Haas Automation, Inc.

UMC-750

Operator's Manual Supplement
96-8250
Revision A
January 2014
English
Original Instructions

To get translated versions of this Manual:
1. Go to ***www.HaasCNC.com***
2. See *Owner Resources* (bottom of page)
3. Select *Manuals and Documentation*

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

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, (ii) any Haas Product was improperly repaired or serviced by Customer, an unauthorized service technician, or other unauthorized person, (iii) Customer or any person makes or attempts to make any modification to any Haas Product without the prior written authorization of Manufacturer, and/or (iv) any Haas Product was used for any non-commercial use (such as personal or household use). This warranty does not cover damage or defect due to an external influence or matters beyond the reasonable control of Manufacturer, including, but not limited to, theft, vandalism, fire, weather condition (such as rain, flood, wind, lightning, or earthquake), or acts of war or terrorism.

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

Limitation of Liability and Damages

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

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

Entire Agreement

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

Transferability

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

Miscellaneous

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

Customer Feedback

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

You also can find an electronic copy of this manual and other useful information on our website under the "Owner's Resources" tab. Join Haas owners online and be a part of the greater CNC community at these sites:

atyourservice.haascnc.com
At Your Service: The Official Haas Answer and Information Blog

www.facebook.com/HaasAutomationInc
Haas Automation on Facebook

www.twitter.com/Haas_Automation
Follow us on Twitter

www.linkedin.com/company/haas-automation
Haas Automation on LinkedIn

www.youtube.com/user/haasautomation
Product videos and information

www.flickr.com/photos/haasautomation
Product photos and information

Customer Satisfaction Policy

Dear Haas Customer,

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

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

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

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

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

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

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

International:

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

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

Declaration of Conformity

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

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

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

Person authorized to compile technical file:

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

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

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

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

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

ETL LISTED
CONFORMS TO
NFPA STD 79
ANSI/UL STD 508
UL SUBJECT 2011
CERTIFIED TO
CAN/CSA STD C22.2 N O.73
9700845

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

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

Original Instructions

How to Use This Manual

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

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

Declaration of Warnings

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

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

Text Conventions Used in this Manual

Description	Text Example
Code Block text gives program examples.	`G00 G90 G54 X0. Y0.;`
A **Control Button Reference** gives the name of a control key or button that you are to press.	Press **[CYCLE START]**.
A **File Path** describes a sequence of file system directories.	*Service > Documents and Software >...*
A **Mode Reference** describes a machine mode.	MDI
A **Screen Element** describes an object on the machine's display that you interact with.	Select the **`SYSTEM`** tab.
System Output describes text that the machine control displays in response to your actions.	`PROGRAM END`
User Input describes text that you should enter into the machine control.	`G04 P1.;`

Contents

Chapter 1: Introduction

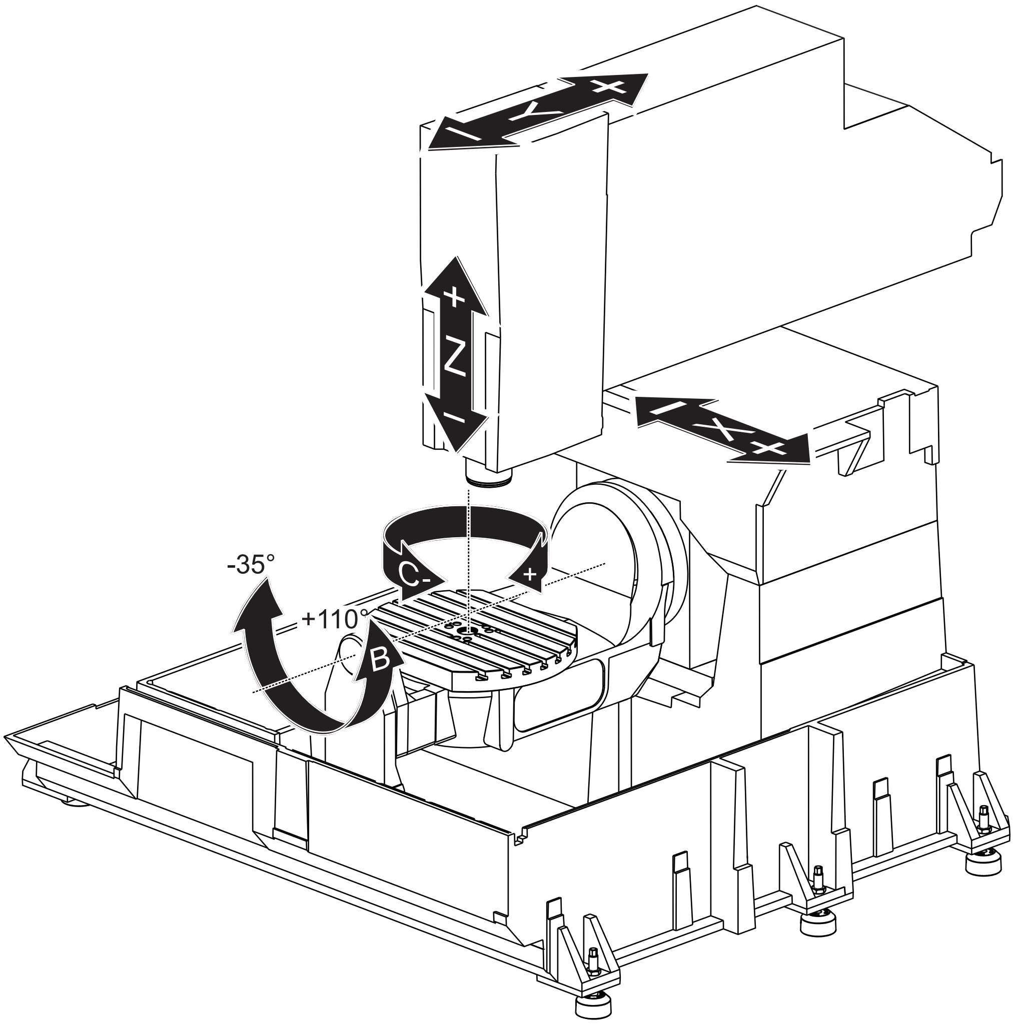
1.1 Overview

This operator's manual supplement describes the unique features and functions of the UMC-750. Refer to the Mill Operator's Manual (P/N 96-8000) for machine and control operation.

Specific details about the UMC-750 itself, including information that is beyond the scope of this document, can be found at www.HaasCNC.com.

1.2 Axis Definitions

This diagram illustrates the (5) axes available on the UMC-750.



1.3 UMC-750 Specifications

Travels		
	S.A.E	Metric
X Axis	30"	762 mm
Y Axis	20"	508 mm
Z Axis	20"	508 mm
C-Axis Rotation	360° Rotation	
B-Axis Tilt	-35° to +110°	
Spindle Nose to Table (~ min.)	4"	102 mm
Spindle Nose to Table (~ max.)	24"	610 mm
For detailed machine dimensions, including work envelope information, refer to the UMC-750 Machine Layout Drawing on www.haascnc.com.		

Table		
Width	19.7"	500 mm
Length	24.8"	630 mm
T-Slot Width	5/8"	16 mm
T-Slot Center Distance	2.48"	63 mm
Number of Standard T-Slots	7	
Max. Weight on Table (evenly distributed)	660 lb	300 kg

General Requirements		
Air Required	4 scfm, 100 psi	113 L/min, 6.9 bar
Coolant Capacity	75 gal	284 L
Power Requirement, Low Voltage	195-260 VAC / 100A	
Power Requirement, High Voltage	354-488 VAC / 50A	
Machine Weight	18,000 lb	8165 kg

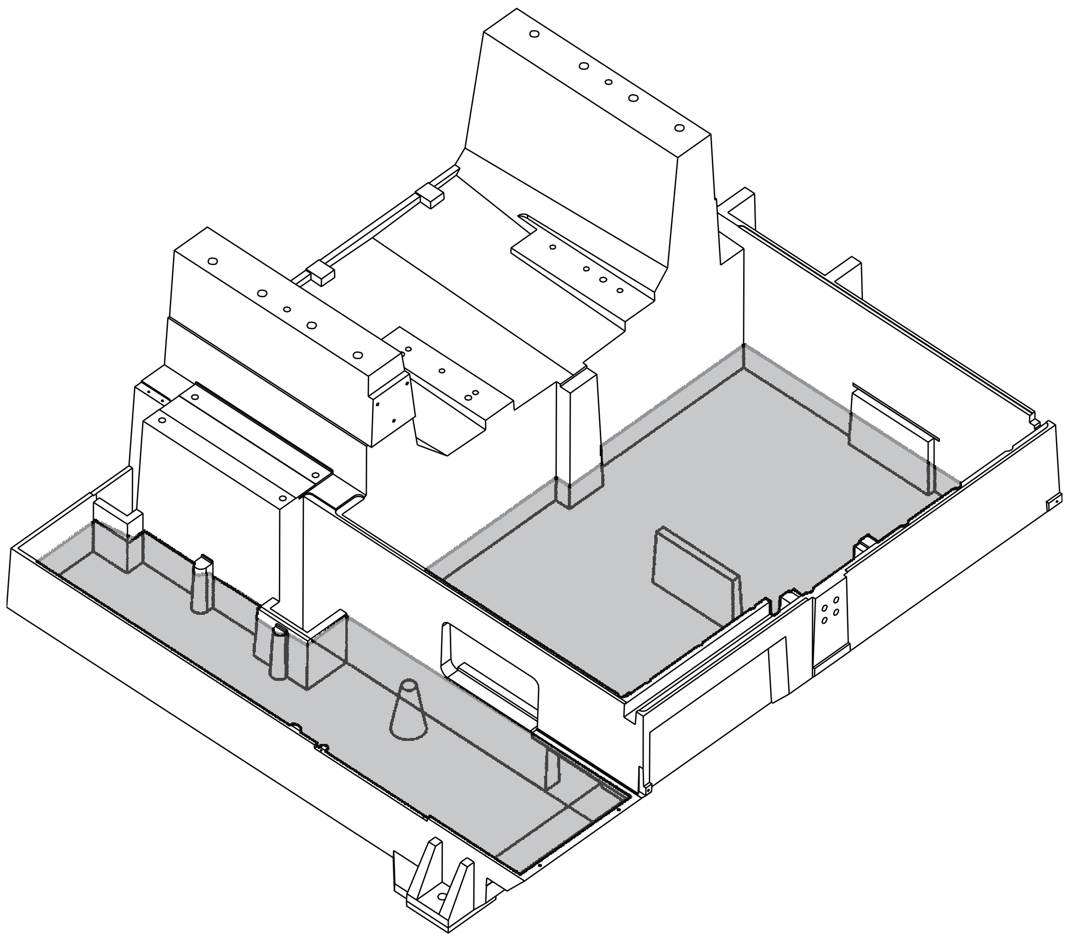
Standard Features
Tool Center Point Control (TCPC), Dynamic Work Offsets (DWO), Remote Jog Handle*, Second Home*, Macros*, Spindle Orientation (SO)*, Coordinate Rotation and Scaling (COORD)*, TSC-Ready, Wireless Intuitive Probing System (WIPS) *Refer to the Mill Operator’s Manual (96-8000) for information on these features.

Chapter 2: Integrated Coolant Tank

2.1 Introduction

The UMC-750 coolant tank is integrated into the machine base.

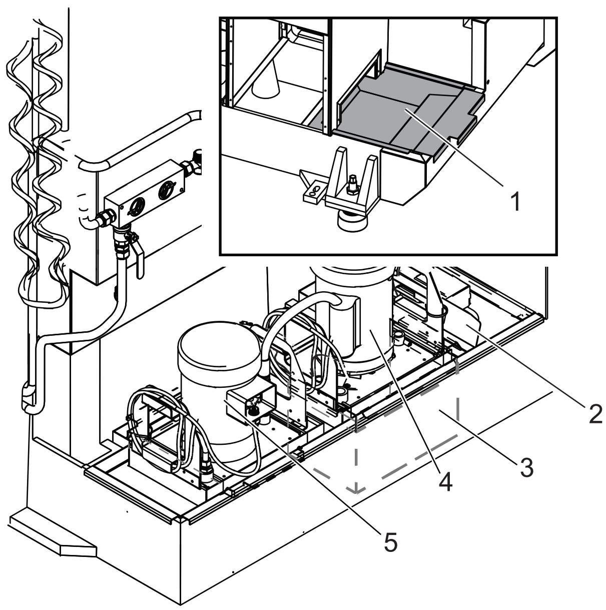
F2.1: UMC -750 Integrated Coolant Tank



2.1.1 Coolant Pump Location

The coolant pumps are on the tool changer side of the machine, behind the chip conveyor. The standard coolant box filter is mounted below the standard coolant pump.

F2.2: UMC-750 Coolant Pump Location: [1] Chip Tray, [2] Gate Filter, [3] Coolant Box Filter, [4] Standard Coolant Pump, [5] TSC Coolant Pump

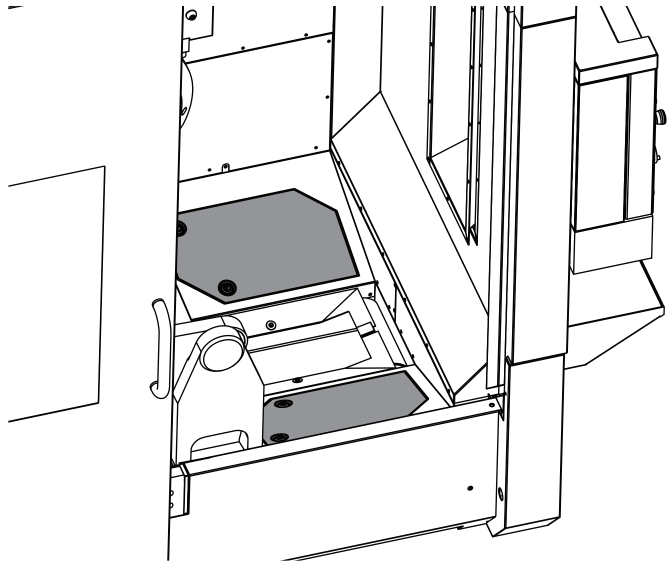


2.2 Coolant Tank Clean-Out

To clean out the coolant tank:

1. Remove the coolant pumps.
2. Remove and empty the coolant box filter.
3. Remove and empty the chip tray.
4. Remove and clean the gate filter.
5. Use a wet/dry vacuum or similar device to remove the used coolant from the tank.
6. For more thorough cleaning, open the coolant tank access panels in the bottom of the work space.

F2.3: Coolant Tank Access Panels



7. Add coolant to the tank and install the gate filter, coolant box filter, chip tray, and coolant pumps. Install the coolant tank access panels if you removed them.

Chapter 3: Wireless Intuitive Probing System (WIPS)

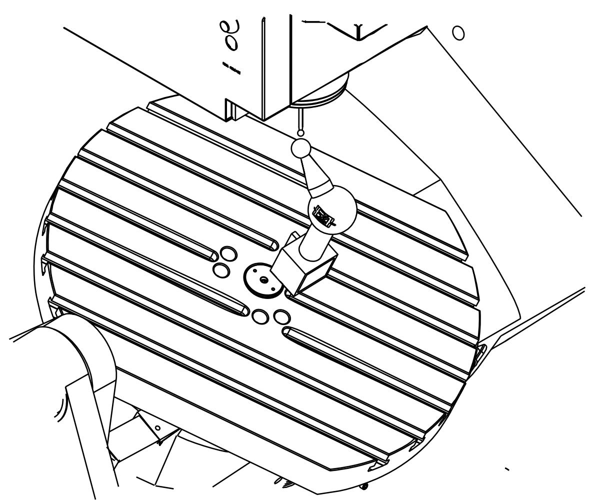
3.1 UMC WIPS Basics

The Wireless Intuitive Probing System (WIPS) comes standard with the UMC-750. This system can perform all of the standard probe routines found in the WIPS templates, and also includes special probe routines specifically for the UMC. These special probe routines use a tooling ball on a magnetic base to automatically find the machine's centers of rotation. Refer to page **11** for more information on this process.

Normally, you use WIPS to set tool and work offsets, but the UMC-750 includes a master gage length tool in case you need to set offsets manually (if, for example, a probe stylus breaks or the batteries lose power). The gage length tool included with your machine has a unique length that is etched on the tool, and also stored as Setting 244 in the machine's control.

NOTE: *If you set the tool length offsets manually, you must also set the Z-Axis work offset manually.*

F3.1: UMC-750 Tooling Ball



Chapter 4: Machine Rotary Zero Point (MRZP) Offsets

4.1 Introduction

The Machine Rotary Zero Point (MRZP) Offsets are control parameters that define the centers of rotation for the rotary table relative to the home positions of the linear axes. Parameters 1306, 1307, and 1308 define:

1306 – Machine Rotary Zero Point X Offset

The location of the B-Axis rotary center point, relative to the X-Axis home position.

1307 – Machine Rotary Zero Point Y Offset

The location of the C-Axis rotary center point, relative to the Y-Axis home position.

1308 – Machine Rotary Zero Point Z Offset

The location of the B-Axis rotary center point, relative to the Z-Axis home position.

The value stored in each of these parameters is the distance from the home position of a linear axis to the center of rotation of a rotary axis. The units are in inches, scaled by 10,000. For example, the default value for 1306 is -150000, representing an offset of -15.0" from machine X Zero. These parameter values are always in inches, regardless of the units selected in Setting 9.

The initial MRZP offsets are set at the factory. Over time, you may need to check and adjust these parameters, due to environmental factors, machine use, or in the event of a machine crash. This section tells you how to make sure the MRZP offsets are correct.

NOTE: *These instructions assume that the probe system is installed and correctly calibrated. Refer to the Haas WIPS Manual (96-10002) for the calibration procedure.*

4.1.1 Check MRZP Offsets with WIPS

The MRZP offsets can change over time. To make sure the UMC-750 MRZP Offsets are correct:

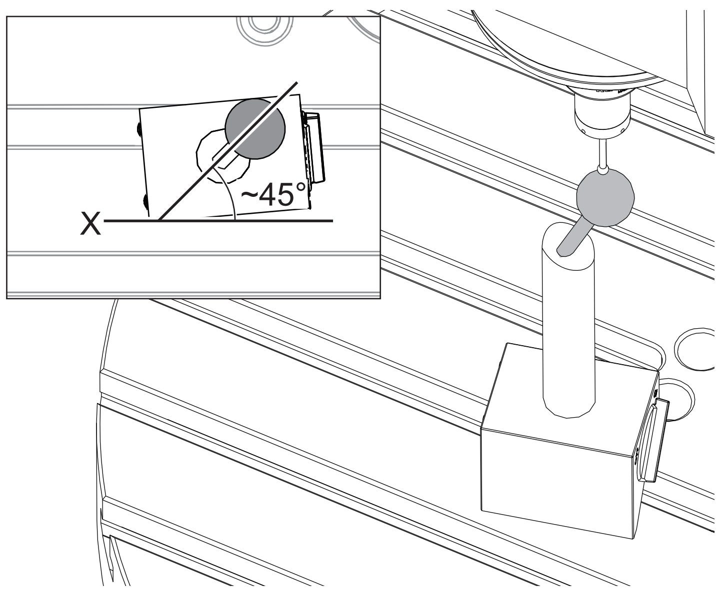
1. Record the original value of Parameter 1314, and then change it to 0. Refer to page **13** for more information on changing parameter values.

WARNING: *Parameters define machine operations in very specific ways. If you must change parameters, change only those for which you have specific and explicit factory instructions, such as those given here.*

2. Place the tooling ball on the table.

IMPORTANT: *To make sure the tooling ball post does not interfere with the probe, position the ball post at an angle of approximately 45 degrees to the X Axis.*

F4.1: Tooling Ball Set at 45 Degrees Relative to X



3. Place the work probe in the spindle.
4. Position the work probe over the gage ball.
5. Press **[MDI/DNC]**. Press **[PROGRAM]**, and then select the `VQC` tab.
6. Highlight `MRZP SET` from the VQC Menu, and then press **[ENTER]**.
7. Highlight `FINISH MRZP SET`, and then press **[ENTER]**.
8. Follow the prompts to generate the probe program. Enter MDI mode and press **[CYCLE START]**.

9. The probe automatically places values in macro variables 106 through 108 and 121 through 123. These variables show the machine rotary zero point axis travel distance from the home position in the X, Y, and Z Axes.
10. If the MRZP locations have changed through foundation settling or other external influences on the machine, enter the values from macro variables 106, 107, and 108 into Parameters 1306, 1307, and 1308. Remember that 10 units = 0.001".
11. Change Parameter 1314 back to its original value.

4.2 Parameter Change

WARNING: *Parameters define machine operations in very specific ways. Never change a parameter, except under explicit factory instructions. Incorrectly set parameters can cause severe damage to your machine and void the warranty.*

1. Change Setting 7 to `OFF`.
   a. Press **[SETTING/GRAPHIC]** until the Settings menu appears.
   b. Press **[7]**, and then press the **[DOWN]** cursor arrow.
   c. Press the **[RIGHT]** cursor arrow to change the setting to `OFF`.
   d. Press**[ENTER]** to save the change.
2. Press **[EMERGENCY STOP]**.
3. Press **[PARAMETER/DIAGNOSTIC]**.
4. Type the number of the parameter that you want to change.
5. Press the **[DOWN]** cursor arrow.
6. Record the parameter’s current value, in case you need to change it back.
7. Type the new parameter value.
8. Press **[ENTER]** to save the change.
9. Repeat steps 3 through 7 for any other parameters that you need to change.
10. Reset **[EMERGENCY STOP]**.
11. Press **[RESET]**.
12. Change Setting 7 back to `ON`.

Chapter 5: G234 - Tool Center Point Control (TCPC)

5.1 Introduction

G234 Tool Center Point Control (TCPC) is a software feature in the Haas CNC control that allows a machine to correctly run a contouring 4- or 5-axis program when the workpiece is not located in the exact location specified by a CAM-generated program. This eliminates the need to repost a program from the CAM system when the programmed and the actual workpiece locations are different.

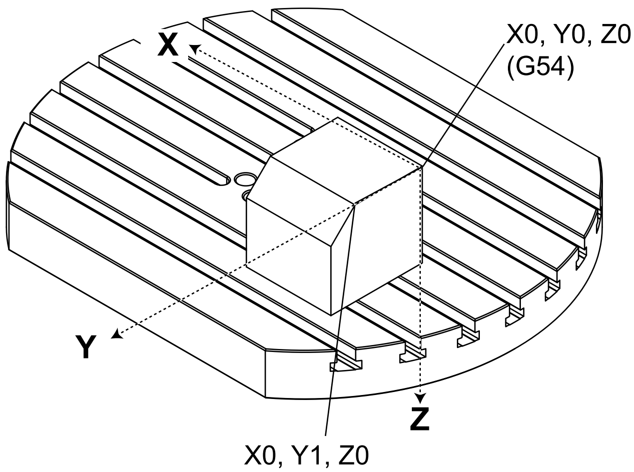
The Haas CNC control combines the known centers of rotation for the rotary table (MRZP) and the location of the workpiece (e.g., active work offset G54) into a coordinate system. TCPC makes sure that this coordinate system remains fixed relative to the table; when the rotary axes rotate, the linear coordinate system rotates with them. Like any other work setup, the workpiece must have a work offset applied to it. This tells the Haas CNC control where the workpiece is located on the machine table.

The conceptual example and illustrations in this section represent a line segment from a full 4- or 5-axis program.

NOTE: *For clarity, the illustrations in this section do not depict workholding. Also, as conceptual, representative drawings, they are not to scale and may not depict the exact axis motion described in the text.*

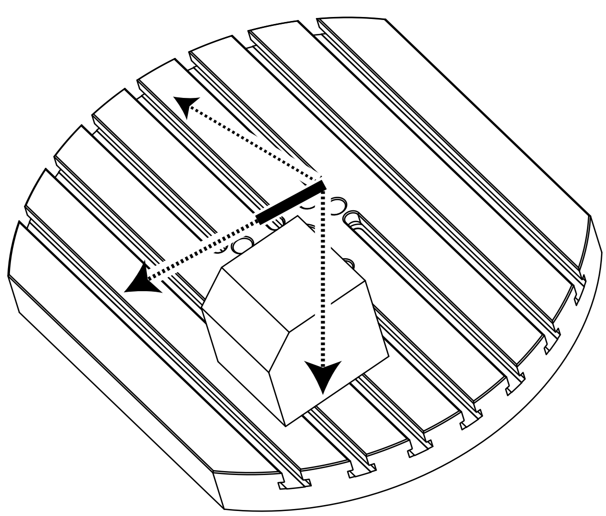
The straight line edge highlighted in Figure **F5.1** is defined by point (X0, Y0, Z0) and point (X0, Y1, Z0). Movement along the Y Axis is all that is required for the machine to create this edge. The location of the workpiece is defined by work offset G54.

F5.1: G234 and G54



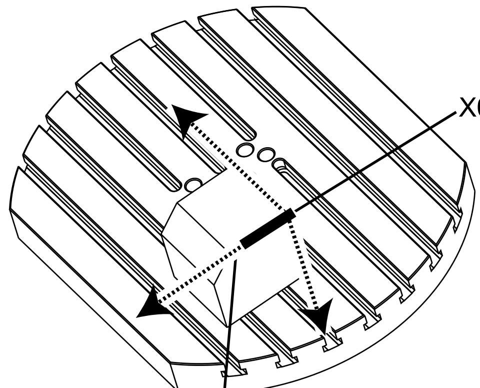
In Figure **F5.2**, the B and C Axes have been rotated 15 degrees each. To create the same edge, the machine will need to make an interpolated move with the X, Y, and Z Axes. Without TCPC, you would need to repost the CAM program for the machine to correctly create this edge.

F5.2: G234 with TCPC Off and the B and C Axes Rotated



TCPC is invoked in Figure **F5.3**. The Haas CNC control knows the centers of rotation for the rotary table (MRZP), and the location of the workpiece (active work offset G54). This data is used to produce the desired machine motion from the original CAM-generated program. The machine will follow an interpolated X-Y-Z path to create this edge, even though the program simply commands a single-axis move along the Y Axis.

F5.3: G234 with TCPC On and the B and C Axes Rotated



G234 Program Example

```
%
O00003 (TCPC SAMPLE)
G20
G00 G17 G40 G80 G90 G94 G98
G53 Z0.
T1 M06
G00 G90 G54 B47.137 C116.354 (POSITION ROTARY AXES)
G00 G90 X-0.9762 Y1.9704 S10000 M03 (POSITION LINEAR
AXES)
G234 H01 Z1.0907 (TCPC ON WITH LENGTH OFFSET 1, APPROACH
IN Z-AXIS)
G01 X-0.5688 Y1.1481 Z0.2391 F40.
X-0.4386 Y0.8854 Z-0.033
X-0.3085 Y0.6227 Z-0.3051
X-0.307 Y0.6189 Z-0.3009 B46.784 C116.382
X-0.3055 Y0.6152 Z-0.2966 B46.43 C116.411
X-0.304 Y0.6114 Z-0.2924 B46.076 C116.44
X-0.6202 Y0.5827 Z-0.5321 B63.846 C136.786
X-0.6194 Y0.5798 Z-0.5271 B63.504 C136.891
X-0.8807 Y0.8245 Z-0.3486
X-1.1421 Y1.0691 Z-0.1701
X-1.9601 Y1.8348 Z0.3884
```

```
G49 (TCPC OFF)
G00 G53 Z0.
G53 B0. C0.
G53 Y0.
M30
%
```

G234 Programmer’s Notes

These key presses and program codes will cancel G234:

- [**EMERGENCY STOP]**
- [**RESET]**
- [**HANDLE JOG]**
- [**LIST PROGRAM]**
- `M02` – Program End
- `M30` – Program End and Reset
- `G43` – Tool Length Compensation +
- `G44` – Tool Length Compensation -
- `G49` – G43 / G44 / G143 Cancel

These codes will NOT cancel G234:

- `M00` – Program Stop
- `M01` – Optional Stop

These key presses and program codes impact G234:

- G234 invokes TCPC and cancels G43.
- When using tool length compensation, either G43 or G234 must be active. G43 and G234 cannot be active at the same time.
- G234 cancels the previous H-code. An H-code must therefore be placed on the same block as G234.
- G234 cannot be used at the same time as G254 (DWO).

These codes ignore 234:

- `G28` – Return to Machine Zero Through Optional Reference Point
- `G29` – Move to Location Thru G29 Reference Point
- `G53` – Non-Modal Machine Coordinate Selection
- `M06` – Tool Change

G234 (TCPC) is intended for simultaneous 4- and 5-axis contouring programs. An active work offset (G54, G55, etc.) is required to use G234.

Chapter 6: G254 - Dynamic Work Offset (DWO)

6.1 Introduction

G254 Dynamic Work Offset (DWO) is similar to TCPC, except that it is designed for use with 3+1 or 3+2 positioning, not for simultaneous 4- or 5-axis machining. If the program does not make use of the B and C Axes, there is no need to use DWO.

CAUTION: *The B-Axis value of the work offset you use with G254 MUST be zero.*

With DWO, you no longer need to set the workpiece in the exact position as programmed in the CAM system. DWO applies the appropriate offsets to account for the differences between the programmed workpiece location and the actual workpiece location. This eliminates the need to repost a program from the CAM system when the programmed and actual workpiece locations are different.

The control knows the centers of rotation for the rotary table (MRZP) and the location of the workpiece (active work offset). This data is used to produce the desired machine motion from the original CAM-generated program. Therefore, it is recommended that G254 be invoked after the desired work offset is commanded, and after any rotational command to position the 4th and 5th axes.

After G254 is invoked, you must specify an X, Y, and Z Axis position before a cutting command, even if it recalls the current position. The program should specify the X and Y Axis position in one block and the Z Axis in a separate block.

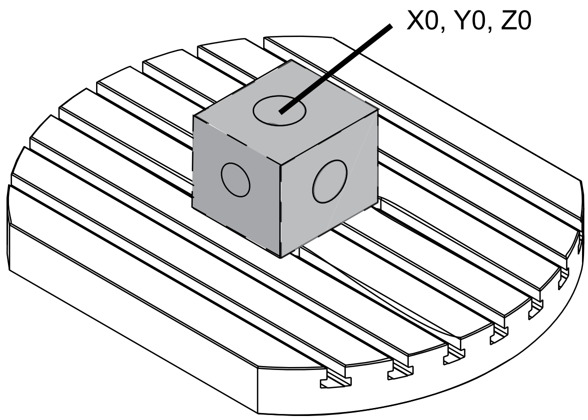
CAUTION: *Cancel G254 with G255 immediately after use and before ANY rotary motion. Be sure to cancel G254 with G255 when your program does simultaneous 4- or 5-axis machining.*

CAUTION: *DWO does not synchronize any motion. It is strongly recommended that you cancel G254 and retract the cutting tool to a safe location when the workpiece is repositioned.*

NOTE: *For clarity, the illustrations in this section do not depict workholding.*

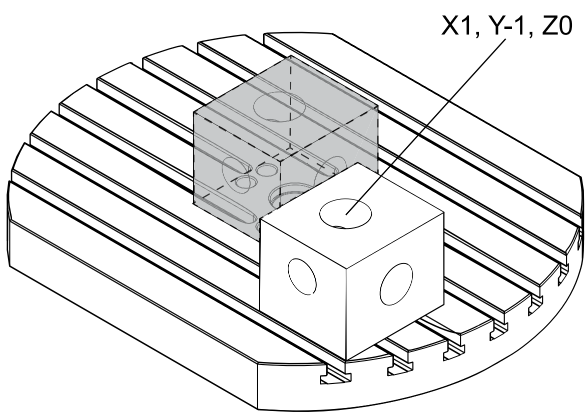
The block in Figure **F6.1** was programmed in the CAM system with the top center hole located at the center of the pallet and defined as X0, Y0, Z0.

F6.1: Original Programmed Position



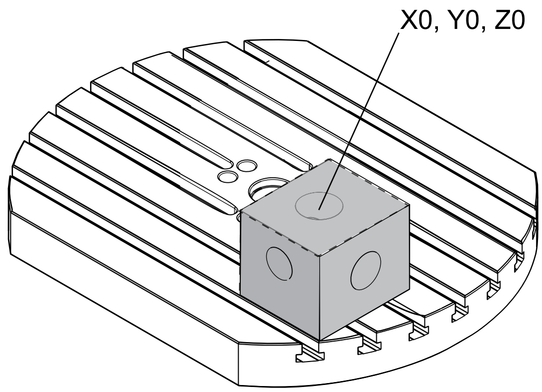
In Figure **F6.2**, the actual workpiece is not located in this programmed position. The center of the workpiece is actually located at X1, Y-1, Z0, and is defined as G54.

F6.2: Center at G54, DWO Off



DWO is invoked in Figure **F6.3**. The control knows the centers of rotation for the rotary table (MRZP), and the location of the workpiece (active work offset G54). The control uses this data to apply the appropriate offset adjustments to make sure that the proper toolpath is applied to the workpiece, as intended by the CAM-generated program. This eliminates the need to repost a program from the CAM system when the programmed and actual workpiece locations are different.

F6.3: Center with DWO On



G254 Program Example

```
%
O00004 (DWO SAMPLE)
G20
G00 G17 G40 G80 G90 G94 G98
G53 Z0.
T1 M06
G00 G90 G54 X0. Y0. B0. C0. (G54 is the active work
offset for the actual workpiece location)
S1000 M03
G43 H01 Z1. (Start feed 1.0 above face of part Z0.)
G01 Z-1.0 F20. (Feed into part 1.0)
G00 G53 Z0.
B90. C0. (ROTARY POSITIONING)
G254 (INVOKE DWO)
X1. Y0.
Z2. (Start feed 1.0 above face of part Z1.0)
G01 Z0. F20. (Feed into part 1.0 )
G00 G53 Z0.
G255 (CANCEL DWO)
B90. C-90. (ROTARY POSITIONING)
G254 (INVOKE DWO)
X1. Y0.
```

```
Z2. (Start feed 1.0 above face of part Z1.0)
G01 Z0. F20. (Feed into part 1.0 )
G255 (CANCEL DWO)
B0. C0.
M30
%
```

G254 Programmer's Notes

These key presses and program codes will cancel G254:

- [**EMERGENCY STOP]**
- [**RESET]**
- [**HANDLE JOG]**
- [**LIST PROGRAM]**
- G255 – Cancel DWO
- M02 – Program End
- M30 – Program End and Reset

These codes will NOT cancel G254:

- M00 – Program Stop
- M01 – Optional Stop

Some codes ignore G254. These codes will not apply rotational deltas:

- *G28 – Return to Machine Zero Through Optional Reference Point
- *G29 – Move to Location Thru G29 Reference Point
- G53 – Non-Modal Machine Coordinate Selection
- M06 – Tool Change

*It is strongly recommended that you not use G28 or G29 while G254 is active, nor when the B and C Axes are not at zero.

1. G254 (DWO) is intended for 3+1 and 3+2 machining, where the B and C Axes are used to position only.
2. An active work offset (G54, G55, etc.) must be applied before G254 is commanded.
3. All rotary motion must be complete before G254 is commanded.
4. After G254 is invoked, you must specify an X-, Y-, and Z-Axis position prior to any cutting command, even if it recalls the current position. It is recommended to specify the X and Y Axes in one block, and the Z Axis in a separate block.
5. Cancel G254 with G255 immediately after use and before ANY rotary motion.
6. Cancel G254 with G255 any time simultaneous 4- or 5-axis machining is performed.
7. Cancel G254 with G255 and retract the cutting tool to a safe location before the workpiece is repositioned.

Chapter 7: Setting Work and Tool Offsets

7.1 Set the B-Axis Work Offset

If the fixture or workpiece requires you to adjust the B Axis to achieve the proper alignment for machining, use this procedure to adjust and record the B-Axis work offset.

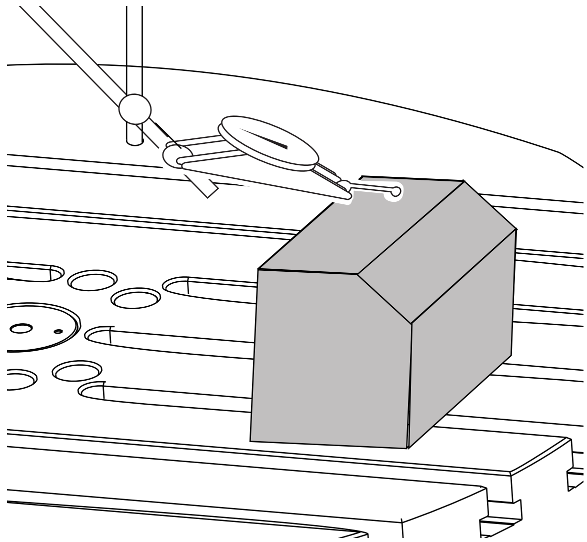
CAUTION: *Do not use a B-Axis offset if your program uses Dynamic Work Offsets (G254). The B-Axis offset value must be zero.*

1. Adjust the B Axis until the workpiece is positioned to the same orientation established in the program. Typically, the top surface of the fixture or workpiece will be perpendicular to the Z Axis.
2. Press **[OFFSET]** until the **`Work Zero Offset`** table appears. Press the cursor arrow keys to scroll to the work offset value used in the program (`G54` in this example).
3. Highlight the **`B AXIS`** column. Press **[PART ZERO SET]** to record the offset.

7.2 Set the C-Axis Work Offset

NOTE: *If the fixture or workpiece requires you to adjust the C Axis to achieve the proper alignment for machining, use the following procedure to adjust and record the C-Axis work offset.*

F7.1: Setting the C-Axis Workpiece Orientation



1. Place the workpiece on the platter (workholding not shown). Adjust the C Axis until the workpiece is positioned to the same orientation established in the program. Typically, a reference feature on the fixture or workpiece is parallel to the X or Y Axis.
2. Press **[OFFSET]** until the **`Work Zero Offset`** table appears. Press the cursor arrow keys to scroll to the work offset value used in the program (`G54` in this example).
3. Highlight the **`C AXIS`** column. Press **[PART ZERO SET]** to record the offset.

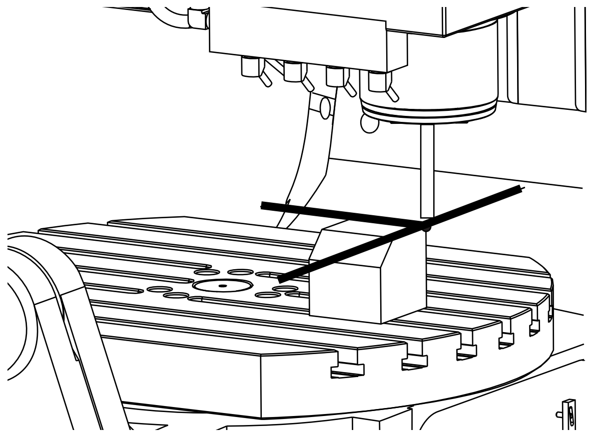
7.3 Set the X-, Y-, and Z-Axis Work Offsets Manually

NOTE: *Use this procedure if the WIPS probe is disabled.*

NOTE: *Refer to the Haas Mill Operator's Manual for basic offset and toolsetting methods.*

1. Jog the X and Y Axes to the zero position established in the program.

F7.2: UMC-750 X- and Y-Axis Zero Position



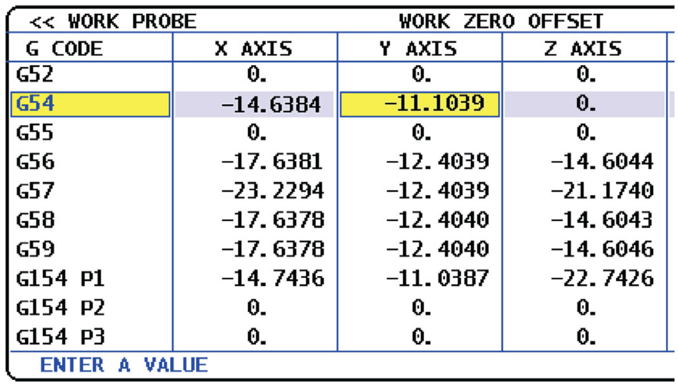
2. Press **[OFFSET]** until the **WORK ZERO OFFSET** table displays. Press the cursor arrow keys to scroll to the work offset value used in the program (G54 in this example).
3. Select the **X AXIS** column of your work coordinate offset and press **[PART ZERO SET]** to set the X-Axis zero position.

F7.3: X-Axis Zero Position Set

<< WORK PROBE		WORK ZERO OFFSET	
G CODE	X AXIS	Y AXIS	Z AXIS
G52	0.	0.	0.
G54	-14.6384	0.	0.
G55	0.	0.	0.
G56	-17.6381	-12.4039	-14.6044
G57	-23.2294	-12.4039	-21.1740
G58	-17.6378	-12.4040	-14.6043
G59	-17.6378	-12.4040	-14.6046
G154 P1	-14.7436	-11.0387	-22.7426
G154 P2	0.	0.	0.
G154 P3	0.	0.	0.
ENTER A VALUE			

4. Press **[PART ZERO SET]** again to set the Y-Axis zero position.

F7.4: Y-Axis Zero Position Set

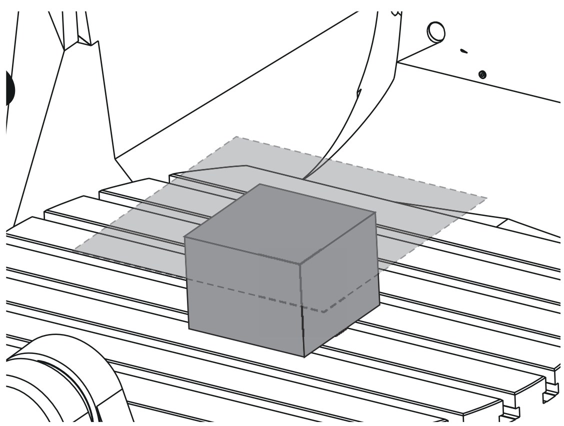


<< WORK PROBE		WORK ZERO OFFSET	
G CODE	X AXIS	Y AXIS	Z AXIS
G52	0.	0.	0.
G54	-14.6384	-11.1039	0.
G55	0.	0.	0.
G56	-17.6381	-12.4039	-14.6044
G57	-23.2294	-12.4039	-21.1740
G58	-17.6378	-12.4040	-14.6043
G59	-17.6378	-12.4040	-14.6046
G154 P1	-14.7436	-11.0387	-22.7426
G154 P2	0.	0.	0.
G154 P3	0.	0.	0.

ENTER A VALUE

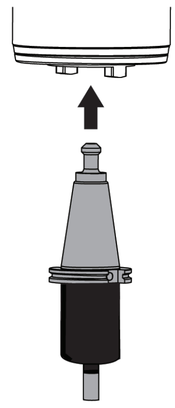
5. Determine a tool set plane to be used for setting all tool length offsets; for example, use the top surface of the workpiece.

F7.5: Example Tool Set Plane (Top of the Part)

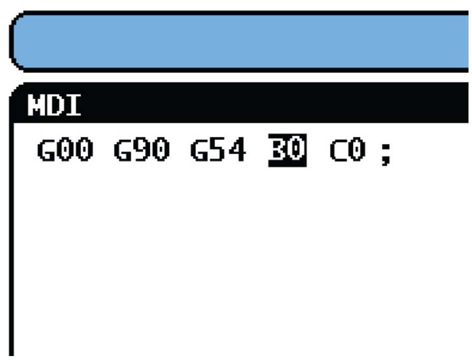


6. Load the master gage tool included with WIPS into the spindle.

F7.6: Master Gage Tool

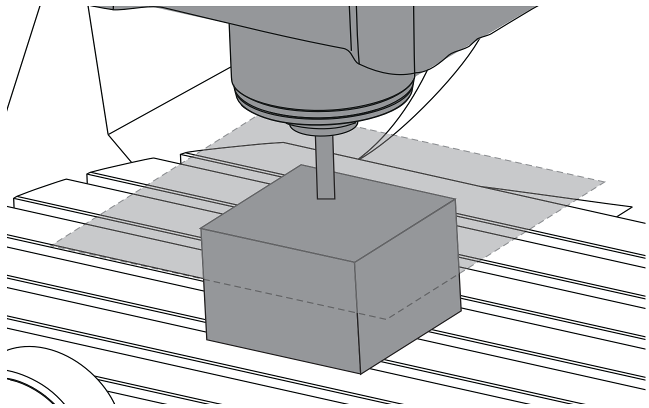


7. Make sure that the B and C Axes are at the same work zero point set earlier. (`G00 G90 G54 B0 C0`)



8. Select the **`Z AXIS`** column of your work coordinate offset.
9. Jog the Z Axis to the tool set plane. Make sure that the end of the gage tool you are using just touches the tool set plane. You will touch-off all of your tools on this surface.

F7.7: Jog the End of the Gage Tool to the Tool Set Plane



10. With the Z-Axis column of the work offset used in the program highlighted (`G54` in this example), press **[PART ZERO SET]**.
11. Subtract the length of the master gage tool supplied with the machine from the Z-Axis work offset. Enter this value as the offset.

    For example, if the Z-Axis work offset is -7.0000 and the master gage tool length is 5.0000, the new Z-Axis work offset is -12.0000.

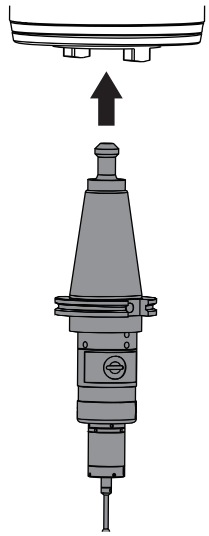
12. Touch-off each of the tools in your program to the Z set plane to establish their length offsets.

7.4 Set the X-, Y-, and Z-Axis Work Offsets with WIPS

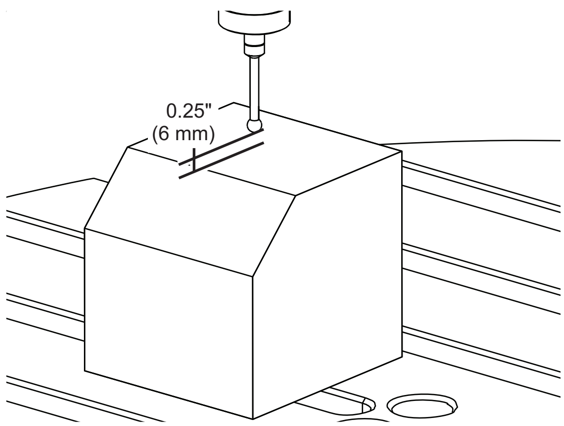
If you are not using the WIPS system, go to the Set the X-, Y-, and Z-Axis Work Offsets Manually section, starting on page **23**.

NOTE: *Make sure that the tool setting probe and the work probe are calibrated. Refer to the Haas WIPS manual (96-10002) for the calibration procedure.*

F7.8: UMC-750 Work Offset with WIPS



F7.9: UMC Z-Axis Work Zero Offset Start



1. Load the work probe into the spindle.
2. Make sure that the B and C Axes are at the same work zero point set earlier. (`G00 G90 G54 B0 C0`). Refer to the Set the B-Axis Work Offset and Set the C-Axis Work Offset sections if these values are not correct.
3. Set the X- and Y-Axis offsets using the standard WIPS templates as appropriate. Refer to the WIPS manual for more information.
4. Position the work probe tip approximately 0.25" (6 mm) above the Z-Axis zero surface.

5. In MDI mode, press **[OFFSET]** until the **WORK ZERO OFFSET** display is active. Select your work coordinate offset (G54, G55, etc.)
6. Press the **[RIGHT]** cursor arrow until you reach the **PROBE ACTION** sub-menu.
7. Type 11, and then press **[ENTER]** to assign the Single Surface probe action to the offset.
8. Press the **[RIGHT]** cursor arrow to access the **WORK PROBE INPUTS** table. Highlight the **Z distance** field.
9. Type -.5 (or -12 if the control is set to metric measurements), and then press **[ENTER]**.
10. Press **[CYCLE START]**. The probe measures the distance to the top of the part and records the value in the Z-Axis work offset.
11. Use the tool setting probe to set each of your tool length offsets.

Chapter 8: C-Axis Rotary Unwind and Setting 247

8.1 C-Axis Rotary Unwind

This feature allows you to return the C Axis to zero within 360 degrees, saving time and motion. The C Axis will need to have rotated at least 360 degrees for the unwind feature to be a benefit.

For example, if the C Axis has rotated a total of 960 degrees through the course of a program, a C-axis zero return command without the unwind feature will cause the axis to rotate back through all 960 degrees of rotation before the Haas CNC control considers the axis at home.

With C-Axis Rotary Unwind enabled, the C Axis rotates toward zero just enough to reach its home position; all previous revolutions are ignored. In the example of 960 degrees of rotation, the C Axis rotates a negative 240 degrees and stops at the machine home position.

To use this feature, Setting 108 must be set to **ON**, and Parameter 498:10 must be set to **1**. The unwind command must be an incremental (`G91`) Home command (`G28`).

For example:

```
G54 G01 F100. C960. (C AXIS TURNS 960 DEGREES CLOCKWISE)
G28 G91 C0. (C AXIS ROTATES 240 DEGREES
COUNTER-CLOCKWISE TO HOME)
```

8.2 247 - Simultaneous XYZ Motion in Tool Change

Setting 247 is a control feature that requires the Z Axis to move to the tool change position first, followed by the X and Y Axes. If Setting 247 is **OFF**, the Z Axis will retract first, followed by X- and Y-Axis motion. This feature can be useful in avoiding tool collisions for some fixture configurations. If Setting 247 is **ON**, the axes will move simultaneously. This may cause collisions between the tool and the workpiece, due to B- and C-Axis rotations. It is strongly recommended that this setting remain `OFF` on the UMC-750, due to the high potential for collisions.

Chapter 9: Other Information

9.1 Machine and Post-Processor Information

NOTE: *Refer to the machine layout drawing available at www.HaasCNC.com for UMC dimensions, weight, and anchoring patterns.*

NOTE: *Refer to the sections on `G234` Tool Center Point Control, `G254` Dynamic Work Offset, and C-Axis Rotary Unwind before you configure a post processor for this machine. If you have questions, e-mail them to applications@haascnc.com.*

Mill Type	5-Axis Vertical Mill
Model Number	UMC-750
Number of Simultaneous Axes	Five (X, Y, Z, B, C)
Axes Configuration	Table / Table with C Axis mounted on B Axis
Axis Travel Limit	X 30.00" (762 mm) Y 20.00" (508 mm) Z 20.00" (508 mm) B +110°/-35° C ± 13,320°

Control Specifications	
Programmed Unit	inch / mm
Data Transmission	DNC, FNC, USB, Hard Drive, NET SHARE (with Ethernet option)
Data Port	RS-232, USB, RJ45 (Ethernet option)

Program Format	G and M code
Circular Interpolation Code	`G02` (CW) and `G03` (CCW) I, J, K, or R. When I, J, and K are used to specify arc center, R cannot be used. 2D mode only, not supported during 5-axis motion. The I, J, K addresses are the distances from the starting point to the center of the arc. Only I, J, or K specific to the selected plane are allowed (`G17` uses IJ, `G18` uses IK and `G19` uses JK). The X, Y, and Z commands specify the end point of the arc. If the X, Y, or Z location for the selected plane is not specified, the endpoint of the arc is the same as the starting point for that axis. The R-value defines the distance from the starting point to the center of the circle. Use a positive R-value for radii of 180° or less, and a negative R-value for radii more than 180°.

Optional Through-Spindle Coolant	`M88` TSC on, `M89` TSC off.	
Flood coolant	`M08` Flood on, `M09` Flood off	
Program Identifier	`O12345`	Letter O followed by 5 digits 0-9
Program Structure	`%`	Beginning and end of the file
	`Onnnn`	Program number

General Notes	
Parameter 1306 Machine Rotary Zero Point X Offset	Stores the location of the B-Axis rotary center point in the X Axis
Parameter 1307 Machine Rotary Zero Point Y Offset	Stores the location of the C-Axis rotary center point in the Y Axis
Parameter 1308 Machine Rotary Zero Point Z Offset	Stores the location of the B-Axis rotary center point in the Z Axis
3D+ Cutter Compensation	Cutter compensation use of `G41`, `G42` is not supported in the 3D toolpaths. `G141` cutter compensation is supported for 3D and 5-axis. `G141` requires X, Y, Z, I, J, and K be output, and is only supported in `G90`. `G40` cancels `G141` (see below).

Rotary Axis Brakes	M10 Engage 4th-Axis Brake (B-axis) M11 Release 4th-Axis Brake (B-axis) M12 Engage 5th-Axis Brake (C-axis) M13 Release 5th-Axis Brake (C-Axis). The post needs to command M11 and M13 just before any simultaneous rotary and linear axis motion begins. This will unlock the brakes to the B and C Axes.The Post needs to command M10 and M12 to lock the B and C Axes brakes after any simultaneous rotary and linear axis motion. B/C brakes must be engaged when doing any non 5-axis motion.
Inverse Time Feed Mode	G93 A feedrate is required for each motion block. Maximum feed value is F45000.0000
Feed Per Minute Mode	G94 Maximum feedrate is 650.00 ipm (16.5 m/min)
Using a tight cut tolerance (or linearization tolerance) in the 3D and 5-axis CAM toolpaths will allow smooth flowing contours and more accurate parts. On all Haas machines, in G00 Rapid Motion Positioning mode, each axis specified moves at its maximum speed until that axis reaches its target position. Generally, rapid motion will not be in a straight line, and all axes will not necessarily complete their motions at the same time. The machine will wait until all motions are complete before starting the next command. To achieve linear positioning, program the positioning movement with a G01 and the maximum feedrate (650 ipm / 16.5 m/min).	

9.1.1 G141 3D+ Cutter Compensation (Group 07)

X - X-Axis command
Y - Y-Axis command
Z - Z-Axis command
A - A-Axis command (optional)
B - B-Axis command (optional)
D - Cutter Size Selection (modal)
I - X-Axis cutter compensation direction from program path
J - Y-Axis cutter compensation direction from program path
K - Z-Axis cutter compensation direction from program path
F - Feedrate

This feature performs three-dimensional cutter compensation.

The form is:

```
G141 Xnnn Ynnn Znnn Innn Jnnn Knnn Fnnn Dnnn
```

Subsequent lines can be:

```
G01 Xnnn Ynnn Znnn Innn Jnnn Knnn Fnnn ;
```

Or

```
G00 Xnnn Ynnn Znnn Innn Jnnn Knnn ;
```

Some CAM systems are able to output the `X`, `Y`, and `Z` with values for `I`, `J`, `K`. The `I`, `J`, and `K` values tell the control the direction in which to apply the compensation at the machine. Similar to other uses of `I`, `J`, and `K`, these are incremental distances from the `X`, `Y`, and `Z` point called.

The `I`, `J`, and `K` specify the normal direction, relative to the center of the tool, to the contact point of the tool in the CAM system. The `I`, `J`, and `K` vectors are required by the control to be able to shift the toolpath in the correct direction. The value of the compensation can be in a positive or negative direction.

The offset amount entered in radius or diameter (Setting 40) for the tool will compensate the path by this amount, even if the tool motions are 2 or 3 axes. Only `G00` and `G01` can use `G141`. A `Dnn` will have to be programmed; the D-code selects which tool wear diameter offset to use. A feedrate must be programmed on each line if in `G93` Inverse Time Feed mode.

With a unit vector, the length of the vector line must always equal 1. In the same way that a unit circle in mathematics is a circle with a radius of 1, a unit vector is a line that indicates a direction with a length of 1. Remember, the vector line does not tell the control how far to move the tool when a wear value is entered, just the direction in which to go.

Only the endpoint of the commanded block is compensated in the direction of I, J, and K. For this reason, this compensation is recommended only for surface toolpaths having a tight tolerance (small motion between blocks of code). G141 compensation does not prohibit the toolpath from crossing over itself when excessive cutter compensation is entered. The tool will be offset, in the direction of the vector line, by the combined values of the tool offset geometry plus the tool offset wear. If compensation values are in diameter mode (Setting 40), the move will be half the amount entered in these fields.

For best results, program from the tool center using a ball nose endmill.

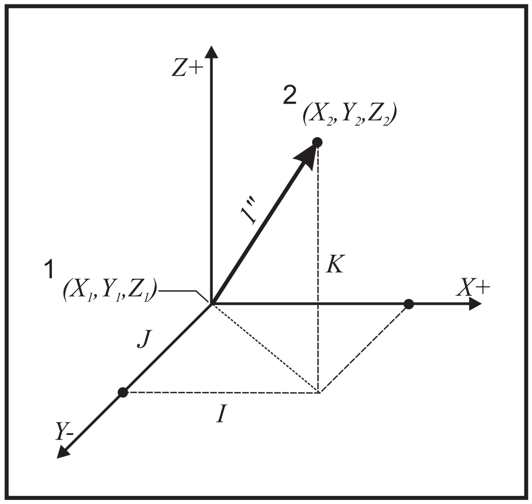
G141 Example:

```
N1 T1 M06 ;
N2 G00 G90 G54 X0 Y0 Z0 A0 B0 ;
N3 G141 D01 X0.Y0. Z0. (RAPID POSIT WITH 3 AX C COMP) ;
N4 G01 G93 X.01 Y.01 Z.01 I.1 J.2 K.9747 F300. (FEED INV
TIME) ;
N5 X.02 Y.03 Z.04 I.15 J.25 K.9566 F300. ;
N6 X.02 Y.055 Z.064 I.2 J.3 K.9327 F300. ;
... ;
N10 X2.345 Y.1234 Z-1.234 I.25 J.35 K.9028 F200. (LAST
MOTION) ;
N11 G94 F50. (CANCEL G93) ;
N12 G0 G90 G40 Z0 (Rapid to Zero, Cancel Cutter Comp) ;
N13 X0 Y0 ;
N14 M30 ;
```

In the above example, we can see where the I, J, and K were derived by plugging the points into the following formula:

AB = $[(x_2-x_1)^2 + (y_2-y_1)^2 + (z_2-z_1)^2]$, a 3D version of the distance formula. Looking at line N5, we will use 0.15 for x_2, 0.25 for y_2, and 0.9566 for Z_2. Because I, J, and K are incremental, we will use 0 for x_1, y_1, and z_1.

F9.1: Unit Vector Example: The commanded line endpoint [1] is compensated in the direction of the vector line [2](I,J,K), by the amount of the Tool Offset Wear.




```
AB=[(.15)² + (.25)² + (.9566)²]
AB=[.0225 + .0625 + .9151]
AB=1
AB=1
```

A simplified example is listed below:

```
N1 T1 M06 ;
N2 G00 G90 G54 X0 Y0 ;
N3 G43 H01 Z1. ;
N4 G141 D01 X0. Y0. Z0. (RAPID POSIT WITH 3 AX C COMP) ;
N5 G01 X10. Y0 I0. J-1. K0. F300. ;
N6 G40 Z1.0 (Rapid to Zero, Cancel Cutter Comp) ;
N7 M30 ;
```

In this case, if the wear value (DIA) for `T01` is set to -.02, then the tool will move from `X0. Y0. Z0.` (Line `N4`) to `X10. Y.01`. The `J` value told the control to compensate the endpoint of the programmed line only in the Y Axis.

Line `N5` could have been written using only the `J-1.` (not using `I0. K0.`), but a `Y` value must be entered if a compensation is to be made in this axis (`J` value used).

Index