1 (Rapid to 1st position) ;
M08 (Coolant on) ;
G96 S200 (CSS on) ;
(BEGIN CUTTING BLOCKS) ;
G94 G41 X1.0 Z-0.5 K-0.577 F.03 (Begin G94 w/ TNC) ;

```

```

Z-0.6 (Optional additional pass) ;
Z-0.7 (Optional additional pass) ;
(BEGIN COMPLETION BLOCKS) ;
G00 G40 X3.1 Z0.1 M09 (TNC off, coolant off) ;
G97 S500 (CSS off) ;
G53 X0 (X home) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%

```

5.9.9 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.

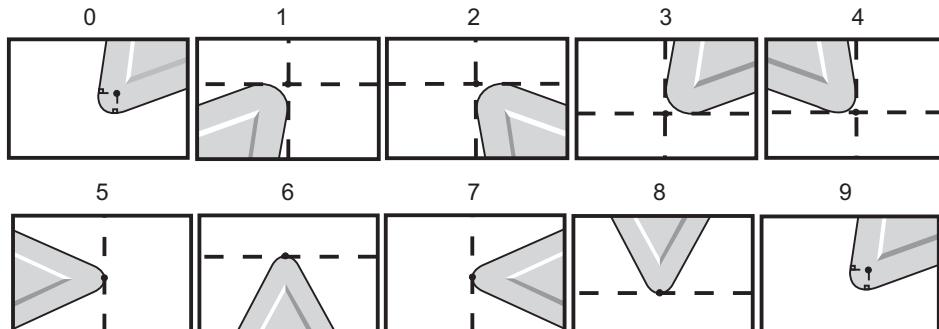
The tool tip direction of each tool is coded as a single integer from 0 to 9. The tip direction code is found next to the radius offset on the geometry offsets page. It is recommended that a tip direction be specified for all tools using tool nose compensation. The following figure is a summary of the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

F5.30: 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

5.9.10 Programming Without Tool Nose Compensation

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

5.9.11 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.

5.9.12 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:

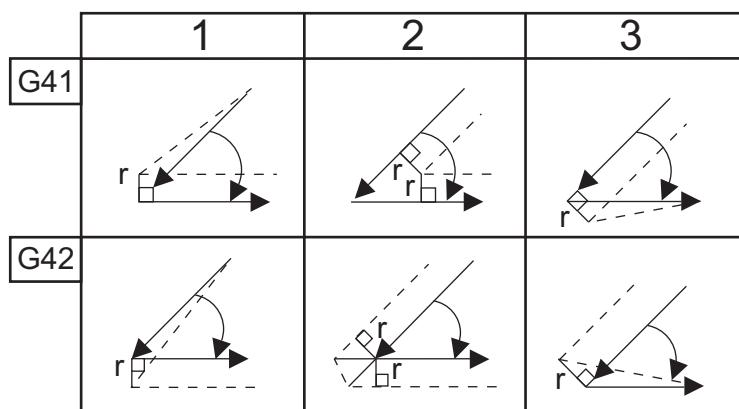
1. linear to linear
2. linear to circular
3. circular to linear
4. circular to circular

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

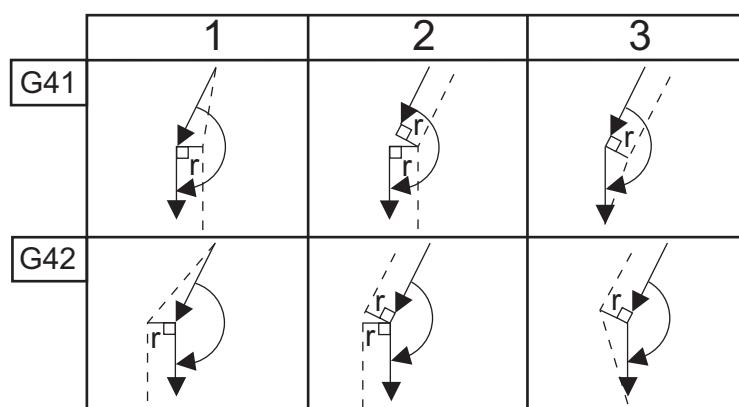
Two FANUC compensation types are supported, Type A and Type B. The default compensation is Type A.

F5.31: TNC Linear-to-Linear (Type A): [1] Approach, [2], Mode to mode, [3] Departure.

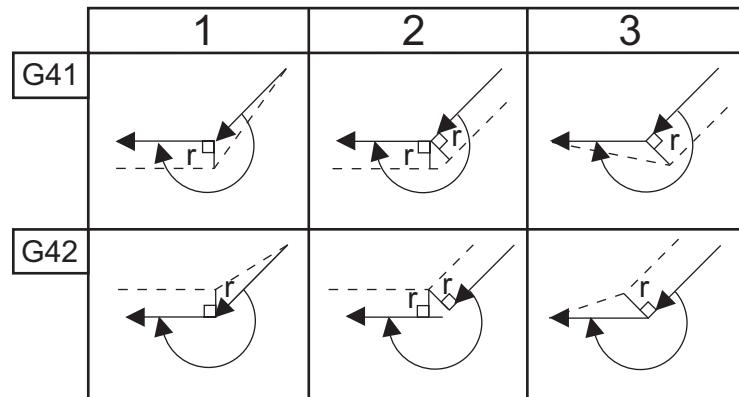
<90



>=90, <180



>180



F5.32: 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			

F5.33: TNC Circular-to-Linear (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			

Tool Radius And Angle Chart (1/32 RADIUS)

The X measurement calculated is based on part diameter.

ANGLE	Xc CROSS	Zc LONGITUDI NAL	ANGLE	Xc CROSS	Zc LONGITUDI NAL
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.	.0110	.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

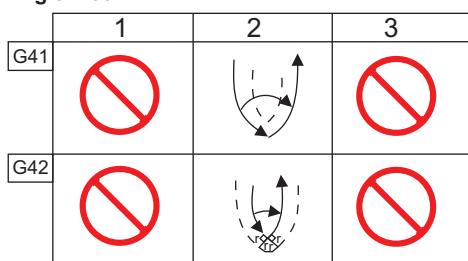
Tool Nose Compensation

ANGLE	Xc CROSS	Zc LONGITUDI NAL	ANGLE	Xc CROSS	Zc LONGITUDI NAL
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

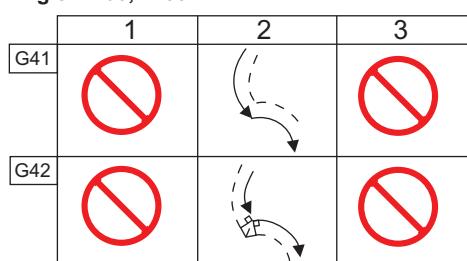
ANGLE	Xc CROSS	Zc LONGITUDI NAL	ANGLE	Xc CROSS	Zc LONGITUDI NAL
43.	.0354	.0189	88.	.0614	.0011
44.	.036	.0186	89.	.062	.0005
45.	.0366	.0183			

F5.34: TNC Circular-to-Circular (Type A): [1] Approach, [2], Mode to mode, [3] Departure.

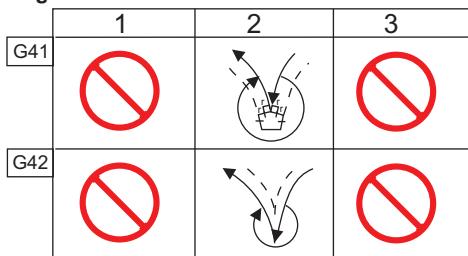
Angle: <90



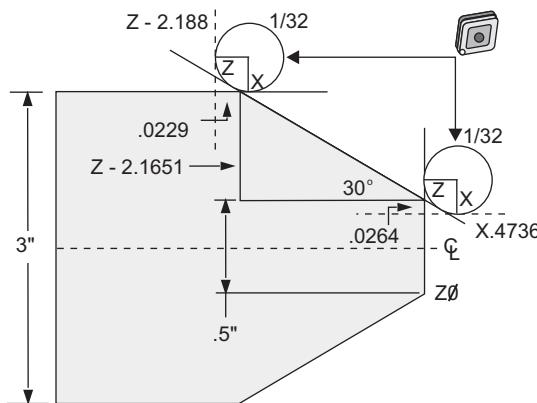
Angle: >=90, <180



Angle: >180

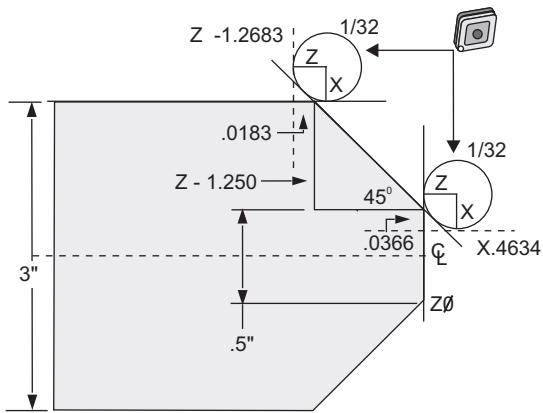


F5.35: Tool Nose Radius Calculation, 1/32, Compensation value for 30 degree angle.



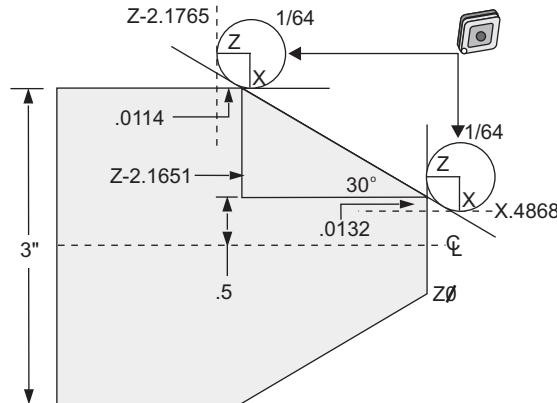
Code	Compensation (1/32 tool nose radius)
G0 X0 Z.1	
G1 Z0	
X.4736	(X.5-0.0264 compensation)
X 3.0 Z-2.188	(Z-2.1651+0.0229 compensation)

F5.36: 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+.0183 compensation)

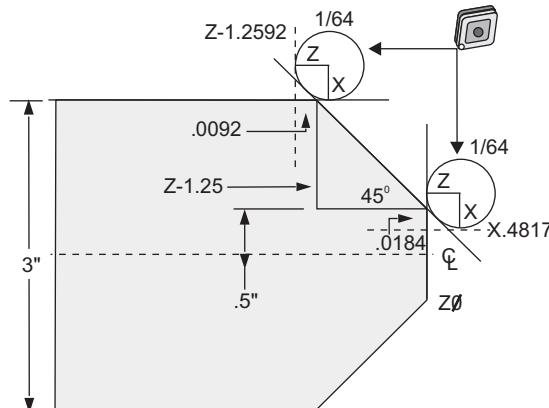
F5.37: Tool Nose Radius Calculation, 1/64, Compensation value for 30 degree angle.



5

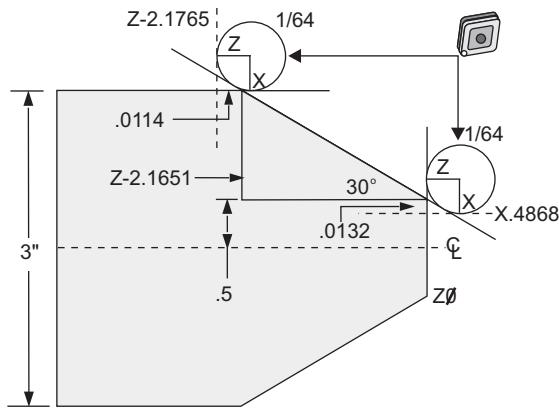
Code	Compensation (1/64 tool nose radius)
G0 X0 Z.1	
G1 Z0	
X.4868	(X.5-.0132 compensation)
X 3.0 Z-2.1765	(Z-2.1651+.0114 compensation)

F5.38: 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)

F5.39: 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 LONGITUDI NAL	ANGLE	Xc CROSS	Zc LONGITUDI NAL
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

Tool Nose Compensation

ANGLE	Xc CROSS	Zc LONGITUDI NAL	ANGLE	Xc CROSS	Zc LONGITUDI NAL
22.	.0102	.0126	67.	.0248	.0053
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

ANGLE	Xc CROSS	Zc LONGITUDI NAL	ANGLE	Xc CROSS	Zc LONGITUDI NAL
44.	.018	.0093	89.	.031	.0003
45.	.0184	.0092			

5.10 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.

5.10.1 Effective Coordinate System

5

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.

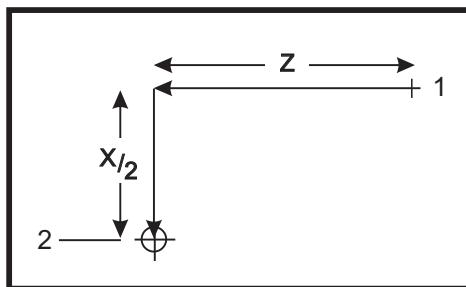
YASNAC Work Coordinate Shift - YASNAC controls discuss a work coordinate shift. It serves the same function as the common coordinate system. When Setting 33 is set to **YASNAC**, it is found on the **Work Offsets** display page as T00.

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 adjust 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.

F5.40: 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) ;
.
.
.
%
```

5.10.2 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.

5.10.3 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.

5.11 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

5.11.1 Live Image Stock Setup

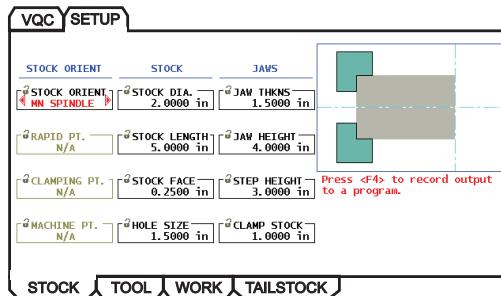
Data values for stock and jaw dimensions are stored in the Stock Setup screen. Live Image applies this stored data to each tool.



NOTE:

Turn Setting 217 ON (refer to page 389) to show the chuck jaws in the display.

F5.41: Stock Setup Screen



To enter stock and jaw values:

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 **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.

5.11.2 Program Example

```
%001000 ;
;
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/GRAFIC**], typing in the first **LIVE IMAGE** setting (202) and pressing the [**UP**] cursor arrow.

Refer to page 388 for more information.

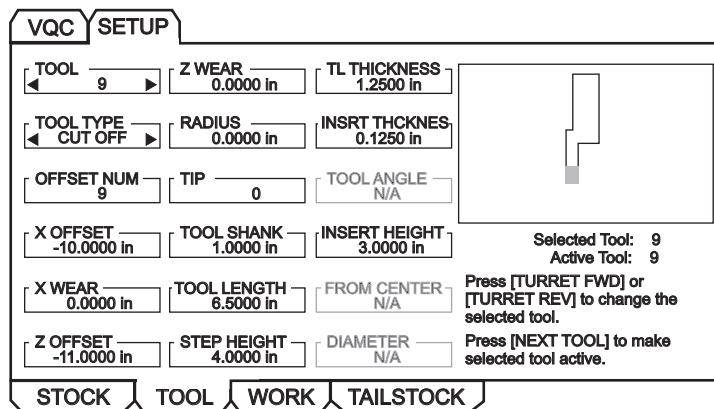
5.11.3 Live Image Tool Setup

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.*

F5.42: Tool Setup



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

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:

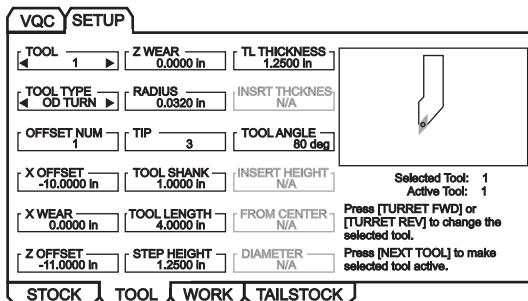
```
%  
o40002 (LIVE IMAGE TOOL SETUP);  
(G54 X0 is at the center of rotation) ;  
(Z0 is on face of the part) ;  
(T1 is an OD cutting tool) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;
```

```

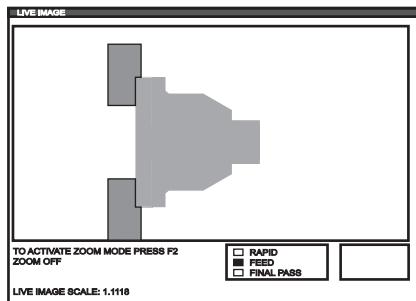
G50 S1000 (Limit spindle to 1000 RPM) ;
G97 S500 M03 (CSS off, Spindle on CW) ;
G00 G54 X6.8 Z0.1 (Rapid to 1st position) ;
M08 (Coolant on) ;
G96 S200 (CSS on) ;
(BEGIN CUTTING BLOCKS) ;
G71 P1 Q2 D0.25 U0.02 W0.005 F0.025 (Begin G71) ;
N1 G00 G40 X2. (Begin toolpath, TNC off) ;
G01 X2.75 Z0. (Linear feed) ;
G01 X3. Z-0.125 (Linear feed) ;
G01 X3. Z-1.5 (Linear feed) ;
G01 X4.5608 Z-2.0304 (Linear feed) ;
G03 X5. Z-2.5606 R0.25 (Feed CCW) ;
G01 X5. Z-3.75 (Linear feed) ;
G02 X5.5 Z-4. R0.25 (Feed CW) ;
G01 X6.6 Z-4. (Linear feed) ;
N2 G01 G40 X6.8 Z-4. (Linear feed) ;
(BEGIN COMPLETION BLOCKS) ;
G97 S500 (CSS off) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%

```

F5.43: [1] T101 Settings , and [2] Part worked from T101 Settings.



1



2

Sample Tool Setup Screens

F5.44: Tool Setup: [1] Drill, [2] ID Bore

VQC SETUP			
<input type="button" value="TOOL 2 >"/>	Z WEAR 0.0000 In	TL THICKNESS N/A	
<input type="button" value="TOOL TYPE DRILL >"/>	RADIUS 0.0000 In	INSRT THICKNES N/A	
OFFSET NUM 2	TIP 7	TOOL ANGLE 175 deg	
X OFFSET -10.0000 In	TOOL SHANK N/A	INSERT HEIGHT N/A	
X WEAR 0.0000 In	TOOL LENGTH 5.0000 In	FROM CENTER N/A	
Z OFFSET -11.0000 In	STEP HEIGHT N/A	DIAMETER 2.0000 In	
Selected Tool: 2 Active Tool: 2			
Press [TURRET FWD] or [TURRET REV] to change the selected tool.			
Press [NEXT TOOL] to make selected tool active.			
STOCK	TOOL	WORK	TAILSTOCK

1

VQC SETUP			
<input type="button" value="TOOL 3 >"/>	Z WEAR 0.0000 In	TL THICKNESS N/A	
<input type="button" value="TOOL TYPE ID BORE >"/>	RADIUS 0.0320 In	INSRT THICKNES N/A	
OFFSET NUM 3	TIP 2	TOOL ANGLE 80 deg	
X OFFSET -10.0000 In	TOOL SHANK N/A	INSERT HEIGHT N/A	
X WEAR 0.0000 In	TOOL LENGTH 6.0000 In	FROM CENTER 1.0000 In	
Z OFFSET -11.0000 In	STEP HEIGHT N/A	DIAMETER 1.5000 In	
Selected Tool: 3 Active Tool: 3			
Press [TURRET FWD] or [TURRET REV] to change the selected tool.			
Press [NEXT TOOL] to make selected tool active.			
STOCK	TOOL	WORK	TAILSTOCK

2

F5.45: Tool Setup: [1] OD Grove, [2] ID Groove

VQC SETUP			
<input type="button" value="TOOL 5 >"/>	Z WEAR 0.0000 In	TL THICKNESS 1.2500 In	
<input type="button" value="TOOL TYPE OD GROOVE >"/>	RADIUS 0.0000 In	INSRT THICKNES 0.1250 In	
OFFSET NUM 5	TIP 0	TOOL ANGLE N/A	
X OFFSET -10.0000 In	TOOL SHANK 1.0000 In	INSERT HEIGHT 0.3500 In	
X WEAR 0.0000 In	TOOL LENGTH 4.0000 In	FROM CENTER N/A	
Z OFFSET -11.0000 In	STEP HEIGHT N/A	DIAMETER 1.6250 In	
Selected Tool: 5 Active Tool: 5			
Press [TURRET FWD] or [TURRET REV] to change the selected tool.			
Press [NEXT TOOL] to make selected tool active.			
STOCK	TOOL	WORK	TAILSTOCK

1

VQC SETUP			
<input type="button" value="TOOL 6 >"/>	Z WEAR 0.0000 In	TL THICKNESS N/A	
<input type="button" value="TOOL TYPE ID GROOVE >"/>	RADIUS 0.0000 In	INSRT THICKNES 0.1250 In	
OFFSET NUM 6	TIP 0	TOOL ANGLE N/A	
X OFFSET -10.0000 In	TOOL SHANK N/A	INSERT HEIGHT N/A	
X WEAR 0.0000 In	TOOL LENGTH 6.0000 In	FROM CENTER 1.0000 In	
Z OFFSET -11.0000 In	STEP HEIGHT N/A	DIAMETER 1.5000 In	
Selected Tool: 6 Active Tool: 6			
Press [TURRET FWD] or [TURRET REV] to change the selected tool.			
Press [NEXT TOOL] to make selected tool active.			
STOCK	TOOL	WORK	TAILSTOCK

2

F5.46: Tool Setup: [1] OD Thread, [2] ID Thread

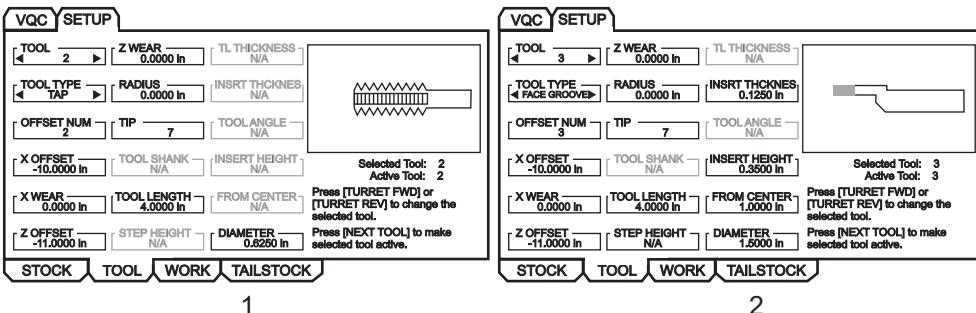
VQC SETUP			
<input type="button" value="TOOL 7 >"/>	Z WEAR 0.0000 In	TL THICKNESS 1.2500 In	
<input type="button" value="TOOL TYPE OD THREAD >"/>	RADIUS 0.0000 In	INSRT THICKNES N/A	
OFFSET NUM 7	TIP 0	TOOL ANGLE 60 deg	
X OFFSET -10.0000 In	TOOL SHANK 1.0000 In	INSERT HEIGHT 0.1250 In	
X WEAR 0.0000 In	TOOL LENGTH 4.0000 In	FROM CENTER N/A	
Z OFFSET -11.0000 In	STEP HEIGHT 1.2500 In	DIAMETER N/A	
Selected Tool: 7 Active Tool: 7			
Press [TURRET FWD] or [TURRET REV] to change the selected tool.			
Press [NEXT TOOL] to make selected tool active.			
STOCK	TOOL	WORK	TAILSTOCK

1

VQC SETUP			
<input type="button" value="TOOL 8 >"/>	Z WEAR 0.0000 In	TL THICKNESS N/A	
<input type="button" value="TOOL TYPE ID THREAD >"/>	RADIUS 0.0000 In	INSRT THICKNES N/A	
OFFSET NUM 8	TIP 0	TOOL ANGLE 60 deg	
X OFFSET -10.0000 In	TOOL SHANK N/A	INSERT HEIGHT 0.1250 In	
X WEAR 0.0000 In	TOOL LENGTH 6.0000 In	FROM CENTER 1.0000 In	
Z OFFSET -11.0000 In	STEP HEIGHT N/A	DIAMETER 1.5000 In	
Selected Tool: 8 Active Tool: 8			
Press [TURRET FWD] or [TURRET REV] to change the selected tool.			
Press [NEXT TOOL] to make selected tool active.			
STOCK	TOOL	WORK	TAILSTOCK

2

F5.47: Tool Setup: [1] Tap, [2] Face Groove



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

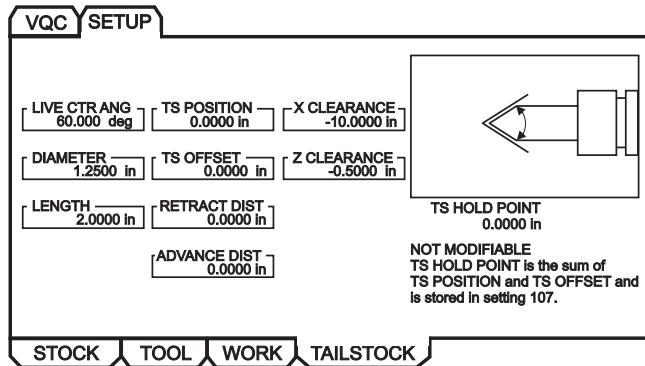
5.11.4 Tailstock Setup (Live Image)

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.

F5.48: Tailstock Setup Screen



- Press [**MDI/DNC**], then [**PROGRAM**] to enter **IPS JOG** mode.
- 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**.

LIVE CTR ANG, **DIAMETER** and **LENGTH** match Settings 220-222. **X CLEARANCE** matches Setting 93. **Z CLEARANCE** matches Setting 94. **RETRACT DIST** matches

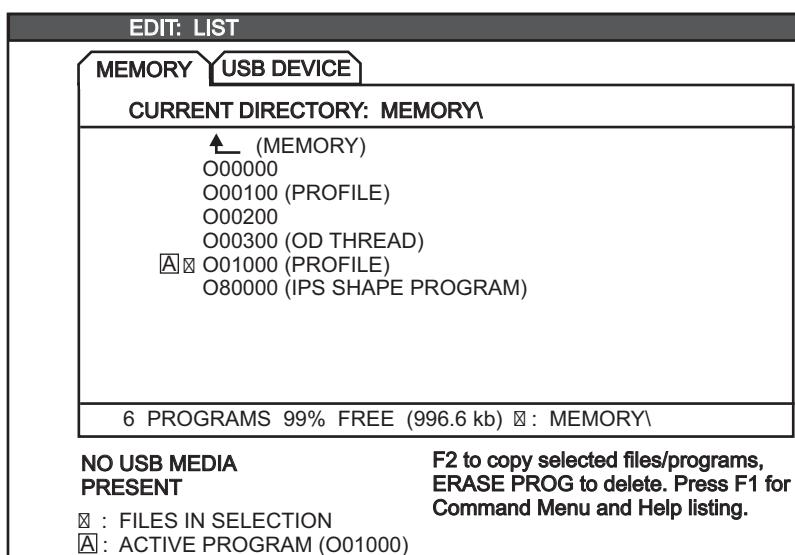
Setting 105. **ADVANCE DIST** matches Setting 106. **TS HOLD POINT** is a combination of **TS POSITION** and **TS OFFSET** and matches Setting 107.

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. Highlight **TS POSITION**, press **[Z FACE MEASURE]** to take the value of the B axis and place it in **TS POSITION**. Highlight **X CLEARANCE**, press **[X DIAMETER MEASURE]** to take the value of the X Axis and place it in **X CLEARANCE**. Highlight **z CLEARANCE**, press **[Z FACE MEASURE]** to take the value of the Z Axis and place it in **z CLEARANCE**.
5. Highlight **X CLEARANCE** and press **[ORIGIN]** to set the clearance to max travel. Highlight **z CLEARANCE** and press **[ORIGIN]** to set clearance to zero.

5.11.5 Operation

Pick a program to run:

F5.49: Current Directory Memory Screen

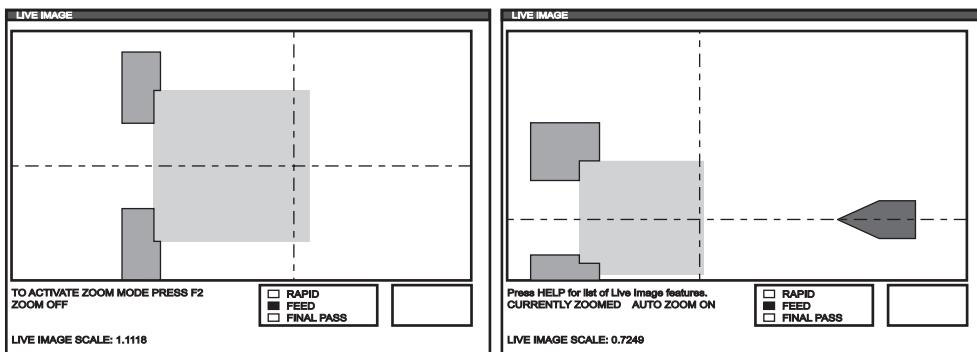


1. Select the desired program by pressing **[LIST PROGRAM]** to display the **EDIT: LIST** screen. Select the **MEMORY** tab and press **[ENTER]** to display **CURRENT DIRECTORY: MEMORY** screen.
2. Select a program (i.e., O01000) and press **[ENTER]** to choose it as the active program.

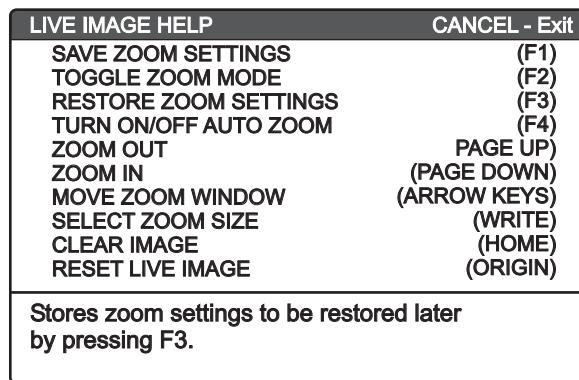
5.11.6 Run Part

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

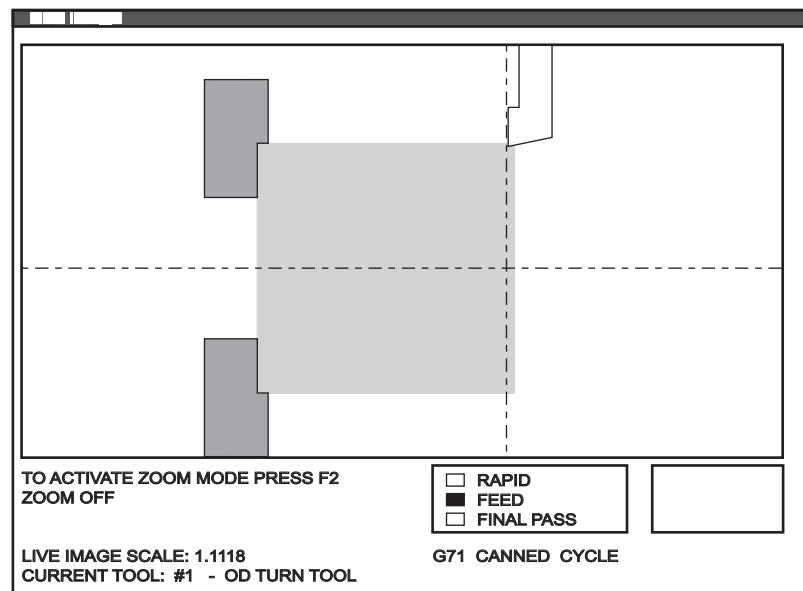
F5.50: Live Image Screen with Stock Drawn



F5.51: Live Image Feature List



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

F5.52: Live Image Tool working the Part**NOTE:**

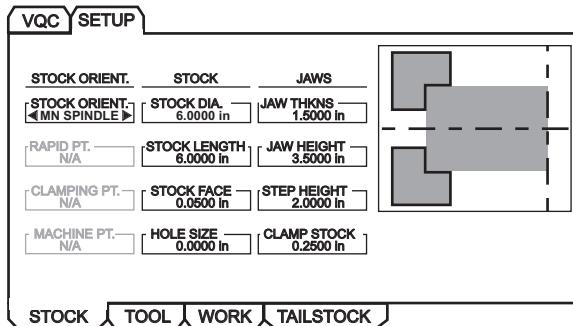
Data displayed on the screen while the program is running includes: program, main spindle, machine position, and timers and counters.

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] Select **Help Active Window Commands** for a pop-up menu with 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.

5.11.7 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.

F5.53: 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.

5.12 Tailstock Setup and Operation

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.

5.12.1 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.

5.13 Subprograms

5

Subprograms (subprograms):

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

When you use M97:

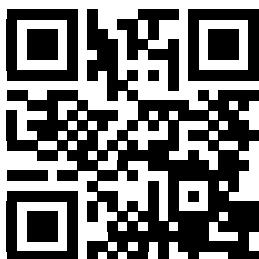
- The P code (nnnnn) is the same as the program location (Onnnnn) of the subprogram.
- The subprogram must be within the main program

When you use M98:

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

5.14 More Information Online

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



Chapter 6: Options Programming

6.1 Introduction

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

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

6.2 Macros (Optional)

6.2.1 Macros Introduction

**NOTE:**

This control feature is optional; call your HFO for information.

6

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

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

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

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 351)

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

Look-ahead

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

```
#1101 = 1 ;
G04 P1. ;
#1101 = 0 ;
```

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

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

Block Look-Ahead and Block Delete

The Haas control uses block look-ahead to read and prepare for blocks of code that come after the current block of code. This lets the control transition smoothly from one motion to the next. G103 limits how far ahead the control looks at blocks of code. The P_{nn} address code in G103 specifies how far ahead the control is allowed to look. For additional information, refer to G103 on page 313.

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

Using a

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

before a block with

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

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

6.2.2 Operation Notes

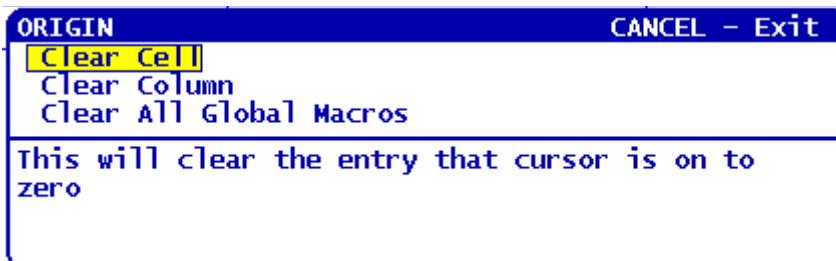
You can save or load macro variables through the RS-232 or USB port, much like settings, and offsets.

Variable Display Page

The macro variables #1 - #999 are displayed and modified through the Current Commands display.

1. Press **[CURRENT COMMANDS]** and use **[PAGE UP]/[PAGE DOWN]** to get to the **Macro Variables** page.
As the control interprets a program, the variable changes and results are displayed on the **Macro Variables** display page.
2. Enter a value and then press **[ENTER]** to set the macro variable. Press **[ORIGIN]** to clear macro variables, this displays the ORIGIN Clear entry popup. Make a selection from the choices and press **[ENTER]**.

- F6.1: [ORIGIN] Clear Entry Popup. **Clear Cell** - Clears the highlighted cell to zero.
Clear Column - Clears the active cursor column entries to zero.
Clear All Global Macros - Clears Global Macro entries (Macro 100-199, Macro 500-699, and Macro 800-999) to zero.



3. Entering the macro variable number and pressing up or down arrow searches for that variable.
4. The variables displayed represent values of the variables when the program runs. At times, this may be up to 15 blocks ahead of actual machine actions. Debugging programs is easier when a G103 P1 is inserted at the beginning of a program to limit block buffering and when G103 P1 is removed after debugging is completed.

Display User Defined Macros 1 and 2

You can display the values of any two user-defined macros (**Macro Label 1**, **Macro Label 2**).



NOTE:

The names Macro Label 1 and Macro Label 2 are changeable labels. just highlight the name, key in new name, and press [ENTER].

To set which two macro variables will display under **Macro Label 1** and **Macro Label 2** on the **Operation Timers & Setup** display window:

1. Press [**CURRENT COMMANDS**].
2. Press [**PAGE UP**] or [**PAGE DOWN**] to reach the **Operation Timers & Setup** page.
3. Use arrow keys to pick the **Macro Label 1** or **Macro Label 2** entry field (to the right of the label).
4. Key in the variable number (without #) and press [**ENTER**].

The field to the right of the entered variable number displays the current value.

Macro Arguments

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

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

Alphabetic Addressing

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

Alternate Alphabetic Addressing

Address	Variable	Address	Variable	Address	Variable
A	1	K	12	J	23
B	2	I	13	K	24

Address	Variable	Address	Variable	Address	Variable
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 subprograms according to this table:

Integer Argument Passing (no decimal point)

Address	Variable	Address	Variable	Address	Variable
A	.0001	J	.0001	S	1.
B	.0001	K	.0001	T	1.
C	.0001	L	1.	U	.0001
D	1.	M	1.	V	.0001
E	1.	N	-	W	.0001
F	1.	O	-	X	.0001
G	-	P	-	Y	.0001

Address	Variable		Address	Variable		Address	Variable
H	1.		Q	.0001		Z	.0001
I	.0001		R	.0001			

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

Example:

G65 P2000 I1 J2 K3 I4 J5 K6;

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

Macro Variables

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

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

Local Variables

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

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

Address:	W	X	Y	Z							
Alternate:	J	K	I	J	K	I	J	K	I	J	K

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

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

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

Global Variables

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

Sometimes, factory-installed options use global variables. For example, probing, pallet changers, etc.


CAUTION:

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

System Variables

System variables let you interact with a variety of control conditions. System variable values can change the function of the control. When a program reads a system variable, it can modify its behavior based on the value in the variable. Some system variables have a Read Only status; this means that you cannot modify them. A brief table of system variables follows with an explanation of their use.

VARIABLES	USAGE
#0	Not a number (read only)
#1-#33	Macro call arguments

VARIABLES	USAGE
#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)
#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

Macros (Optional)

VARIABLES	USAGE
#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)
#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

VARIABLES	USAGE
#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
#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

VARIABLES	USAGE
#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)	G112 (G154 P3) additional work offsets
#7061-#7066 (#14061-#14066)	G113 (G154 P4) additional work offsets
#7081-#7086 (#14081-#14086)	G114 (G154 P5) additional work offsets
#7101-#7106 (#14101-#14106)	G115 (G154 P6) additional work offsets
#7121-#7126 (#14121-#14126)	G116 (G154 P7) additional work offsets
#7141-#7146 (#14141-#14146)	G117 (G154 P8) additional work offsets
#7161-#7166 (#14161-#14166)	G118 (G154 P9) additional work offsets
#7181-#7186 (#14181-#14186)	G119 (G154 P10) additional work offsets
#7201-#7206 (#14201-#14206)	G120 (G154 P11) additional work offsets
#7221-#7226 (#14221-#14221)	G121 (G154 P12) additional work offsets

VARIABLES	USAGE
#7241-#7246 (#14241-#14246)	G122 (G154 P13) additional work offsets
#7261-#7266 (#14261-#14266)	G123 (G154 P14) additional work offsets
#7281-#7286 (#14281-#14286)	G124 (G154 P15) additional work offsets
#7301-#7306 (#14301-#14306)	G125 (G154 P16) additional work offsets
#7321-#7326 (#14321-#14326)	G126 (G154 P17) additional work offsets
#7341-#7346 (#14341-#14346)	G127 (G154 P18) additional work offsets
#7361-#7366 (#14361-#14366)	G128 (G154 P19) additional work offsets
#7381-#7386 (#14381-#14386)	G129 (G154 P20) additional work offsets
#8550	Tool/tool group id
#8552	Maximum recorded vibrations
#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

VARIABLES	USAGE
#14401-#14406	G154 P21 additional work offsets
#14421-#14426	G154 P22 additional work offsets
#14441-#14446	G154 P23 additional work offsets
#14461-#14466	G154 P24 additional work offsets
#14481-#14486	G154 P25 additional work offsets
#14501-#14506	G154 P26 additional work offsets
#14521-#14526	G154 P27 additional work offsets
#14541-#14546	G154 P28 additional work offsets
#14561-#14566	G154 P29 additional work offsets
#14581-#14586	G154 P30 additional work offsets
.	
⋮	
#14781 - #14786	G154 P40 additional work offsets
⋮	
#14981 - #14986	G154 P50 additional work offsets
⋮	
#15181 - #15186	G154 P60 additional work offsets
⋮	
#15381 - #15386	G154 P70 additional work offsets

VARIABLES	USAGE
⋮	
#15581 - #15586	G154 P80 additional work offsets
⋮	
#15781 - #15786	G154 P90 additional work offsets
⋮	
#15881 - #15886	G154 P95 additional work offsets
#15901 - #15906	G154 P96 additional work offsets
#15921 - #15926	G154 P97 additional work offsets
#15941 - #15946	G154 P98 additional work offsets
#15961-#15966	G154 P99 additional work offsets

6.2.3 System Variables In-Depth

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

Variables #550 through #580

These variables store probe calibration data. If these variables are overwritten, you will need to calibrate the probe again.

1-Bit Discrete Inputs

You can connect inputs designated as spare to external devices.

1-Bit Discrete Outputs

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

Maximum Axis Loads

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

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

Tool Offsets

Use these 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 overrides the Single Block function in G-code. When #3003 has a value of 1, the control executes each G-code command continuously even though the Single Block function is ON. When #3003 has a value of zero, Single Block operates as normal. You must press **[CYCLE START]** to execute each line of code in single block mode.

```
#3003=1 ;
G54 G00 X0 Z0 ;
G81 R0.2 Z-0.1 F.002 L0 ;
S2000 M03 ;
#3003=0 ;
T02 M06 ;
Q.05 G83 R0.2 Z-1. F.001 L0 ;
X0. Z0. ;
```

Variable #3004

Variable #3004 overrides specific control features during operation.

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

```
%  
    (Approach code - [FEED HOLD] allowed) ;  
#3004=1 (Disables [FEED HOLD]) ;  
    (Non-stoppable code - [FEED HOLD] not allowed) ;  
#3004=0 (Enables [FEED HOLD]) ;  
    (Depart code - [FEED HOLD] allowed) ;  
%
```

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

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

#3006 Programmable Stop

You can add stops to the program that act like an M00 - The control stops and waits until you press **[CYCLE START]**, then the program continues with the block after the #3006. In this example, the control displays the first 15 characters of the comment on the lower-left part of the screen.

```
#3006=1 (comment here) ;
```

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

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

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

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

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

#5001-#5006 Last Target Position

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

Axis Position Variables

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

#5021-#5026 Current Machine Coordinate Position

To get the current machine axis positions, call macro variables #5021- #5025 corresponding to axis X, Z, Y, A, and B, respectively.



NOTE:

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

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

**NOTE:**

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

The value of #5042 (Z) has tool length compensation applied to it.

#5061-#5069 Current Skip Signal Position

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

The value of #5062 (Z) has tool length compensation applied to it.

#5081-#5086 Tool Length Compensation

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

#6996-#6999 Parameter Access With Macro Variables

These macro variables can access parameters 1 to 1000 and any of the parameter bits, as follows:

#6996: Parameter Number

#6997: Bit Number (optional)

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

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

**NOTE:**

Variables #6998 and #6999 are read-only.

Usage

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

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

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

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

**NOTE:**

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

Pallet Changer Variables

The status of the pallets from the Automatic Pallet Changer is checked with these variables:

#7501-#7506	Pallet priority
#7601-#7606	Pallet status
#7701-#7706	Part program numbers assigned to pallets
#7801-#7806	Pallet usage count
#3028	Number of pallet loaded on receiver

Work Offsets

Macro expressions can read and set all work offsets. This lets you preset coordinates to approximate locations, or set coordinates to values based upon the results of skip signal locations and calculations. When any of the offsets are read, the interpretation look-ahead queue is stopped until that block is executed.

#5201- #5206	G52 X, 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

Macros (Optional)

#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
#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

#8550-#8567 Tooling

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



NOTE:

Macro variables #1801-#2000 give access to the same data as #8550-#8567.

6.2.4 Variable Usage

All variables are referenced with a number sign (#) followed by a positive number: #1, #101, and #501.

Variables are decimal values that are represented as floating point numbers. If a variable has never been used, it can take on a special **undefined** value. This indicates that it has not been used. A variable can be set to **undefined** with the special variable #0. #0 has the value of undefined or 0.0 depending on its context. Indirect references to variables can be accomplished by enclosing the variable number in brackets: # [<Expression>]

The expression is evaluated and the result becomes the variable accessed. For example:

```
#1=3 ;
# [#1]=3.5 + #1 ;
```

This sets the variable #3 to the value 6.5.

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

In the block:

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

the variables can be set to the following values:

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

and replaced by:

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

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

6.2.5 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 this 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><-><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 example, this 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. If #1=.123456, then G01 X#1 would move the machine tool to .1235 on the X Axis. If the control is in the metric mode, the machine would be moved to .123 on the X axis.

When an undefined variable is used to replace an address value, that address reference is ignored. For example:

```
(#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
ABS[]	Decimal	Decimal	Absolute value
ROUND[]	Decimal	Decimal	Round off a decimal
FIX[]	Decimal	Integer	Truncate fraction

Function	Argument	Returns	Notes
ACOS[]	Decimal	Degrees	Arc cosine
ASIN[]	Decimal	Degrees	Arcsine
#[]	Integer	Integer	Variable Indirection
DPRNT[]	ASCII text	External Output	

Notes on Functions

The function ROUND works differently depending on the context that it is used. When used in arithmetic expressions, any number with a fractional part greater than or equal to .5 is rounded up to the next whole integer; otherwise, the fractional part is truncated from the number.

```
%  
#1=1.714 ;  
#2=ROUND[#1] (#2 is set to 2.0) ;  
#1=3.1416 ;  
#2=ROUND[#1] (#2 is set to 3.0) ;  
%
```

When ROUND is used in an address expression, the result is rounded to the significant precision. For metric and angle dimensions, three-place precision is the default. For inch, four-place precision is the default.

```
%  
#1= 1.00333 ;  
G00 X[ #1 + #1 ] ;  
(Table X Axis moves to 2.0067) ;  
G00 X[ ROUND[ #1 ] + ROUND[ #1 ] ] ;  
(Table X Axis moves to 2.0066) ;  
G00 A[ #1 + #1 ] ;  
(Axis rotates to 2.007) ;  
G00 A[ ROUND[ #1 ] + ROUND[ #1 ] ] ;  
(Axis rotates to 2.006) ;  
D[1.67] (Diameter 2 is made current) ;  
%
```

Fix vs. Round

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

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

Operators

Operators have (3) categories: Boolean, Arithmetic, and Logical.

Boolean Operators

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

EQ - Equal To

NE - Not Equal To

GT - Greater Than

LT - Less Than

GE - Greater Than or Equal To

LE - Less Than or Equal To

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

Example	Explanation
IF [#1 EQ 0.0] GOTO100 ;	Jump to block 100 if value in variable #1 equals 0.0.
WHILE [#101 LT 10] DO1 ;	While variable #101 is less than 10 repeat loop DO1..END1.
#1=[1.0 LT 5.0] ;	Variable #1 is set to 1.0 (TRUE).
IF [#1 AND #2 EQ #3] GOTO1 ;	If variable #1 AND variable #2 are equal to the value in #3 then control jumps to block 1.

Arithmetic Operators

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

+	- Unary plus	+1.23
-	- Unary minus	-[COS[30]]
+	- Binary addition	#1=#1+5
-	- Binary subtraction	#1=#1-1
*	- Multiplication	#1=#2*#3

/	- Division	#1=#2/4
MOD	- Remainder	#1=27 MOD 20 (#1 contains 7)

Logical Operators

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

OR - logically OR two values together

XOR - Exclusively OR two values together

AND - Logically AND two values together

Examples:

```
%  
#1=1.0 ;  
#2=2.0 ;  
#3=#1 OR #2 ;  
%
```

Here the variable #3 will contain 3.0 after the OR operation.

```
%  
#1=5.0 ;  
#2=3.0 ;  
IF [[#1 GT 3.0] AND [#2 LT 10]] GOTO1 ;  
%
```

Here control will transfer to block 1 because #1 GT 3.0 evaluates to 1.0 and #2 LT 10 evaluates to 1.0, thus 1.0 AND 1.0 is 1.0 (TRUE) and the GOTO occurs.



NOTE:

To achieve your desired results, be very careful when you use logical operators.

Expressions

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

Arithmetic Expressions

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

Examples of Arithmetic expressions:

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

Conditional Expressions

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

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

```
N50 M99 P10 ;
```

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

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

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

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

Assignment Statements

Assignment statements let you modify variables. The format of the assignment statement is:

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

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

```
%  
O50001 (INITIALIZE A SEQUENCE OF VARIABLES) ;  
N1 IF [#2 NE #0] GOTO2 (B=base variable) ;  
#3000=1 (Base variable not given) ;
```

```
N2 IF [#19 NE #0] GOTO3 (S=size of array) ;
#3000=2 (Size of array not given) ;
N3 WHILE [#19 GT 0] DO1 ;
#19=#19-1 (Decrement count) ;
#[#2+#19]=#22 (V=value to set array to) ;
END1 ;
M99 ;
%
```

You could use the above macro to initialize three sets of variables as follows:

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

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

Control Statements

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

Unconditional Branch (GOTOnnn and M99 Pnnnn)

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

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

Computed branching allows the program to transfer control to another line of code in the same subprogram. The control can compute the block while the program runs, using the GOTO [expression] form, or it can pass the block in through a local variable, as in the GOTO#n form.

The GOTO rounds the variable or expression result that is associated with the Computed branch. For instance, if variable #1 contains 4.49 and the program contains a GOTO#1 command, the control attempts to transfer to a block that contains N4. If #1 contains 4.5, then the control transfers to a block that contains N5.

Example: You could develop this code skeleton into a program that adds serial numbers to parts:

```
%  
O50002 (COMPUTED BRANCHING) ;
(D=Decimal digit to engrave) ;
;  
IF [[#7 NE #0] AND [#7 GE 0] AND [#7 LE 9]] GOTO99 ;
```

```

#3000=1 (Invalid digit) ;
;
N99;
#7=FIX[#7] (Truncate any fractional part) ;
;
GOTO#7 (Now engrave the digit) ;
;
N0 (Do digit zero) ;
M99 ;
;
N1 (Do digit one) ;
;
M99 ;
%

```

With the above subprogram, you would use this call to engrave the fifth digit:

```
G65 P9200 D5 ;
```

Computed GOTOS using expression could be used to branch processing based on the results of reading hardware inputs. For example:

```

%
GOTO [[#1030*2]+#1031] ;
N0 (1030=0, 1031=0) ;
...M99 ;
N1 (1030=0, 1031=1) ;
...M99 ;
N2 (1030=1, 1031=0) ;
...M99 ;
N3 (1030=1, 1031=1) ;
...M99 ;
%
#1030 and #1031.

```

Conditional Branch (IF and M99 Pnnnn)

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

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

As discussed, <conditional expression> is any expression that uses any of the six Boolean operators EQ, NE, GT, LT, GE, or LE. The brackets surrounding the expression are mandatory. In the Haas control, it is not necessary to include these operators. For example:

```
IF [#1 NE 0.0] GOTO5 ;
```

could also be:

```
IF [#1] GOTO5 ;
```

In this statement, if the variable #1 contains anything but 0.0, or the undefined value #0, then branching to block 5 occurs; otherwise, the next block is executed.

In the Haas control, a <conditional expression> is also used with the M99 Pnnnn format. For example:

```
G00 X0 Y0 [#1EQ#2] M99 P5;
```

Here, the conditional is for the M99 portion of the statement only. The machine tool is instructed to go to X0, Y0 whether or not the expression evaluates to True or False. Only the branch, M99, is executed based on the value of the expression. It is recommended that the IF GOTO version be used if portability is desired.

Conditional Execution (IF THEN)

Execution of control statements can also be achieved by using the IF THEN construct. The format is:

```
IF [<conditional expression>] THEN <statement> ;
```



NOTE:

To preserve compatibility with FANUC syntax THEN may not be used with GOTOn.

This format is traditionally used for conditional assignment statements such as:

```
IF [#590 GT 100] THEN #590=0.0 ;
```

Variable #590 is set to zero when the value of #590 exceeds 100.0. In the Haas control, if a conditional evaluates to FALSE (0.0), then the remainder of the IF block is ignored. This means that control statements can also be conditioned so that we could write something like:

```
IF [#1 NE #0] THEN G01 X#24 Y#26 F#9 ;
```

This executes a linear motion only if variable #1 has been assigned a value. Another example is:

```
IF [#1 GE 180] THEN #101=0.0 M99 ;
```

This says that if variable #1 (address A) is greater than or equal to 180, then set variable #101 to zero and return from the subprogram.

Here is an example of an IF statement that branches if a variable has been initialized to contain any value. Otherwise, processing continues and an alarm is generated. Remember, when an alarm is generated, program execution is halted.

```
%  
N1 IF [#9NE#0] GOTO3 (TEST FOR VALUE IN F) ;  
N2 #3000=11 (NO FEED RATE) ;  
N3 (CONTINUE) ;  
%
```

Iteration/Looping (WHILE DO END)

Essential to all programming languages is the ability to execute a sequence of statements a given number of times or to loop through a sequence of statements until a condition is met. Traditional G coding allows this with the use of the L address. A subprogram can be executed any number of times by using the L address.

```
M98 P2000 L5 ;
```

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

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

This executes the statements between DOn and ENDn as long as the conditional expression evaluates to True. The brackets in the expression are necessary. If the expression evaluates to False, then the block after ENDn is executed next. WHILE can be abbreviated to WH. The DOn-ENDn portion of the statement is a matched pair. The value of n is 1-3. This means that there can be no more than three nested loops per subprogram. A nest is a loop within a loop.

Although nesting of WHILE statements can only be up to three levels, there really is no limit since each subprogram can have up to three levels of nesting. If there is a need to nest to a level greater than 3, then the segment containing the three lowest levels of nesting can be made into a subprogram thus overcoming the limitation.

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

```
%  
#3001=0 (WAIT 500 MILLISECONDS) ;  
WH [#3001 LT 500] D01 ;  
END1 ;  
<Other statements>  
#3001=0 (WAIT 300 MILLISECONDS) ;  
WH [#3001 LT 300] D01 ;  
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,

```
%  
D01 ;  
<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).

6.2.6 G65 Macro Subprogram Call Option (Group 00)

G65 is the command that calls a subprogram with the ability to pass arguments to it. The format follows:

```
G65 Pnnnn [Lnnnn] [arguments] ;
```

Arguments italicized in square brackets are optional. See the Programming section for more details on macro arguments.

The G65 command requires a **P** address corresponding to a program number currently in the control's memory. When the **L** address is used the macro call is repeated the specified number of times.

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

Example 1:

```
%  
G65 P1000 (Call subprogram 1000 as a macro) ;  
M30 (Program stop) ;  
O01000 (Macro Subprogram) ;  
...  
M99 (Return from Macro Subprogram) ;  
%
```

Aliasing

Aliased codes are user defined G and M-codes that reference a macro program. There are 10 G alias codes and 10 M alias codes available to users.

Aliasing is a means of assigning a G-code or M-code to a G65 P##### sequence. For instance, in the previous Example 2, it would be easier to write:

```
G06 X.5 Y.25 Z.05 F10. T10 ;
```

When aliasing, variables can be passed with a G-code; variables cannot be passed with an M-code.

Here, an unused G code has been substituted, G06 for G65 P9010. In order for the previous block to work, the parameter associated with subprogram 9010 must be set to 06 (Parameter 91).

**NOTE:**

G00, G65, G66, and G67 cannot be aliased. All other codes between 1 and 255 can be used for aliasing.

Program numbers 9010 through 9019 are reserved for G code aliasing. This table lists which Haas parameters are reserved for macro subprogram aliasing.

F6.2: G- and M-code Aliasing

Haas Parameter	O Code	Haas Parameter	O Code
81	9000	91	9010
82	9001	92	9011
83	9002	93	9012
84	9003	94	9013
85	9004	95	9014
86	9005	96	9015
87	9006	97	9016
88	9007	98	9017
89	9008	99	9018
90	9009	100	9019

Setting an aliasing parameter to 0 disables aliasing for the associated subprogram. If an aliasing parameter is set to a G-code and the associated subprogram is not in memory, then an alarm will be given. When a G65 macro, Aliased-M or Aliased-G code is called, the control first looks for the sub-program in **MEM**. If it is not found in **MEM**, the control then looks for the sub-program on the active drive (**USB**, **HDD**). An alarm occurs if the sub-program is not found.

When a G65 macro, Aliased-M or Aliased-G code is called, the control looks for the sub-program in memory and then in any other active drive if the sub-program cannot be located. The active drive may be memory, USB drive or hard drive. An alarm occurs if the control does not find the sub-program in either memory or an active drive.

6.2.7 Communication With External Devices - DPRNT[]

Macros allow additional capabilities to communicate with peripheral devices. With user provided devices you can digitize parts, provide runtime inspection reports, or synchronize controls. The commands provided for this are **POPN**, **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. These formats are illegal: [00] [54] [45] [36] /* not legal formats */

A decimal point is printed out between the whole part and the fractional part. The fractional part is rounded to the least significant place. When zero places are reserved for the fractional part, then no decimal point is printed out. Trailing zeros are printed if there is a fractional part. At least one place is reserved for the whole part, even when a zero is used. If the value of the whole part has fewer digits than have been reserved, then leading spaces are output. If the value of the whole part has more digits than have 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 look-ahead interpretation to one block, you would include this command at the start of your program: This causes the control to look ahead (2) blocks.

```
G103 P1 ;
```

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

Editing

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

The DPRNT [] function can be edited much like a comment. It can be deleted, moved as a whole item, or individual items within the bracket can be edited. Variable references and format expressions must be altered as a whole entity. If you wanted to change [24] to [44], place the cursor so that [24] is highlighted, enter [44] and press [**ENTER**]. Remember, you can use the [**HANDLE JOG**] control to maneuver through long DPRNT [] expressions.

Addresses with expressions can be somewhat confusing. In this case, the alphabetic address stands alone. For instance, this block contains an address expression in X:

```
G01 G90 X [COS [90]] Y3.0 (CORRECT) ;
```

Here, the X and brackets stand-alone and are individually editable items. It is possible, through editing, to delete the entire expression and replace it with a floating-point constant.

```
G01 G90 X 0 Y3.0 (WRONG) ;
```

The above block will result in an alarm at runtime. The correct form looks as follows:

```
G01 G90 X0 Y3.0 (CORRECT) ;
```



NOTE:

There is no space between the X and the Zero (0). REMEMBER when you see an alpha character standing alone it is an address expression.

6.2.8 Fanuc-Style Macros Not Included

This section lists the FANUC macro features that are not available on the Haas control.

M Aliasing Replace G65 Pnnnn with Mnn PROGS 9020-9029.

G66	Modal call in every motion block
G66.1	Modal call in every motion block

G67	Modal cancel
M98	Aliasing, T code PROG 9000, VAR #149, enable bit
M98	Aliasing, B Code PROG 9028, VAR #146, enable bit
SKIP/N	N=1..9
#3007	Mirror image on flag each axis
#4201-#4320	Current block modal data
#5101-#5106	Current servo deviation

Names for Variables for Display Purposes:

ATAN []/[]	Arctangent, FANUC version
BIN []	Conversion from BCD TO BIN
BCD []	Conversion from BIN TO BCD
FUP []	Truncate fraction ceiling
LN []	Natural logarithm
EXP []	Base E Exponentiation
ADP []	Re-Scale variable to whole number
BPRNT []	

GOTO-nnnn

Searching for a block to jump in the negative direction, i.e. backwards through a program, is not necessary if you use unique N address codes.

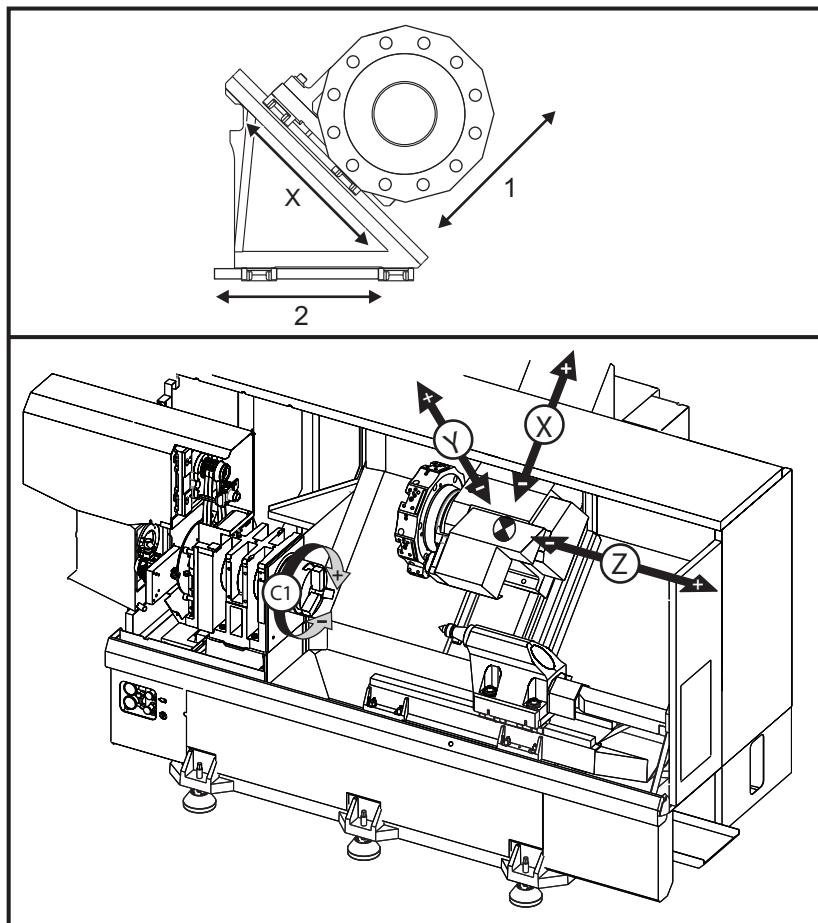
A block search is made starting from the current block being interpreted. When the end of the program is reached, searching continues from the top of the program until the current block is encountered.

6.3 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 261, for programming information.

F6.3: Y-Axis Motion: [1] Y-Axis compound motion, [2] Horizontal plane.



6.3.1 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

6.3.2 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.

6.3.3 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-codes 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 the 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 268.

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 **249** for more information.

Axial Only Cycles:

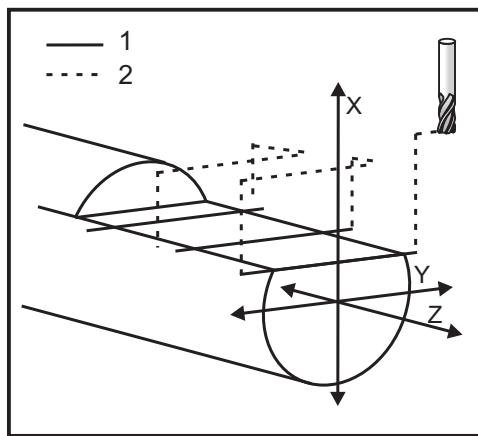
- Drilling: G74, G81, G82, G83,
- Boring: G85, G89,
- Tapping: G95, G186,

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:

F6.4: Y-axis Milling Program Example: [1] Feed, [2] Rapid.



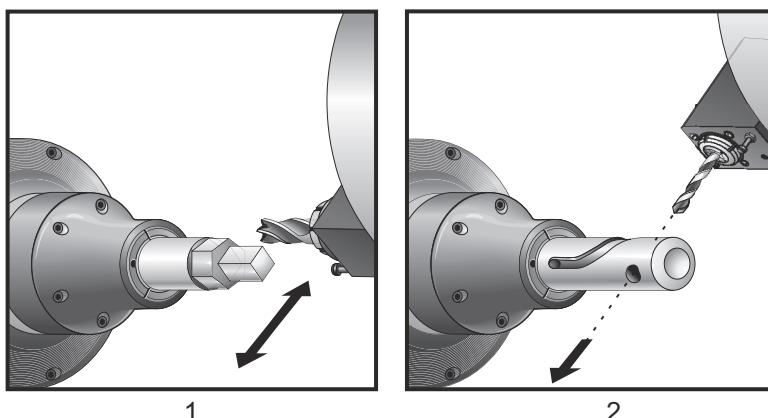
```
%  
o50004 (Y AXIS MILLING) ;  
(G54 X0 Y0 is at the center of rotation) ;  
(Z0 is on face of the part) ;  
(T1 is an end mill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G19 (Call YZ plane) ;  
G98 (Feed per min) ;  
M154 (Engage 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) ;  
%
```

6.4 Live Tooling

This option is not field installable.

F6.5: Axial and Radial Live Tooling: [1] Axial Tool, [2] Radial Tool.



6.4.1 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.

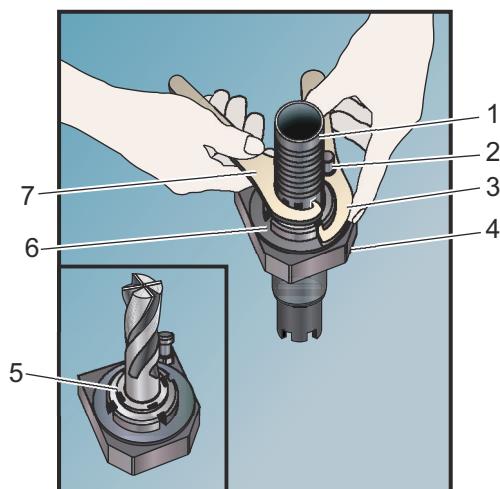
For the best milling accuracy, use the spindle clamp M Codes (M14 - Main Spindle / M114 - Secondary Spindle) before machining. The spindle automatically unclamps when a new main spindle speed is commanded or [RESET] is pressed.

Maximum live tooling drive speed is 6000 RPM.

Haas live tooling is designed for medium duty milling, e.g.: 3/4" diameter end mill in mild steel max.

6.4.2 Live Tooling Cutting Tool Installation

F6.6: 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. Tighten 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.

6.4.3 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.

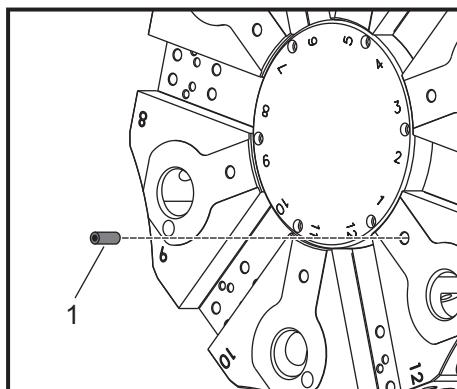
Adjustment set screws are standard on all radial live tool heads. An alignment dowel pin is included in Haas radial live tool kits.

Mounting and Alignment

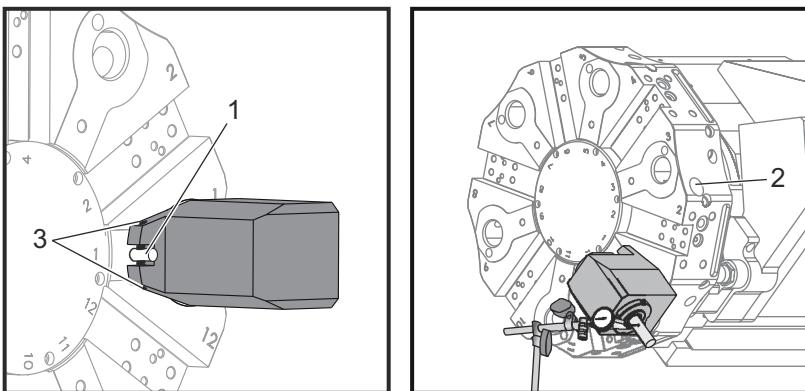
To mount and install live tools:

1. Install the alignment dowel pin that comes with the Haas-supplied live tooling holder on the turret.

F6.7: Install the 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. Tighten the VDI allen bolt [2] 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.

F6.8: 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 [2] 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.

6.4.4 Live Tooling M-codes

The following M-Codes are used in Live Tooling. Also, refer to the M-codes section starting on page 337.

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 354 for a complete description of these M-codes.

6.5 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.

6.5.1 Cartesian to Polar Transformation (G112)

Cartesian to Polar coordinate programming 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 **248** for more information on modal G-codes.

6.5.2 Cartesian Interpolation

Cartesian coordinate commands are interpreted into movements of the linear axis (turret movements) and spindle movements (rotation of the workpiece).

Example Program

```
%  
o51120 (CARTESIAN INTERPOLATION) ;  
(G54 X0 Y0 is at the center of rotation) ;  
(Z0 is on face of the part) ;  
(T1 is an end mill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G20 G40 G80 G97 G99 (Safe startup) ;  
G17 G112 (Call XY plane, XY to XC interpretation) ;  
G98 (Feed per min) ;  
M154 (Engage C-Axis) ;  
G00 G54 X2.35 C0. Y0. Z0.1 ;  
(Rapid to 1st position) ;  
P1500 M133 (Live tool CW at 1500 RPM) ;
```

```
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
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) ;
(BEGIN COMPLETION BLOCKS) ;
G113 (Cancel G112) ;
M155 (Disengage C axis) ;
M135 (Live tool off) ;
G18 (Return to XZ plane) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 (Z home) ;
M30 (End program) ;
%
```

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.

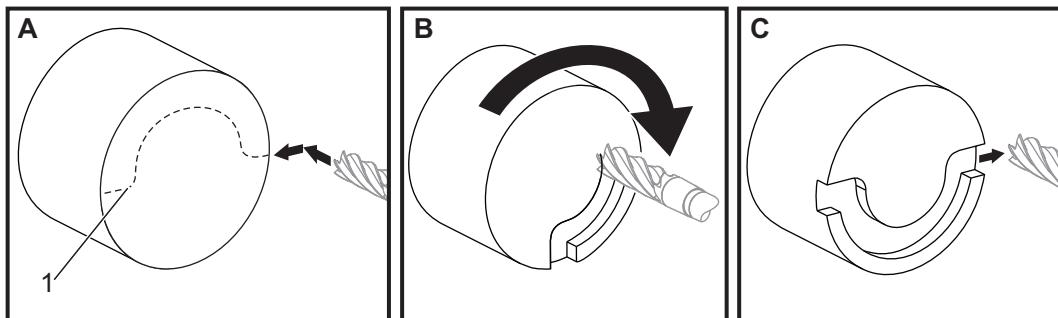
The lathe automatically disengages the spindle brake when you command the C-Axis to move and to reengage it afterwards if the M codes are still active.

C-Axis incremental moves are possible using the H address code as shown in this example:

```
G0 C90. (C-Axis moves to 90. deg.) ;
H-10. (C-Axis moves to 80. deg. from the previous 90 deg
position) ;
```

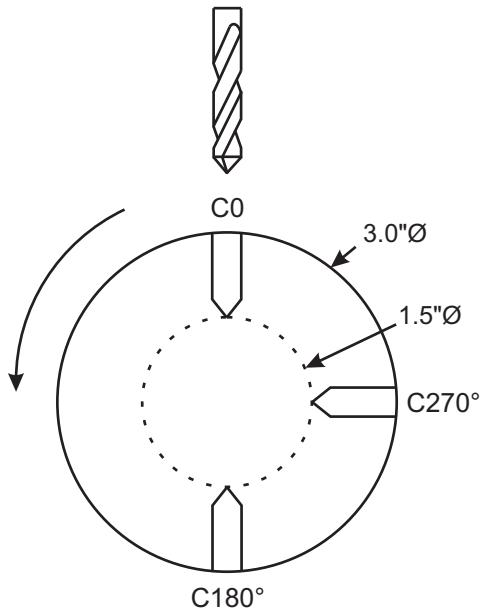
Sample Programs

- F6.9:** Cartesian Interpolation Example 1. (A) Projected Cutting Path (B) The C Axis turns 180 degrees to cut the arc shape. (C) The endmill feeds 1" out of the workpiece.



```
%o51121 (CARTESIAN INTERPOLATION EX 1) ;
(G54 X0 Y0 is at the center of rotation) ;
(Z0 is on 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) ;
G98 (Feed per min) ;
M154 (Engage C Axis) ;
G00 G54 X2. C90 Z0.1 (Rapid to 1st position) ;
P1500 M133 (Live tool CW at 1500 RPM) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G01 Z-0.1 F6.0 (Feed to Z depth) ;
X1.0 (Feed to Position 2) ;
C180. F10.0 (Rotate to cut arc) ;
X2.0 (Feed back to Position 1) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.5 M09 (Rapid retract, coolant 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) ;
%
```

F6.10: Cartesian Interpolation Example 2



%
o51122 (CARTESIAN INTERPOLATION EX 2);
(G54 X0 Y0 is at the center of rotation) ;
(Z0 is on 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 depth) ;
(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) ;

```

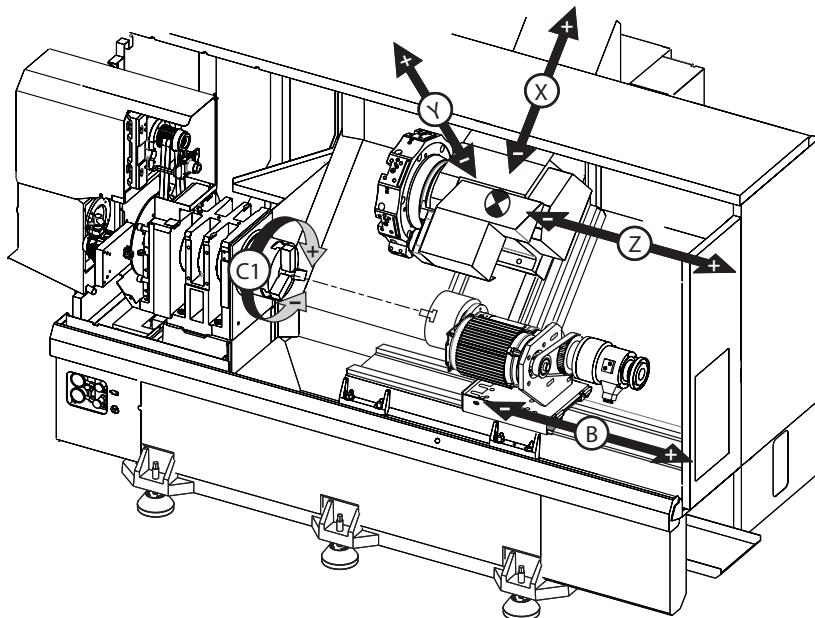
M135 (Live tool off) ;
G18 (Return to XZ plane) ;
G53 X0 (X home) ;
G53 Z0 (Z home) ;
M30 (End program) ;
%

```

6.6 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.

F6.11: Dual Spindle Lathe with an Optional Y Axis



6.6.1 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

F6.12:

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	0.0000
G199 R PHASE OFS			
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:

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 **322** 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.

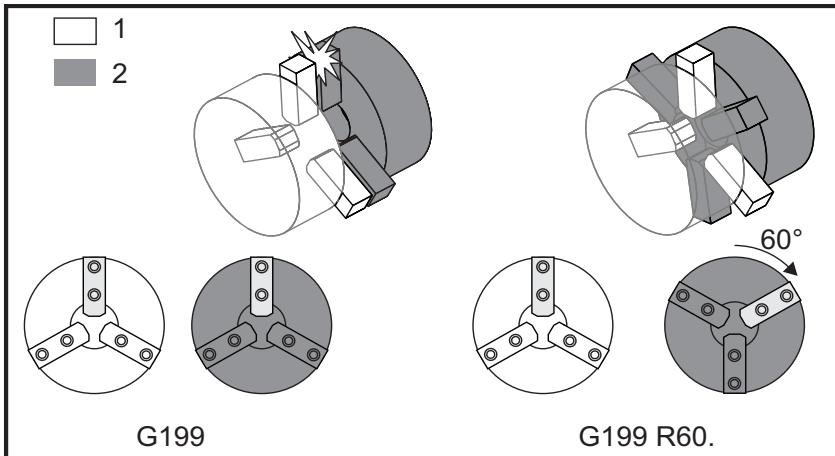
LOAD % - This shows the current load percent for each spindle.

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 **F6.13** for an example.

F6.13: 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.

6.6.2 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 260 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 384 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.

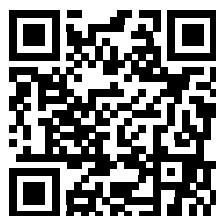
6.7 More Information Online

You can find programming information for other optional equipment in the online Haas Resource Center, including:

- High Pressure Coolant (HPC)
- Automatic Tool Setting Probe
- Servo Auto Door

To access the site, go to www.HaasCNC.com and select the **Haas Resource Center**.

You can also scan this QR code with your mobile device to go directly to the options programming section of the Resource Center.



Chapter 7: G-codes

7.1 Introduction

This chapter gives detailed descriptions of the G-codes that you use to program your machine.



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



NOTE: *The sample programs in this manual represent a very conservative programming style. The samples are intended to demonstrate safe and reliable programs, and they are not necessarily the fastest or most efficient way to operate a machine. The sample programs use G-codes that you might choose not to use in more efficient programs.*

7.1.1 List of G-codes

Code	Description	Group	Page
G00	Rapid Motion Positioning	01	250
G01	Linear Interpolation Motion	01	250
G02	CW Circular Interpolation Motion	01	256
G03	CCW Circular Interpolation Motion	01	256
G04	Dwell	00	259
G09	Exact Stop	00	259
G10	Set Offsets	00	260
G14	Secondary Spindle Swap	17	260

Code	Description	Group	Page
G15	Secondary Spindle Cancel	17	260
G17	XY Plane	00	261
G18	XZ Plane	02	261
G19	YZ Plane	02	261
G20	Select Inches	06	261
G21	Select Metric	06	261
G28	Return To Machine Zero Point	00	261
G29	Return From Reference Point	00	262
G31	Skip Function	00	262
G32	Thread Cutting	01	263
G40	Tool Nose Compensation Cancel	07	265
G41	Tool Nose Compensation (TNC) Left	07	266
G42	Tool Nose Compensation (TNC) Right	07	266
G50	Set Global coordinate Offset FANUC, YASNAC	00	266
G51	Cancel Offset (YASNAC)	00	267
G52	Set Local Coordinate System FANUC	00	268
G53	Machine Coordinate Selection	00	268
G54	Coordinate System #1 FANUC	12	268
G55	Coordinate System #2 FANUC	12	268
G56	Coordinate System #3 FANUC	12	268
G57	Coordinate System #4 FANUC	12	268
G58	Coordinate System #5 FANUC	12	268
G59	Coordinate System #6 FANUC	12	268

Code	Description	Group	Page
G61	Exact Stop Modal	15	268
G64	Exact Stop Cancel G61	15	268
G65	Macro Subprogram Call Option	00	268
G70	Finishing Cycle	00	268
G71	O.D./I.D. Stock Removal Cycle	00	270
G72	End Face Stock Removal Cycle	00	279
G73	Irregular Path Stock Removal Cycle	00	285
G74	End Face Grooving Cycle	00	287
G75	O.D./I.D. Grooving Cycle	00	289
G76	Threading Cycle, Multiple Pass	00	292
G80	Canned Cycle Cancel	09	296
G81	Drill Canned Cycle	09	296
G82	Spot Drill Canned Cycle	09	297
G83	Normal Peck Drilling Canned Cycle	09	298
G84	Tapping Canned Cycle	09	300
G85	Boring Canned Cycle	09	303
G86	Bore and Stop Canned Cycle	09	304
G87	Bore and Manual Retract Canned Cycle	09	304
G88	Bore and Dwell and Manual Retract Canned Cycle	09	305
G89	Bore and Dwell Canned Cycle	09	306
G90	O.D./I.D. Turning Cycle	01	306
G92	Threading Cycle	01	307
G94	End Facing Cycle	01	310

Code	Description	Group	Page
G95	Live Tooling Rigid Tap (Face)	09	311
G96	Constant Surface Speed On	13	312
G97	Constant Surface Speed Off	13	312
G98	Feed Per Minute	10	312
G99	Feed Per Revolution	10	312
G100	Disable Mirror Image	00	312
G101	Enable Mirror Image	00	312
G102	Programmable Output to RS-232	00	313
G103	Limit Block Lookahead	00	313
G105	Servo Bar Command	09	314
G110	Coordinate System #7	12	315
G111	Coordinate System #8	12	315
G112	XY to XC interpretation	04	312
G113	Cancel G112	04	316
G114	Coordinate System #9	12	316
G115	Coordinate System #10	12	316
G116	Coordinate System #11	12	316
G117	Coordinate System #12	12	316
G118	Coordinate System #13	12	316
G119	Coordinate System #14	12	316
G120	Coordinate System #15	12	316
G121	Coordinate System #16	12	316
G122	Coordinate System #17	12	316

Code	Description	Group	Page
G123	Coordinate System #18	12	316
G124	Coordinate System #19	12	316
G125	Coordinate System #20	12	316
G126	Coordinate System #21	12	316
G127	Coordinate System #22	12	316
G128	Coordinate System #23	12	316
G129	Coordinate System #24	12	316
G154	Select Work Coordinates P1-99	12	317
G159	Background Pickup / Part Return		318
G160	APL Axis Command Mode Only		319
G161	APL Axis Command Mode Off		319
G184	Reverse Tapping Canned Cycle For Left Hand Threads	09	319
G186	Reverse Live Tool Rigid Tap (For Left Hand Threads)	10	320
G187	Accuracy Control	00	320
G195	Forward Live Tool Radial Tapping (Diameter)	00	321
G196	Reverse Live Tool Radial Tapping (Diameter)	00	321
G198	Disengage Synchronous Spindle Control	00	310
G199	Engage Synchronous Spindle Control	00	322
G200	Index on the Fly	00	324
G211	Manual Tool Setting		325
G212	Auto Tool Setting		325
G241	Radial Drill Canned Cycle	09	325
G242	Radial Spot Drill Canned Cycle	09	327

Code	Description	Group	Page
G243	Radial Normal Peck Drilling Canned Cycle	09	328
G245	Radial Boring Canned Cycle	09	330
G246	Radial Bore and Stop Canned Cycle	09	331
G247	Radial Bore and Manual Retract Canned Cycle	09	332
G248	Radial Bore and Dwell and Manual Retract Canned Cycle	09	333
G249	Radial Bore and Dwell Canned Cycle	09	334

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.

Each G-code has a group number. Each group of codes contains 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 are either modal or non-modal. A modal G-code stays in effect until the end of the program or until you command another G-code from the same group. A non-modal G-code affects only the line it is in; it does not affect the next program line. Group 00 codes are non-modal; the other groups are modal.


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

Canned cycles simplify part programming. Most common Z-axis repetitive operations, such as drilling, tapping, and boring, have canned cycles. When active, a canned cycle executes at every new axis position. Canned cycles execute axis motions as rapid commands (G00) and the canned cycle operation is performed after the axis motion. Applies to G17, G19 cycles, and Y-axis movements on Y-axis lathes.

Using Canned Cycles

Modal canned cycles stay in effect after you define them, and they execute in the Z-axis for each position of the X, Y, and C-Axes.



NOTE:

X, Y, or C-Axis positioning moves during a canned cycle are rapid moves.

Canned cycles operate differently, depending on whether you use incremental (U,W) or absolute (X, Y, or C) positions.

If you define a loop count (Lnn code number) in the canned cycle block, the canned cycle repeats that many times with an incremental (U or W) move between each cycle.

Enter the number of repeats (L) each time you want to repeat a canned cycle. The control does not remember the number of repeats (L) for the next canned cycle.

You should not use spindle control M-codes while a canned cycle is active.

7

Canceling a Canned Cycle

G80 cancels all canned cycles. G00 or G01 code also cancel a canned cycle. A canned cycle stays active until G80, G00, or G01 cancels it.

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.

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.

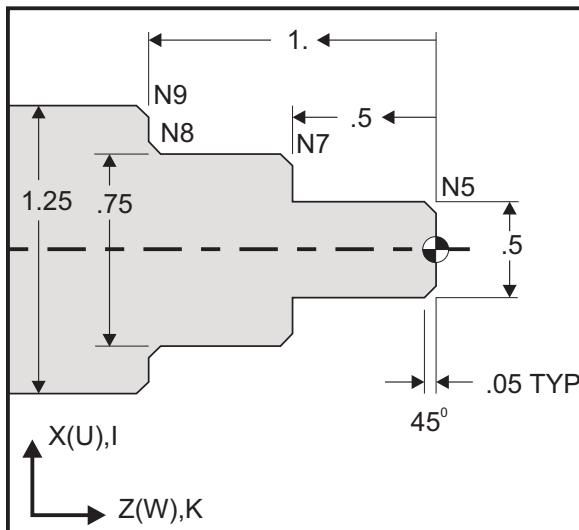
Corner Rounding and Chamfering

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 after 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 after 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 after 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.

F7.1: Chamfering


```
%  
o60011 (G01 CHAMFERING) ;  
(G54 X0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is an OD cutting tool) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;
```

```
G50 S1000 (Limit spindle to 1000 RPM) ;
G97 S500 M03 (CSS off, Spindle on CW) ;
G00 G54 X0 Z0.25 (Rapid to 1st position) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G01 Z0 F0.005 (Feed to Z0) ;
N5 G01 X0.50 K-0.050 (Chamfer 1) ;
G01 Z-0.5 (Linear feed to Z-0.5) ;
N7 G01 X0.75 K-0.050 (Chamfer 2) ;
N8 G01 Z-1.0 I0.050 (Chamfer 3) ;
N9 G01 X1.25 K-0.050 (Chamfer 4) ;
G01 Z-1.5 (Feed to Z-1.5) ;
(BEGIN COMPLETION BLOCKS) ;
G00 X1.5 M09 (Rapid Retract, Coolant off) ;
G53 X0 (X home) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%
```

This 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, +/-)

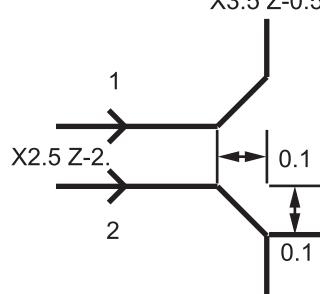
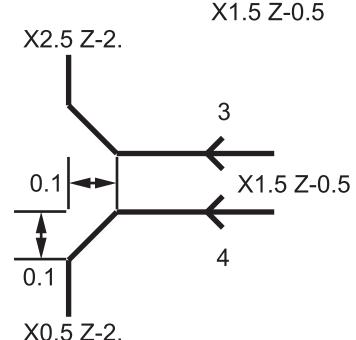
K = chamfering, X to Z (Z axis direction, +/-)

R = corner rounding (X or Z axis direction, +/-, Radius value)

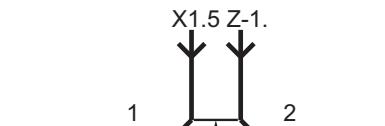
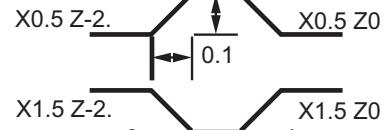
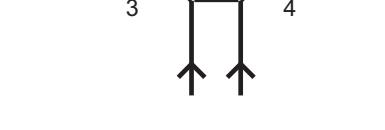
Notes:

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(current position + i) = Ui
Z(current position + k) = Wk
X(current position + r) = Ur
Z(current position + r) = Wr
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).

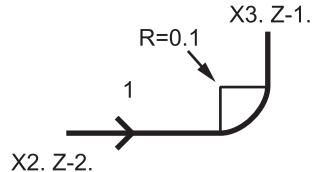
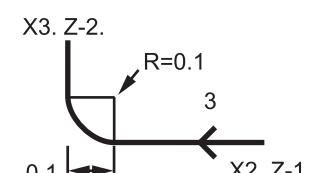
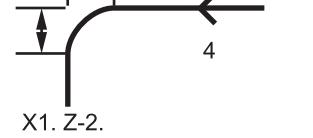
F7.2: Chamfering Code Z to X: [A] Chamfering, [B] Code/Example, [C] Movement.

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;	
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;	
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;	
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;	

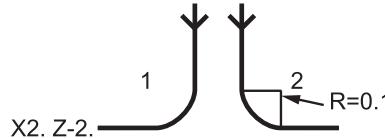
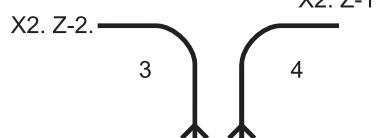
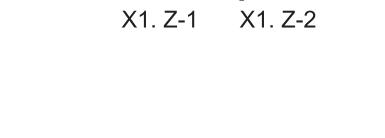
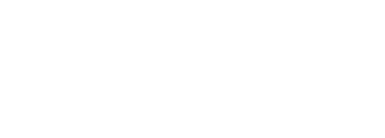
F7.3: Chamfering Code X to Z: [A] Chamfering, [B] Code/Example, [C] Movement.

A	B	C	
1. X- to Z-	X1.5 Z-1.; G01 X0.5 K-0.1; Z-2.;	X1.5 Z-1.; G01 X0.7; X0.5 Z-1.1; Z-2.	
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.;	

F7.4: Corner Rounding Code Z to X: [A] Corner rounding, [B] Code/Example, [C] Movement.

A	B	C	
1. Z+ to X+	X2. Z-2.; G01 Z-1 R.1; X3.;	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.;	

F7.5: Corner Rounding Code X to Z: [A] Corner rounding, [B] Code/Example, [C] Movement.

A	B	C	
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.;	
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.;	
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.;	
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.;	

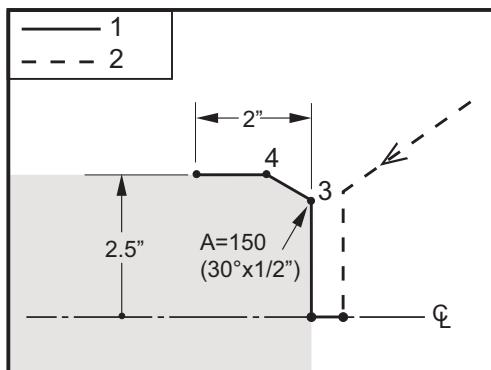
Rules:

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.

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.

F7.6: G01 Chamfering with A: [1] Feed, [2] Rapid, [3] Start Point, [4] Finish Point.



```
%  
o60012 (G01 CHAMFERING WITH 'A') ;  
(G54 X0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is an OD cutting tool) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G50 S1000 (Limit spindle to 1000 RPM) ;  
G97 S500 M03 (CSS off, Spindle on CW) ;  
G00 G54 X4. Z0.1 (Rapid to clear position) ;  
M08 (Coolant on) ;  
X0 (Rapid to center of diameter) ;  
(BEGIN CUTTING BLOCKS) ;
```

```
G01 Z0 F0.01 (Feed towards face) ;
G01 X4. (position 3) ;
X5. A150. (position 4) ;
Z-2. (Feed to back of part) ;
(BEGIN COMPLETION BLOCKS) ;
G00 X6. M09 (Rapid Retract, Coolant off) ;
G53 X0 (X home) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%
```

**NOTE:**

A -30 = A150; A -45 = A135

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

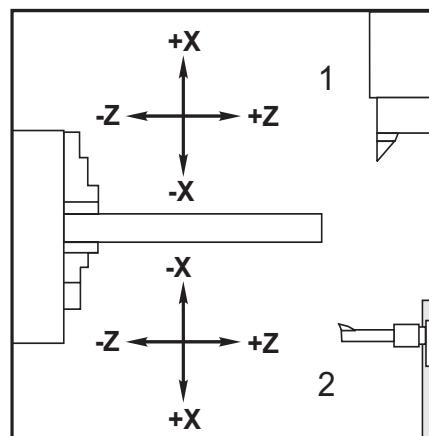
***Z** - Z-axis absolute motion command

* indicates optional

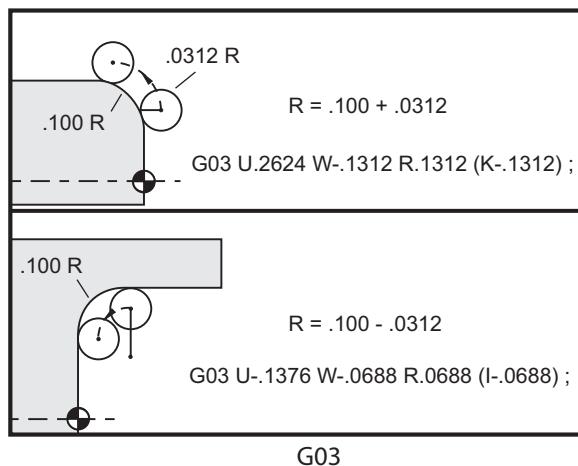
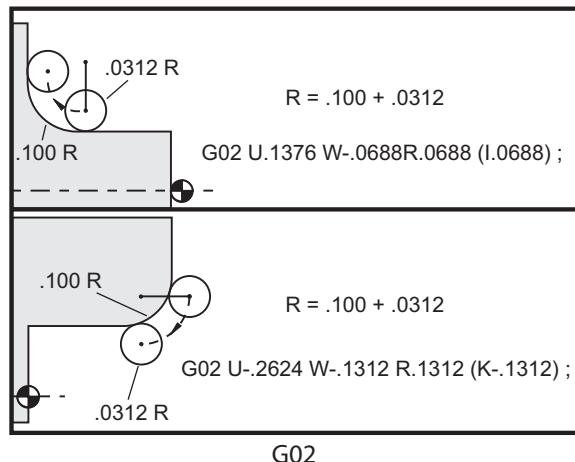
These G codes are used to specify a circular motion (CW or CCW) of the linear axes (Circular motion is possible in the X and Z axes as selected by G18). The X and Z values are used to specify the end point of the motion and can use either absolute (X and Z) or incremental motion (U and W). If either 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.

F7.7: G02 Axis Definitions: [1] Turret Lathes, [2] Table Lathes.



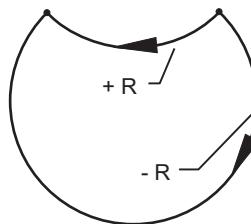
F7.8: G02 and G03 Programs



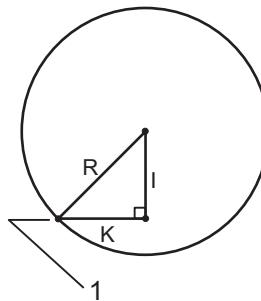
R is used to specify the radius of the arc. With a positive R , the control will generate a path of 180 degrees or less; to generate a radius of over 180 degrees, specify a negative R . X or Z is required to specify an endpoint if different from the starting point.

The following lines cut an arc of less than 180 degrees:

```
G01 X3.0 Z4.0 ;
G02 Z-3.0 R5.0 ;
```

F7.9: 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.

F7.10: G02 Defined X and Z: [1] Start.

7

G04 Dwell (Group 00)

P - The dwell time in seconds or milliseconds

G04 specifies a delay or dwell in the program. The block with G04 delay for the time specified by the P address code. For example:

```
G04 P10.0. ;
```

Delays the program for 10 seconds.

**NOTE:**

G04 P10. is a dwell of 10 seconds; G04 P10 is a dwell of 10 milliseconds. Make sure you use decimal points correctly so that you specify the correct dwell time.

G09 Exact Stop (Group 00)

G09 code is used to specify a controlled axes stop. It affects only the block in which it is commanded. It is non-modal and does not affect the blocks that come after the block where it is commanded. Machine moves decelerate to the programmed point before the control processes the next command.

G10 Set Offsets (Group 00)

G10 lets you set offsets within the program. G10 replaces manual offset entry (i.e. Tool length and diameter, and work coordinate offsets).

L - Selects offset category.

- L2 Work coordinate origin for 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, so that the secondary spindle reacts to commands normally used for the main spindle. For example, M03, M04, M05 and M19 affect the secondary spindle, and M143, M144, M145, and M119 (secondary spindle commands) cause an alarm.



NOTE:

G50 limits the secondary spindle speed, and G96 sets the secondary spindle surface feed value. These G-codes adjust the secondary spindle speed when there is motion in the X Axis. G01 Feed Per Rev feeds based on the secondary spindle.

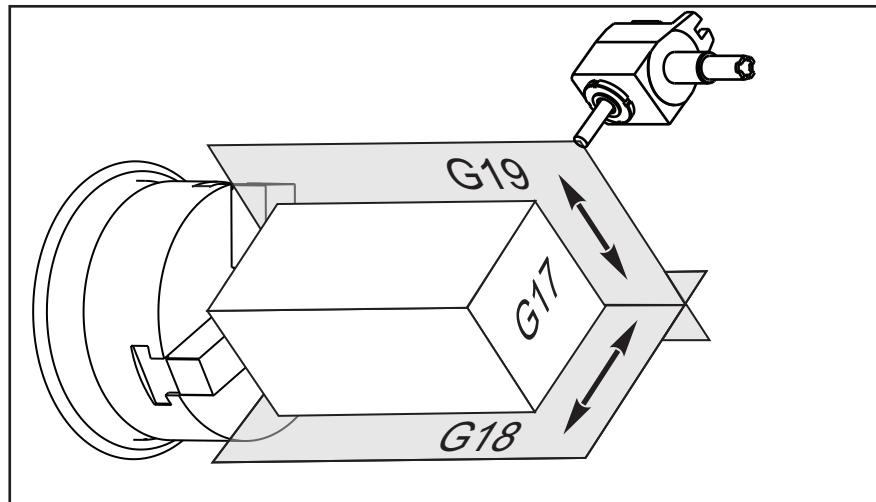
G14 automatically activates Z-Axis mirroring. If the Z Axis is already mirrored (Setting 47 or G101), the mirror function is canceled.

G14 is canceled by G15, M30, at the end of a program, or when you press [RESET].

G17 XY Plane / G18 XZ Plane / 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 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.

F7.11: G17, G18, and G19 Plane Selection



Program format with tool nose compensation:

```
G17 G01 X_ Y_ F_ ;
G40 G01 X_ Y_ I_ J_ F_ ;
```

G20 Select Inches / G21 Select Metric (Group 06)

Use G20 (inch) and G21 (mm) codes to make sure that the inch/metric selection is set correctly for the program. Use Setting 9 to select between inch and metric programming. G20 in a program causes an alarm if Setting 9 is not set to inch.

G28 Return to Machine Zero Point (Group 00)

The G28 code returns all axes (X, Y, Z, 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.

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)

G29 moves the axes to a specific position. The axes selected in this block are moved to the G29 reference point saved in G28, and then moved to the location specified in the G29 command.

G31 Feed Until Skip (Group 00)

(This G-code is optional and requires a probe.)

This G-code is used to record a probed location to a macro variable.



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)

F - Feedrate in inches (mm) per minute

Q - Thread Start Angle (optional). See example on the following page.

U/W - X/Z-axis incremental positioning command. (Incremental thread depth values are user specified)

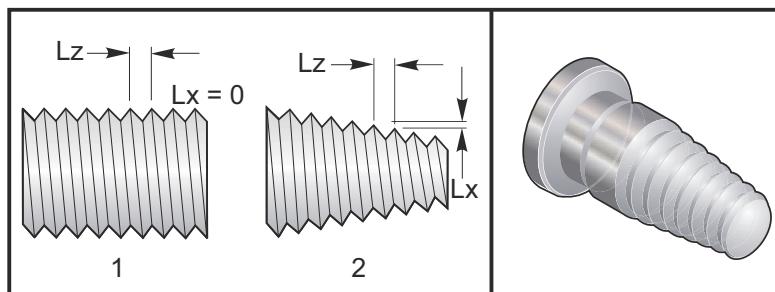
X/Z - X/Z-axis absolute positioning command. (Thread depth values are user specified)



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.

F7.12: 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).

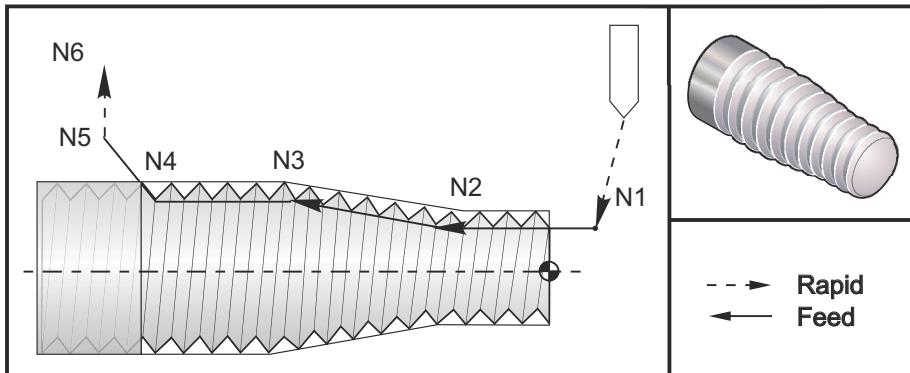


NOTE:

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 G94.

F7.13: Straight-to-Taper-to-Straight Thread Cutting Cycle**NOTE:**

Example is for reference only. Multiple passes are usually required to cut actual threads.

```
%  
o60321 (G32 THREAD CUTTING WITH TAPER) ;  
(G54 X0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is an OD thread tool) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G50 S1000 (Limit spindle to 1000 RPM) ;  
G97 S500 M03 (CSS off, Spindle on CW) ;  
N1 G00 G54 X0.25 Z0.1 (Rapid to 1st position) ;  
M08 (coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
N2 G32 Z-0.26 F0.065 (Straight thread, Lead = .065) ;  
N3 X0.455 Z-0.585 (Blend to tapered thread) ;  
N4 Z-0.9425 (Blend back to straight thread) ;  
N5 X0.655 Z-1.0425 (Pull off at 45 degrees) ;  
(BEGIN COMPLETION BLOCKS) ;  
N6 G00 X1.2 M09 (Rapid Retract, Coolant off) ;  
G53 X0 (X home) ;  
G53 Z0 M05 (Z home, spindle off) ;
```

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

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.

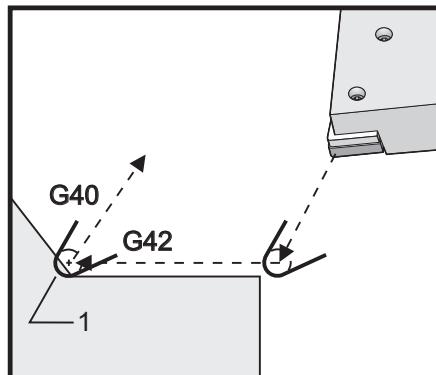
G40 Tool Nose Compensation Cancel (Group 07)

***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.

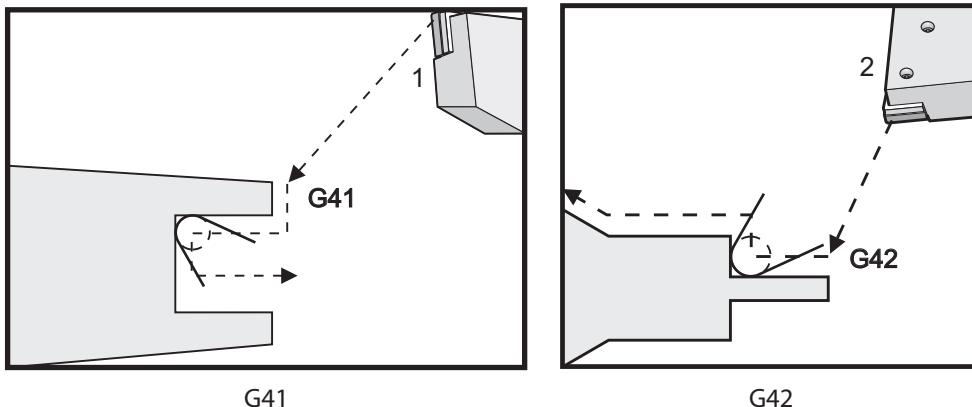
F7.14: G40 TNC Cancel: [1] Overcut.



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 Tnxxx 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.

F7.15: 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 - Limit spindle speed to specified value

T - Apply tool shift offset (YASNAC)

G50 performs several functions. It sets and shifts the global coordinate and it limits 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 becomes 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. For 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 is shifted by the amount and direction specified in **u** or **w**. The current effective coordinate displayed changes by this amount in the opposite direction. This method is often used to place the part zero outside of the work cell. For example:

```
G50 W-1.0 (Effective coordinates are shifted left 1.0) ;
```

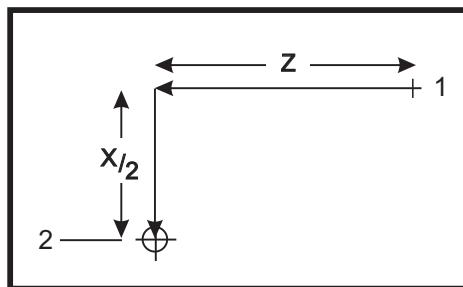
To set a YASNAC style work coordinate shift, specify G50 with a T value (Setting 33 must be set to **YASNAC**). The global coordinate is set to the X and Z values in the **Tool Shift offset** page. Values for the T address code are, Txxyy where xx is between 51 and 100 and yy is between 00 and 50. For 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, Txxyy 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.

```
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) ;
```

or,

```
G51 (Cancel Offsets) ;
G50 T5700 (Apply Tool Shift 57) ;
T707 M3 (Change to Tool 7 and apply Tool Wear 07) ;
```

- F7.16:** G50 YASNAC Tool Shift: [1] Machine (0,0), [2] Spindle centerline.



G50 Spindle Speed Limit

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 cancels the existing tool wear and work coordinate shifts and returns 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 317.

G61 Exact Stop Mode (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 G61 Cancel (Group 15)

G64 code cancels exact stop and selects the normal cutting mode.

G65 Macro Subprogram Call Option (Group 00)

G65 is described in the Macros programming section.

G70 Finishing Cycle (Group 00)

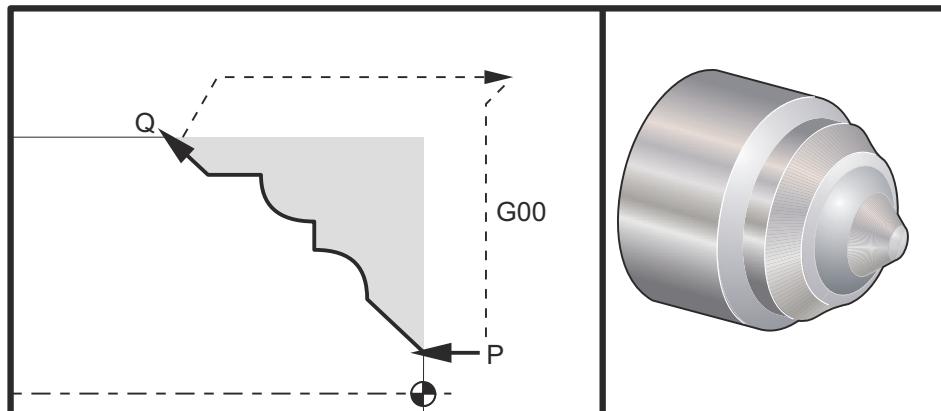
The G70 Finishing Cycle can be used to finish cut paths that are rough cut with stock removal cycles such as G71, G72 and G73.

P - Starting Block number of routine to execute

Q - Ending Block number of routine to execute

G18 Z-X plane must be active

F7.17: G70 Finishing Cycle: [P] Starting block, [Q] Ending Block.



```

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.

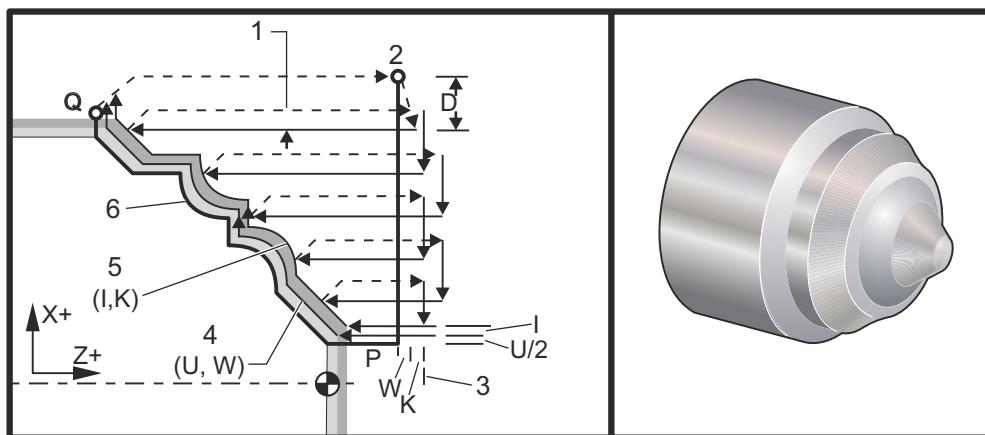
The G70 cycle is usually used after a G71, G72 or G73 has been performed using the blocks specified by P and Q. Any F, S, or T codes with the PQ block are effective. After execution of the Q block, a rapid (G00) is executed returning the machine to the start position that was saved before the starting of the G70. The program then returns to the block following the G70 call. A subprogram in the PQ sequence is acceptable providing that the subprogram 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.

F7.18: 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.

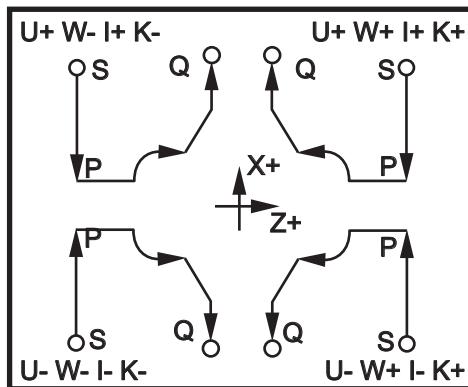
Two types of machining paths are addressed with a G71 command. The first type of path (Type 1) is when the X-Axis of the programmed path does not change direction. The second type of path (Type 2) allows the X-Axis to change direction. For both Type 1 and Type 2, the programmed path of the Z-axis cannot 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. When in YASNAC mode, include R1 on the G71 command block to select Type 2 roughing.

**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 276, note that the *P1* block (indicated by the comment in parentheses) contains the same Z-Axis position as the start position *G00* block above.

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.

F7.19: G71 Address Relationships**Type 1 Details**

When Type 1 is specified, 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, 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.

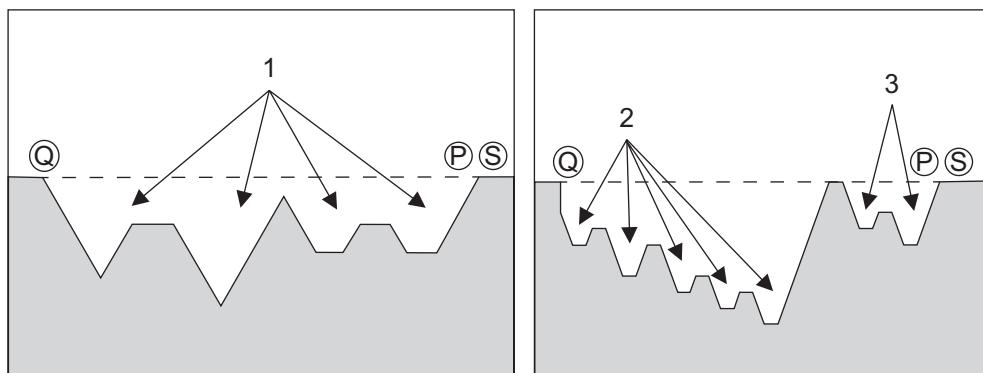
Type 2 roughing, when Setting 33 is set to **YASNAC**, must include $R1$ (with no decimal) on the $G71$ command block.

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 follows the path defined by PQ . The tool then retracts parallel to the X Axis by a distance defined in Setting 73 (Canned 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.

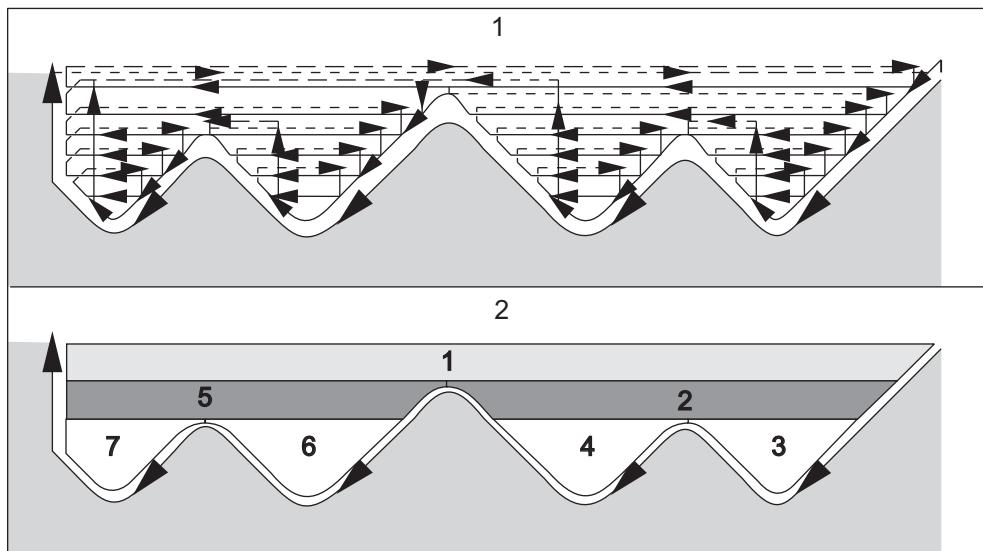
Troughs

- F7.20:** Single Nest with (4) Troughs [1] and Two Nests: one with (5) Troughs [2] and one with (2) Troughs [3].

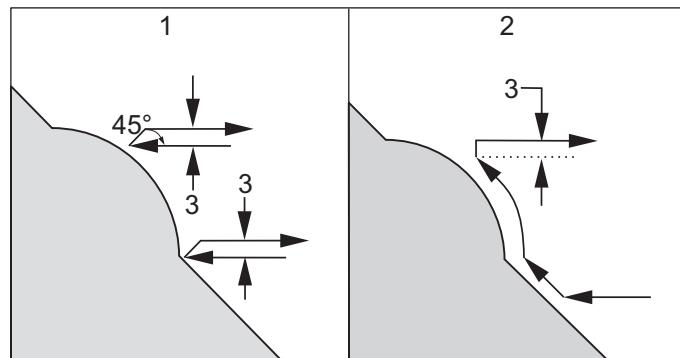


A trough can be defined as a change in direction which creates a concave surface in the material being cut. There can be no more than 10 troughs per cycle. If the part has more than 10 troughs, create another cycle. The following figures illustrate the sequence of roughing cuts (Type 1 and 2) for PQ paths with multiple troughs. All material above the troughs is roughed first, followed by the troughs in the direction of Z.

F7.21: Path for Type 2 Roughing: [1] Cutter path, [2] Region Sequence.



F7.22: 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.

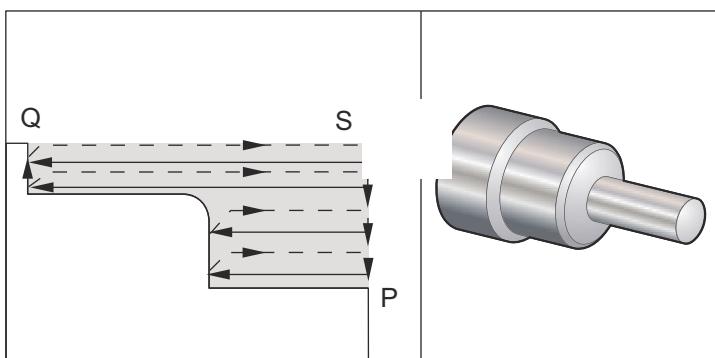


CAUTION:

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.

F7.23: G71 Basic G-code Example: [S] Start Point, [P] Starting block, [Q] Ending block.



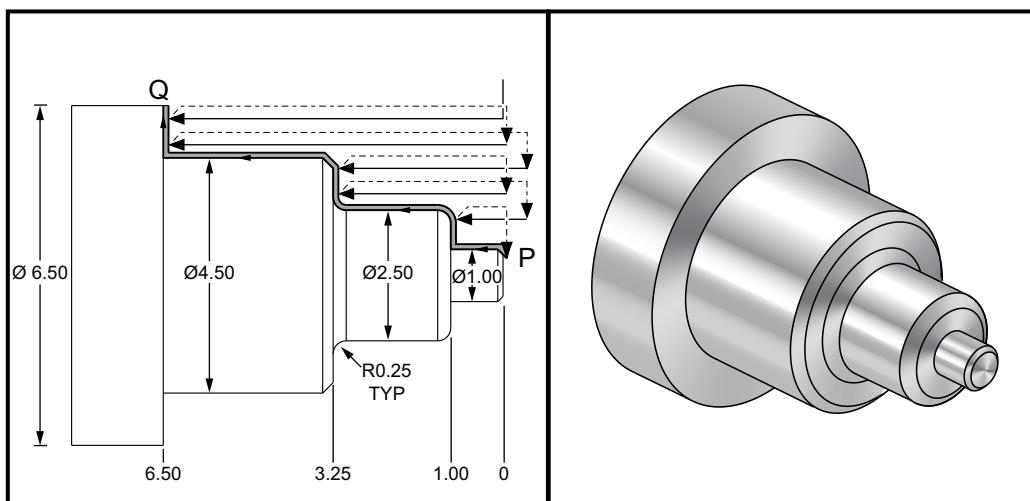
```
%  
O60711(G71 ROUGHING CYCLE) ;  
(G54 X0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is an OD cutting tool) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;
```

```

G50 S1000 (Limit spindle to 1000 RPM) ;
G97 S500 M03 (CSS off, Spindle on CW) ;
G00 G54 X6. Z0.1 (S - Rapid to 1st position) ;
M08 (Coolant on) ;
G96 S750 (CSS on) ;
(BEGIN CUTTING BLOCKS) ;
G71 P1 Q2 D0.15 U0.005 W0.005 F0.014 (Begin G71) ;
(Stock removal cycle leaving stock allowance) ;
N1 G00 X2. (P - Begin toolpath) ;
G01 Z-3. F0.006 (Linear feed to Z-3.) ;
X3.5 (Linear feed to X3.5) ;
G03 X4. Z-3.25 R0.25 (CCW arc) ;
G01 Z-6. (Linear feed to Z-6.) ;
N2 X6. (Q - End of toolpath) ;
G70 P1 Q2 (Finish pass) ;
(BEGIN COMPLETION BLOCKS) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%

```

F7.24: G71 Type 1 Stock Removal Example



```

%
O60712(G71 FANUC TYPE 1 EXAMPLE) ;
(G54 X0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is an OD cutting tool) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G50 S1000 (Limit spindle to 1000 RPM) ;

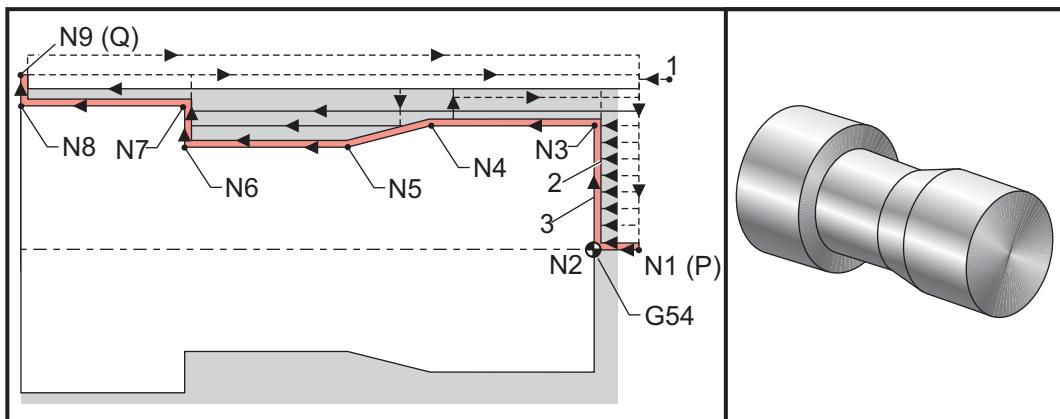
```

```

G97 S500 M03 (CSS off, spindle on CW) ;
G00 G54 X6.6 Z0.1 (Rapid to 1st position) ;
M08 (Coolant on) ;
G96 S200 (CSS on) ;
(BEGIN CUTTING BLOCKS) ;
G71 P1 Q2 D0.15 U0.01 W0.005 F0.012 (Begin G71) ;
(Stock removal cycle leaving stock allowance) ;
N1 G00 X0.6634 (P1 - Begin toolpath) ;
G01 X1. Z-0.1183 F0.004 (Linear feed chamfer) ;
Z-1. (Linear feed) ;
X1.9376 (Linear feed) ;
G03 X2.5 Z-1.2812 R0.2812 (CCW arc round) ;
G01 Z-3.0312 (Linear feed) ;
G02 X2.9376 Z-3.25 R0.2188 (CW arc round) ;
G01 X3.9634 (Linear feed) ;
X4.5 Z-3.5183 (Linear feed chamfer) ;
Z-6.5 (Linear feed) ;
N2 X6.0 (Q2 - End of toolpath) ;
G70 P1 Q2 (Finish pass) ;
(BEGIN COMPLETION BLOCKS) ;
G97 S500 (CSS off) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%

```

F7.25: 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.



```

%
O0125 (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 ;
%

```

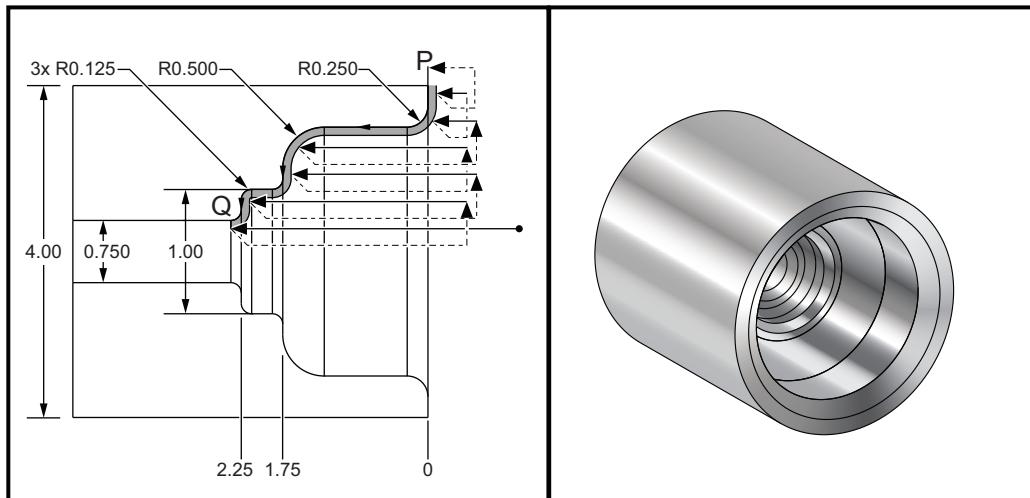
G71 I.D. Stock Removal Example



NOTE:

Make sure the start position of the tool is positioned below the diameter of the part you wish to start roughing out, before you define a G71 on an I.D. with this cycle.

F7.26: G71 I.D. Stock Removal Example



**NOTE:**

This example program and illustration assume that the workpiece starts with a 0.75" through-hole for the boring bar to enter.

```
%  
o60713 (G71 ID ROUGHING) ;  
(G54 X0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is an ID cutting tool) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G50 S1000 (Limit spindle to 1000 RPM) ;  
G97 S500 M03 (CSS off, spindle on CW) ;  
G00 G54 X0.7 Z0.1 (Rapid to clear position) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G71 P1 Q2 U-0.01 W0.002 D0.08 F0.01 (Begin G71) ;  
(Negative U indicates ID roughing) ;  
N1 G00 X4.1 Z0.1 (P1 - Begin toolpath) ;  
G01 Z0 ;  
X3. ,R.25 F.005 ;  
Z-1.75 ,R.5 ;  
X1.5 ,R.125 ;  
Z-2.25 ,R.125 ;  
X.75 ,R.125 ;  
Z-2.375 ;  
N2 X0.73 (Q2 - End of toolpath);  
G70 P1 Q2 ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 G53 X0 M09 (X home, coolant off) ;  
G53 Z0 M05 (Z home, spindle off) ;  
M30 (End program) ;  
%
```

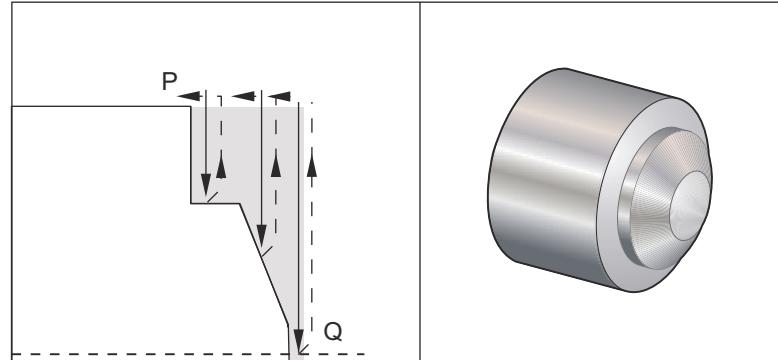
G72 End Face Stock Removal Cycle (Group 00)

- ***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.

F7.27: G72 Basic G Code Example: [P] Starting block, [1] Start position, [Q] Ending block.



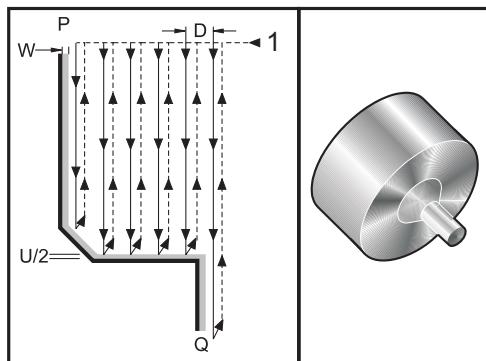
```
%  
O60721 (G72 END FACE STOCK REMOVAL EX 1) ;  
(G54 X0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is an end face cutting tool) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G50 S1000 (Limit spindle to 1000 RPM) ;  
G97 S500 M03 (CSS, spindle on CW) ;  
G00 G54 X6. Z0.1 (Rapid to clear position) ;  
M08 (Coolant on) ;  
G96 S200 (CSS on) ;  
(BEGIN CUTTING BLOCKS) ;  
G72 P1 Q2 D0.075 U0.01 W0.005 F0.012 (Begin G72) ;  
N1 G00 Z-0.65 (P1 - Begin toolpath);  
G01 X3. F0.006 (1st position);  
Z-0.3633 (Face Stock Removal);
```

```

X1.7544 Z0. (Face Stock Removal) ;
X-0.0624 ;
N2 G00 Z0.02 (Q2 - End toolpath) ;
G70 P1 Q2 (Finish Pass) ;
(BEGIN COMPLETION BLOCKS) ;
G97 S500 (CSS off) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%

```

F7.28: G72 Tool Path: [P] Starting block, [1] Start position, [Q] Ending block.



```

%
O60722(G72 END FACE STOCK REMOVAL EX 2) ;
(G54 X0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is an end face cutting tool) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G50 S1000 (Limit spindle to 1000 RPM) ;
G97 S500 M03 (CSS, spindle on CW) ;
G00 G54 X4.05 Z0.2 (Rapid to 1st position) ;
M08 (Coolant on) ;
G96 S200 (CSS on) ;
(BEGIN CUTTING BLOCKS) ;
G72 P1 Q2 U0.03 W0.03 D0.2 F0.01 (Begin G72) ;
N1 G00 Z-1.(P1 - Begin toolpath) ;
G01 X1.5 (Linear feed) ;
X1. Z-0.75 (Linear feed) ;
G01 Z0 (Linear feed) ;
N2 X0(Q2 - End of toolpath) ;
G70 P1 Q2 (Finishing cycle) ;
(BEGIN COMPLETION BLOCKS) ;
G97 S500 (CSS off) ;

```

```

G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%

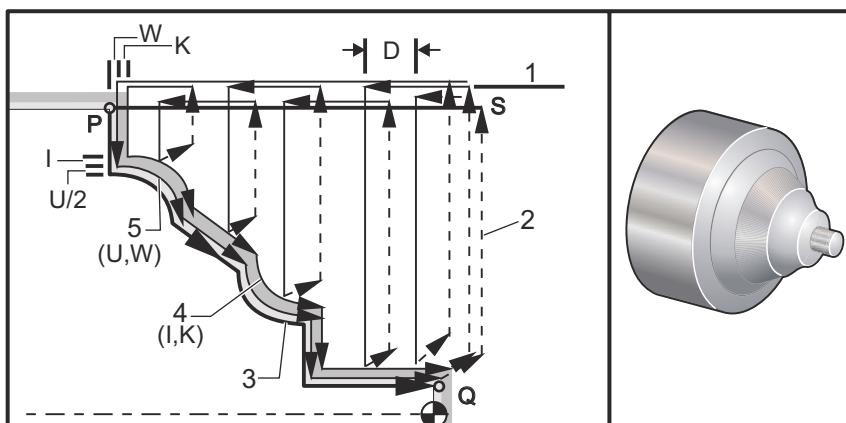
```

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).

F7.29: 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.

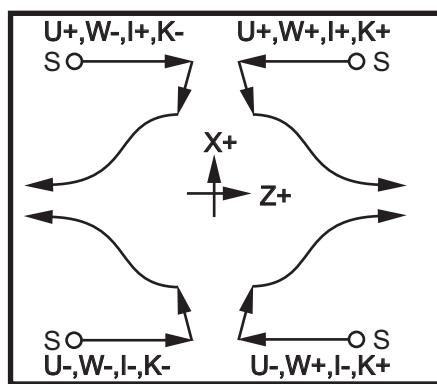


The G72 consists of a roughing phase and a finishing phase. The roughing and finishing phase are handled differently for Type 1 and Type 2. Generally the roughing phase consists of repeated passes along the X-axis at the specified feed rate. The finishing phase consists of a pass along the programmed 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.

F7.30: 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 P_Q. 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.

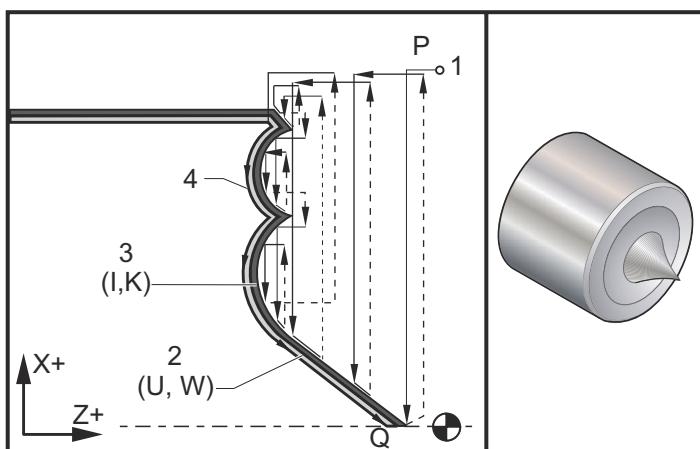


CAUTION:

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 the following Figure, 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.

F7.31: G72 End Face Removal: [P] Starting block, [1] Start position, [Q] Ending block, [2] Finishing allowance, [3] Roughing allowance, [4] Programmed path.



```
%  
O60723(G72 END FACE REMOVAL) ;  
(G54 X0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is an end face grooving tool) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G50 S1000 (Limit spindle to 1000 RPM) ;  
G97 S500 M03 (CSS off, spindle on CW) ;  
G00 G54 X2.1 Z0.1 (Rapid to clear position) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G72 P1 Q2 D0.06 I0.02 K0.01 U0.0 W0.01 F0.015 (Begin G72) ;  
N1 G01 Z-0.46 X2.1 F0.005 (P1 - Begin toolpath) ;  
X2. (1st position);  
G03 X1.9 Z-0.45 R0.2 (Tool path) ;  
G01 X1.75 Z-0.4 (Linear feed) ;  
G02 X1.65 Z-.4 R0.06 (Feed CW) ;  
G01 X1.5 Z-0.45 (Linear feed) ;  
G03 X1.3 Z-0.45 R0.12 (Feed CCW) ;  
G01 X1.17 Z-0.41 (Linear feed) ;  
G02 X1.03 Z-0.41 R0.1 (Feed CW) ;  
G01 X0.9 Z-0.45 (Linear feed) ;  
G03 X0.42 Z-0.45 R0.19 (Feed CCW) ;  
G03 X0.2 Z-0.3 R0.38 (Feed CCW) ;  
N2 G01 X0.01 Z0 (Q2 - End of tool path) ;  
G70 P1 Q2 (Finish Pass) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 G53 X0 M09 (X home, coolant off) ;
```

```
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%
```

G73 Irregular Path Stock Removal Cycle (Group 00)

D - Number of cutting passes, positive integer

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

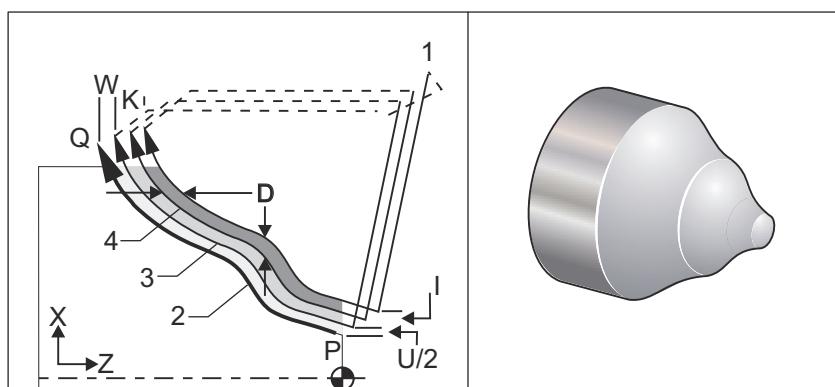
***U** - X-axis size and direction of G73 finish allowance, diameter

***W** - Z-axis size and direction of G73 finish allowance

* indicates optional

G18 Z-X plane must be active

F7.32: 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.

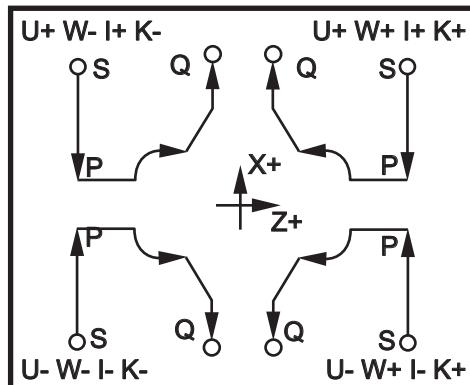
The offset of the first rough cut is determined by $(U/2 + I)$ for the X Axis, and by $(W + K)$ for the Z Axis. Each successive roughing pass moves incrementally closer to the final roughing finish pass by an amount of $(I/(D-1))$ in the X Axis, and by an amount of $(K/(D-1))$ in the Z Axis. The last 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.

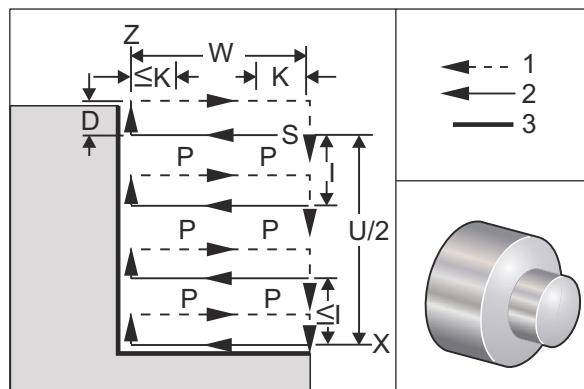
The value of D must be a positive integral number. If the D value includes a decimal, an alarm is generated. The four quadrants of the ZX plane can be machined if the following signs for U, I, W, and K are used.

F7.33: 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

F7.34: G74 End Face Grooving Cycle Peck Drilling: [1] Rapid, [2] Feed, [3] Programmed Path, [S] Start position, [P] Peck retraction (Setting 22).



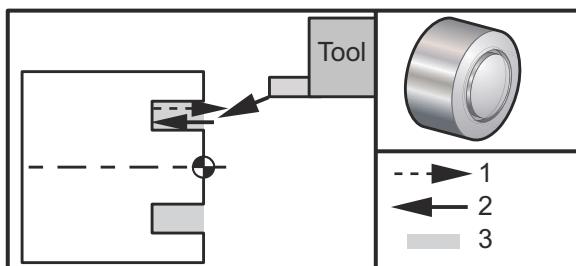
7

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**.

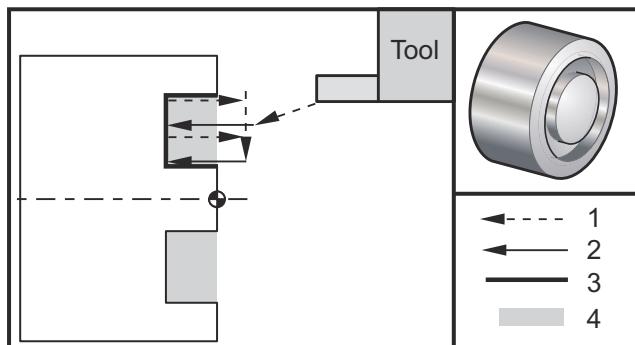
When **K** is added to a G74 block, pecking is performed at each interval specified by **K**, the peck is a rapid move opposite the direction of feed with a distance defined by Setting 22. The **D** code can be used for grooving and turning to provide material clearance when returning to starting plane **S**.

F7.35: G74 End Face Grooving Cycle: [1] Rapid, [2] Feed, [3] Groove.



```
%  
O60741 (G74 END FACE) ;  
(G54 X0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is an end face cutting tool) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G50 S1000 (Limit spindle to 1000 RPM) ;  
G97 S500 M03 (CSS off, Spindle on CW) ;  
G00 G54 X3. Z0.1 (Rapid to 1st position) ;  
M08 (Coolant on) ;  
G96 S200 (CSS on) ;  
(BEGIN CUTTING BLOCKS) ;  
G74 Z-0.5 K0.1 F0.01 (Begin G74) ;  
(BEGIN COMPLETION BLOCKS) ;  
G97 S500 (CSS off) ;  
G00 G53 X0 M09 (X home, coolant off) ;  
G53 Z0 M05 (Z home, spindle off) ;  
M30 (End program) ;  
%
```

F7.36: G74 End Face Grooving Cycle (Multiple Pass): [1] Rapid, [2] Feed, [3] Programmed path, [4] Groove.



```

%
O60742 (G74 END FACE MULTI PASS) ;
(G54 X0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is an end face cutting tool) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G50 S1000 (Limit spindle to 1000 RPM) ;
G97 S500 M03 (CSS off, spindle on CW) ;
G00 G54 X3. Z0.1 (Rapid to 1st position) ;
M08 (Coolant on) ;
G96 S200 (CSS on) ;
(BEGIN CUTTING BLOCKS) ;
G74 X1.75 Z-0.5 I0.2 K0.1 F0.01 (Begin G74) ;
(BEGIN COMPLETION BLOCKS) ;
G97 S500 (CSS off) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%

```

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

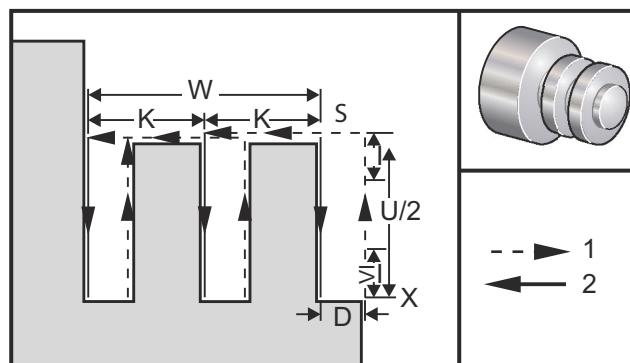
W - Z-axis incremental distance to furthest peck cycle

X - X-axis absolute location total pecking depth (diameter)

Z - Z-axis absolute location to furthest peck cycle

* indicates optional

F7.37: G75 O.D./I.D. Grooving Cycle: [1] Rapid, [2] Feed, [S] Start position.



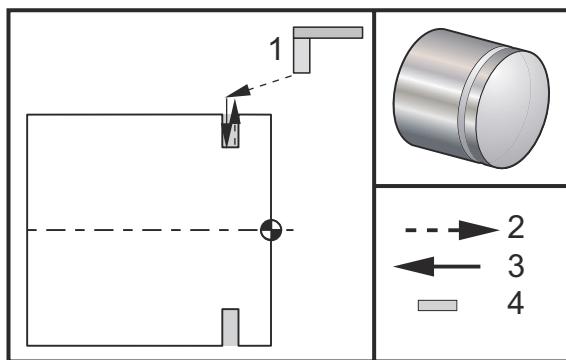
The G75 canned cycle can be used for grooving an outside diameter. When a Z, or W, code is added to a G75 block and Z is not the current position, then a minimum of two pecking cycles occur. One at the current location and another at the Z location. The K code is the incremental distance between Z axis 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.

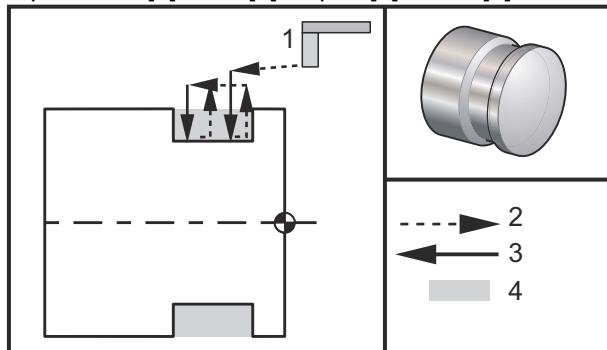
F7.38: G75 O.D. Single Pass



```
%  
O60751 (G75 OD GROOVE CYCLE) ;  
(G54 X0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is an OD groove tool) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G50 S1000 (Limit spindle to 1000 RPM) ;  
G97 S500 M03 (CSS off, spindle on CW) ;  
G00 G54 X4.1 Z0.1 (Rapid to 1st position) ;  
M08 (Coolant on) ;  
G96 S200 (CSS on) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 Z-0.75 F0.05 (Feed to Groove location) ;  
G75 X3.25 I0.1 F0.01 (Begin G75) ;  
(BEGIN COMPLETION BLOCKS) ;  
G97 S500 (CSS off) ;  
G00 G53 X0 M09 (X home, coolant off) ;  
G53 Z0 M05 (Z home, spindle off) ;  
M30 (End program) ;  
%
```

The following program is an example of a G75 program (Multiple Pass):

F7.39: G75 O.D. Multiple Pass: [1] Tool, [2] Rapid, [3] Feed, [4] Groove.



```
%  
O60752 (G75 OD GROOVE CYCLE 2) ;  
(G54 X0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is an OD groove tool) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G50 S1000 (Limit spindle to 1000 RPM) ;  
G97 S500 M03 (CSS off, spindle on CW) ;  
G00 G54 X4.1 Z0.1 (Rapid to 1st position) ;  
M08 (Coolant on) ;  
G96 S200 (CSS on) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 Z-0.75 F0.05 (Feed to Groove location) ;  
G75 X3.25 Z-1.75 I0.1 K0.2 F0.01 (Begin G75) ;  
(BEGIN COMPLETION BLOCKS) ;  
G97 S500 (CSS off) ;  
G00 G53 X0 M09 (X home, coolant off) ;  
G53 Z0 M05 (Z home, spindle off) ;  
M30 (End program) ;  
%
```

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

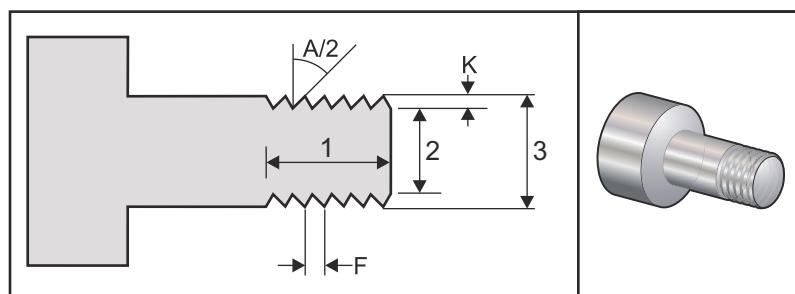
***W** - Z-axis incremental distance, start to maximum thread length

***X** - X-axis absolute location, maximum thread Depth Diameter

***Z** - Z-axis absolute location, maximum thread length

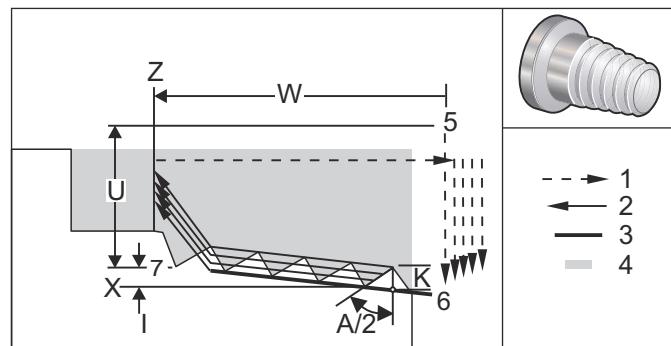
* indicates optional

F7.40: 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.

F7.41: 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.

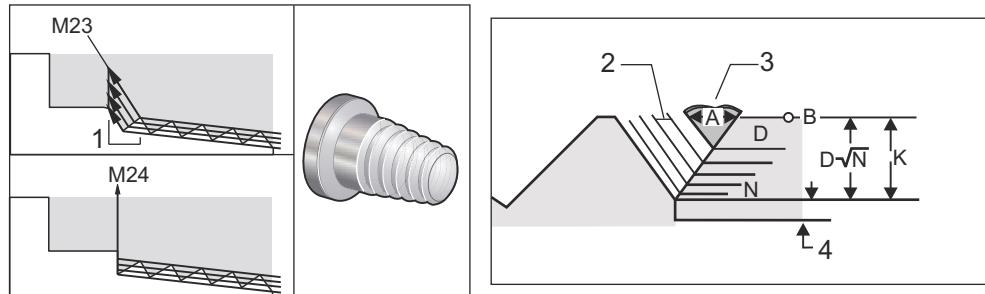
The tool nose angle for the thread is specified in A . The value can range from 0 to 120 degrees. If A is not used, 0 degrees is assumed. To reduce chatter while threading use $A59$ when cutting a 60 degree included thread.

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.

F7.42:

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.

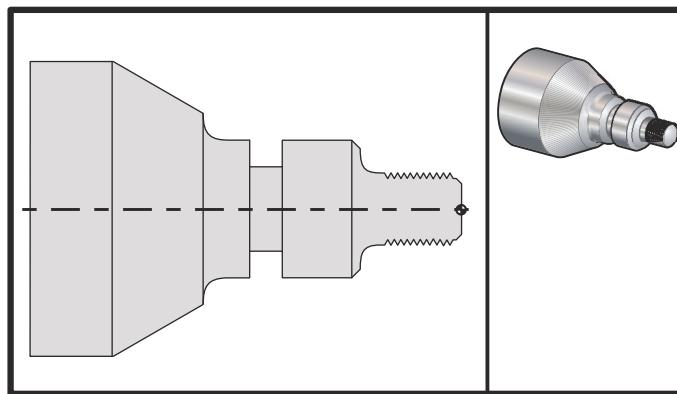
Four options for G76 Multiple Thread Cutting are available:

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

F7.43: G76 Thread Cutting Cycle, Multiple Pass



```
%  
o60761 (G76 THREAD CUTTING MULTIPLE PASSES) ;  
(G54 X0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is an OD thread tool) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G50 S1000 (Limit spindle to 1000 RPM) ;  
G97 S500 M03 (CSS off, Spindle on CW) ;  
G00 G54 X1.2 Z0.3 (Rapid to 1st position) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;
```

```
G76 X0.913 Z-0.85 K0.042 D0.0115 F0.0714 (Begin G76) ;
(BEGIN COMPLETION BLOCKS) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%
```

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).

```
% 
o60762 (G76 MULTI START THREAD CYCLES) ;
(G54 X0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is an OD thread tool) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G50 S1000 (Limit spindle to 1000 RPM) ;
G97 S400 M03 (CSS off, spindle on CW) ;
G00 G54 X1.1 Z0.5 (Rapid to clear position) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G76 X.913 Z-.850 K.042 D.0115 F.2142 (1st cycle) ;
G00 X1.100 Z.5714 (Z0.5 + Z0.0714) ;
```

```
G76 X.913 Z-.850 K.042 D.0115 F.2142 (2nd cycle) ;
G00 X1.100 Z.6428 (Z0.5714 + Z0.0714) ;
G76 X.913 Z-.850 K.042 D.0115 F.2142 (3rd Cycle) ;
(BEGIN COMPLETION BLOCKS) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%
```

G80 Canned Cycle Cancel (Group 09)

G80 cancels all active canned cycles.



NOTE:

G00 or G01 also cancel canned cycles.

G81 Drill Canned Cycle (Group 09)

***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

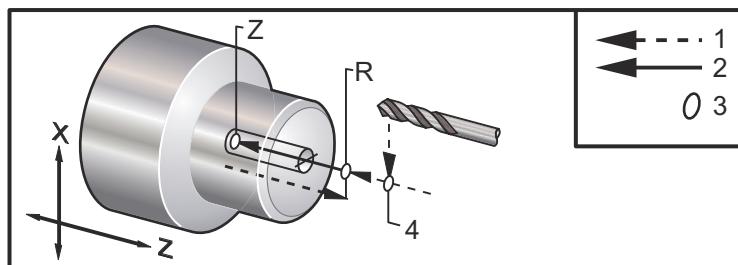
***Z** - Position of bottom of hole

* indicated optional

Also see G241 for radial drilling and G195/G196 for radial tapping with live tooling.

F7.44:

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

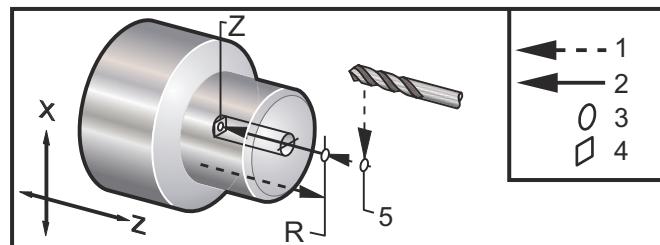
***Z** - Position of bottom of hole

* indicates optional

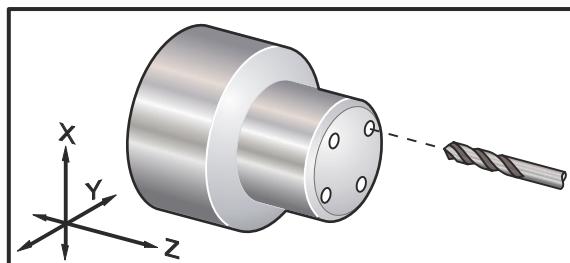
This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X will cause this canned cycle to be executed.

Also, see G242 for radial live tool spot drilling.

F7.45: 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.



F7.46: G82 Y-Axis Drill



```
%  
o60821 (G82 LIVE SPOT DRILL CYCLE) ;  
(G54 X0 Y0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is a spot drill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;
```

```
G98 (Feed per min) ;
M154 (Engage C Axis) ;
G00 G54 X1.5 C0. Z1. (Rapid to 1st position) ;
P1500 M133 (Live tool CW at 1500 RPM) ;
M08 (coolant on) ;
(BEGIN CUTTING CYCLE) ;
G82 C45. Z-0.25 F10. P80 (Begin G82) ;
C135. (2nd position) ;
C225. (3rd position) ;
C315. (4th position) ;
(BEGIN COMPLETION BLOCKS) ;
M155 (C axis disengage) ;
M135 (Live tool off) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 (Z home) ;
M30 (End program) ;
%
```

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:

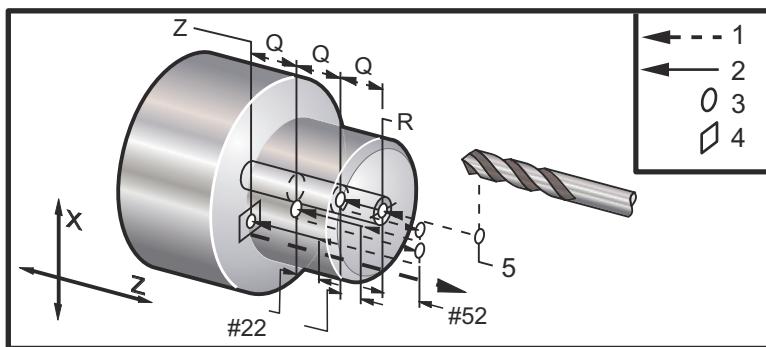
$$2 \times 60000 / 1500 = 80$$

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

F7.47: 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.

```
%  
o60831 (G83 NORMAL PECK DRILLING) ;  
(G54 X0 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) ;  
G50 S1000 (Limit spindle to 1000 RPM) ;  
G97 S500 M03 (CSS off, spindle on CW) ;  
G00 G54 X0 Z0.25 (Rapid to 1st position) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G83 Z-1.5 F0.005 Q0.25 R0.1 (Begin G83)  
(BEGIN COMPLETION BLOCKS)  
G00 G53 X0 M09 (X home, coolant off) ;  
G53 Z0 M05 (Z home, spindle off) ;  
M30 ;
```

```
%  
%  
(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.
- When G84 tapping on a lathe, it is simplest to use G99 Feed Per Revolution.
- The Lead is the distance traveled along a screw's axis, with each full revolution.
- The feedrate, when using G99, is equal to the Lead of the tap.
- 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.

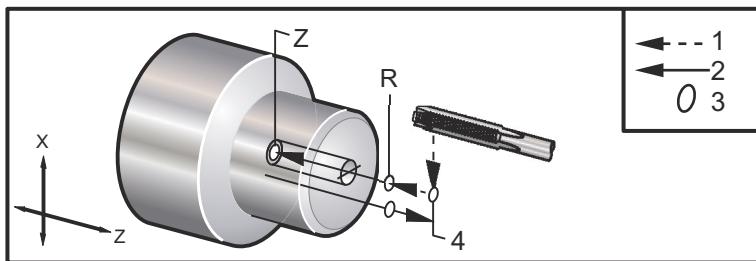
- In Inch Mode (G99, with Setting 9 = INCH), the feedrate is the Inch equivalent of the lead, in inches.
- The lead (and G99 feedrate) of an M10 x 1.0mm tap is 1.0mm, or .03937" (1.0/25.4=.03937).

Examples:

1. The lead of a 5/16-18 tap is 1.411 mm ($1/18 \times 25.4 = 1.411$), or .0556" ($1/18 = .0556$)
2. 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 page 260 for more information.
3. For Axial Live-Tool tapping, use a G95 or G186 command.
4. For Radial Live-Tool tapping, use a G195 or G196 command.
5. For Reverse Tapping (left-hand thread) on the Main or Secondary Spindle, refer to page 319.

More programming examples, in both Inch and Metric, are shown below:

F7.48: 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.



```
%  
o60841 (IMPERIAL TAP, SETTING 9 = MM) ;  
(G54 X0 is at the center of rotation) ;  
(Z0 is on the face of the part)  
(T1 is a 1/4-20 Tap) ;  
G21 (ALARM if setting 9 is not MM) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G40 G80 G99 (Safe startup) ;  
G00 G54 X0 Z12.7 (Rapid to 1st position) ;  
M08 (Coolant on) ;  
S800 (RPM OF TAP CYCLE) ;  
(BEGIN CUTTING BLOCK) ;  
G84 Z-12.7 R12.7 F1.27 (1/20*25.4 = 1.27) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 G53 X0 M09 (X home, coolant off) ;  
G53 Z0 M05 (Z home, spindle off) ;  
M30 (End program) ;
```

```
%  
%  
o60842 (METRIC TAP, SETTING 9 = MM) ;  
(G54 X0 is at the center of rotation) ;  
(Z0 is on the face of the part)  
(T1 is an M8 x 1.25 Tap) ;  
G21 (ALARM if setting 9 is not MM) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G40 G80 G99 (Safe startup) ;  
G00 G54 X0 Z12.7 (Rapid to 1st position) ;  
M08 (Coolant on) ;  
S800 (RPM OF TAP CYCLE) ;  
(BEGIN CUTTING BLOCK) ;  
G84 Z-12.7 R12.7 F1.25 (Lead = 1.25) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 G53 X0 M09 (X home, coolant off) ;  
G53 Z0 M05 (Z home, spindle off) ;  
M30 (End program) ;  
%  
  
%  
o60843 (IMPERIAL TAP, SETTING 9 = IN) ;  
(G54 X0 is at the center of rotation) ;  
(Z0 is on the face of the part)  
(T1 is a 1/4-20 Tap) ;  
G20 (ALARM if setting 9 is not INCH) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G00 G54 X0 Z0.5 (Rapid to 1st position) ;  
M08 (Coolant on) ;  
S800 (RPM OF TAP CYCLE) ;  
(BEGIN CUTTING BLOCK) ;  
G84 Z-0.5 R0.5 F0.05 (Begin G84) ;  
(1/20 = .05) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 G53 X0 M09 (X home, coolant off) ;  
G53 Z0 M05 (Z home, spindle off) ;  
M30 (End program) ;  
%  
  
%  
o60844 (METRIC TAP, SETTING 9 = IN) ;  
(G54 X0 is at the center of rotation) ;  
(Z0 is on the face of the part)
```

```

(T1 is an M8 x 1.25 Tap) ;
G20 (ALARM if setting 9 is not INCH) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G00 G54 X0 Z0.5 (Rapid to 1st position) ;
M08 (Coolant on) ;
S800 (RPM OF TAP CYCLE) ;
(BEGIN CUTTING BLOCK) ;
G84 Z-0.5 R0.5 F0.0492 (1.25/25.4 = .0492) ;
(BEGIN COMPLETION BLOCKS) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%

```

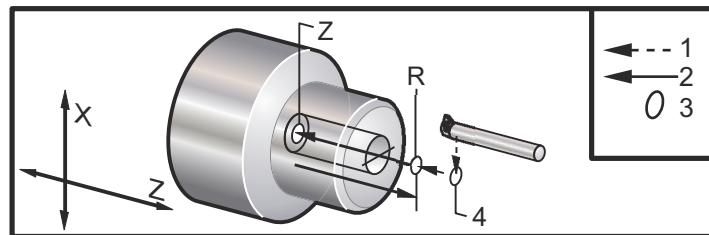
G85 Boring Canned Cycle (Group 09)



NOTE: This cycle feeds in and feeds out.

F - Feed Rate
 $*\text{L}$ - Number of repeats
 $*\text{R}$ - Position of the R plane
 $*\text{W}$ - Z-axis incremental distance
 $*\text{X}$ - X-axis motion command
 $*\text{Y}$ - Y-axis motion command
 Z - Position of bottom of hole
* indicates optional

F7.49: 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)

**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

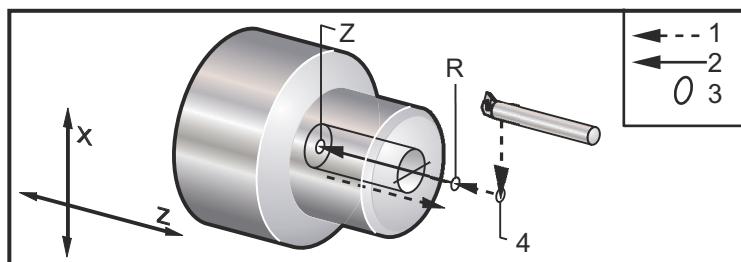
***Z** - Position of bottom of hole

* 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.

F7.50:

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

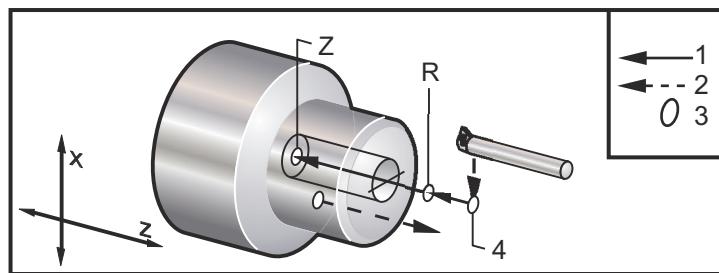
***Y** - Y-axis motion command

***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.

- F7.51:** 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

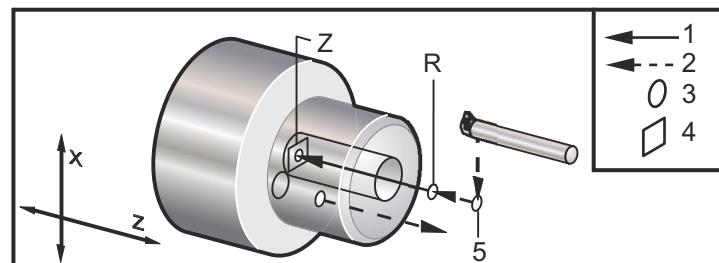
***Y** - Y-axis motion command

***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.

- F7.52:** 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

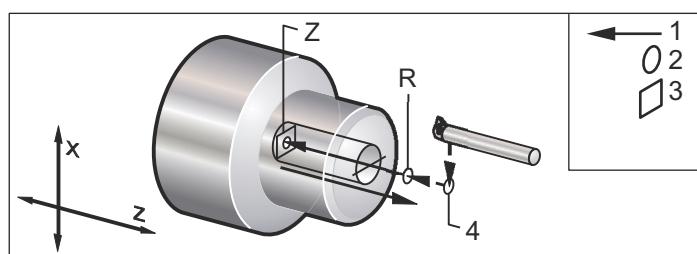
***X** - X-axis motion command

***Y** - Y-axis motion command

***Z** - Position of bottom of hole

* indicates optional

F7.53: 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

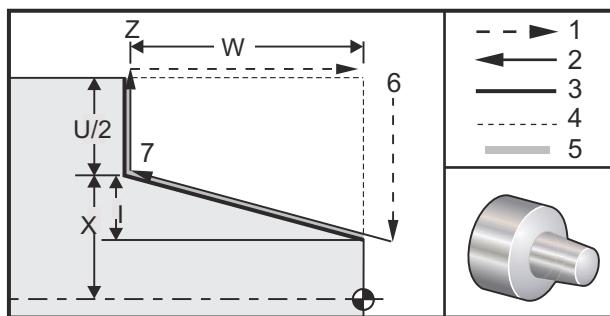
***W** - Z-axis incremental distance to target

X - X-axis absolute location of target

Z - Z-axis absolute location of target

*indicates optional

- F7.54:** 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.

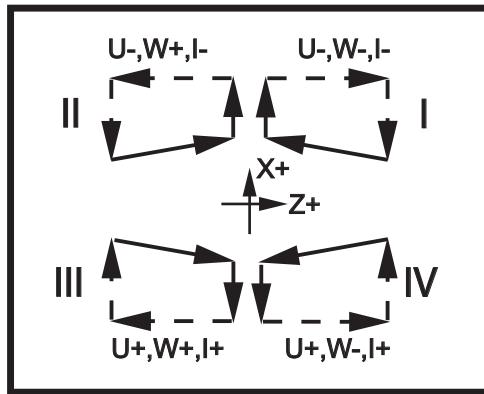


G90 is used for simple turning, however, multiple passes are possible by specifying the X locations of additional passes.

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.

- F7.55:** 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

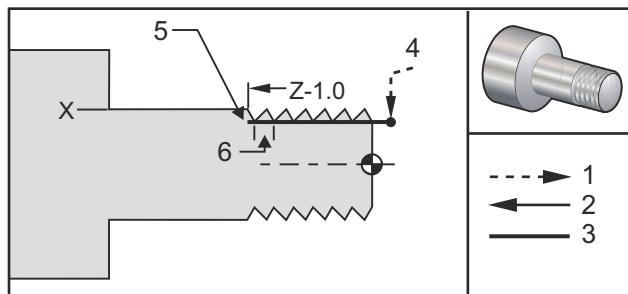
Z - Z-axis absolute location of target

* 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.

F7.56: 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).



```
%  
O60921 (G92 THREADING CYCLE) ;  
(G54 X0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is an OD thread tool) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G50 S1000 (Limit spindle to 1000 RPM) ;  
G97 S500 M03 (CSS off, Spindle on CW) ;  
G00 G54 X0 Z0.25 (Rapid to 1st position) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
X1.2 Z.2 (Rapid to clear position) ;  
G92 X.980 Z-1.0 F0.0833 (Begin Thread Cycle) ;  
X.965 (2nd pass) ;  
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) ;
(BEGIN COMPLETION BLOCKS) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%

```

Example Using Start Thread Angle Q

```

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.

G93 Inverse Time Feed Mode (Group 05)

F - Feed Rate (strokes per minute)

This G code specifies that all F (feedrate) values are interpreted as strokes per minute. In other words the time (in seconds) to complete the programmed motion using G93 is, 60 (seconds) divided by the F value.

G93 is generally used in 4 and 5-axis work when the program is generated using a CAM system. G93 is a way of translating the linear (inches/min) feedrate into a value that takes rotary motion into account. When G93 is used, the F value will tell you how many times per minute the stroke (tool move) can be repeated.

When G93 is used, feedrate (F) is mandatory for all interpolated motion blocks. Therefore each non-rapid motion block must have its own feedrate (F) specification.

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

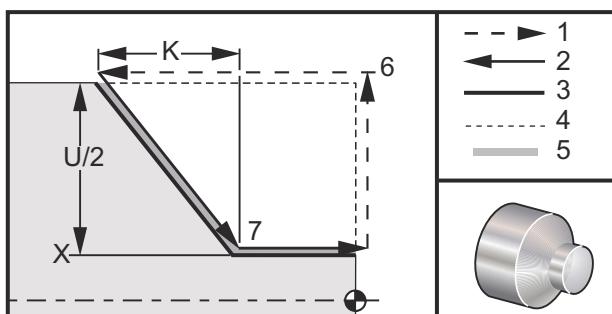
***W** - Z-axis incremental distance to target

X - X-axis absolute location of target

Z - Z-axis absolute location of target

*indicates optional

F7.57: G94 End Facing Cycle: [1] Rapid, [2] Feed, [3] Programmed path, [4] Cut allowance, [5] Finish allowance, [6] Start position, [7] Target.

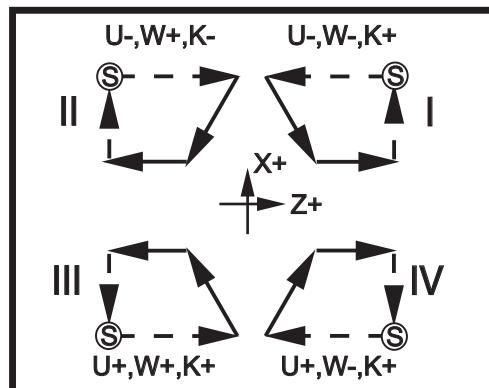


Straight end facing cuts can be made by specifying **X**, **Z** and **F**. By adding **K** a cone-shaped face is cut. The amount of coning is referenced from the target. That is **K** is added to the value of **X** at the target.

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.

F7.58: 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.

```
%  
o60951 (G95 LIVE TOOLING RIGID TAP) ;  
(G54 X0 Y0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is a 1/4-20 tap) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
M154 (Engage C Axis) ;  
G00 G54 X1.5 C0. Z0.5 (Rapid to 1st position) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING CYCLE) ;  
S500 (Select tap RPM) ;  
G95 C45. Z-0.5 R0.5 F0.05 (Tap to Z-0.5) ;  
C135. (next position) ;  
C225. (next position) ;  
C315. (last position) ;  
(BEGIN COMPLETION BLOCKS) ;  
M155 (Disengage C Axis) ;  
G00 G53 X0 M09 (X home, coolant off) ;  
G53 Z0 (Z home) ;  
M30 (End program) ;  
%
```

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 ($RPM=3.82 \times SFM/DIA$). 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 command tells the control NOT to 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 Disable / G101 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 and 47 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. No axis motion occurs.

**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 Look-Ahead (Group 00)

G103 specifies the maximum number of blocks the control looks ahead (Range 0-15), for example:

G103 [P..] ;

During machine motions, the control prepares future blocks (lines of code) ahead of time. This is commonly called “Block Look-ahead.” While the control executes the current block, it has already interpreted and prepared the next block for continuous motion.

A program command of G103 P0, or simply G103, disables block limiting. A program command of G103 Pn limits look-ahead to n blocks.

G103 is useful for debugging macro programs. The control interprets Macro expressions during look-ahead time. If you insert a G103 P1 into the program, the control interprets macro expressions (1) block ahead of the currently executing block.

It is best to add several empty lines after a G103 P1 is called. This ensures that no lines of code after the G103 P1 are interpreted until they are reached.

G103 affects cutter compensation and High Speed Machining.

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 cutoff subprogram

Q - Bar Feeder setup mode

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.

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 cutoff 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 J value, the distance specified by J.
 - 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 cutoff 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 cutoff subprogram, unclamps the workholding, then pushes the bar the distance specified by #3100 or J, then clamps the workholding.
- h) The program continues.

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.

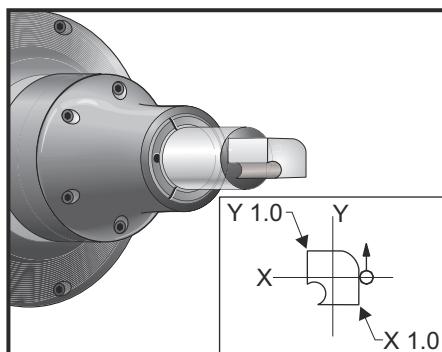


NOTE:

Mill-style Cutter Compensation becomes active when G112 is used. Cutter Compensation (G41, G42) must be canceled (G40) before exiting G112.

G112 Program Example

F7.59: G112 XY to XC Interpretation



```
%  
o61121 (G112 XY TO XC INTERPRETATION) ;  
(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) ;
```

```
G17 G112 (Call XY plane, XY to XC interpretation) ;
G98 (Feed per min) ;
M154 (Engage C Axis) ;
G00 G54 X0.875 C0. Z0.1 ;
(Rapid to 1st position) ;
P1500 M133 (Live tool CW at 1500 RPM) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G1 Z0. F15. (Feed towards face) ;
Y0.5 F5. (Linear feed) ;
G03 X.25 Y1.125 R0.625 (Feed CCW) ;
G01 X-0.75 (Linear feed) ;
G03 X-0.875 Y1. R0.125 (Feed CCW) ;
G01 Y-0.25 (Linear Feed) ;
G03 X-0.75 Y-0.375 R0.125 (Feed CCW) ;
G02 X-0.375 Y-0.75 R0.375 (Feed CW) ;
G01 Y-1. (Linear feed) ;
G03 X-0.25 Y-1.125 R0.125 (Feed CCW) ;
G01 X0.75 (Linear feed) ;
G03 X0.875 Y-1. R0.125 (Feed CCW) ;
G01 Y0. (Linear feed) ;
G00 Z0.1 (Rapid retract) ;
(BEGIN COMPLETION BLOCKS) ;
G113 (Cancel G112) ;
M155 (Disengage C Axis) ;
M135 (Live tool off) ;
G18 (Return to XZ plane) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 (Z home) ;
M30 (End program) ;
%
```

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-P99 (Group 12)

This feature provides 99 additional work offsets. G154 with a P value from 1 to 99 activates additional work offsets. For example G154 P10 selects work offset 10 from the list of additional work offsets.


NOTE:

G110 to G129 refer to the same work offsets as G154 P1 through P20; they can be selected by using either method.

When a G154 work offset is active, the heading in the upper right work offset will show the G154 P value.

G154 work offsets format

```
#14001-#14006 G154 P1 (also #7001-#7006 and G110)
#14021-#14026 G154 P2 (also #7021-#7026 and G111)
#14041-#14046 G154 P3 (also #7041-#7046 and G112)
#14061-#14066 G154 P4 (also #7061-#7066 and G113)
#14081-#14086 G154 P5 (also #7081-#7086 and G114)
#14101-#14106 G154 P6 (also #7101-#7106 and G115)
#14121-#14126 G154 P7 (also #7121-#7126 and G116)
#14141-#14146 G154 P8 (also #7141-#7146 and G117)
#14161-#14166 G154 P9 (also #7161-#7166 and G118)
#14181-#14186 G154 P10 (also #7181-#7186 and G119)
#14201-#14206 G154 P11 (also #7201-#7206 and G120)
#14221-#14221 G154 P12 (also #7221-#7226 and G121)
#14241-#14246 G154 P13 (also #7241-#7246 and G122)
#14261-#14266 G154 P14 (also #7261-#7266 and G123)
#14281-#14286 G154 P15 (also #7281-#7286 and G124)
#14301-#14306 G154 P16 (also #7301-#7306 and G125)
#14321-#14326 G154 P17 (also #7321-#7326 and G126)
#14341-#14346 G154 P18 (also #7341-#7346 and G127)
#14361-#14366 G154 P19 (also #7361-#7366 and G128)
#14381-#14386 G154 P20 (also #7381-#7386 and G129)
#14401-#14406 G154 P21
#14421-#14426 G154 P22
#14441-#14446 G154 P23
#14461-#14466 G154 P24
#14481-#14486 G154 P25
#14501-#14506 G154 P26
#14521-#14526 G154 P27
#14541-#14546 G154 P28
#14561-#14566 G154 P29
#14581-#14586 G154 P30
#14781-#14786 G154 P40
#14981-#14986 G154 P50
#15181-#15186 G154 P60
```

```
#15381-#15386 G154 P70  
#15581-#15586 G154 P80  
#15781-#15786 G154 P90  
#15881-#15886 G154 P95  
#15901-#15906 G154 P96  
#15921-#15926 G154 P97  
#15941-#15946 G154 P98  
#15961-#15966 G154 P99
```

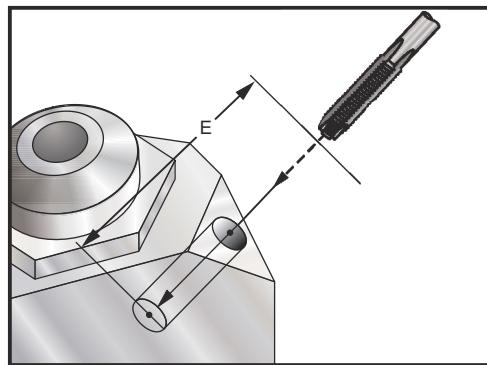
G155 5-Axis Reverse Tap Canned Cycle (Group 09)

G155 performs only floating taps. G174 is available for 5-axis reverse rigid tapping.

- E** - Specifies the distance from the start position to the bottom of the hole (must be a positive value)
- F** - Feedrate
- L** - Number of repeats
- A** - A-Axis tool starting position
- B** - B-Axis tool starting position
- X** - X-Axis tool starting position
- Y** - Y-Axis tool starting position
- Z** - Z-Axis tool starting position
- S** - Spindle Speed

A specific X, Y, Z, A, B position must be programmed before the canned cycle is commanded. This position is used as the Initial Start position. The control automatically starts the spindle counterclockwise before this canned cycle.

F7.60: G155 5-Axis Reverse Tap Canned Cycle



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. For 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

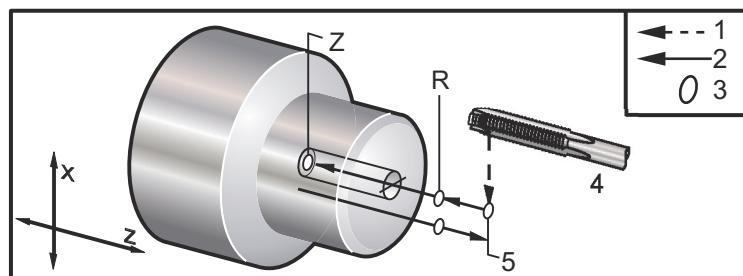
***Z** - Position of bottom of hole (optional)

* 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.

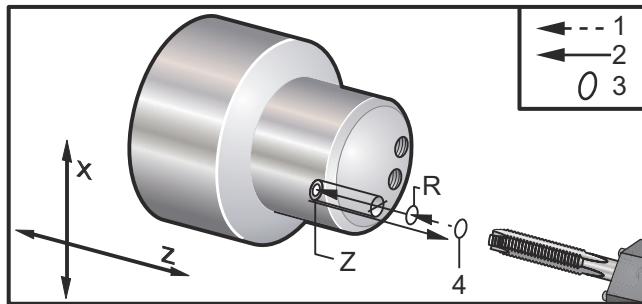
- F7.61:** G184 Reverse Tapping Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Left-hand tap, [5] Starting plane, [R] R plane, [Z] Position of the bottom of the hole.



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
 $*\mathbf{X}$ - Part Diameter X-axis motion command
 $*\mathbf{Y}$ - Y-axis motion command
Z - Position of bottom of hole
 * indicates optional

- F7.62:** 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

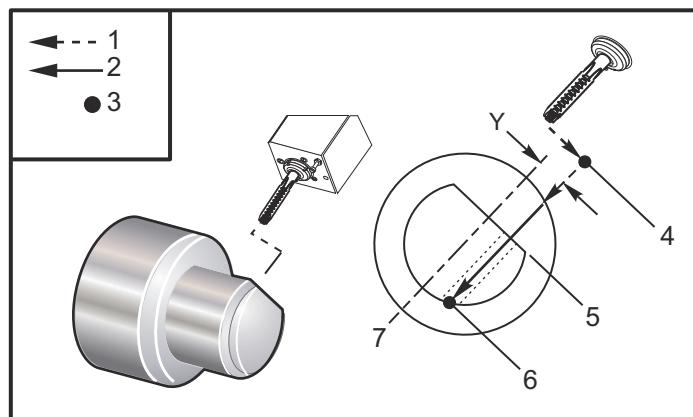
X - Position of the X Axis at the bottom of hole

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.

F7.63: 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.



```
%  
o61951 (G195 LIVE RADIAL TAPPING) ;  
(G54 X0 Y0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is a tap) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
M154 (Engage C Axis) ;  
G00 G54 X3.25 Z-0.75 C0. (Start Point) ;  
M08 (coolant on) ;  
(BEGIN CUTTING BLOCK) ;  
S500 (Select tap RPM) ;  
G195 X2. F0.05 (Taps to X2., bottom of hole) ;  
G00 C180. (Index C-Axis. New Start Point) ;
```

```
G195 X2. F0.05 (Taps to X2., bottom of hole) ;
G00 C270. Y-1. Z-1. ;
(Optional YZ-axis positioning, new start point) ;
G195 X2. F0.05 (Taps to X2., bottom of part) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.25 M09 (Rapid retract, coolant off) ;
M155 (Disengage C Axis) ;
G53 X0 Y0 (X & Y home) ;
G53 Z0 (Z home) ;
M30 (End program) ;
%
```

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. 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 **238**.

```
%  
o61991 (G199 SYNC SPINDLES) ;  
(G54 X0 Y0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G20 G40 G80 G99 (Safe startup) ;  
G00 G54 X2.1 Z0.5 ;  
G98 M08 (Feed per min, turn coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 Z-2.935 F60. (Linear feed) ;  
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 (Feed per rev) ;  
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 (Dwell for .5 sec) ;  
G00 B-29.25 (Feed secondary spindle onto part) ;  
M110 (secondary spindle chuck clamp) ;  
G04 P0.3 (Dwell for .3 sec) ;  
M08 (Turn coolant on) ;  
G97 S500 M03 (Turn spindle on at 500 RPM, CSS off) ;  
G96 S400 (CSS on, RPM is 400) ;  
G01 X1.35 F0.0045 (Linear feed) ;  
X-.05 (Linear feed) ;  
G00 X2.1 M09 (Rapid retract) ;  
G00 B-28. (Rapid secondary spindle to face of part) ;  
G198 (Synch spindle off) ;  
M05 (Turn off main spindle) ;  
G00 G53 B-13.0 (Secondary spindle to cut position);  
G00 G53 X-1. Y0 Z-11. (Rapid to 1st position) ;  
(*****second side of part*****)  
G55 G99 (G55 for secondary spindle work offset) ;  
G00 G53 B-13.0 ;  
G53 G00 X-1. Y0 Z-11. ;  
G14 ;  
T101 (Select tool and offset 1) ;  
G50 S2000 (limit spindle to 1000 RPM);  
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. ;
(BEGIN COMPLETION BLOCKS) ;
G00 G53 X0 M09 (X home) ;
G53 Z0 (Z home) ;
G28 H0. (Unwind C-Axis) ;
M30 (End program) ;
%
```

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 page 97.

G211 Manual Tool Setting / G212 Auto Tool Setting

These G-codes are used in probing applications for both automatic and manual probes (SS and ST lathes only).

G241 Radial Drill Canned Cycle (Group 09)

C - C-Axis absolute motion command

F - Feed Rate

R - Position of the R plane (diameter)

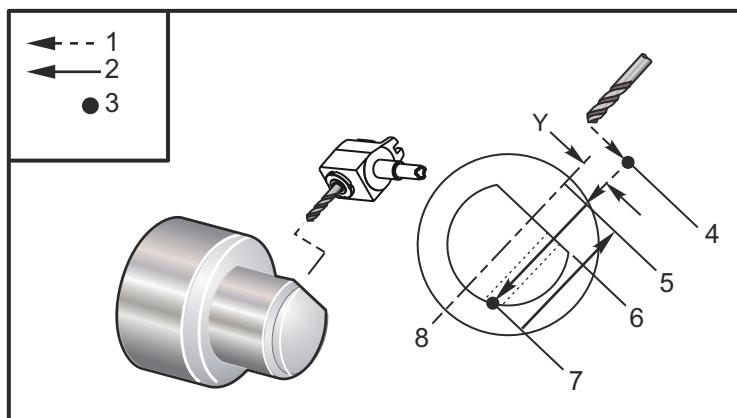
***X** - Position of bottom of hole (diameter)

***Y** - Y-axis absolute motion command

***Z** - Z-axis absolute motion command

* indicates optional

F7.64: 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.



```
%  
o62411 (G241 RADIAL DRILLING) ;  
(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) ;  
G98 (Feed per min) ;  
M154 (Engage C Axis) ;  
G00 G54 X5. Z-0.75 (Rapid to 1st position) ;  
P1500 M133 (Live tool CW at 1500 RPM) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G241 X2.1 Y0.125 Z-1.3 C35. R4. F20. (Begin G241) ;  
X1.85 Y-0.255 Z-0.865 C-75. (next position) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z1. M09 (Rapid retract, coolant off) ;  
M155 (Disengage C Axis) ;  
M135 (Live tool off) ;  
G53 X0 Y0 (X & Y Home) ;  
G53 Z0 (Z Home) ;  
M30 (End program) ;  
%
```

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)

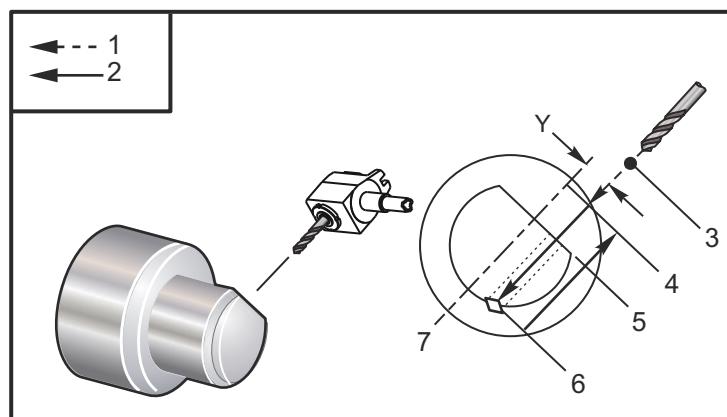
***Y** - Y-axis motion command

***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.

- F7.65:** 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.



```
%  
o62421 (G242 RADIAL SPOT DRILL) ;  
(G54 X0 Y0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is a spot drill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G98 (Feed per min) ;  
M154 (Engage C Axis) ;  
G00 G54 X5. Y0.125 Z-1.3 (Rapid to 1st position) ;  
P1500 M133 (Live tool CW at 1500 RPM) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G241 X2.1 Y0.125 Z-1.3 C35. R4. P0.5 F20. ;  
(Drill to X2.1) ;  
X1.85 Y-0.255 Z-0.865 C-75. P0.7 (next position) ;  
(BEGIN COMPLETION BLOCKS) ;
```

```

G00 Z1. M09 (Rapid retract, coolant off) ;
M155 (Disengage C Axis) ;
M135 (Live tool off) ;
G53 X0 Y0 (X & Y Home) ;
G53 Z0 (Z Home) ;
M30 (End program) ;
%

```

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)

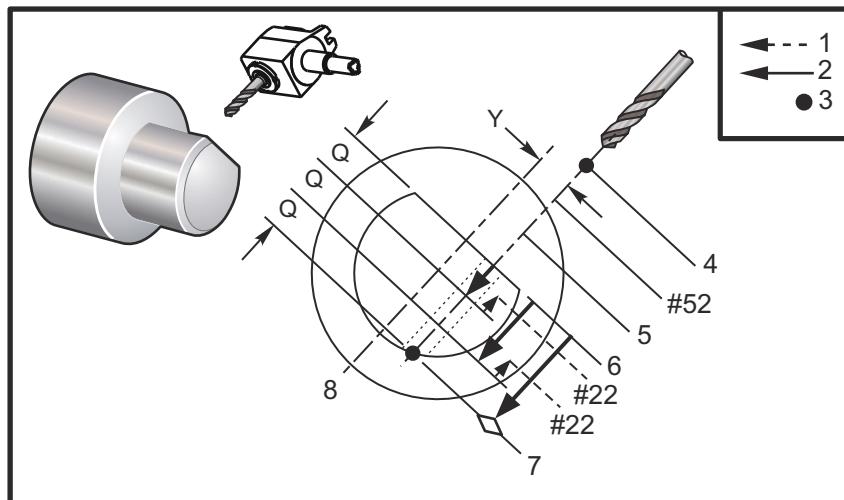
***X** - Position of bottom of hole (Diameter)

***Y** - Y-axis absolute motion command

***Z** - Z-axis absolute motion command

* indicates optional

F7.66: 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.

```
%  
o62431 (G243 RADIAL PECK DRILL CYCLE) ;  
(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) ;  
G98 (Feed per min) ;  
M154 (Engage C Axis) ;  
G00 G54 X5. Y0.125 Z-1.3 (Rapid to 1st position) ;  
P1500 M133 (Live tool CW at 1500 RPM) ;  
M08 (Coolant on) ;  
G243 X2.1 Y0.125 Z-1.3 C35. R4. Q0.25 F20. ;  
(Drill to X2.1) ;  
X1.85 Y-0.255 Z-0.865 C-75. Q0.25 (Next position);  
G00 Z1. (Rapid retract) ;  
M135 (Live tool off) ;  
G00 G53 X0 M09(X home, coolant off) ;  
G53 Z0 ;  
M00 ;  
(G243 - RADIAL WITH I,J,K PECK DRILLING) ;  
M154 (Engage C Axis) ;  
G00 G54 X5. Y0.125 Z-1.3 (Rapid to 1st position) ;  
P1500 M133 (Live tool CW - 1500 RPM) ;  
M08 (Coolant on) ;  
G243 X2.1 Y0.125 Z-1.3 I0.25 J0.05 K0.1 C35. R4. F5. ;  
(Drill to X2.1) ;  
X1.85 Y-0.255 Z-0.865 I0.25 J0.05 K0.1 C-75. ;  
(next position) ;  
(BEGIN COMPLETION BLOCKS) ;  
M155 (Disengage C Axis) ;  
M135 (Turn live tool off) ;  
G00 G53 X0 Y0 M09 (X & Y home, coolant off) ;  
G53 Z0 (Z home) ;  
M30 (End program) ;  
%
```

G245 Radial Boring Canned Cycle (Group 09)

C - C-Axis absolute motion command

F - Feed Rate

R - Position of the **R** plane (Diameter)

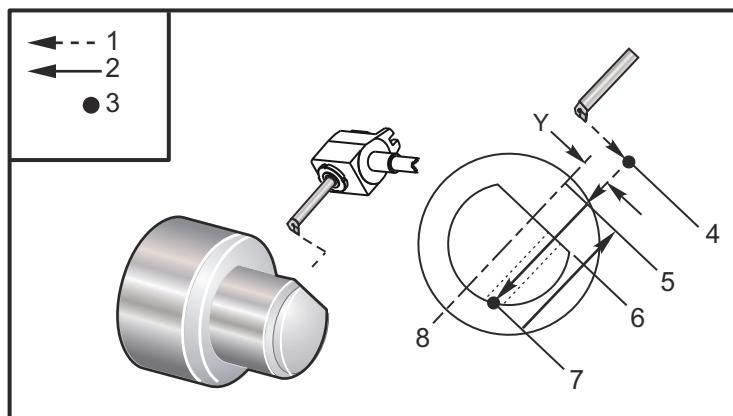
***X** - Position of bottom of hole (Diameter)

***Y** - Y-axis absolute motion command

***Z** - Z-axis absolute motion command

* indicates optional

F7.67: G245 Radial Boring Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Starting point, [5] R plane, [6] Part surface, [Z] Bottom of the hole, [8] Centerline.



```
%  
o62451 (G245 RADIAL BORING) ;  
(G54 X0 Y0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is a boring tool) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G98 (Feed per min) ;  
M154 (Engage C Axis) ;  
G00 G54 X5. Y0.125 Z-1.3 (Rapid to 1st position) ;  
P500 M133 (Live tool CW at 500 RPM) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G245 X2.1 Y0.125 Z-1.3 C35. R4. F20. ;  
(Bore to X2.1) ;  
X1.85 Y-0.255 Z-0.865 C-75. (next position) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z1. M09 (Rapid retract, coolant off) ;  
M155 (Disengage C Axis) ;  
M135 (live tool off) ;
```

```
G53 X0 Y0 (X & Y home) ;
G53 Z0 (Z home) ;
M30 (End program) ;
%
```

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.

```
%  
o62461 (G246 RADIAL BORE AND STOP) ;  
(G54 X0 Y0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is a boring tool) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G98 (Feed per min) ;  
M154 (Engage C Axis) ;  
G00 G54 X5. Y0.125 Z-1.3 (Rapid to 1st position) ;  
P500 M133 (Live tool CW at 500 RPM) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G246 X2.1 Y0.125 Z-1.3 C35. R4. F20. ;  
(Bore to X2.1) ;  
X1.85 Y-0.255 Z-0.865 C-75. (next position) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z1. M09 (Rapid retract, coolant off) ;  
M155 (Disengage C Axis) ;  
M135 (Live tool off) ;  
G53 X0 Y0 (X & Y Home) ;  
G53 Z0 (Z Home) ;  
M30 (End program) ;  
%
```

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.

```
%  
o62471 (G247 RADIAL BORE AND MANUAL RETRACT) ;  
(G54 X0 Y0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is a boring tool) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G98 (Feed per minute) ;  
M154 (Engage C Axis) ;  
G00 G54 X5. Y0.125 Z-1.3 (Rapid to 1st position) ;  
P500 M133 (Live tool CW at 500 RPM) ;  
M08 (coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G247 X2.1 Y0.125 Z-1.3 C35. R4. F20. ;  
(Bore to X2.1) ;  
X1.85 Y-0.255 Z-0.865 C-75. (next position) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z1. M09 (Rapid retract, Coolant off) ;  
M155 (Disengage C Axis) ;  
M135 (Live tool off) ;  
G53 X0 Y0 (X & Y Home) ;  
G53 Z0 (Z Home) ;  
M30 (End program) ;  
%
```

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.

```
%  
o62481 (G248 RADIAL BORE, DWELL, MANUAL RETRACT) ;  
(G54 X0 Y0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is a boring tool) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G98 (Feed per minute) ;  
M154 (Engage C Axis) ;  
G00 G54 X5. Y0.125 Z-1.3 (Rapid to 1st position) ;  
P500 M133 (Live tool CW at 500 RPM) ;  
M08 (coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G248 X2.1 Y0.125 Z-1.3 C35. R4. P1. F20. ;  
(Bore to X2.1) ;  
X1.85 Y-0.255 Z-0.865 C-75. (next position) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z1. M09 (Rapid retract, coolant off) ;  
M155 (Disengage C Axis) ;  
M135 (Live tool off) ;  
G53 X0 Y0 (X & Y Home) ;  
G53 Z0 (Z Home) ;  
M30 (End program) ;  
%
```

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

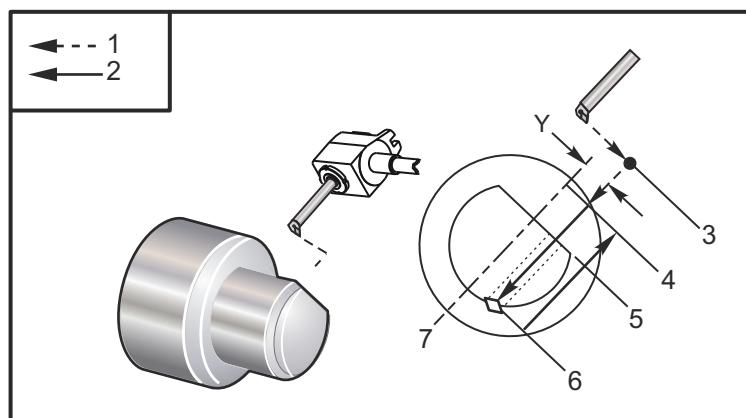
***X** - Position of bottom of hole

***Y** - Y-axis motion command

***Z** - Z-axis motion command

* indicates optional

F7.68: 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.

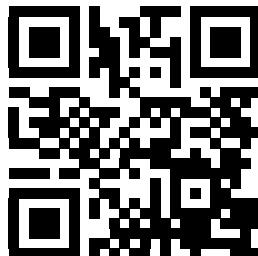


```
%  
o62491 (G249 RADIAL BORE AND DWELL) ;  
(G54 X0 Y0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(T1 is a boring tool) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
G98 (Feed per minute) ;  
M154 (Engage C Axis) ;  
G00 G54 X5. Y0.125 Z-1.3 (Rapid to 1st position) ;  
P500 M133 (Live tool CW at 500 RPM) ;  
M08 (coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G249 X2.1 Y0.125 Z-1.3 C35. R4. P1.35 F20. ;  
(Bore to X2.1) ;  
X1.85 Y-0.255 Z-0.865 C-75. P1.65 (next position) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z1. M09 (Rapid retract, Coolant off) ;  
M155 (Disengage C Axis) ;  
M135 (Live tool off) ;
```

```
G53 X0 Y0 (X & Y home) ;  
G53 Z0 (Z home) ;  
M30 (End program) ;  
%
```

7.2 More Information Online

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



Chapter 8: M-codes

8.1 Introduction

This chapter gives detailed descriptions of the M-codes that you use to program your machine.



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



NOTE: *The sample programs in this manual represent a very conservative programming style. The samples are intended to demonstrate safe and reliable programs, and they are not necessarily the fastest or most efficient way to operate a machine. The sample programs use G-codes that you might choose not to use in more efficient programs.*

8.1.1 List of M-codes

Code	Description	Page
M00	Stop Program	341
M01	Stop Program	341
M02	Program End	342
M03	Spindle On Fwd	342
M04	Spindle On Rev	342
M05	Spindle Stop	342
M08	Coolant On	342
M09	Coolant Off	342

Code	Description	Page
M10	Chuck Clamp	342
M11	Chuck Unclamp	342
M12	Auto Jet Air Blast On (Optional)	342
M13	Auto Jet Air Blast Off (Optional)	342
M14	Main Spindle Brake On (Optional C-Axis)	342
M15	Main Spindle Brake Off (Optional C-Axis)	342
M17	Turret Rotation Fwd	342
M18	Turret Rotation Rev	342
M19	Orient Spindle (Optional)	343
M21	Tailstock Advance (Optional)	344
M22	Tailstock Retract (Optional)	344
M23	Chamfer Out of Thread On	345
M24	Chamfer Out of Thread Off	345
M30	End of Program and Reset	345
M31	Chip Auger Forward (Optional)	345
M33	Chip Auger Stop (Optional)	345
M36	Parts Catcher On (Optional)	345
M37	Parts Catcher Off (Optional)	345
M38	Spindle Speed Variation On	345
M39	Spindle Speed Variation Off	345
M41	Low Gear (Optional)	346
M42	High Gear (Optional)	346
M43	Turret Unlock (Service Use Only)	346

Code	Description	Page
M44	Turret Lock (Service Use Only)	346
M51	User M Turn On (Optional)	346
M52	User M Turn On (Optional)	346
M53	User M Turn On (Optional)	346
M54	User M Turn On (Optional)	346
M55	User M Turn On (Optional)	346
M56	User M Turn On (Optional)	346
M57	User M Turn On (Optional)	346
M58	User M Turn On (Optional)	346
M59	Set Output Relay	346
M61	User M Turn Off (Optional)	347
M62	User M Turn Off (Optional)	347
M63	User M Turn Off (Optional)	347
M64	User M Turn Off (Optional)	347
M65	User M Turn Off (Optional)	347
M66	User M Turn Off (Optional)	347
M67	User M Turn Off (Optional)	347
M68	User M Turn Off (Optional)	347
M69	Clear Output Relay	347
M76	Display Disable	347
M77	Display Enable	347
M78	Alarm if Skip Signal Found	347
M79	Alarm if Skip Signal Not Found	347

Code	Description	Page
M85	Automatic Door Open (Optional)	347
M86	Automatic Door Close (Optional)	347
M88	High Pressure Coolant On (Optional)	348
M89	High Pressure Coolant Off (Optional)	348
M95	Sleep Mode	348
M96	Jump If No Signal	348
M97	Local Subprogram Call	349
M98	Subprogram Call	349
M99	Subprogram Return Or Loop	350
M104	Probe Arm Extend (Optional)	351
M105	Probe Arm Retract (Optional)	351
M109	Interactive User Input	351
M110	Secondary Spindle Chuck Clamp (Optional)	342
M111	Secondary Spindle Chuck Unclamp (Optional)	342
M112	Secondary Spindle Air Blast On (Optional)	353
M113	Secondary Spindle Air Blast Off (Optional)	353
M114	Secondary Spindle Brake On (Optional)	353
M115	Secondary Spindle Brake Off (Optional)	353
M119	Secondary Spindle Orient (Optional)	354
M121	User M-codes (Optional)	354
M122	User M-codes (Optional)	354
M123	User M-codes (Optional)	354
M124	User M-codes (Optional)	354

Code	Description	Page
M125	User M-codes (Optional)	354
M126	User M-codes (Optional)	354
M127	User M-codes (Optional)	354
M128	User M-codes (Optional)	354
M133	Live Tool Fwd (Optional)	354
M134	Live Tool Rev (Optional)	354
M135	Live Tool Stop (Optional)	354
M143	Secondary Spindle Forward (Optional)	354
M144	Secondary Spindle Reverse (Optional)	354
M145	Secondary Spindle Stop (Optional)	354
M154	C-Axis Engage (Optional)	354
M155	C-Axis Disengage (Optional)	354

About M-codes

M-codes are miscellaneous machine commands that do not command axis motion. The format for an M-code is the letter M followed by two to three digits; for example M03.

Only one M-code is allowed per line of code. All M-codes take effect at the end of the block.

M00 Stop Program

The M00 code stops a program. It stops the axes, spindle, and turns off the coolant (including auxiliary coolant). The next block after the M00 is highlighted when viewed in the program editor. Press **[CYCLE START]** to continue program operation from the highlighted block.

M01 Optional Program Stop

M01 works the same as M00, except the optional stop feature must be on. Press **[OPTION STOP]** to toggle the feature on and off.

M02 Program End

M02 ends a program.

**NOTE:**

The most common way of ending a program is with an M30.

M03/M04/M05 Spindle 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 379 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. M12 Srrr Pnnn (rrr in RPM and nnn is in milliseconds) turns the air blast on for the specified time, rotates the spindle at the specified speed while the air blast is on, then turns off both the spindle and the air blast automatically. The air blast command for the secondary spindle is 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.

M19 Orient Spindle (Optional)

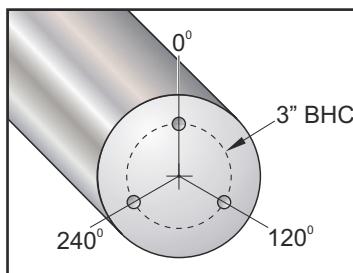
M19 adjusts the spindle to a fixed position. The spindle only orients to the zero position without the optional M19 orient spindle feature.

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

F8.1: M19 Orient Spindle Bolt Hole Circle Example: 3 holes at 120 degrees on 3" BHC.



```
%  
o60191 (M19 ORIENT SPINDLE) ;  
(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) ;  
G00 G54 X3.0 Z0.1 ;  
G98 (Feed per minute) ;  
M19 P0 (Orient spindle) ;  
M14 (Turn on main spindle brake) ;  
P2000 M133 (Live tool on - 2000 RPM) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G01 Z-0.5 F40.0 (Linear feed) ;  
G00 Z0.1 (Rapid retract) ;  
M19 P120 (Orient spindle) ;  
M14 (Turn on main spindle brake) ;
```

```
G01 Z-0.5 (Linear feed) ;  
G00 Z0.1 (Rapid retract) ;  
M19 P240 (Orient spindle) ;  
M14 (Turn on main spindle brake) ;  
G01 Z-0.5 (Linear Feed) ;  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z0.1 M09 (Rapid retract, Coolant off) ;  
M15 (Turn off main spindle brake) ;  
M135 (Turn live tool off) ;  
G53 X0 (X home) ;  
G53 Z0 (Z home & C unwind) ;  
M30 (End program) ;  
%
```

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:**

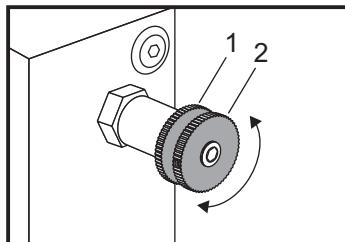
ST10 does not use any settings (105, 106, 107).

Adjust pressure using the valves on the HPU (except ST-40, which uses Setting 241 to define hold pressure). For information on proper ST tailstock pressure, refer to pages 93 and 93.

**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.

F8.2: Set Screw Hold Pressure Valve: [1] Locking knob, [2] Adjustment knob.



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

```
%  
o60381 (M38/39-SSV-SPINDLE SPEED VARIATION) ;  
(G54 X0 Y0 is at the center of rotation) ;  
(Z0 is on the face of the part) ;  
(BEGIN PREPARATION BLOCKS) ;  
T101 (Select tool and offset 1) ;  
G00 G18 G20 G40 G80 G99 (Safe startup) ;  
S1000 M3 (Turn spindle CW at 1000 RPM) ;  
G04 P3. (Dwell for 3 seconds) ;  
M38 (SSV ON) ;
```

```
G04 P60. (Dwell for 60 seconds) ;
M39 (SSV OFF) ;
G04 P5. (Dwell for 5 seconds) ;
G00 G53 X0 (X home) ;
G53 Z0 (Z home & C unwind) ;
M30 (End program) ;
%
```

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.

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

M69 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.

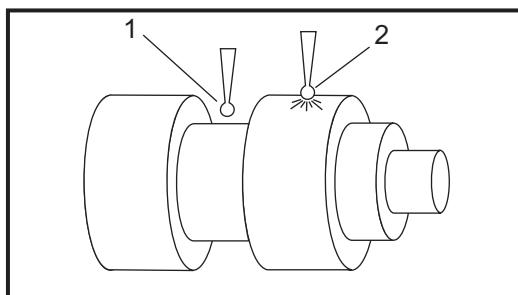
M76/M77 Display Disable/Enable

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.

F8.3: 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.



DANGER: *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. The format of the M95 command is: M95 (hh:mm).

The comment immediately following M95 must contain the duration, in hours and minutes, that you want the machine to sleep. For example, if the current time were 6 p.m. and you want the machine to sleep until 6:30 a.m. the next day, command M95 (12:30). The line(s) after M95 should be axis moves and spindle warm-up commands.

M96 Jump If No 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).

```
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 (subprogram) 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 subprograms within a program as it does not require a separate program. The subprogram must end with an *M99*. An *Lnn* code in the *M97* block will repeat the subprogram call *nn* times.

```
%  
O69701 (M97 LOCAL SUBPROGRAM CALL) ;  
M97 P1000 L2 (L2 will run the N1000 line twice) ;  
M30 ;  
N1000 G00 G90 G55 X0 Z0 (N line that will run after M97 P1000  
is run) ;  
S500 M03 ;  
G00 Z-.5 ;  
G01 X.5 F100. ;  
G03 ZI-.5 ;  
G01 X0 ;  
Z1. F50. ;  
G28 U0 ;  
G28 W0 ;  
G90 ;  
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.

```
%O69801 (M98 SUBPROGRAM CALL) ;M98 P100 L4 (L4 repeats  
subprogram 4 times) ;M30 (End of program) ;%
```

This is the subprogram itself.

```
%  
O69800 (SUBPROGRAM FOR M98 EX o69801) ;  
G00 G90 G55 X0 Z0 ;  
S500 M03 ;  
G00 Z-.5 ;  
G01 X.5 F100. ;
```

```

G03 Z1-.5 ;
G01 X0 ;
Z1. F50. ;
G28 U0 ;
G28 W0 ;
G90 ;
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 subprogram and then using M99 P#nnn after the subprogram 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

P - A number in the range (500-599) representing the macro variable of the same name. 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.

T8.1: Values for ASCII Characters

32		space	59	;	semicolon
33	!	exclamation mark	60	<	less than
34	"	double quotation mark	61	=	equals
35	#	number sign	62	>	greater than
36	\$	dollar sign	63	?	question mark
37	%	percent sign	64	@	at sign
38	&	ampersand	65-90	A-Z	capitol letters
39	,	closed single quote	91	[open square bracket
40	(open parenthesis	92	\	backslash
41)	close parenthesis	93]	closed square bracket
42	*	asterisk	94	^	caret
43	+	plus sign	95	_	underscore
44	,	comma	96	'	open single quote
45	-	minus sign	97-122	a-z	lowercase letters
46	.	period	123	{	open curly bracket
47	/	slash	124		vertical bar

48-57	0-9	numbers	125	}	closed curly bracket
58	:	colon	126	~	tilde

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.

```
%  
O61091 (57 M109_01 Interactive User Input) ;  
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.

```
%  
O61092 (58 M109_02 Interactive User Input) ;  
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. M112 Srrr Pnnn (rrr is in RPM and nnn is in milliseconds) turns the air blast on for the specified time, rotates the spindle at the specified speed while the air blast is on, then turns off both the spindle and the air blast automatically.

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

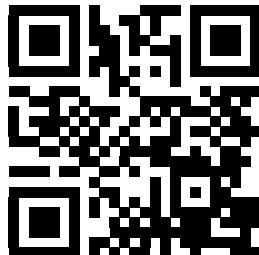
The subspindle speed is controlled with an P address code, for example, P1200 commands a spindle speed of 1200 RPM.

M154/M155 C-Axis Engage/Disengage (Optional)

This M code is used to engage or disengage the optional C-Axis motor.

8.2 More Information Online

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



Chapter 9: Settings

9.1 Introduction

This chapter gives detailed descriptions of the settings that control the way that your machine works.

9.1.1 List of Settings

Setting	Description
1	Auto Power Off Timer
2	Power Off at M30
4	Graphics Rapid Path
5	Graphics Drill Point
6	Front Panel Lock
7	Parameter Lock
8	Prog Memory Lock
9	Dimensioning
10	Limit Rapid at 50%
11	Baud Rate Select
12	Parity Select
13	Stop Bit
14	Synchronization
16	Dry Run Lock Out
17	Opt Stop Lock Out
18	Block Delete Lock Out
19	Feedrate Override Lock

Setting	Description
20	Spindle Override Lock
21	Rapid Override Lock
22	Can Cycle Delta Z
23	9xxx Progs Edit Lock
24	Leader To Punch
25	EOB Pattern
26	Serial Number
28	Can Cycle Act w/o X/Z
31	Reset Program Pointer
32	Coolant Override
33	Coordinate System
36	Program Restart
37	RS-232 Data Bits
39	Beep @ M00, M01, M02, M30
41	Add Spaces RS-232 Out
42	M00 After Tool Change
43	Cutter Comp Type
44	Min F in Radius TNC %
45	Mirror Image X-axis
47	Mirror Image Z-axis
52	G83 Retract Above R
53	Jog w/o Zero Return
55	Enable DNC from MDI

Setting	Description
56	M30 Restore Default G
57	Exact Stop Canned X-Z
58	Cutter Compensation
59	Probe Offset X+
60	Probe Offset X-
61	Probe Offset Z+
62	Probe Offset Z-
63	Tool Probe Width
64	T. Ofs Meas Uses Work
65	Graph Scale (Height)
66	Graphics X Offset
68	Graphics Z Offset
69	DPRNT Leading Spaces
70	DPRNT Open/CLOS DCode
72	Can Cycle Cut Depth
73	Can Cycle Retraction
74	9xxx Progs Trace
75	9xxx Progs Single BLK
76	Foot Pedal Lock Out
77	Scale Integer F
81	Tool at Auto Off
82	Language
83	M30/Resets Overrides

Setting	Description
84	Tool Overload Action
85	Maximum Corner Rounding
86	Thread Finish Allowance
87	TNN Resets Override
88	Reset Resets Overrides
90	Graph Z Zero Location
91	Graph X Zero Location
92	Chuck Clamping
93	Tailstock X Clearance
94	Tailstock Z Clearance
95	Thread Chamfer Size
96	Thread Chamfer Angle
97	Tool Change Direction
98	Spindle Jog RPM
99	Thread Minimum Cut
100	Screen Saver Delay
101	Feed Override -> Rapid
102	C Axis Diameter
103	CYC START/FH Same Key
104	Jog Handle to SNGL BLK
105	TS Retract Distance
106	TS Advance Distance
107	TS Hold Point

Setting	Description
109	Warm-Up Time in MIN.
110	Warmup X Distance
112	Warmup Z Distance
113	Tool Change Method
114	Conveyor Cycle Time (minutes)
115	Conveyor On Time (minutes)
118	M99 Bumps M30 CNTRS
119	Offset Lock
120	Macro Var Lock
121	Foot Pedal TS Alarm
122	Secondary Spindle Chuck Clamping
131	Auto Door
132	Jog Before TC
133	REPT Rigid Tap
142	Offset Chng Tolerance
143	Machine Data Collect
144	Feed Override->Spindle
145	TS at Part for CS
156	Save Offset with PROG
157	Offset Format Type
158	X Screw Thermal COMP%
159	Y Screw Thermal COMP%
160	Z Screw Thermal COMP%

Setting	Description
162	Default To Float
163	Disable .1 Jog Rate
164	Powerup SP Max RPM
165	SSV Variation (RPM)
166	SSV CYCLE (0.1) SECS
167-186	Periodic Maintenance
187	Machine Data Echo
196	Conveyor Shutoff
197	Coolant Shutoff
198	Background Color
199	Backlight Timer
201	Show Only Work and Tool Offsets In Use
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

Setting	Description
214	Show Rapid Path Live Image
215	Show Feed Path Live Image
216	Servo and Hydraulic Shutoff
217	Show Chuck Jaws
218	Show Final Pass
219	Auto Zoom to Part
220	TS Live Center Angle
221	Tailstock Diameter
222	Tailstock Length
224	Flip Part Stock Diameter
225	Flip Part Stock Length
226	SS Stock Diameter
227	SS Stock Length
228	SS Jaw Thickness
229	SS Clamp Stock
230	SS Jaw Height
231	SS Jaw Step Height
232	G76 Default P Code
233	SS Clamping Point
234	SS Rapid Point
235	SS Machine Point
236	FP Z Stock Face
237	SS Z Stock Face

Setting	Description
238	High Intensity Light Timer (minutes)
239	Worklight Off Timer (minutes)
240	Tool Life Warning
241	Tailstock Hold Force
242	Air Water Purge Interval (minutes)
243	Air Water Purge On-Time (seconds)
245	Hazardous Vibration Sensitivity
249	Enable Haas Startup Screen
900	CNC Network Name
901	Obtain Address Automatically
902	IP Address
903	Subnet Mask
904	Default Gateway
905	DNS Server
906	Domain/Workgroup Name
907	Remote Server Name
908	Remote Share Path
909	User Name
910	Password
911	Access To CNC Share
912	Floppy Tab Enabled
913	Hard Drive Tab Enabled
914	USB Tab Enabled

Setting	Description
915	Net Share
916	Second USB Tab Enabled

Introduction to Settings

The setting pages contain values that control machine operation and that you may need to change.

Settings are presented in tabbed menus. For information on navigating tabbed menus in the Haas control, refer to page **48**. The on-screen settings are organized into groups.

Use the **[UP]** and **[DOWN]** cursor arrow keys to highlight a setting. To quickly access a setting, with the Settings display active on the screen, type the setting number and press the **[DOWN]** cursor arrow.

Some settings have numerical values that fit in a given range. To change the value of these settings, type the new value and press **[ENTER]**. Other settings have specific available values that you select from a list. For these settings, use the **[LEFT]** and **[RIGHT]** cursor arrow keys to display the choices. Press **[ENTER]** to change the value. The message near the top of the screen tells you how to change the selected setting.

1 - Auto Power Off Timer

This setting is used to automatically power-down the machine after a period of idle time. The value entered in this setting is the number of minutes the machine remains idle until it is powered down. The machine does not power down while a program is running, and the time (number of minutes) starts back at zero anytime a button is pressed or the **[HANDLE JOG]** control is used. The auto-off sequence gives the operator a 15-second warning before power down, at which time pressing any button stops the power down.

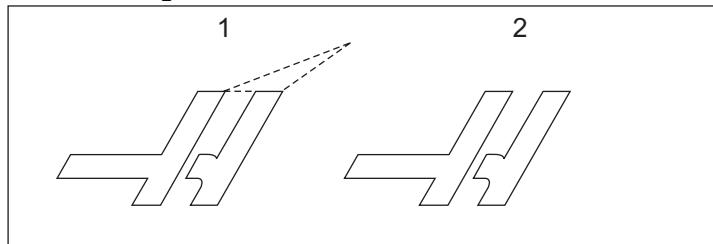
2 - Power Off at M30

If this setting is set to **ON**, the machine powers down at the end of a program (**M30**). The machine gives the operator a 15-second warning once an **M30** is reached. Press any key to interrupt the power-off sequence.

4 - Graphics Rapid Path

This setting changes the way a program is viewed in the Graphics mode. When it is **OFF**, rapid, non-cutting tool motions do not leave a path. When it is **ON**, rapid tool motions leave a dashed line on the screen.

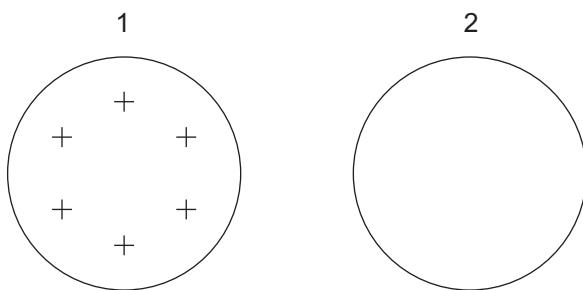
- F9.1:** Setting 4 -Graphics Rapid Path: [1] All Rapid Tool Motions Shown with a Dashed Line When **ON**. [2] **Only Cut Lines Shown When OFF**.



5 - Graphics Drill Point

This setting changes the way a program is viewed in Graphics mode. When it is **ON**, motion in the Z-Axis will leave an **X** mark on the screen. When it is **OFF**, no additional marks are shown on the graphics display.

- F9.2:** Setting 5, Graphics Drill Point: [1] X Mark Displays When **ON**. [2] No X Marks Shown When **OFF**.



6 - Front Panel Lock

When set to **ON**, this Setting disables the Spindle [**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:

*Each time the control is powered up, this setting is set to **ON**.*

8 - Prog Memory Lock

This setting locks out the memory editing functions ([ALTER], [INSERT], etc.) when it is set to **ON**. This also locks out MDI. Editing functions are not restricted by this setting.

9 - Dimensioning

This setting selects between inch and metric mode. When it is set to **INCH**, the programmed units for X and Z are inches, to 0.0001". When it is set to **MM**, programmed units are millimeters, to 0.001 mm. All offset values are converted when this setting is changed from inches to 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	Metric
Feed	in/min and in/rev	mm/min and mm/rev
Max Travel	Varies by axis and model	
Minimum programmable dimension	.0001	.001

Axis jog key	Inch	Metric
.0001	.0001 in/jog click	.001 mm/jog click
.001	.001 in/jog click	.01 mm/jog click
.01	.01 in/jog click	.1 mm/jog click
.1	.1 in/jog click	1 mm/jog click

10 - Limit Rapid at 50%

Turning this setting **ON** limits the machine to 50% of its fastest non-cutting axis motion (rapids). This means, if the machine can position the axes at 700 inches per minute (ipm), it is limited to 350 ipm when this setting is **ON**. The control displays a 50% rapid override message, when this setting is **ON**. When it is **OFF**, the highest rapid speed of 100% is available.

11 - Baud Rate Select

This setting allows the operator to change the rate at which data is transferred to/from the serial port (RS-232). This applies to the upload/download of programs, etc., and to DNC functions. This setting must match the transfer rate from the personal computer.

12 - Parity Select

This setting defines parity for the RS-232 Serial Port. When set to **NONE**, no parity bit is added to the serial data. When set to **ZERO**, a 0 bit is added. **EVEN** and **ODD** work like normal parity functions. Make sure you know what your system needs, for example, **XMODEM** must use 8 data bits and no parity (set to **NONE**). This setting must match the parity from the personal computer.

13 - Stop Bit

This setting designates the number of stop bits for the RS-232 Serial Port. It can be **1** or **2**. This setting must match the number of stop bits from the personal computer.

14 - Synchronization

This setting changes the synchronization protocol between sender and receiver for the RS-232 Serial Port. This setting must match the synchronization protocol from the personal computer.

When set to **RTS/CTS**, the signal wires in the serial data cable are used to tell the sender to temporarily stop sending data while the receiver catches up.

When set to **XON/XOFF**, the most common setting, ASCII character codes are used by the receiver to tell the sender to temporarily stop.

The selection **DC CODES** is like **XON/XOFF**, except that paper tape punch or reader start/stop codes are sent.

XMODEM is a receiver-driven communications protocol that sends data in blocks of 128 bytes. **XMODEM** has added reliability as each block is checked for integrity. **XMODEM** must use 8 data bits and no parity.

16 - Dry Run Lock Out

The Dry Run feature is not available when this setting is turned **ON**.

17 - Opt Stop Lock Out

The Optional Stop feature is not available when this setting is **ON**.

18 - Block Delete Lock Out

The Block Delete feature is not available when this setting is **ON**.

19 - Feedrate Override Lock

The feedrate override buttons are disabled when this setting is turned **ON**.

20 - Spindle Override Lock

The spindle speed override keys are disabled when this setting is turned **ON**.

21 - Rapid Override Lock

The axis rapid override keys are disabled when this setting is turned **ON**.

22 - Can Cycle Delta Z

This setting specifies the distance to retract the Z Axis to clear chips during a G73 irregular path stock removal cycle. The range is 0.0000 to 29.9999 inches (0-760 mm).

23 - 9xxx Progs Edit Lock

Turning this setting **ON** prevents the 9000 series of programs from being viewed in memory, edited, or deleted. 9000 series programs cannot be uploaded or downloaded while this setting is **ON**.



NOTE:

9000 series programs are usually macro programs.

24 - Leader To Punch

This setting is used to control the leader (the blank tape at the beginning of a program) sent to a paper tape punch device connected to the RS-232 Serial Port.

25 - EOB Pattern

This setting controls the EOB (End of Block) pattern when data is sent and received to/from the serial port (RS-232). This setting must match the EOB pattern from the personal computer. The choices are CR LF, LF ONLY, LF CR CR, and CR ONLY.

26 - Serial Number

This is the serial number of the machine. It cannot be changed.

28 - Can Cycle Act w/o X/Y

This is an **ON/OFF** setting. The preferred setting is **ON**.

When it is **OFF**, the initial canned cycle definition block requires an **X** or **Y** code for the canned cycle to be executed.

When it is **ON**, the initial canned cycle definition block causes one cycle to be executed even when there is no X or Y code in the block.

**NOTE:**

When an L0 is in that block, it will not execute the canned cycle on the definition line. This setting has no effect on G72 cycles.

31 - Reset Program Pointer

When this setting is **OFF**, [RESET] does not change the position of the program pointer. When it is **ON**, pressing [RESET] moves the program pointer to the beginning of the program.

32 - Coolant Override

This setting controls how the coolant pump operates. The **NORMAL** selection allows the operator to turn the pump on and off manually or with M-codes. The **OFF** selection gives the message **FUNCTION LOCKED** if an attempt is made to turn the coolant on manually or from a program. The **IGNORE** selection ignores all programmed coolant commands, but the pump can be turned on manually.

33 - Coordinate System

This setting changes the way 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 make sure that the tools, offsets, G and M codes, and axis positions are set correctly before the program starts at the block where the cursor is positioned. The control processes these M codes 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 Setting 36 is **OFF** the control starts the program, but it does not check the conditions of the machine. Having this setting **OFF** may save time when running a proven program.

37 - RS-232 Data Bits

This setting is used to change the number of data bits for the Serial Port (RS-232). This setting must match the data bits from the personal computer. Normally 7 data bits should be used but some computers require 8. **XMODEM** must use 8 data bits and no parity.

39 - Beep @ M00, M01, M02, M30

Turning this setting **ON** causes the keyboard beeper to sound when an **M00**, **M01** (with Optional Stop active), **M02**, or an **M30** is found. The beeper continues until a button is pressed.

41 - Add Spaces RS-232 Out

When this setting is **ON**, spaces are added between address codes when a program is sent out via the RS-232 serial port. This can make a program much easier to read/edit on a personal computer (PC). When it is set to **OFF**, programs sent out the serial port have no spaces and are more difficult to read.

42 - M00 After Tool Change

Turning this setting **ON** stops the program after a tool change and a message is displayed stating this. **[CYCLE START]** must be pressed to continue the program.

43 - Cutter Comp Type

This controls how the first stroke of a compensated cut begins and the way the tool is cleared from the part. The selections can be **A** or **B**; see the Tool Nose Compensation section on page 143.

44 - Min F in Radius TNC %

Minimum feedrate in radius tool nose compensation percent affects the feedrate when cutter compensation moves the tool towards the inside of a circular cut. This type of cut slows 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.

49 - Skip Same Tool Change

In a program, the same tool may be called in the next section of a program or a subprogram. The control does two tool changes and finishes with the same tool in the spindle. Turning this setting **ON** skips same-tool changes; a tool change only occurs if a different tool is placed in the spindle.

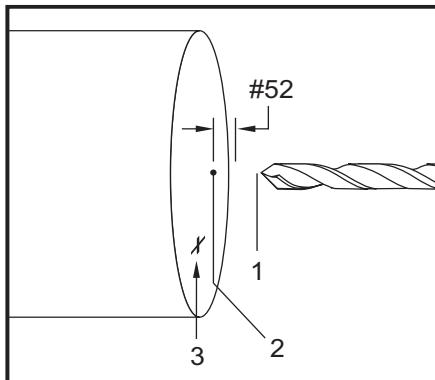

NOTE:

This setting only affects machines with carousel (umbrella) tool changers.

52 - G83 Retract Above R

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.

- F9.3:** 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**.

55 - Enable DNC from MDI

Turning this setting **ON** makes the DNC feature available. DNC is selected in the control by pressing **[MDI/DNC]** twice.

The DNC Direct Numerical Control feature is not available when Setting 55 is set to **OFF**.

56 - M30 Restore Default G

When this setting is **ON**, ending a program with M30 or pressing [**RESET**] returns all modal G-codes to their defaults.

57 - Exact Stop Canned X-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** makes the XZ motion come to an exact stop.

58 - Cutter Compensation

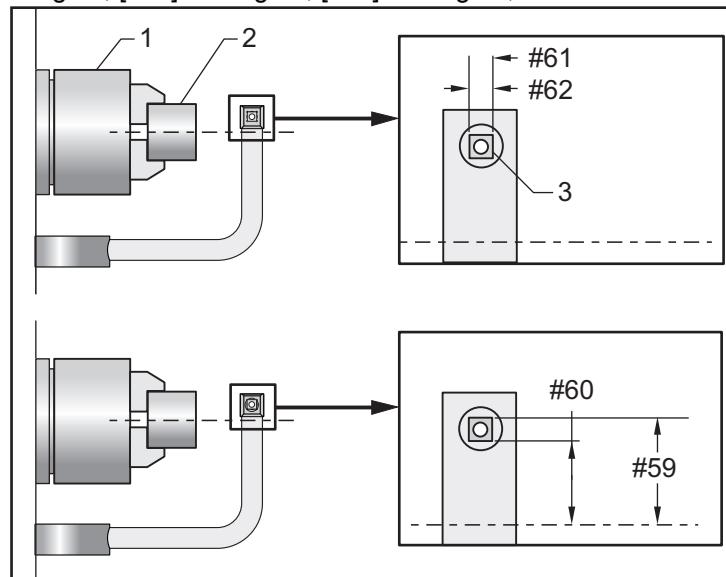
This setting selects the type of cutter compensation used (**FANUC** or **YASNAC**). Refer to the tool functions section on page 138.

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.

- F9.4:** 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 - T. Ofs Meas Uses Work

The (Tool Offset Measure Uses Work) setting changes the way the [Z FACE MEASURE] key works. When this is **ON**, the entered tool offset is the measured tool offset plus the work coordinate offset (Z-axis). When it is **OFF**, the tool offset equals the Z machine position.

65 - Graph Scale (Height)

This setting specifies the height of the work area that is displayed on the Graphics mode screen. The default value for this setting is the total X travel.

Total X travel = Parameter 6/Parameter 5

Scale = Total X travel/Setting 65

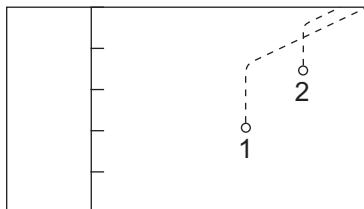
66 - Graphics X Offset

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.

F9.5: 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 uses leading spaces. This example illustrates control behavior when this setting is **OFF** or **ON**.

```
%  
#1 = 3.0 ;  
G0 G90 X#1 ;  
DPRNT[X#1[44]] ;  
%
```

OUTPUT

OFF	ON
X3.0000	X 3.0000

Notice the space between the X and the 3 when the setting is **ON**. Information can be easier to read when this setting is **ON**.

70 - DPRNT Open/CLOS DCode

This setting controls whether the **POPEN** and **PCLOS** statements in macros send DC control codes to the serial port. When the setting is **ON**, these statements send DC control codes. When it is **OFF**, the control codes are suppressed. The default value is **ON**.

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 (**O9xxxx**). When the setting is **OFF**, the control does not display the 9000 series code.

75 - 9xxxx Progs Single BLK

When Setting 75 is **ON** and the control is operating in Single Block mode, then the control stops at each block of code in a macro program (**O9xxxx**) and waits for the operator to press **[CYCLE START]**. When Setting 75 is **OFF** the macro program is run continuously, the control does not pause at each block, even if Single Block is **ON**. The default setting is **ON**.

When Setting 74 and Setting 75 are both **ON**, the control acts normally. That is, all blocks executed are highlighted and displayed, and when in Single-Block mode there is a pause before each block is executed.

When Setting 74 and Setting 75 are both **OFF**, the control executes 9000 series programs without displaying the program code. If the control is in Single-Block mode, no single-block pause occurs during the running of the 9000 series program.

When Setting 75 is **ON** and Setting 74 is **OFF**, 9000 series programs are displayed as they are executed.

76 - 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 you always use a decimal point.) This setting helps operators run programs developed on a control other than Haas. For example **F12** becomes:

- 0.0012 units/minute with Setting 77 **OFF**
- 12.0 units/minute with Setting 77 **ON**

There are 5 feedrate settings. This chart shows the effect of each setting on a given F10 address.

INCH		MILLIMETER	
DEFAULT	(.0001)	DEFAULT	(.001)
INTEGER	F1 = F1	INTEGER	F1 = F1
.1	F10 = F1.	.1	F10 = F1.
.01	F10 = F.1	.01	F10 = F.1
.001	F10 = F.01	.001	F10 = F.01
.0001	F10 = F.001	.0001	F10 = F.001

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 with the **[LEFT]** and **[RIGHT]** cursor arrows, then press **[ENTER]**.

83 - M30/Resets Overrides

When this setting is **ON**, an M30 restores any overrides (feedrate, spindle, rapid) to their default values (100%).

84 - Tool Overload Action

When a tool becomes overloaded, Setting 84 designates the control response. These settings cause specified actions (refer to the Additional Tooling Set-up on page 85):

- **ALARM** causes the machine to stop.
- **FEEDHOLD** displays the message *Tool Overload* and the machine stops in a feedhold situation. Press any key to clear the message.
- **BEEP** causes an audible noise (beep) from the control.
- **AUTOFEE**D causes the control to automatically limit the feedrate based on the tool load.



NOTE:

When tapping (rigid or floating), the feed and spindle overrides are locked out, so the AUTOFEE setting is ineffective (the control appears to respond to the override buttons, by displaying the override messages).



CAUTION:

Do not use the AUTOFEE setting when thread milling or auto reversing tapping heads, as it can cause unpredictable results or even a crash.

The last commanded feedrate is restored at the end of program execution, or when the operator presses **[RESET]** or turns **OFF** the **AUTOFEE** setting. The operator can use **[FEEDRATE OVERRIDE]** while the **AUTOFEE** setting is selected. These keys are recognized by the **AUTOFEE** setting as the new commanded feedrate as long as the tool load limit is not exceeded. However, if the tool load limit has already been exceeded, the control ignores **[FEEDRATE OVERRIDE]**.

85 - Maximum Corner Rounding

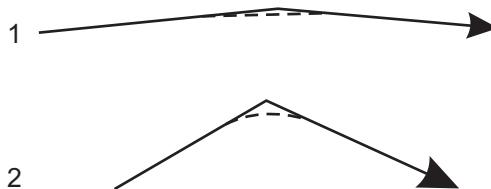
This setting defines the machining accuracy tolerance around corners. The initial default value is 0.05". This means that the control keeps the radii of corners no bigger than 0.05".

Setting 85 causes the control to adjust feeds around corners to meet the tolerance value. The lower the value of Setting 85, the slower the control feeds around corners to meet the tolerance. The higher the value of Setting 85, the faster the control feeds around corners, up to the commanded feedrate, but it could round the corner off to a radius up to the tolerance value.

**NOTE:**

The angle of the corner also affects the change to the feedrate. The control can cut shallow corners within tolerance at a higher feedrate than it can with tighter corners.

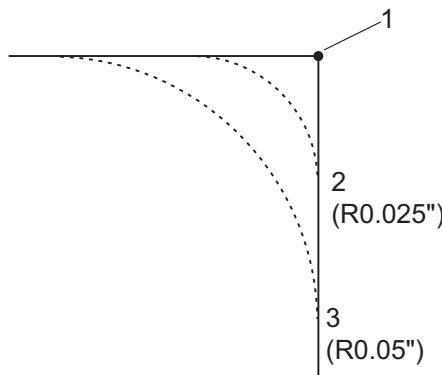
- F9.6:** The control can cut corner [1] within tolerance at a higher feedrate than it can cut corner [2].



If Setting 85 has a value of zero, the control acts as if exact stop is active in each motion block.

Refer also to G187 – Accuracy Control (Group 00) on page 320.

- F9.7:** Assume that the commanded feedrate is too high to achieve corner [1]. If Setting 85 has a value of 0.025, then the control slows the feedrate enough to achieve corner [2] (with a radius of 0.025"). If Setting 85 has a value of 0.05, then the control slows the feedrate enough to achieve corner [3]. The feedrate to achieve corner [3] is faster than the feedrate to achieve corner [2].



86 - 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 programmed values or defaults (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 -8.0000.

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.

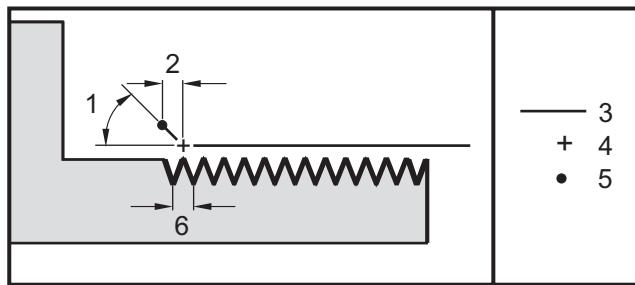
95 - Thread Chamfer Size

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).*

- F9.8:** 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 this setting has a value of zero, the screen saver is disabled. A nonzero value specifies the number of minutes before the screen saver starts. Press [CANCEL] to exit the screen saver. The screen saver does not start if the control is in Sleep, Jog, Edit, or Graphics mode.

101 - Feed Override -> Rapid

Pressing [HANDLE CONTROL FEED], with this setting ON, will cause the jog handle to affect both the feedrate and the rapid rate overrides. Setting 10 affects the maximum rapid rate. The rapid rate cannot exceed 100%. Also, [+10% FEEDRATE], [-10% FEEDRATE], and [100% FEEDRATE] change the rapid and feed rate together.

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 automatically turns off.

104 - Jog Handle to SNGL BLK

The [HANDLE JOG] control can single-step through a program when this setting is ON. Reversing the [HANDLE JOG] control direction generates a feed hold.

This setting cannot be turned on while Setting 103 is ON. When one of them is set to ON, the other automatically turns off.

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 control gives this warning:

CAUTION! Warm up Compensation is specified!

Do you wish to activate

Warm up Compensation (Y/N) ?

If a Y is entered, the control immediately applies the total compensation (Setting 110, 111, 112), and the compensation begins to decrease as the time elapses. For instance, after 50% of the time in Setting 109 has elapsed, the compensation distance is 50%.

To restart the time period, it is necessary to power the machine off and on, and then answer YES to the compensation query at start-up.



CAUTION:

Changing Setting 110, 111, or 112 while compensation is in progress can cause a sudden movement of up to 0.0044 inch.

The amount of remaining warm-up time is displayed on the bottom right hand corner of the Diagnostics Inputs 2 screen using the standard hh:mm:ss format.

110/112 - 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 in the Lathe Operator's Manual.

114 - Conveyor Cycle (minutes)

Setting 114 Conveyor Cycle Time is the interval that the conveyor turns on automatically. For example, if setting 114 is set to 30, the chip conveyor turns on every half an hour. On-time should be set no greater than 80% of cycle time. Refer to Setting 115 on page 383.

NOTE: *The [CHIP FWD] button (or M31) starts the conveyor in the forward direction and starts the cycle.*

The [CHIP STOP] button (or M33) stops the conveyor and cancels the cycle.

115 - Conveyor On-time (minutes)

Setting 115 Conveyor On-Time is the amount of time the conveyor runs. For example, if setting 115 is set to 2, the chip conveyor runs for 2 minutes, then turns off. On-time should be set no greater than 80% of cycle time. Refer to Setting 114 Cycle Time on page 383.

NOTE: *The [CHIP FWD] button (or M31) starts the conveyor in the forward direction and starts the cycle.*

The [CHIP STOP] button (or M33) stops the conveyor and cancels the cycle.

118 - M99 Bumps M30 CNTRS

When this setting is ON, an M99 adds one to the M30 counters (these are visible after pressing [CURRENT COMMANDS]).



NOTE: *M99 only increases the counters as it occurs in a main program, not in a sub-program.*

119 - Offset Lock

Turning the setting ON does not allow the values in the Offset display to be altered. However, programs that alter offsets with macros or G10 are permitted to do so.

120 - Macro Var Lock

Turning this setting ON does not allow the macro variables to be altered. However, programs that alter macro variables can do so.

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).

**NOTE:**

The M-codes work only while the machine receives a cell-safe signal from a robot. For more information, contact a robot integrator.

The door closes when **[CYCLE START]** is pressed and opens when the program reaches an M00, M01 (with Optional Stop turned on) or M30 and the spindle has stopped turning.

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 - REPT Rigid Tap

This setting (Repeat Rigid Tap) ensures that the spindle is oriented during tapping so that the threads line up when a second tapping pass is programmed in the same hole.

**NOTE:**

This setting must be ON when a program commands peck tapping.

142 - Offset Chng Tolerance

This setting generates a warning message if an offset is changed by more than the amount entered for this setting. If an attempt is made to change an offset by more than the entered amount (either positive or negative), the control gives this prompt: *XX changes the offset by more than Setting 142! Accept (Y/N)?*

If **Y** is entered, the control updates the offset as usual, otherwise, the change is rejected.

143 - Machine Data Collect

This setting enables the user to extract data from the control using one or more Q commands sent through the RS-232 port, and to set Macro variables by using an E command. This feature is software-based and requires an additional computer to request, interpret, and store data from the control. A hardware option also allows the reading of machine status. For detailed information, refer to the Machine Data Collection section on page 78.

144 - Feed Override->Spindle

This setting is intended to keep the chip load constant when an override is applied. When this setting is **ON**, any feedrate override is also applied to the spindle speed, and the spindle overrides are disabled.

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.

156 - Save Offset with PROG

When this setting is **ON**, the control includes the offsets in the program file when you save it. The offsets appear in the file before the final % sign, under the heading 0999999.

When you load the program back into memory, the control prompts *Load Offsets (Y/N?)*. Press **Y** if you want to load the saved offsets. Press **N** if you do not want to load them.

157 - Offset Format Type

This setting controls the format in which offsets are saved with programs.

When it is set to **A** the format looks like what is displayed on the control, and contains decimal points and column headings. Offsets saved in this format can be edited on a PC and later reloaded.

When it is set to **B**, each offset is saved on a separate line with an **N** value and a **V** value.

158,159,160 - X, Y, Z Screw Thermal COMP%

These settings can be set from -30 to +30 and adjust the existing screw thermal compensation by -30% to +30% accordingly.

162 - Default To Float

When this setting is **ON**, the control adds a decimal point to values entered without a decimal point for certain address codes. When the setting is **OFF**, values after address codes that do not include decimal points are taken as machinist's notation; for example, thousandths or ten-thousandths.

	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 these address codes:

X, Y, Z, A, B, C, E, F, I, J, K, U, W

Including A, D, and R except when:

- the **A** value (tool angle) is in a G76 block. If a G76 A value containing a decimal point is found during program execution, Alarm 605 - Invalid Tool Nose Angle is generated.
- the **D** value is in a G73 block.
- the **R** value is in a G71 block in YASNAC mode.



NOTE:

This setting affects the interpretation of all programs entered manually, 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 clamp the spindle speed each time the machine is powered on. It causes a G50 Snnn command to be executed at power on time, where nnn is the value from Setting 164. If nnn contains zero, or a value equal to or greater than parameter 131 MAX SPINDLE RPM, Setting 164 has 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

When this setting is **ON**, the data collection Q commands issued from the user's PC are shown on the PC screen. When this setting is **OFF**, the PC screen does not show these commands.

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

This setting is the amount of time to wait without activity before Coolant flow stops. Units are minutes.

198 - Background Color

Specifies the background color for inactive display panes. Range is 0 to 254. The default value is 235.

199 - Backlight Timer

This setting is the time in minutes after which the machine display backlight turns off when there is no input at the control (except in JOG, GRAPHICS, or SLEEP mode or when an alarm is present). Press any key to restore the screen (**[CANCEL]** is preferred).

201 - Show Only Work and Tool Offsets In Use

When this setting is **ON** only the Work and Tool offsets used by the running program display. The program must run in the graphics mode first to turn on this feature.

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 off the servomotors and hydraulic pump, if equipped, after the specified number of minutes has elapsed without activity, such as running a program, jogging, button 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.

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 appear with an orange background in the ATM screen, and the beacon light flashes 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 begins.

243 - Air Water Purge On-Time (seconds)

This setting specifies the duration of the purge of condensates in the system air reservoir. The units are seconds. When the time specified by Setting 242 lapses, starting from midnight, the purge is begun for the number of seconds specified by Setting 243.

245 - Hazardous Vibration Sensitivity

This setting selects from three sensitivity levels (**LOW**, **MEDIUM**, or **HIGH**) for the hazardous vibration sensor (if installed). This setting defaults to **HIGH** each time the machine is powered up.

249 - Enable Haas Startup Screen

If this setting is **ON**, the screen shows startup instructions each time the machine is powered on. You can turn Setting 249 **ON** or **OFF** through the settings page, or you can press **[F1]** at the startup screen to turn it off.

900 - CNC Network Name

This setting contains the control name you would like to show up on the network.

901 - Obtain Address Automatically

Retrieves a TCP/IP address and subnet mask from a DHCP server on a network (Requires a DHCP server). When DHCP is on, TCP/IP, SUBNET MASK and GATEWAY entries are no longer required and has ******* entered.

**NOTE:**

The ADMIN section at the end provides the IP address from DHCP.

The machine must be turned off and back on for changes to this setting to take effect.

**NOTE:**

To get IP settings from DHCP:

1. At the control, press **[LIST PROGRAM]**.
2. Press **[CANCEL]**.
3. Press the right arrow for the Hard Drive directory and press **[ENTER]**.
4. Type in **ADMIN** and press **[INSERT]**.

5. Select ADMIN folder and press **[ENTER]**.
6. Copy the ipconfig.txt file to disk or USB and read it on a Windows computer.

902 - IP Address

This setting is needed on a network with static TCP/IP addresses (DHCP Off). The network administrator assigns an address (example 192.168.1.1). The machine must be turned off and back on for changes to this setting to take effect.

**NOTE:**

The address format for Subnet Mask, Gateway and DNS is XXX.XXX.XXX.XXX (example 255.255.255.255). Do not end the address with a period. The max address is 255.255.255.255; no negative numbers.

903 - Subnet Mask

This setting is needed on a network with static TCP/IP addresses. The network administrator assigns a mask value. The machine must be turned off and back on for changes to this setting to take effect.

904 - Default Gateway

This setting is needed to gain access through routers. The network administrator assigns an address. The machine must be turned off and back on for changes to this setting to take effect.

905 - DNS Server

This setting contains the Domain Name Server or Domain Host Control Protocol IP address on the network. The machine must be turned off and back on for changes to this setting to take effect.

906 - Domain/Workgroup Name

This setting is the CNC control workgroup or domain. The machine must be turned off and back on for changes to this setting to take effect.

907 - Remote Server Name

For Haas machines with WINCE FV 12.001 or higher, this setting contains the NETBIOS name from the computer where the share folder resides. IP address is not supported.

908 - Remote Share Path

This setting contains the name of the shared network folder. To rename the shared folder after a host name is selected, enter the new shared folder name and press [ENTER].



NOTE:

Do not use spaces in the shared folder name.

909 - User Name

This setting is the name used to logon to the server or domain (using a user domain account). The machine must be turned off and back on for changes to this setting to take effect. User Names are case sensitive and cannot contain spaces.

910 - Password

This setting is the password used to logon to the server. The machine must be turned off and back on for changes to this setting to take effect. Passwords are case sensitive and cannot contain spaces.

911 - Access to CNC Share

This setting is used for the CNC hard drive read/write privileges. **OFF** stops the hard drive from being networked. **FULL** allows read/write access to the drive from the network. Turning off this setting and Setting 913 disables network card communication.

912 - Floppy Tab Enabled

Refer to Setting 914 USB Tab Enabled for this functionality. (Older software used this setting to turn off/on access to the USB floppy drive. When set to **OFF**, the USB floppy drive is not accessible.)

913 - Hard Drive Tab Enabled

This setting turns off/on access to the hard drive. When set to **OFF**, the hard drive is not accessible. Turning off this setting and CNC Share (Setting 911) disables network card communication.

914 - USB Tab Enabled

Turns access to the USB port **OFF/ON**. When set to **OFF** the USB port will not be accessible.

915 - Net Share

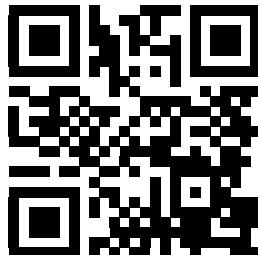
This setting turns off/on access to the server drive. When set to **OFF** access to the sever from the CNC control is not possible.

916 - Second USB Tab Enabled

This setting turns off/on access to the secondary USB port. When set to OFF the USB port will not be accessible.

9.2 More Information Online

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



Chapter 10: Maintenance

10.1 Introduction

Regular maintenance is important to make sure that your machine has a long and productive life with minimal downtime. The most common maintenance tasks are simple and you can do them yourself. You can also ask your HFO about their comprehensive preventive maintenance program for complex maintenance tasks.

10.2 Maintenance Monitor

The Haas control features a maintenance monitor to tell you when you need to do certain maintenance tasks. There are (14) included maintenance items, and (6) spare items that you can designate yourself.

10.2.1 Maintenance Settings

Settings 167-186 control the default maintenance interval for each maintenance item. The Maintenance Monitor page shows only maintenance items that have a default interval (non-zero).

Maintenance intervals have (3) possible unit values:

- On-time (hours): The control counts down this interval while the machine power is on.
- Motion time (hours): The control counts down this interval only while the specified component is in motion.
- Tool changes (each): The control counts this interval down by (1) after each tool change.

You can change each setting to increase or decrease the default interval. At the end of each maintenance interval, the control will show a *Maintenance Due* message and icon. Go to the maintenance monitor page to see the maintenance required.

F10.1: Maintenance Settings Tab

GENERAL	PROGRAM	I/O	CONTROL PANEL	SYSTEM	MAINTENANCE	POWER SETTINGS
MAINT DEFALTS						
167 Coolant Replacement default in power-on hours					1000	
168 Control Air Filter Replacement default in power-on hours					0	
169 Oil Filter Replacement default in power-on hours					2500	
170 Gearbox Oil Replacement default in power-on hours					5000	
171 Coolant Tank Level Check default in power-on hours					20	
172 Way Lube Level Check default in motion-time hours					250	
173 Gearbox Oil Level Check default in power-on hours					250	
174 Seals/Wipers Inspection default in motion-time hours					250	
175 Air Supply Filter Check default in power-on hours					40	
176 Hydraulic Oil Level Check default in power-on hours					100	
177 Hydraulic Filter Replacement default in motion_time hours					150	
178 Grease Fittings default in motion_time hours					250	
179 Grease Chuck default in motion_time hours					0	
180 Grease Tool Changer Cams default in tool-changes					1000	
181 Spare Maintenance Setting #1 default in power-on hours					0	
182 Spare Maintenance Setting #2 default in power-on hours					0	
183 Spare Maintenance Setting #3 default in motion-time hours					0	
184 Spare Maintenance Setting #4 default in motion-time hours					0	
185 Spare Maintenance Setting #5 default in tool-changes					0	
186 Spare Maintenance Setting #6 default in tool-changes					0	

10.2.2 The Maintenance Monitor Page

To find the Maintenance Monitor Page:

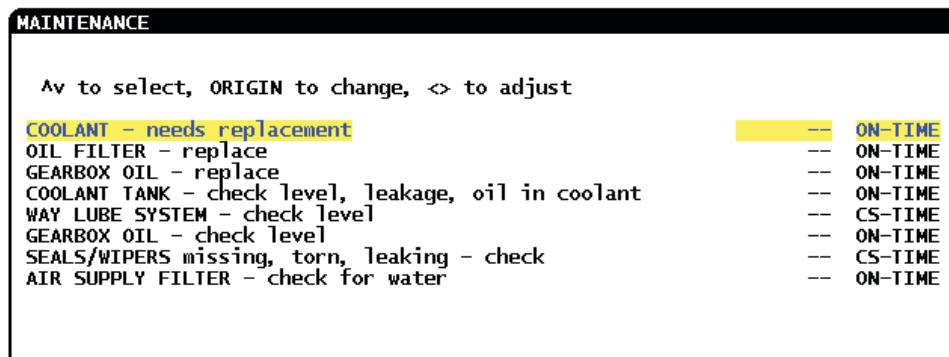
1. Press [CURRENT COMMANDS].
2. Press [PAGE UP] or [PAGE DOWN] until you see the Maintenance page.

F10.2: The Maintenance Page

MAINTENANCE	
<i>Av to select, ORIGIN to change, <> to adjust</i>	
COOLANT - needs replacement	-- ON-TIME
OIL FILTER - replace	-- ON-TIME
GEARBOX OIL - replace	-- ON-TIME
COOLANT TANK - check level, leakage, oil in coolant	-- ON-TIME
WAY LUBE SYSTEM - check level	-- CS-TIME
GEARBOX OIL - check level	-- ON-TIME
SEALS/WIPERS missing, torn, leaking - check	-- CS-TIME
AIR SUPPLY FILTER - check for water	-- ON-TIME

10.2.3 Start, Stop, or Adjust Maintenance Monitoring

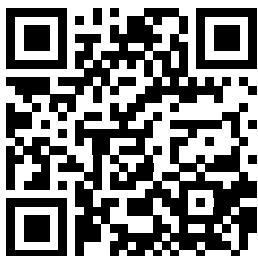
To start or stop monitoring on the maintenance page:



1. Use the [UP] or [DOWN] cursor arrow keys to highlight a maintenance item.
Maintenance items that show -- instead of a number are not currently monitored.
2. Press [ORIGIN] to start monitoring the item. The -- changes into the default maintenance interval.
3. To adjust the current interval count, use the [RIGHT] or [LEFT] cursor arrow key.
On-time and motion-time intervals increase or decrease by (1) when you press the [RIGHT] or [LEFT] cursor arrow key. Tool-change intervals increase or decrease by (25).
4. Press [ORIGIN] again to stop monitoring the item. The maintenance interval changes into --.

10.3 More Information Online

For detailed maintenance procedures, drawings of machine components, and other useful information, visit the Haas Automation Resource Center at diy.HaasCNC.com. You can also scan this code with your mobile device to go directly to the maintenance information in the Resource Center.



Chapter 11: Other Equipment

11.1 Introduction

Some Haas machines have unique characteristics which are beyond the scope of this manual to describe. These machines come with a printed manual addendum, but you can also download them at www.haascnc.com.

11.2 Office Lathe

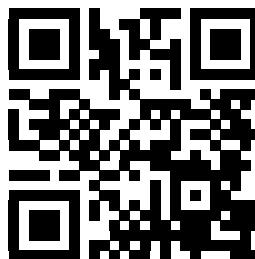
The Office Lathe series are compact small-scale lathes that can fit through a standard door frame and run on single-phase power.

11.3 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.

11.4 More Information Online

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



Index

A

absolute positioning	138
active codes	38
active codes display	
current commands	36
active program	72
advanced editor	110
edit menu	113
modify menu	116
pop-up menu	111
program menu	111
search menu	115
text selection	113
Advanced Tool Management	37
auto door (option)	
override	20
automatic tool offset setting	174
axis motion	
circular	141
linear	141
axis overload timer	103

B

background edit	108
barstock	
safety and	3
basic program example	
completion code blocks	137
cutting code blocks	137
preparation block	136
Basic Programming	135
completion code blocks	137
cutting code blocks	137
preparation	136

basic programming

absolute vs. incremental	138
beacon light	
status	20

C

C-Axis	
jog the	31
c axis	233
calculator	
circle	51
circle-circle tangent	53
circle-line tangent	52
triangle	51
chuck	
installation of	89
removal of	90
safety and	3
chuck foot pedal	86
circular interpolation	141
clipboard	
copy to	114
cut to	114
paste from	115
Collet installation	90
communications	
RS-232	78
control cabinet	
secure latches	2
control display	
active codes	38
active pane	34
basic layout	34
offsets	36
tailstock	38

control pendant	19, 20
detail.....	15
front panel controls	19
USB port.....	20
coolant	
operator override	33
setting 32 and	370
coolant level gauge	39
coolant tank assembly	
detail.....	16
coordinate system	
automatic tool offset setting	174
effective.....	173
FANUC	173
FANUC child coordinate.....	173
FANUC common coordinate	173
FANUC work coordinate	173
global.....	175
YASNAC machine coordinate	173
YASNAC work coordinate	173
coordinate systems	173
copying files.....	73
current commands	36
additional setup.....	85
D	
data collection	78
spare M-codes	80
with RS-232	79
deleting programs	73
Departure move.....	147
device manager.....	70
program selection	72
DIR FULL message.....	74
direct numerical control (DNC).....	81
operating notes	83
display	
graphics	38
settings	38
distance to go position	43
DNC.....	81
doors	
interlocks	2
DPRNT	
DNC and	83

Drawtube	
clamping force adjustment	88
cover plate.....	88
warnings	86
drip mode	83
dry run	103
Dual spindle	237
finding R value.....	240
R phase offset	239
secondary spindle	237
synchronization control display.....	238
synchronized spindle control.....	237
duplicating a program	74
dxf importer	133
chain and group	134
part origin	133
toolpath selection	134

E

edit keys	
ALTER.....	108
DELETE.....	108
INSERT	108
UNDO.....	108
editing	
highlight code	108

F

Features	
axis overload timer	102
background edit	102
dry run	102
Graphics	102
feed hold	
as override.....	33
file directory system	71
directory creation	72
navigation.....	72

file numerical control (FNC).....	81	Intuitive Programming System (IPS)	
display footer	120	dxf importer and.....	133
display line numbers	121		
display modes.....	119		
FNC editor	118		
loading a program.....	118		
menus.....	119		
opening multiple programs	120		
file numerical control (FNC) editor			
text selection	124		
files			
copying	73		
folder, See directory structure			
foot pedals			
chuck	86	alpha keys	29
steady rest	91	cursor keys	24
tailstock	97	display keys	24
jog keys	30	key groups.....	22
mode keys	25	numeric keys.....	28
numeric keys.....	28		
G			
gauges display			
coolant	39	linear interpolation	141
G-codes	243	Live Imaging	
cutting	141	flipped manually.....	184
graphics mode	102	machining.....	182
H		operation.....	181
hazards		program example	176
environmental	4	stock setup	175
help		tool setup	177
calculator	50	Live tooling.....	229
drill table.....	50	cartesian coordinate commands	233
keyword search.....	49	cartesian interpolation example	235
tabbed menu	49	cartesian m-codes	234
help function	48	cartesian programming example	233
High-Pressure Coolant		cartesian to polar	233
HPC	16	cartesian to polar programming	233
I		c-axis	229
icon bar	58	m133/m134/m135 fwd/rev/stop	232
incremental positioning	138	m19 orient spindle	232
input bar	44	mounting and alignment	231
interpolation motion		mounting in turret	231
circular	141	programming notes	230
linear	141		
M			
M30 counters.....	39		
machine			
environmental limits	4		
machine components	13		

machine data	
back up and recover.....	75
backup	76
restore	77
machine position.....	43
machine power-up	69
macro variables	
#3006 programmable stop.....	204
#4001-#4021 last block group codes ...	205
#5001-#5006 last target position	205
#5021-#5026 current machine coordinate po-	
sition	205
#5041-#5046 current work coordinate posi-	
tion	206
#5081-#5086 tool length compensation	206
#6996-#6999 parameter access	206
#8550-#8567 tooling	209
axis position	205
current commands display.....	36
tool offsets	202
macros	
1-bit discrete outputs	201
g- and m-codes	187
look ahead	188
M30 counters and.....	39
round off	188
settings	188
variables	193
main spindle display	47
maintenance	397
current commands	37
manual data input (MDI)	109
material	
fire risk	4
M-codes.....	337
coolant commands.....	140
program stop	140
spindle commands.....	140
memory lock	20
mode display	35
O	
O09xxx program numbers	107
offsets	
displays	36
offsets:	129
operating modes	35
operation	
device manager	70
dry run	103
unattended	4
operator position	43
optional stop.....	341
overrides	33
disabling	33
P	
part setup	85
part zero	101
setting for z axis.....	101
position display	43
axis selection	43
current commands	36
positions	
distance to go.....	43
machine	43
operator	43
work (G54)	43
power on	69
program	
active	72
line numbers removal.....	116
program number	
change	75
program numbers	
change in memory.....	75
O09xxx	107
Onnnnn format	73
program optimizer.....	131
screen	132
program selection	72
programming	
subprograms	185

programs	
.nc file extension	73
basic editing	107
basic search	77
deleting	73
duplication	74
file naming	73
maximum number of	74
running.....	104
transfer.....	72
R	
RS-232.....	78
cable length.....	78
data collection.....	79
DNC and	81
DNC settings	82
running programs.....	104
run-stop-jog-continue.....	104
S	
safety	
decals	8
during operation	2
electrical.....	2
electrical panel.....	2
eye and ear protection	2
hazardous material	2
introduction	1
keyswitch operation	5
part loading/unloading	3
robot cells	6
tool loading/unloading.....	3
safety decals	
other	10
standard layout	8
safety modes	
setup	5
search	
basic.....	77
second home	20
Secondary spindle	
clamping.....	241
m-codes	240
spindle swap	241
secondary spindle programming	240
Servo tailstock	
power failure	94
start up	94
Settings	357
setup mode	
keyswitch	20
shop roles	
machine cleaner	3
spindle load meter	47
ST-20 minimal lube panel	
detail	15
steady rest foot pedal	91
subprograms	185
subprograms, See subprograms	
Synchronized Spindle Control (SSC)	241
T	
tabbed menus	
basic navigation.....	48
Tailstock	180
tailstock	
advance point.....	96
cancel restricted zone	99
foot pedal	97
hold force	93
hold point	96
jogging.....	99
motion	95
programming.....	92, 184, 185
restricted zone	97
resume operation	93
retract point	96
Setting 94 and	98
settings	95
ST-40 servo brake engage	94
ST-40 servo operation.....	93
X-axis clearance plane	98
tailstock display.....	38
text selection	
advanced editor and	113
FNC editor and	124
timers and counters display	39

tips and tricks	
calculator	131
operation	130
programming	128
settings and parameters.....	130
TNC	
approach and departure.....	147
approach move	147
canned cycles.....	150
concept	145
Ex1-standard interpolation.....	150
Ex3-G72 roughing canned cycle	154
Ex4-G73 roughing canned cycle	156
Ex5-G90 modal rough turning cycle.....	157
Ex6-G94 modal rough turning cycle.....	158
G71 roughing.....	153
general.....	143
geometry	161
Imaginary Tool Tip	159
manually calculating.....	160
programming	143
radius wear offset.....	148
tool length.....	149
using.....	146
without	160
tool functions	138
FANUC coordinate system	138
load or change tools.....	139
YASNAC coordinate system	139
tool life display	
current commands	37
tool load limits	85
Tool Nose Compensation	147
tool nose compensation TNC	143
tool offset	84
manual entry	84
manually set.....	84
setting	83
Tool offsets. See Tool offset	
tool turret	
air pressure	99
eccentric locating cam buttons	100
load or change tools.....	101
operations.....	99
protective caps.....	100
touch off the tools	83
U	
unattended operation	
fire risk and	4
USB device	70
W	
work (G54) position.....	43
work offsets.....	207
workholding.....	85
safety and	3
workpiece	
safety	3
X	
x and z axes	
jogging.....	31
x offset to centerline	
Hybrid BOT and VDI	85
setting	85
Y	
y axis	224
jogging.....	31
travel envelope	225
vdi turret and	226
Y-Axis	
operation and programming	226