



Haas Automation, Inc.

Scan Code to get  
the latest version  
of this document



Translation  
Available



# Mill Operator's Manual

Next Generation Control  
96-8210  
Revision T  
March 2023  
English  
Original Instructions

---

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



---

© 2023 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.



This product uses Java Technology from Oracle Corporation and we request that you acknowledge that Oracle owns the Java Trademark and all Java related Trademarks and agree to comply with the trademark guidelines at [www.oracle.com/us/legal/third-party-trademarks/index.html](http://www.oracle.com/us/legal/third-party-trademarks/index.html).

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

---

# LIMITED WARRANTY CERTIFICATE

Haas Automation, Inc.

Covering Haas Automation, Inc. CNC Equipment

Effective September 1, 2010

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

## **Limited Warranty Coverage**

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

## **Repair or Replacement Only**

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

## **Disclaimer of Warranty**

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

---

## **Limits and Exclusions of Warranty**

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

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

## **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](http://www.HaasCNC.com). Use the "Contact Us" link and send your comments to the Customer Advocate.

Join Haas owners online and be a part of the greater CNC community at these sites:

-  [haasparts.com](http://haasparts.com)  
Your Source for Genuine Haas Parts
-  [www.facebook.com/HaasAutomationInc](http://www.facebook.com/HaasAutomationInc)  
Haas Automation on Facebook
-  [www.twitter.com/Haas\\_Automation](http://www.twitter.com/Haas_Automation)  
Follow us on Twitter
-  [www.linkedin.com/company/haas-automation](http://www.linkedin.com/company/haas-automation)  
Haas Automation on LinkedIn
-  [www.youtube.com/user/haasautomation](http://www.youtube.com/user/haasautomation)  
Product videos and information
-  [www.flickr.com/photos/haasautomation](http://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](mailto: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](mailto:customerservice@HaasCNC.com)

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



---

# Declaration of Conformity

Product: Mill (Vertical and Horizontal)\*

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

Manufactured By: Haas Automation, Inc.

2800 Sturgis Road, Oxnard, CA 93030

**805-278-1800**

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

- Machinery Directive 2006/42/EC
- Electromagnetic Compatibility Directive 2014/30/EU
- Low Voltage Directive 2014/35/EC
- Additional Standards:
  - EN 12417:2001+A2:2009
  - EN 60204-1:2018
  - EN ISO 13849-1:2015
  - ISO 10218:1-2:2011 (if robot included)

RoHS2: COMPLIANT (2011/65/EU) by Exemption per producer documentation.

Exempt by:

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

Person authorized to compile technical file:

Kristine De Vries

Phone: +32 (2) 4272151

Address:

Haas Automation Europe  
Mercuriusstraat 28  
B-1930 Zaventem  
Belgium

---

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

- *OSHA 1910.212 - General Requirements for All Machines*
- *ANSI B11.5-1983 (R1994) Drilling, Milling, and Boring Machines*
- *ANSI B11.19-2019 Performance Requirements for Risk Reduction Measures*
- *ANSI B11.23-2002 Safety Requirements for Machining Centers and Automatic Numerically Controlled Milling, Drilling, and Boring Machines*
- *ANSI B11.TR3-2000 Risk Assessment and Risk Reduction - A Guideline to Estimate, Evaluate, and Reduce Risks Associated with Machine Tools*

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

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



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



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

## Original Instructions

---

# User's Operator Manual and other Online Resources

This manual is the operation and programming manual that applies to all Haas Mills.

An English language version of this manual is supplied to all customers and is marked "**Original Instructions**".

For many other areas of the world, there is a translation of this manual marked "**Translation of Original Instructions**".

This manual contains an unsigned version of the EU required "**Declaration Of Conformity**". European customers are provided a signed English version of the Declaration of Conformity with Model Name and Serial Number.

Besides this manual, there is a tremendous amount of additional information online at: [www.haascnc.com](http://www.haascnc.com) under the Service section.

Both this manual and the translations of this manual are available online for machines up to approximately 15 years old.

The CNC control of your machine also contains all of this manual in many languages and can be found by pressing the [**HELP**] button.

Many machine models come with manual supplement that is also available online.

All machine options also have additional information online.

Maintenance and service information is available online.

The online "**Installation Guide**" contains information and check list for Air & Electrical requirements, Optional Mist Extractor, Shipping Dimensions, weight, Lifting Instructions, foundation and placement, etc.

Guidance on proper coolant and Coolant Maintenance is located in the Operators Manual and Online.

Air and Pneumatic diagrams are located on the inside of the lubrication panel door and CNC control door.

Lubrication, grease, oil and hydraulic fluid types are listed on a decal on the machine's lubrication panel.

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

---

## Text Conventions Used in this Manual

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



---

# Contents

User's Operator Manual and other Online Resources . . . . .	xi
How to Use This Manual . . . . .	xii
<b>Chapter 1 Safety . . . . .</b>	<b>1</b>
<b>1.1 General Safety Notes . . . . .</b>	<b>1</b>
<b>1.1.1 Summary of Types of Operation for Haas Automation Machine Tools . . . . .</b>	<b>2</b>
<b>1.1.2 Read Before Operating . . . . .</b>	<b>4</b>
<b>1.1.3 Machine Environmental Limits . . . . .</b>	<b>7</b>
<b>1.1.4 Machine Noise Limits . . . . .</b>	<b>8</b>
<b>1.2 Unattended Operation . . . . .</b>	<b>8</b>
<b>1.3 Door Rules - Run / Setup Mode . . . . .</b>	<b>9</b>
<b>1.3.1 Robot Cells . . . . .</b>	<b>11</b>
<b>1.4 Haas Robot Package . . . . .</b>	<b>11</b>
<b>1.5 Mist Extraction / Enclosure Evacuation . . . . .</b>	<b>13</b>
<b>1.6 Spindle Safety Limit . . . . .</b>	<b>14</b>
<b>1.7 Modifications to the Machine . . . . .</b>	<b>15</b>
<b>1.8 Improper Coolants . . . . .</b>	<b>15</b>
<b>1.9 Safety Decals . . . . .</b>	<b>16</b>
<b>1.9.1 Decal Symbols Reference . . . . .</b>	<b>17</b>
<b>1.9.2 Other Safety Information . . . . .</b>	<b>21</b>
<b>1.9.3 More Information Online . . . . .</b>	<b>21</b>
<b>Chapter 2 Introduction . . . . .</b>	<b>23</b>
<b>2.1 Vertical Mill Overview . . . . .</b>	<b>23</b>
<b>2.2 EC-1600 Overview . . . . .</b>	<b>29</b>
<b>2.2.1 EC-400, EC-400PP Overview . . . . .</b>	<b>32</b>
<b>2.3 Control Pendant . . . . .</b>	<b>36</b>
<b>2.3.1 Pendant Front Panel . . . . .</b>	<b>36</b>
<b>2.3.2 Pendant Right Side, and Top Panels . . . . .</b>	<b>37</b>
<b>2.3.3 Keyboard . . . . .</b>	<b>38</b>
<b>2.3.4 Control Display . . . . .</b>	<b>50</b>
<b>2.4 Tabbed Menu Basic Navigation . . . . .</b>	<b>73</b>
<b>2.5 LCD Touchscreen Overview . . . . .</b>	<b>74</b>
<b>2.5.1 LCD Touchscreen - Navigation Tiles . . . . .</b>	<b>76</b>
<b>2.5.2 LCD Touchscreen - Selectable Boxes . . . . .</b>	<b>78</b>
<b>2.5.3 LCD Touchscreen - Virtual Keyboard . . . . .</b>	<b>80</b>

---

<b>2.5.4</b>	LCD Touchscreen - Program Editing . . . . .	81
<b>2.5.5</b>	LCD Touchscreen - Maintenance . . . . .	82
<b>2.6</b>	<b>Help . . . . .</b>	<b>82</b>
<b>2.6.1</b>	Active Icon Help . . . . .	83
<b>2.6.2</b>	Active Window Help . . . . .	83
<b>2.6.3</b>	Active Window Commands . . . . .	83
<b>2.6.4</b>	Help Index . . . . .	83
<b>2.6.5</b>	More Information Online . . . . .	83
<b>Chapter 3</b>	<b>Control Icons . . . . .</b>	<b>85</b>
<b>3.1</b>	Next Generation Control Icon Guide . . . . .	85
<b>3.2</b>	More Information Online . . . . .	101
<b>Chapter 4</b>	<b>Operation . . . . .</b>	<b>103</b>
<b>4.1</b>	Machine Power-On . . . . .	103
<b>4.2</b>	Spindle Warm-Up . . . . .	104
<b>4.3</b>	Device Manager ([LIST PROGRAM]) . . . . .	104
<b>4.3.1</b>	Device Manager Operation . . . . .	105
<b>4.3.2</b>	File Display Columns . . . . .	106
<b>4.3.3</b>	Create a New Program . . . . .	107
<b>4.3.4</b>	Create A Container . . . . .	108
<b>4.3.5</b>	Select the Active Program . . . . .	109
<b>4.3.6</b>	Check Mark Selection . . . . .	109
<b>4.3.7</b>	Copy Programs . . . . .	109
<b>4.3.8</b>	Edit a Program . . . . .	110
<b>4.3.9</b>	File Commands . . . . .	111
<b>4.4</b>	Screen Capture . . . . .	112
<b>4.5</b>	Error Report . . . . .	112
<b>4.6</b>	Full Machine Backup . . . . .	113
<b>4.6.1</b>	Selected Machine Data Backup . . . . .	116
<b>4.7</b>	Restoring a Full Machine Backup . . . . .	117
<b>4.7.1</b>	Restoring Selected Backups . . . . .	118
<b>4.8</b>	Basic Program Search . . . . .	119
<b>4.9</b>	Locate the Last Program Error . . . . .	120
<b>4.10</b>	Safe Run Mode . . . . .	120
<b>4.11</b>	Tooling . . . . .	123
<b>4.11.1</b>	Tool Holders . . . . .	123
<b>4.11.2</b>	Advanced Tool Management Introduction . . . . .	124
<b>4.11.3</b>	Tool Usage Display . . . . .	130
<b>4.11.4</b>	Tool Info Display . . . . .	132
<b>4.12</b>	WorkHolding Tab- Overview . . . . .	133
<b>4.12.1</b>	E-Vise Setup . . . . .	134
<b>4.12.2</b>	Hydraulic Vise - Setup . . . . .	137

---

<b>4.12.3</b>	Pneumatic Vise - Setup . . . . .	141
<b>4.13</b>	Tool Changers . . . . .	143
<b>4.13.1</b>	Loading the Tool Changer . . . . .	143
<b>4.13.2</b>	Umbrella Tool Changer Recovery . . . . .	150
<b>4.13.3</b>	SMTS Programming Notes . . . . .	150
<b>4.13.4</b>	SMTS Recovery . . . . .	151
<b>4.13.5</b>	SMTS Door Switch Panel . . . . .	152
<b>4.14</b>	Pallet Changer - Introduction . . . . .	153
<b>4.14.1</b>	Pallet Changer Warnings and Cautions . . . . .	153
<b>4.14.2</b>	Maximum Pallet Loads . . . . .	153
<b>4.14.3</b>	Operator Load Station (EC-400) . . . . .	154
<b>4.14.4</b>	Sub-Panel Controls . . . . .	154
<b>4.14.5</b>	Pallet Replacement . . . . .	154
<b>4.14.6</b>	Pallet Storage . . . . .	155
<b>4.14.7</b>	Pallet Schedule Table . . . . .	156
<b>4.14.8</b>	Pallet Pool/Changer Recovery . . . . .	157
<b>4.15</b>	RJH-Touch Overview . . . . .	159
<b>4.15.1</b>	RJH-Touch Operation Mode Menu . . . . .	160
<b>4.15.2</b>	RJH-Touch Manual Jogging . . . . .	161
<b>4.15.3</b>	Tool Offsets with RJH-Touch . . . . .	162
<b>4.15.4</b>	Work Offsets with the RJH-Touch . . . . .	163
<b>4.16</b>	Part Setup . . . . .	164
<b>4.16.1</b>	Jog Mode . . . . .	164
<b>4.16.2</b>	Setting Offsets . . . . .	164
<b>4.17</b>	Run-Stop-Jog-Continue . . . . .	172
<b>4.18</b>	Graphics Mode . . . . .	174
<b>4.19</b>	More Information Online . . . . .	175
<b>Chapter 5</b>	<b>Programming . . . . .</b>	<b>177</b>
<b>5.1</b>	Create / Select Programs for Editing . . . . .	177
<b>5.2</b>	Program Edit Modes . . . . .	177
<b>5.2.1</b>	Basic Program Editing . . . . .	177
<b>5.2.2</b>	Manual Data Input (MDI) . . . . .	180
<b>5.2.3</b>	Background Edit . . . . .	181
<b>5.2.4</b>	Program Editor . . . . .	181
<b>5.3</b>	Basic Programming . . . . .	186
<b>5.3.1</b>	Preparation . . . . .	187
<b>5.3.2</b>	Cutting . . . . .	189
<b>5.3.3</b>	Completion . . . . .	189
<b>5.3.4</b>	Absolute vs. Incremental Positioning (G90, G91) . . . . .	190
<b>5.4</b>	Tool and Work Offset Calls . . . . .	194
<b>5.4.1</b>	G43 Tool Offset . . . . .	194
<b>5.4.2</b>	G54 Work Offsets . . . . .	194

---

<b>5.5</b>	Miscellaneous Codes . . . . .	195
<b>5.5.1</b>	Tool Functions (Tnn) . . . . .	196
<b>5.5.2</b>	Spindle Commands . . . . .	196
<b>5.5.3</b>	Program Stop Commands . . . . .	196
<b>5.5.4</b>	Coolant Commands . . . . .	197
<b>5.6</b>	Cutting G-codes . . . . .	197
<b>5.6.1</b>	Linear Interpolation Motion . . . . .	197
<b>5.6.2</b>	Circular Interpolation Motion. . . . .	198
<b>5.7</b>	Cutter Compensation . . . . .	199
<b>5.7.1</b>	General Description of Cutter Compensation . . . . .	200
<b>5.7.2</b>	Entry and Exit from Cutter Compensation . . . . .	203
<b>5.7.3</b>	Feed Adjustments in Cutter Compensation . . . . .	204
<b>5.7.4</b>	Circular Interpolation and Cutter Compensation. . . . .	206
<b>5.8</b>	Canned Cycles . . . . .	209
<b>5.8.1</b>	Drilling Canned Cycles . . . . .	209
<b>5.8.2</b>	Tapping Canned Cycles . . . . .	209
<b>5.8.3</b>	Boring and Reaming Cycles . . . . .	210
<b>5.8.4</b>	R Planes. . . . .	210
<b>5.9</b>	Special G-codes . . . . .	210
<b>5.9.1</b>	Engraving . . . . .	211
<b>5.9.2</b>	Pocket Milling . . . . .	211
<b>5.9.3</b>	Rotation and Scaling. . . . .	211
<b>5.9.4</b>	Mirror Image . . . . .	211
<b>5.10</b>	Subprograms . . . . .	212
<b>5.10.1</b>	External Subprogram (M98) . . . . .	212
<b>5.10.2</b>	Local Subprogram (M97) . . . . .	215
<b>5.10.3</b>	External Subprogram Canned Cycle Example (M98) . .	216
<b>5.10.4</b>	External Subprograms With Multiple Fixtures (M98) . .	218
<b>5.10.5</b>	Setting Up Search Locations . . . . .	220
<b>5.10.6</b>	More Information Online. . . . .	220
<b>Chapter 6</b>	<b>Options Programming . . . . .</b>	<b>221</b>
<b>6.1</b>	Introduction . . . . .	221
<b>6.2</b>	Feature List . . . . .	221
<b>6.2.1</b>	Purchased Options Enable/Disable . . . . .	222
<b>6.2.2</b>	Option Tryout . . . . .	222
<b>6.3</b>	Rotation and Scaling . . . . .	222
<b>6.4</b>	Visual Programming System (VPS) . . . . .	223
<b>6.4.1</b>	VPS Example . . . . .	224
<b>6.5</b>	Rigid Tapping . . . . .	226
<b>6.6</b>	M19 Spindle Orientation. . . . .	226
<b>6.7</b>	High-Speed Machining . . . . .	226
<b>6.8</b>	Additional Memory Options . . . . .	226

---

<b>6.9</b>	<b>Probing . . . . .</b>	<b>227</b>
<b>6.9.1</b>	Check Tool Probe . . . . .	227
<b>6.9.2</b>	Check Work Probe. . . . .	228
<b>6.9.3</b>	Probe Example . . . . .	229
<b>6.9.4</b>	Probe Use with Macros . . . . .	230
<b>6.9.5</b>	VPS Probe Operations. . . . .	231
<b>6.9.6</b>	Probe Troubleshooting. . . . .	232
<b>6.10</b>	<b>Maximum Spindle Speed . . . . .</b>	<b>232</b>
<b>6.11</b>	<b>Compensation Tables . . . . .</b>	<b>232</b>
<b>6.12</b>	<b>4th and 5th Axis Programming . . . . .</b>	<b>233</b>
<b>6.12.1</b>	New Rotary Configuration . . . . .	233
<b>6.12.2</b>	TCPC/DWO Activation. . . . .	239
<b>6.12.3</b>	Machine Rotary Zero Point (MRZP) . . . . .	240
<b>6.12.4</b>	Creating Five-Axis Programs . . . . .	244
<b>6.12.5</b>	Tilt Axis Center-of-Rotation Offset (Tilting Rotary Products)	
	246	
<b>6.13</b>	<b>Macros (Optional) . . . . .</b>	<b>247</b>
<b>6.13.1</b>	Macros Introduction . . . . .	248
<b>6.13.2</b>	Operation Notes . . . . .	251
<b>6.13.3</b>	Macro Variable Display Page . . . . .	251
<b>6.13.4</b>	Display Macro Variables in the Timers And Counters Window	
	252	
<b>6.13.5</b>	Macro Arguments . . . . .	253
<b>6.13.6</b>	Macro Variables . . . . .	255
<b>6.13.7</b>	Macro Variables Table. . . . .	257
<b>6.13.8</b>	System Variables In-Depth . . . . .	264
<b>6.13.9</b>	Variable Usage . . . . .	275
<b>6.13.10</b>	Address Substitution. . . . .	276
<b>6.13.11</b>	Communication With External Devices - DPRNT[ ] . . . . .	288
<b>6.13.12</b>	G65 Macro Subprogram Call Option (Group 00) . . . . .	291
<b>6.13.13</b>	Aliasing . . . . .	292
<b>6.13.14</b>	More Information Online . . . . .	294
<b>6.14</b>	<b>Pallet Pool M-Codes. . . . .</b>	<b>295</b>
<b>6.14.1</b>	M46 Qn Pmm Jump to Line . . . . .	295
<b>6.14.2</b>	M48 Validate That The Current Program is Appropriate for	
	Loaded Pallet . . . . .	295
<b>6.14.3</b>	M50 Pallet Change Sequence. . . . .	295
<b>6.14.4</b>	M199 Pallet / Part Load or Program End . . . . .	295
<b>Chapter 7</b>	<b>G-codes . . . . .</b>	<b>297</b>
<b>7.1</b>	<b>Introduction . . . . .</b>	<b>297</b>
<b>7.1.1</b>	List of G-codes. . . . .	297

---

<b>Chapter 8</b>	<b>M-codes . . . . .</b>	<b>409</b>
8.1	Introduction . . . . .	409
8.1.1	List of M-codes . . . . .	409
8.1.2	More Information Online . . . . .	437
<b>Chapter 9</b>	<b>Settings . . . . .</b>	<b>439</b>
9.1	Introduction . . . . .	439
9.1.1	List of Settings . . . . .	439
9.2	Network Connection . . . . .	497
9.2.1	Network Icon Guide . . . . .	498
9.2.2	Network Connection Terms and Responsibilities . . . . .	499
9.2.3	Wired Connection Setup . . . . .	500
9.2.4	Wired Network Settings . . . . .	501
9.2.5	Wireless Connection Setup . . . . .	501
9.2.6	Wireless Network Settings . . . . .	504
9.2.7	Net Share Settings . . . . .	504
9.2.8	Haas Drop . . . . .	506
9.2.9	Haas Connect . . . . .	507
9.2.10	Remote Display View . . . . .	507
9.2.11	Machine Data Collection . . . . .	509
9.3	User Positions . . . . .	512
9.4	More Information Online . . . . .	514
<b>Chapter 10</b>	<b>Other Equipment . . . . .</b>	<b>515</b>
10.1	More Information Online . . . . .	515
	<b>Index . . . . .</b>	<b>517</b>

# Chapter 1: Safety

## 1.1 General Safety Notes



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

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



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

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

The work area must be adequately illuminated to allow clear view and safe operation of the machine. This includes the operator work area and all areas of the machine that might be accessed during maintenance or cleaning. Adequate illumination is the responsibility of the user.

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

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

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

## 1.1.1 Summary of Types of Operation for Haas Automation Machine Tools

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

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

Operation of this machine consists of the following:

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

Maintenance only consists of the following:

1. Adding and maintaining condition of coolant
  - Adding coolant and maintaining coolant concentration is required at regular intervals. This is a normal operator function and is either done from a safe location outside of the work enclosure or with the doors open and the machine stopped.

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

Service only consists of the following:

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

## 1.1.2 Read Before Operating



### DANGER:

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

Basic safety:

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

Electrical safety:

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

## Operation Safety:



### DANGER:

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

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



### CAUTION:

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

### Release of person trapped in the machine:

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

### Recover from a jam or blockage:

- Of the chip conveyor - Follow the cleaning instructions on the Haas service site (go to [www.haascnc.com](http://www.haascnc.com) click on the Service tab). If necessary, close the doors and reverse the conveyor so the jammed part or material is accessible, and remove. Use lifting equipment or get assistance for lifting heavy and awkward parts.
- Of a tool and material/part - Close the doors, press [RESET] to clear any displayed alarms. Jog the axis so the tool and material are clear.
- Of the Automatic Tool Changer/tool and spindle - Press [RECOVER] and follow the on-screen instructions.

- If the alarms do not reset or you are unable to clear a blockage, contact your Haas Factory Outlet (HFO) for assistance.

Follow these guidelines when you work with the machine:

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

Periodic inspection of machine safety features:

- Press any of the Emergency Stop buttons to verify all motion is stopped and a 107 EMERGENCY STOP Alarm is displayed.
- A monthly inspection of each Emergency Stop button should be performed.
- Inspect door interlock mechanism for proper fit and function.
- Inspect safety windows and enclosure for damage or leaks.
- Verify all enclosure panels are in place.

Door Safety Interlock inspection:

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

Door Safety Interlock verification:

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

Machine Enclosure and Safety Glass inspection and testing:

Routine Inspection:

- Visually inspect the enclosure and safety glass for any signs of distortion, breakage or other damage.
- Replace the Lexan windows after 7 years or if they are damaged or severely scratched.
- Keep all safety glass and machine windows clean to allow proper viewing of the machine during operations.
- A daily visual inspection of the machine enclosure to verify all panels are in place should be performed.

Testing of machine enclosure:

- No testing of the machine enclosure is necessary.

### 1.1.3 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 (-20.0 °C)	158 °F (70.0 °C)
Ambient Humidity	20% relative, non-condensing	90% relative, non-condensing
Altitude	Sea Level	6,000 ft. (1,829 m)



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

#### Machine with Haas Robot Package

Machine and robot environment is intended to be a machine shop or industrial installation. Shop lighting is the user's responsibility.

## 1.1.4 Machine Noise Limits



**CAUTION:**

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

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

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



**NOTE:**

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

## 1.2 Unattended Operation

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

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

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

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

## 1.3 Door Rules - Run / 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 Run and in Setup mode, the enclosure doors are locked closed during CNC program execution, spindle rotation or axis movement. The doors automatically unlock when the machine is not in cycle. Many machine functions are unavailable with the door open.

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



**CAUTION:** *Haas Robot Package: Care should be taken in Setup mode - Only time Robot can move while cage is open. For more information on the Haas Robot Package refer to the "Haas Robot Package" on page 11*



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

### T1.2: Mill - Run/Setup Mode Restrictions

Machine Function	RUN Mode	SETUP Mode
Air Blast (AAG) On	Not allowed.	Not allowed.
Axis Jog using the Pendant Handle Jog	Not allowed.	Allowed.
Axis Jog using the RJH Handle Jog	Not allowed.	Allowed.
Axis Jog using the RJH shuttle knob	Not allowed.	Not allowed.
Axis Rapid using Home G28 or Second Home	Not allowed.	Not allowed.
Axis Zero Return	Not allowed.	Not allowed.

Machine Function	RUN Mode	SETUP Mode
Automatic Pallet Change	Not allowed.	Not allowed.
APC Operation Buttons	Not allowed.	Not allowed.
Chip Conveyor [CHIP FWD, REV]	Not allowed.	Not allowed.
[COOLANT] button on pendant	Not allowed.	Allowed.
[COOLANT] button on RJH.	Not allowed.	Allowed.
Move Programmable Coolant Spigot	Not allowed.	Allowed.
Orient Spindle	Not allowed.	Not allowed.
Run a program, [CYCLE START] button on the pendant	Not allowed.	Not allowed.
Run a program [CYCLE START] button on the RJH	Not allowed.	Not allowed.
Run a program (Pallet)	Not allowed.	Not allowed.
Spindle [FWD] / [REV] button on the pendant	Not allowed.	Not allowed.
Spindle [FWD] / [REV] on the RJH	Not allowed.	Not allowed.
Tool Change [ATC FWD] / [ATC REV].	Not allowed.	Not allowed.
Tool Release from Spindle	Allowed.	Allowed.
Through Spindle Coolant (TSC) On	Not allowed.	Not allowed.
Tool Air Blast (TAB) On	Not allowed.	Not allowed.

**DANGER:**

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

### 1.3.1 Robot Cells

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

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

## 1.4 Haas Robot Package

The Haas designed work cell made up of a CNC machine and a robot has been evaluated for CE compliance. Changes or variations to the Haas cell design should be re-evaluated for compliance to the applicable standards and are the responsibility of the user/integrator.

Robot is controlled by the CNC it is paired with. It does not respond to any external commands as it could cause hazardous conditions. Do not leave a network connection plugged into the robot controller. No remote control is permitted.

### Run Mode

When in Run mode, unlocking the interlock and opening up the gate stops all motion, and no motion can be started until it is closed and locked.

### Setup Mode

In Set up mode, speed limited motion such as jogging an axis of the robot can be performed in order to program a robot's path of motion, pick up position, set a robot's tool center point, (an offset), load and pick up parts from the spindle or part holder in the machine, etc. The robot program will also be able to be run slowly to prove out the programmed path of the robot.



**NOTE:**

*The max robot speed in setup mode is 7.9 in/sec (200mm/sec).*

### Robot Motion

Manual motion is only permitted in set up mode and is recommended only with limited speed of motion. High speed is permitted but 500mm clearance for the safety of the operator is required. This requires the set up and verification of DCS/keep out zone(s).



**WARNING:**

*Startup of automatic motion with someone in the fenced in area is controlled by the operator. The hazardous area is visible from the remote jog pendant outside of the fenced in area surrounding the robot. Verify nobody is in the fenced in area before starting an automatic operation.*

## **Lighting**

Level of task lighting required for the robot installation is to be provided by the end user. The robot does not require any lighting. But, light will be required for a user to load or unload parts or to perform maintenance, or service.

## **Installation**

The Haas Robot Package installation procedure is located on the website. This procedure describes and verifies the functionality of the enabling device at install, i.e the buttons and how the remote jog handle is intended to be used. Scan the code below with your mobile device to go directly to the procedure.

**F1.1:** Haas Robot Package 1 - VMC Installation



## **Haas Robot - FANUC Dual Check Safety (DCS)**

Each Haas Robot is provided with FANUC's DCS System. This allows the user to define speed and positional limits to the robot. If the robot goes outside of these limits, DCS will stop the robot and remove power to the motors.



**NOTE:**

*Verify proper setup of the DCS zones at the time of robot installation and every 6 months. It should also be verified after a new job has been set up. Scan the code below with your mobile device to go directly to the procedure.*

**F1.2:** Haas Robot - FANUC Dual Check Safety - Setup**Operation**

Recommendation of PPE including but not limited to gloves to protect from sharp edges and chips, eye glasses for eye protection, steel toed shoes etc. when handling parts or loading/unloading parts for the robot to handle.



**WARNING:** *Loss of air pressure may cause the gripper to allow a part being held to slip or be dropped. Users should review grippers that they purchase and determine how they will respond to a loss of power or air pressure, so they know how to minimize any potential hazard.*

The Haas Robot - Quick Start Guide is located on the website. This procedure helps the user set up a job with the Haas Robot. Scan the code below with your mobile device to go directly to the procedure.

**F1.3:** Haas Robot - Quick Start Guide

## 1.5 Mist Extraction / Enclosure Evacuation

Some models have a provision installed that will allow for a mist extractor to be attached to the machine. There is also an optional enclosure exhaust system available that helps keep the mist out of the machine enclosure.

It is entirely up to the owner/operator to determine if and what type of mist extractor is best suited for the application.

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

## 1.6 Spindle Safety Limit

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

### F1.4: Spindle Safety Limit Popup [1]



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

### T1.3: Spindle Maximum Manual Speed Parameter Values

Machine / Spindle Option	Spindle Maximum Manual Speed
Mills	5000
TL	1000
ST-10 through ST-20	2000
ST-30 through ST-35	1500

Machine / Spindle Option	Spindle Maximum Manual Speed
ST-40	750
Live Tool	2000



**NOTE:** *These values can not be changed.*

## 1.7 Modifications to the Machine

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

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

## 1.8 Improper Coolants

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

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

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

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

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



**NOTE:**

*Be sure to maintain your coolant mixture to keep the coolant concentrate at acceptable levels. Improperly maintained coolant mixtures can allow machine components to rust. Rust damage is not covered by your warranty.*

Ask your HFO or your coolant dealer if you have questions about the specific coolant that you plan to use.

## 1.9

# Safety Decals

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



**NOTE:**

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

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

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

## F1.5:

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

1



2



3



## 1.9.1 Decal Symbols Reference

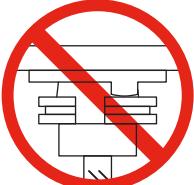
This section gives explanations and clarifications for the safety symbols you will see on your machine.

### T1.4: Hazard Symbols – Yellow Triangles

Symbol	Description
	<p>Moving parts can entangle, trap, crush, and cut. Keep all parts of your body away from machine parts when they move, or whenever motion is possible. Motion is possible when the power is on and the machine is not in <b>[EMERGENCY STOP]</b>. Secure loose clothing, hair, etc. Remember that automatically controlled devices can start at any time.</p>
	<p>Do not touch rotating tools. Keep all parts of your body away from machine parts when they move, or whenever motion is possible. Motion is possible when the power is on and the machine is not in <b>[EMERGENCY STOP]</b>. Sharp tools and chips can easily cut skin.</p>
	<p>The Regen is used by the spindle drive to dissipate excess power and will get hot. Always use care around the Regen.</p>
	<p>There are high voltage components on the machine that can cause electrical shock. Always use care around high voltage components.</p>

Symbol	Description
	<p>There are high voltage components on the machine that can cause electrical shock.</p> <p>Take care to avoid opening electrical enclosure unless components are de-energized or proper personal protective equipment is worn. Arc flash ratings are on the nameplate.</p>
	<p>Long tools are dangerous, especially at spindle speeds higher than 5000 RPM. The tools can break and eject from the machine.</p> <p>Remember that machine enclosures are intended to stop coolant and chips. Enclosures may not stop broken tools or thrown parts.</p> <p>Always check your setup and tooling before you start machining.</p>
	<p>Machining operations can create hazardous chips, dust or mist. This is function of the materials being cut, the metalworking fluid and cutting tools used and the machining speeds/feeds.</p> <p>It is up to the owner/operator of the machine to determine if personal protective equipment such as safety goggles or a respirator is required and also if a mist extraction system is needed.</p> <p>Some models have a provision for connecting a mist extraction system. Always read and understand the Safety Data Sheets (SDS) for the workpiece material, the cutting tools and the metalworking fluid.</p>

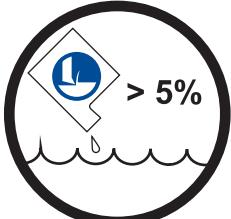
## T1.5: Prohibited Action Symbols – Red Circles with Slash-Through

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

**T1.6:** Recommended Action Symbols – Green Circles

Symbol	Description
	Keep the machine doors closed.
	Always wear safety glasses or goggles when you are near a machine. Airborne debris can cause eye damage. Always wear hearing protection when you are near a machine. Machine noise can exceed 70 dBA.
	Make sure the spindle dogs are correctly aligned with the cutouts in the toolholder V-flange.
	Note the location of the tool release button. Press this button only when you are holding the tool. Some tools are very heavy. Handle these tools carefully; use both hands and have someone press the tool release button for you.

**T1.7:** Informational Symbols – Black Circles

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

## 1.9.2 Other Safety Information

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

## 1.9.3 More Information Online

For updated information, including tips, tricks, maintenance procedures, and more, visit the Haas Service page at [www.HaasCNC.com](http://www.HaasCNC.com).

For the most current Operator’s and Service Manuals scan the code below with your mobile device:



## **Safety Decals**

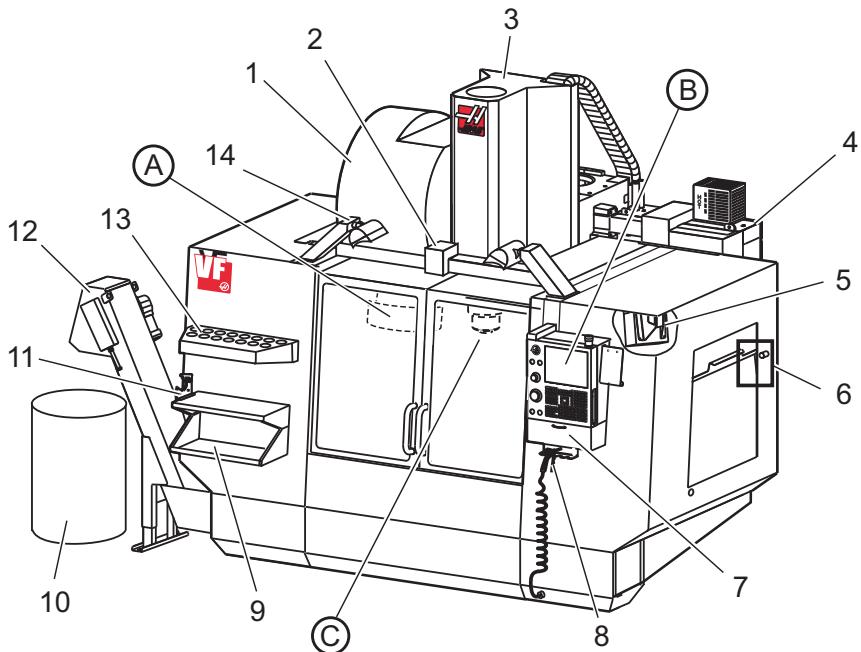
---

# Chapter 2: Introduction

## 2.1 Vertical Mill Overview

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

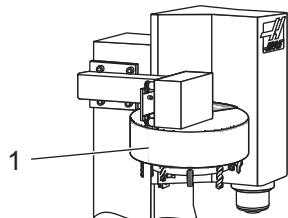
F2.1: Vertical Mill Features (front view)



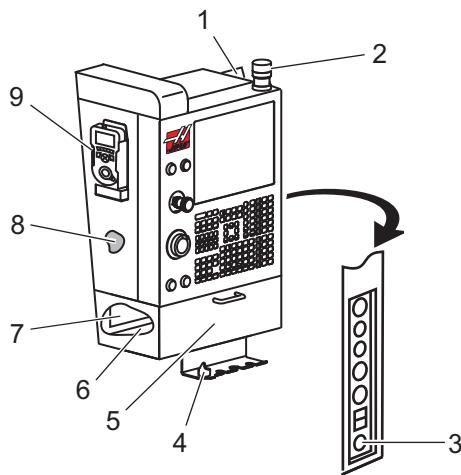
- 1. Side Mount Tool Changer (optional)
  - 2. Auto Door (optional)
  - 3. Spindle Assembly
  - 4. Electrical Control Box
  - 5. Work Light (2X)
  - 6. Window Controls
  - 7. Storage Tray
  - 8. Air Gun
  - 9. Front Work Table
  - 10. Chip Container
  - 11. Tool Holding Vise
  - 12. Chip Conveyor (optional)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

**F2.2:** Detail A

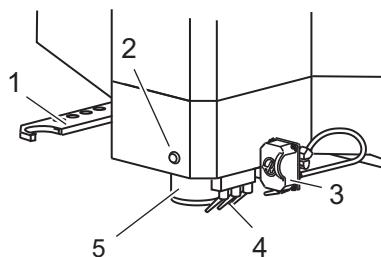
1. Umbrella-Style Tool Changer

**F2.3:** Detail B

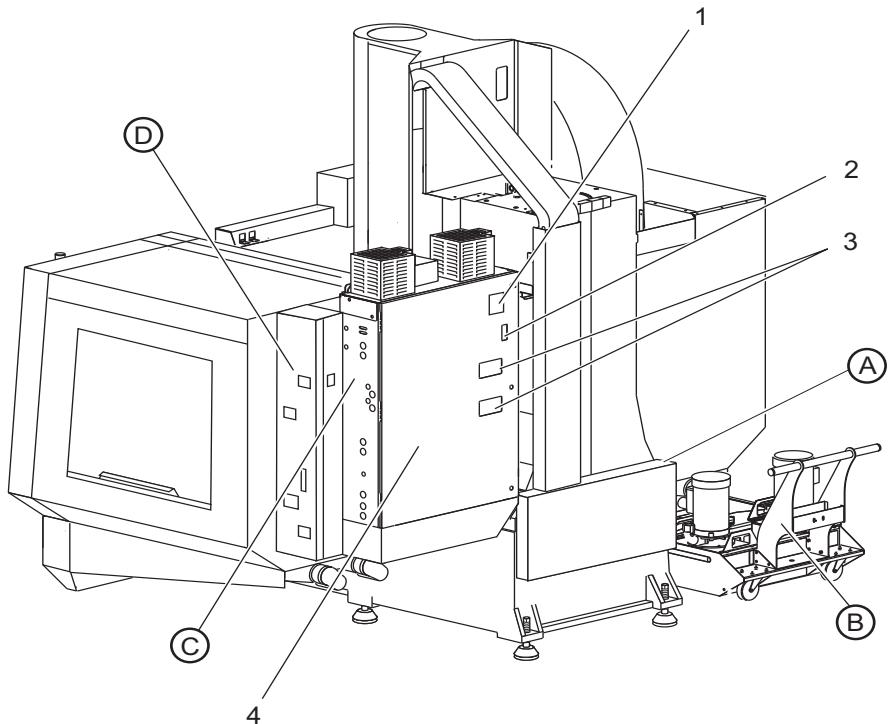
1. Clipboard  
2. Work Beacon  
3. Hold To Run (where equipped)  
4. Vise Handle Holder  
5. Storage Pull Down Access Door  
6. Tool Tray  
7. G- and M-code Reference List  
8. Operator's Manual and Assembly Data (stored inside)  
9. Remote Jog Handle

**F2.4:** Detail C

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

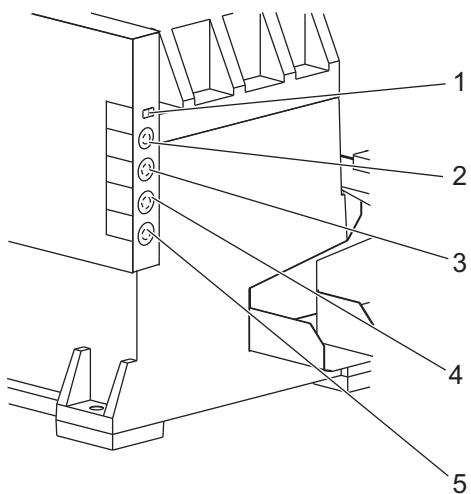


F2.5: Vertical Mill Features (rear view)

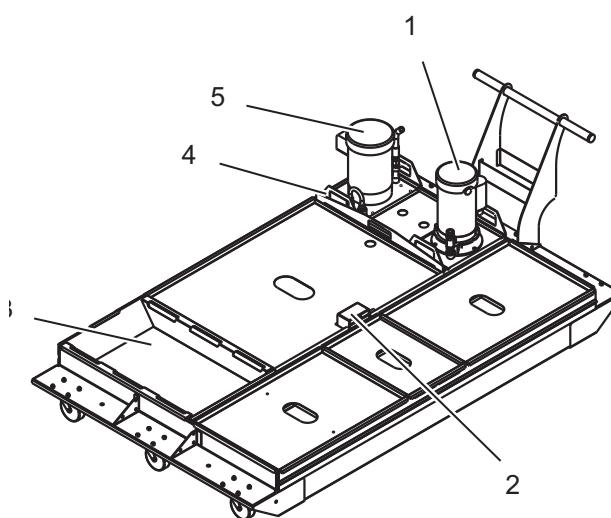


1. Data Plate
2. Main Circuit Breaker Switch
3. Vector Drive Fan (runs intermittently)
4. Control Cabinet

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

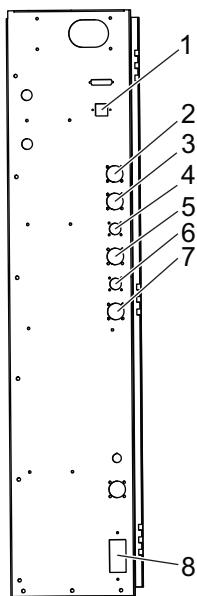
**F2.6:** Detail A - Electrical Connectors

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

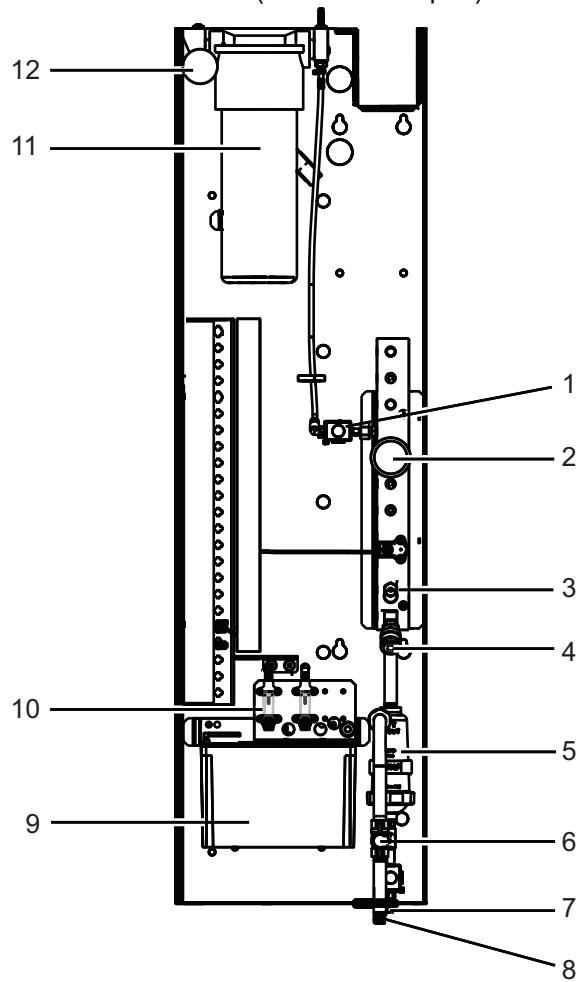
**F2.7:** Detail B

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

**F2.8:** Detail C



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

**F2.9:** Detail D (access door open)

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

**NOTE:**

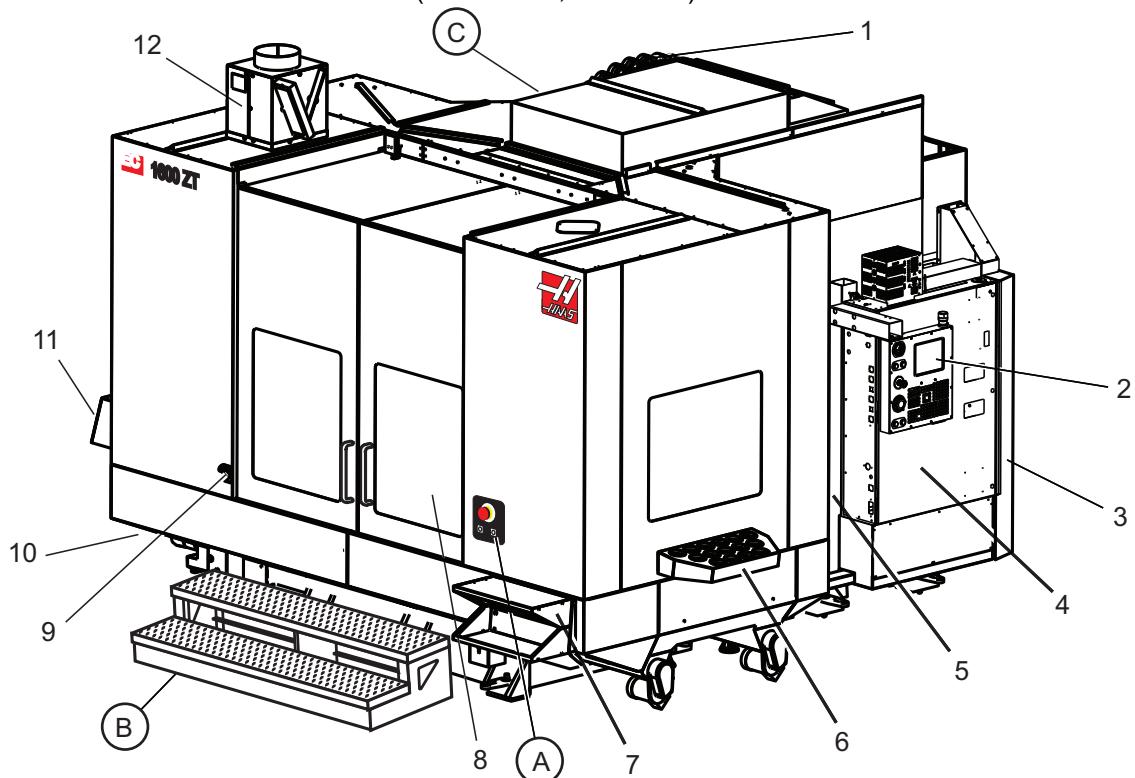
*More details are shown on the decals inside of the access door.*

## 2.2 EC-1600 Overview

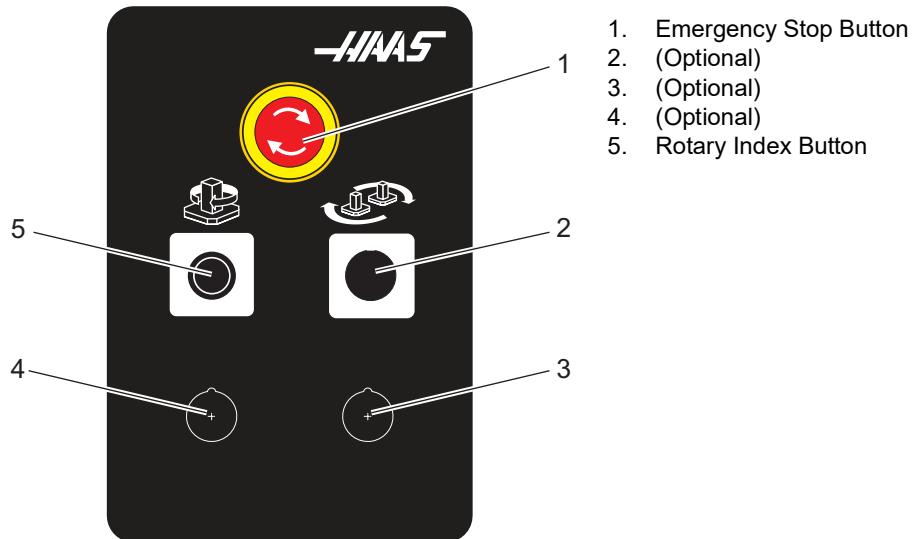
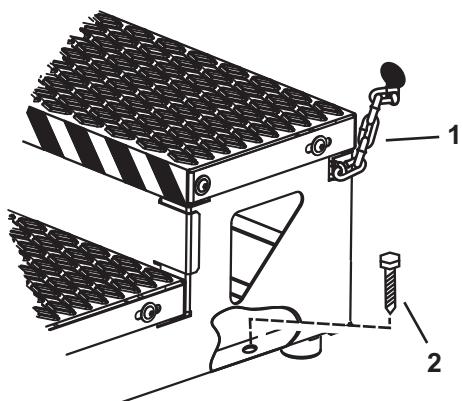
The following figures show some of the standard and optional features of your EC-1600 horizontal mill. Some features are common with the vertical mill.

**NOTE:**

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

**F2.10:** Horizontal Mill Features (EC-1600ZT, front view)

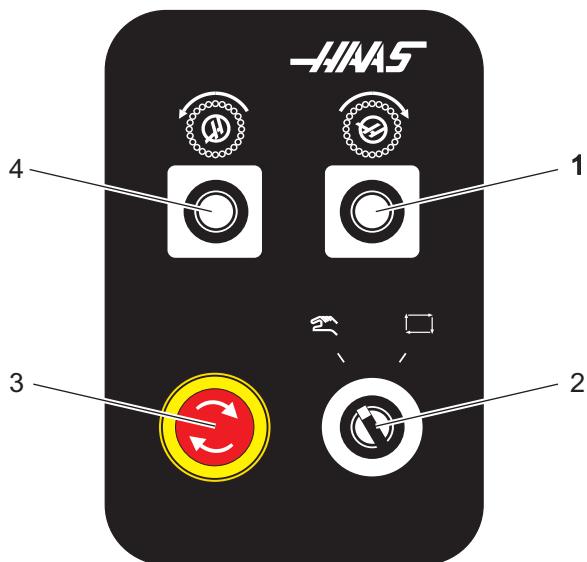
- 1. Side Mount Tool Changer SMT
  - 2. Control Pendant
  - 3. Consolidated Air-Lubrication Module (CALM)
  - 4. Electrical Control Box
  - 5. Operator Spindle Access Door
  - 6. Tool Tray
  - 7. Front Work Table
  - 8. Work Access Doors
  - 9. Air Gun Holder
  - 10. Coolant Tank Assembly (movable)
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

**F2.11:** Detail A**F2.12:** Detail B

1. Chain to Enclosure

2. Floor Anchor Bolt

Secure the work platform with chains to the enclosure or bolts to the floor.

**F2.13:** Detail C

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

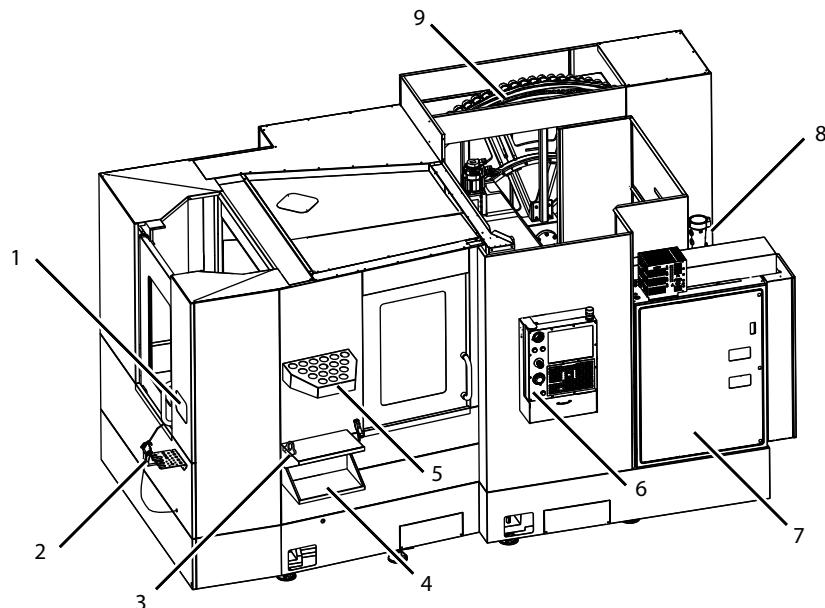
## 2.2.1 EC-400, EC-400PP Overview

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

**NOTE:**

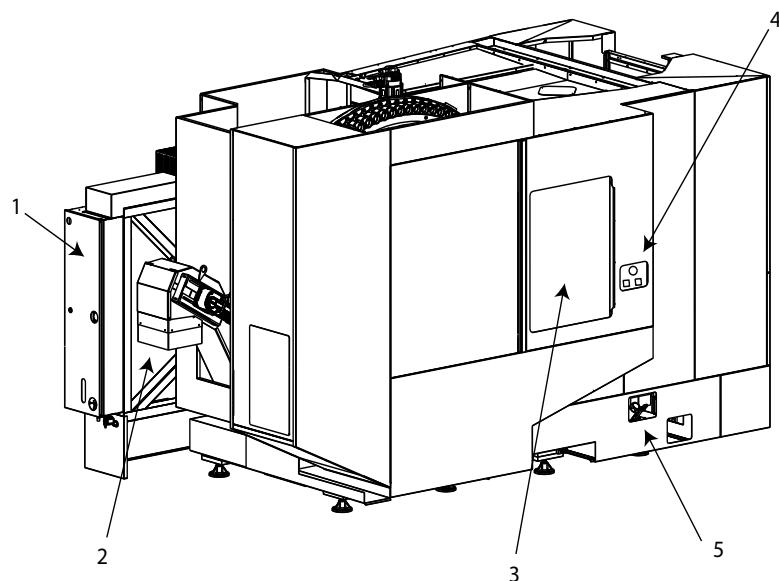
*These figures are representative only; your machine's appearance may vary depending on the model and installed options.*

**F2.14:** Horizontal Mill Features (EC-400, front view)



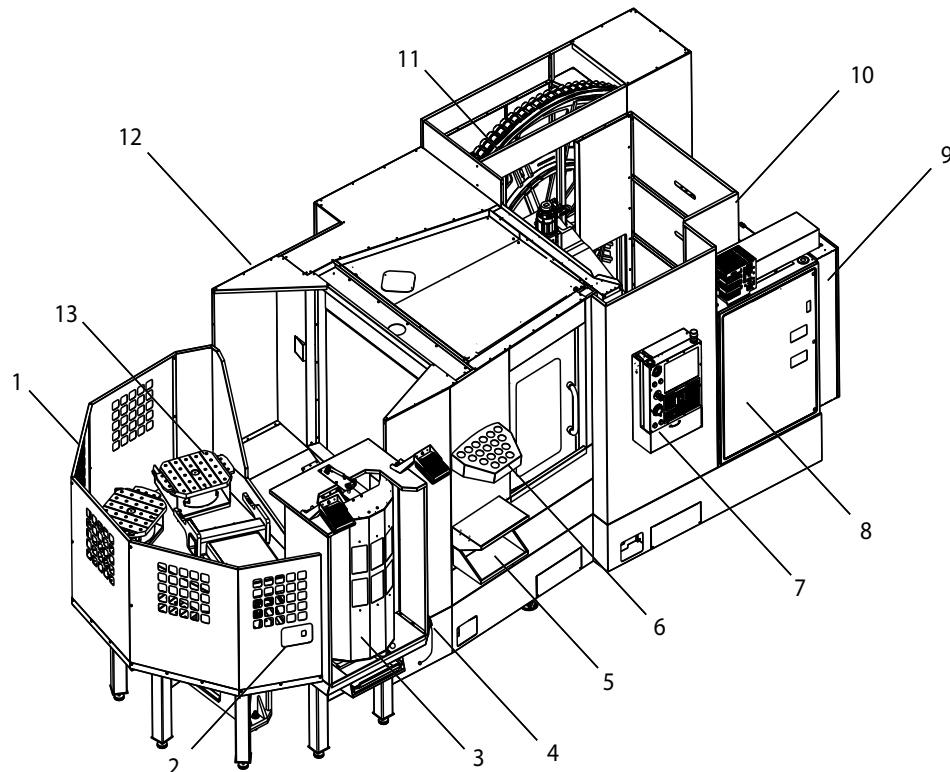
1. Load Station Emergency Stop
2. Air Gun
3. Tool Holding Vise
4. Front Table
5. Tool Crib
6. Control Pendant
7. Electrical Cabinet
8. Coolant Filters
9. Side Mount Tool Changer

**F2.15:** Horizontal Mill Features (EC-400, Rear Left view)



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

**F2.16:** Horizontal Mill Features (EC-400PP)



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

## 2.3 Control Pendant

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

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

### 2.3.1 Pendant Front Panel

#### T2.1: Front Panel Controls

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

## 2.3.2 Pendant Right Side, and Top 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, and offsets.
Setup Mode		In the locked position, this keyswitch enables all machine safety features. Unlock allows setup (refer to "Setup Mode" in the Safety section of this manual for details).
Second Home		Press to rapid all axes to the coordinates specified in settings 268 - 270. (Refer to "Settings 268 - 270" in the Settings section of this manual for details).
Auto Door Override		Press this button to open or close the Auto Door (if equipped).
Worklight		These buttons toggle the internal worklight and High Intensity Lighting (if equipped).

### 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.

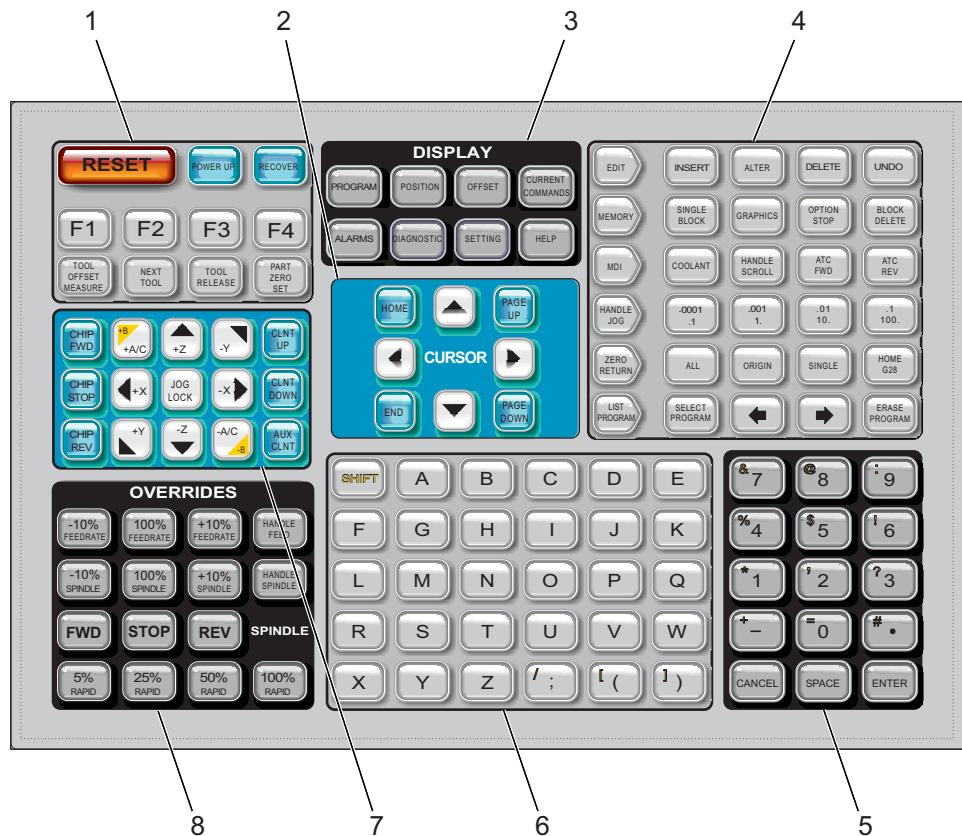
<b>Beacon Light</b>	
Flashing Green	The machine is stopped, but is in a ready state. Operator input is required to continue.
Flashing Red	A fault has occurred, or the machine is in Emergency Stop.
Flashing Yellow	A tool has expired, and the Tool Wear warning icon is shown.

### 2.3.3 Keyboard

Keyboard keys are grouped into these functional areas:

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

**F2.17:** Mill Keyboard: [1] Function Keys, [2] Cursor Keys, [3] Display Keys, [4] Mode Keys, [5] Numeric Keys, [6] Alpha Keys, [7] Jog Keys, [8] Override Keys.



## Function Keys

**T2.4:** List of Function Keys and How They Operate

Name	Key	Function
Reset	[RESET]	Clears alarms. Clears input text. Sets overrides to default values if Setting 88 is ON.
Power up	[POWER UP]	Zero returns all axes and initializes the machine control.
Recover	[RECOVER]	Enters tool changer recovery mode.

Name	Key	Function
F1- F4	[F1 - F4]	These buttons have different functions depending on the tab that is active.
Tool Offset Measure	[TOOL OFFSET MEASURE]	Records tool length offsets during part setup.
Next Tool	[NEXT TOOL]	Selects the next tool from the tool changer.
Tool Release	[TOOL RELEASE]	Releases the tool from the spindle when in MDI, ZERO RETURN, or HAND JOG mode.
Part Zero Set	[PART ZERO SET]	Records work coordinate offsets during part setup.

## Cursor Keys

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

### 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 keys to see the machine displays, operational information, and help pages.

### 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]	Displays the Tool Offset and Work Offset tabbed menu.
Current Commands	[CURRENT COMMANDS]	Displays menus for Devices, Timers, Macros, Active Codes, Calculators, Advanced Tool Management (ATM), Tool Table, and Media.
Alarms	[ALARMS]	Displays the Alarm viewer and Message screens.
Diagnostics	[DIAGNOSTIC]	Displays tabs for Features, Compensation, Diagnostics, and Maintenance.
Settings	[SETTING]	Displays and allows changing of user settings.
Help	[HELP]	Displays help information.

## Mode Keys

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



### NOTE:

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

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

Name	Key	Function
Edit	[EDIT]	Lets you edit programs in the editor. You can access the Visual Programming System (VPS) from the EDIT tabbed menu.
Insert	[INSERT]	Enters text from the input line or the clipboard into the program at the cursor position.
Alter	[ALTER]	Replaces the highlighted command or text with text from the input line or the clipboard.   <b>NOTE:</b> [ALTER] does not work for offsets.
Delete	[DELETE]	Deletes the item that the cursor is on, or deletes a selected program block.
Undo	[UNDO]	Undoes up to the last 40 edit changes, and deselects a highlighted block.   <b>NOTE:</b> [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. You run programs in 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].
Graphics	[GRAPHICS]	Opens Graphics mode.
Optional Stop	[OPTION STOP]	Toggles optional stop on or off. When optional stop is on, the machine stops when it reaches M01 commands.
Block Delete	[BLOCK DELETE]	Toggles Block Delete On or Off. When Block Delete is On, the control ignores (does not execute) the code following a Forward Slash (/), on that same line.

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

Name	Key	Function
Manual Data Input	[MDI]	In MDI mode, you run unsaved programs or blocks of code entered from the control. Shows <i>EDIT:MDI</i> in upper left display.
Coolant	[COOLANT]	Turns the optional coolant on and off. Also, [SHIFT] + [COOLANT] turns on and off the optional Auto Air Gun / Minimum Quantity Lubrication functions.
Handle Scroll	[HANDLE SCROLL]	Toggles Handle Scroll mode. This lets you use the jog handle to move the cursor in menus while the control is in jog mode.
Automatic Tool Changer Forward	[ATC FWD]	Rotates the tool carousel to the next tool.
Automatic Tool Changer Reverse	[ATC REV]	Rotates the tool carousel to the previous tool.

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

Name	Key	Function
Handle Jog	[HANDLE JOG]	Enters Jog mode.
.0001/.1 .001/1 .01/10 .1/100	[.0001 /.1], [.001 / 1.], [.01 / 10.], [.1 / 100.]	Selects the increment for each click of the jog handle. When the mill is in MM mode the first number is multiplied by ten when jogging the axis (e.g., .0001 becomes 0.001 mm). The bottom number sets speed after you press [JOG LOCK] and an axis jog key or you press and hold an axis jog key. Shows <i>SETUP:JOG</i> in the upper left of the 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: Operator, Work G54, Machine, and Dist (distance) To Go. Select the tab to switch between the categories. Shows <i>SETUP: ZERO</i> in the upper-left display.
All	[ALL]	Returns all axes to machine zero. This is similar to [POWER UP], except a tool change does not occur.
Origin	[ORIGIN]	Sets selected values to zero.
Single	[SINGLE]	Returns one axis to machine zero. Press the desired axis letter on the Alpha keyboard and then press [SINGLE].
Home G28	[HOME G28]	Returns all axes to zero in rapid motion. [HOME G28] will also home a single axis in the same manner as [SINGLE].
		 <b>CAUTION:</b> <i>Make sure the axis motion paths are clear when you press this key. There is no warning or prompt before axis motion begins.</i>

## 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.
Select Programs	[SELECT PROGRAM]	Makes the highlighted program the active program.
Back	[BACK ARROW],	Navigates to the screen you were on before the current one. This key operates like the BACK button on a web browser.

Name	Key	Function
Forward	<b>[FORWARD ARROW]</b>	Navigates to the screen you went to after the current screen, if you have used the back arrow. This key operates like the FORWARD button on a web browser.
Erase Program	<b>[ERASE PROGRAM]</b>	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	<b>[0]-[9]</b>	Types numbers.
Minus sign	<b>[ - ]</b>	Adds a minus (-) sign to the input line.
Decimal point	<b>[ . ]</b>	Adds a decimal point to the input line.
Cancel	<b>[CANCEL]</b>	Deletes the last character typed.
Space	<b>[SPACE]</b>	Adds a space to input.
Enter	<b>[ENTER]</b>	Answers prompts and writes input.
Special Characters	Press <b>[SHIFT]</b> , 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.
+	<b>[SHIFT], then [ - ]</b>	Inserts +
=	<b>[SHIFT], then [0]</b>	Inserts =
#	<b>[SHIFT], then [ . ]</b>	Inserts #
*	<b>[SHIFT], then [1]</b>	Inserts *
'	<b>[SHIFT], then [2]</b>	Inserts '

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

Name	Key	Function
Left Bracket	[SHIFT], then [(	Inserts [
Right Bracket	[SHIFT], then ()]	Inserts ]

## Jog Keys

**T2.15:** List of Jog Keys and How They Operate

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

## Override Keys

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

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

## Override Usage

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

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

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

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

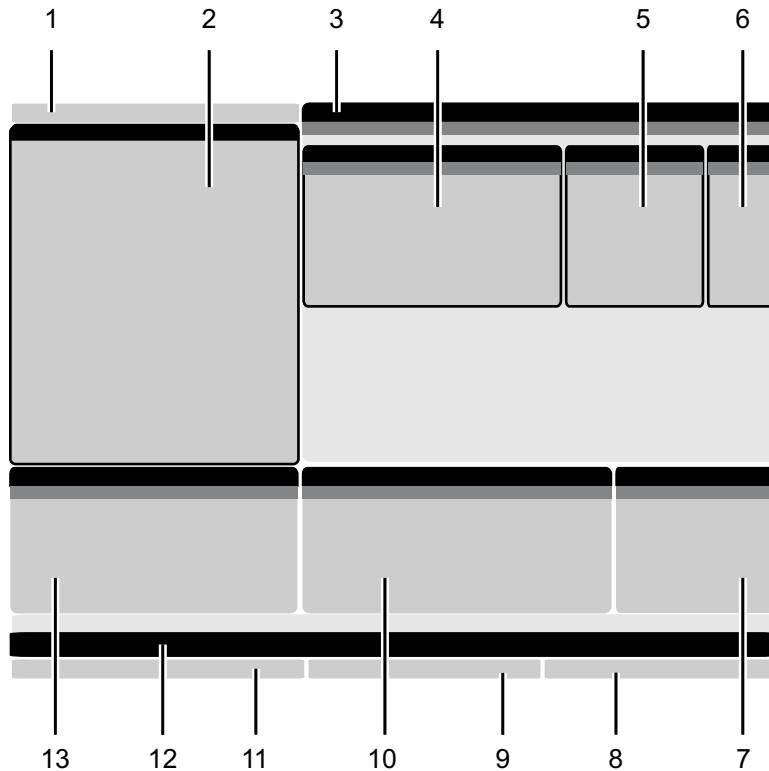
You can override the standard coolant setting by pressing **[COOLANT]**. The coolant pump remains either on or off until the next M-code or operator action (see Setting 32).

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

## 2.3.4 Control Display

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

F2.18: Basic Control Display Layout in **Operation:Mem** Mode (While a Program Runs)



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

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

## Mode, Network, and Time Status Bar

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

- F2.19:** The Mode, Network, and Time Status bar shows [1] the current machine mode, [2] network status icons, and [3] the current time.



### Mode [1]

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

- T2.17:** Mode, Key Access, and Mode Display

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

### Network [2]

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

**T2.18:** Networking Icons and Associated Network Status

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

### Time [3]

The right side of the bar shows the current time in hh:mm:ss format. To set the time, refer to the Time Adjustment section on page 54.

## Offsets Display

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

### T2.19: Offset Tables

Name	Function
<b>TOOL</b>	Display and work with tool numbers and tool length geometry.
<b>WORK</b>	Display and work with part zero locations.

## Current Commands

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

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

**Timers Display** -This page shows:

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

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

**Macros Display** -This page shows a list of the macro variables and their values. The control updates these variables as programs run. You can modify the variables in this display; refer to the Variable Display Page on page 251.

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

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

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

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

**Media** -This page contains the **Media Player**.

## Timer and Counter Reset

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

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



**TIP:**

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

## Time Adjustment

Follow this procedure to adjust the date or time.

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

## Current Commands - Active Codes

### F2.20: Active Codes Display Example

Current Commands						
Devices	Timers	Macro Vars	Active Codes	ATM	Calculator	Media
G-Codes	Address Codes		DHMT Codes	Speeds & Feeds		
G00	N	0	D 00	Programmed Feed Rate	0.	
G18	X	0.	H 00	Actual Feed Rate	0.	
G90	Y	0.	M 00	Programmed Spindle Speed	0.	
G113	Z	0.	T 00	Commanded Spindle Speed	0.	
G20	I	0.		Actual Spindle Speed	0.	
G40	J	0.		Coolant Spigot Position		
G49	K	0.				
G80	P	0				
G99	Q	0.				
G50	R	0.				
G54	O	000000				
G97	A	0.				
G64	B	0.				
G69	C	0.				
	U	0.				
	V	0.				
	W	0.				
	E	0.				

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

## Advance Tool Management (ATM)

### F2.21: Advance Tool Management Display Example

07:03:51 N0 3.9);

Current Commands										
	Devices	Timers	Macro Vars	Active Codes	ATM	Calculator	Media			
<b>F4</b>	To Switch Boxes				Allowed Limits			Active Tool: 1		
	Group	Expired Count	Tool Order	Holes Limit	Usage Limit	Life Warn %	Load Limit	Expired Action	Feed Limit	Total Time Limit
All	-	-	-	-	-	-	-	-	-	
Expired	0	-	-	-	-	-	-	-	-	
No Group	-	-	-	-	-	-	-	-	-	
Add Group	-	-	-	-	-	-	-	-	-	

Tool Data For Group: All									
Tool	Offset	Life	Holes Count	Usage Count	Usage Limit	Max Load %	Load Limit %	Feed Time	Total Time
1	0	100%	0	0	0	0%	0%	0:00:00	0:00:00
2	0	100%	0	0	0	0%	0%	0:00:00	0:00:00
3	0	100%	0	0	0	0%	0%	0:00:00	0:00:00
4	0	100%	0	0	0	0%	0%	0:00:00	0:00:00
5	0	100%	0	0	0	0%	0%	0:00:00	0:00:00
6	0	100%	0	0	0	0%	0%	0:00:00	0:00:00

**INSERT** Add Group

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

For more information, refer to:

- AdvanceTool Management Introduction
- AdvanceTool Management Macros
- SaveAdvance Tool Management Tables
- RestoreAdvance Tool Management Tables

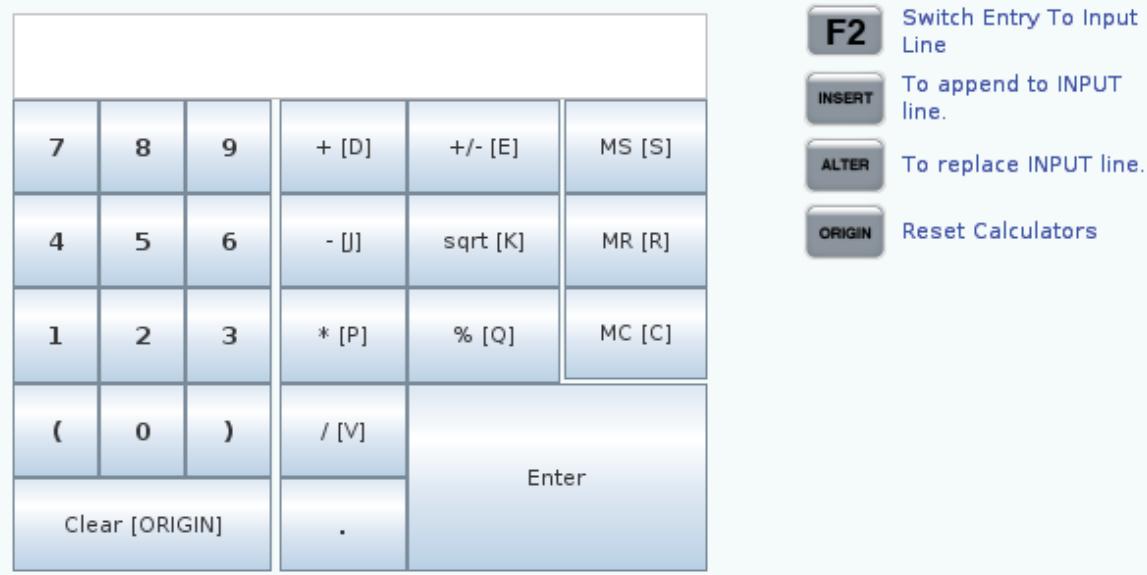
## Calculator

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

- Select the calculator tab in the **[CURRENT COMMANDS]** menu.
- Select the calculator tab that you want to use: **Standard, Milling, or Tapping**.

## Standard Calculator

### F2.22: Standard Calculator Display



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

- Use the number keys to type operands into the calculator.
- To insert an arithmetic operator, use the letter key that appears in brackets next to the operator you want to insert. These keys are:

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

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

**NOTE:**

*These options are available for all calculators.*

Press [ENTER] to return the result of your calculation.

Press [INSERT] to append the data or the result to the end of the input line.

Press [ALTER] to move the data or the result to the input line. This overwrites the current contents of the input line.

Press [ORIGIN] to reset the calculator.

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

### Milling/Turning Calculator

#### F2.23: Milling/turning Calculator Display

Cutter Diameter	*****.****	in	<b>F2</b> Switch Entry To Input Line
Surface Speed	*****.****	ft/min	<b>INSERT</b> To append to INPUT line.
RPM	*****.****		<b>ALTER</b> To replace INPUT line.
Flutes	*****.****		<b>DELETE</b> Clear current input
Feed	*****.****	in/min	<b>ORIGIN</b> Reset Calculators
Chip Load	*****.****	in/tth	
Work Material	◀ ▶ No Material Selected		<b>F3</b> Copy Value From Standard Calculator
Tool Material	◀ ▶ Please Select Work Material		<b>F4</b> Paste Current Value To Standard Calculator
Cut Width	*****.****	in	
Cut Depth	*****.****	in	

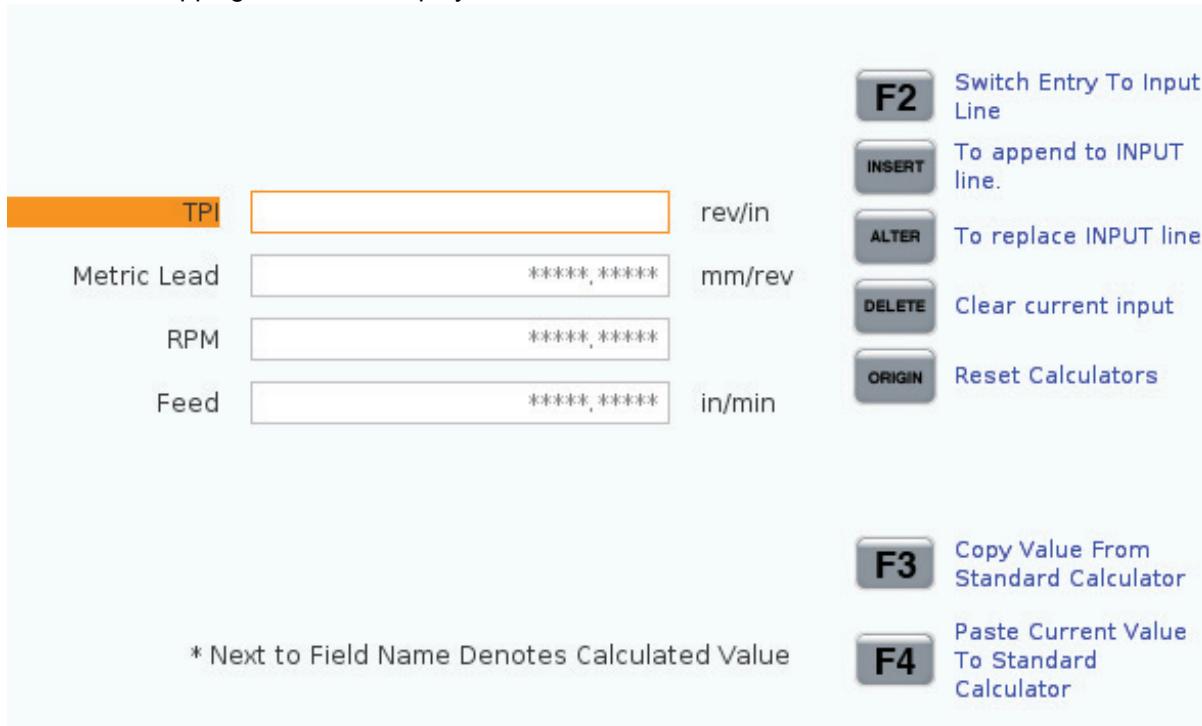
Enter a value from 0 - 1000.0000  
 \* Next to Field Name Denotes Calculated Value

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

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

### Tapping Calculator

**F2.24:** Tapping Calculator Display



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

- Use the cursor arrow keys to move from field to field.

- Type known values in the appropriate fields. You can also press [F3] to copy a value from the standard calculator.
- When the calculator has enough information, it puts calculated values in the appropriate fields.

## Media Display

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

- Providing visual cues or work instructions during program operation
- Providing images to aid part inspection at certain points in a program
- Demonstrating procedures with video

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

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



**NOTE:**

*M130 uses the subprogram search settings, Settings 251 and 252 in the same way that M98 does. You can also use the Insert Media File command in the editor to easily insert an M130 code that includes the file path. Refer to page 183 for more information.*

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

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

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

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

**T2.20:** Permitted media file formats

Standard	Profile	Resolution	Bitrate
MPEG-2	Main-High	1080 i/p, 30 fps	50 Mbps
MPEG-4 / XviD	SP/ASP	1080 i/p, 30 fps	40 Mbps

Standard	Profile	Resolution	Bitrate
H.263	P0/P3	16 CIF, 30fps	50 Mbps
DivX	3/4/5/6	1080 i/p, 30fps	40 Mbps
Baseline	8192 x 8192	120 Mpixel/sec	-
PNG	-	-	-
JPEG	-	-	-

**NOTE:**

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

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

**F2.25:** Media Display Example - Work Video Instruction during a Program



**T2.21:** Troubleshooting

Symptom	Possible Cause	Corrective Action
Error <i>Encrypted PDF</i> is generated.	The PDF file is encrypted.	The Haas control does not support encryption or passwords. Decrypt or remove password protection for the PDF on a computer.
Error <i>File not found</i> is generated.	The File could not be found at the location specified.	Verify that the file is where you expect it to be. Use the M130 path generator in the Editor to verify that the path specified in your program is correct.

Symptom	Possible Cause	Corrective Action
Error <i>Invalid File</i> is generated.	The control had a problem identifying or interacting with the file.	If the PDF is on USB or Netshare try copying it to the My Media folder on your control inside User Data and test opening it from there. If you already copied it onto the local control verify the file size or try to copy it to another place in the control to make sure the file was not corrupted. Verify that the PDF can be opened on your computer.
Error <i>Unable to Render</i> is generated	The control could read the image but it did not have the available space to display it.	Sometimes this message is generated when the control is busy performing other actions in the background. Try the following things: "Change pages in the template" "Stop running a program (if possible)" "Reload the template" "Zoom out" if none of those steps worked the PDF page you are displaying might be too large for the control to process. Try editing the specific page to a smaller size on a computer before loading it again.

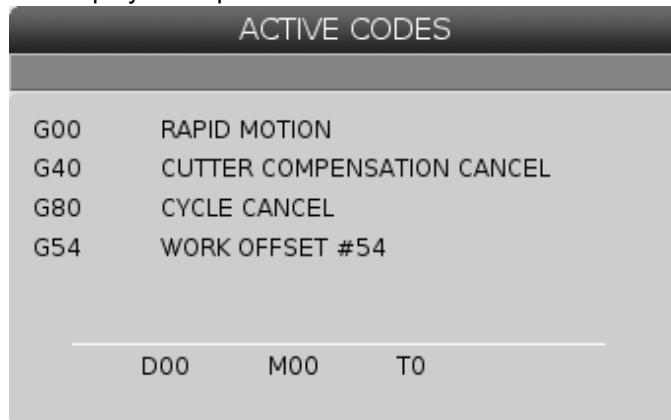
## Settings/Graphics Display Function

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

To use Graphics mode, select the **GRAPHICS** tab. Graphics shows an on-screen representation of your part program. The axes do not move, so you do not risk tool or part damage from programming errors.

### Active Codes

F2.26: 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 M-code. If an alarm is active, this shows a quick display of the active alarm instead of the active codes.

## Active Tool

F2.27: Active Tool Display Example

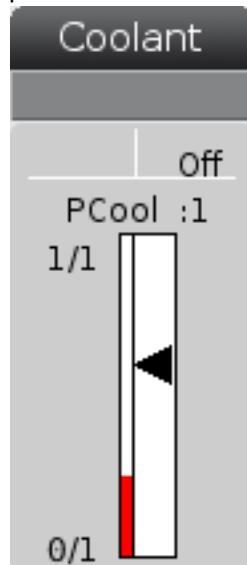


This display gives information about the current tool in the spindle. This information includes:

- The tool number
- The offset number
- The type of tool (if specified in the tool offsets table)
- The tool group number (if specified in the ATM table)
- Maximum tool load (the highest load, in percent, that has been put on the tool)
- The remaining percentage of tool life or tool group
- An example image of the tool type (if specified)
- The next tool pocket number and the tool number currently in that pocket

## Coolant Display

F2.28: Coolant Level Display Example



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

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

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

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

## Timers & Counters Display

**F2.29:** Timers & Counters Display Example

Timers And Counters	
This Cycle:	0:00:00
Last Cycle:	0:00:00
Remaining	0:00:00
M30 Counter #1:	0
M30 Counter #2:	0
Loops Remaining:	0

The timer section of this display gives information about cycle times (This Cycle, Last Cycle, and Remaining).

The counter section has two M30 counters and 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 **54** for information on how to reset the timers and counters.
- Loops Remaining: shows the number of subprogram loops remaining to complete the current cycle.

## Alarms and Messages Display

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

Press **[ALARMS]**, then select a display tab:

- The **ACTION ALARM** tab shows the alarms that currently affect machine operation. Use **[PAGE UP]** and **[PAGE DOWN]** to see the other active alarms.
- The **MESSAGES** tab shows the messages page. The text you put on this page stays there when you power the machine off. You can use this to leave messages and information for the next machine operator, etc.
- The **ALARM HISTORY** tab shows a list of the alarms that have recently affected machine operation. You can also search for an alarm number or alarm text. To do this type in the alarm number or the desired text and press **[F1]**.

- The **ALARM VIEWER** tab shows a detailed description of all the alarms. You can also search for an alarm number or alarm text. To do this type in the alarm number or the desired text and press **[F1]**.
- The **KEY HISTORY** tab shows up to the last 2000 keystrokes.

## Add Messages

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

1. Press **[ALARMS]**, select the **MESSAGES** tab, and press the **[DOWN]** cursor arrow key.
2. Type your message.

Press **[CANCEL]** to backspace and delete. Press **[DELETE]** to delete an entire line. Press **[ERASE PROGRAM]** to delete the entire message.

## Alarm Notifications

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

To set up alarm alerts, press **[SETTING]** and choose the **NOTIFICATIONS** tab.

## 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

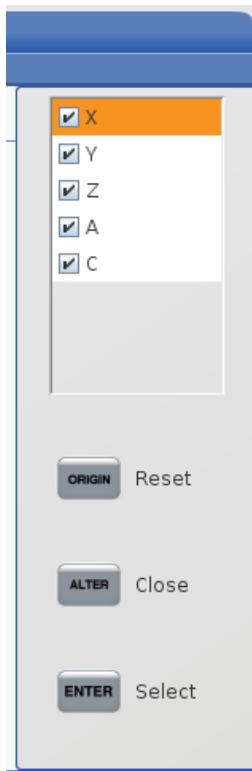
The Position display shows the current axis position relative to four reference points (Work, Distance-to-go, Machine, and Operator). In any mode, press [**POSITION**] and use cursor keys to access the different reference points displayed in tabs. The last tab display shows all the reference points on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

### T2.22: Axis Position Reference Points

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

### Axis Display Selection

You can add or remove axes in the Positions displays. While a **Positions** display tab is active, press [**ALTER**]. The axis display selection window comes in from the right side of the screen.

**F2.30:** Axis Display Selector

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

**NOTE:**

*You can display a maximum of (5) axes.*

## Input Bar

**F2.31:** 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.

## Special Symbol Input

Some special symbols are not on the keypad.

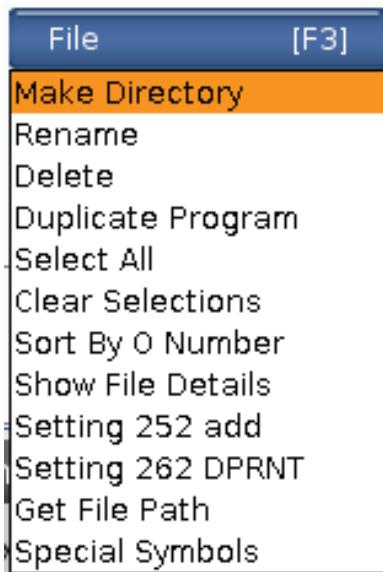
### T2.23: Special Symbols

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

Do these steps to enter special symbols:

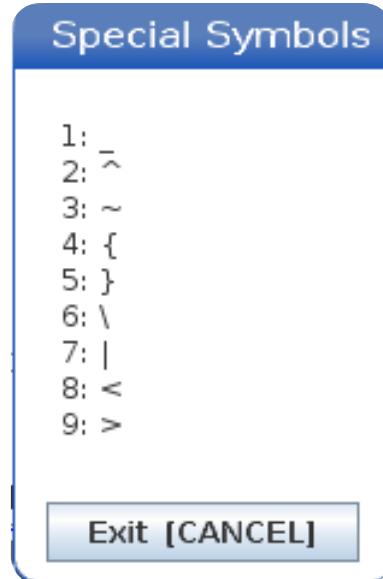
1. Press [LIST PROGRAMS] and select a storage device.
2. Press [F3].

The [FILE] drop-down menu shows:



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

The **SPECIAL SYMBOLS** pick list shows:



4. Enter a number to copy the associated symbol to the **INPUT:** bar.

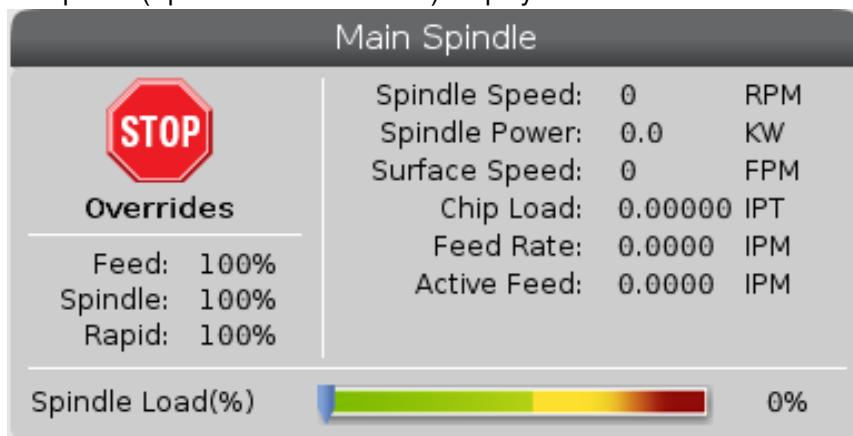
For example, to change a directory's name to MY\_DIRECTORY:

1. Highlight the directory with the name that you want to change.
2. Type MY.

3. Press [F3].
4. Select **SPECIAL SYMBOLS** and press [**ENTER**].
5. Press [1].
6. Type DIRECTORY.
7. Press [F3].
8. Select **RENAME** and press [**ENTER**].

## Main Spindle Display

**F2.32:** Main Spindle (Speed and Feed Status) Display



The first column of this display gives you information about feedrate, spindle, and rapid overrides.

The second column displays the current spindle speed in rpm and spindle load in kW. The spindle load value reflects the actual spindle power to the tool. The next values presented are linked: surface speed of the rotating tool in fpm, the actual chip load in in/tth, and the programmed feed rate in in/min. Active feed rate displays the actual feed rate including any manual overrides.

The spindle load meter indicates the spindle load as a percentage of motor capacity.

## 2.4 Tabbed Menu Basic Navigation

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

1. Press a display or mode key.  
The first time you access a tabbed menu, the first tab (or sub-tab) is active. The highlight cursor is at the first available option in the tab.
2. Use the cursor keys or the [**HANDLE JOG**] control to move the highlight cursor within an active tab.

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



**NOTE:**

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

The current tab becomes inactive.

4. Use the cursor keys to highlight a tab or a sub-tab, and press the [DOWN] cursor arrow key to use the tab.



**NOTE:**

*You cannot make the tabs active in the POSITIONS tabbed display.*

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

## 2.5 LCD Touchscreen Overview

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



**NOTE:**

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

**T2.24:** Touchscreen Settings

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

- F2.33:** Touchscreen status icons - [1] Software does not support Touchscreen [2] Touchscreen is Disabled, [3] Touchscreen is Enabled.



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

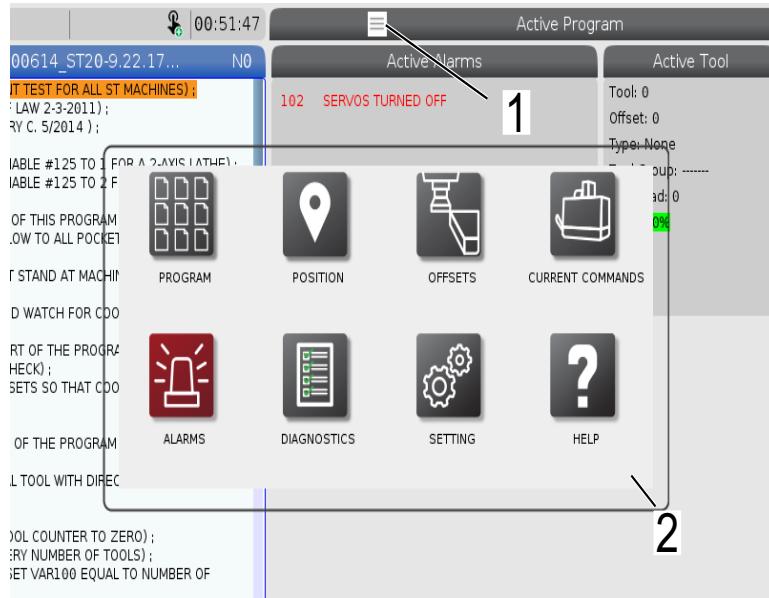
- T2.25:** Functions excluded from Touchscreen

Functions	Touchscreen
[RESET]	Not available
[EMERGENCY STOP]	Not available
[CYCLE START]	Not available
[FEED HOLD]	Not available

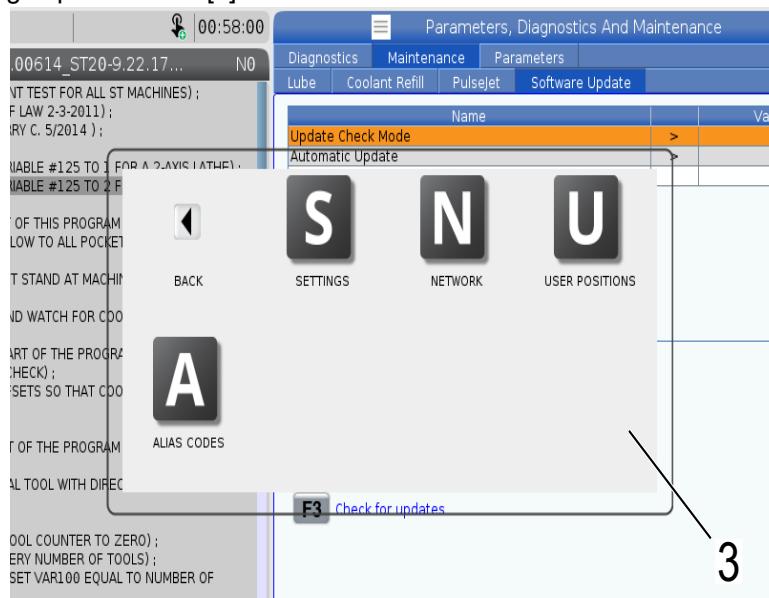
## 2.5.1 LCD Touchscreen - Navigation Tiles

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

**F2.34:** [1] Menu Panel Icon, [2] Display Icons.

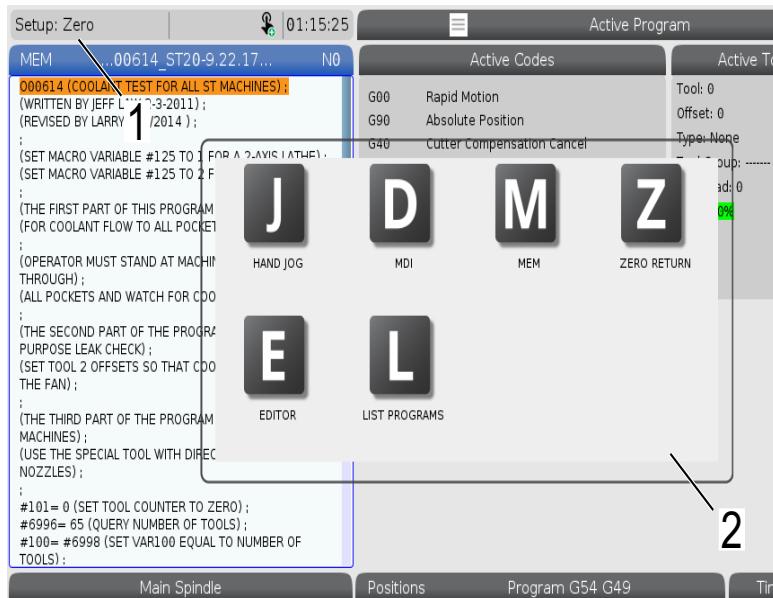


**F2.35:** Settings options Icons [3].



- Press and hold the display icon, to navigate to a specific tab. For example if you want to go to the Network page, press and hold the [SETTINGS] icon until the settings options [3] are shown.
- Press the back icon to go back to the main menu.
- To close the popup box touch anywhere else outside the popup box.

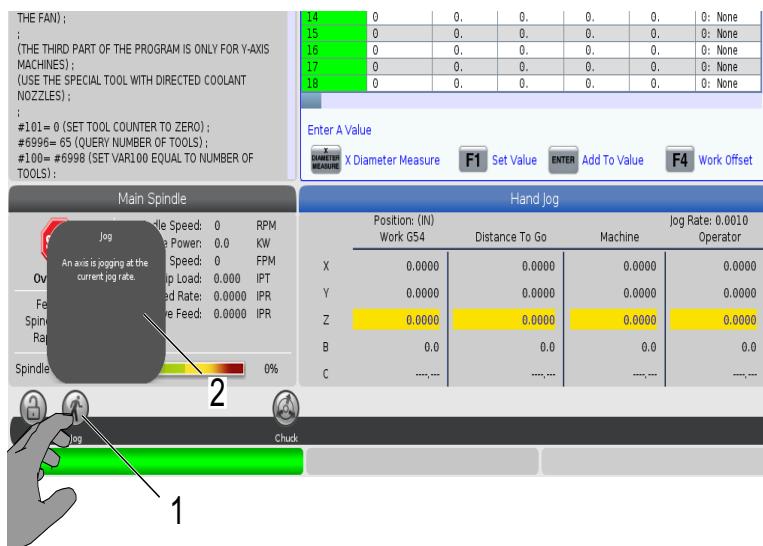
#### F2.36: Operation Mode Panel



- Press the upper left corner [1] of the screen to have the operation mode panel popup box [2] to appear. Press the mode icon to put the machine in that mode.

## 2.5.2 LCD Touchscreen - Selectable Boxes

### F2.37: Icon Help



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

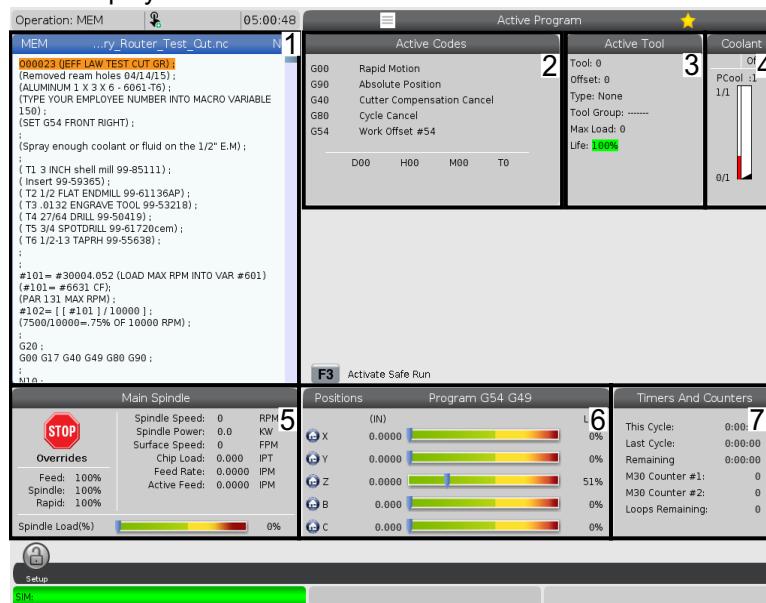
### F2.38: Selectable tables and function buttons.

The screenshot shows a 'Offsets' table window. A hand icon [1] is shown touching the table, with callout arrows pointing to specific rows [2]. The table has columns for Tool Offset, Length Geometry(H), Length Wear(H), Diameter Geometry(D), and Diameter Wear(D). The rows are numbered 1 through 18. At the bottom of the table, there is an 'Enter A Value' input field and function keys: F1 Tool Offset Measure, F2 Set Value, F3 Enter, F4 Add To Value, and F5 Work Offset.

Tool Offset	Length Geometry(H)	Length Wear(H)	Diameter Geometry(D)	Diameter Wear(D)
1	0.	0.	0.	0.
2	0.	0.	0.	0.
3	0.	0.	0.	0.
4	0.	0.	0.	0.
5	0.	0.	0.	0.
6	0.	0.	0.	0.
7	0.	0.	0.	0.
8	0.	0.	0.	0.
9	0.	0.	0.	0.
10	0.	0.	0.	0.
11	0.	0.	0.	0.
12	0.	0.	0.	0.
13	0.	0.	0.	0.
14	0.	0.	0.	0.
15	0.	0.	0.	0.
16	0.	0.	0.	0.
17	0.	0.	0.	0.
18	0.	0.	0.	0.

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

### F2.39: Selectable Display Boxes

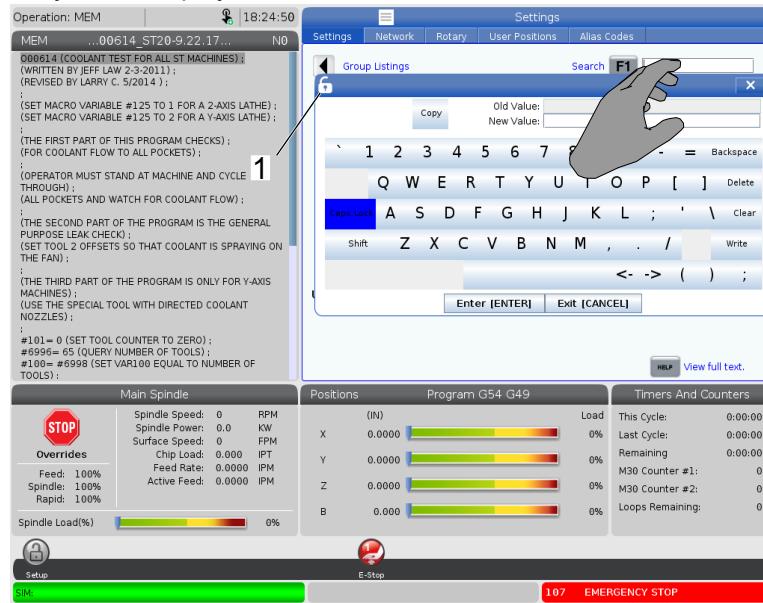


- Display boxes [1 - 7] are selectable. For example if you want to go to the Maintenance tab, press the coolant display box [4].

## 2.5.3 LCD Touchscreen - Virtual Keyboard

The virtual keyboard allows you to input text on the screen, without using the keypad. To enable this function set setting 396 - Virtual Keyboard Enabled to On.

### F2.40: Virtual Keyboard Display



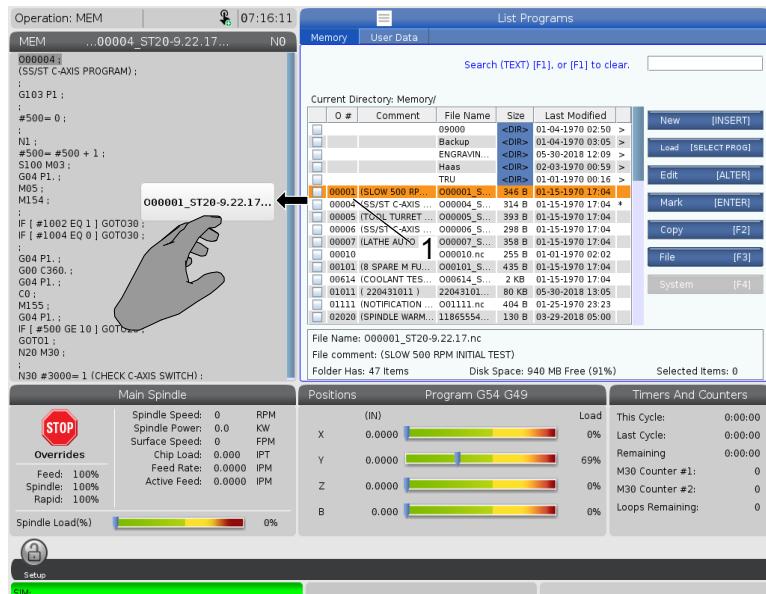
Press and hold any input line for the virtual keyboard to appear.

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

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

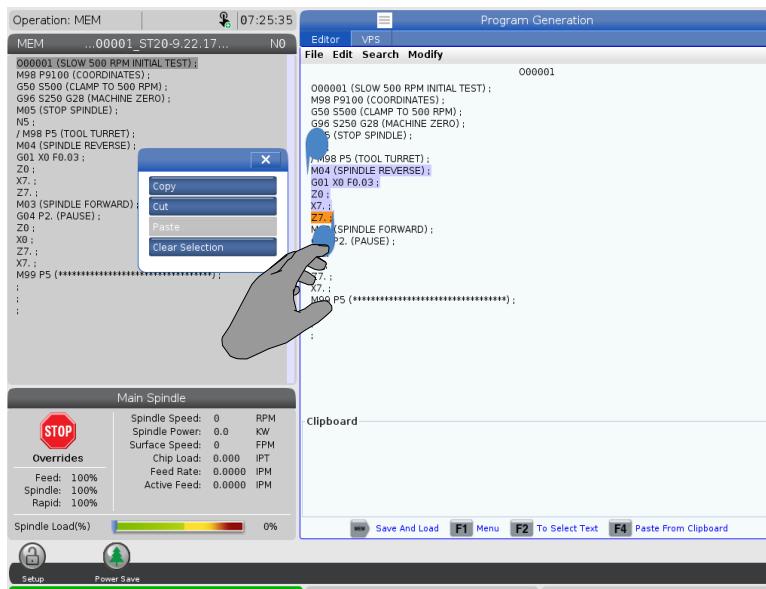
## 2.5.4 LCD Touchscreen - Program Editing

### F2.41: Drag and Drop from List Program



- You can drag and drop programs from [LIST PROGRAM] to [MEM] by dragging the file [1] over to the [MEM] display.

### F2.42: Copy, Cut, and Paste Handle Bars

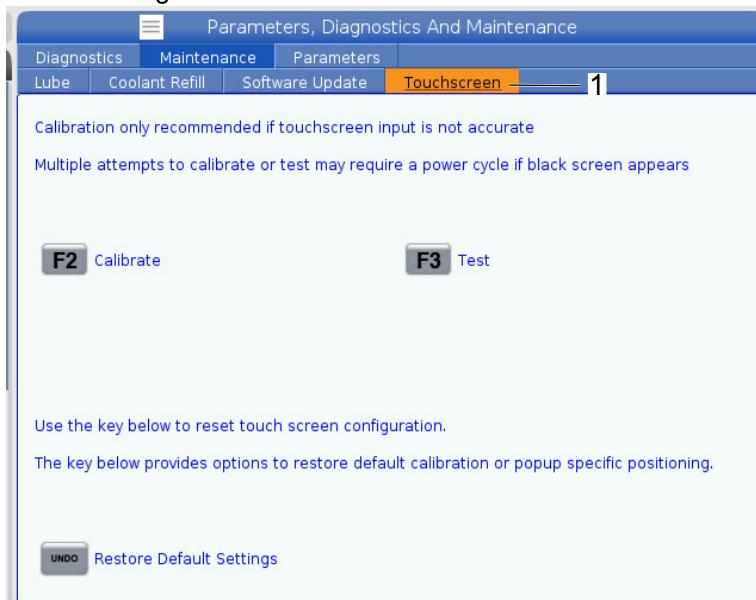


- In edit mode you can drag your fingers across the code to use the handle bars to copy, cut and paste a section of the program.

## 2.5.5 LCD Touchscreen - Maintenance

Use the touchscreen configuration page to calibrate, test, and restore default settings. The touchscreen configuration is located in the maintenance section. Press **[DIAGNOSTIC]** to go to the Maintenance and navigate to the Touchscreen tab.

F2.43: Touchscreen Configuration Tab



## 2.6 Help

Use the **[HELP]** key on the control when you need to access information about machine functions, commands, or programming printed in this manual.

To open a help topic:

1. Press **[HELP]**. You are presented with icon options for different help information. (Press **[HELP]** again to exit the **Help** window.)
2. Use the cursor arrows or **[HANDLE JOG]** control to highlight an icon option, then press **[ENTER]**. Press the **[UP]** or **[DOWN]** cursor arrows or turn the **[HANDLE JOG]** control to scroll through pages larger than the screen.
3. Press **[HOME]** to go to the top directory level or top of a page.

4. To search for help content by keyword, type your search term in the input field, then press [F1] to execute the search. Search results for the keyword appear in the **HELP** window.
5. Press the [**LEFT**]/[**RIGHT**] cursor arrow keys to go to the previous/next page on the contents pages.

## 2.6.1 Active Icon Help

Displays a list of the currently active icons.

## 2.6.2 Active Window Help

Displays the help system topic related to the currently active window.

## 2.6.3 Active Window Commands

Displays a list of the available commands for the active window. You can use the keys listed in parentheses, or you can select a command from the list.

## 2.6.4 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.

## 2.6.5 More Information Online

For updated information, including tips, tricks, maintenance procedures, and more, visit the Haas Service page at [www.HaasCNC.com](http://www.HaasCNC.com).

For the most current Operator's and Service Manuals scan the code below with your mobile device:





# Chapter 3: Control Icons

## 3.1 Next Generation Control Icon Guide

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.

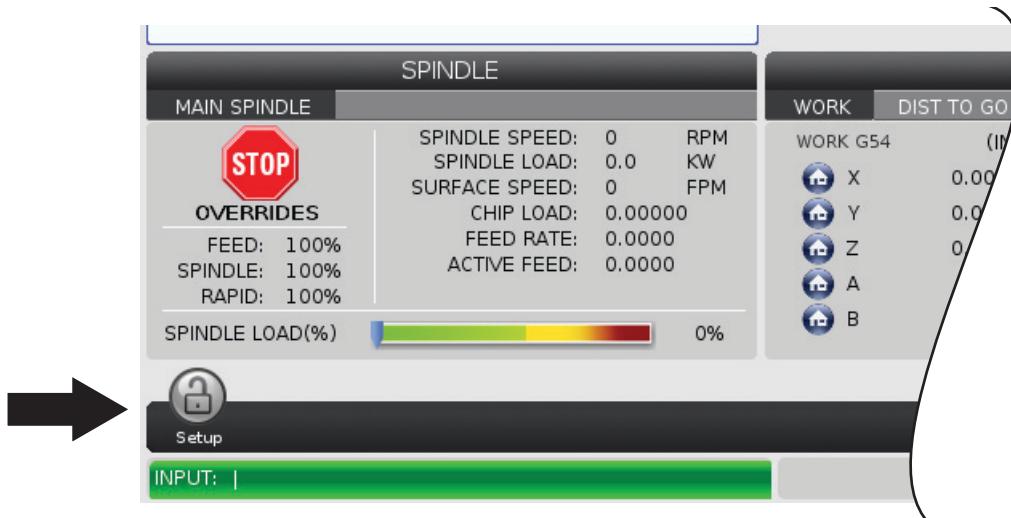
For the most current control icon information scan the code below with your mobile device.

**F3.1:** Mill - Control Icon Guide



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

**F3.2:** Icon Bar Location



## T3.1: Mill Control Icons

Name	Icon	Meaning
Setup		Setup mode is locked; the control is in Run mode. Most machine functions are disabled or limited while the machine doors are open.
Setup		Setup mode is unlocked; the control is in Setup mode. Most machine functions are available, but may be limited while the machine doors are open.
Cycle Door		The door must be cycled at least once to ensure that the door sensor is working. This icon appears after [POWER UP] If the user has not yet cycled the door.
Door Open		Warning, door is open.
Pallet Load Door Open		The pallet load station door is open.

Name	Icon	Meaning
Light Curtain Breach		This icon appears when the machine is idle and the light curtain is triggered. It also appears when a program is running and the light curtain is running. This icon disappears when the obstacle is removed from the light curtain line of sight.
Light Curtain Hold		This icon appears when a program is running and the light curtain is triggered. This icon will clear the next time [CYCLE START] is pressed.
Running		The machine is running a program.
Jog		An axis is jogging at the current jog rate.
APL Mode		This icon appears when the machine is in APL Mode.
Power Saving		The power-saving servos-off feature is active. Setting 216, SERVO AND HYDRAULIC SHUTOFF, designates the time period allowed before this feature activates. Press a key to activate the servos.

Name	Icon	Meaning
Jog	A circular icon with a downward-pointing arrow in the center, indicating a jog operation.	This icon appears while the control returns to the workpiece during a run-stop-jog-continue operation.
Jog	A circular icon with a red prohibition sign over a person jogging, indicating a jog hold.	You have pressed <b>[FEED HOLD]</b> during the return portion of a run-stop-jog-continue operation.
Jog	A circular icon with an upward-pointing arrow in the center, indicating a jog away operation.	This icon prompts you to jog away during a run-stop-jog-continue operation.
Feed Hold	A circular icon with a red prohibition sign over a right-pointing arrow, indicating feed hold.	The machine is in feed hold. Axis motion has stopped, but the spindle continues to turn.
Feed	A circular icon with a right-pointing arrow in the center, indicating a cutting move.	The machine is executing a cutting move.
Rapid	A circular icon with a silhouette of a rabbit jumping, indicating a rapid axis move.	The machine is executing a non-cutting axis move (G00) at the fastest possible rate. Overrides can affect the actual rate.

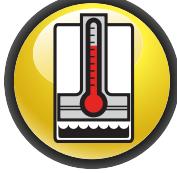
Name	Icon	Meaning
Dwell		The machine is executing a dwell (G04) command.
Restart		The control scans the program before a restart if Setting 36 is ON.
Singbk Stop		<b>SINGLE BLOCK</b> mode is active, and the control needs a command to continue.
Door Hold		Machine motion has stopped because of door rules.
Jog Lock		Jog lock is active. If you press an axis key, that axis moves at the current jog rate until you press [JOG LOCK] again, or the axis reaches its limit.
Remote Jog		The optional remote jog handle is active.

Name	Icon	Meaning
Vector Jog		For five-axis machines, the tool will jog along the vector defined by the rotary positions.
Low Gearbox Oil Flow		This icon appears when low gearbox oil flow persists for 1 minute.
Low Gearbox Oil		The control detected a low gearbox oil level.   <b>NOTE:</b> <i>In software version 100.19.000.1100 and higher the control will monitor the gearbox oil level condition when spindle fan is turned OFF. After the spindle fan turns off, there is a delay before the gearbox oil level monitoring will begin. Press [RESET] to clear the low gearbox oil icon.</i>
Rotary Lube		Check and fill the rotary table lubrication oil reservoir.
Dirty TSC/HPFC Filter		Clean the Through-Spindle Coolant or High-Pressure Flood Coolant filter.

Name	Icon	Meaning
Low Coolant Concentrate		Fill the concentrate reservoir for the coolant refill system.
PulseJet Low Oil		This icon appears when the system detects a low oil condition on the PulseJet oil reservoir.
Low Lube		The spindle lubrication oil system detected a low oil condition, or the axis ball screw lubrication system detected a low grease or low pressure condition.
Low Oil		The rotary brake oil level is low.
Residual Pressure		Before a lubrication cycle the system detected residual pressure from the grease pressure sensor. This can be caused by an obstruction in the axes grease lubrication system.
Mist Filter		Clean the mist extractor filter.

Name	Icon	Meaning
Vise Clamp		This icon appears when the vise is commanded to clamp.
Low Coolant (Warning)		Coolant level is low.
Low PulseJet Oil		The PulseJet oil level is low.
Mist Condenser		This icon appears when the mist condenser is turned on.
Low Air Flow		Inch Mode - Air flow is not sufficient for correct machine operation.
Low Air Flow		Metric Mode - Air flow is not sufficient for correct machine operation.

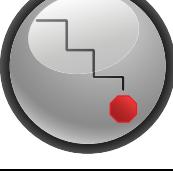
Name	Icon	Meaning
Spindle		When you press <b>[HANDLE SPINDLE]</b> , the jog handle varies the spindle override percentage.
Feed		When you press <b>[HANDLE FEED]</b> , the jog handle varies the feed rate override percentage.
Handle Scroll		When you press <b>[HANDLE SCROLL]</b> , the jog handle scrolls through the text.
Mirroring		Mirroring mode is active. Either G101 is programmed or Setting 45, 46, 47, 48, 80, or 250 (mirror image of axis X, Y, Z, A, B, or C) is set to ON.
Brake		A rotary axis brake, or a combination of rotary axis brakes, is unclamped.

Name	Icon	Meaning
Brake		A rotary axis brake, or a combination of rotary axis brakes, is clamped.
HPU Oil Low		The HPU oil level is low. Check the oil level and add the recommended oil for the machine.
HPU Oil Temperature (Warning)		The oil temperature is too high to reliably operate the HPU.
Spindle Fan Failed		This icon appears when the spindle fan stops operating.
Electronics Overheat (Warning)		This icon appears when the control has detected cabinet temperatures are approaching levels that are potentially dangerous to the electronics. If the temperature reaches or exceeds this recommended level alarm 253 ELECTRONICS OVERHEAT will be generated. Inspect the cabinet for clogged air filters and correctly operating fans.

Name	Icon	Meaning
Electronics Overheat (Alarm)		This icon appears when the electronics remains in the overheat state for too long. The machine will not operate until the condition is corrected. Inspect the cabinet for clogged air filters and correctly operating fans.
Transformer Overheat (Warning)		This icon appears when the transformer is detected to be overheated for more than 1 second.
Transformer Overheat (Alarm)		This icon appears when the transformer remains in the overheat state for too long. The machine will not operate until the condition is corrected.
Low Voltage (Warning)		The PFDM detects low incoming voltage. If the condition continues, the machine cannot continue operation.
Low Voltage (Alarm)		The Power Fault Detect Module (PDFM) detects incoming voltage that is too low to operate. The machine will not operate until the condition is corrected.

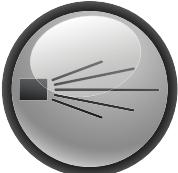
Name	Icon	Meaning
High Voltage (Warning)		The PFDM detects incoming voltage above a set limit, but still within operating parameters. Correct the condition to prevent damage to machine components.
High Voltage (Alarm)		The PFDM detects incoming voltage that is too high to operate and could cause damage to the machine. The machine will not operate until the condition is corrected.
Surge Protector Fault Detected		Indicates a Surge Protector Fault has been detected. This icon is active until the fault has been cleared.  <b>WARNING:</b> <i>If continuing to use the machine in this state. The electronics are capable of being damaged due to any electrical surge.</i>
Robot Battery is Low		Robot Battery is Low. Please replace the pulse coder batteries as soon as possible. Do NOT turn off the robot, otherwise it may require remastering. Reference 9156.062 ROBOT COMMAND FAILED SRVO-062 BZAL alarm in service documentation for more information.
High Air (Warning)		The air pressure to the machine is too high to reliably operate pneumatic systems. Correct this condition to prevent damage to or incorrect operation of pneumatic systems. You may need to install a regulator at the machine's air input.

Name	Icon	Meaning
Low Air (Alarm)		The air pressure to the machine is too low to operate pneumatic systems. The machine will not operate until the condition is corrected. You may need a higher-capacity air compressor.
Low Air (Warning)		The air pressure to the machine is too low to reliably operate pneumatic systems. Correct this condition to prevent damage to or incorrect operation of pneumatic systems.
High Air (Alarm)		The air pressure to the machine is too high to operate pneumatic systems. The machine will not operate until the condition is corrected. You may need to install a regulator at the machine's air input.
Pendant E-Stop		<b>[EMERGENCY STOP]</b> on the pendant has been pressed. This icon disappears when <b>[EMERGENCY STOP]</b> is released.
APC E-Stop		<b>[EMERGENCY STOP]</b> on the pallet changer has been pressed. This icon disappears when <b>[EMERGENCY STOP]</b> is released.
Tool Changer E-Stop		<b>[EMERGENCY STOP]</b> on the tool changer cage has been pressed. This icon disappears when <b>[EMERGENCY STOP]</b> is released.

Name	Icon	Meaning
Auxiliary E-Stop		[EMERGENCY STOP] on an auxiliary device has been pressed. This icon disappears when [EMERGENCY STOP] is released.
Remote Jog Handle-XL (RJH-XL) E-Stop		[EMERGENCY STOP] on the RJH-XL has been pressed. This icon disappears when [EMERGENCY STOP] is released.
Single Blk		<b>SINGLE BLOCK</b> mode is active. The control executes programs (1) block at a time. Press [CYCLE START] to execute the next block.
Tool Life (Warning)		The tool life remaining is below Setting 240, or the current tool is the last one in its tool group.
Tool Life (Alarm)		The tool or tool group has expired, and no replacement tools are available.
Opt Stop		<b>OPTIONAL STOP</b> is active. The control stops the program at each M01 command.

Name	Icon	Meaning
Blk Delete		<b>BLOCK DELETE</b> is active. When Block Delete is On, the control ignores (does not execute) the code following a Forward Slash (/), on that same line.
TC Door Open		The side-mount tool changer door is open.
TC Manual Mode		This icon appears when the tool carousel is in manual mode via the auto/manual switch. This switch is only on machines with tool cages.
TL CCW		The side-mount tool changer carousel is rotating counter-clockwise.
TL CW		The side-mount tool changer carousel is rotating clockwise.
Tool Change		A tool change is in progress.

Name	Icon	Meaning
Tool Unclamped		The tool in the spindle is unclamped.
Probe		The probe system is active.
Conveyor Fwd		The conveyor is active and moving forward.
Conveyor Rev		The conveyor is active and moving in reverse.
TSC		The Through-Spindle Coolant (TSC) system is active.
TAB		The Tool Air Blast (TAB) system is active.

Name	Icon	Meaning
Air Blast		The Auto Air Gun is active.
HIL Light		Indicates the optional High Intensity Lights (HIL) are turned ON and the doors are open. Duration is determined by Setting 238.
Coolant		The main coolant system is active.

## 3.2 More Information Online

For updated information, including tips, tricks, maintenance procedures, and more, visit the Haas Service page at [www.HaasCNC.com](http://www.HaasCNC.com).

For the most current Operator's and Service Manuals scan the code below with your mobile device:





# Chapter 4: Operation

## 4.1 Machine Power-On

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

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

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

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



**WARNING:**

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

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



After the first [**POWER UP**], the axes move toward their home positions. The axes then move slowly until the machine finds the home switch for each axis. This establishes the machine home position.

6. Press any of the following:
  - a. [**CANCEL**] to dismiss the screen.
  - b. [**CYCLE START**] to run current program.
  - c. [**HANDLE JOG**] for manual operation.

## 4.2 Spindle Warm-Up

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

Your machine includes a 20-minute warm-up program (009220) in the program list. If you use the spindle at consistent high speeds, you should run this program every day.

## 4.3 Device Manager (**[LIST PROGRAM]**)

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

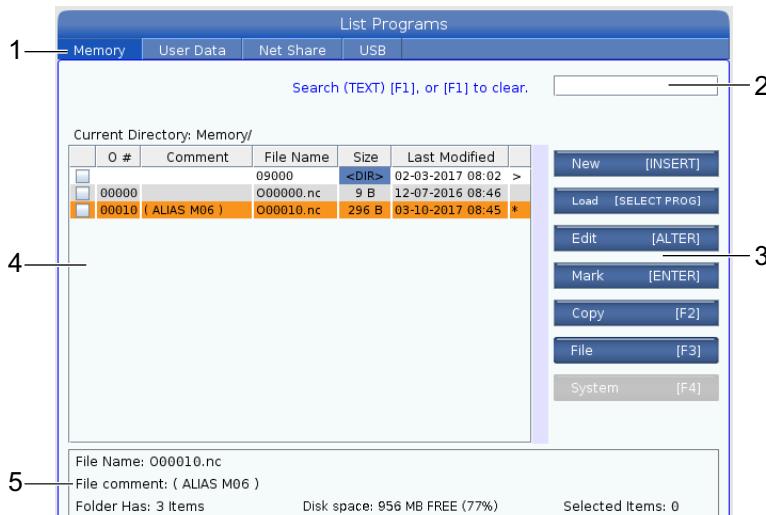
In the tabbed menu at the top of the display, the device manager (**[LIST PROGRAM]**) shows you only the available memory devices. For example, if you do not have a USB memory device connected to the control pendant, the tabbed menu does not show a **USB** tab. For more information on navigating tabbed menus, refer to page **73**.

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

### 4.3.1 Device Manager Operation

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

- F4.1:** Device Manager Initial Screen Example: [1] Available Device Tabs, [2] Search Box, [3] Function Keys, [4] File Display, [5] File Comments (only available in **Memory**).



Use the cursor arrow keys to navigate the directory structure:

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

## 4.3.2 File Display Columns

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

### F4.2: Program/Directory List Example

**Current Directory: Memory**

	O #	Comment	File Name	Size	Last Modified	
			TEST	<DIR>	2015/11/23 08:54	>
			programs	<DIR>	2015/11/23 08:54	>
	00010		000010.nc	130 B	2015/11/23 08:54	
	00030		000030.nc	67 B	2015/11/23 08:54	*
	00035		000035.nc	98 B	2015/11/23 08:54	
	00045		NEXTGENte...	15 B	2015/11/23 08:54	
	09001 (ALIAS M89)		O9001.nc	94 B	2015/11/23 08:54	

The columns are:

- File selection check box (no label): Press **ENTER** to toggle a check mark on and off in the box. A check mark in a box indicates that the file or directory selected for operations on multiple files (usually copy or delete).
- Program O Number (**O #**): This column lists the program numbers of the programs in the directory. The letter 'O' is omitted in the column data. Only available in the **Memory** tab.
- File comment (**Comment**): This column lists the optional program comment that appears in the first line of the program. Only available in the **Memory** tab.
- File Name (**File Name**): This is an optional name that the control uses when you copy the file to a memory device other than the control. For example, if you copy program **000045** to a USB memory device, the filename in the USB directory is **NEXTGENtest.nc**.
- File Size (**size**): This column shows the amount of storage space that the file takes up. Directories in the list have the designation **<DIR>** in this column.



#### NOTE:

*This column is hidden by default, press the [F3] button and select Show File Details to display this column.*

- Date Last Modified (**Last Modified**): This column shows the last date and time when the file was changed. The format is YYYY/MM/DD HR:MIN.

**NOTE:**

*This column is hidden by default, press the [F3] button and select Show File Details to display this column.*

- Other Information (no label): This column gives you some information about a file's status. The active program has an asterisk (\*) in this column. A letter **E** in this column means that the program is in the program editor. A greater-than symbol (>) indicates a directory. A letter **S** indicates that a directory is part of Setting 252 (refer to page 471 for more information). Use the [**RIGHT**] or [**LEFT**] cursor arrow keys to enter or exit the directory.

### 4.3.3 Create a New Program

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

**F4.3:** Create New Program Popup Menu Example: [1] Program O number field, [2] File Name field, [3] File comment field.



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

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

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

**NOTE:**

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

- **File Name** (optional): Type a filename for the new program. This is the name the control uses when you copy the program to a storage device other than memory.
- **File comment** (optional): Type a descriptive program title. This title goes into the program as a comment in the first line with the O number.

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

#### 4.3.4

## Create A Container

The control has the ability to group files together and create a zip file, you can also unzip the files.

**To zip the files:**

1. Press **[LIST PROGRAM]**.
2. Navigate and highlight a.nc file.
3. Press **[SELECT PROGRAM]**.
4. Press **[F3]** and select Create Container.
5. Select the programs you will like to zip.

**NOTE:**

*You can press **[ALTER]** to change the save location.*

**NOTE:**

*Any files that the control cannot find will be marked in red and must be unchecked from the container before it can package the files.*

6. Press **[F4]** to start packing.

**To unzip the files:**

1. Select the \*.hc.zip file and press **[F3]**.
2. Press **[F4]** to extract the files.

**NOTE:**

*When unzipping the control will overwrite existing files and they will be highlighted in red. If you don't want to overwrite existing files make sure you unchecked the file before extracting.*

### 4.3.5 Select the Active Program

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

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

### 4.3.6 Check Mark Selection

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

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

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

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

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

**NOTE:**

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

**NOTE:**

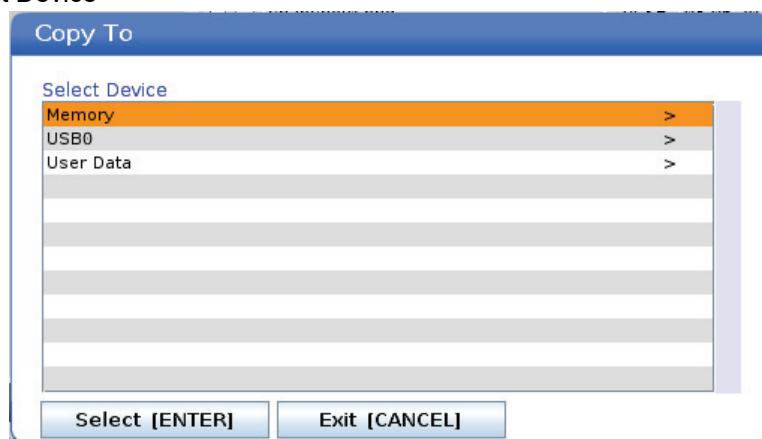
*If you have not selected multiple files with check marks, the control does operations only on the currently highlighted directory or file. If you have selected files, the control does operations only on the selected files and not on the highlighted file, unless it is also selected.*

### 4.3.7 Copy Programs

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

1. To copy a single program, highlight it in the device manager program list and press **[ENTER]** to assign a check mark. To copy multiple programs, check-mark all the programs you want to copy.
2. Press **[F2]** to start the copy operation.  
The Select Device popup appears.

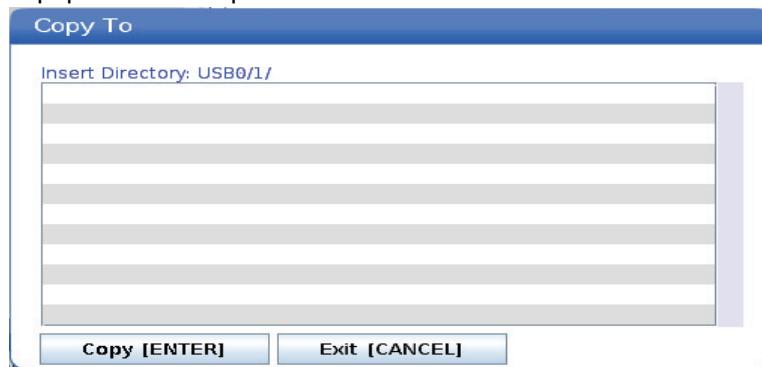
**F4.4:** Select Device



3. Use the cursor arrow keys to select the destination directory. **[RIGHT]** cursor to enter the chosen directory.

The **Insert Directory**: Copy popup menu appears.

**F4.5:** Copy Popup Menu Example



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

### 4.3.8 Edit a Program

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

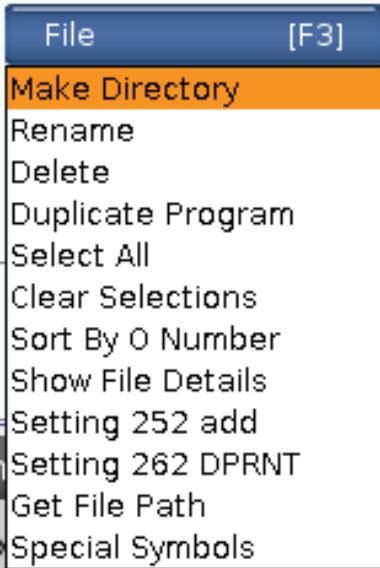
The program has the designation **E** in the far-right column of the file display list when it is in the editor, unless it is also the active program.

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

### 4.3.9 File Commands

Press **[F3]** to access the file commands menu in the device manager. The list of options appears under the **File [F3]** drop-down menu in the device manager. Use the cursor arrow keys or the jog handle to highlight a command, and then press **[ENTER]**.

**F4.6:** The File Commands Menu



- **Make Directory:** creates a new subdirectory in the current directory. Type a name for the new directory, and then press **[ENTER]**.
- **Rename:** changes the name of a program. The **Rename** popup menu has the same options as the new program menu (File Name, O Number, and File Title).
- **Delete:** deletes files and directories. When you confirm the operation, the control deletes the highlighted file, or all of the files with check-mark selection.
- **Duplicate Program:** makes a copy of a file in the current location. The **Save As** popup menu asks you to specify a new program name before you can complete this operation.
- **Select All:** adds check-marks to all of the files/directories in the **Current Directory**.
- **Clear Selections:** removes check-marks from all of the files/directories in the **Current Directory**.
- **Sort By O Number:** sorts the program list by O number. Use this menu item again to sort by filename. By default, the program list is sorted by filename. Only available in the **Memory** tab.

- **Setting 252 add / Setting 252 remove:** adds or removes a custom sub-program search location to the list of locations. See Setting Up Search Locations section for more information.
- **Setting 262 DPRNT:** adds a custom destination file path for DPRNT.
- **Get File Path:** places the path and the name of the selected file in parenthesis on the input bar.
- **Special Symbols:** accesses text symbols that are not available on the keyboard. Enter the number of the character you want to use to put it in the input bar. The special characters are: \_ ^ ~ { } \ < >

## 4.4

## Screen Capture

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

1. Enter a filename if desired. If no filename is entered, the system will use the default filename (see note).
2. Press **[SHIFT]**.
3. Press **[F1]**.



### NOTE:

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

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

## 4.5

## Error Report

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

1. Press [SHIFT].
2. Press [F3].

**NOTE:**

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

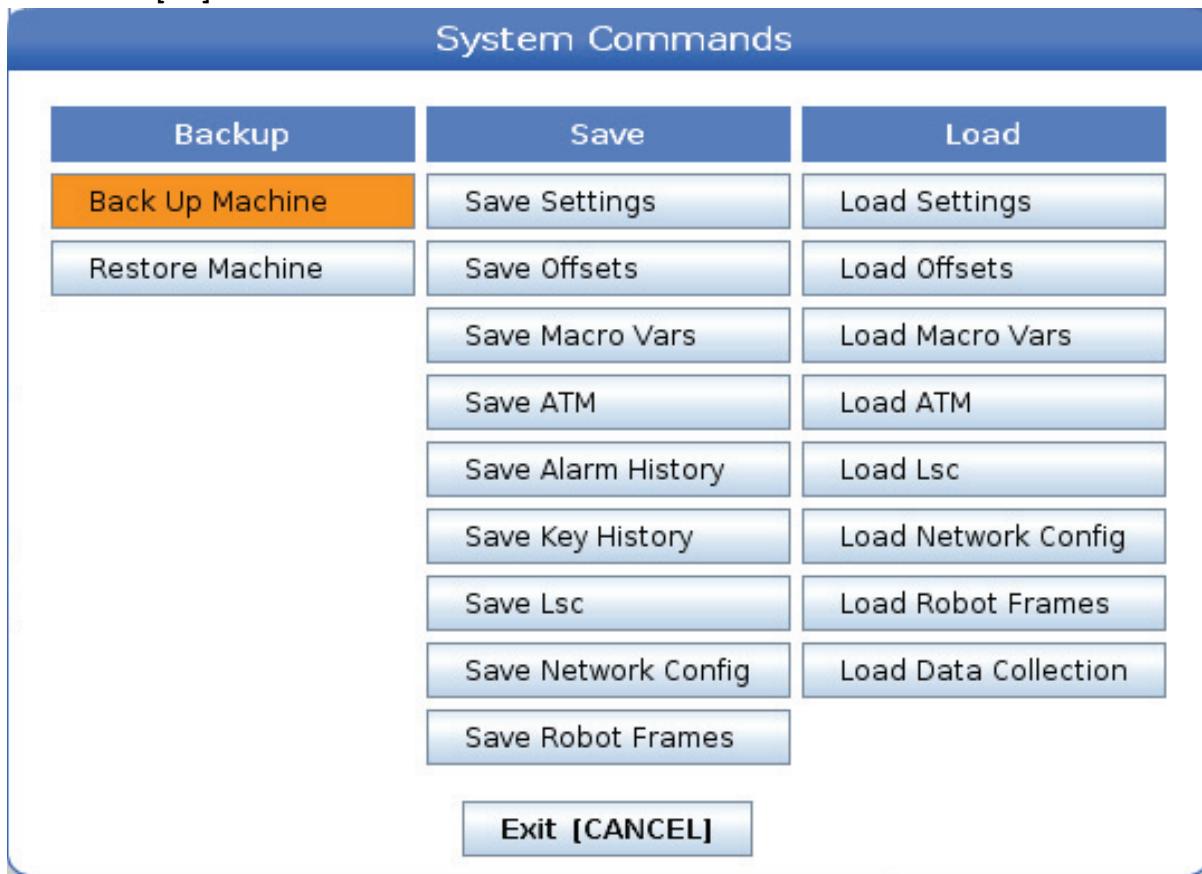
The control saves the error report to your USB device or control memory. The error report is a zip file that includes a screen capture, the active program, and other information used for diagnostics. Generate this error report when an error or an alarm occurs. E-mail the error report to your local Haas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

## 4.6 Full Machine Backup

The backup function makes a copy of your machine's settings, programs, and other data so that you can easily restore them.

You create and load backup files with the **System [F4]** drop-down menu.

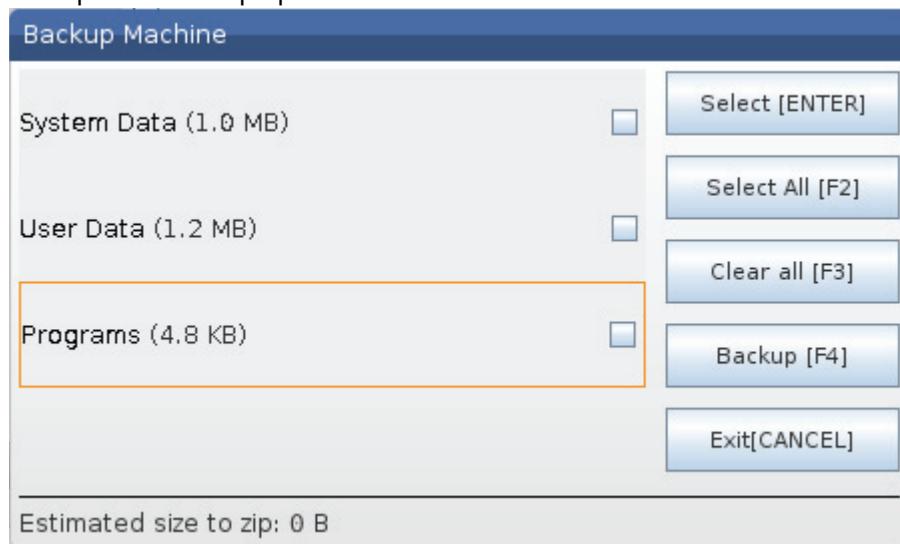
## F4.7: [F4] Menu Selections



To create a full machine backup:

1. Press **[LIST PROGRAM]**.
2. Navigate to the **USB** or **Network Device**.
3. Press **[F4]**.
4. Select **Backup Machine** and press **[ENTER]**.

## Backup Machine Pop-up Menu



5. Highlight the data to back up and press **[ENTER]** to apply a check mark. Press **[F2]** to select all data. Press **[F3]** to clear all check marks.
6. Press **[F4]**.  
The control saves the backup you selected in a zip file labeled **HaasBackup (mm-dd-yyyy) .zip** where mm is the month, dd is the day, and yyyy is the year.

## T4.1: Default File Names in the Zip File

Selected Backup	Data Saved	Name of File (Folder)
System Data	Settings	(Serial Number)
System Data	Offsets	OFFSETS.OFS
System Data	Alarm History	AlarmHistory.txt
System Data	Advanced Tool Management (ATM)	ATM.ATM
System Data	Key History	KeyHistory.HIS
Programs	Memory Files and Folders	(Memory)
User Data	User Data Files and Folders	(User Data)

## 4.6.1 Selected Machine Data Backup

To back up selected information from your machine:

1. If a USB is used, insert a USB memory device into the **[USB]** port on the right side of the control pendant. If **Net Share** is used, make sure **Net Share** is correctly set up.
2. Using the **[LEFT]** and **[RIGHT]** cursors navigate to **USB** in the Device Manager.
3. Open the destination directory. If you want to create a new directory for your backup data, refer to page **111** for instructions.
4. Press **[F4]**.
5. Select the menu option for the data you want to back up, and press **[ENTER]**.
6. Type a filename in the **Save As** pop-up menu. Press **[ENTER]**. The message **SAVED** is displayed after saving is complete. If the name exists you can overwrite or type a new name.

The file types for backups are listed in the following table.

**T4.2:** Menu Selection and File Name for Backup

<b>F4 Menu Selection</b>	<b>Save</b>	<b>Load</b>	<b>Created File</b>
Settings	yes	yes	USB0/serialnumber/CONFIGURATION/ serialnumber_us.xml
Offsets	yes	yes	filename.OFS
Macro Vars	yes	yes	filename.VAR
ATM	yes	yes	filename.ATM
Lsc	yes	yes	filename.LSC
Network Config	yes	yes	filename.xml
Alarm History	yes	no	filename.txt
Key History	yes	no	filename.HIS

**NOTE:**

*When you back up settings, the Control does not prompt for a file name. It saves the file in a subdirectory:*

- USB0/machine serial number/CONFIGURATION/machine serial number\_us.xml

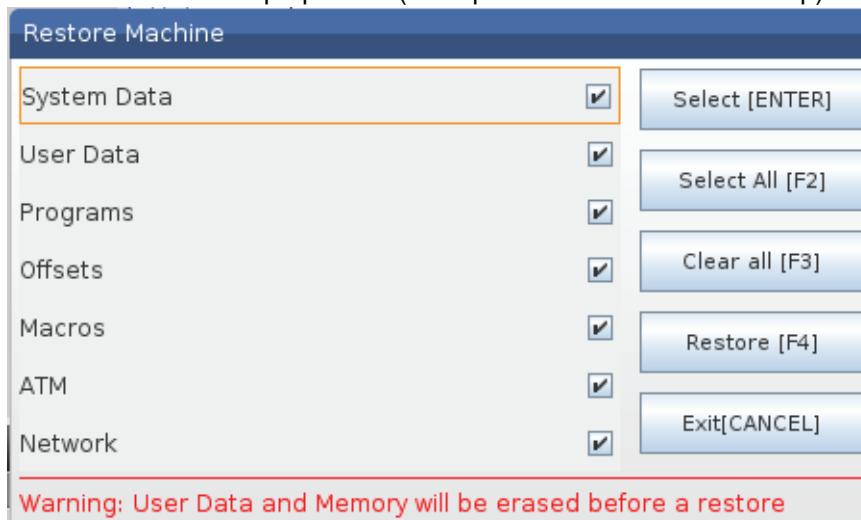
## 4.7 Restoring a Full Machine Backup

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. Navigate to **USB** in the Device Manager.
3. Press **[EMERGENCY STOP]**.
4. Open the directory that contains the backup you want to restore.
5. Highlight the HaasBackup zip file to load.
6. Press **[F4]**.
7. Select **Restore Machine** and press **[ENTER]**.

The Restore Machine pop-up window shows what types of data can be selected to restore.

**F4.8:** **Restore Machine** Pop-up menu (example shows an all data backup)



8. Highlight the data to restore and press **[ENTER]** to apply a check mark. Press **[F2]** to select all data. Press **[F3]** to clear all selectors.



**NOTE:**

A restore can be stopped at any time by pressing [CANCEL] or [RESET] except when restoring System Data.



**WARNING:**

User data and memory are erased prior to a restore.

9. Press F4.

Each data area restored is checked-off and initialized.

#### 4.7.1 Restoring Selected Backups

This procedure tells you how to restore selected data backups from 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. Navigate to **USB** in the Device Manager.
3. Press **[EMERGENCY STOP]**.
4. Open the directory that contains the files you want to restore.
5. Highlight or type in the name of the file to be restored. Typed file name takes precedence over highlighted file name.



**NOTE:**

Type the backup name with or without a file extension (e.g., MACROS or MACROS.VAR)

6. Press **[F4]**.
7. Highlight the backup type to load and press **[ENTER]**.

The highlighted file or the typed name file is loaded on the machine. The message *Disk Done* displays after loading is complete.



**NOTE:**

Settings load the moment you pick Settings from the System [F4] drop-down menu. Highlighting or typing is not required.

## 4.8 Basic Program Search

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



**NOTE:**

*This is a quick-search function that finds the first match in the search direction that you specify. You can use the Editor for a more full-featured search. Refer to page 181 for more information on the 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.



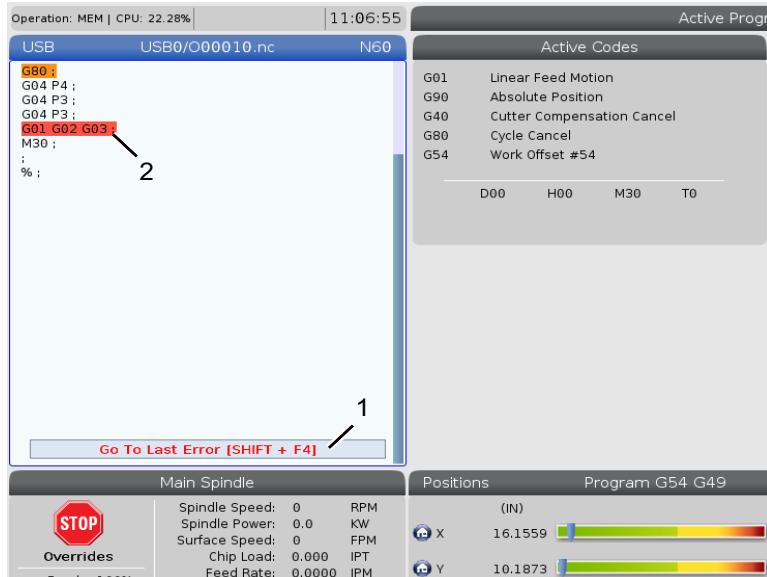
**NOTE:**

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

## 4.9 Locate the Last Program Error

Starting in software version 100.19.000.1100 the control can find the last error in a program. Press **[SHIFT] + [F4]** to display the last line of G-code that generated the error.

**F4.9:** Press **[SHIFT] + [F4]** [1] to display the last G-code error [2].



## 4.10 Safe Run Mode

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



**NOTE:**

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

### Safe Run Supported Machines

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

**Common causes for crashes are:**

- Incorrect tool offsets.
- Incorrect work offsets.
- Wrong tool in the spindle.

**NOTE:**

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

**Safe Run does the following:**

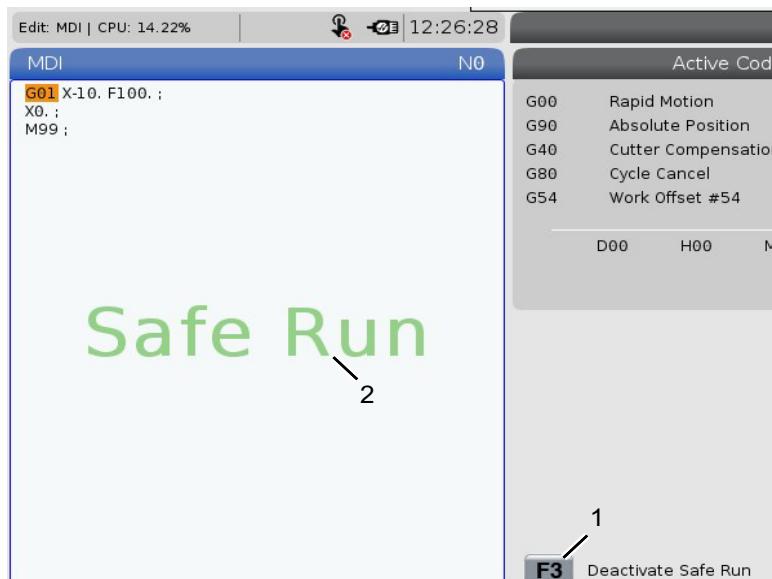
- Slow down the speed of the motion.
- Increases the position error sensitivity.
- When a crash is detected, the control will immediately reverse the axis by a small amount. This will prevent the motor from continuing to drive into the object it has crashed into as well as relieve pressure from the crash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

**NOTE:**

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

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

### F4.10: Safe Run Mode

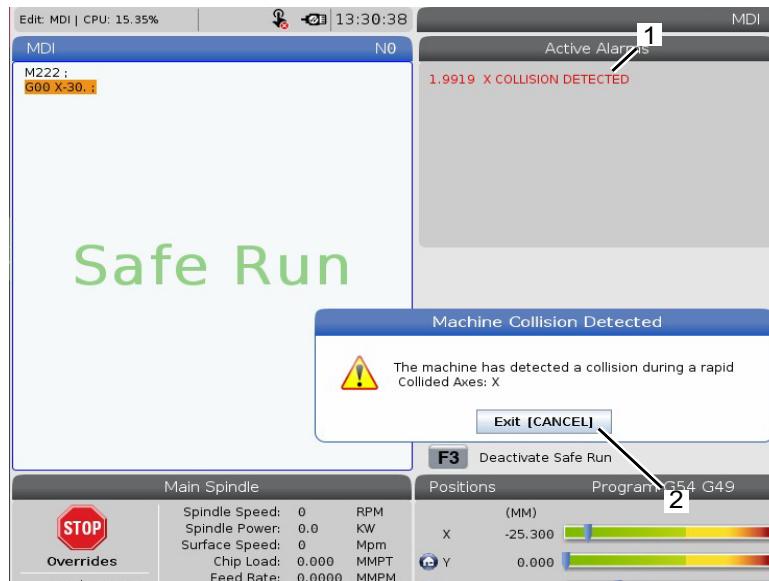


If your machine supports Safe Run, you will see a new icon in MDI with the text *F3 Activate Safe Run* [1]. Press **[F3]** to turn Safe Run on/off. Safe Run Active state is noted by a water mark [2] in the program panel.

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

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

#### F4.11: Safe Run Mode



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

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

## 4.11 Tooling

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

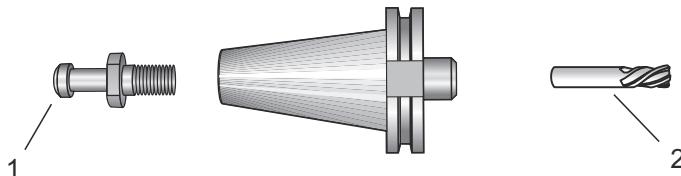
### 4.11.1 Tool Holders

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

## Tool Holder Care

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

F4.12: Tool Holder Assembly, 40-Taper CT Example: [1] Pull Stud, [2] Tool (Endmill).



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

## Pull Studs

A pull stud (sometimes called a retention knob) secures the tool holder into the spindle. Pull studs are threaded into the top of the tool holder and are specific to the type of spindle. Refer to the 30, 40, and 50-taper spindle and tooling information on the Haas Service website for descriptions of the pull studs you need.



**CAUTION:**

*Do not use short-shaft or pull studs with a sharp right-angle (90-degree) head; they will not work and will cause serious damage to the spindle.*

### 4.11.2 Advanced Tool Management Introduction

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

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

When a tool expires:

- The beacon will flash.
- ATM puts the expired tool in the EXP group
- Tool groups that contain the tool appear with a red background.

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

- F4.13:** Advanced Tool Management Window: [1] Active window label, [2] Allowed Limits window, [3] Tool Group window, [4] Tool Data window, [5] Help text

The screenshot shows the ATM window with the following tabs: Devices, Timers, Macro Vars, Active Codes, ATM (highlighted in orange), Tool Table, and Calcul... (partially visible). A callout labeled F4 To Switch Boxes points to the ATM tab.

**Allowed Limits:**

Group	Expired Count	Tool Order	Holes Limit	Usage Limit	Life Warn %	Expired Action	Feed Limit	Total Time Limit
All	-	-	-	-	-	-	-	-
Expired	4	-	-	-	-	-	-	-
No Group	-	-	-	-	-	-	-	-
1001	1 / 5	Newest	99999	99999	100	Alarm	1000:00	1000:00
1002	0 / 0	Ordered	99999	99999	100	Feedhold	100:00	100:00
Add Group	-	-	-	-	-	-	-	-

**Tool Data For Group: All**

Tool	Life	Holes Count	Usage Count	Usage Limit	H-Code	D-Code	Feed Time	Total Time
1	0%	100	50	25	1	1	0:00:00	0:00:00
2	0%	50	25	25	2	2	0:00:00	0:00:00
3	0%	30	10	10	3	3	0:00:00	0:00:00
4	95%	10	5	100	4	4	0:00:00	0:00:00
5	0%	0	0	0	5	5	0:00:00	0:00:00
6	100%	0	0	0	0	0	0:00:00	0:00:00

**Help Text:**

INSERT Add Group

### Allowed Limits

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

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

The columns in the **ALLOWED LIMITS** table are:

- **GROUP** - Displays the tool group's ID number. this is the number you use to specify the tool group in a program.
- **EXP #** - Tells you how many tools in the group are expired. If you highlight the **ALL** row, you see a list of all of the expired tools in all groups.
- **ORDER** - Specifies the tool to use first. If you select **ORDERED**, ATM uses the tools in tool number order. You can also have ATM automatically use the **NEWEST** or **OLDEST** tool in the group.

- **USAGE** - The maximum number of times the control can use a tool before it expires.
- **HOLEs** - The maximum number of holes a tool is allowed to drill before it expires.
- **WARN** - The minimum value for tool life remaining in the group before the control gives a warning message.
- **LOAD** - The allowed load limit for tools in the group before the control does the **ACTION** that the next column specifies.
- **ACTION** - The automatic action when a tool reaches its maximum tool load percentage. Highlight the tool action box to change and press **[ENTER]**. Use the **[UP]** and **[DOWN]** cursor keys to select the automatic action from the pull down menu (**ALARM**, **FEEDHOLD**, **BEEP**, **AUTOFEED**, **NEXT TOOL**).
- **FEED** - The total amount of time, in minutes, that the tool can be in a feed.
- **TOTAL TIME** - The total amount of time, in minutes, that the control can use a tool.

### Tool Data

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

- **TOOL#** - Shows the tool numbers used in the group.
- **LIFE** - The percentage of life left in a tool. This is calculated by the CNC control, using actual tool data and the allowed limits the operator entered for the group.
- **USAGE** - The total number of times that a program has called the tool (number of tool changes).
- **HOLES** - The number of holes the tool has drilled/ tapped/ bored.
- **LOAD** - The maximum load, in percent, exerted on the tool.
- **LIMIT** - The maximum load allowed for the tool
- **FEED** - The amount of time, in minutes, the tool has been in a feed.
- **TOTAL** - The total amount of time, in minutes, the tool has been used.
- **H-CODE** - The tool-length code to use for the tool. You can edit this only if Setting 15 is set to **OFF**.
- **D-CODE** - The diameter code to use for the tool.

**NOTE:**

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

## Tool Group Setup

To add a tool group:

1. Select the **ALLOWED LIMITS** table.
2. Use the cursor arrow keys to highlight an empty row.

3. Type the group identification number (between 1000 and 2999) that you want to use for the new tool group.
4. Press [ENTER].

## Manage Tools in a Group

To add, change, or delete a tool in a group:

1. Highlight the group you want to work with in the ALLOWED LIMITS table.
2. Press [F4] to switch to the TOOL DATA table.
3. Use the cursor arrow keys to highlight an empty row.
4. Type an available tool number between 1 and 200.
5. Press [ENTER].
6. To change a tool number assigned to a group, use the cursor keys to highlight the tool number you want to change.
7. Type a new tool number.



**NOTE:**

*You can type 0 if you want to delete the tool from the tool group.*

8. Press [ENTER].

## Tool Group Usage

To use a tool group in a program, substitute the tool group ID number for the tool number and for the H-codes and D-codes in the program. Refer to this program for an example of the program format.

Example:

```
%  
O30001 (Tool change ex-prog);  
(G54 X0 Y0 is top right corner of part) ;  
(Z0 is on top of the part) ;  
(Group 1000 is a drill) ;(T1000 PREPARATION BLOCKS) ;  
T1000 M06 (Select tool group 1000) ;  
G00 G90 G40 G49 G54 (Safe startup) ;  
X0 Y0 (Rapid to 1st position) ;  
S1000 M03 (Spindle on CW) ;  
G43 H1000 Z0.1 (Tool group offset 1000 on) ;  
M08 (Coolant on) ;  
(T1000 CUTTING BLOCKS) ;  
G83 Z-0.62 F15. R0.1 Q0.175 (Begin G83);
```

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

## Advanced Tool Management Macros

Advanced Tool Management (ATM) can use macros to obsolete a tool within a tool group. Macros 8001 to 8200 represent tools 1 through 200. You can set one of these macros to 1 to expire a tool. For example:

8001 = 1 (this expires tool 1)

8001 = 0 (this makes tool 1 available)

Macro variables 8500 - 8515 enable a G code program to obtain tool group information. When you specify a tool group ID number with macro 8500, the control returns the tool group information in macro variables #8501 through #8515. Refer to the variables #8500 - #8515 in the Macros chapter for the macro variable data label information.

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

## Save Advanced Tool Management Tables

You can save to USB the variables associated with Advanced Tool Management (ATM).

To save the ATM information:

1. Select the USB device in the Device Manager (**[LIST PROGRAM]**).
2. Type a filename on the input line.
3. Press **[F4]**.
4. Highlight **SAVE ATM** in the pop-up menu.
5. Press **[ENTER]**.

## Restore Advanced Tool Management Tables

You can restore from USB the variables associated with Advanced Tool Management (ATM).

To restore the ATM information:

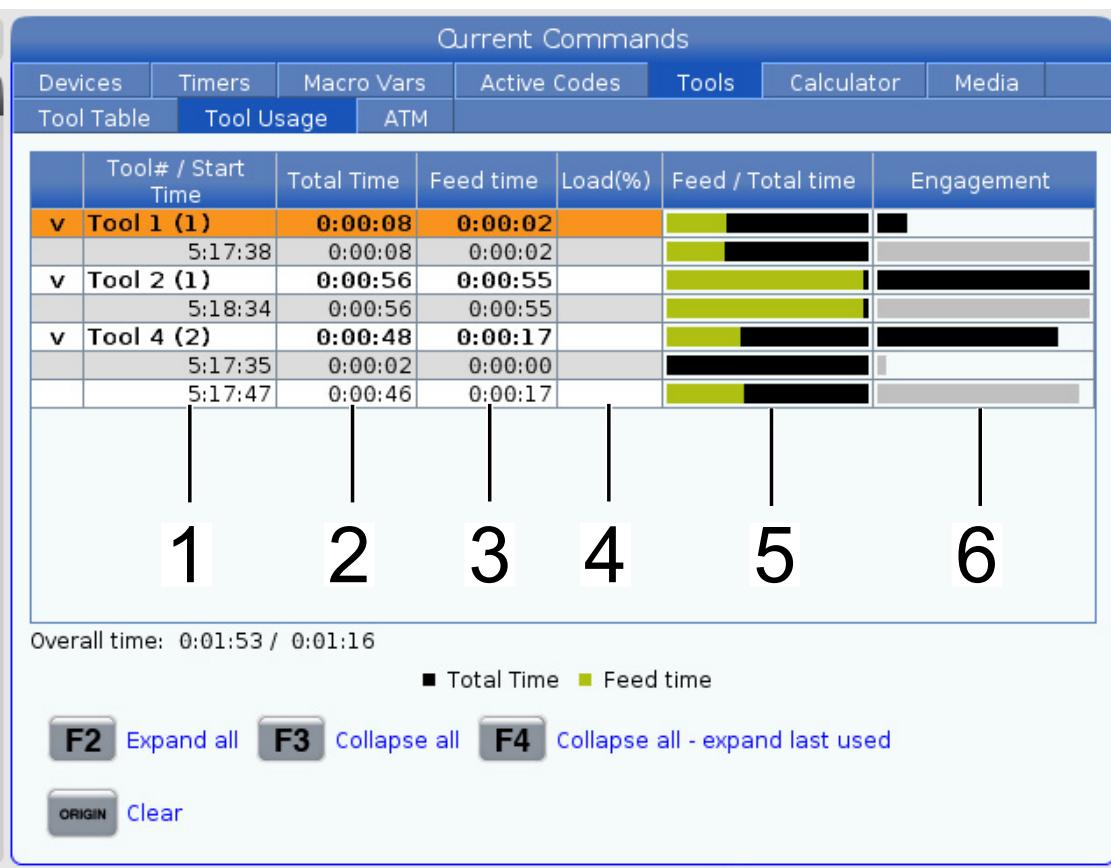
1. Select the USB device in the Device Manager (**[LIST PROGRAM]**).
2. Press **[F4]**.
3. Highlight **LOAD ATM** in the pop-up menu.
4. Press **[EMERGENCY STOP]**.
5. Press **[ENTER]**.

### 4.11.3 Tool Usage Display

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

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

#### F4.14: Tool Usage Display



1. **Start Time** - When the tool was inserted to the spindle.
  2. **Total Time** - The total time the tool has been in the spindle.
  3. **Feed time** - Tool usage time.
  4. **Load%** - The maximum load of the Spindle during a tool usage.

**NOTE:**

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

5. **Feed/Total time** - A graphic representation of the feed time of the tool over the total time.
6. **Engagement:**
  - **Black Bar**- The tool usage versus another tools.
  - **Gray Bar** - This bar shows how long the tool was used in this usage related to other usages.

### Macros Interface

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

#### T4.3: Tool Usage Macro Variables

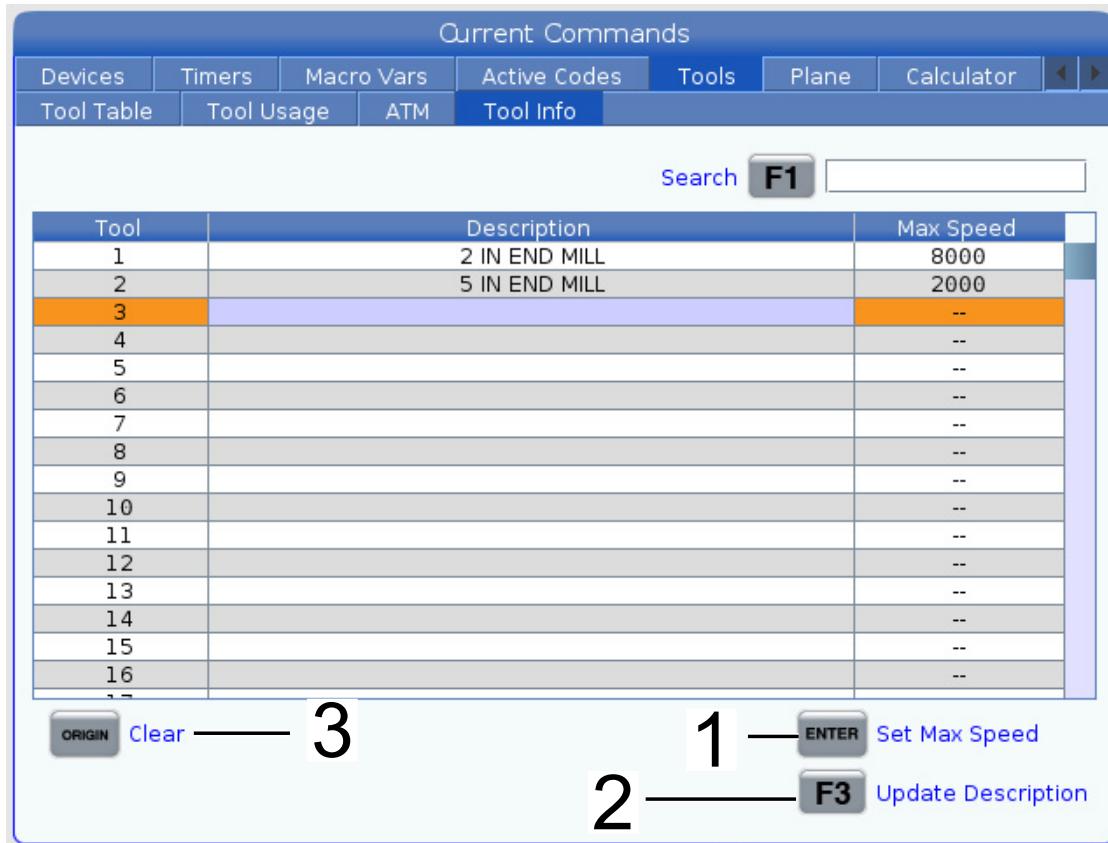
Macro Variable	Function
#8608	Set the desired tool
#8609	Current tool number - if result more 0 (the tool was used)
#8610	Total time mentioned in #8609 tool number
#8611	Feed time of mentioned tool number
#8612	Total time
#8605	Next usage of a tool
#8614	Usage start time stamp
#8615	Usage Total time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

#### 4.11.4 Tool Info Display

The Tool Info display tab allows user to specify maximum RPM for a tool. You can also specify a tool description in the same table.

To get to the Tool Info display press [**CURRENT COMMANDS**] then go to Tools and then Tool Info tab.

**F4.15:** Tool Info Display



1. To set the max speed for a tool highlight the tool row and type in the desired max speed and press [**ENTER**].
2. To set the tool description highlight the tool row and type in the description and press [**F3**].
3. To clear the tool max speed or description press [**ORIGIN**].

## 4.12 WorkHolding Tab- Overview

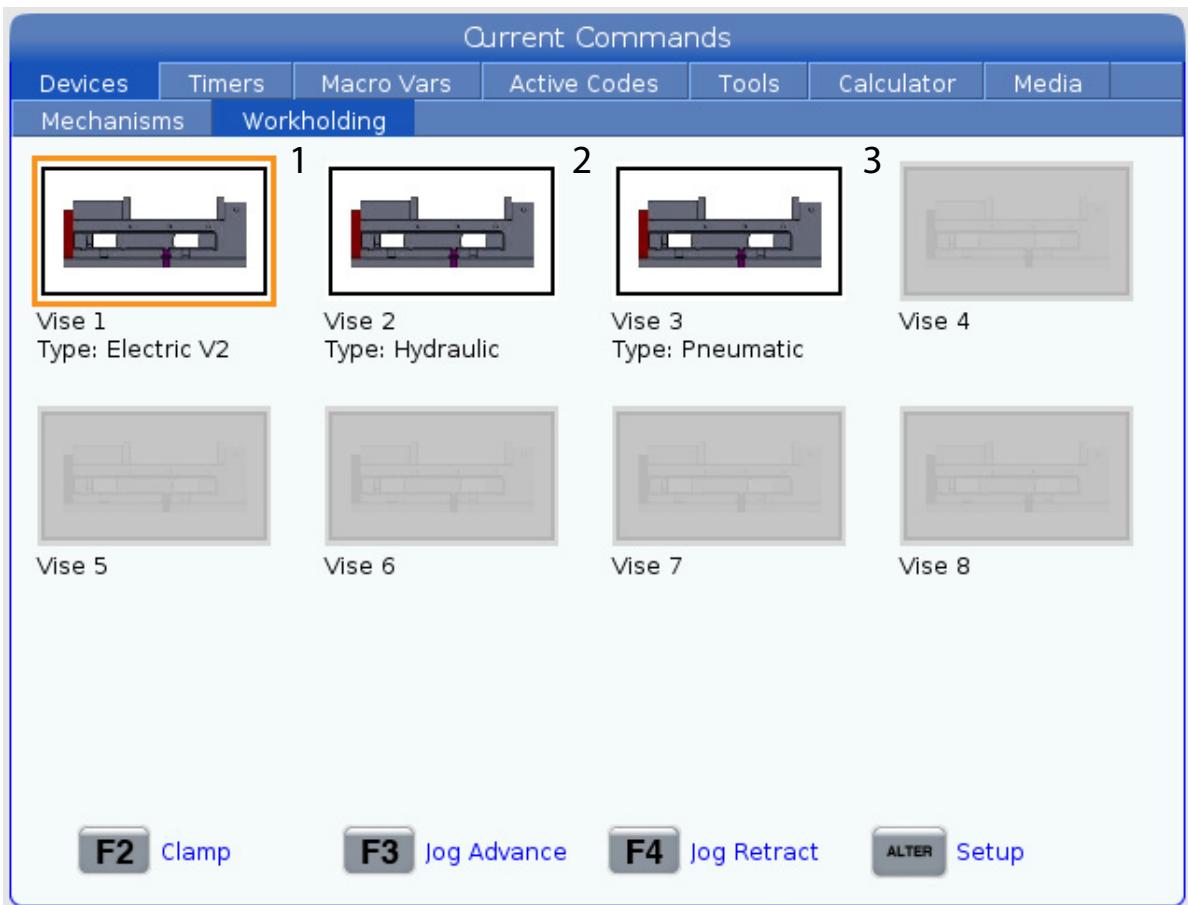
Starting in software version 100.20.000.1110 a workholding tab was added to the control to support multiple workholding devices. The control will support Haas E-Vise [1], Hydraulic [2] and Pneumatic [3] vises.

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


**NOTE:**

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

**F4.16:** Workholding Display Tab - Electric [1], Hydraulic [2], and Pneumatic [3] Vises



You can actuate up to 8 workholding devices.

To access the Workholding page press **[CURRENT COMMANDS]** and navigate to Devices > Workholding.

From the Workholding display tab you will be able to:

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

### 4.12.1 E-Vise Setup

To enable the E-Vise press **[CURRENT COMMANDS]** and navigate to Devices > Workholding.

Select one of the workholding devices and press **[ALTER]** to enter the setup page.

## F4.17: Electric Vise Setup page

**Current Commands**

Devices	Timers	Macro Vars	Active Codes	Tools	Plane	Calculator	◀	▶
Mechanisms	Automatic Part Loader	Robot	Workholding					
Name				Value		Units		
1 Workholding 1		>		Haas Electric V2				
2 Vise Clamping Time					4.0	Sec		
3 Vise Unclamping Time					4.0	Sec		
4 Electric Vise Clamp Strength					6			
5 Electric Vise Clamp Type		>			O.D.			

**Description**

Defines the type of vise that is associated with the selected workholding.  
Please unclamp workholding and remove any material.  
Press [E-STOP] and select the workholding type.

**ALTER** [To vises view](#)

1. Workholding - This setting defines the type of vise that is associated with the selected workholding. Press **[E-STOP]** and set this setting to Haas Electric V2.
2. Vise Clamping Time - This setting defines amount of time the vise will move towards the clamp position. If the E-Vise does not detect a part within the specified time the machine will generate an alarm.

**NOTE:**

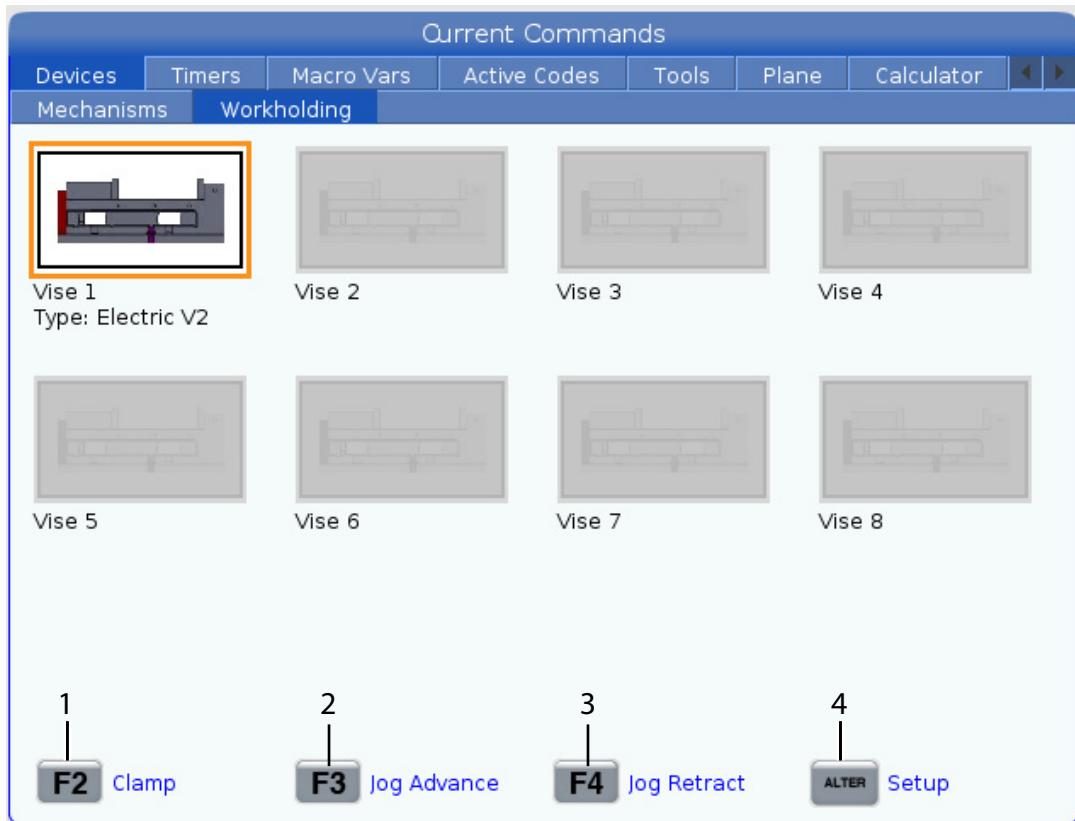
*This setting does not affect jogging.*

3. Vise Unclamping Time- This setting defines the amount of time the vise will move in the unclamp direction.
4. Electric Vise Clamp Strength - This setting only applies to E-vises. This setting defines the amount of force used during clamping. The value specified in this setting will determine which pair of voltage % and Current limit parameters will be used.

5. Electric Vise Clamp Type - This setting only applies to E-vises. This setting defines whether the workholding clamps the Outer Diameter (O.D) or the inner diameter (I.D) of a workpiece.

Press **[ALTER]** to exit the setup mode.

### F4.18: Workholding - Electric Vise Display



1. Clamp/Unclamp - Press **[F2]** to Clamp or Unclamp the E-Vise.  
Use M70 P1 / M71 P1 in the program to clamp or unclamp the E-vise.  
For more information refer to "M70 Workholding Clamp / M71 Workholding Unclamp" on page 422.
2. Jog Advance - Press **[F3]** to advance the E-vise to the clamp direction.
3. Jog Retract - Press **[F4]** to Retract the E-vise to the unclamp direction.
4. Setup - Press **[ALTER]** to enter the E-vise setup page.

## 4.12.2 Hydraulic Vise - Setup

To enable a Hydraulic Vise press **[CURRENT COMMANDS]** and navigate to Devices > Workholding.

### F4.19: Hydraulic Vise Setup page

	Name	Value	Units
1	Workholding 1	>	Custom Hydraulic Vise
2	Vise Clamping Time		4.0 Sec
3	Vise Unclamping Time		4.0 Sec
	Electric Vise Clamp Strength		6
	Clamp Type	>	O.D.

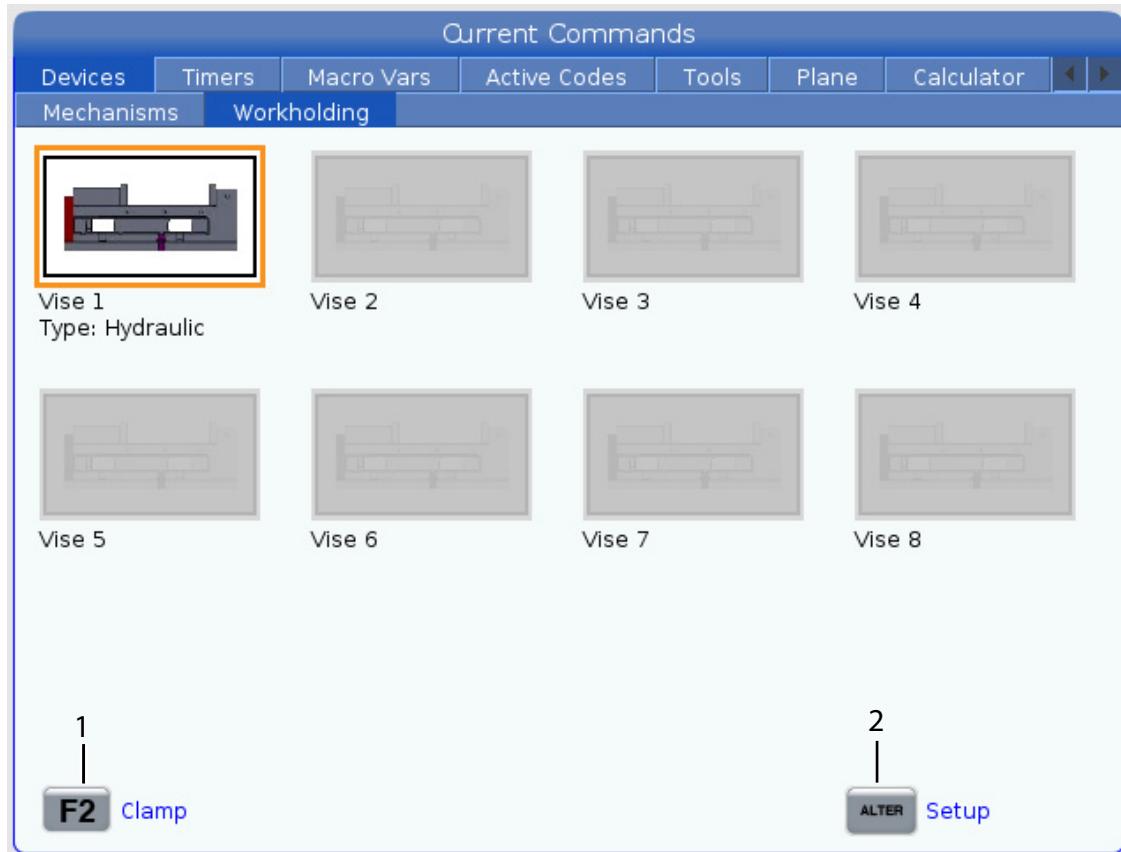
**Description**

Defines the type of vise that is associated with the selected workholding.  
Please unclamp workholding and remove any material.  
Press [E-STOP] and select the workholding type.

4 — To vises view

1. Workholding - This setting defines the type of vise that is associated with the selected workholding. Press **[E-STOP]** and set this setting to Custom Hydraulic Vise.
2. Vise Clamping Time - This setting defines amount of time the control waits for the vise to be considered clamped.
3. Vise Unclamping Time- This setting defines the amount of time the control waits for the vise to be considered unclamped.
4. Press **[ALTER]** to exit the setup mode.

### F4.20: Workholding - Hydraulic Vise Display



1. Clamp/Unclamp - Press **[F2]** to Clamp or Unclamp the Vise.  
In the program use M70 P1 to clamp and M71 P1 the vise. When commanding the M70 Pn / M71 Pn code the control will toggle an I/O output. Make sure the solenoid valve is connected to the correct output see the table below.

### T4.4: Hydraulic / Pneumatic Vise Outputs

M-Code Clamp / Unclamp	I/O Output (see note)
M70 P1 (Clamp) / M71 P1 (Unclamp)	176 - User Vise 1 Unclamp
M70 P2 (Clamp) / M71 P2 (Unclamp)	102 - User Vise 2 Unclamp
M70 P3 (Clamp) / M71 P3 (Unclamp)	121 - User Vise 3 Unclamp
M70 P4 (Clamp) / M71 P4 (Unclamp)	123 - User Vise 4 Unclamp

M-Code Clamp / Unclamp	I/O Output (see note)
M70 P5 (Clamp) / M71 P5 (Unclamp)	124 - User Vise 5 Unclamp
M70 P6 (Clamp) / M71 P6 (Unclamp)	125 - User Vise 6 Unclamp
M70 P7 (Clamp) / M71 P7 (Unclamp)	131 - User Vise 7 Unclamp
M70 P8 (Clamp) / M71 P8 (Unclamp)	132 - User Vise 8 Unclamp



**NOTE:** *The output will show a 1 when the user vise is unclamped and 0 when clamped.*

For more information refer to “M70 Workholding Clamp / M71 Workholding Unclamp” on page 422.

2. Setup - Press **[ALTER]** to enter the Hydraullic vise setup page.

#### **UMC Procedures:**

The UMC Hydraulic Workholding / Vise installation procedures can be found on the Haas Service website. You can also scan the codes below with your mobile device to go directly to the procedure.

The AD0499 procedure shows you how to install the UMC Hydraulic Workholding union.

**F4.21:** UMC Hydraulic Workholding - Installation - AD0499



The AD0514 procedure shows you how to install and remove a Hydraulic Vise on a UMC.

**F4.22:** UMC Hydraulic Vise - Installation/ Removal - AD0514



### 4.12.3 Pneumatic Vise - Setup

To enable a Pneumatic Vise press **[CURRENT COMMANDS]** and navigate to Devices > Workholding.

**F4.23:** Pneumatic Vise Setup page

**Current Commands**

Devices	Timers	Macro Vars	Active Codes	Tools	Plane	Calculator																				
Mechanisms	Automatic Part Loader	Robot	Workholding				◀	▶																		
<table border="1" style="width: 100%; border-collapse: collapse;"> <thead> <tr> <th style="width: 30%;">Name</th> <th style="width: 30%;">Value</th> <th style="width: 40%;">Units</th> </tr> </thead> <tbody> <tr> <td>1 Workholding 1</td> <td>&gt;</td> <td>Custom Pneumatic Vise</td> </tr> <tr> <td>2 Vise Clamping Time</td> <td></td> <td>4.0 Sec</td> </tr> <tr> <td>3 Vise Unclamping Time</td> <td></td> <td>4.0 Sec</td> </tr> <tr> <td>Electric Vise Clamp Strength</td> <td></td> <td>6</td> </tr> <tr> <td>Electric Vise Clamp Type</td> <td>&gt;</td> <td>O.D.</td> </tr> </tbody> </table>									Name	Value	Units	1 Workholding 1	>	Custom Pneumatic Vise	2 Vise Clamping Time		4.0 Sec	3 Vise Unclamping Time		4.0 Sec	Electric Vise Clamp Strength		6	Electric Vise Clamp Type	>	O.D.
Name	Value	Units																								
1 Workholding 1	>	Custom Pneumatic Vise																								
2 Vise Clamping Time		4.0 Sec																								
3 Vise Unclamping Time		4.0 Sec																								
Electric Vise Clamp Strength		6																								
Electric Vise Clamp Type	>	O.D.																								

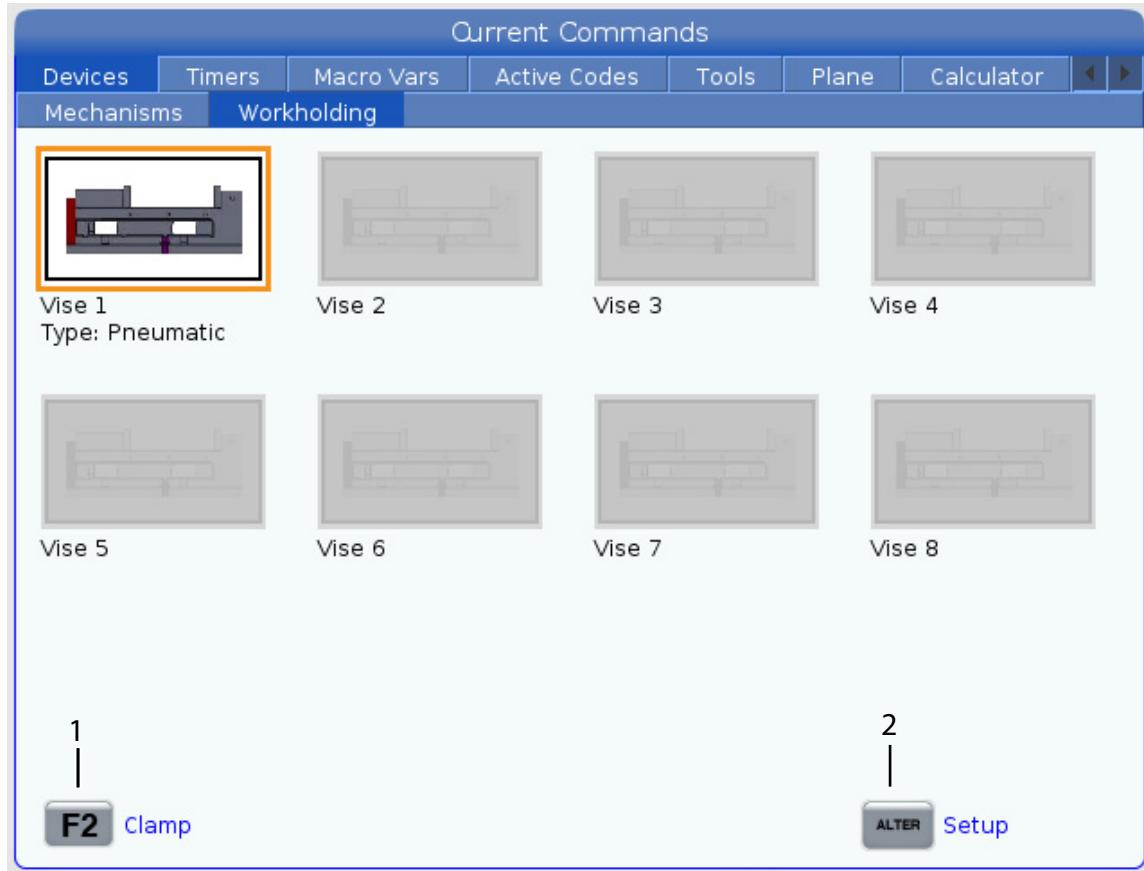
**Description**

Defines the type of vise that is associated with the selected workholding.  
Please unclamp workholding and remove any material.  
Press **[E-STOP]** and select the workholding type.

4 —  To vises view

1. Workholding - This setting defines the type of vise that is associated with the selected workholding. Press **[E-STOP]** and set this setting to Custom Pneumatic Vise.
2. Vise Clamping Time - This setting defines amount of time the control waits for the vise to be considered clamped.
3. Vise Unclamping Time- This setting defines the amount of time the control waits for the vise to be considered unclamped.
4. Press **[ALTER]** to exit the setup mode.

### F4.24: Workholding - Pneumatic Vise Display



1. Clamp/Unclamp - Press **[F2]** to Clamp or Unclamp the Vise.  
In the program use M70 P1 to clamp and M71 P1 the vise. When commanding the M70 Pn / M71 Pn code the control will toggle an I/O output. Make sure the solenoid valve is connected to the correct output see the table below.

### T4.5: Hydraulic / Pneumatic Vise Outputs

M-Code Clamp / Unclamp	I/O Output (see note)
M70 P1 (Clamp) / M71 P1 (Unclamp)	176 - User Vise 1 Unclamp
M70 P2 (Clamp) / M71 P2 (Unclamp)	102 - User Vise 2 Unclamp
M70 P3 (Clamp) / M71 P3 (Unclamp)	121 - User Vise 3 Unclamp
M70 P4 (Clamp) / M71 P4 (Unclamp)	123 - User Vise 4 Unclamp

M-Code Clamp / Unclamp	I/O Output (see note)
M70 P5 (Clamp) / M71 P5 (Unclamp)	124 - User Vise 5 Unclamp
M70 P6 (Clamp) / M71 P6 (Unclamp)	125 - User Vise 6 Unclamp
M70 P7 (Clamp) / M71 P7 (Unclamp)	131 - User Vise 7 Unclamp
M70 P8 (Clamp) / M71 P8 (Unclamp)	132 - User Vise 8 Unclamp



**NOTE:** *The output will show a 1 when the user vise is unclamped and 0 when clamped.*

For more information refer to “M70 Workholding Clamp / M71 Workholding Unclamp” on page 422.

2. Setup - Press **[ALTER]** to enter the Hydrualic vise setup page.

## 4.13 Tool Changers

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

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

### 4.13.1 Loading the Tool Changer



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

**NOTE:**

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

**WARNING:**

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

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

**CAUTION:**

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

## Tool Table

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



**NOTE:**

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

1. To access the Tool Pocket Table, press [**CURRENT COMMANDS**] and choose the **Tool Table** tab.

**F4.25:** Tool Table Display

Active Tool			31	—	2	Next Pocket			2	—	3
Pocket	Category	Tool									
Spindle		31									
1		13									
2*	11	12									
3		8									
4		4									
5		9									
6		7									
7		22									
8		15									
9		5									
10		3									
11		6									
12		14									
13		16									
14		30									
15		10									
16		17									
17		18									
18		19									
19		23									
20		20									

\* Indicates Current Tool Changer Pocket  
Green indicates a large pocket. Yellow indicates an extra large pocket.

1. **Tool Table** tab (highlighted in orange).

2. Set pocket as large [L]

3. Set pocket as heavy [H]

4. Set pocket as XL [X]

5. Clear category [SPACE]

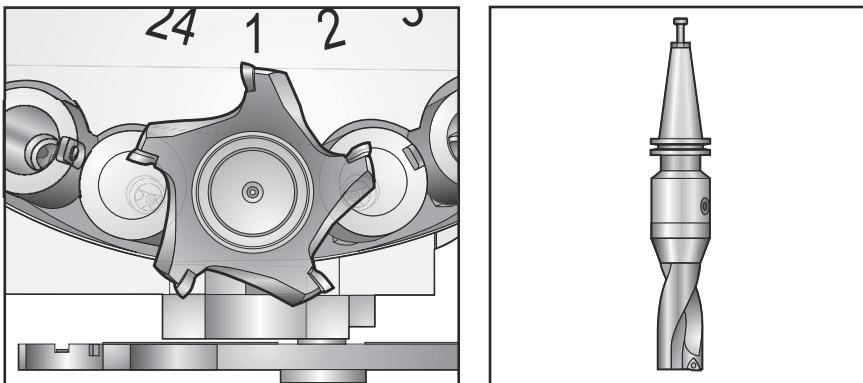
6. Set tool [###] + [ENTER]

7. Clear tool [0] + [ENTER]

8. Reset table [ORIGIN]

2. **Active Tool**- Tells you the tool number that is installed in the spindle.
3. **Active Pocket** - this show you the next pocket number.

F4.26: A Large (left), and a Heavy (not Large) Tool (right)



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



**CAUTION:**

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

5. **Set pocket as heavy [H]** - Use this flag when a Heavy, Small Diameter 40-taper tool (4 lb or heavier) or a 50-taper tool (12 lb or heavier) is loaded into the spindle. Scroll to the pocket of interest and press **[H]** to set the flag.
6. **Set pocket as XL [X]** - Use this flag when two adjacent pockets at each side of the tool are needed. Scroll to the pocket of interest and press **[X]** to set the flag.



**NOTE:**

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

7. **Clear category [Space]** - Highlight the desired tool and press **[SPACE]** to clear the flag.
8. **Set tool [###] + [Enter]** - Highlight the desired pocket and type in the tool number + **[ENTER]** to set the desired tool number.



**NOTE:**

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

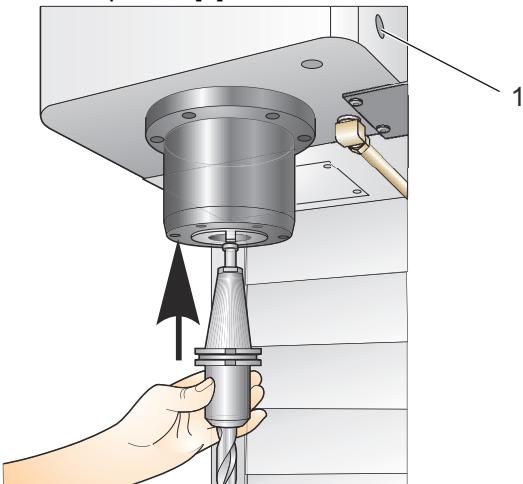
9. **Clear tool [0] + [Enter]**- Highlight the desired pocket and press **[0] + [ENTER]** to clear out the tool number.
10. **Reset table [Origin]** - Press **[ORIGIN]** with the cursor in the center column to use the **[ORIGIN]** menu. This menu lets you:
  - **Sequence All Pockets** - Makes all of the tool numbers sequential based on their pocket location, starting with 1.
  - **Zero All Pockets** - Removes all of the tool numbers from all of the pocket numbers.
  - **Clear Category Flags** - Removes the category designations from all of the tools.
11. \* Indicates current tool changer pocket.

## Tool Loading for a Side-Mount Tool Changer

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

1. Make sure your tool holders have the correct pull stud type for the mill.
2. Press **[CURRENT COMMANDS]** then navigate to the **TOOL TABLE** tab and press the **[DOWN]** cursor. Refer to the “Tool Table” on page 145 to set the correct tool information into the tool table.
3. Insert tool 1 (pull stud first) into the spindle.

**F4.27:** Inserting a Tool Into the Spindle: [1] Tool release button.



4. Turn the tool so that the two cutouts in the tool holder line up with the tabs of the spindle.
5. Push the tool upward and press the tool release button.
6. When the tool is fitted into the spindle, release the tool release button.

## High-Speed Side-Mount Tool Changer

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

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

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

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

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

Large tools are assumed to be heavy.

Heavy tools are not assumed to be large.

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

## Using '0' for a Tool Designation

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

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

## Moving Tools in the Carousel

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



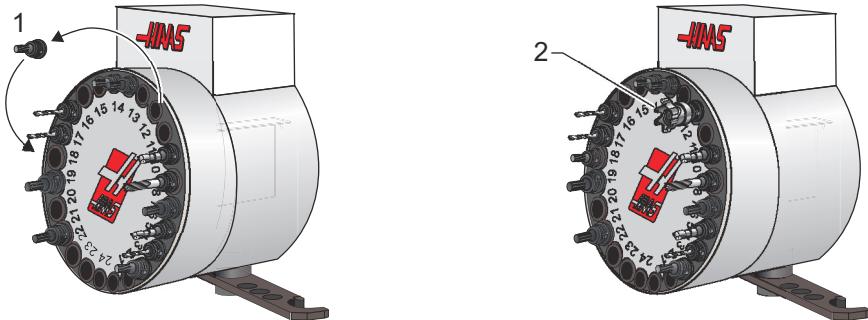
**CAUTION:**

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

## Moving Tools

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

- F4.28:** Making Room for Large Tools: [1] Tool 12 to Pocket 18, [2] Large Tool in Pocket 12.



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



**NOTE:**

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

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

## Umbrella Tool Changer

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

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

### 4.13.2 Umbrella Tool Changer Recovery

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



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

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

### 4.13.3 SMT Programming Notes

#### Tool Pre-Call

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

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

Program commands that stop look-ahead:

- Work offset selections ( G54, G55, etc.)
- G103 Limit Block Buffering, when programmed without a P address or with a nonzero P address
- M01 Optional Stop
- M00 Stop Program

- Block Delete Slashes ( / )
- A large number of program blocks executed at high speed

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

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

#### 4.13.4 SMTCA Recovery

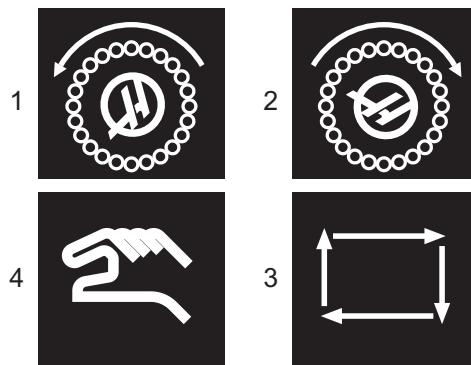
If a problem occurred during a tool change, a tool changer recovery needs to be performed. Enter the tool changer recovery mode by:

1. Press **[RECOVER]** and navigate to the **TOOL CHANGER RECOVERY** tab.
2. Press **[ENTER]**. If there is no Alarm, the control first attempts an automatic recovery. If there is an alarm press **[RESET]** to clear the alarms and repeat from step 1.
3. At the **VMSTC TOOL RECOVERY** screen, press **[A]** to begin automatic recovery or **[E]** to exit.
4. If the automatic recovery fails, press **[M]** to continue for a manual recovery.
5. In manual mode, follow the instructions and answer the questions to perform a proper tool changer recovery. The entire tool changer recovery process must be completed before exiting. Start the routine from the beginning if you exit the routine early.

## 4.13.5 SMTc Door Switch Panel

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

- F4.29:** Tool Changer Door Switch Panel Symbols: [1] Rotate Tool changer Carousel Counter-Clockwise, [2] Rotate Tool Changer Carousel Clockwise, [3] Tool Change Switch - Automatic Operation, [4] Tool Change Switch - Manual Operation Selection.



### SMTc Door Operation

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

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

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

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

## 4.14 Pallet Changer - Introduction

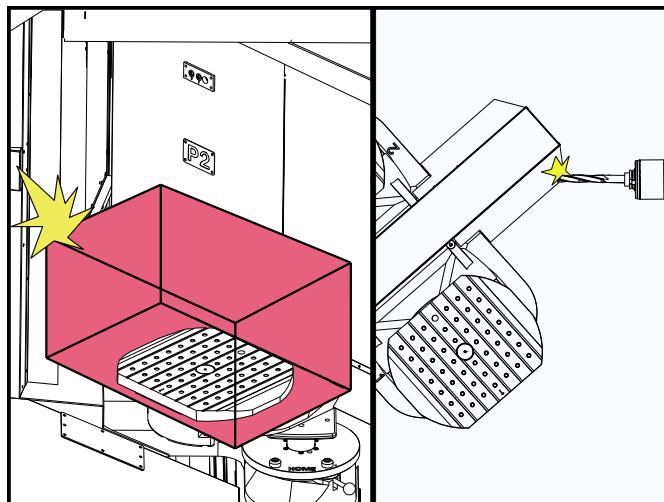
The pallet changer is commanded through a CNC program. The M50 (execute pallet change) function consists of unlocking, lifting and rotating the pallets, then lowering and locking the pallets again. The pallet changer rotates the pallets 180°, then back; it does not continuously rotate in the same direction.

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

### 4.14.1 Pallet Changer Warnings and Cautions

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

F4.30: EC-400 Shown



### 4.14.2 Maximum Pallet Loads

EC-400 - Full 4th Axis - 1000lbs per pallet

### 4.14.3 Operator Load Station (EC-400)

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



**NOTE:**

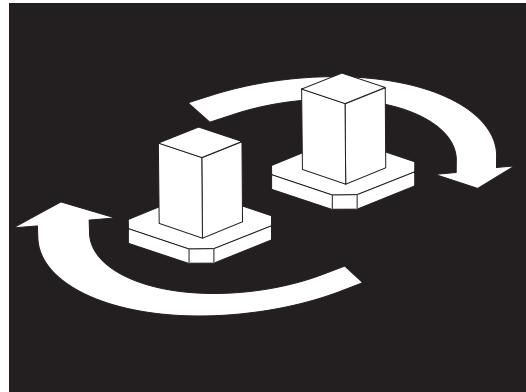
*The load station pallet must be at home to do a pallet change.*

### 4.14.4 Sub-Panel Controls

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

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

F4.31: Pallet Ready Button Symbol



### 4.14.5 Pallet Replacement

The pallets can be loaded into the mill through the load station. Note the orientation of the pallet; the pallet can only be loaded one way. The locating holes on the pallets are drilled on the rear of the pallet, lining up with the pins in the APC.

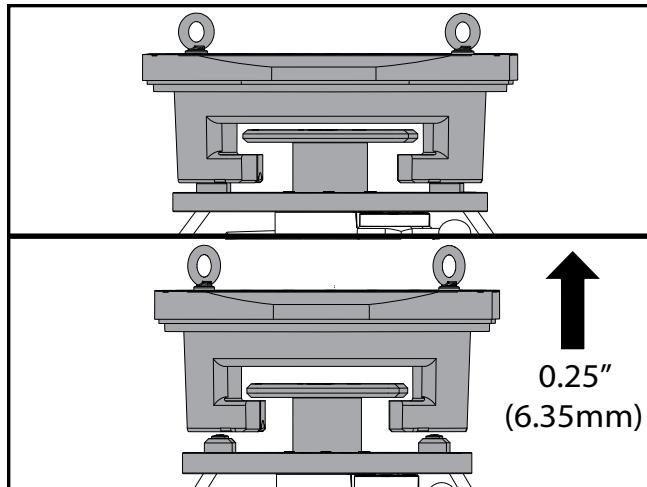


**NOTE:**

*On 2 pallet machines the engraved arrow points towards the operator (out) when in the home position. On a pallet pool machine, the arrow points away from the operator (in).*

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

2. Attach a suitable lifting device to the top of the fixture or use eye bolts threaded into the pallet holes.



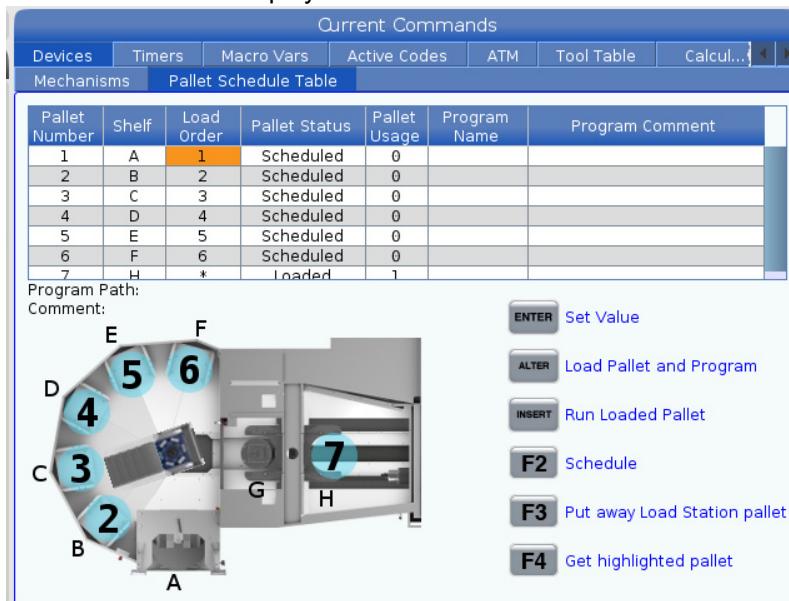
3. Lift the pallet approximately 0.25" (6.35mm) to position it above the load station pins, but below the load station lock plate. Pull the pallet towards you until it has cleared the load station.

#### 4.14.6 Pallet Storage

When removing the pallet, be sure to set it on a soft surface such as a wooden pallet. The bottom side of the pallet has machined surfaces that must be protected. Spray a light coat of oil on the top and bottom of the pallet to protect from rust.

## 4.14.7 Pallet Schedule Table

### F4.32: Pallet Schedule Table - Display



The Pallet Schedule Table incorporates a number of features to help the user with their routine.

**Load Order and Pallet Status:** These two features work together to show which pallet is currently in the machining area. Enter a number for the load order and press **[ENTER]** on the Pallet Status field to select the pallet status. The choices are: 0: Unscheduled, 1: Scheduled, 2: Missing and 3: Completed.

**Comment:** To add a user comment to a pallet, highlight the pallet number field and press **[ENTER]**. A box will appear, type in the desired comment and press **[ENTER]**.

**Pallet Usage:** This feature gives the number of times the specific pallet has been loaded into the machining area. Press **[ORIGIN]** button to clear the value.

**Program Number:** This detail shows which program number has been assigned to the pallet. To select a program, highlight the Program Name field then press **[ENTER]** and navigate to the program.

**Program Comment:** This area displays the comments that are written in the part program. This can only be changed by editing the comments in the program.

Command Prompts:

**[ENTER]** The function changes depending where the highlighter is located. It is used to enter a user comment, set a value in the field and to view options for the field.

**[ALTER]** Load Pallet and Program. This will load the selected pallet into the machine and call the assigned program to memory.

**[INSERT]** Run loaded program. This will start the machine running in Pallet Scheduler Mode. The machine will continue to run all scheduled pallets in the PST until they are completed. For more information on Pallet Scheduler Mode refer to M199 Pallet / Part Load or Program End in the M-code section.

**[F2]** Schedule Load Station pallet. This will set the status of the Load Station pallet to Scheduled.

**[F3]** Put away Load Station pallet. This will return the Load Station pallet to the pallet pool.

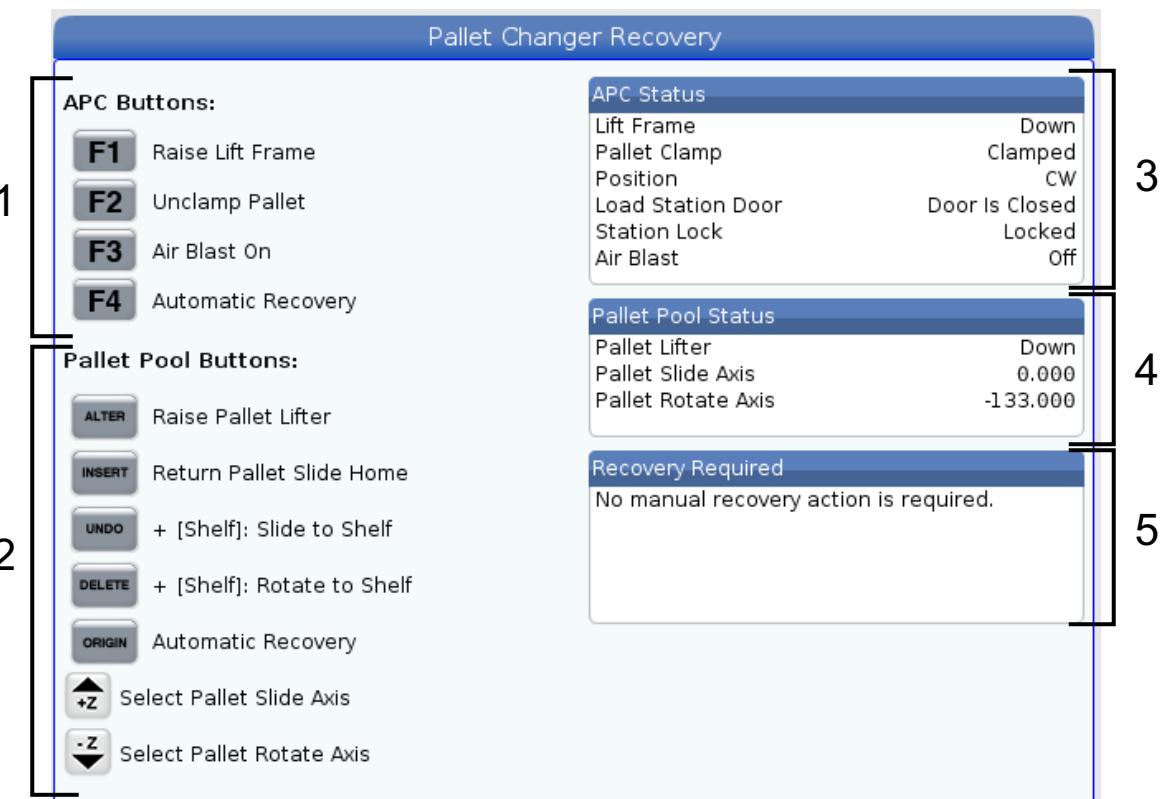
**[F4]** Get highlighted pallet. This will bring the selected pallet to the Load Station.

#### 4.14.8 Pallet Pool/Changer Recovery

If the Pallet Pool or Pallet Changer cycles get interrupted you must enter **[RECOVER]** mode to correct or complete the cycle.

Press the **[RECOVER]** button and press 2 for Pallet Changer. The recovery page will display the input and output values.

- F4.33:** Pallet Changer Recovery Display: APC Functions [1], Pallet Pool Functions [2], APC status [3], Pallet Pool Status [4], Message Box [5].



Command Prompts:

**[F1]** APC Up. This will lift the pallet changer H-Frame if the pallet is not clamped.

**[F2]** Unclamp. This will unclamp the pallet in the machine from the receiver.

**[F3]** Air Blast. This will activate the air blast under the pallet to remove any chips or debris.

**[F4]** Attempt Auto Recovery. This will attempt to automatically correct or complete the pallet changer or pallet pool operation.

**[ALTER]** Raise Pallet Lifter. This will lift the pallet pool lifter assembly.

**[INSERT]** Return Pallet Slide Home. This will return the lifter to its home position.

**[UNDO]** Pallet Pool Slide to Shelf. This will slide the pallet pool lifter to the selected shelf location. Example: Press **[A]** then **[UNDO]** to slide the lifter to shelf position A.

**[DELETE]** Pallet Pool Rotate to Shelf. This will rotate the pallet pool lifter to the selected shelf location. Example: Press **[A]** then **[DELETE]** to rotate the lifter to shelf position A.

**[ORIGIN]** Attempt Auto Recovery. This will attempt to automatically correct or complete the pallet pool operation.

**[+Z]** Select Pallet Slide Axis. This will select the PS axis in handle jog mode.

**[-Z]** Select Pallet Rotate Axis. This will select the PR axis in handle jog mode.

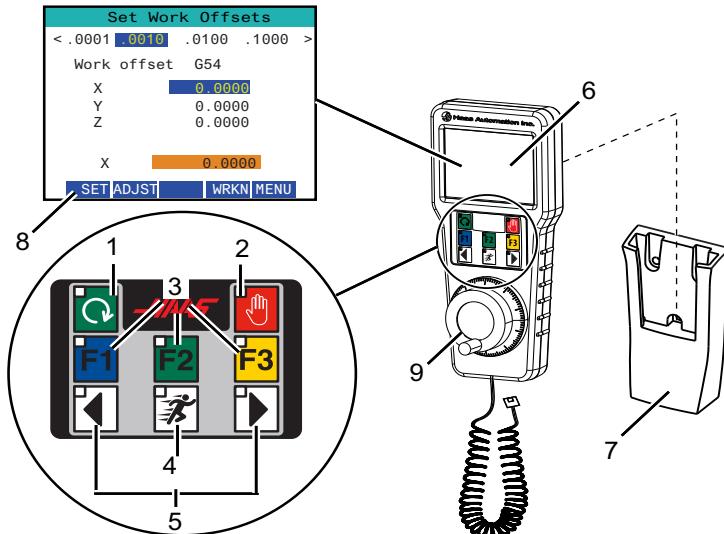
**[Q]** Exit to Pallet Schedule Table. This will take you out of the recovery mode and put you into the pallet schedule table screen.

## 4.15 RJH-Touch Overview

The Remote Jog Handle (RJH-Touch) is an optional accessory that gives you hand-held access to the control for faster and easier setups.

Your machine must have Next Generation Control software 100.19.000.1102 or higher to use all the RJH-Touch functions. The next sections explain how to operate the RJH-Touch.

- F4.34:** Remote Jog Handle [1] Cycle Start Key, [2] Feed Hold Key, [3] Function Keys, [4] Rapid Jog Key, [5] Jog Direction Keys, [6] Touch Screen, [7] Holster, [8] Function Tabs, [9] Handle Jog Wheel.



This illustration shows these components:

1. Cycle Start. Has the same function as **[CYCLE START]** on the control pendant.
2. Feed Hold. Has the same function as **[FEED HOLD]** on the control pendant.
3. Function Keys. These keys are for future use.

4. Rapid Jog button. This key doubles the jogging speed when pressed simultaneously with one of the jog direction buttons.
5. Jog Direction Keys. These keys work the same as keypad jog arrow keys. You can press and hold to jog the axis.
6. LCD Touch Screen Display.
7. Holster. To activate the RJH, lift it out of the holster. To deactivate the RJH, put it back into the holster.
8. Function Tabs. These tabs have different functions in different modes. Press the function tab that corresponds to the function you want to use.
9. Handle Jog Wheel. This Handle Jog works like the jog handle on the control pendant. Each click of the handle jog moves the selected axis one unit of the selected jog rate.

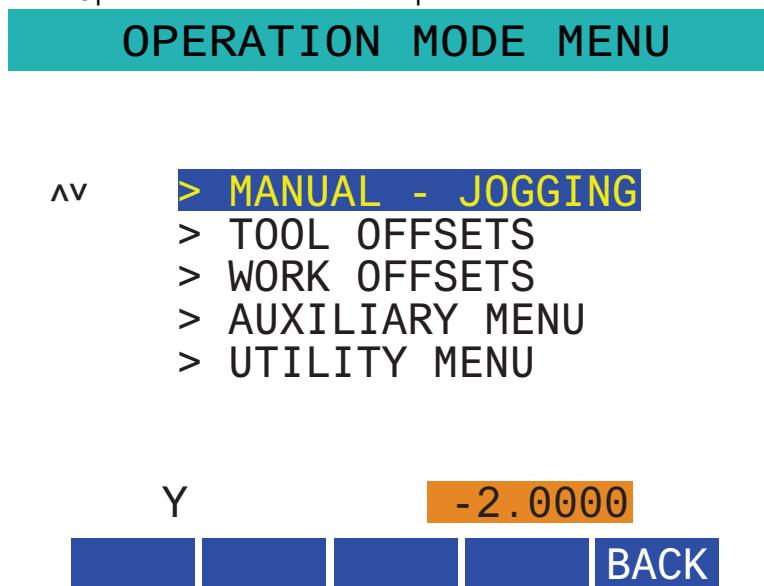
Most RJH functions are available in Handle Jog mode. In other modes, the RJH screen displays information about the active or MDI program.

#### 4.15.1 RJH-Touch Operation Mode Menu

The operation mode menu lets you quickly select the RJH mode. When you select a mode on the RJH, the control pendant also changes to that mode.

Press the [MENU] function key in most RJH modes to access this menu.

**F4.35:** RJH-Touch Operation Mode Menu Example



The menu options are:

- **MANUAL - JOGGING** puts the RJH and the machine control in **HANDLE JOG** mode.
- **TOOL OFFSETS** puts the RJH and the machine control in **TOOL OFFSET** mode.
- **WORK OFFSETS** puts the RJH and the machine control in **WORK OFFSETS** mode.

- **AUXILIARY MENU** brings up the Auxiliary Menu for the RJH.

**NOTE:**

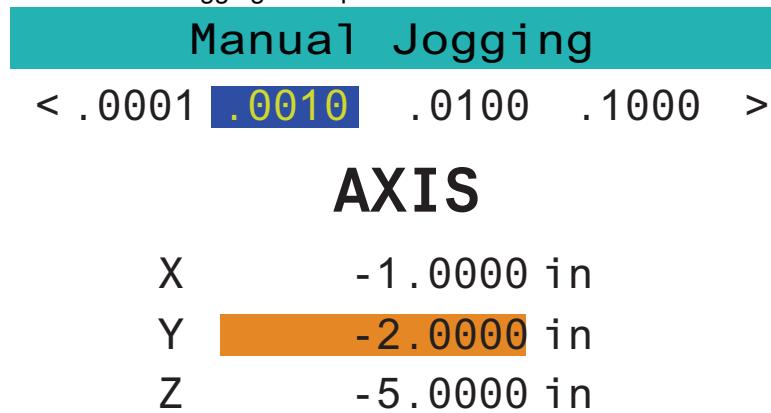
*The flash light feature is not available with the RJH-Touch.*

- **UTILITY MENU** brings up the Utility Menu for the RJH. This menu contains only diagnostic information.

## 4.15.2 RJH-Touch Manual Jogging

The manual jogging screen on the RJH allows you to select the axis and jog rate.

**F4.36:** RJH-Touch Manual Jogging Example.



**WORK** **TO GO** **MACH** **OPER** **MENU**

- Press **[MENU]** on the screen.
- Press **Manual Jogging** on the screen.
- Press the **.0001**, **.0010**, **.0100**, or **.1000** on the screen to change the jog rate.
- Press the axis position on the screen or press **[F1]**/ **[F3]** on the RJH to change the axis.
- Turn the handle jog wheel to jog the axis.
- Press **[WORK]** on the screen to show the Program positions.
- Press **[TO GO]** on the screen to show the Distance to go positions.
- Press **[MACH]** on the screen to show the Machine position.
- Press **[OPER]** on the screen to display the Operator position.

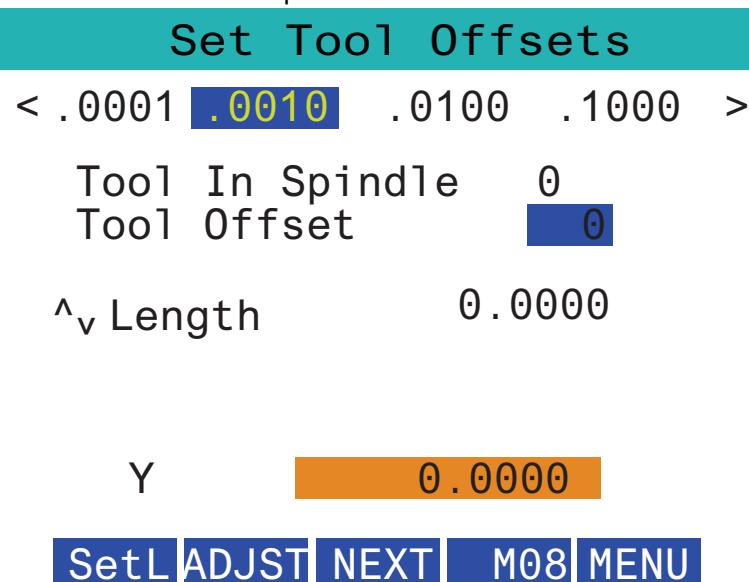
### 4.15.3 Tool Offsets with RJH-Touch

This section describes the controls you use on the RJH to set tool offsets.

For more information about the process of setting tool offsets, refer to page 169.

To access this function on the RJH, press [OFFSET] on the control pendant and select the **Tool Offsets** page, or select **TOOL OFFSETS** from the RJH operation mode menu (refer to page 160).

**F4.37:** RJH Tool Offsets Screen Example



- Press the .0001, .0010, .0100, or .1000 on the screen to change the jog rate.
- Press the axis position on the screen or press [F1]/[F3] on the RJH to change the axis.
- Press the [NEXT] on the screen to change to the next tool.
- To change the tool offset, highlight the **TOOL OFFSET** field and use the handle to change the value.
- Use the jog handle to jog the tool to the desired position. Press the **[SETL]** function key to record the tool length.
- To adjust the tool length, for example, if you want to subtract from the tool length the thickness of the paper you used to touch off the tool:
  - a) Press the **[ADJST]** button on the screen.
  - b) Use the handle jog to change the value (positive or negative) to add to the tool length.
  - c) Press the **[ENTER]** button on the screen.

- If your machine has the Programmable Coolant option, you can adjust the spigot position for the tool. Highlight the **COOLANT POS** field and use the handle jog to change the value. You can use the **[M08]** button on the screen to turn on the coolant and test the spigot position. Press the button on the screen again to turn off the coolant.

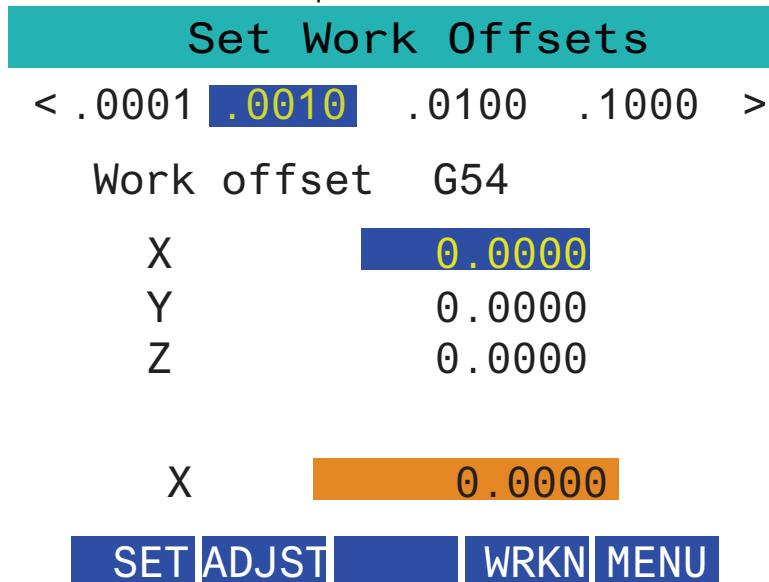
#### 4.15.4 Work Offsets with the RJH-Touch

This section describes the controls you use on the RJH-Touch to set work offsets.

For more information about the process of setting work offsets, refer to page 171

To access this function on the RJH-Touch, press **[OFFSET]** on the control pendant and select the **Work Offsets** page, or select **WORK OFFSETS** from the RJH operation mode menu (refer to page 160).

**F4.38:** RJH Work Offsets Screen Example



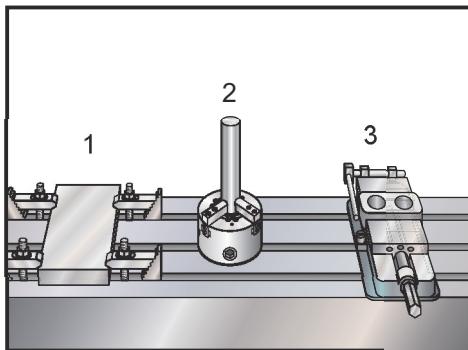
- Press the **.0001**, **.0010**, **.0100**, or **.1000** on the screen to change the jog rate.
- Press the axis position on the screen or press **[F1]/ [F3]** on the RJH to change the axis.
- To change the work offset number, press the **[WRKN]** button on the screen and use the handle jog knob to select a new offset number. Press the **[ENTER]** button on the screen to set the new offset.
- To move the axes use the handle jog wheel.
- When you reach the offset position in an axis, press the **[SET]** button on the screen to record the offset position.

- To adjust an offset value:
  - a) Press the **[ADJST]** function key.
  - b) Use the pulse knob to change the value (positive or negative) to add to the offset.
  - c) Press the **[ENTER]** function key.

## 4.16 Part Setup

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

**F4.39:** Part Setup Examples: [1] Toe clamp, [2] Chuck, [3] Vise.



### 4.16.1 Jog Mode

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

To enter jog mode:

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

### 4.16.2 Setting Offsets

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

## Tool Offsets

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

F4.40: Tool Offset Display

Tool	Work	Length Geometry(H)	Length Wear(H)	Diameter Geometry(D)	Diameter Wear(D)	Coolant Position
1 Spindle	0.	0.	0.	0.	0.	2
2	0.	0.	0.	0.	0.	2
3	0.	0.	0.	0.	0.	2
4	0.	0.	0.	0.	0.	2
5	0.	0.	0.	0.	0.	2
6	0.	0.	0.	0.	0.	2
7	0.	0.	0.	0.	0.	2
8	0.	0.	0.	0.	0.	2
9	0.	0.	0.	0.	0.	2
10	0.	0.	0.	0.	0.	2
11	0.	0.	0.	0.	0.	2
12	0.	0.	0.	0.	0.	2
13	0.	0.	0.	0.	0.	2
14	0.	0.	0.	0.	0.	2
15	0.	0.	0.	0.	0.	2
16	0.	0.	0.	0.	0.	2
17	0.	0.	0.	0.	0.	2
18	0.	0.	0.	0.	0.	2

Enter A Value

TOOL OFFSET MEAS      F1 Set Value      ENTER Add To Value      F4 Work Offset

1. Active Tool: - This tells you which tool is in the spindle.
2. Tool Offset (T) - This is the list of tool offsets. There is a maximum of 200 tool offsets available.
3. Length Geometry (H), Length Wear (H) - These two columns are tied to the G43 (H) values in the program. If you command a  
G43 H01;  
from within a program for tool #1, the program will use the values from these columns.



**NOTE:**

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

4. Diameter Geometry (D), Diameter Wear (D) - These two columns are used for cutter compensation. If you command a  
`G41 D01;`  
from within a program, the program will use the values from these columns.

**NOTE:**

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

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

**NOTE:**

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

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

**F4.41:** Tool Offsets Display Continued. Press the [RIGHT] arrow key to display this page.

Offsets

Tool	Work	7	8	9	10	11	12
Active Tool: 1							
Tool Offset	Flutes	Actual Diameter	Tool Type	Tool Material	Tool Pocket	Category	
1 Spindle	0	0.	None	User	Spindle		
2	0	0.	None	User	1	*	
3	0	0.	None	User	2		
4	0	0.	None	User	3		
5	0	0.	None	User	4		
6	0	0.	None	User	5		
7	0	0.	None	User	6		
8	0	0.	None	User	7		
9	0	0.	None	User	8		
10	0	0.	None	User	9		
11	0	0.	None	User	10		
12	0	0.	None	User	11		
13	0	0.	None	User	12		
14	0	0.	None	User	13		
15	0	0.	None	User	14		
16	0	0.	None	User	15		
17	0	0.	None	User	16		
18	0	0.	None	User	17		

Enter A Value

 Tool Offset Measure
**F1** Set Value
**ENTER** Add To Value
**F4** Work Offset

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

**NOTE:**

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

8. Actual Diameter - This column is used by the control to calculate the correct Surface Speed value displayed at the Main Spindle screen.
9. Tool Type - This column is used by the control to decide which probe cycle to use to probe this tool. Press **[F1]** to view the options: None, Drill, Tap, Shell Mill, End Mill, Spot Drill, Ball Nose and Probe. When this field is set to Drill, Tap, Spot Drill, Ball Nose and Probe, the probe will probe along the tool centerline for the length. When this field is set to Shell Mill or End Mill the probe will probe at the tools edge.
10. Tool Material - This column is used for calculations by the VPS feeds and speeds library. Press **[F1]** to view the options: User, Carbide, Steel. Press **[ENTER]** to set the material, or press **[CANCEL]** to exit.
11. Tool Pocket - This column shows you what pocket the tool is currently in. This column is read only.
12. Tool Category - This column shows if the tool is set up as large, heavy or extra large. To make a change, highlight the column and press **[ENTER]**. The Tool Table will be displayed. Follow the instructions on the screen to make tool table changes.

- F4.42:** Tool Offsets Display Continued. Press the [RIGHT] arrow key to display this page. The values in this page are used by the probe.

Offsets					
Tool	Work	13	14	15	16
Active Tool: 1 Coolant Position: 1					
Tool Offset	Approximate Length	Approximate Diameter	Edge Measure Height	Tool Tolerance	Probe Type
1 Spindle	0.	0.	0.	0.	None
2	0.	0.	0.	0.	None
3	0.	0.	0.	0.	None
4	0.	0.	0.	0.	None
5	0.	0.	0.	0.	None
6	0.	0.	0.	0.	None
7	0.	0.	0.	0.	None
8	0.	0.	0.	0.	None
9	0.	0.	0.	0.	None
10	0.	0.	0.	0.	None
11	0.	0.	0.	0.	None
12	0.	0.	0.	0.	None
13	0.	0.	0.	0.	None
14	0.	0.	0.	0.	None
15	0.	0.	0.	0.	None
16	0.	0.	0.	0.	None
17	0.	0.	0.	0.	None
18	0.	0.	0.	0.	None

Enter A Value

Automatic Probe Options  
   Add To Value  

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

**NOTE:**

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

14. Approximate Diameter - This column is used by the probe. The value in this field tells the probe the diameter of the tool.
15. Edge Measure Height - This column is used by the probe. The value in this field is the distance below the tip of the tool that the tool needs to move, when the tool diameter is probed. Use this setting when you have a tool with a large radius or when you are probing a diameter on a chamfer tool.
16. Tool Tolerance - This column is used by the probe. The value in this field is used for checking tool breakage and wear detection. Leave this field blank if you are setting the length and diameter on the tool.

17. Probe Type - This column is used by the probe. You can select the probe routine you want to perform on this tool.  
 The choices are: 0 - No tool probing to be performed., 1- Length probing (Rotating),, 2 - Length probing (Non-Rotating),, 3 - Length and Diameter probing (Rotating).  
 Press [TOOL OFFSET MEASURE] to set automatic probe options.

## Set a Tool Offset

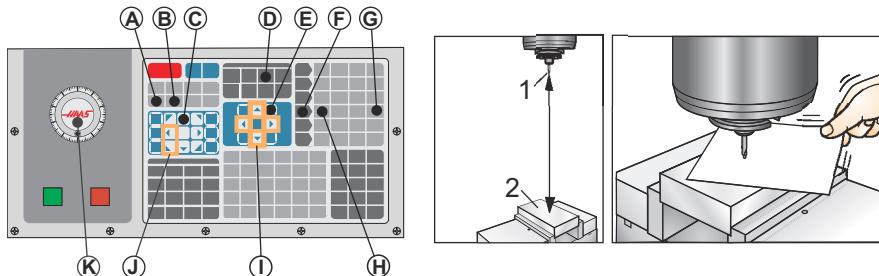
The next step is to touch off the tools. This defines the distance from the tip of the tool to the top of the part. Another name for this is Tool Length Offset, which is designated as **H** in a line of machine code. The distance for each tool is entered into the **TOOL OFFSET** table.



**NOTE:**

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

- F4.43:** Setting Tool Offset. With the Z Axis at its home position, Tool Length Offset is measured from the tip of the tool [1] to the top of the part [2].



1. Load the tool in the spindle [1].
2. Press [**HANDLE JOG**] [F].
3. Press [.1/100.] [G] (The mill moves at a fast rate when the handle is turned).
4. Select between the X and Y axes [J], and with the use the jog handle [K] to move the tool near the center of the part.
5. Press [+Z] [C].
6. Jog the Z Axis approximately 1" above the part.
7. Press [.0001/.1] [H] (The mill moves at a slow rate when the handle is turned).
8. Place a sheet of paper between the tool and the work piece. Carefully move the tool down to the top of the part, as close as possible, and still be able to move the paper.
9. Press [OFFSET] [D] and select the **TOOL** tab.
10. Highlight the **H (length) Geometry** value for position #1.
11. Press [TOOL OFFSET MEASURE] [A]. A pop box may appear if the change is Greater Than Setting 142! Accept (Y/N). Press [Y] to accept.



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

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

## Work Offsets

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

**F4.44:** Work Offsets Display

The screenshot shows a software interface titled "Offsets". At the top, there are tabs for "Tool" and "Work", with "Work" being the active tab. Below the tabs is a table with three columns: "G Code", "X Axis", and "Y Axis". The "Z Axis" column is partially visible on the right. The first row of the table is highlighted in orange and contains "G52". The subsequent rows are greyed out, showing other G codes like G54, G55, etc. To the right of the table, a column labeled "Work Material" shows "No Material Selected" for all entries. At the bottom of the table, there are several buttons: "F1 To view options.", "F3 Probing Actions", "F4 Tool Offsets", "Enter A Value", and "ENTER Add To Value".

G Code	X Axis	Y Axis	Z Axis	Work Material
G52	0.	0.	0.	No Material Selected
G54	0.	0.	0.	No Material Selected
G55	0.	0.	0.	No Material Selected
G56	0.	0.	0.	No Material Selected
G57	0.	0.	0.	No Material Selected
G58	0.	0.	0.	No Material Selected
G59	0.	0.	0.	No Material Selected
G154 P1	0.	0.	0.	No Material Selected
G154 P2	0.	0.	0.	No Material Selected
G154 P3	0.	0.	0.	No Material Selected
G154 P4	0.	0.	0.	No Material Selected
G154 P5	0.	0.	0.	No Material Selected
G154 P6	0.	0.	0.	No Material Selected
G154 P7	0.	0.	0.	No Material Selected
G154 P8	0.	0.	0.	No Material Selected
G154 P9	0.	0.	0.	No Material Selected
G154 P10	0.	0.	0.	No Material Selected
G154 P11	0.	0.	0.	No Material Selected

1. G Code - This column displays all the available work offset G-codes. For more information on these work offsets See "G52 Set Work Coordinate System (Group 00 or 12)" on page 344., See "G54-G59 Select Work Coordinate System #1 - #6 (Group

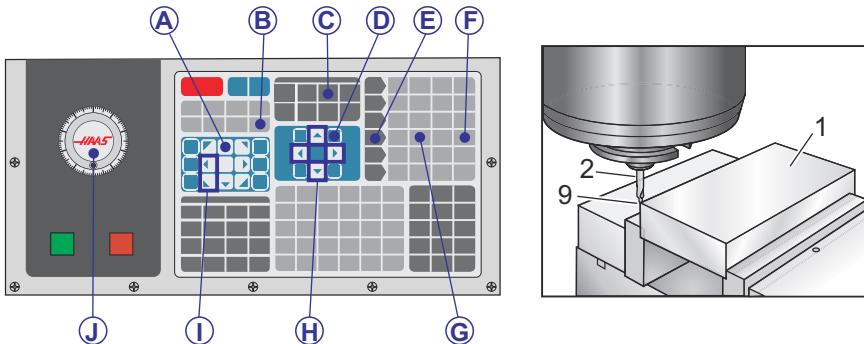
12)" on page 344.. See "G92 Set Work Coordinate Systems Shift Value (Group 00)" on page 366.

2. X, Y, Z, Axis - This column displays the work offset value for each axis. If rotary axis are enabled the offsets for these will be displayed on this page.
3. Work Material - This column is used by the VPS feeds and speeds library.
4. These functions buttons allow you to set the offset values. Type in the desired work offset value and press [F1] to set the value. Press [F3] to set a probing action. Press [F4] to toggle from work to tool offset tab. Type in a value and press [ENTER] to add to the current value.

## Set a Work Offset

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

**F4.45:** Part Zero Set



1. Place the material [1] in the vise and tighten.
2. Load a pointer tool [2] in the spindle.
3. Press [HANDLE JOG] [E].
4. Press [.1/100.] [F] (The mill moves at a fast speed when the handle is turned).
5. Press [+Z] [A].
6. Use the jog handle [J] to move the Z-Axis approximately 1" above the part.
7. Press [.001/1.] [G] (The mill moves at a slow speed when the handle is turned).
8. Jog the Z-Axis approximately. 0.2" above the part.
9. Select between the X and Y axes [I] and jog the tool to the upper left corner of the part (see illustration [9]).
10. Navigate to the [OFFSET]>WORK [C] tab and press the [DOWN] cursor key [H] to activate the page. You can press [F4] to toggle between Tool offsets and Work offsets.
11. Navigate to the G54 X Axis location.

**CAUTION:**

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

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

## 4.17 Run-Stop-Jog-Continue

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

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

**CAUTION:**

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

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



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

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

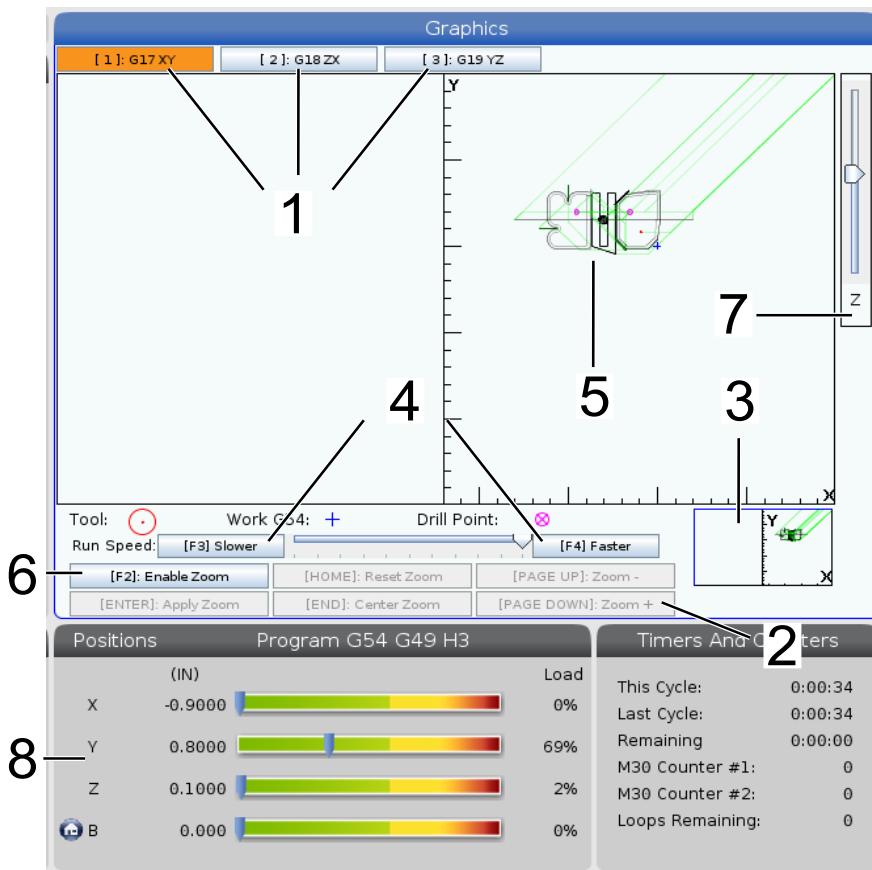


**CAUTION:** *If Setting 36 is ON, the control scans the program to make sure the machine is in the correct state (tools, offsets, G- and M-codes, etc.) to safely continue the program. If Setting 36 is OFF, the control does not scan the program. This can save time, but it could cause a crash in an unproven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

## 4.18 Graphics Mode

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

**F4.46:** Graphics Mode Display



- Axis Planes** Press **[1]** to view the graphics in G17 plane, press **[2]** for G18 plane or press **[3]** to view in G19 plane.
- Key Help Area** The lower-left part of the graphics display pane is the function key help area. This area shows you the function keys that you can use, and a description of what they do.
- Locator Window** The lower-right part of the pane displays the simulated machine table area, and it shows where the simulated view is zoomed and focused.
- Graphics Speed** Press **[F3]** or **[F4]** to run the desired graphics speed.
- Tool Path Window** The large window in the center of the display gives a simulated view of the work area. It displays a cutting tool icon and simulated tool paths.

**NOTE:**

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

**NOTE:**

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

6. **Zoom** Press **[F2]** to display a rectangle (zoom window) that shows the area that the zoom operation will move to. Use **[PAGE DOWN]** to decrease the size of the zoom window (zoom in), and use **[PAGE UP]** to increase the size of the zoom window (zoom out). Use the cursor arrow keys to move the zoom window to the location you want to zoom, and press **[ENTER]** to complete the zoom. The control scales the tool path window to the zoom window. Run the program again to display the tool path. Press **[F2]** and then **[HOME]** to expand the Tool Path window to cover the entire work area.
7. **Z-Axis Part Zero Line** The horizontal line on the Z-Axis bar at the top-right corner of the graphics screen gives the position of the current Z-axis work offset plus the length of the current tool. While a program simulation runs, the shaded portion of the bar indicates the depth of the simulated Z-Axis motion relative to the Z-Axis work zero position.
8. **Position Pane** The position pane displays axis locations just as it would during a live part run.

To run a program in graphics mode:

1. Press **[GRAPHICS]**.
2. Press **[CYCLE START]**.

**NOTE:**

*Graphics mode does not simulate all machine functions or motions.*

## 4.19 More Information Online

For updated information, including tips, tricks, maintenance procedures, and more, visit the Haas Service page at [www.HaasCNC.com](http://www.HaasCNC.com).

For the most current Operator's and Service Manuals scan the code below with your mobile device:



# Chapter 5: Programming

## 5.1 Create / Select Programs for Editing

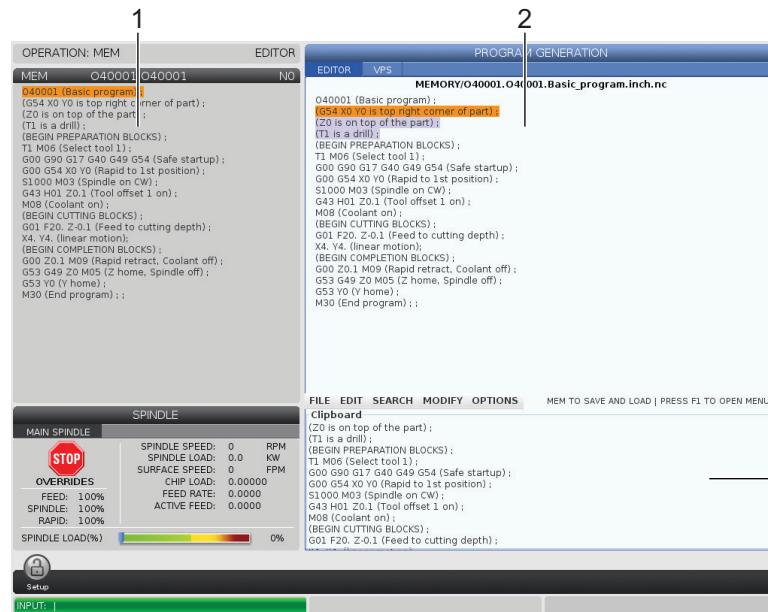
You use the Device Manager (**[LIST PROGRAM]**) to create and select programs for editing. Refer to page 107 to create a new program. Refer to page 109 to choose an existing program to edit.

## 5.2 Program Edit Modes

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

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

- F5.1:** Example Edit Panes. [1] Active Program / MDI Pane, [2] Program Edit Pane, [3] Clipboard Pane



### 5.2.1 Basic Program Editing

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

1. To write a program, or make changes to a program:
  - a. To edit a program in MDI, press **[MDI]**. This is **EDIT:MDI** mode. The program is displayed on the Active pane.
  - b. To edit a numbered program, select it in the Device Manager (**[LIST PROGRAM]**), then press **[EDIT]**. This is **EDIT:EDIT** mode. The program is displayed in the Program Generation pane.
2. To highlight code:
  - a. Use the cursor arrow keys or the jog handle to move the highlight cursor through the program.
  - b. You can interact with single pieces of code or text (cursor highlighting), blocks of code, or multiple blocks of code (block selection). Refer to the Block Selection section for more information.
3. To add code to the program:
  - a. Highlight the code block you want the new code to follow.
  - b. Type the new code.
  - c. Press **[INSERT]**. Your new code appears after the block you highlighted.
4. To replace code:
  - 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:
  - a. Highlight the text you want to delete.
  - b. Press **[DELETE]**. The text that you highlighted is removed from the program.
6. Press **[UNDO]** to reverse up to the last (40) changes.



**NOTE:**

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



**NOTE:**

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

## Block Selection

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

To select a block:

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



**NOTE:**

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



**NOTE:**

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

2. Press **[F2]** to start your selection.
3. Use the cursor arrow keys or the jog handle to expand the selection.
4. Press **[F2]** to complete the selection.

## Actions with a Block Selection

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



**NOTE:**

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



**NOTE:**

*These are actions available in MDI and the Program Editor. You cannot use **[UNDO]** to reverse these actions.*

1. To copy and paste the selection:
  - a. Move the cursor to the location where you want to put a copy of the text.
  - b. Press **[ENTER]**.

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



**NOTE:**

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

2. To move the selection:
  - a. Move the cursor to the location where you want to move the text.
  - b. Press **[ALTER]**.

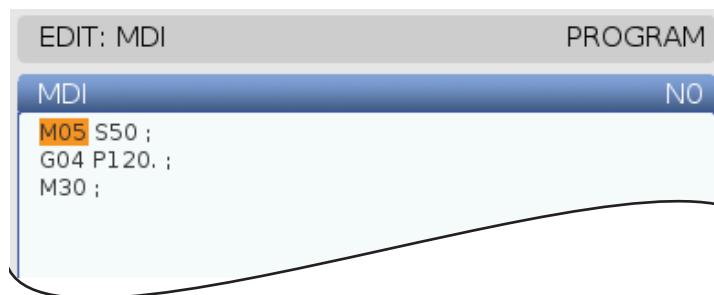
The control removes the text from its current location and puts it in the line after the current line.

3. Press **[DELETE]** to delete the selection.

## 5.2.2 Manual Data Input (MDI)

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

### F5.2: MDI Input Page Example



1. Press **[MDI]** to enter **MDI** mode.
2. Type your 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 put the cursor at the beginning of the program.
  - b. Type a new program number. Program numbers must follow the standard program number format (**O**nnnnn).
  - c. Press **[ALTER]**.
  - d. In the RENAME pop-up window, you can type a filename and file title for the program. Only the O number is required.
  - e. Press **[ENTER]** to save the program to memory.
4. Press **[ERASE PROGRAM]** to delete everything from the MDI input page.

### 5.2.3 Background Edit

Background edit lets you edit a program while a program runs. If you edit the active program, background edit creates a copy of the program until you overwrite the active program, save the edited program as a new program, or discard the program. The changes you make do not affect the program while it runs.

Background edit notes:

- Press **[PROGRAM]** or **[MEMORY]** to exit background edit.
  - You cannot use **[CYCLE START]** during a background edit session. If the running program contains a programmed stop, you must exit background edit before you can use **[CYCLE START]** to continue the program.
1. To edit the active program, press **[EDIT]** while the program runs.  
A copy of the active program appears in the **PROGRAM GENERATION** pane on the right-hand side of the screen.
  2. To edit a different program while the active program runs:
    - a. Press **[LIST PROGRAM]**.
    - b. Select the program that you want to edit.
    - c. Press **[ALTER]**.

The program appears in the **PROGRAM GENERATION** pane on the right-hand side of the screen.

3. Edit the program.
4. The changes that you make to the active program do not affect the program while it runs.
5. If you background edit the active program, after the program finishes running, when you attempt to leave the screen, a popup gives the option to overwrite the program or discard your changes.
  - Pick the first item, Overwrite After End of Program, in the popup and press **[ENTER]** to overwrite the active program with your changes.
  - Pick the second item, Discard Changes, in the popup and press **[ENTER]** to discard all of your changes.

### 5.2.4 Program Editor

The Program Editor is a full-featured editing environment with access to powerful functions in an easy-to-use pull-down menu. You use the Program Editor for normal editing.

Press **[EDIT]** to enter edit mode and use the Program Editor.

F5.3: Example Program Editor Screen. [1] Main Program Display, [2] Menu Bar, [3] Clipboard



## Program Editor Pull-Down Menu

The Program Editor uses a pull-down menu to give you easy access to editor functions in (5) categories: **File**, **Edit**, **Search**, and **Modify**. This section describes the categories and the choices you can make when you select them.

To use the pull-down menu:

1. Press **[EDIT]** to start the Program Editor.
2. Press **[F1]** to access the pull-down menu.

The menu opens for the last category you used. If you have not yet used the pull-down menu, the **File** menu opens by default.

3. Use the **[LEFT]** and **[RIGHT]** cursor arrow keys to highlight a category. When you highlight a category, the menu appears below the category name.

4. Use the [UP] and [DOWN] cursor arrow keys to select an option inside of the current category.
5. Press [ENTER] to execute the command.

Some menu commands require further input or confirmation. In these cases, an input window or confirmation pop-up appears on the screen. Type your input in the field(s) where applicable, and then press [ENTER] to confirm the action or [UNDO] to close the popup and cancel the action.

## File Menu

The **File** menu has these options:

- **New**: Creates a new program. In the pop-up menu fields, type an O number (required), a filename (optional), and a file title (optional). For more information about this menu, refer to “Create a New Program” in the Operation section of this manual.
- **Set To Run**: Saves the program and puts it in the active program pane on the left-hand side of the screen. You can also press [MEMORY] to use this function.
- **Save**: Saves the program. The program’s filename and path change from red to black to show that the changes are saved.
- **Save As**: You can save the file under any filename. The program’s new filename and path will change from red to black to show that the changes are saved.
- **Discard Changes**: Reverts any changes you’ve made since the file was last saved.

## Edit Menu

The **Edit** menu has these options:

- **Undo**: Reverses the last edit operation, up to the last (40) edit operations. You can also press [UNDO] to use this function.
- **Redo**: Reverses the last undo operation, up to the last (40) undo operations.
- **Cut Selection To Clipboard**: Removes the selected lines of code from the program and puts them in the clipboard. Refer to “Block Selection” to learn how to make a selection.
- **Copy Selection To Clipboard**: Puts the selected lines of code in the clipboard. This operation does not remove the original selection from the program.
- **Paste From Clipboard**: Puts a copy of the clipboard contents below the current line. This does not remove the clipboard contents.
- **Insert File Path (M98)**: Lets you select a file from a directory and creates the path with the M98.
- **Insert Media File (M130)**: Lets you select a media file from a directory and creates the path with the M130.
- **Insert Media File (\$FILE)**: Lets you select a media file from a directory and creates the path with the \$FILE tag.
- **Special Symbols**: Inserts a special symbol.

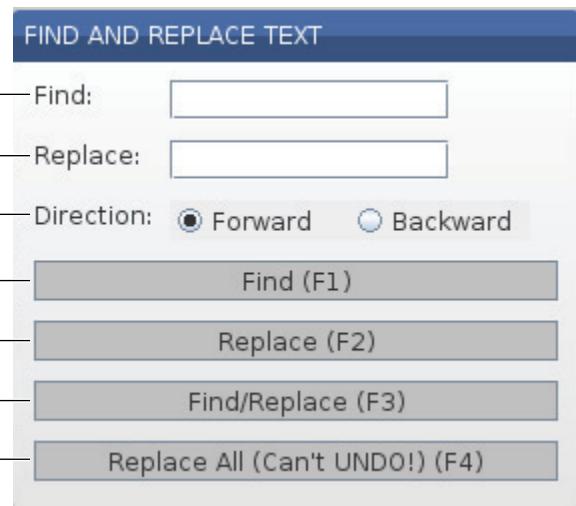
## Search Menu

The **Search** menu gives you access to the **Find And Replace Text** function. This function lets you quickly find code in the program and optionally replace it. To use it:

**NOTE:**

*This function searches for program code, not text. You cannot use this function to find text strings (like comments).*

- F5.4:** Find and Replace Menu Example: [1] Text to Find, [2] Replacement Text, [3] Search Direction, [4] Find Option, [5] Replace Option, [6] Find and Replace Option, [7] Replace all Option



### Specify Your Find / Replace Code

1. Press **[ENTER]** in the editor pull-down menu to open the **Find And Replace Text** menu. Use the cursor arrow keys to move between the fields in the menu.
2. In the **Find** field, Type the code you want to search for.
3. If you want to replace some or all of the found code, type the replacement code in the **Replace** field.
4. Use the **[LEFT]** and **[RIGHT]** cursor arrow keys to choose the search direction. **Forward** searches the program below the cursor position, **Backward** searches the program above the cursor position.

After you specify at least the code that you want to search for and the direction you want to search, press the function key for the search mode that you want to use:

#### Find Code ( **[F1]** )

Press **[F1]** to find the search term.

The control searches the program in the specified direction and highlights the first occurrence of the search term. Each time you press [F1], the control searches for the next occurrence of the search term, in the search direction that you specified, until it reaches the end of the program.

### Replace Code ( [F2] )

After the search function finds an occurrence of your search term, you can press [F2] to replace that code with the contents of the **Replace** field.



**NOTE:**

*If you press [F2] with no text in the Replace field, the control deletes that occurrence of your search term.*

### Find and Replace( [F3] )

Press [F3] instead of [F1] to start the find and replace operation. For each occurrence of your search term, press [F3] if you want to replace it with the text in the **Replace** field.

### Replace All( [F4] )

Press [F4] to replace all of the search term occurrences in (1) step. You cannot undo this.

## MODIFY Menu

The Modify menu has commands that let you make quick changes to an entire program, or to selected lines within a program.

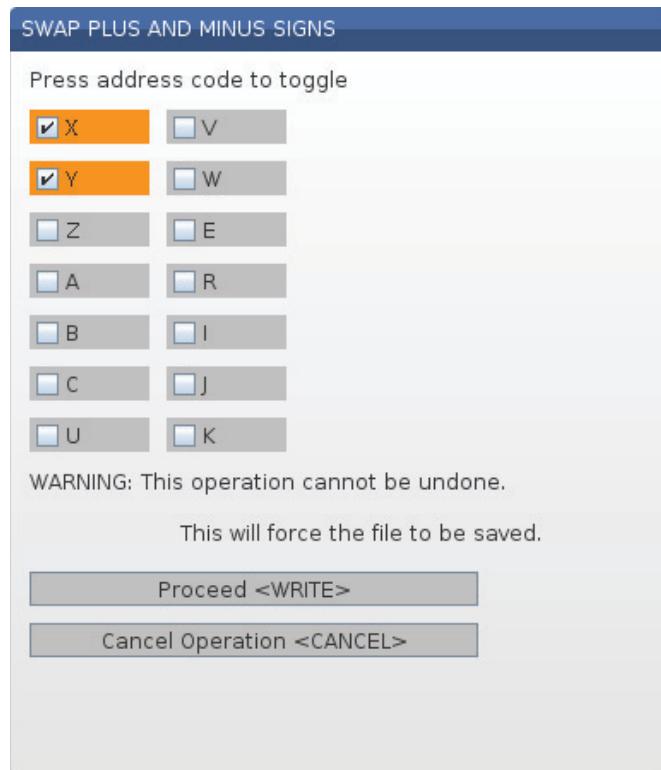


**NOTE:**

*You cannot use [UNDO] to reverse Modify operations. The operations also automatically save the program. If you are not sure you want to keep the changes that you make, be sure to save a copy of the original program.*

- **Remove All Line Numbers:** Automatically removes all N-code line numbers from the program or the selected program blocks.
- **Renumber All Lines:** Automatically adds N-code line numbers to the program or the selected program blocks. Enter the line number you want to start with and the increment to use between line numbers, then press [ENTER] to continue or press [UNDO] to cancel and to return to the editor.
- **Reverse + And - Signs:** Changes positive values for selected address codes to negative, or negative values to positive. Press the letter key for the address codes that you want to reverse to toggle selections in the pop-up menu. Press [ENTER] to execute the command or [CANCEL] to return to the editor.

F5.5: Reverse Plus and Minus Signs Menu



- **Reverse X And Y:** Changes the X address codes in the program to Y address codes, and changes Y address codes to X address codes.

## 5.3 Basic Programming

A typical CNC program has (3) parts:

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

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

**NOTE:**

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

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

### 5.3.1 Preparation

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

Preparation Code Block	Description
%	Denotes the beginning of a program written in a text editor.
O40001 (Basic program) ;	O40001 is the name of the program. Program naming convention follows the Onnnnn format: The letter "O", or "o" is followed by a 5-digit number.

Preparation Code Block	Description
(G54 X0 Y0 is top right corner of part) ;	Comment
(Z0 is on top of the part) ;	Comment
(T1 is a 1/2" end mill) ;	Comment
(BEGIN PREPARATION BLOCKS) ;	Comment
T1 M06 (Select tool 1) ;	Selects tool T1 to be used. M06 commands the tool changer to load Tool 1 (T1) into the spindle.
G00 G90 G17 G40 G49 G54 (Safe startup) ;	<p>This is referred to as a safe startup line. It is good machining practice to place this block of code after every tool change. G00 defines axis movement following it to be completed in Rapid Motion mode. G90 defines axis movements following it to be completed in absolute mode (refer to page <b>190</b> for more information). G17 defines the cutting plane as the XY plane. G40 cancels Cutter Compensation. G49 cancels tool length compensation. G54 defines the coordinate system to be centered on the Work Offset stored in G54 on the Offset display.</p>
X0 Y0 (Rapid to 1st position) ;	X0 Y0 commands the table to move to the position X = 0.0 and Y = 0.0 in the G54 coordinate system.
S1000 M03 (Spindle on CW) ;	<p>M03 turns the spindle on in a clockwise direction. It takes the address code Snnnn, where nnnn is the desired spindle RPM. On machines with a gearbox, the control automatically selects high gear or low gear, based on the commanded spindle speed. You can use an M41 or M42 to override this. Refer to page <b>420</b> for more information on these M-codes.</p>
G43 H01 Z0.1 (Tool offset 1 on) ;	G43 H01 turns on Tool Length Compensation +. The H01 specifies to use the length stored for Tool 1 in the Tool Offset display. Z0.1 commands the Z Axis to Z=0.1.
M08 (Coolant on) ;	M08 commands the coolant to turn on.

## 5.3.2 Cutting

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

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

## 5.3.3 Completion

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

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

### 5.3.4 Absolute vs. Incremental Positioning (G90, G91)

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

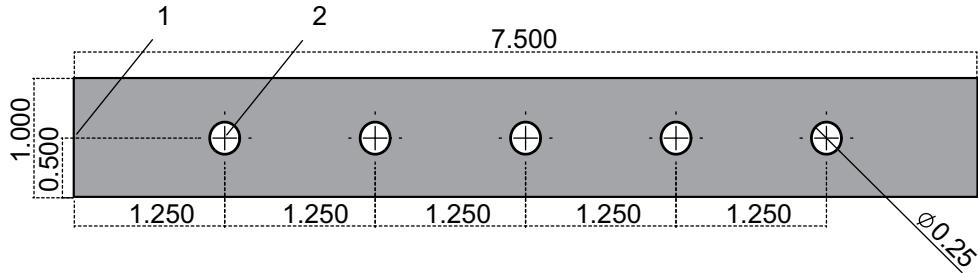
When you command axis motion after a G90 code, the axes move to that position relative to the origin of the coordinate system currently in use.

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

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

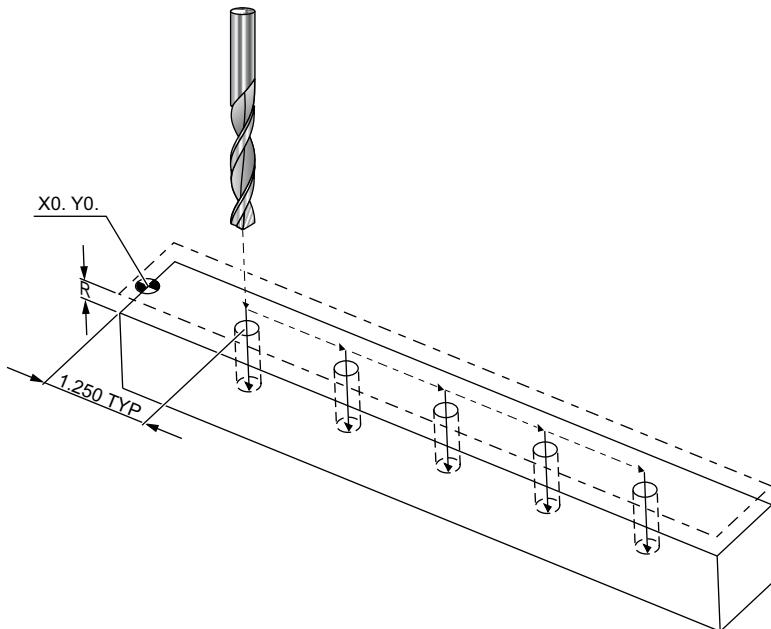
Figure F5.6 shows a part with 5 equally spaced Ø0.25" (13 mm) diameter holes. The hole depth is 1.00" (25.4 mm) and the spacing is 1.250" (31.75 mm) apart.

**F5.6:** Absolute / Incremental Sample Program. G54 X0. Y0. for Incremental [1], G54 for Absolute [2]



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

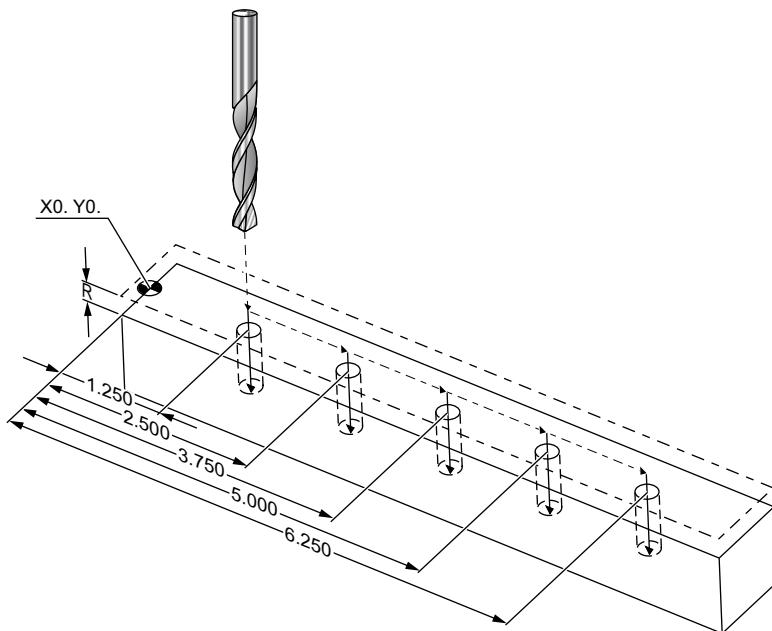
**F5.7:** Mill Incremental Positioning Example.



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

```
N23 S1000 M03 (Spindle on CW) ;
N24 G43 H02 Z0.1(Tool offset 2 on) ;
N25 M08 (Coolant on) ;
N26 (T2 CUTTING BLOCKS) ;
N27 G99 G91 G81 F21.4 X1.25 Z-1.1 L5 ;
N28 G80 (Cancel G81) ;
N29 (T2 COMPLETION BLOCKS) ;
N30 G00 Z0.1 M09 (Rapid retract, clnt off) ;
N31 G53 G90 G49 Z0 M05 (Z home, spindle off) ;
N32 G53 Y0 (Y home) ;
N33 M30 (End program) ;
%
```

### F5.8: Mill Absolute Positioning Example



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

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

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

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

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

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

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

## 5.4 Tool and Work Offset Calls

### 5.4.1 G43 Tool Offset

The G43 Hnn Tool Length Compensation command should be used after every tool change. It adjusts the Z-Axis position to account for the length of the tool. The Hnn argument specifies which tool length to use. For more information see Setting Tool Offsets on page **169** in the Operation section.



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

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

### 5.4.2 G54 Work Offsets

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

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

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

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

```
%  
O40005 (Work offsets ex-prog) ;  
(G54 X0 Y0 is center left of part) ;  
(Z0 is on top of the part) ;  
(T1 is a drill) ;  
(BEGIN PREPARATION BLOCKS) ;
```

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

## 5.5 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 **409**, for a list of all M-codes with descriptions.



**NOTE:**

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

### 5.5.1 Tool Functions (Tnn)

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

**CAUTION:**

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

You can command a tool change with the X, Y, and Z Axes in any position. The control will bring the Z Axis up to the machine zero position. The control moves the Z Axis to a position above machine zero during a tool change, but it never moves below machine zero. At the end of a tool change, the Z Axis is at machine zero.

### 5.5.2 Spindle Commands

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

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

**NOTE:**

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

- M05 commands the spindle to stop.

**NOTE:**

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

### 5.5.3 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.*

### 5.5.4 Coolant Commands

Use M08 to command standard coolant on. Use M09 to command standard coolant off. Refer to page 414 for more information on these M-codes.

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

## 5.6 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

### 5.6.1 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 309 for more information on G01.

## 5.6.2 Circular Interpolation Motion

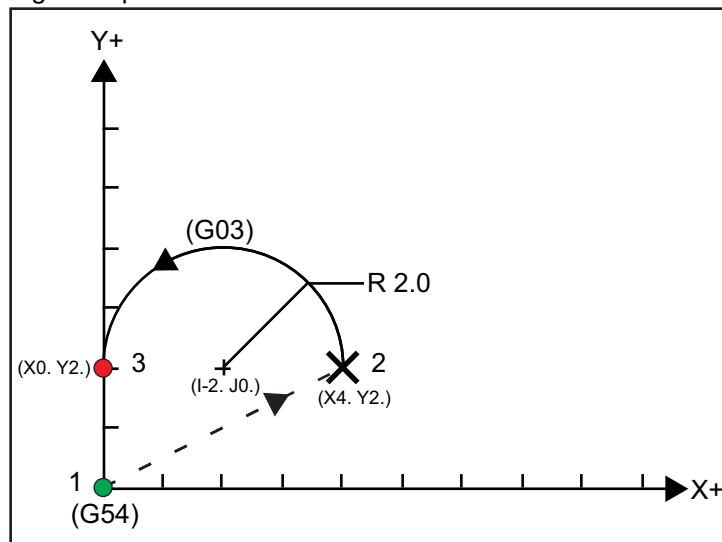
G02 and G03 are the G-codes for circular cutting motions. Circular Interpolation Motion has several optional address codes to define the arc or circle. The arc or circle begins cutting from the current cutter position [1] to the geometry specified within the G02/ G03 command.

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]:

**F5.9:** 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 ;  
%
```

## 5.7 Cutter Compensation

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

## 5.7.1 General Description of Cutter Compensation

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

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

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

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

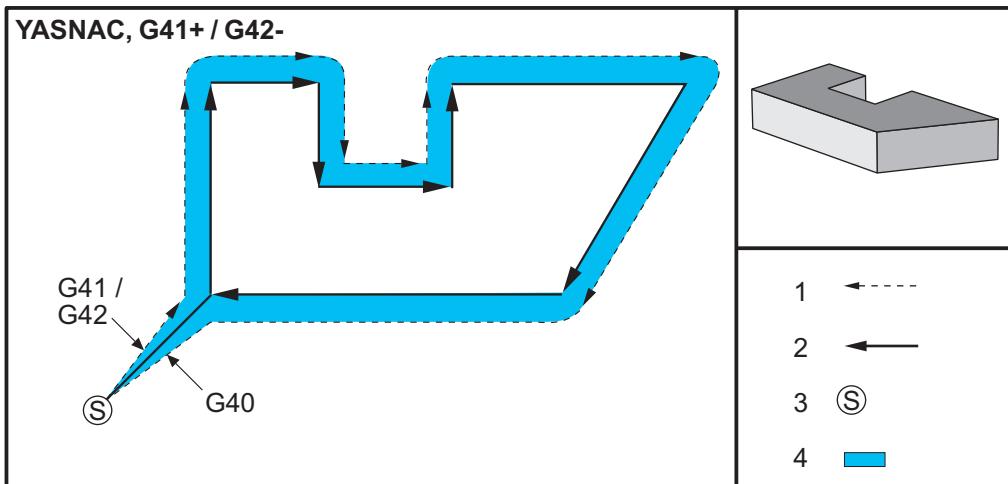
The control operates on one motion block at a time. However, it will look ahead at the next (2) blocks that have X or Y motions. The control checks these (3) blocks of information for interference. Setting 58 controls how this part of cutter compensation works. Available Setting 58 values are Fanuc or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

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

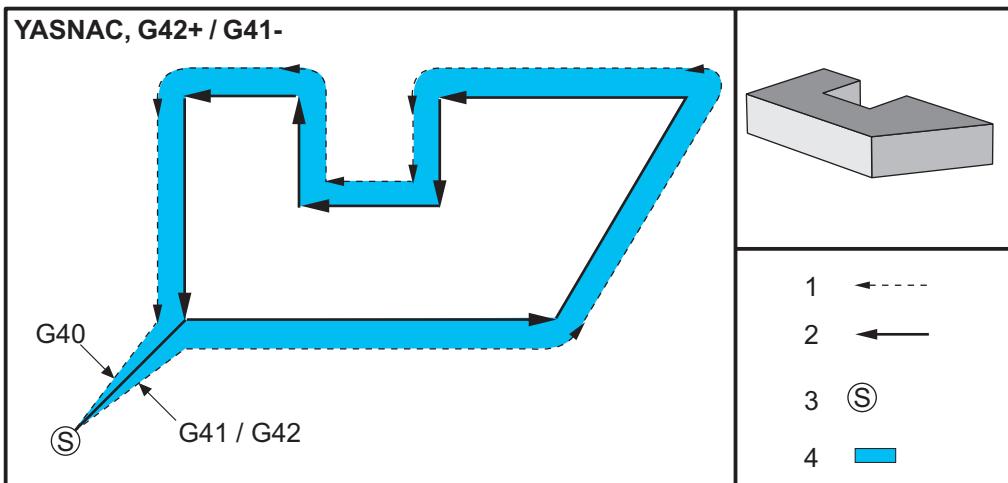
If Setting 58 is set to Fanuc, the control does not require that the tool cutting edge be placed along all edges of the programmed contour, preventing overcutting. However the control will generate an alarm if the cutter's path is programmed so that it will overcut. The control joins outside angles less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

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

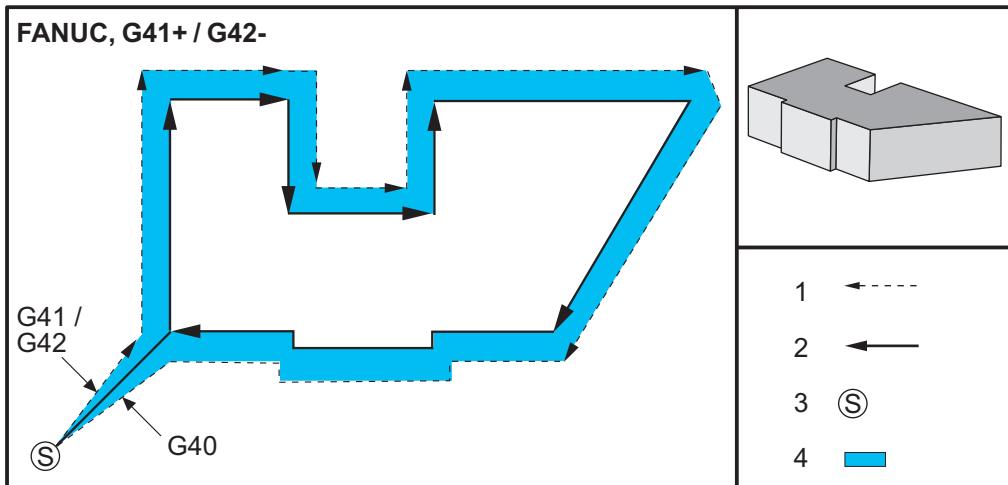
- F5.10:** Cutter Compensation, YASNAC Style, G41 with a Positive Tool Diameter or G42 with a Negative Tool Diameter: [1] Tool Path Actual Center, [2] Programmed Tool Path, [3] Start Point, [4] Cutter Compensation. G41 / G42 and G40 are commanded at the start and end of the tool path.



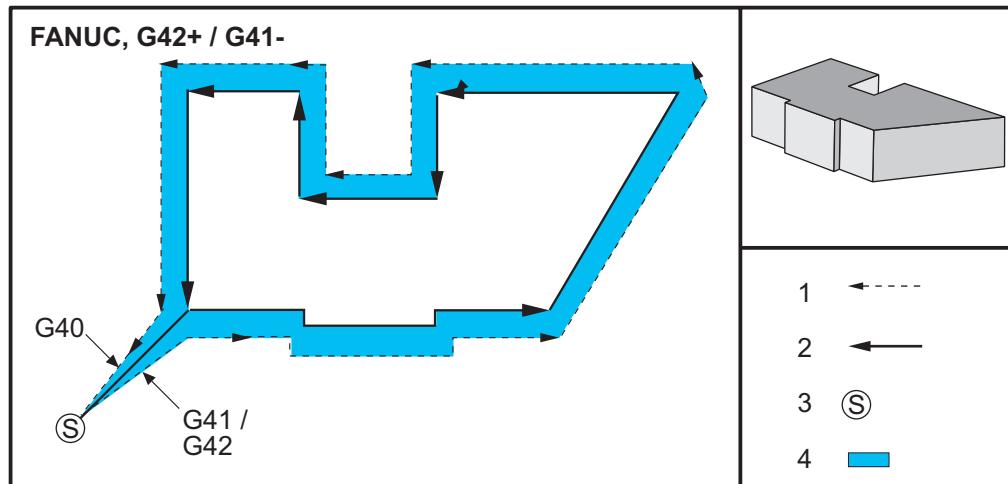
- F5.11:** Cutter Compensation, YASNAC Style, G42 with a Positive Tool Diameter or G41 with a Negative Tool Diameter: [1] Tool Path Actual Center, [2] Programmed Tool Path, [3] Start Point, [4] Cutter Compensation. G41 / G42 and G40 are commanded at the start and end of the tool path.



- F5.12: Cutter Compensation, FANUC Style, G41 with a Positive Tool Diameter or G42 with a Negative Tool Diameter: [1] Tool Path Actual Center, [2] Programmed Tool Path, [3] Start Point, [4] Cutter Compensation. G41 / G42 and G40 are commanded at the start and end of the tool path.



- F5.13: Cutter Compensation, FANUC Style, G42 with a Positive Tool Diameter or G41 with a Negative Tool Diameter: [1] Tool Path Actual Center, [2] Programmed Tool Path, [3] Start Point, [4] Cutter Compensation. G41 / G42 and G40 are commanded at the start and end of the tool path.



## 5.7.2 Entry and Exit from Cutter Compensation

When entering and exiting cutter compensation, or when changing from left side to right side compensation, there are special considerations to be aware of. Cutting should not be performed during any of these moves. To activate cutter compensation, a nonzero  $D$  code must be specified with either G41 or G42 and G40 must be specified in the line that cancels cutter compensation. In the block that turns on cutter compensation, the starting position of the move is the same as the programmed position, but the ending position will be offset, to either the left or right of the programmed path, by the amount entered in the, radius/diameter, offset column.

In the block that turns off cutter compensation, the starting point is offset and the ending point is not offset. Similarly, when changing from left to right or right to left side compensation, the starting point of the move needed to change cutter compensation direction will be offset to one side of the programmed path and end at a point that is offset to the opposite side of the programmed path. The result of all this is that the tool moves through a path that may not be the same as the intended path or direction.

If cutter compensation is turned on or off in a block without any X-Y move, there is no change made to cutter compensation until the next X or Y move is encountered. To exit from cutter compensation, you must specify G40.

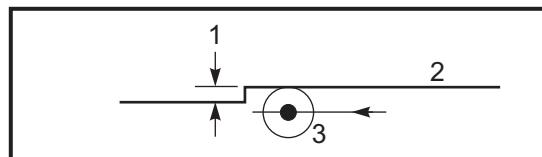
You should always turn off cutter compensation in a move that clears the tool away from the part being cut. If a program is terminated with cutter compensation still active, an alarm is generated. In addition, you cannot turn cutter compensation on or off during a circular move (G02 or G03); otherwise an alarm will be generated.

An offset selection of D0 will use zero as the offset value and have the same effect as turning off cutter compensation. If a new  $D$  value is selected while cutter compensation is already active, the new value will take effect at the end of the proceeding move. You cannot change the  $D$  value or change sides during a circular motion block.

When turning on cutter compensation in a move that is followed by a second move at an angle of less than 90 degrees, there are two ways of computing the first motion: cutter compensation type A and type B (Setting 43). Type A is the default in Setting 43 and is what is normally needed; the tool moves directly to the offset start point for the second cut. Type B is used when clearance around a fixture or clamp is needed, or in rare cases when part geometry demands it. The diagrams in this section illustrate the differences between type A and type B for both Fanuc and Yasnac settings (Setting 58).

## Improper Cutter Compensation Application

- F5.14: Improper Cutter Compensation: [1] Move is less than cutting comp radius, [2] Workpiece, [3] Tool.



**NOTE:**

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

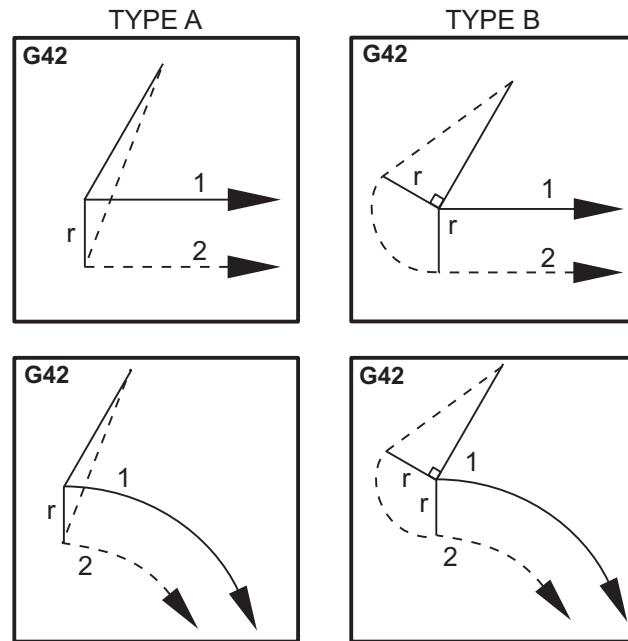
### 5.7.3 Feed Adjustments in Cutter Compensation

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

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

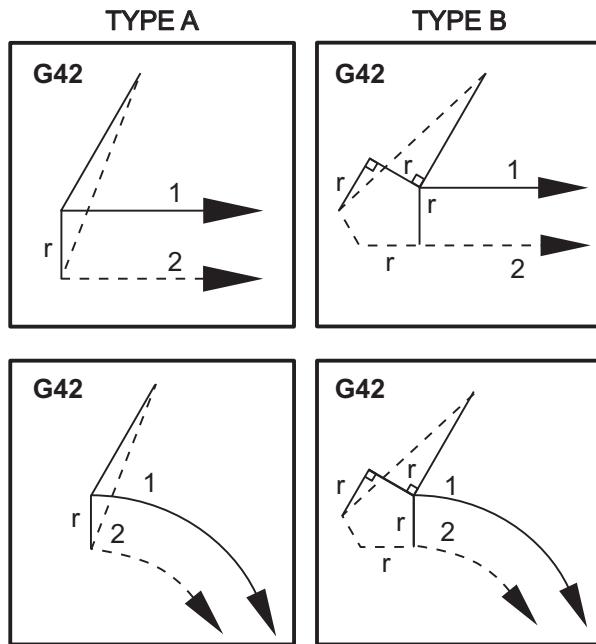
## Cutter Compensation Entry (Yasnac)

**F5.15:** Cutter Compensation Entry (Yasnac) Type A and B: [1] Programmed Path, [2] Tool Center Path, [r] Tool Radius



## Cutter Compensation Entry (Fanuc style)

**F5.16:** Cutter Compensation Entry (Fanuc style) Type A and B: [1] Programmed Path, [2] Tool Center Path, [r] Tool Radius



### 5.7.4 Circular Interpolation and Cutter Compensation

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

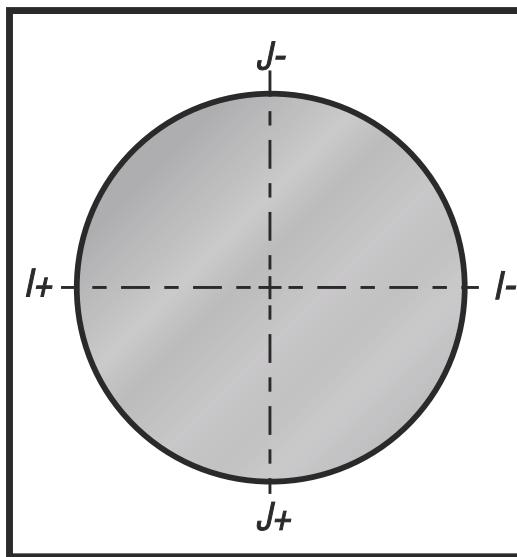
Using G02 and G03, we can program the machine to cut circular moves and radii. Generally, when programming a profile or a contour, the easiest way to describe a radius between two points is with an  $R$  and a value. For complete circular moves (360 degrees), an  $I$  or a  $J$  with a value must be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

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

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

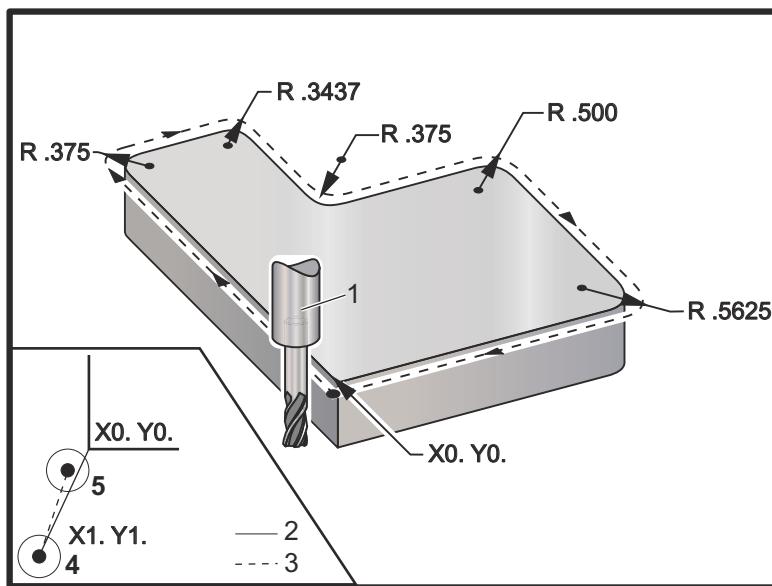
1. Cutter compensation must be turned ON during a G01 X, Y move that is equal to or greater than the cutter radius, or the amount being compensated.
2. When an operation using cutter compensation is done, the cutter compensation will need to be turned OFF, using the same rules as the turn ON process, i.e., what is put in must be taken out.
3. In most machines, during cutter compensation, a linear X,Y move that is smaller than the cutter radius may not work. (Setting 58 - set to Fanuc - for positive results.)
4. Cutter compensation cannot be turned ON or OFF in a G02 or G03 arc movement.
5. With cutter compensation active, machining an inside arc with a radius less than what is defined by the active D value causes the machine to alarm. Can not have too big of a tool diameter if the radius of arc is too small.

**F5.17:** Circle Sections



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

- F5.18:** Circular Interpolation G02 and G03: [1] 0.250" diameter endmill, [2] Programmed path, [3] Center of Tool, [4] Start Position, [5] Offset Tool Path.



### Programming exercise showing tool path.

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

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

---

```

G03 X2.375 Y2.75 R0.375 (Corner rounding) ;
G01 X3.5 (Linear motion) ;
G02 X4. Y2.25 R0.5 (Corner rounding) ;
G01 Y0.4375 (Linear motion) ;
G02 X3.4375 Y-0.125 R0.5625 (Corner rounding) ;
G01 X-0.125 (Linear motion) ;
G40 X-1. Y-1. (Last position, cutter comp off) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%

```

## 5.8 Canned Cycles

Canned cycles are G-codes that do repetitive operations such as drilling, tapping, and boring. You define a canned cycle with alphabetic address codes. While the canned cycle is active, the machine does the defined operation every time you command a new position, unless you specify not to.

### 5.8.1 Drilling Canned Cycles

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

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

### 5.8.2 Tapping Canned Cycles

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

- The G84 Tapping Canned Cycle is the normal tapping cycle. It is used for tapping right-hand threads.

- G74 Reverse Tap Canned Cycle is the reverse thread tapping cycle. It is used for tapping left-hand threads.

### **5.8.3 Boring and Reaming Cycles**

There are (5) boring canned cycles. All boring canned cycles can be looped in G91, Incremental Programming mode.

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

### **5.8.4 R Planes**

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

## **5.9 Special G-codes**

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

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

## 5.9.1 Engraving

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

Refer to page 333 for more information on engraving.

## 5.9.2 Pocket Milling

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

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

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

Refer to page 5 for more information on the pocket milling G-codes.

## 5.9.3 Rotation and Scaling



### NOTE:

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

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

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

Refer to page 345 for more information on the rotation and scaling G-codes.

## 5.9.4 Mirror Image

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

Refer to page 370 for more information on the mirror image G-codes.

## 5.10 Subprograms

Subprograms:

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

When you use M97:

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

When you use M98:

- The P code (nnnnn) is the same as the program number (Onnnnn) of the subprogram.
- If the subprogram is not in memory, the file name must be Onnnnn.nc. The file name must contain the O, leading zeros and .nc for the machine to find the subprogram.
- The subprogram must reside in the active directory, or in a location specified in Settings 251/252. Refer to page 471 for more information on subprogram search locations.

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

### 5.10.1 External Subprogram (M98)

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

When your program calls an M98 subprogram, the control looks for the subprogram in the main program's directory. If the control cannot find the subprogram in the main program's directory, it then looks in the location specified in Setting 251. Refer to page 470 for more information. An alarm occurs if the control cannot find the subprogram.

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

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

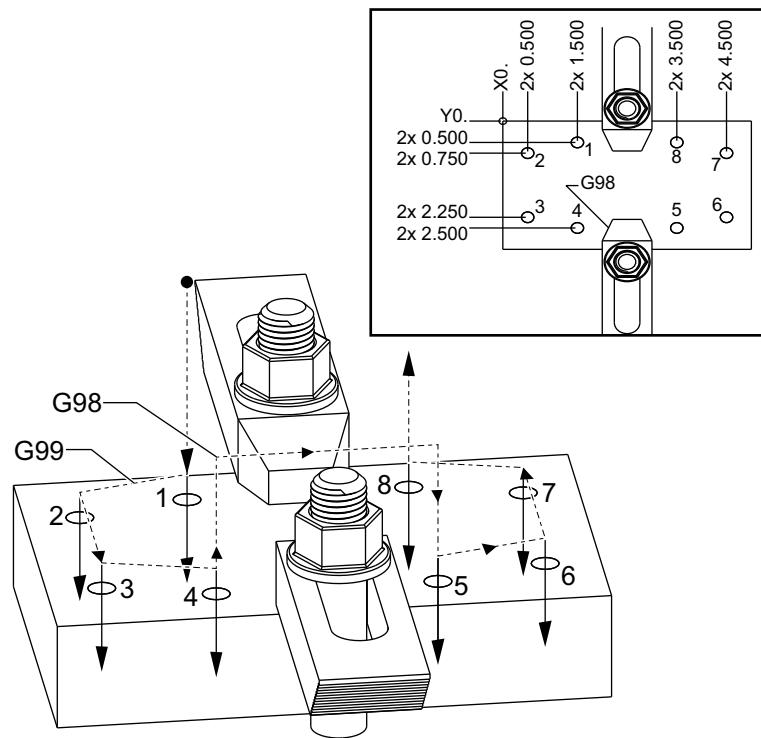
1. G81 Spot drill at each position
2. G83 Peck drill at each position
3. G84 Tap at each position

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

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

```
G00 G54 X1.5 Y-0.5 (Rapid to 1st position) ;  
S750 M03 (Spindle on CW) ;  
G43 H03 Z1. (Tool offset 3 on) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;  
G84 G99 Z-0.6 R0.1 F37.5 (Begin G84) ;  
M98 P40008 (Call external subprogram);  
(BEGIN COMPLETION BLOCKS) ;  
G00 Z1. M09 (Rapid retract, Coolant off) ;  
G53 G49 Z0 M05 (Z home, Spindle off) ;  
G53 Y0 (Y home) ;  
M30 (End program) ;  
%
```

F5.19: Subprogram Pattern



### Subprogram

```
%  
O40008 (Subprogram) ;  
X0.5 Y-0.75 (2nd position) ;  
Y-2.25 (3rd position) ;
```

```

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

```

## 5.10.2 Local Subprogram (M97)

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

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

### Local Subprogram Example

```

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

```

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

### 5.10.3 External Subprogram Canned Cycle Example (M98)

%  
O40010 (M98\_External sub canned cycle ex) ;

```
(G54 X0 Y0 is at the top left of the part) ;
(Z0 is on top of the part) ;
(T1 is a spot drill) ;
(T2 is a drill) ;
(T3 is a tap) ;
(BEGIN PREPARATION BLOCKS) ;
T1 M06 (Select tool 1) ;
G00 G90 G40 G49 G54(Safe startup) ;
X0.565 Y-1.875 (Rapid to 1st position) ;
S1275 M03 (Spindle on CW) ;
G43 H01 Z0.1 (Tool offset 1 on) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G82 Z-0.175 P0.03 R0.1 F10. (Begin G82) ;
M98 P40011 (Call external subprogram) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z1. M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
M01 (optional stop) ;
(BEGIN PREPARATION BLOCKS) ;
T2 M06 (Select tool 2) ;
G00 G90 G40 G49 (Safe startup) ;
G54 X0.565 Y-1.875 ;
(Rapid back to 1st position) ;
S2500 M03 (Spindle on CW) ;
G43 H02 Z0.1 (Tool offset 2 on) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G83 Z-0.72 Q0.175 R0.1 F15. (Begin G83) ;
M98 P40011 (Call external subprogram) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z1. M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
M01 (optional stop) ;
(BEGIN PREPARATION BLOCKS) ;
T3 M06 (Select tool 3) ;
G00 G90 G40 G49 (Safe startup) ;
G54 X0.565 Y-1.875 ;
(Rapid back to 1st position) ;
S900 M03 (Spindle on CW) ;
G43 H03 Z0.1 (Tool offset 3 on) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G84 Z-0.6 R0.2 F56.25 (Begin G84) ;
M98 P40011 (Call external subprogram) ;
G80 G00 Z1. M09 (Cancel canned cycle) ;
(BEGIN COMPLETION BLOCKS) ;
```

```
G00 Z1. M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%
```

### **Subprogram**

```
%  
O40011 (M98_Subprogram X,Y Locations) ;  
X1.115 Y-2.75 (2nd position) ;  
X3.365 Y-2.875 (3rd position) ;  
X4.188 Y-3.313 (4th position) ;  
X5. Y-4. (5th position) ;  
M99 ;  
%
```

## **5.10.4 External Subprograms With Multiple Fixtures (M98)**

Subprograms can be useful when cutting the same part in different X and Y locations within the machine. For example, there are six vises mounted on the table. Each of these vises uses a new X, Y zero. They are referenced in the program using the G54 through G59 work offsets in absolute coordinates. Use an edge finder or an indicator to establish the zero point on each part. Use the part zero set key in the work offset page to record each X, Y location. Once the X, Y zero position for each workpiece is in the offset page, the programming can begin.

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

### **Main Program**

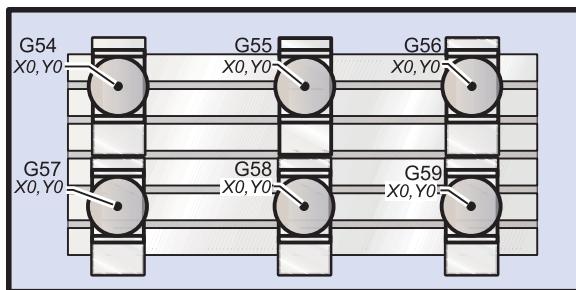
```
%  
O40012 (M98_External sub multi fixture);  
(G54-G59 X0 Y0 is center of each part) ;  
(G54-G59 Z0 is on top of the part) ;  
(T1 is a drill) ;  
(BEGIN PREPARATION BLOCKS) ;  
T1 M06 (Select tool 1) ;  
G00 G90 G40 G49 G54(Safe startup) ;  
X0 Y0 (Rapid to 1st position) ;  
S1500 M03 (Spindle on CW) ;  
G43 H01 Z0.1 (Tool offset 1 on) ;  
M08 (Coolant on) ;  
(BEGIN CUTTING BLOCKS) ;
```

```

M98 P40013 (Call external subprogram) ;
G55 (Change work offset) ;
M98 P40013 (Call external subprogram) ;
G56 (Change work offset) ;
M98 P40013 (Call external subprogram) ;
G57 (Change work offset) ;
M98 P40013 (Call external subprogram) ;
G58 (Change work offset) ;
M98 P40013 (Call external subprogram) ;
G59 (Change work offset) ;
M98 P40013 (Call external subprogram) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.1 M09 (Rapid retract, Coolant off) ;
G53 G49 Z0 M05 (Z home, Spindle off) ;
G53 Y0 (Y home) ;
M30 (End program) ;
%

```

#### F5.20: Subprogram Multiple Fixture Drawing



#### Subprogram

```

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

```

## 5.10.5 Setting Up Search Locations

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

To build a list of search locations in Setting 252:

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

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

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

## 5.10.6 More Information Online

For updated information, including tips, tricks, maintenance procedures, and more, visit the Haas Service page at [www.HaasCNC.com](http://www.HaasCNC.com).

For the most current Operator's and Service Manuals scan the code below with your mobile device:



# 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 Feature List

The Feature List contains both standard and purchasable options.

### F6.1: Features Tab

Parameters, Diagnostics And Maintenance

Diagnostics		Maintenance	Parameters
Features	Compensation	Activation	
Search (TEXT) [F1], or [F1] to clear. <input type="text"/>			
Feature	Status	Date:	
<input type="checkbox"/> Machine	Feature Disabled	Remaining 5 Days 1 hr	
<input checked="" type="checkbox"/> Macros	Purchased	Acquired 05-20-16	
<input checked="" type="checkbox"/> Rotation And Scaling	Purchased	Acquired 05-20-16	
<input checked="" type="checkbox"/> Rigid Tapping	Purchased	Acquired 05-20-16	
<input type="checkbox"/> TCPC and DWO	Tryout Available		
<input checked="" type="checkbox"/> M19 Spindle Orient	Purchased	Acquired 05-20-16	
<input type="checkbox"/> High Speed Machining	Tryout Available		
<input checked="" type="checkbox"/> VPS Editing	Purchased	Acquired 05-20-16	
<input checked="" type="checkbox"/> Fourth Axis	Purchased	Acquired 05-20-16	
<input type="checkbox"/> Fifth Axis	Feature Disabled	Purchase Required	
<input checked="" type="checkbox"/> Max Memory: 1GB	Purchased	Acquired 05-20-16	
<input checked="" type="checkbox"/> Wireless Networking	Purchased	Acquired 05-20-16	
<input checked="" type="checkbox"/> Compensation Tables	Purchased	Acquired 05-20-16	
<input type="checkbox"/> Through Spindle Coolant	Feature Disabled	Purchase Required	
<input checked="" type="checkbox"/> Max Spindle Speed: 8100 RPM	Purchased	Acquired 05-20-16	

\*Tryout time is only updated while Feature is enabled.

**ENTER** Turn On/Off Feature      **F4** Purchase Feature With Entered Activation Code.

To access the list:

1. Press [DIAGNOSTIC].
2. Navigate to the **Parameters** and then the **Features** tab. (Purchased options are marked in green and their status is set as PURCHASED.)

### 6.2.1 Purchased Options Enable/Disable

To enable or disable a purchased option:

1. Highlight the option on the **FEATURES** tab.
2. Press [**ENTER**] to turn ON/OFF the option.

If the featured option is turned OFF, the option is not available.

### 6.2.2 Option Tryout

Some options have a 200-hour tryout available. The FEATURES tab Status column shows the options available for tryout.



**NOTE:**

*If an option does not have a tryout, the Status column shows FEATURE DISABLED, and you must purchase the option to use it.*

To start tryout:

1. Highlight the feature.
2. Press [**ENTER**]. Press [**ENTER**] again to disable the option and stop the timer.

The status of the feature changes to TRYOUT ENABLED, and the date column shows the remaining hours left in the tryout period. When the tryout period expires, the status changes to EXPIRED. You cannot extend the tryout time for expired options. You must purchase them to use them.



**NOTE:**

*Tryout time is only updated while the option is enabled.*

## 6.3 Rotation and Scaling

Rotation lets you rotate a pattern to another location or around a circumference. Scaling reduces or enlarges a toolpath or pattern.

## 6.4 Visual Programming System (VPS)

VPS lets you quickly build programs from program templates. To access VPS, press [EDIT] and then select the **VPS** tab.

- F6.2:** VPS Starting Screen. [1] Recently Used Templates, [2] Template Directory Window, [3] [ENTER] to Load a Template, [4] [F4] to Switch Between Recently Used and Template Directory.



In the template directory window, you can choose from the **VPS** or **CUSTOM** directories. Highlight a directory name and press the **[RIGHT]** cursor arrow to see the directory's contents.

The VPS starting screen also lets you choose templates you have recently used. Press **[F4]** to change to the Recently Used window and highlight a template from the list. Press **[ENTER]** to load the template.

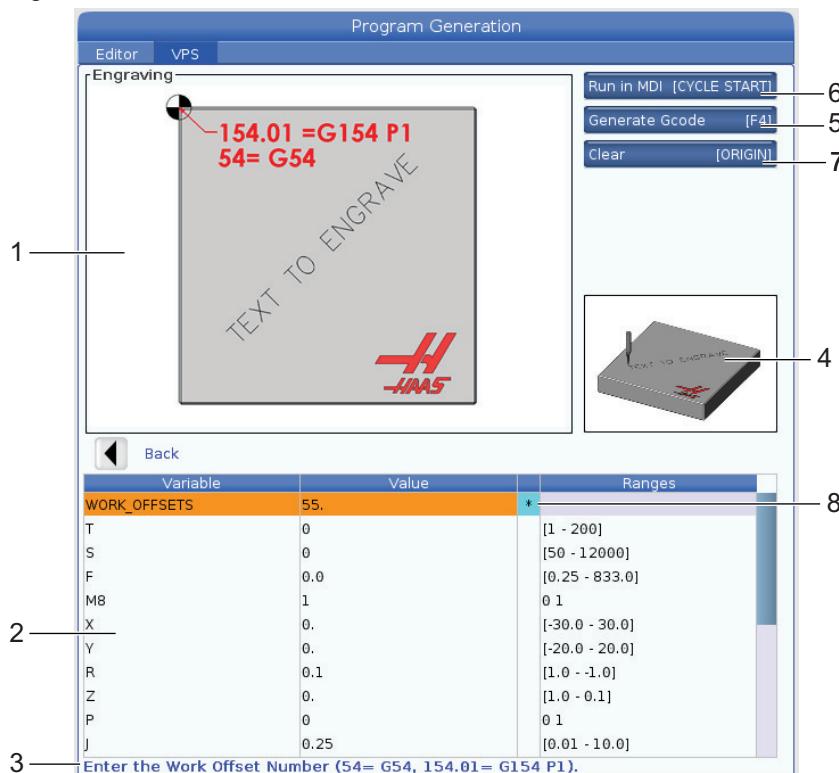
## 6.4.1 VPS Example

When you use VPS, you choose a template for the function you want to program, and then enter variables to create a program. The default templates include probing and part features. You can also create custom templates. Contact the Applications department at your HFO for help with custom templates.

In this example, we use a VPS template to program the engraving cycle from the G47 program example in this manual. The G47 description starts on page 333. VPS templates all work in the same way: You first fill in values for the template variables, and you then output a program.

1. Press **[EDIT]**, and then select the **VPS** tab.
2. Use the cursor arrow keys to highlight the **VPS** menu option. Press the **[RIGHT]** cursor arrow key to select the option.
3. Highlight and select the **Engraving** option from the next menu.

**F6.3:** VPS Engraving Program Generation Window Example. [1] Variable Illustration, [2] Variables Table, [3] Variable Description Text, [4] Template Illustration, [5] Generate G-code **[F4]**, [6] Run in MDI **[CYCLE START]**, [7] Clear **[ORIGIN]**, [8] Default value was changed indicator.



4. In the Program Generation window, use the **[UP]** and **[DOWN]** cursor arrow keys to highlight the variable rows.
5. Type a value for the highlighted variable and press ENTER. The control will display an asterisk (\*) next to variable if the default value is changed. To set the variable back to default press the **[ORIGIN]** button. Press the DOWN cursor arrow key to move to the next variable.

To generate the example engraving cycle, we use these variable values. Note that all of the position values are given in work coordinates.

<b>Variable</b>	<b>Description</b>	<b>Value</b>
<b>WORK_OFFSETS</b>	Work Offset Number	54
<b>T</b>	Tool Number	1
<b>S</b>	Spindle Speed	1000
<b>F</b>	Feedrate	15.
<b>M8</b>	Coolant (1 - YES / 0 - NO)	1
<b>X</b>	Starting X Position	2.
<b>Y</b>	Starting Y Position	2.
<b>R</b>	R-Plane Height	0.05
<b>Z</b>	Z Depth	-0.005
<b>P</b>	Text or Serial Number Switch (0 - Text, 1 - Serial Number)	0
<b>J</b>	Text Height	0.5
<b>I</b>	Text Angle (Degrees from Horizontal)	45.
<b>TEXT</b>	Text to Engrave	TEXT TO ENGRAVE

6. With all of the variables entered, you can press **[CYCLE START]** to immediately run the program in MDI, or F4 to output the code to either the clipboard or MDI without running the program.

This VPS template creates a program with the specified variables to engrave the text:

```
%  
O11111 ;  
(Engraving) ;  
( TOOL 1 ) ;  
( SPINDLE 1000 RPM / FEED 15. ) ;  
( DEPTH -0.005 ) ;  
T1 M06 ;  
G00 G90 G54 X2. Y2. S1000 M03 ;  
G43 Z0.05 H1 ;  
M08 ;  
G00 G90 G54 X2. Y2. ;  
( TEXT ENGRAVING : TEXT TO ENGRAVE ) ;  
G47 E7.5000 F15. I45. J5 P0 R0.05 Z-0.005 (TEXT TO ENGRAVE) ;  
G0 Z0.05 M09 ;  
M05 ;  
G91 G28 Z0. ;  
G91 G28 Y0. ;  
M01 ( END ENGRAVING ) ;  
%
```

## 6.5 Rigid Tapping

This option synchronizes the spindle's RPM with the feedrate during a tapping operation.

## 6.6 M19 Spindle Orientation

The Spindle Orientation lets you position the spindle to a programmed angle. This option provides inexpensive, accurate positioning. For more information about M19, refer to page **415**.

## 6.7 High-Speed Machining

The Haas high-speed machining option allows faster feedrates and more complex toolpaths. HSM uses a motion algorithm called Acceleration Before Interpolation combined with full look-ahead to provide contouring feeds up to 1200 ipm (30.5 m/min) without risk of distortion to the programmed path. This reduces cycle times, improves accuracy, and smooths motion.

## 6.8 Additional Memory Options

This option expands the on-board solid-state memory and allows the control to store, run, and edit large programs directly at the machine.

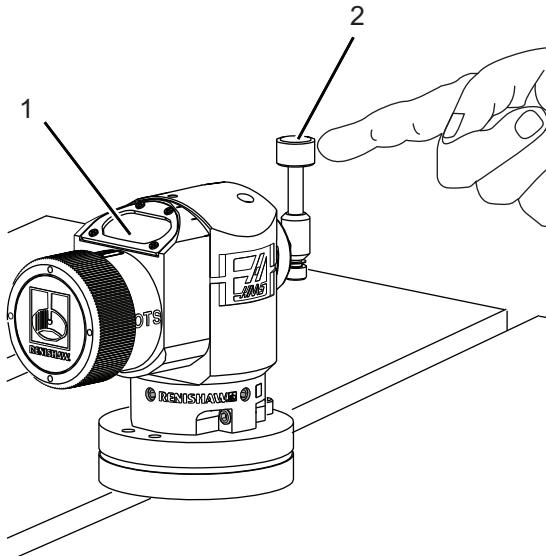
## 6.9 Probing

You can use an optional probe system to set offsets, check work, measure tools, and check tools. This section describes basic probe use and troubleshooting.

### 6.9.1 Check Tool Probe

Do these steps to make sure the tool probe operates correctly:

**F6.4:** Tool Probe Test



1. In MDI mode, run:

```
M59 P2 ;  
G04 P1.0 ;  
M59 P3 ;
```

This turns on tool probe communication, delays one second, and turns on the tool probe. The LED [1] on the tool probe flashes green.

2. Touch the stylus [2].

The machine makes a "beep" sound and the LED becomes red [1]. This tells you that the tool probe is started.

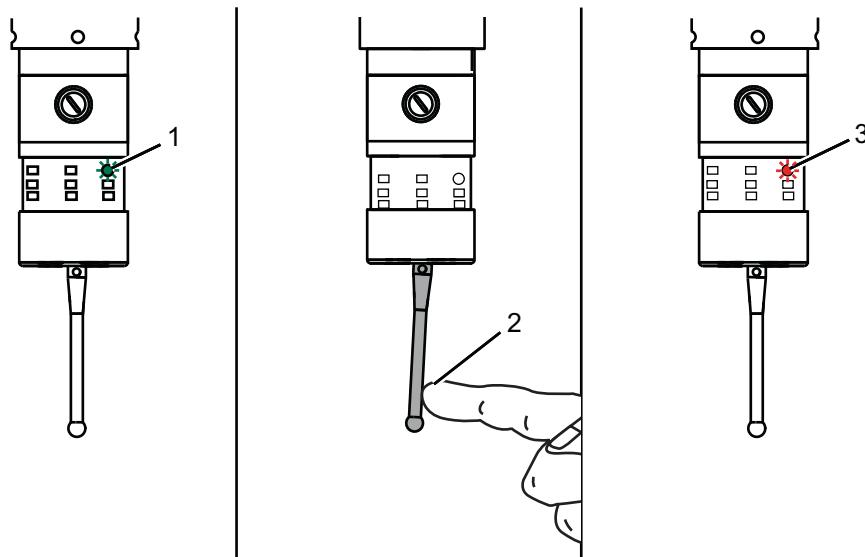
3. Press [RESET] to deactivate the probe.

The probe LED [1] turns off.

## 6.9.2 Check Work Probe

Do these steps to make sure the work probe operates correctly:

### F6.5: Work Probe Test



1. Select the work probe with a tool change, or manually insert the work probe into the spindle.
2. In MDI mode, run M69 P2 ;  
This starts communication with the work probe.
3. in MDI mode: run M59 P3 ;  
The probe LED flashes green [1].
4. Touch the stylus [2].  
The machine makes a "beep" sound and the LED becomes red [3]. This tells you that the work probe is started.
5. Press **[RESET]** to deactivate the probe.  
The work probe LED turns off [1].

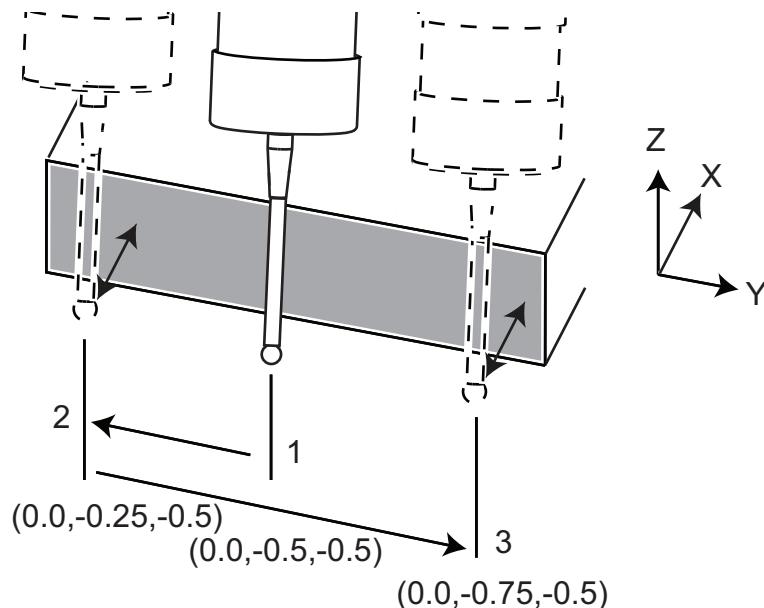
### 6.9.3 Probe Example

You can use a probe to check your part for correct dimensions during the machining process. For example, this program uses the work probe to check for squareness. The program uses G65 to call 9XXXXX macro programs created specifically for probing. You can find more information on these programs in the Renishaw manuals on the web at [www.haascnc.com](http://www.haascnc.com) then click on the Service tab.

The program does the following:

1. After a tool change, home, and adding tool length compensation, the system turns on the work probe and moves to a safe start location.
2. The probe stylus moves adjacent to the surface at the required Z-Axis point to provide a central start position [1].
3. The cycle makes two measurements, symmetrically about the start position, to establish the surface angle [2], [3].
4. Finally, the probe stylus moves to its safe out position, turns off the probe, and returns home.

**F6.6:** Check for Square: [1] Safe Move Position, [2] First Measurement, [3] Second Measurement



Example:

```
%  
O00010 (CHECK FOR SQUARE) ;  
T20 M06 (PROBE) ;
```

```

G00 G90 G54 X0. Y0. ;
G43 H20 Z6. ;
G65 P9832 (WORK PROBE ON) ;
G65 P9810 Z-0.5 F100. (SAFE MOVE) ;
G65 P9843 Y-0.5 D0.5 A15. (ANGLE MEAS.) ;
G65 P9810 Z6. F100. (SAFE OUT) ;
G65 P9833 (WORK PROBE OFF) ;
G00 G90 G53 Z0. ;
M01 ;
;
;
( PART PROGRAM ) ;
G00 G90 G54 X0. Y0. ;
T2 M06 (1/2" END MILL) ;
G00 G90 G43 H02 Z1.5 ;
G68 R#189 ;
G01 X-2. F50. ;
M30 ;
%

```

## 6.9.4 Probe Use with Macros

Macro statements select and turn on and off the probe the same as M-codes.

### T6.1: Probe Macro Values

M-Code	System Variable	Macro Value	Probe
M59 P2 ;	#12002	1.000000	Tool Probe Selected
M69 P2 ;	#12002	0.000000	Work Probe Selected
M59 P3 ;	#12003	1.000000	Probe Enable
M69 P3 ;	#12003	0.000000	Probe Disable

If you assign the system variable to a viewable global variable, you can see the macro value change in the **Macro Vars** tab under **[CURRENT COMMANDS]**.

For example,

```

M59 P3 ;
#10003=#12003 ;

```

The global variable #10003 shows the output from M59 P3 ; as 1.000000. This means that either the tool probe or work probe is on.

## 6.9.5 VPS Probe Operations

VPS offers templates to simplify probe operations in (3) categories: Tool Setting, Spindle Probe, and Calibration. Select PROBING in the VPS menu and then select a template. Fill in the variable fields to generate the probing code. Refer to the VPS section of this manual, starting on page 223, for more information on using VPS templates.

### VPS Probe Example (Complete Probe Calibration)

F6.7: Complete Probe Calibration Screen



To calibrate the tool probe:

1. In VPS, select **PROBING > CALIBRATION >Complete Probe Calibration**.
2. Go to each variable and enter the correct value per the screen instructions.
3. Press **[CYCLE START]** to run the program, or F4 to generate the code to the clipboard or MDI.

## 6.9.6 Probe Troubleshooting

If you cannot get the tool or work probe to beep or flash, do these steps:

1. In **[MDI]** mode, run **M69 P2 ;** to select the spindle work probe or **M59 P2 ;** to select the table tool probe.
2. Run **M59 P3 ;** to make the probe flash.
3. To check the I/O values for the probe, press **[DIAGNOSTIC]** and select the **Diagnostics** tab, then the **I/O** tab.
4. Type **PROBE** and press **[F1]** to search for I/O items that contain the word “probe.”
5. Check the table for proper probe values. For example, **Output 2** with a value of 0 selects the work probe.

Type	Number	M-code	Name	Value	Probe
OUTPUT	2	<b>M69 P2 ;</b>	PROBE_SELECT_TO_PROBE	0	work
OUTPUT	2	<b>M59 P2 ;</b>	PROBE_SELECT_TO_PROBE	1	tool
OUTPUT	3	<b>M69 P3 ;</b>	PROBE_ENABLE_TO_PROBE	0	Off
OUTPUT	3	<b>M59 P3 ;</b>	PROBE_ENABLE_TO_PROBE	1	flashing

6. If you are using the correct I/O values in your programs, but the probe does not flash or beep, check the batteries in the probes, and then check the wired connections to the control.

## 6.10 Maximum Spindle Speed

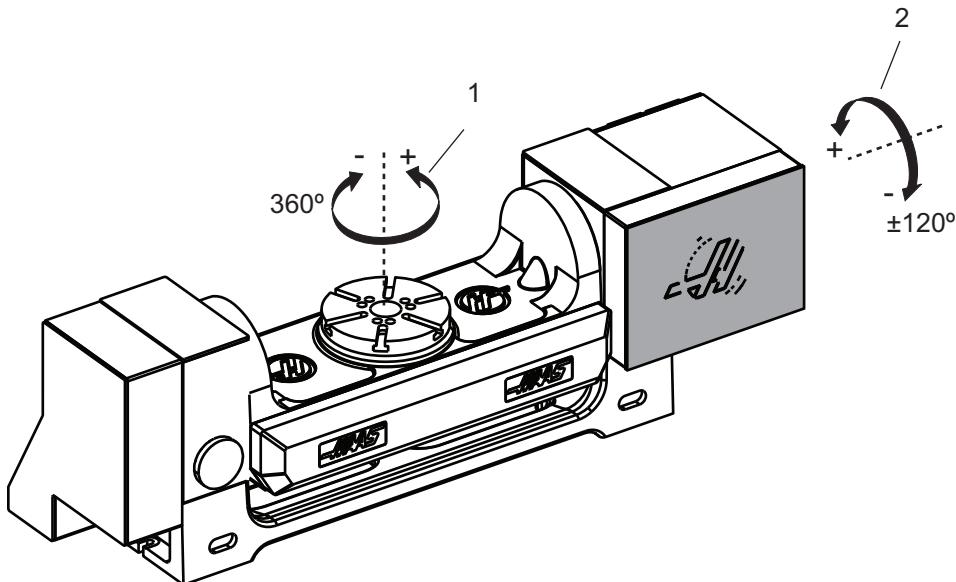
This option increases the maximum speed at which you can run the machine spindle.

## 6.11 Compensation Tables

With this option, the control stores a compensation table to correct for small errors in the rotary worm gear, as well as small errors in X, Y, and Z.

## 6.12 4th and 5th Axis Programming

**F6.8:** Axis Motion on an Example Rotary Trunnion Unit: [1] Rotary Axis, [2] Tilt Axis



### 6.12.1 New Rotary Configuration

When you install a rotary unit to your machine, you must:

- Designate the correct rotary model so that the machine control can load the correct parameters.
- Assign an axis letter (A, B, or C) to each new axis.
- Tell the machine which physical connection (4th or 5th axis) to use for each axis.

You do these tasks on the Rotary Selection page:

1. Press **[SETTING]**.
2. Select the **Rotary** tab.



**NOTE:**

*Make sure the machine is not in Hand Jog mode when you go to the Rotary Selection page. The control does not allow changes to rotary configuration in Hand Jog mode.*

When you go to the Rotary Selection page to install a rotary unit for the first time, both the 4th and 5th axes are disabled and have no rotary model selections. This process assigns a rotary model axis, and an axis letter, to the 4th and 5th axes.

**NOTE:**

To use Tool Center Point Control (TCPC) and Dynamic Work Offsets (DWO), your axis definitions and rotary installation must match the ANSI standard, where the A, B, and C Axes each rotate about the X, Y, and Z Axes, respectively. Refer to page 394 for more information about TCP. Refer to page 239 for more information about DWO.

**F6.9:** Rotary Selections Page. [1] Current Rotary Selections, [2] Select New Rotaries Table.

Axis	Configuration	Name	Model	Direction
4th Axis	Disabled	-----	-----	Normal
5th Axis	Disabled	-----	-----	Normal

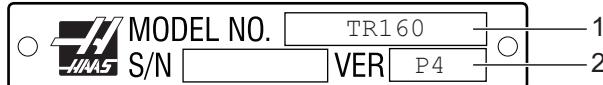
4th Axis	5th Axis	Name	Model
--	--	--	HA2CTS-B
--	--	--	HA2TS-P3
--	--	--	HA5C-P1
--	--	--	HA5C-P3
--	--	--	HA5C2-B
--	--	--	HA5C2-P3
--	--	--	HA5C3-HDH
--	--	--	HA5C3-P3
--	--	--	HA5C4-HDH
--	--	--	HA5C4-P3
--	--	--	HA5CS-B
--	--	--	HA5CS-P3

## Rotary Model Selection

In this procedure, you select your particular rotary model from the list of models in the control, so that the control can load the correct parameters for that unit. In this example, we have a TR160 unit installed on the table, with the tilt axis parallel to X.

We want to configure both the rotary (platter) and tilt (trunnion) axes. The rotary axis is physically connected to the 5th axis at the control cabinet. We want to designate the rotary axis c. The tilt axis is physically connected to the 4th axis at the control cabinet. We want to designate the tilt axis A.

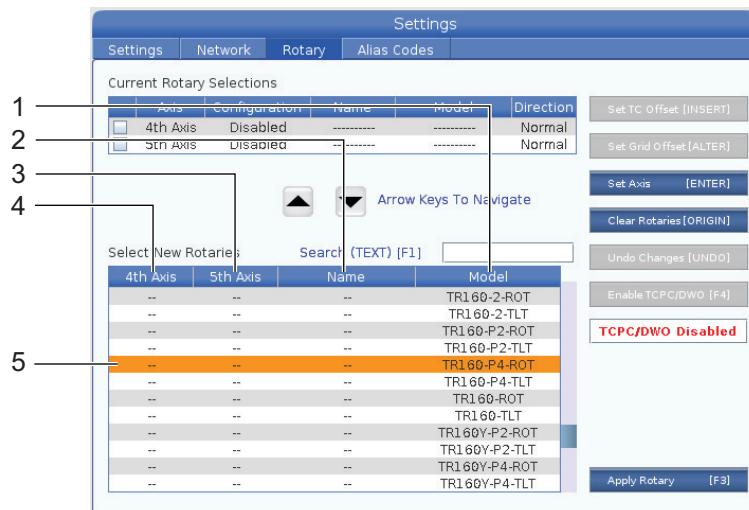
- Find the nameplate on your rotary unit. Record the values in the "MODEL NO." (model number) and "VER" (version) fields. On our example nameplate, we find that the model number is **TR160**, and the version is **P4**.

**F6.10:** Example rotary nameplate. [1] Model Number, [2] Version

2. On the Rotary Selection page, use the [CURSOR] keys or the jog handle to browse the list of rotary models to find your model.

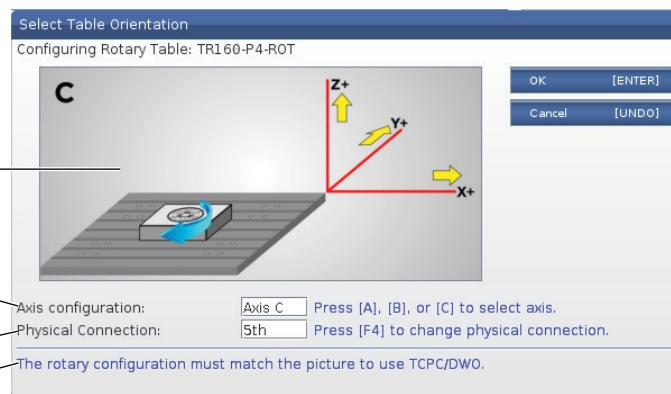
Dual-axis rotary units have two entries in the list: one for the rotary axis (**ROT**), and one for the tilt axis (**TLT**). Be sure to select the rotary model that matches both the model number and version on the nameplate. In the example below, the cursor highlights the rotary axis of the model that matches our example nameplate (**TR160-P4-ROT**).

- F6.11:** Rotary Selection Example. [1] Model Column, [2] Name Column, [3] Fifth Axis Column, [4] Fourth Axis Column, [5] Current Selection (Highlighted).



3. Press [**ENTER**]. The **Select Table Orientation** window appears.

- F6.12:** The **Select Table Orientation** Window. [1] Orientation Example Image, [2] Axis Configuration (Letter Assignment), [3] Physical Connection, [4] The Rotary Configuration Must Match the Picture to use TCPC/DWO.



4. Press **[A]**, **[B]**, or **[C]** to change the axis letter.

5. Press [F4] to toggle the physical connection setting between 4th and 5th.
6. Press [ENTER] to save the configuration to the **Select New Rotaries** table, or press [UNDO] to cancel.
7. Repeat steps 2-6 for the tilt axis, if applicable. In this example, we would now set up the TR160 tilt axis (**TR160-P4-TLT**).
8. After you complete the axis configuration, press [**EMERGENCY STOP**], and then press [F3] to apply the rotary parameters.
9. Cycle power.

## Custom Rotary Configurations

When you change a tool change offset or grid offset for an installed rotary, the control saves this information as a custom rotary configuration. You give this configuration a name that appears in the **Name** column of the **Current Rotary Selections** and **Select New Rotaries** tables.

The control keeps the default values in the base configuration and makes your custom configuration an option in the list of available rotaries. After you define a custom configuration for an axis, the control saves future changes to the same custom configuration name.

- F6.13:** Custom Rotary Configurations [1] In the **Current Rotary Selections** Table, and [2] in the **Select New Rotaries** Table.

The screenshot shows the 'Settings' menu with the 'Rotary' tab selected. There are two tables displayed:

**1. Current Rotary Selections:**

Axis	Configuration	Name	Model	Direction
4th Axis	A Axis	CUSTOM	TR160-P4-TLT	Normal
5th Axis	C Axis	CUSTOM	TR160-P4-ROT	Normal

**2. Select New Rotaries:**

4th Axis	5th Axis	Name	Model
A Axis	--	CUSTOM	TR160-P4-TLT
--	C Axis	CUSTOM	TR160-P4-ROT
--	--	--	HA2CTS-B
--	--	--	HA2CTS-PR

Buttons on the right side of the screen include: Set TC Offset [INSERT], Set Grid Offset [ALTER], Toggle Enable [ENTER], Clear Rotaries [ORIGIN], Undo Changes [UNDO], Disable TCP/C/DWO [F4], and **TCPC/DWO Enabled**.

The custom rotary configurations appear as options in the Select New Rotaries table. You can select them in the same way you would select a base rotary configuration. You can also save more than one custom configuration for the same rotary unit:

1. Start again with the base configuration of the installed rotary.
2. Configure the TC offset and grid offsets as needed.
3. Save this configuration with a new name.

You can also transfer custom rotary configurations to other machines. The control saves custom rotary files in the **User Data / My Rotary** folder in the Device Manager (**[LIST PROGRAM]**). You can transfer these files to the **User Data / My Rotary** folder on a different machine to make those configurations available in the **Select New Rotaries** table on that machine.

#### F6.14: Custom Rotary Files in the **User Data** Tab



## Rotary Tool Change Offset

After you define a rotary unit's axes in your machine control, you can set the tool change offset. This defines axis positions that put the rotary platter perpendicular to its defined axis.

1. In Handle Jog mode, jog the axes to make the platter face perpendicular to its defined axis. Use an indicator to verify perpendicularity.
2. Press **[SETTING]** and select the **Rotary** tab.
3. Highlight one of the axes in the **Current Rotary Selections** table.
4. Press **[INSERT]** to define the current axis position as the tool change offset position.
5. Type a name for your custom configuration, if prompted. You will see a prompt for a configuration name only when you make changes to a base configuration for the first time. Otherwise, the control saves your changes to the current custom configuration.

## Rotary Grid Offset

You use the rotary grid offset to set new zero positions for your rotary unit.

1. In Handle Jog mode, jog the axes to the positions you want to use as the offset positions.
2. Press **[SETTING]** and select the **Rotary** tab.
3. Highlight one of the axes in the **Current Rotary Selections** table.
4. Press **[ALTER]** to define the current axis positions as the grid offset positions.
5. Type a name for your custom configuration, if prompted. You will see a prompt for a configuration name only when you make changes to a base configuration for the first time. Otherwise, the control saves your changes to the current custom configuration.

## Disabling and Enabling Rotary Axes

A disabled rotary axis does not move, but it stays configured. Disabling a rotary axis is a good way to temporarily stop using a rotary axis without removing it completely from the machine.


**NOTE:**

*You can also disable and enable built-in rotary axes this same way.*

Enabled rotary axes appear with a filled-in checkbox in the **Current Rotary Selections** table.

- F6.15:** [1] Enabled Rotary Axis, [2] Disabled Rotary Axis.

Current Rotary Selections					
	Axis	Configuration	Name	Model	Direction
1	<input checked="" type="checkbox"/> 4th Axis	A Axis	Base	TR160-P4-TLT	Normal
2	<input type="checkbox"/> 5th Axis	C Axis	Base	TR160-P4-ROT	Normal
<b>ENTER</b>		Toggle axis enabled. Disabled remain configured but will not move.			

1. Highlight the axis you want to disable or enable.
2. Press **[EMERGENCY STOP]**.
3. Press **[ENTER]**.


**NOTE:**

*The control must not be in Jog mode when you disable an axis. If you get a Wrong Mode message, press **[MEMORY]** to change modes, and then press **[SETTING]** to return to the Rotary page.*

The control toggles the enabled state of the rotary axis.

4. Release **[EMERGENCY STOP]** to continue operation.

## 6.12.2 TCPC/DWO Activation

You can use Tool Center Point Control (TCPC) and Dynamic Work Offsets (DWO) if your rotary configuration is correct, and you have correctly set Machine Rotary Zero Point (MRZP) settings (255-257). Refer to page 239 for more information about TCPC. Refer to page 239 for more information about DWO.

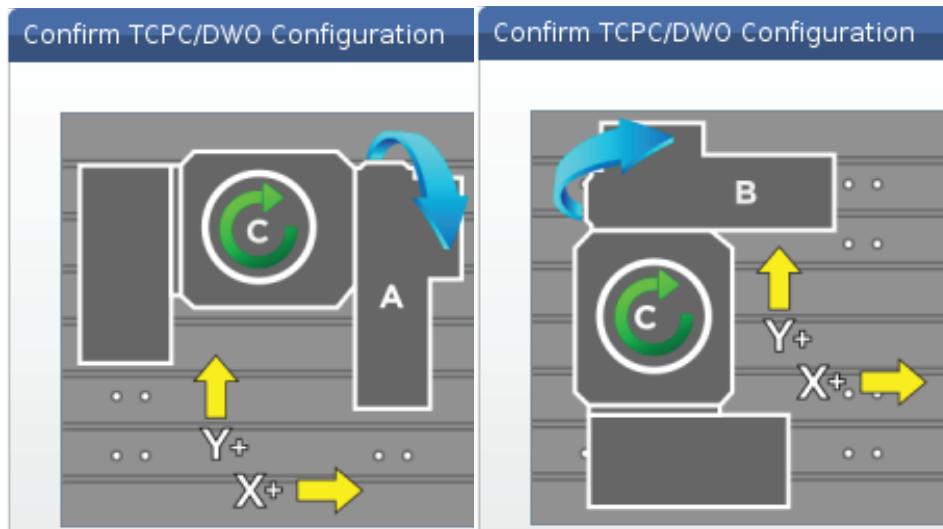

**NOTE:**

*To use Tool Center Point Control (TCPC) and Dynamic Work Offsets (DWO), your axis definitions and rotary installation must match the ANSI standard, where the A, B, and C Axes each rotate about the X, Y, and Z Axes, respectively. When you activate TCPC/DWO, you must confirm that your configuration is correct.*

1. On the **Rotary** page, press **[F4]**.

The **Confirm TCPC/DWO Configuration** pop-up window appears.

- F6.16:** Confirm TCPC/DWO Configuration Pop-up Window. [1] A- and C-Axis Configuration, [2] B- and C-Axis Configuration



1

2

2. If your rotary configuration matches the diagram, press **[ENTER]** to confirm so. This activates TCPC/DWO.

If your configuration does not match the diagram, you must adjust it to match; for example, you may need to redefine the axis letters or change the rotary unit's orientation.

3. After you activate TCPC/DWO, press F3 to save the rotary configuration. If you do not save the configuration, then TCPC/DWO is deactivated when you turn off the machine.

### 6.12.3 Machine Rotary Zero Point (MRZP)

The Machine Rotary Zero Point (MRZP) Offsets are control settings that define the centers of rotation for the rotary table relative to the home positions of the linear axes. The control uses the MRZP for Tool Center Point Control (TCPC) and Dynamic Work Offsets (DWO) for 4th- and 5th-axis machining. MRZP uses Settings 255, 256, and 257 to define the zero point.

**255** – Machine Rotary Zero Point X Offset

**256** – Machine Rotary Zero Point Y Offset

**257** – Machine Rotary Zero Point Z Offset

The value stored in each of these settings is the distance from the home position of a linear axis to the center of rotation of a rotary axis. The units are in current machine units (as defined by Setting 9).



**NOTE:**

*In machines with built-in 4th and 5th axes, such as the UMC-750, the initial MRZP offsets are set at the factory. You do not have to set initial values for these machines.*

You do the MRZP adjustment procedures when:

- You install a new rotary unit in a mill, and you want to use TCPC/DWO.
- The machine has been crashed.
- The machine's level has changed.
- You want to make sure that the MRZP settings are correct.

MRZP adjustment consists of (2) stages: rough and finish. The rough stage establishes MRZP values that the control uses for the finish stage. Generally, you do the rough stage only at new installations, or when you are not sure if the current MRZP settings are close enough to correct for the finish set procedure.

Both the rough and finish MRZP procedures use the work probe to generate values in macro variables, which you then transfer to the correct settings. You must change the values manually because the setting values cannot be set via macro. This protects them from accidental change in the middle of a program.

**NOTE:**

*These instructions assume that the probe system is installed and correctly calibrated.*

## MRZP Rough Set

This procedure establishes basic values for MRZP, which you then refine with the finish set process.

**NOTE:**

*You should do this procedure only at new rotary installations, or when you are unsure if your current MRZP values are close enough to do the finish set procedure.*

To do this procedure, you need to know the diameter of the center bore in your rotary platter.

1. Load or command the work probe into the spindle.
2. Jog the probe tip to approximately 0.4" (10 mm) above the approximate center of the ring gauge or bored hole.
3. Press **[EDIT]**.
4. Select the **VPS** tab, then use the **[RIGHT]** cursor arrow key to select **Probing, Calibration, MRZP Calibration**, and then **MRZP Rough Set**.
5. Highlight variable **C**, and then type the diameter of the ring gauge or bored hole. Press **[ENTER]**.
6. Highlight variable **H**, and then type the approximate distance between the rotary platter surface and the trunnion's center of rotation. Press **[ENTER]**.

**NOTE:**

*This distance is approximately 2" on a UMC-750; refer to your rotary unit's layout drawing to find this dimension for other units, or follow the procedure on page 246.*

7. Press **[CYCLE START]** to immediately run the probe program in MDI, or press **[F4]** to choose to output the probing program to the clipboard or MDI to run later.
8. When the probing program runs, it automatically places values in macro variables #10121, #10122, and #10123. These variables show the machine rotary zero point axis travel distance from the home position in the X, Y, and Z Axes. Record the values.



**NOTE:**

Press [**CURRENT COMMANDS**] and select the **Macro Vars** tab to view the variables. When the cursor is in the window, you can type a macro variable number and press the [**DOWN**] cursor arrow key to jump to that variable.

9. Enter the values from macro variables #10121, #10122, and #10123 into Settings 255, 256, and 257, respectively.
10. Do the Finish MRZP Set procedure.

## MRZP Finish Set

Follow this procedure to get final values for the MRZP settings. You can also use this procedure to check your current setting values against new readings, to make sure the current values are correct.

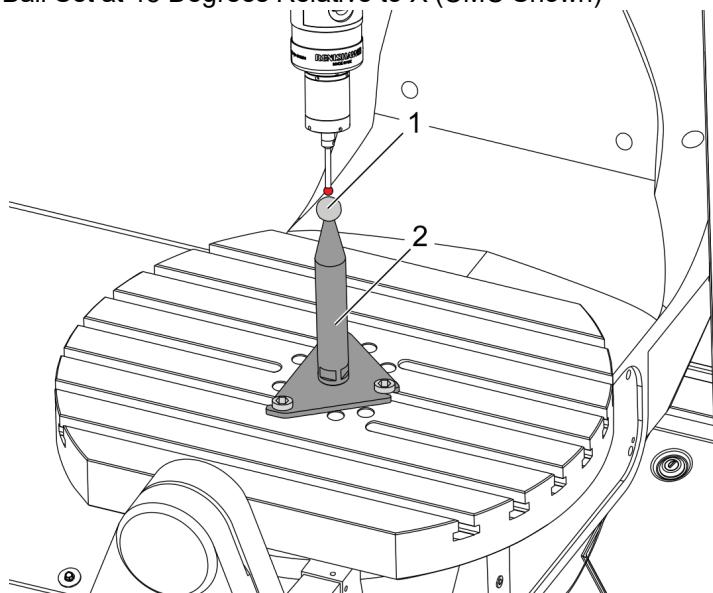
If you want to use this procedure to check your current setting values, make sure that the setting values you start with are close to correct to begin with. Values of zero generate an alarm. If the settings are too far off, the probe will not contact the gauge ball when it rotates positions during the cycle. The MRZP rough set process establishes appropriate starting values, so if you are unsure of the current values, you should do the MRZP rough set process first.

To do this procedure, you need a gauge ball with a magnetic base.

1. Place the gauge ball on the table.

**IMPORTANT:** So that the gauge ball post does not interfere with the probe, position the ball post at an angle of approximately 45 degrees to the X Axis.

**F6.17:** Gauge Ball Set at 45 Degrees Relative to X (UMC Shown)



2. Load or command the work probe into the spindle.
3. Position the work probe above the tooling ball.
4. Press [EDIT].
5. Select the **VPS** tab, then use the [**RIGHT**] cursor arrow key to select **Probing**, **Calibration**, **MRZP Calibration**, and then **MRZP Finish Set**.
6. Highlight variable **B**, and then type the gauge ball diameter. Press [**ENTER**].
7. Press [**CYCLE START**] to immediately run the probe program in MDI, or press [**F4**] to choose to output the probing program to the clipboard or MDI to run later.
8. When the probing program runs, it automatically places values in macro variables #10121, #10122, and #10123. These variables show the machine rotary zero point axis travel distance from the home position in the X, Y, and Z Axes. Record the values.



**NOTE:**

Press [**CURRENT COMMANDS**] and select the **Macro Vars** tab to view the variables. When the cursor is in the variable list, you can type a macro variable number and press the [**DOWN**] cursor arrow key to jump to that variable.

9. Enter the values from macro variables #10121, #10122, and #10123 into Settings 255, 256, and 257, respectively.

## 6.12.4 Creating Five-Axis Programs

### Offsets

1. Press **[OFFSET]** and select the **WORK** tab.
2. Jog the axes to the zero point of the workpiece. Refer to page **171** for jogging information.
3. Highlight the axis and offset number.
4. Press **[PART ZERO SET]** and the current machine position is automatically stored in that address.

**CAUTION:**

*If you use automatically generated tool length offsets, you should leave the Z-Axis work offset values at zero. Nonzero Z-Axis work offset values interfere with automatically generated tool length offsets, and can cause a machine crash.*

5. The X and Y work coordinate offsets are always given as negative values from machine zero. Work coordinates are entered into the table as a number only. To enter an X value of **X-2.00** into **G54**, highlight the **X Axis** column in the **G54** row, type **-2.0**, and press **[F1]** to set the value.

### Five-Axis Programming Notes

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

### G-codes

**G93** inverse time feed mode must be in effect for simultaneous 4- or 5-axis motion; however, if your mill supports Tool Center Point Control (**G234**), you may use **G94** (feed per minute). Refer to **G93** on page **367** for more information.

Limit the post processor (CAD/CAM software) to a maximum **G93 F** value of 45000. This is the maximum allowable feedrate in **G93** inverse time feed mode.

## M-codes

**IMPORTANT:** When doing any non 5-axis motion, engage the rotary axes brakes. Cutting with the brakes off causes excessive wear in the gear sets.

M10/M11 engages/disengages the fourth axis brake.

M12/M13 engages/disengages the fifth axis brake.

When in a 4 or 5 axis cut, the machine pauses between blocks. This pause is due to the Rotary Axes brakes releasing. To avoid this dwell and allow for smoother program execution, program an M11 and/or M13 before the G93. The M-codes disengage the brakes, resulting in a smoother and uninterrupted flow of motion. Remember that if the brakes are never re-engaged, they remain off indefinitely.

## Settings

Settings used for 4th and 5th axis programming include:

For the 4th axis:

- Setting 34 - 4th Axis Diameter

For the 5th axis:

- Setting 79 - 5th-Axis Diameter

For the Axis mapped to the 4th or 5th Axis:

- Setting 48 - Mirror Image A-Axis
- Setting 80 - Mirror Image B-Axis
- Setting 250 - Mirror Image C-Axis

Setting 85 - Maximum Corner Rounding should be set to 0.0500 for 5-axis cutting. Settings lower than 0.0500 move the machine closer to an exact stop and cause uneven motion.

You can also use G187 Pn Ennnn to set the smoothness level in the program to slow the axes down. G187 temporarily overrides Setting 85. Refer to page 394 for more information.

## Jogging the 4th and 5th Axes

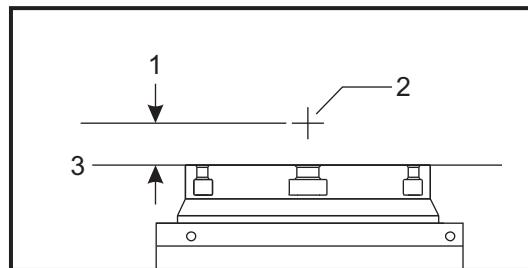
Jogging the rotary axes works like jogging the linear axes: you select an axis and a jog rate, and then you use the jog handle or jog keys to move the axis. In Hand Jog mode, press the **[+A/C +B]** or **[-A/C -B]** jog key to select the 4th axis. To select the 5th axis, press **[SHIFT]** and then **[+A/C +B]** or **[-A/C -B]**.

The control remembers the last rotary axis you selected, and **[+A/C +B]** or **[-A/C -B]** continue to select that axis until you select the other axis. For example, after you select the 5th axis as described above, each time you press **[+A/C +B]** or **[-A/C -B]** selects the 5th axis for jogging. To select the 4th axis again, press SHIFT and then **[+A/C +B]** or **[-A/C -B]**. Now, each subsequent press of **[+A/C +B]** or **[-A/C -B]** selects the 4th axis.

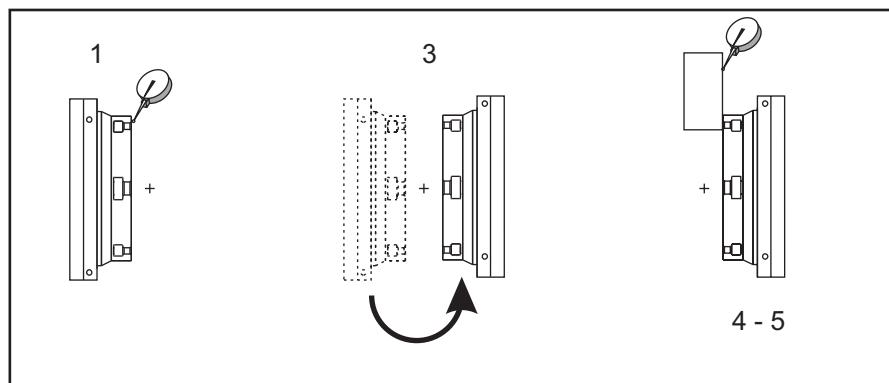
### 6.12.5 Tilt Axis Center-of-Rotation Offset (Tilting Rotary Products)

This procedure determines the distance between the plane of the rotary axis platter and the tilt axis centerline on tilting rotary products. Some CAM software applications require this offset value. You also need this value to rough-set the MRZP offsets. Refer to page **241** for more information.

- F6.18:** Tilt Axis Center-of-Rotation Offset Diagram (side view): [1] Tilt Axis Center-of-Rotation Offset, [2] Tilt Axis, [3] Plane of the Rotary Axis Platter.



- F6.19:** Tilt Axis Center-of-Rotation Illustrated Procedure. Numeric labels in this diagram correspond to the step numbers in the procedure.



1. Jog the tilt axis until the rotary platter is vertical. Attach a dial indicator to the machine spindle (or other surface independent of table motion) and indicate the platter face. Set the dial indicator to zero.



**NOTE:**

*The rotary unit's orientation on the table determines which linear axis to jog in these steps. If the tilt axis is parallel to the X Axis, use the Y Axis in these steps. If the tilt axis is parallel to the Y Axis, use the X Axis in these steps.*

2. Set the X- or Y-Axis operator position to zero.
3. Jog the tilt Axis 180 degrees.
4. Indicate the platter face from the same direction as the first indication:
  - a. Hold a 1-2-3 block against the platter face.
  - b. Indicate the face of the block that rests against the platter face.
  - c. Jog the X or Y Axis to zero the indicator against the block.
5. Read the new X- or Y-Axis operator position. Divide this value by 2 to determine the tilt axis center-of-rotation offset value.

## 6.13 Macros (Optional)

## 6.13.1 Macros Introduction

**NOTE:**

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

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

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

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

- **Tools For Immediate, On-Table Fixturing** - You can semi-automate many setup procedures to assist the machinist. You can reserve tools for immediate situations that you did not anticipate in your application design. For instance, suppose a company uses a standard clamp with a standard bolt hole pattern. If you discovered, after setup, that a fixture needs an additional clamp, and suppose that you programmed macro subprogram 2000 to drill the bolt pattern of the clamp, then you only need this two-step procedure to add the clamp to the fixture:
  - a) Jog the machine to the X, Y, and Z coordinates and angle where you want to place the clamp. Read the position coordinates from the machine display.
  - b) Execute this command in MDI mode:

```
G65 P2000 Xnnn Ynnn Znnn Ann ;
```

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

- **Simple Patterns That Are Repeated**- You can define and store repeated patterns with macros. For example:
  - a) Bolt hole patterns
  - b) Slotting
  - c) Angular patterns, any number of holes, at any angle, with any spacing
  - d) Specialty milling such as soft jaws
  - e) Matrix Patterns, (e.g. 12 across and 15 down)
  - f) Fly-cutting a surface, (e.g. 12 inches by 5 inches using a 3 inch fly cutter)
- **Automatic Offset Setting Based On The Program** - With macros, coordinate offsets can be set in each program so that setup procedures become easier and less error-prone (macro variables #2001–2800).

- **Probing** - Using a probe enhances the capabilities of the machine, some examples are:
  - a) Profiling of a part to determine unknown dimensions for machining.
  - b) Tool calibration for offset and wear values.
  - c) Inspection prior to machining to determine material allowance on castings.
  - d) Inspection after machining to determine parallelism and flatness values as well as location.

## Useful G and M Codes

M00, M01, M30 - Stop Program

G04 - Dwell

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

M29 - Set output relay with M-FIN.

M59 - Set output relay.

M69 - Clear output relay.

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

M97 Pxx - Local Sub Routine Call

M98 Pxx - Sub Program Call

M99 - Sub Program Return or Loop

G103 - Block Lookahead Limit. No cutter comp allowed.

M109 - Interactive User Input (refer to page 429)

## Round Off

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

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

it may give a false reading. A safer way of programming this would be

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

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

## Look-ahead

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

```
#12012 = 1 ;
G04 P1. ;
#12012 = 0 ;
```

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

```
G103 P1 (See the G-code section of the manual for a further
explanation of G103) ;
;
#12012=1 ;
G04 P1. ;
;
;
;
#12012=0 ;
```

## Block Look-Ahead and Block Delete

The Haas control uses block look-ahead to read and prepare for blocks of code that come after the current block of code. This lets the control transition smoothly from one motion to the next. G103 limits how far ahead the control looks at blocks of code. The Pnnaddress code in G103 specifies how far ahead the control is allowed to look. For additional information, refer to G103 on page **371**.

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

Using a

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

before a block with

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

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

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

## 6.13.2 Operation Notes

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

## 6.13.3 Macro Variable Display Page

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

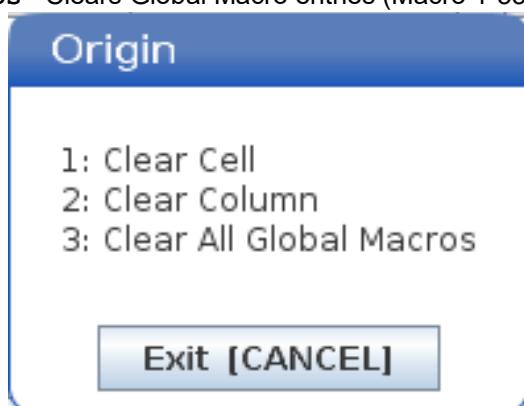


**NOTE:**

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

1. Press **[CURRENT COMMANDS]** and use navigation keys to get to the **Macro Vars** page.  
As the control interprets a program, the variable changes and results are displayed on the **Macro Vars** display page.
2. Enter a value (maximum is 999999.000000) and then press **[ENTER]** to set the macro variable. Press **[ORIGIN]** to clear macro variables, this displays the Origin clear entry popup. Press number 1 - 3 to make a selection or press **[CANCEL]** to exit.

- F6.20: Origin Clear Entry Popup. 1: **Clear Cell** - Clears the highlighted cell to zero. 2: **Clear Column** - Clears the active cursor column entries to zero. 3: **Clear All Global Macros** - Clears Global Macro entries (Macro 1-33, 10000-10999) to zero.



3. To search for a variable, enter the macro variable number and press the up or down arrow.
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. A G103 without the P value can be added after the macro variable blocks in the program. For a macro program to operate correctly it is recommended that the G103 P1 be left in the program during loading of variables. For more details about G103 see the G-code section of the manual.

#### 6.13.4 Display Macro Variables in the Timers And Counters Window

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

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

1. Press [**CURRENT COMMANDS**].
2. Use the navigation keys to select the **TIMERS** page.
3. Highlight the **Macro Label #1** name or **Macro Label #2** name.
4. Key in a new name, and press [**ENTER**].
5. Use arrow keys to pick the **Macro Assign #1** or **Macro Assign #2** entry field (corresponding to your chosen **Macro Label** name).
6. Key in the macro variable number (without #) and press [**ENTER**].

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

## 6.13.5 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

**T6.2:** Alphabetic Address Table

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

Alternate Alphabetic Addressing

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

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

G65 P9910 A1 B2 C3 ;

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

### Integer Argument Passing (no decimal point)

Address	Variable	Address	Variable	Address	Variable
A	.0001	J	.0001	S	1.
B	.0002	K	.0001	T	1.
C	.0003	L	1.	U	.0001

Address	Variable	Address	Variable	Address	Variable
D	1.	M	1.	V	.0001
E	1.	N	-	W	.0001
F	1.	O	-	X	.0001
G	-	P	-	Y	.0001
H	1.	Q	.0001	Z	.0001
I	.0001	R	.0001		

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

Example:

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

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

### 6.13.6 Macro Variables

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

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

## Local Variables

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

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

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

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

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

## Global Variables

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


**NOTE:**

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

Sometimes, factory-installed options use global variables, for example, probing and pallet changers, etc. Refer to the Macro Variables Table on page 257 for global variables and their use.


**CAUTION:**

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

## System Variables

System variables let you interact with a variety of control conditions. System variable values can change the function of the control. When a program reads a system variable, it can modify its behavior based on the value in the variable. Some system variables have a Read Only status; this means that you cannot modify them. Refer to the Macro Variables Table on page 257 for a list of system variables and their use.

### 6.13.7 Macro Variables Table

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

NGC Variable	Legacy Variable	Usage
#0	#0	Not a number (read only)
#1- #33	#1- #33	Macro call arguments
#10000- #10149	#100- #149	General-purpose variables saved on power off
#10150- #10199	#150- #199	Probe values (if installed)

<b>NGC Variable</b>	<b>Legacy Variable</b>	<b>Usage</b>
#10200- #10399	N/A	General-purpose variables saved on power off
#10400- #10499	N/A	General-purpose variables saved on power off
#10500- #10549	#500-#549	General-purpose variables saved on power off
#10550- #10599	#550-#599	Probe calibration data (if installed)
#10600- #10699	#600- #699	General-purpose variables saved on power off
#10700- #10799	N/A	General-purpose variables saved on power off
#700- #749	#700- #749	Hidden variables for internal use only
#709	#709	Used for the Fixture Clamp Input. Do not use for general purpose.
#10800- #10999	#800- #999	General-purpose variables saved on power off
#11000- #11063	N/A	64 discrete inputs (read only)
#1064- #1068	#1064- #1068	Maximum axis loads for X, Y, Z, A, and B Axes, respectively
#1080- #1087	#1080- #1087	Raw analog to digital inputs (read only)
#1090- #1098	#1090- #1098	Filtered analog to digital inputs (read only)
#1098	#1098	Spindle load with Haas vector drive (read only)
#1264- #1268	#1264- #1268	Maximum axis loads for C, U, V, W, and T-axes respectively
#1601- #1800	#1601- #1800	Number of flutes on tools #1 through 200
#1801- #2000	#1801- #2000	Maximum recorded vibrations of tools 1 through 200
#2001- #2200	#2001- #2200	Tool length offsets
#2201- #2400	#2201- #2400	Tool length wear
#2401- #2600	#2401- #2600	Tool diameter/radius offsets
#2601- #2800	#2601- #2800	Tool diameter/radius wear
#3000	#3000	Programmable alarm
#3001	#3001	Millisecond timer

NGC Variable	Legacy Variable	Usage
#3002	#3002	Hour timer
#3003	#3003	Single block suppression
#3004	#3004	Override <b>[FEED HOLD]</b> control
#3006	#3006	Programmable stop with message
#3011	#3011	Year, month, day
#3012	#3012	Hour, minute, second
#3020	#3020	Power on timer (read only)
#3021	#3021	Cycle start timer
#3022	#3022	Feed timer
#3023	#3023	Present part timer (read only)
#3024	#3024	Last complete part timer (read only)
#3025	#3025	Previous part timer (read only)
#3026	#3026	Tool in spindle (read only)
#3027	#3027	Spindle RPM (read only)
#3028	#3028	Number of pallets loaded on receiver
#3030	#3030	Single block
#3032	#3032	Block delete
#3033	#3033	Opt stop
#3034	N/A	Safe Run (read only)
#3196	#3196	Cell safe timer
#3201- #3400	#3201- #3400	Actual diameter for tools 1 through 200
#3401- #3600	#3401- #3600	Programmable coolant positions for tools 1 through 200
#3901	#3901	M30 count 1

NGC Variable	Legacy Variable	Usage
#3902	#3902	M30 count 2
#4001- #4021	#4001- #4021	Previous block G-code group codes
#4101- #4126	#4101- #4126	Previous block address codes.   <b>NOTE:</b> (1) Mapping of 4101 to 4126 is the same as the alphabetic addressing of Macro Arguments section; e.g., the statement X1.3 sets variable #4124 to 1.3.
#5001- #5006	#5001- #5006	Previous block end position
#5021- #5026	#5021- #5026	Present machine coordinate position
#5041- #5046	#5041- #5046	Present work coordinate position
#5061- #5069	#5061- #5069	Present skip signal position - X, Y, Z, A, B, C, U, V, W
#5081- #5086	#5081- #5086	Present tool offset
#5201- #5206	#5201- #5206	G52 work offsets
#5221- #5226	#5221- #5226	G54 work offsets
#5241- #5246	#5241- #5246	G55 work offsets
#5261- #5266	#5261- #5266	G56 work offsets
#5281- #5286	#5281- #5286	G57 work offsets
#5301- #5306	#5301- #5306	G58 work offsets
#5321- #5326	#5321- #5326	G59 work offsets
#5401- #5500	#5401- #5500	Tool feed timers (seconds)
#5501- #5600	#5501- #5600	Total tool timers (seconds)

<b>NGC Variable</b>	<b>Legacy Variable</b>	<b>Usage</b>
#5601- #5699	#5601- #5699	Tool life monitor limit
#5701- #5800	#5701- #5800	Tool life monitor counter
#5801- #5900	#5801- #5900	Tool load monitor maximum load sensed so far
#5901- #6000	#5901- #6000	Tool load monitor limit
#6001- #6999	#6001- #6999	Reserved. Do not use.
#6198	#6198	NGC/CF flag
#7001- #7006	#7001- #7006	G110 (G154 P1) additional work offsets
#7021- #7026	#7021- #7026	G111 (G154 P2) additional work offsets
#7041- #7386	#7041- #7386	G112 - G129 (G154 P3 - P20) additional work offsets
#7501- #7506	#7501- #7506	Pallet priority
#7601- #7606	#7601- #7606	Pallet status
#7701- #7706	#7701- #7706	Part program numbers assigned to pallets
#7801- #7806	#7801- #7806	Pallet usage count
#8500	#8500	Advanced Tool Management (ATM) group ID
#8501	#8501	ATM percent of available tool life of all tools in the group
#8502	#8502	ATM total available tool usage count in the group
#8503	#8503	ATM total available tool hole count in the group
#8504	#8504	ATM total available tool feed time (in seconds) in the group
#8505	#8505	ATM total available tool total time (in seconds) in the group
#8510	#8510	ATM next tool number to be used
#8511	#8511	ATM percent of available tool life of the next tool
#8512	#8512	ATM available usage count of the next tool
#8513	#8513	ATM available hole count of the next tool

## Macros (Optional)

---

NGC Variable	Legacy Variable	Usage
#8514	#8514	ATM available feed time of the next tool (in seconds)
#8515	#8515	ATM available total time of the next tool (in seconds)
#8550	#8550	Individual tool ID
#8551	#8551	Number of flutes of tools
#8552	#8552	Maximum recorded vibrations
#8553	#8553	Tool length offsets
#8554	#8554	Tool length wear
#8555	#8555	Tool diameter offsets
#8556	#8556	Tool diameter wear
#8557	#8557	Actual diameter
#8558	#8558	Programmable coolant position
#8559	#8559	Tool feed timer (seconds)
#8560	#8560	Total tool timers (seconds)
#8561	#8561	Tool life monitor limit
#8562	#8562	Tool life monitor counter
#8563	#8563	Tool load monitor maximum load sensed so far
#8564	#8564	Tool load monitor limit
#9000	#9000	Thermal comp accumulator
#9000- #9015	#9000- #9015	Reserved (duplicate of axis thermal accumulator)
#9016#9016	#9016#9016	Thermal spindle comp accumulator
#9016- #9031	#9016- #9031	Reserved (duplicate of axis thermal accumulator from spindle)
#10000- #10999	N/A	General purpose variables
#11000- #11255	N/A	Discrete inputs (read only)