



VMC QUICK START-UP GUIDE



The purpose of this quick-reference guide is to assist the operator in setting up and running a HAAS mill in as little time as possible. Refer to the Operators Manual for more programming and operating information. **IMPORTANT!!** Be sure ALL Installation procedures have been completed before operating the machine.

POWERING ON THE MACHINE

- Press the green **POWER ON** button in the upper left hand corner of the control.
- With the front doors closed, press the **POWER UP / RESTART** button. This will zero return all available axes.



GENERAL INFORMATION

HELP FUNCTION

This function is selected by pressing the **HELP/CALC** display button. This will bring a mini-manual up on the screen. The A..Z keys can then be used to find specific information on 26 topic areas.

- Pressing **HELP/CALC** key a second time will select the calculator function.

CREATING A PROGRAM

- Press the **LIST PROG** key. This will show a list of any programs presently in the control.



- Press the **O** (letter, not number) key, then nnnn, where nnnn can be any four numbers. Then press the **WRITE** key. This will create a program named Onnnn. Program names must follow this format.



The new program may now be created on the line following the program number. All further entries are made by typing a letter followed by a numeric value and pressing the **WRITE**, **INSERT** or **ALTER** key.

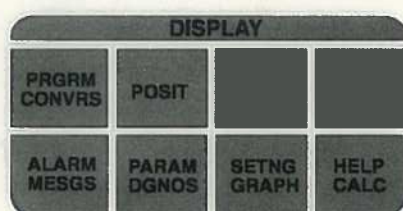


To make changes to an existing program, use the following keys:

- The **INSERT** key will insert any text to the right of the current cursor location.
- The **ALTER** key will replace highlighted information with the desired text.
- The **DELETE** key will delete any highlighted text.

LOCATING X AND Y ZERO POINTS OF THE PART

- Use an edge finder or indicator to locate the part zero point with the jog handle.
- Press the **OFSET** key and **PAGE UP** until the Work Coordinate page appears.



MANUAL DATA INPUT (MDI)

Another way of writing and editing programs, without using the **EDIT** mode is in **MDI** (Manual Data Input) mode. **MDI** is a scratch pad memory that can execute many lines of instruction without having to disturb your main program in memory.

- A program in **MDI** can be saved as a normal program by placing the cursor at the beginning of the first line, typing Onnnn (new program number), then pressing **ALTER**. This will add that program name to the program list and clear **MDI**.

EMERGENCY STOP

This button will instantly stop all motion of the machine including the servo motors, the spindle, the tool changer, and the coolant pump. It will also stop any auxiliary axes.

CYCLE START

This button will start a program running in **MEM** or **MDI** mode, continue motion after a **FEED HOLD**, or continue after a **SINGLE BLOCK** stop.

FEED HOLD

This button will stop all axis motion until the **CYCLE START** button is pressed. **FEED HOLD** will not stop the spindle, the tool changer, or the coolant pump. It also does not stop motion of any auxiliary axes.

CONTINUED

- Use the cursor arrows to get to G54 X or any of the other available work offsets.
- Push the **PART ZERO SET** key and the X-axis value will be stored as this offset.



- The cursor will automatically move to the G54 Y location. Repeat the steps above to set the G54 Y.
 - Usually the Z axis value will not have to be set and should be zero.
 - Repeat the steps to set the G54 A.

LOADING A TOOL INTO THE TOOL CAROUSEL

- Press the **MDI** button. Enter "T1", then press the **ATC FWD**, instead of **WRITE** key.



- Press the **TOOL RELEASE** button, then place tool #1 into the spindle.

SETTING THE TOOL OFFSETS

- Press the **OFFSET** key and **PAGE DOWN** to the Tool Offset page. Cursor to Tool #1.
- Use the jog handle to accurately position the tool tip to Z0. The top of your part will usually be Z0.
- Press the **TOOL OFFSET MESUR** key and the Z value will be stored in tool offset #1.
- Close the machine doors. Press the **NEXT TOOL** key and the Z-axis will retract to tool change position and tool #2 (empty) will be installed in the spindle.
- Put tool #2 into the spindle and jog to Z0 as you did for tool #1. The cursor will automatically be on Offset #2, so press the **TOOL OFFSET MESUR** key. Close the machine doors and press **NEXT TOOL**.
- To return to tool #1, press the **MDI** key, enter T1, and press **ATC FWD**.

RUNNING A PROGRAM

- Press the **LIST PROG** key. Use the cursor to find the desired program and then press the **SELECT PROG** key. The program list will include the program name and the first comment.
- Press the **CYCLE START** button to begin execution. To start a program other than at the beginning, scan to the desired starting place using the down arrow or **PAGE DOWN** key, and then press the **CYCLE START** button.

The **GRAPHICS** function is a visual dry run of your part program without moving the axes and risking tool damage from programming errors. To select **GRAPHICS** mode press **MEM** or **MDI** and the **SETG / GRAPH** key twice.

GENERAL INFORMATION

CURSOR KEYS

These keys give the user the ability to move to various screens and fields in the control. They are used extensively in CNC programs.

- The **HOME** key generally moves the cursor to the topmost item on the screen.
- The arrow keys move one item, block or field.
- The **PAGE UP** and **PAGE DOWN** keys are used to change displays up or down one page in the editor, or zoom in or out when in graphics.
- The **END** key generally moves the cursor to the bottom most item on the screen.

JOG KEYS

These keys give the user the ability to jog a certain axis in either direction.

- When a key is pressed briefly, that axis is selected for use by the jogging handle.
- When a key is pressed and held down, that axis is moved as long as the key is held down.
- If a "+" key is pressed and held, that axis is moved so that the tool position is changed in a positive direction relative to the work coordinates.
- If a "-" key is pressed and held, the axis is moved so that the tool position is changed in a negative direction relative to the work coordinates. The jog keys are locked out if the machine is running.

- The A axis key selects the B axis when used with the **SHIFT** key and the control is configured with a fifth-axis option.

- When the **JOG LOCK** key is pressed prior to one of the above keys, the axis is moved in a continuous motion without the need to hold the axis key depressed. Another press of the key stops jogging motion.

QUICK-TAB DIRECTORY

1. BASIC OPERATION

- G CODES
- M CODES

2. ALARMS

3. PARAMETERS

- SETTINGS

**4. MAINTENANCE &
INSTALLATION**

5. TECHNICAL REFERENCE

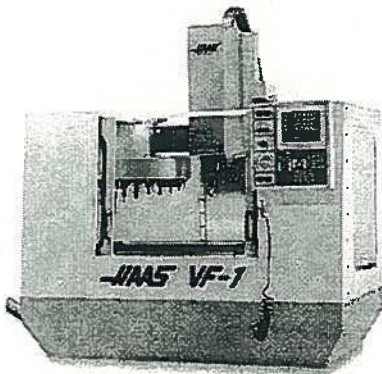
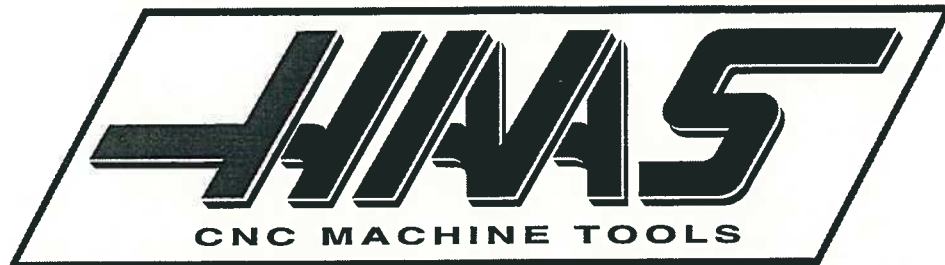
6. ASSEMBLY DRAWINGS

- WIRING DIAGRAMS

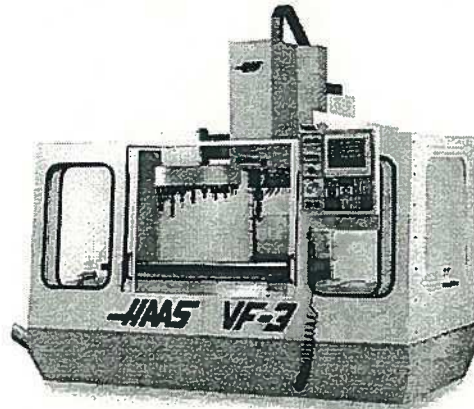
7. INSPECTION REPORT

8. INDEX

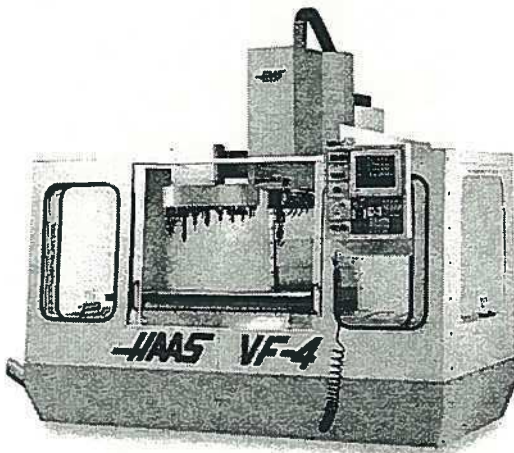
VF-SERIES OPERATORS MANUAL



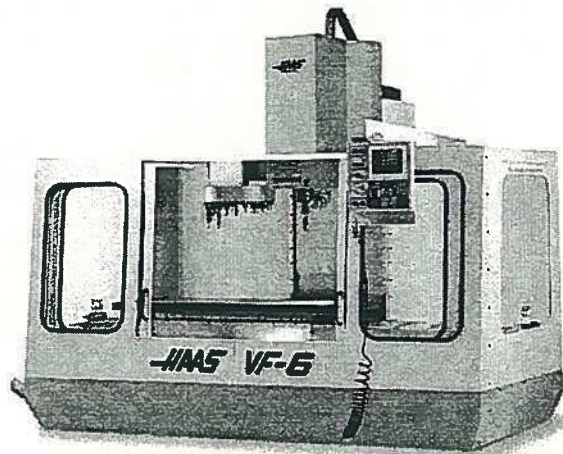
VF-1



VF-3



VF-4



VF-6

HAAS AUTOMATION, INC.

9601 LURLINE AVE.

CHATSWORTH CA 91331

TEL. 818-885-6050

FAX. 818-885-5281

805 278 1800
1-800-331-6746

JUNE 1996



VF-SERIES

OPERATORS MANUAL

WARRANTY REGISTRATION CERTIFICATE

The HAAS VF-Series Machining Centers are warranted by Haas Automation as follows:

All components are warranted against defects in material and workmanship for a period of one year from date of purchase, effective January 1, 1996:

An optional two year warranty can be purchased from your dealer.

EXCLUSIONS: Items subject to wear from machining parts, such as paint, windows, light bulbs, seals, etc. are excluded.

This warranty is void if the unit is subject to misuse, neglect, accident, improper installation, or improper application. Haas Automation is not liable for any additional or incidental damage to parts, fixtures, machines, or loss of time that may be caused by malfunctions.

Warranty service is available from your dealer. Units out of warranty will be repaired promptly and at reasonable cost.

"Date of purchase" is the date the machine is shipped to the original purchaser.

If you have a problem with your unit, a rereading of the manuals might solve the problem. If you still have a problem, a phone call to your dealer should solve the problem. **Please call your dealer first.** If you cannot get the problem taken care of with either of the above, a phone call to Haas maybe able to solve the problem.

Haas Automation, Inc.
9601 Lurline Ave.
Chatsworth, CA 91311
Phone: ~~(818) 885-6050~~ 805 278 1800
FAX: (818) 885-8372

In order to record the end user of this machine for updates and for product safety notices, we must have the machine registration returned immediately. Please fill out completely and mail to the above address to ATTENTION (VF-0, VF-1, etc. — whichever is applicable) REGISTRATIONS. **Please include a copy of your invoice** to validate your warranty date and to cover any additional options you may have purchased.

Company Name: STANFORD UNIVERSITY Contact Name: ALEXANDER COOPER
Address: Bldg 530, Rm. 128, STANFORD UNIVERSITY, STANFORD CA 94305-3030
Dealer: SELWAY MACHINE TOOL COMPANY, UNION CITY, CA
Date Purchased: 2/3/97
Model No.: VF-OE Serial Number: 9345
Telephone: (415) 723-0565 FAX: (415) 723-5034

IMPORTANT NOTICE!!! PLEASE READ IMMEDIATELY!!!

This machine is equipped with an electronically-recorded serial number that cannot be altered. This is done to protect you in case of theft and to track machines when sold to other owners. After approximately 800 hours of use, the machine will automatically shut down if it has not been unlocked by Haas Automation. To unlock the machine, we must have the above registration with the serial number and the authorization from your dealer. You will receive a number from Haas that you will write in over the serial number on setting page (#26). The authorization from the dealer will come upon final acceptance of the machine. If, for any reason, the serial number of the machine is erased in memory, the machine will revert back to 200 hour limit for your protection.

[The main body of the page is a large, empty rectangular frame, indicating that the table of contents text is either missing or has been redacted.]

CONTENTS

CHAPTER I

INTRODUCTION	1
1. THE COORDINATE SYSTEM	2
2. MACHINE HOME	4
3. ABSOLUTE AND INCREMENTAL POSITIONING	4
4. PROGRAMMING WITH CODES	5
5. MACHINE DEFAULTS	7
6. PROGRAM FORMAT	7
7. CANNED CYCLES	9
8. OPERATORS CONTROL PANEL	10
9. KEYBOARD	13
10. POWER ON/OFF	19
10.1 POWER ON	19
10.2 POWER OFF	20
11. MANUAL OPERATION	20
11.1 MDI	20
11.2 HANDLE/JOG	20
12. AUTOMATIC OPERATION	21
12.1 OPERATION MODE	21
12.2 PROGRAM SELECTION	22
12.3 STARTING AUTOMATIC OPERATION	22
12.4 PROGRAM RESTART	22
12.5 STOPPING AUTOMATIC OPERATION	23
12.6 EMERGENCY STOP SWITCH	23
12.7 WORKING LIGHTS	24
13. PART PROGRAM STORAGE AND EDIT	24
13.1 CREATING PROGRAMS	24
13.2 EDITING PROGRAMS	25
13.3 BACKGROUND EDIT	26
13.4 DELETING PROGRAMS	26
13.5 SPECIAL FUNCTION KEYS	27
13.6 THE UNDO KEY	27
13.7 BLOCK OPERATIONS	27
14. PART PROGRAM INPUT / OUTPUT	28
14.1 DATA INPUT / OUTPUT (RS-232)	28
14.2 DATA INPUT / OUTPUT (OPTIONAL RS-422)	31
14.3 DIRECT NUMERICAL CONTROL (DNC)	31
14.4 FLOPPY OPERATION	32
14.5 FLOPPY - DIRECT NUMERICAL CONTROL (DNC)	33

CHAPTER II

PROGRAMMING	35
1. PROGRAM STRUCTURE	35
2. ALPHABETICAL ADDRESS CODES	36
3. PREPARATORY FUNCTIONS (G CODES)	41
3.1 RAPID POSITION COMMANDS	43
3.2 INTERPOLATION COMMANDS	43
3.3 MISCELLANEOUS G CODES	45
3.4 PROGRAMMABLE OFFSET SETTING	45
3.5 CIRCULAR POCKET MILLING	46
3.6 CIRCULAR PLANE SELECTION	47
3.7 INCH / METRIC SELECTION (G20, G21)	48
3.8 REFERENCE POINT DEFINITION AND RETURN.....	49
3.9 SKIP FUNCTION (G31)	49
3.10 AUTOMATIC TOOL MEASUREMENT (G35, G37).....	49
3.11 AUTOMATIC WORK OFFSET MEASUREMENT (G36 , G136).....	50
3.12 CUTTER COMPENSATION	51
3.13 TOOL LENGTH COMPENSATION.....	51
3.14 ENGRAVING	51
3.15 COORDINATE ROTATION AND SCALING	53
3.16 WORK COORDINATE SYSTEM SELECTION	58
3.17 MORE MISCELLANEOUS G CODES	58
3.18 BOLT HOLE PATTERNS.....	59
3.19 CANNED CYCLES	60
3.20 ABSOLUTE / INCREMENTAL SELECTION	74
3.21 MORE WORK COORDINATE SELECTION.....	74
3.22 CANNED CYCLE AUXILIARY FUNCTIONS.....	74
3.23 PROGRAMMABLE MIRROR IMAGE (G100, G101).....	75
3.24 PROGRAMMABLE OUTPUT TO RS-232 (G102)	75
3.25 BLOCK LOOKAHEAD LIMIT FUNCTION (G103)	76
3.26 MORE WORK COORDINATE SELECTION.....	76
3.27 GENERAL PURPOSE POCKET MILLING FUNCTION.....	77
3.28 IN POSITION ACCURACY (G 187).....	78
4. MISCELLANEOUS FUNCTIONS (M FUNCTIONS).....	79
4.1 M CODE SUMMARY	79
4.2 M CODE DETAILED DESCRIPTION	80
5. SUBROUTINES	85
6. TOOL FUNCTIONS (TN)	86
7. SPINDLE SPEED FUNCTIONS	88
7.1 SPINDLE SPEED COMMANDS.....	88
7.2 RIGID TAPPING CONTROL OF SPINDLE	88
8. PROGRAMMING EXAMPLES	89
8.1 G81 DRILLING CANNED CYCLE.....	89
8.2 G82, G83, G84 CANNED CYCLES.....	90
8.3 SUBPROGRAMS AND CANNED CYCLES	91
8.4 SUBPROGRAMS WITH MULTIPLE FIXTURES	92
8.5 LOOPING CANNED CYCLES.....	93
8.6 MODIFYING CANNED CYCLES.....	94

8.7	SPECIAL CANNED CYCLES.....	95
8.8	CIRCULAR INTERPOLATION AND CUTTER COMPENSATION.....	97
8.9	PROGRAMMING USING I AND J.....	101
8.10	CIRCULAR POCKET MILLING.....	101
8.11	GENERAL PURPOSE POCKET MILLING.....	104
8.12	PROGRAMMABLE MIRROR IMAGE.....	108
8.13	THREAD MILLING.....	111
8.14	SINGLE-POINT THREAD MILLING.....	114
8.15	FOURTH AXIS PROGRAMMING.....	115
8.16	FORMULAS.....	117
9.	CUTTER COMPENSATION DESCRIPTION.....	118
9.1	GENERAL DESCRIPTION OF CUTTER COMPENSATION.....	118
9.2	ENTRY AND EXIT FROM CUTTER COMPENSATION.....	121
9.3	FEED ADJUSTMENTS IN CUTTER COMPENSATIONS.....	121
10.	QUICKCODE.....	124
10.1	INTRODUCTION.....	124
10.2	QUICKCODE TERMINOLOGY.....	125
10.3	USAGE AND FEATURES.....	126
10.4	CONVERSATIONAL QUICKCODE.....	127
10.5	A SAMPLE QUICKCODE SESSION.....	128
10.6	THE QUICKCODE SOURCE FILE.....	134
10.7	A SAMPLE QUICKCODE SOURCE FILE.....	137
11.	MACROS.....	138
11.1	INTRODUCTION.....	138
11.2	MACRO SUBROUTINE CALL (G65).....	139
11.3	ALIASING.....	140
11.4	MACRO ARGUMENTS.....	141
11.5	MACRO CONSTANTS.....	142
11.6	MACRO VARIABLES.....	142
11.7	SYSTEM VARIABLES IN-DEPTH.....	144
11.8	ADDRESS CONSTANT SUBSTITUTION.....	148
11.9	MACRO STATEMENTS.....	149
11.10	COMMUNICATION WITH EXTERNAL DEVICES - DPRNT[].....	156
11.11	RUNTIME EXECUTION.....	157
11.12	OPERATION NOTES.....	158
11.13	FANUC-STYLE MACRO FEATURES NOT INCLUDED IN HAAS CNC CONTROL.....	159

CHAPTER III

MACHINE OPERATION	161
1. WORK COORDINATE SYSTEM	161
2. SETUP PROCEDURES	162
3. DRY RUN OPERATION	162
4. DISPLAYS	163
4.1 PROGRAM DISPLAYS	165
4.2 POSITION DISPLAYS	166
4.3 TOOL OFFSETS DISPLAY	166
4.4 CURRENT COMMANDS DISPLAY	168
4.5 ALARMS / MESSAGES DISPLAY	169
4.6 PARAMETER / DIAGNOSTIC DISPLAY	170
4.7 SETTING / GRAPHIC DISPLAY FUNCTION	171
4.8 HELP / CALCULATOR FUNCTION	173
5. TRAVEL LIMITS	175
6. FEED/RAPID/SPINDLE OVERRIDES	175
7. WHAT TO DO WHEN ALARMS OCCUR	177
7.1 ALARMS	177
7.2 ALARM LIST	177
8. PARAMETERS	195
8.1 PARAMETER LIST	195
8.2 LEAD SCREW COMPENSATION	211
9. SETTINGS	212
10. SETTING UP A FOURTH AXIS	224
11. AUXILIARY AXIS CONTROL	225
12. FIFTH AXIS CONTROL	226
12.1 THE COORDINATE SYSTEM	226
12.2 ALPHABETICAL ADDRESS CODES	227
12.3 PREPARATORY FUNCTIONS (G CODES)	227
12.4 INCH/METRIC SELECTION (G20,G21)	227
12.5 WORK COORDINATE SYSTEMS	227
12.6 M CODE SUMMARY	228
12.7 HANDLE/JOG	228
12.8 SETTINGS	228
12.9 TOOL OFFSETS	229
12.10 PARAMETER DISPLAY	229
12.11 DIAGNOSTIC DATA DISPLAY	230
12.12 PARAMETERS	231
12.13 SETTING UP A FIFTH AXIS	231
13. AUTOMATIC ACCELERATION/DECELERATION	232
13.1 CONSTANT ACCELERATION	232
13.2 EXPONENTIAL ACCELERATION	232
13.3 ACCELERATION IN FEED MOTIONS	232
13.4 ACCELERATION IN RAPID MOVES	233
13.5 ACCELERATION/DECELERATION IN CIRCULAR MOVES	233
13.6 FANUC 6M, 10M, AND 15M COMPATIBILITY	233
14. HIGH SPEED MACHINING	234
14.1 CONTOURING ACCURACY	235

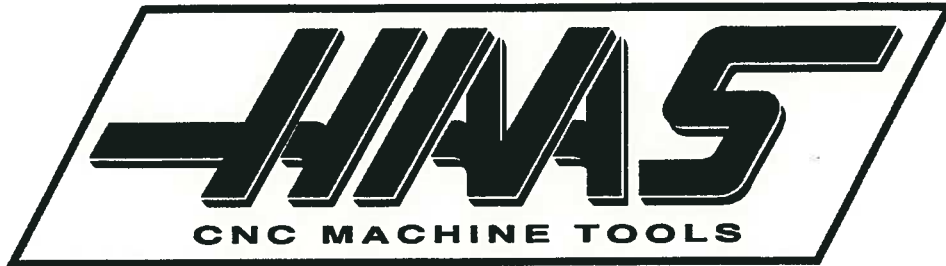
15. TAPPING WITH THE VF SERIES CNC MILL	236
15.1 RIGID TAPPING	236
15.2 USING FLOATING TAP HOLDERS	236
15.3 AUTOREVERSING TAPPING HEADS	237
15.4 THREAD MILLING	237
16. PROGRAMMABLE COOLANT SPIGOT	237
16.1 OPERATING THE COOLANT SPIGOT	237
16.2 OFFSET DISPLAY WITH SPIGOT ENABLED	237
16.3 SPIGOT PROGRAM COMMANDS	238
16.4 SPIGOT PARAMETERS	238
16.5 M CODE SUMMARY	238
16.6 M CODE DETAILED DESCRIPTION	238
17. AUTOMATIC CHIP CONVEYOR	239
17.1 CONVEYOR KEYBOARD COMMANDS	239
17.2 CONVEYOR PROGRAM COMMANDS	239
17.3 CONVEYOR PARAMETERS	239
17.4 M CODE SUMMARY	239
17.5 M CODE DETAILED DESCRIPTION	239
18. THROUGH THE SPINDLE COOLANT (TSC)	240
18.1 OPERATION	240
18.2 PROGRAM COMMANDS	240
18.3 TSC PARAMETERS	241
18.4 M CODE SUMMARY	241
18.5 M CODE DETAILED DESCRIPTION	241
18.6 ALARM DESCRIPTION	242
18.7 SAMPLE PROGRAM	242
 CHAPTER IV	
MAINTENANCE SCHEDULE	243
 CHAPTER V	
INSTALLATION MANUAL	245
 CHAPTER VI	
TECHNICAL REFERENCE	275
 CHAPTER VII	
ASSEMBLY DRAWINGS	307
 CHAPTER VIII	
ELECTRICAL DIAGRAMS	325
 CHAPTER IX	
INSPECTION REPORT	349
 INDEX	353



[Blank area for Table of Contents]

CHAPTER IX

MACHINE INSPECTION REPORT



	VF-0		VF-3
X	VF-0E		VF-4
	VF-E		VF-6
	VF-1		VF-8
	VF-2		

SERIAL NUMBER	9345
INSPECTOR	Danny
DATE	1/22/97

HAAS AUTOMATION, INC.
9601 LURLINE AVE.
CHATSWORTH, CA 91311
TEL: ~~(818) 885-6050~~ FAX: (818) 885-5281
(805) 278-1800

VMC INSPECTION REPORT

1

INSPECTOR	Danny White
DATE	1/22/97
SERIAL #	9345

VF-0

VF-1

VF-4

VF-0E

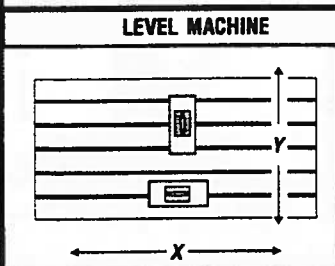
VF-2

VF-6

VF-E

VF-3

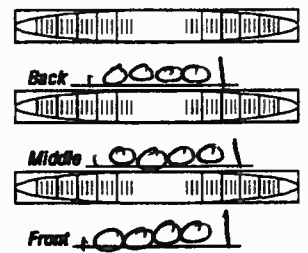
VF-8



1. FLATNESS OF TABLE TRAVEL MOVEMENTS

READ LEVEL AT RIGHT/CENTER, LEFT FOR BOTH X AND Y AXIS DIRECTIONS. RECORD LARGEST READING.

MEASURED VALUE: X-AXIS .0001
 (NTE 0.0005 / 10")
 Y-AXIS Draw in bubble positions.



2. PARALLELISM OF TABLE TO X-AXIS MOVEMENT

MOUNT INDICATOR TO SPINDLE AND MOVE X-AXIS OVER FULL RANGE.

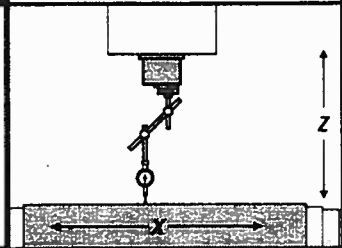
MEASURED VALUE: .0003
 (NTE 0.0005 / 10") VF-0/0E/E1/2/3/4

MEASURED VALUE: _____
 (NTE 0.0012 / 10") VF-6/8

FULL TRAVEL READING 0.001 MAX. .0005

FULL TRAVEL READING 0.0015 MAX. _____

MOVE X-AXIS OVER FULL RANGE



3. PARALLELISM OF TABLE TO Y-AXIS MOVEMENT

MOUNT INDICATOR TO SPINDLE AND MOVE Y-AXIS OVER FULL RANGE.

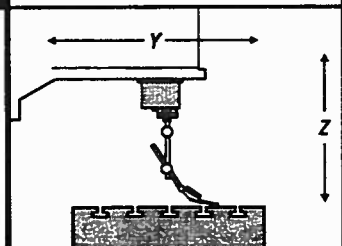
MEASURED VALUE: .0002
 (NTE 0.0005 / 10") VF-0/0E/E1/2/3/4

MEASURED VALUE: _____
 (NTE 0.0012 / 10") VF-6/8

FULL TRAVEL READING 0.001 MAX. .0004

FULL TRAVEL READING 0.0015 MAX. _____

MOVE Y-AXIS OVER FULL RANGE

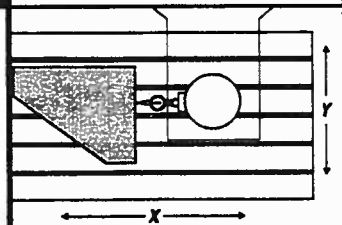


4. INSPECT SQUARENESS OF X-AXIS TO Y-AXIS

ALIGN PRECISION SQUARE WITH X-AXIS INDICATE ALONG Y-AXIS.

MEASURED VALUE: .0003
 (NTE 0.0005 / 10")

X-AXIS INDICATE ALONG Y-AXIS

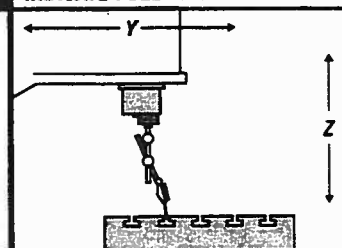


5. T-SLOT STRAIGHTNESS

PLACE INDICATOR IN T-SLOT OF MILL TABLE. INDICATE OVER FULL TRAVEL OF X-AXIS.

MEASURED VALUE: .0002
 (NTE 0.0005 / 10")
 FULL TRAVEL READING 0.0010 MAX.

INDICATE FULL TRAVEL OF X-AXIS



VMC INSPECTION REPORT

2

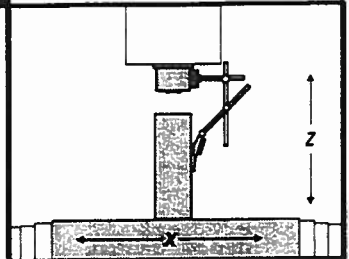
6.A SQUARENESS OF Z-AXIS MOTION TO X-AXIS

PLACE A CYLINDER SQUARE CENTERED ON TABLE. ADJUST CYLINDER TOP TO WITHIN 0.0000 FOR X-Y MOTION (NO MEASURABLE CHANGE IN X/Y MOTION). INDICATE THE SIDE OF THE CYLINDER SQUARE IN X DIRECTION. MOVE Z.

MEASURED VALUE: .0002
(NTE 0.0005 / 10")

Z-AXIS TILT RELATIVE TO X-AXIS

PARALLEL TO X-AXIS



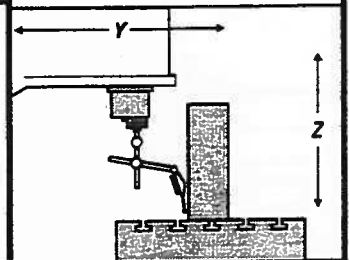
6.B SQUARENESS OF Z-AXIS MOTION TO Y-AXIS

INDICATE THE SIDE OF THE CYLINDER SQUARE IN Y DIRECTION. MOVE Z.

MEASURED VALUE: .0001
(NTE 0.0005 / 10")

Z-AXIS TILT RELATIVE TO Y-AXIS

PARALLEL TO Y-AXIS

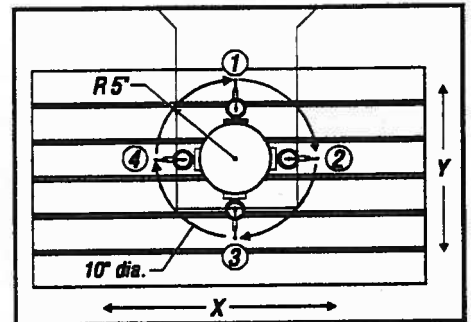


7. SPINDLE SWEEP

- (1) POSITION INDICATOR WITHIN 5" RADIUS.
- (2) SET ZERO AT A QUADRANT.
- (3) MEASURE EACH 3 REMAINING POSITIONS.

(X) MEASURED DIFFERENCE: .0002
(NTE 0.0005 / 10" dia.)

(Y) MEASURED DIFFERENCE: ∅
(NTE 0.0005 / 10" dia.)

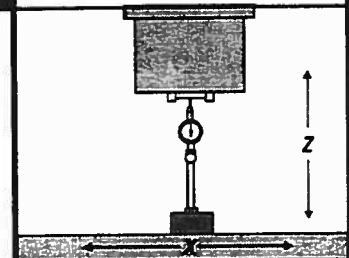


SPINDLE SWEEP X/Z PLANE

8.A SPINDLE RUNOUT AT TAPER

MEASURED VALUE: ∅
(RUNOUT OF TAPER 0.0002 TIR)

RUNOUT OF TAPER

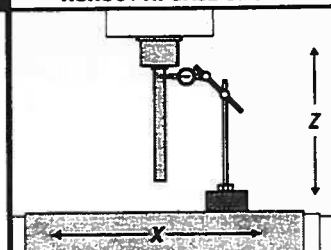


8.B SPINDLE RUNOUT ON TEST TOOL BAR

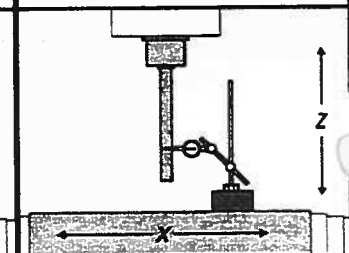
(1) MEASURED VALUE: ∅
(RUNOUT AT BASE OF BAR 0.0005 TIR)

(2) MEASURED VALUE: .0001
(RUNOUT AT 6" FROM BASE 0.001 TIR)

RUNOUT AT BASE OF BAR



RUNOUT AT 6 INCHES FROM BASE



DYNAMIC ISO 230-1

Circularity: 0.000268 in
Max: +0.000131 in, 113.6°
Min: -0.000136 in, 53.3°

9345XY.RTB

By: SASSAN HOOSHYAR

Machine: VF-0E

Date: 08:35 Jan 23 1997

Length: 5.905512 in
Radius: 5.905512 in
Best Radius: 5.905178 in
Centre Off X: -0.000031 in
Centre Off Y: +0.000001 in
Sample: 31.25 per sec
Feed: 50.0000 in/min

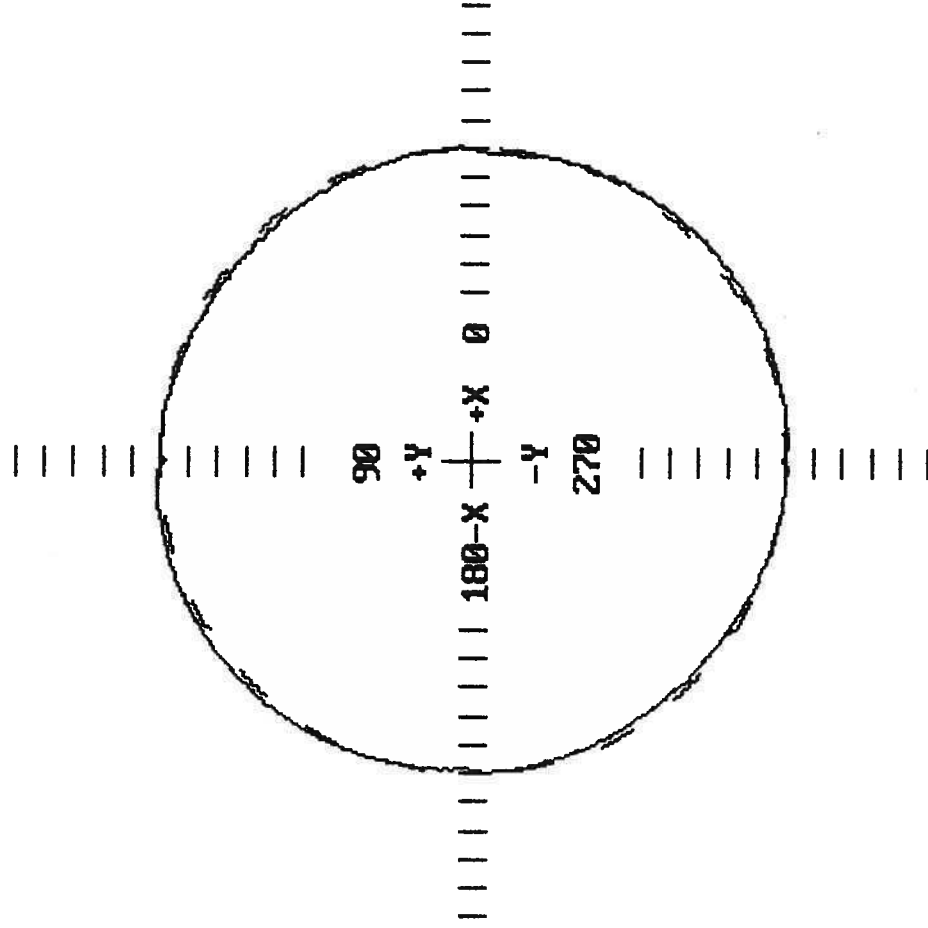
Start End

Machine 100.0° 100.0°

Data 0.0° 0.0°

1 run, Counter-clockwise

Renishaw Ballbar System



Scale: 0.000500 in/div

CHAPTER I

I. INTRODUCTION

This section provides the basic programming and operation principles necessary to begin operating the machine. The remainder of this manual is divided into more detailed **Programming** and **Operation** sections.

In an "NC" (Numerically Controlled) machine, the tool is controlled by a code system that enables it to be operated with minimal supervision and with a great deal of repeatability. "CNC" (Computerized Numerical Control) is the same type of operating system, with the exception that the machine tool is monitored by a computer.

The same principles used in operating a manual machine are used in programming an NC or CNC machine. The main difference is that instead of cranking handles to position a slide to a certain point, the dimension is stored in the memory of the machine control once. The control will then move the machine to these positions each time the program is run.

The operation of the VF-Series Vertical Machining Center requires that a part program be designed, written, and entered into the memory of the control. The most common way of writing part programs is off-line, that is, away from the CNC in a facility that can save the program and send it to the CNC control. The most common way of sending a part program to the CNC is via an RS-232 interface. The HAAS VF-Series Vertical Machining Center has an RS-232 interface that is compatible with most existing computers and CNC's.

In order to operate and program a CNC controlled machine, a basic understanding of machining practices and a working knowledge of math is necessary. It is also important to become familiar with the control console and the placement of the keys, switches, displays, etc., that are pertinent to the operation of the machine.

This manual can be used as both an operator's manual and as a programmer's manual. It is intended to give a **basic** understanding of CNC programming and it's applications. It is not intended as an in-depth study of all ranges of machine use, but as an overview of common and potential situations facing CNC programmers. Much more training and information is necessary before attempting to program on the machine.

The programming section of this manual is meant as a supplementary teaching aid to users of the HAAS Vertical Machining Center. The information in this section may apply in whole or in part to the operation of other CNC machines. Its use is intended only as an aid in the operation of the HAAS Vertical Machining Center.

1. THE COORDINATE SYSTEM

The first diagram we are concerned with is called a NUMBER LINE. This number line has a reference point zero that is called ABSOLUTE ZERO and may be placed at any point along the line.



Fig. 1-1 Horizontal number line.

The number line also has numbered increments on either side of absolute zero. Moving away from zero to the right are positive increments. Moving away from zero to the left are negative increments. The "+", or positive increments, are understood, therefore no sign is needed.



We use positive and negative along with the increment's value to indicate its relationship to zero on the line. In the case of the previous line, if we choose to move to the third increment on the minus (-) side of zero, we would call for -3. If we choose the second increment in the plus range, we would call for 2. Our concern is with distance and direction from zero.

Remember that zero may be placed at any point along the line, and that once placed, one side of zero has negative increments and the other side has positive increments.

The next illustration (Fig. 1-3) shows the three directions of travel on a vertical machining center. To carry the number line idea a little further, imagine such a line placed along each axis of the machine.

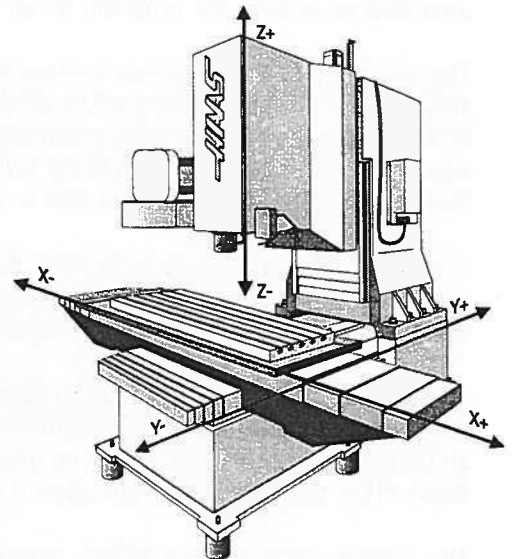


Fig. 1-2

Fig. 1-3 VF-1 showing X, Y, and Z axis lines.

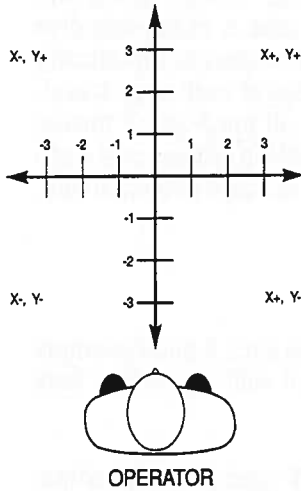
The first number line is easy to conceive as belonging to the left-to-right, or "X", axis of the machine. If we place a similar number line along the front-to-back, or "Y", axis, the increments toward the operator are the negative increments, and the increments on the other side of zero away from the operator are the positive increments.

The final axis of travel on our machine is the up-and-down, or "Z", axis. When we place a number line on the Z travel, the positive increments are up — above zero — and the negative values are down — below zero. Actually, the increments on each number line on the HAAS machining centers equals .0001 inches. Also, while a line theoretically travels infinitely in either direction once established, the three lines placed along the X, Y, and Z axes of the machine do not have unlimited accessibility. That is to say, we are limited by the range of travel on the machine.

VMC TRAVELS:

	X-axis	Y-axis	Z-axis
VF-0/ VF-1	20"	16"	20"
VF-2	30"	16"	20"
VF-3	40"	20"	25"
VF-4	50"	20"	25"
VF-6	64"	32"	30"
VF-8	64"	40"	30"

Remember, when we are moving the machine, we are concerned with positioning the spindle. Although the machine table is the moving part, we have to keep in mind our coordinates are based off our theoretical spindle movement.



Keep in mind that the zero position may be placed at any point along each of the three number lines, and in fact will probably be different for each setup of the machine. It is noteworthy to mention here that the Z-axis is usually set with the machine zero position in the full upward position, or the tool change position. This will place all the Z moves in a negative range of travel. However, the work zero in the Z-axis is usually set at the top of the part surface, and this will be entered in the tool length offset as a negative value. The range of travel on the HAAS VF-1, for example, is a total of 20 inches; four of these inches are above tool change position and are listed as a positive tool length offset, and 16 of these inches are below tool change position and are listed as a negative tool length offset.

The diagram shows a top view of the grid as it would appear on the machine tool. This view shows the X and Y axes as the operator faces the machine tool. Note that at the intersection of the two lines, a common zero point is established. The four areas to the sides and above and below the lines are called "QUADRANTS" and make up the basis for what is known as rectangular coordinate programming.

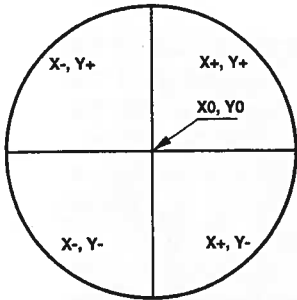
Fig. 1-4 View of X,Y grid from above.

THE TOP LEFT QUADRANT IS =	X- , Y+
THE BOTTOM LEFT QUADRANT IS =	X- , Y-
THE TOP RIGHT QUADRANT IS =	X+ , Y+
THE BOTTOM RIGHT QUADRANT IS=	X+ , Y-

Whenever we set a zero somewhere on the X-axis and somewhere on the Y-axis, we have automatically caused an intersection of the two lines. This intersection where the two zeros come together will automatically have the four quadrants to its sides, above, and below it. How much of a quadrant we will be able to access is determined by where we placed the zero within the travel of the machine axis.

For example, for a VF-1, if we set zero exactly in the middle of the travel of X and Y (table center), we have created four quadrants that are 10 inches by 8 inches in size.

2. MACHINE HOME



The principal may be seen when doing a manual reference return of all machine axes. When a zero return (ZERO RET) is performed at machine start up, all three axes are brought to the extreme positive direction until the limit switch is reached. When this condition is satisfied, the only way to move any of the three axes is in the negative direction. This is because a new zero was set for each of the three axes automatically when the machine was brought Home. This is placed at the edge of each axes travel. In effect, now the positive quadrants cannot be reached, and all the X and Y moves will be found to be in the X-, Y- quadrant. It is only by setting a new part zero somewhere within the travel of each axis that other quadrants are able to be reached.

Fig. 1-5 All four quadrants will have to be accessed to machine this part.

Sometimes it is useful in the machining of a part to utilize more than one of the X, Y quadrants. A good example of this is a round part that has its datum lines running through the center. The setup of such a part may look like Fig. 1-5.

These are just some examples of how to make use of the four quadrants of the X and Y axes on the machine. As more experience is gained in the machine tool programming and setup techniques, each programmer and setup person develops their own methods and style. Some methods will be faster than others, but each individual will have to determine the needs of each job in question, and reflect back on notes and the previous jobs completed.

3. ABSOLUTE AND INCREMENTAL POSITIONING

Up to this point, we have dealt with a system of positioning the tool that is known as absolute programming. In absolute, all coordinate points are given with regard to their relationship to the origin, a fixed zero point, or considered as part zero. This is the most common type of positioning.

Another type of positioning is called incremental positioning. Incremental positioning concerns itself with distance and direction. A new coordinate is entered in terms of its relationship to the previous position, and not from a fixed zero or origin. In other words, after a block of information has been executed, the position that the tool is now at is the new zero point for the next move to be made.

An example of the use of the incremental system is below. Note that to move from X 4.25 to X 2.025 on the scale, an incremental move of X -2.225 was made, even though the move still places the tool on the plus side of the scale. Therefore the move was determined from the last point, with no regard for the zero position. The + and - signs are used in terms of direction, and not in regard to the position of zero.

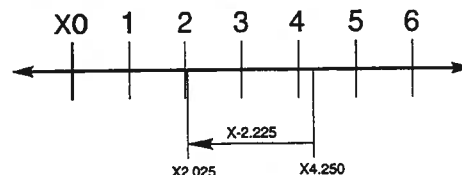


Fig. 1-6 An example of an incremental move.

Keep in mind that when positioning in **absolute**, we are concerned with distance and direction from a fixed zero reference point, and when positioning in **incremental** we are concerned with distance and direction from the last position.

4. PROGRAMMING WITH CODES

A program is written as a set of instructions given in the order they are to be performed. The instructions, if given in English, might look like this:

LINE #1 =	SELECT CUTTING TOOL.
LINE #2 =	TURN THE SPINDLE ON AND SELECT THE RPM.
LINE #3 =	TURN THE COOLANT ON.
LINE #4 =	RAPID TO THE STARTING POSITION OF THE PART.
LINE #5 =	CHOOSE THE PROPER FEED RATE AND MAKE THE CUT(S).
LINE #6 =	TURN OFF THE SPINDLE AND THE COOLANT.
LINE #7 =	RETURN TOOL TO HOLDING POSITION AND SELECT NEXT TOOL.

and so on. But our machine control understands only these messages when given in machine code.

Before considering the meaning and the use of codes, it is helpful to lay down a few guidelines:

- 1) Codes come in groups. Each group of codes will have a specific group number.
- 2) A **G** code from the same group can be replaced by another code in the same group. By doing this the programmer establishes modes of operation. The universal rule here is that codes from the same group cannot be used more than once on the same line.
- 3) There are modal **G** codes which, once established, remain effective until replaced with another code from the same group.
- 4) There are non-modal **G** codes which, once called, are effective only in the calling block, and are immediately forgotten by the control.

The rules above govern the use of all codes for programming the Haas (and other) controls. The concept of grouping codes and rules that apply will have to be remembered if we are to effectively program the machine tool. The following is a discussion of the codes most basic to the operation of the machine.

G CODES:

- G00 Rapid traverse motion; Used for positioning and during non-cutting moves.
NOTE: Machine rapids at 500 inches per minute (IPM).
- G01 Linear interpolation motion; Used for actual machining and metal removal. Governed by programmed feed rate.
- G02 Circular interpolation - Clockwise
- G03 Circular interpolation - Counterclockwise
- G28 Machine home (Rapid traverse)
- G40 Cutter compensation cancel
- G41 Cutter compensation to **left** of path
- G42 Cutter compensation to **right** of path

Programming with Codes

- G43 Read tool length compensation
- G54 Work coordinate #1 (Part zero)
- G80 Canned cycle cancel
- G81 Drill canned cycle
- G82 Spot drill canned cycle
- G83 Peck drill canned cycle
- G84 Tapping canned cycle
- G90 Absolute programming
- G91 Incremental programming
- G98 Initial point return
- G99 Reference plane return

M CODES:

- M00 Program stop. Press CYCLE START button to continue.
- M01 Optional program stop. Press optional stop key on control panel on M01 code.
- M02 End of program. Cannot continue.
- M03 Start spindle forward (Clockwise). Must be accompanied by a spindle speed.
- M04 Start spindle reverse (Counterclockwise). Must have a spindle speed.
- M05 Spindle stop
- M06 Tool change command. Must have a tool number in the same line. This command will automatically stop the spindle.
- M08 Coolant **ON** command
- M09 Coolant **OFF** command
- M30 Program end and rewind to beginning of program
- M97 Local subroutine call
- M98 Subprogram call
- M99 Subprogram return, or loop

NOTE: Only one "M" code can be used per line. The "M" code will be the last item of code to be performed, regardless of where it is located in the line.

5. MACHINE DEFAULTS

A **default** is an automatic function of the machine tool control. When powering up the machine, the control looks for the home position of all axes, then will read the default values or the preset "G" codes. If you have ever wondered why the machine went to the part zero that was entered in the G54 column when it was never specified in the actual program, it is because the machine automatically reads the G54 column upon start-up. That is a **default**.

The defaults for the Haas mill are indicated by an asterisk (*) in the "Preparatory Functions (G Codes)" section of this manual.

The control automatically reads these G codes when power is turned on:

G00	Rapid traverse
G17	X,Y Circular plane selection
G40	Cutter Compensation cancel
G49	Tool length compensation cancel
G54	Work coordinate zero #1 (1 of 26 available)
G64	Exact stop cancel
G80	Canned cycle cancel
G90	Absolute programming
G98	Initial point return

There is no default FEED RATE (F code), but once an F code is programmed, it will apply until another is entered or the machine is turned off.

6. PROGRAM FORMAT

Program format, or style, is an important part of CNC machining. Each individual will format their programs differently and, in most cases, a programmer could not identify a program written by themselves. The point is that a programmer needs to be consistent and efficient, writing code in the way it is listed and in the order it appears in the program. For example:

X, Y, Z is in order of appearance. The machine will read **X, Y, or Z** in any order, but we want to be consistent. Write **X** first, **Y** second, **Z** third.

The first line or block in a program using active G codes should be a tool number and tool change command. This would be a good safety measure.

The second line or block will contain a rapid command (G00), an absolute or incremental command (G90, G91), a work zero for **X** and **Y** (G54), a positioning **X** and **Y** coordinate, a spindle speed command (S____), and a spindle ON clockwise command (M03).

The third line or block will contain a "Read tool length compensation" command (G43), a tool length offset number (H01), a Z-axis positioning move (Z.1), and an optional coolant ON command (M08).

An example program's first three lines will look like this:

```
T1 M06;  
G00 G90 G54 X0 Y0 S2500 M03;  
G43 H01 Z.1 M08;
```


INTRODUCTION

Program Format

All the necessary codes for each operation are listed above. This format is a good practice and will separate your style from other programmers.

QUESTION:

If G00, G90, and G54 are defaults, why do we list them in the second line of a program and for each different tool?

ANSWER:

G00, G90, and G54 are listed for an operator/setup person's aid so he/she can determine if the machine will rapid position, if the machine is in fact in the absolute coordinate mode, and most important, the work zero. The work zero is always different between setups, and multiple work zeros are very common.

QUESTION:

Can we combine the second and third lines, excluding the M08 code? If so, why do we write the lines separate?

ANSWER:

Yes. The four G codes G00, G90, G54, and G43 all belong to different groups. Remember, no two G codes of the same group can be listed on the same line.

The main reason for using two lines is SAFETY. Remember, only one line of information can be executed at a time. The X and Y coordinates will position first, then the tool length and the Z coordinate will execute. If combined, all three axes will move simultaneously, and any interfering clamps or fixtures can be struck and/or destroyed. When combining X, Y, and Z in positioning, chances of crashing the machine are greater.

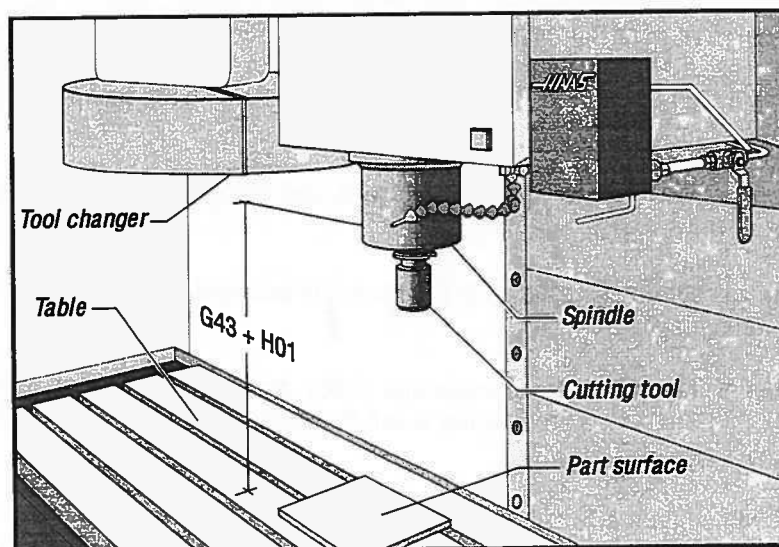


Fig. 1-7 Tool length offset and tool length compensation.

Tool number should always remain numerically matched with the tool length offset number. Setting 15 (the H & T agreement) will ensure the tool number and tool length offset will match. (Ex. T1 in line #1 should have H01 in line #3, and T2 should have H02 in line #3.)

7. CANNED CYCLES

A canned cycle is used to simplify programming of a part. Canned cycles are defined for most common Z-axis repetitive operations such as drilling, tapping, and boring. Once selected, a canned cycle is active until canceled with the G80 code. There are six operations involved in every canned cycle:

- 1) Positioning of X and Y axes (optional A, rotary axis).
- 2) Rapid traverse to the reference plane.
- 3) Drilling, boring, or tapping action.
- 4) Operation at the bottom of the hole.
- 5) Retraction to the reference plane.
- 6) Rapid traverse to the initial starting point.

A canned cycle is presently limited to operations in the Z-axis; that is, only the G17 plane is allowed. This means the canned cycle will be executed in the Z-axis whenever a new position is selected in the X or Y axis. The operation of a canned cycle will vary according to whether incremental (G91) or absolute (G90) is active. Incremental motion in a canned cycle is often useful as a loop count (L) and can be used to repeat the operation with an incremental X or Y move between each cycle. G98 and G99 are modal commands which change the way the canned cycles operate. When G98 (the system default) is active, the Z-axis will be returned to the starting position at the completion of the canned cycle. When G99 is active, the Z-axis will be returned to the reference plane when the canned cycle is completed.

NOTE: If an L0 is in the canned cycle line, the cycle will not execute until the control reads an X or Y location.

For more detailed information on canned cycles, refer to the "G Codes" section of this manual.

8. OPERATORS CONTROL PANEL

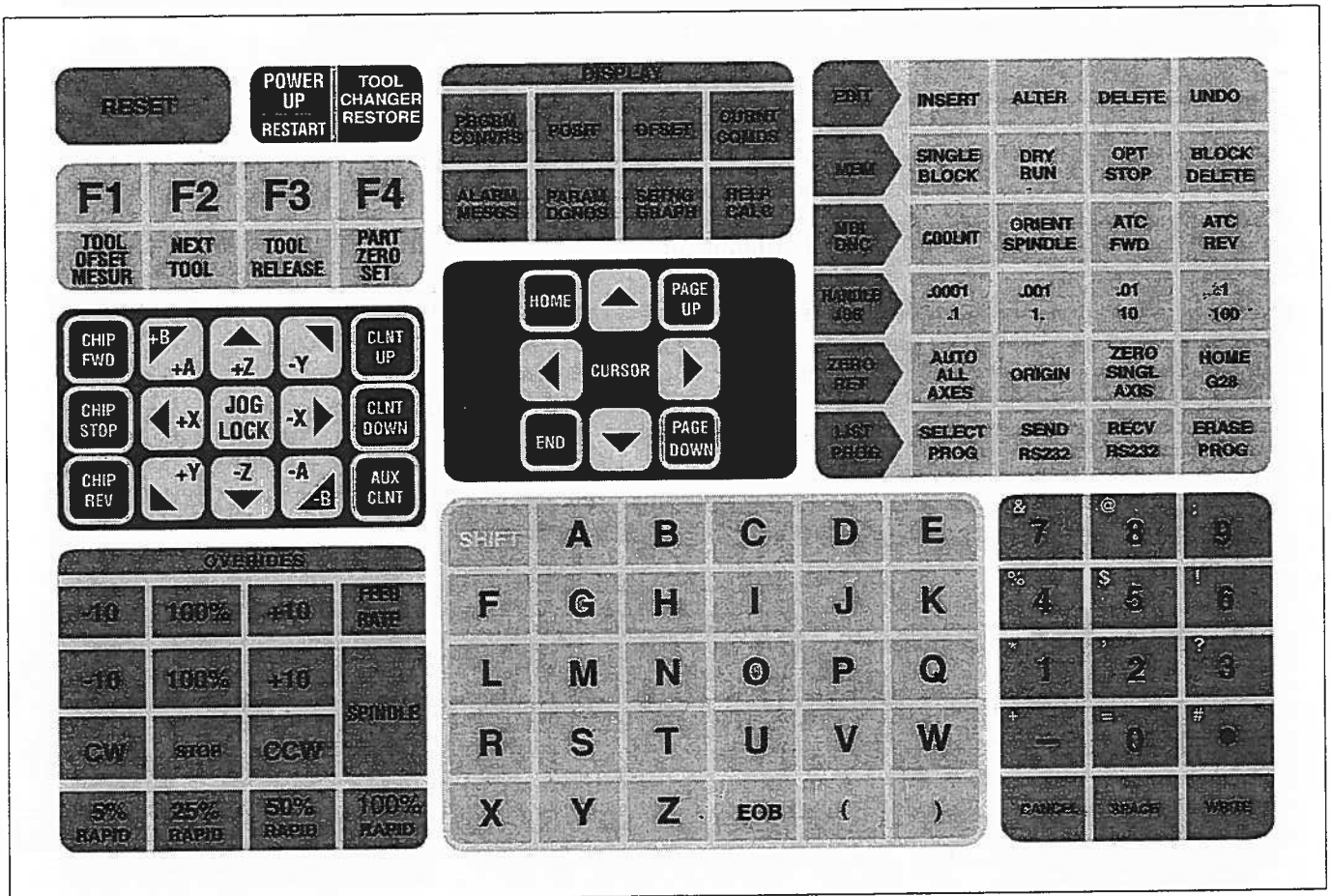


Figure 1-8 Control panel keypad with operating and display keys highlighted.

In operation, it is important to be aware of the operating mode selected for the CNC. There are six operating modes and one simulation mode in this control. The operating mode is selected with the six buttons labeled:

- EDIT To edit a program already in memory
- MEM To run a program stored in memory
- MDI / DNC To directly run manually entered program or to select DNC mode
- HANDLE JOG To use jog keys or jog handle
- ZERO RET To establish machine zero
- LIST PROG To list, send, or receive programs

The Graphics simulation mode is entered with the DISPLAY select buttons.

In MEM or MDI mode, a program can be started with the CYCLE START button. While a program is running, you cannot change to another mode; you must wait until it finishes or press RESET to stop the program.

When already in MDI, a second push of the MDI button will select DNC if the DNC mode is enabled by settings and parameters in your machine.

In any of the above modes, you can select any of the following displays using the six DISPLAY buttons:

PRGRM / CNVRS	To show the program selected
POSIT	To show the axes positions
OFSET	To show or enter working Offsets
CURNT COMDS	To show Current Commands and times
ALARM / MESGS	To show Alarms and user messages
PARAM / DGNOS	To show Parameters or Diagnostic data
SETNG / GRAPH	To show or enter Settings or to select Graphics simulation mode
HELP / CALC	To show the Help data and calculator

In addition to the above displays, when a program is already running, you may press LIST PROG to select a list of the programs in memory. This is useful to determine what programs can be edited in BACKGROUND EDIT. BACKGROUND EDIT is selected from the PROGRAM DISPLAY.

All operation of the CNC is controlled from the operator's panel. The control panel is composed of the CRT display, the keypad, On/Off switch, Load meter, Handle, EMERGENCY STOP, CYCLE START, and FEED HOLD buttons.

The **keypad** is a flat membrane type that requires approximately eight (8) ounces of pressure. The **SHIFT** button replaces the function of the numeric buttons with the white characters in the upper left corner. The **SHIFT** button must be pressed once before each shifted character. Pressing the **SHIFT** button twice will turn off shift.

The **load meter** measures the power to the spindle motor. At 100%, the spindle motor can be operated continuously. The 150% level can be sustained for no more than five (5) minutes. At 180%, the spindle may begin to slow and even stall. A 180% load should be reduced to 150% by reducing spindle speed or decreasing the feed rate.

The **handle** is used to jog one of the axis. Each step of the crank can be 0.0001, 0.001, 0.01 or 0.1 inch. The rotary axis is 0.001, 0.01, 0.1, or 1.0 degree per step. The handle has 100 steps per rotation. When using metric, the smallest handle step is 0.001 mm and the largest is 1.0 mm. As an option, the handle can also be used to move the screen cursor while in EDIT mode. An optional remote jog handle is also available.

The EMERGENCY STOP button will instantly stop all motion of the machine including the servo motors, the spindle, the tool changer, and the coolant pump. It will also stop any auxiliary axes. When Through the Spindle Coolant (TSC) is ON, this button performs a slightly different function, as described in the TSC section of this manual.

CYCLE START will start a program running in MEM or MDI mode, continue motion after a FEED HOLD, or continue after a SINGLE BLOCK stop.

FEED HOLD will stop all axis motion until the CYCLE START is pressed. FEED HOLD will not stop the spindle, the tool changer, or the coolant pump. It will not stop motion of any auxiliary axes.

The SINGLE BLOCK button on the keypad will turn on and off the SINGLE BLOCK condition. When in SINGLE BLOCK, the control will operate one block and stop. Every press of the START button will then operate one more block.

The RESET button on the keypad will always stop motion of the servos, the spindle, the coolant pump, and tool changer. It will also stop the operation of a running program. This is not, however, a recommended method to stop the machine as it may be difficult to continue from that point. SINGLE BLOCK and FEED HOLD provide for continuation of the program. RESET will not stop motion of any auxiliary axes but they will stop at the end of any motion in progress. When Through the Spindle Coolant is ON, this button performs a slightly different function, as described in the TSC section of this manual.

The CRT is the only display or readout device in the control. All status and position data is shown on the CRT.

The F1, F2, F3, and F4 buttons perform different functions depending on what display and mode is selected. The following is a quick summary of the **Fn** buttons:

F1 In EDIT mode and PROGRAM DISPLAY, this will start a block definition.

In LIST PROG mode, F1 will duplicate a program already stored and give it a new name from the command line.

In offsets display, F1 will set the entered value into the offsets.

F2 In EDIT mode, PROGRAM DISPLAY, this will end a block definition.

F3 In EDIT and MDI modes, the F3 key will copy the highlighted circular help line into the data entry line at the bottom of the screen. This is useful when you want to use the solution developed for a circular motion. Push INSERT to add that circular motion command line to your program.

In the calculator Help function, this button copies the value in the calculator window to the highlighted data entry for Trig, Circular, or Milling Help.

F4 In MEM mode and PROGRAM DISPLAY, this will select either BACKGROUND EDIT or PROGRAM REVIEW. BACKGROUND EDIT is selected by entering **Onnnn** with the program number to edit. Program review is selected with just F4. Program review shows the running program on the left half screen and allows the operator to review the program on the right half screen.

In the Calculator Help function, this button uses the highlighted Trig, Circular, or Milling data value to load, add, subtract, multiply, or divide with the calculator.

9. KEYBOARD

The control panel keyboard consists of 130 keys and is divided into nine separate regions. They are:

RESET keys	Three (3) keys
FUNCTION keys	Eight (8) keys
JOG keys	Fifteen (15) keys
OVERRIDES	Thirteen (13) keys
DISPLAYS	Eight (8) keys
CURSOR keys	Eight (8) keys
ALPHA keys	Thirty (30) keys
MODE keys	Thirty (30) keys
NUMERIC keys	Fifteen (15) keys

A detailed description of how and where these keys are used can be found through use of the index. The following are short descriptions of the control panel keys' usage.

RESET KEYS: The RESET keys are in the upper left corner of the control panel.

RESET	Stops all machine motion and places the program pointer to the top of the current program.
POWER UP/ RESTART	Automatically initializes the machine at power up. After initial power up, when this key is pressed, the axes zero return and tool one is put in the spindle.
TOOL CHANGER RESTORE	Restores the tool changer to normal operation after the tool changer has encountered an interruption during a tool change. The button initiates a user prompt screen to assist the operator in recovering from a tool changer crash.

FUNCTION KEYS: Below the reset keys are the function keys. There are eight function keys. They are used to execute special functions implemented throughout the control software.

F1-F4	Used in editing, graphics, background edit, and the help/calculator to execute special functions.
TOOL OFFSET MESUR	Used to record tool length offsets in the offset page during part setup.
NEXT TOOL	Used to select the next tool during part setup.
TOOL RELEASE	Releases the tool from the spindle when in MDI mode. (The remote TOOL RELEASE button is located on the front of the cover to the spindle head. It operates the same as the one on the keypad. It must be held for ½ second before the tool will be released, and the tool will remain released for ½ second after the button is released. While the tool is unclamped, air is forced down the spindle to clear chips, oil, or coolant away from the tool holder.
PART ZERO SET	Used to automatically set work coordinate offsets during part setup.

JOG KEYS: The jog keys are on the left below the function keys. These keys select which axes the jog handle sends signals to and provides for continuous jogging. When a key is pressed briefly, that axis is selected for use by the jogging handle. When a key is pressed and held down, that axis is moved as long as the key is held down. If a "+" key is pressed

INTRODUCTION

Keyboard

and held, the axis is moved so that the tool position is changed in a positive direction relative to the work coordinates. If a "-" key is pressed and held, the axis is moved so that the tool position is changed in a negative direction relative to the work coordinates. The jog keys are locked out if the machine is running.

+A, -A Selects the A axis. Selects the B axis when used with the shift key and control is configured with a fifth-axis option.

+Z, -Z Selects the Z axis.

+Y, -Y Selects the Y axis.

+X, -X Selects the X axis.

JOG LOCK When pressed prior to one of the above keys, the axis is moved in a continuous motion without the need to hold the axis key depressed. Another press of the key stops jogging motion.

To the left side of the jog keys are three keys to control the optional chip auger. If the auger is enabled with Parameter 209, these keys perform the following functions:

CHIP FWD Turns the auger in a direction that removes chips from the work cell.

CHIP STOP Stops auger movement.

CHIP REV Turns the chip auger in the reverse direction.

To the right side of the jog keys are three keys to control the optional automatic spigots. If the spigot is enabled with Parameter 57, these keys perform the following functions:

CLNT UP Pressing this key positions the coolant stream direction one position higher, if possible.

CLNT DOWN Pressing this key positions the coolant stream direction one position lower, if possible.

AUX CLNT Pressing this key while in MDI mode will turn on the Through the Spindle Coolant system, and pressing it again will shut off the system.

OVERRIDES:

The overrides are at the lower left of the control panel. They give the user the ability to override the speed of rapid traverse motion, as well as programmed feeds and spindle speeds.

FEED RATE Not a key.
-10 Decreases current feed rate by 10 percent.
100% Sets control feed rate to programmed feed rate.
+10 Increases current feed rate by 10 percent.
SPINDLE Not a key.
-10 Decreases current spindle speed by 10 percent.
100% Sets spindle speed to programmed speed.
+10 Increases current spindle speed by 10 percent.
CW Starts the spindle in the clockwise direction.
STOP Stops the spindle.

CCW	Starts the spindle in the counterclockwise direction.
5% RAPID	Limits rapid traverse to 5 percent of maximum.
25% RAPID	Limits rapid traverse to 25 percent of maximum.
50% RAPID	Limits rapid traverse to 50 percent of maximum.
100% RAPID	Allows rapid traverse to feed at its maximum.

DISPLAYS:

The display keys are in the center at the top. These eight keys provide access to the different displays and operational information and help routines available to the user. Some of these keys are multi-action keys in that they will display different screens when pressed multiple times. The current display is always displayed on the top left line of the video screen.

PRGRM / CONVRS	Displays the currently selected program. Also used in QuickCode applications.
POSIT	Displays the position of the machine axes. Pressing PAGE UP and PAGE DOWN will show operator, machine, work, and distance-to-go formats in large letter format.
OFFSET	Displays the tool length and radius offsets. PAGE UP will display the values of the axes' work offsets.
CURNT COMDS	Displays the current program, modal program values, and position during run time. Succeeding presses of the PAGE DOWN key will display modal values, system timers, macro variables, tool life and tool load information.
ALARM / MESGS	Shows the full text of an alarm when the alarm message is flashing. Pressing the left or right arrow keys will display an alarm history. Pressing PAGE DOWN will display a page for user messages and notes.
PARAM / DGNOS	Displays and allows changing of parameters that define machine character. Pressing PAGE UP will display lead screw compensation values. Successive PAGE DOWN presses will display general parameters as well as the X, Y, Z, A and B parameters. A second press of the PARAM key will display the first page of diagnostic data. The first page of diagnostic data is discrete inputs and outputs. Pressing PAGE DOWN will display the second page of diagnostic data that consists of additional inputs and analog data.
SETNG / GRAPH	Displays and allows changing of user settings. Pressing the SETNG key twice enables graphics mode where the user can debug the current program and view the program's generated tool path.
HELP / CALC	Displays a brief, on-line manual. Pressing HELP a second time will display the help calculator. There are three pages of calculator help. Pressing the PAGE DOWN key will display milling and tapping help, trigonometry help, or circle help.

CURSOR KEYS:

The cursor keys are in the center of the control panel. They give the user the ability to move to various screens and fields in the control. They are used extensively for editing of CNC programs.

HOME	Context-sensitive key that generally moves the cursor to the topmost item on the screen. In editing, this is the top block of the program. In graphics zoom, it will select full view.
------	--

INTRODUCTION

Keyboard



(UP ARROW)

The up arrow moves up one item, block, or field. In graphics, the zoom window is moved up.

PAGE UP

Used to change displays, move up one page in the editor, or zoom out when in graphics.



(LEFT ARROW)

Used to select individually editable items within the editor, moves cursor to the left. It selects optional data in fields of the settings page and moves the zoom window left when in graphics.



(RIGHT ARROW)

Used to select individually editable items within the editor, moves cursor to the right. It selects optional data in fields of the settings page and moves the zoom window right when in graphics.

END

Context-sensitive key that generally moves the cursor to the bottom most item on the screen. In editing, this is the last block of the program.



(DOWN ARROW)

The down arrow moves down one item, block, or field. In graphics, the zoom window is moved down.

PAGE DOWN

Used to change displays, move down one page in the editor, or zoom closer when in graphics.

ALPHA KEYS:

The alpha keys allow the user to enter the 26 letters of the alphabet along with some special characters.

SHIFT

The shift key provides access to the white characters on the keyboard. Pressing SHIFT and then the white character will result in that character being sent to the control. When a control has a fifth-axis installed, the B axis is selected for jogging by pressing SHIFT and then the +,-A keys.

EOB

This is the END-OF-BLOCK character. It is displayed as a semicolon on the screen and it signifies the end of a programming block. It is the same as a carriage return and then a line feed.

()

The parenthetical brackets are used to separate CNC program commands from user comments. They must always be entered as a pair and may or may not have additional characters separating them. Any time an invalid line of code is received through the RS-232 (or RS-422) port while receiving a program, it is added to the program between these two brackets.

/

The right slash is used as a block delete flag. If this symbol is the first symbol in a block and a BLOCK DELETE is enabled, then that block is ignored at run time. The symbol is also used for division in macro expressions.

In some FANUC compatible controls, the block delete symbol can be used to choose between two options when the "/" symbol is not at the beginning of the

line. For instance, in the following line, T2 is executed when the block delete option is off, and when the block delete option is on, T1 is executed.

```
T1 / T2;
N1 G54
```

This cannot be done on a HAAS control.

A coding method for achieving the same results on a HAAS control is given below:

```
/ T2 M99      (T2 executed when block delete is off)
T1            (T1 executed when block delete is on)
N1 G54
```

[and] Square brackets are used in macro expressions and functions.

MODE KEYS:

The mode keys are in the upper right part of the control panel. These keys change the operational state of the CNC machine tool. There are six major operation modes. The user can enter a specific mode by pressing the desired "arrow" shaped key on the left. The keys in the same row as the pressed mode key are then made available to the user. Otherwise, these keys are not available. The current mode is always displayed on the top line just to the right of the current display on the video screen.

EDIT Selects edit mode.

- INSERT Inserts the text in the input buffer after the current cursor location. Also used to copy blocks of code in a program.
- ALTER Changes the item that the cursor is on to the text in the input buffer. Places an MDI program in the program list.
- DELETE Deletes the item that the cursor is on.
- UNDO Backs out or undoes up to the last 10 edit changes.

MEM Selects MEM mode.

- SINGLE BLOCK Turns single block **on** so that when the cycle start button is pressed, only one block of the program running is executed.
- DRY RUN Used to check actual machine movement without cutting a part. Programmed feeds are replaced by the speed keys in the handle jog row.
- OPT STOP Turns on optional stops. If an M01 code is encountered in the program and is on, then a stop is executed. Depending on the lookahead function, it may not stop immediately. If the program has been interpreted many blocks ahead, and the OPT STOP is pressed, then the nearest M01 may not be commanded. See G103
- BLOCK DELETE Blocks with a slash ("/) as the first item are ignored or not executed when this option is enabled. If a slash is within a block, address codes after the slash will be ignored until after the block, if this option is enabled.

Keyboard

MDI/DNC**Selects MDI or DNC mode.**

COOLNT Turns the coolant on and off.

ORIENT SPINDLE Rotates the spindle to a known position and then locks the spindle. Can be used during setup to indicate parts.

ATC FWD Rotates the tool turret forward to the next sequential tool. If **Tnn** is in the input buffer, the turret will advance to tool **nn**.ATC REV Rotates the tool turret backward to the previous tool. If **Tnn** is in the input buffer, the turret will advance to tool **nn**.**HANDLE JOG****Selects Jogging mode.**

.0001, .1 .0001 inches or .001 mm for each division on the jog handle. For dry run .1 inches/min.

.001, 1. .001 inches or .01 mm for each division on the jog handle. For dry run 1. inches/min.

.01, 10. .01 inches or .1 mm for each division on the jog handle. For dry run 10. inches/min.

.1, 100. 0.1 inches or 1.0 mm for each division on the jog handle. For dry run 100. inches/min.

ZERO RET**Selects Zero Return mode.**

AUTO ALL AXES Searches for all axes' machine zero.

ORIGIN Zeros out various displays and timers.

ZERO SINGL AXIS Returns the axis that is specified in the input buffer to machine zero.

HOME G28 Returns all axes to machine zero. Does not search.

LIST PROG**Selects Program List mode and displays a list of the programs in the control.**

SELECT PROG Makes the highlighted program on the program list the current program. The current program will have an asterisk preceding it in the program list.

SEND RS232 Transmits programs out the RS232 serial port. This key is also used for optional RS-422.

RCV RS232 Receives programs from the RS232 serial port. This key is also used for optional RS-422.

ERASE PROG Erases the highlighted program or the program specified in the input buffer.

NUMERIC KEYS: The numeric keys give the user the ability to enter numbers and a few special characters into the control.

CANCEL The Cancel key is used to delete the last character entered during editing or field input.

SPACE This is a space and can be used to format comments placed into programs.

WRITE This acts as the general purpose enter key. Any time that user needs to change any information in the control, this key is pressed.

-, . Used to negate numbers, or provide decimal precision.

+, =, #, and * These symbols are accessed by first pressing the SHIFT key and then the key with these symbols. They are used in macro expressions.

?, %, \$, !, &, @, and : These are additional symbols, accessed by pressing the SHIFT key. They can be used in program comments.

10. POWER ON/OFF

10.1 POWER ON

There is only one way to turn on this CNC. This is by pressing the green "On" button at the top left of the control panel. The main breaker at the rear of the mill must be on before this button will turn on the mill. Any interruption to power will turn the mill off and this button must be used to turn power back on again.

Upon power up, the machine must find its fixed reference point before any operations can occur. After power on, pressing the POWER UP/RESTART will establish this point. The ZERO RET mode and AUTO ALL AXES button may also be used to initialize the system after all alarms are cleared. A single axis can be selected by first pushing the **X**, **Y**, **Z**, **A**, or **B** key and then the ZERO SINGL AXIS key. The position thus found is used as machine zero. Note that the Z-axis will shift downwards about five inches as the search for zero is finished; so, keep clear.

CAUTION !!!! After power on, the machine does not know its home position or stored stroke limits until it has been zero returned by the POWER UP/RESTART key or the ZERO RET/AUTO ALL AXES key. It is possible to jog the machine with the handle or jog keys at the lower feeds. If it is jogged unchecked in the negative directions, you may damage the sheet metal covers or overload the ball screws. To avoid this, always properly ZERO RET the machine immediately after power on before doing anything else.

NOTE: Tool changer goes to tool #1 first, then to tool designated in Setting 81.

After initializing, all Position displays are reset to zero. The ORIGIN key will zero out the operator display only when that display is selected and in JOG mode. During ZERO RET, the Distance To Go display contains a diagnostic value.

The HOME G28 key should be used any time after the initial power up. This will return the Z-axis first and then the **X**, **Y**, **A**, and **B** axes all at rapid rate. If the Z-axis is positioned above the machine zero, the **X**, **Y**, **A**, and **B** axes are moved first. This key will work in any of the operating modes. The manual G28 button does

not use any intermediate return point the way the programmed G28 does. Any auxiliary axes (C, U, ...) are returned to home after X, Y, Z, A, and B.

NOTE: Repairs to the motor, ball screw, or home switch will affect the zero return point and must be done only by a factory trained technician. Serious damage to the ball screw, way covers, linear guides, or tool changer may occur if the zero return point is not properly set.

10.2 POWER OFF

Pressing the red POWER OFF button will remove power to the machine instantly. The machine can also be programmed to turn off at an end of cycle (M30) or after a preset amount of time that the machine sees no activity. These are Settings 1 and 2.

A sustained overvoltage condition or sustained overheat condition will also shut this machine off automatically. If either of these conditions exists for 4.5 minutes, the machine will start the 30 second auto-shutdown. Alarm 176 is displayed when an overheat shutdown begins and Alarm 177 is displayed when an overvoltage shutdown begins.

Any power interruption, including the rear cabinet main circuit breaker, will also turn this machine off. Power must be restored and the POWER ON button pressed to restore operation.

11. MANUAL OPERATION

11.1 MDI

Manual data input allows you to enter data that can be executed on a line by line basis instantly without having to use the EDIT and MEM modes. In this control, MDI is actually a scratch pad memory that can execute many lines of instruction without having to disturb your main program in memory. The data in MDI will be retained even when switching modes or in power off.

Editing with MDI is the same as memory editing.

The MDI mode also allows for manual operation of coolant, spindle, and tool changer.

A program in MDI can be saved as a normal named program in memory by placing the cursor at the beginning of the first line (HOME), typing **Onnnn** (new program number), then pushing ALTER. This will add that name to the program list and clear MDI.

The entire MDI program may be cleared by pressing the ERASE PROG key while in MDI.

A fast way to select a tool is to type **Tnn** and, instead of INSERT, press either ATC FWD or ATC REV. This will directly select that tool.

When DNC is enabled with Setting 55, a second push of the MDI button will put the control into DNC mode.

When the Parameter 57 flag DOOR STOP is set to 1, manual tool change operations are not allowed with the doors open. In addition to this, the maximum spindle speed is 100 RPM.

11.2 HANDLE/JOG

Manually moving the axes is accomplished by pressing the mode button labeled HANDLE JOG and then by using the JOG keys or the Handle to move the axis. Both the JOG buttons and the Handle are enabled simultaneously without needing to select between them.

Jog feed rate or handle resolution is selected by the four keys to the right of the HANDLE JOG key. Jog feeds from .1 inch per minute to 100 inch per minute or handle divisions from .0001 inch to .1 inch are selectable. Auxiliary axes can be manually jogged from the front panel.

During jogging, the FEED RATE override buttons will adjust the rates selected from the keypad. This allows for very fine control of the jog speed. It does not change the handle step size.

In the center of the jog buttons is a key labeled JOG LOCK. This key will cause the axis you are jogging to continue jogging even after you release the key. Press this key and then press the selected axis motion key to start. Motion will stop as soon as the JOG LOCK button is pressed again, another axis is selected or RESET is pressed. This feature is handy, for example, when you are slow milling the soft jaws of a vise.

In order to select another axis for jogging while using the handle, use +/- X, Y, Z, or A buttons. When one of these buttons is pressed, that axis is selected for HANDLE JOG but does not move unless the button is held down for more than 1/2 second. After 1/2 second, that axis is moved in the selected direction and at the selected feed rate.

All aspects of handle jogging for the fifth axis work as they do for the other axes. The exception is the method of selecting jog between axis A and axis B. By default the '+A' and '-A' keys will select the A axis for jogging. The display will show "JOGGING A AXIS HANDLE .01" while you are jogging the A axis. The B axis can be selected by pressing the SHIFT key, and then either the '+A' or '-A' key. When this is done the control will switch to jogging the B axis and the display will change to "JOGGING B AXIS HANDLE .01".

The axis assigned to the '+A' and '-A' keys will remain selected for jogging even if the operating mode is changed or if the machine is turned off. The selected axis for '+A' and '-A' can be toggled by pressing the shift key prior to pressing the '+A' or '-A' keys.

An optional remote jog handle is now available. It operates exactly the same as the standard jog handle, except that the desired axis and jog increments can be selected by switches on the remote handle.

The jog axis switch on the remote handle may be switched to OFF, X, Y, Z, or A. When it is set to OFF, the standard jog handle on the control works normally. When X, Y, Z, or A is selected, that axis is selected for jogging. The jog increments switch may be switched to X1, X10, or X100. These correspond to the .0001/.1, .001/1., and .01/10 buttons, respectively, on the keypad.

12. AUTOMATIC OPERATION

12.1 OPERATION MODE

There are six modes of operation of the VF Series CNC Mill. They are:

EDIT	Used to make manual changes to a part program.
MEM	Used to run a users part program stored in memory.
MDI/DNC	Used to quickly manually enter and run a program.
HANDLE/JOG	Used to move the axes with the handle or JOG buttons.
ZERO RET	Used to search for machine zero and to return to machine zero automatically.
LIST PROG	Used to list, send, receive and delete programs.

INTRODUCTION

Automatic Operation

Changes to the mode are made by pressing of the buttons on the top right quadrant of the keypad that have the above labels. If an operation is started, such as running a program, you cannot change modes until the operation is stopped. The six mode selection buttons are arranged vertically and, generally, the keys to their right apply only in that selected mode.

12.2 PROGRAM SELECTION

Program selection is done from the LIST PROG mode. This mode will list all of the programs stored in memory and allow you to select one as the main program. This is the program that will be run when you press START in MEM mode. On the LIST PROG display, the program with the " * " is the main program. The selected program is the one that you will see on the EDIT display and is the one that will be run when you press START in MEM mode.

To select an existing program, press the CURSOR **up** or **down** buttons until the program you want is highlighted and then press the SELECT PROG button. The " * " will move to that program.

To select a new program (create a new program) or to select an existing program, you may also enter **Oxxxx** from the keyboard and then press SELECT PROG button.

There is a maximum of 200 programs stored in this control at a time.

12.3 STARTING AUTOMATIC OPERATION

Before you can run a program, it must be loaded in the current memory. To select a program, push the LIST PROG mode key. Use the cursor to find the desired program and then push SELECT PROG. The program list includes the program name and the first comment. If the control was turned off while running, that program will automatically be in current memory and selected.

If the machine has just powered up, you need to first push the POWER UP / RESTART key. This will initialize all axes and the tool changer, display the Current Commands, and go to MEM mode with the control ready to run. Pushing the CYCLE START button in the lower left of the control panel will begin execution.

To start a program other than at the beginning, scan to the block number using the keypad and the **down** arrow or PAGE DOWN until you reach the desired start place. Push the MEM key and CYCLE START to begin. The Program Restart function, selected from the Setting page 36, will change the way a program operates if you start from other than the first block. The setting called Program restart "ON" will ensure that the correct tool and axis positions are selected when you start from part way through a program.

Any errors in your program will cause an alarm and stop the running of the program. Typical alarms are travel limits and missing I, J, and Q codes. Attempts to move outside of the limits of travel will also cause an alarm.

When cutting materials that produce hot chips, use coolant to prevent damage to the plastic windows and enclosure.

At any time that a program is running, the bottom left corner of the CRT will show RUNNING. If it does not show this, the program has completed, has been stopped by the operator, or has been stopped by a fault condition.

12.4 PROGRAM RESTART

The Program Restart function may be selected from the Setting page. It allows a program to be restarted from other than the first block. You do this by using the CURSOR **up** and **down** keys in MEM mode to select the block to start operation and pressing CYCLE START. If Program-Restart is on, program interpretation will begin with the first block but no motion of the machine will occur until execution gets to the selected restart

block. When it gets to the restart block, the axis and tools will be moved to the correct position and normal operation will proceed from there.

12.5 STOPPING AUTOMATIC OPERATION

There are several ways a program can be stopped. They include both normal stops and abnormal, or alarm caused, stops. The normal stops are:

- 1) Normal completion at M00, M01, M02, or M30
- 2) A FEED HOLD stop by the operator. This is continued by pressing CYCLE START again.
- 3) A SINGLE BLOCK stop when operator selected. This is continued by pressing START again.
- 4) Door Hold stop caused by operator opening the enclosure doors. This continues when doors close.

The abnormal stops are:

- 1) **Operator Reset**
This stops all axes' motion, stops the tool changer, turns off the spindle, and turns off the coolant pump. Program operation cannot be continued from the stopping point. If Setting 31 is On, the program pointer is reset to the beginning.
- 2) **Emergency Stop**
This stops all axes' motion, disables the servos, stops the tool changer, turns off the spindle, and turns off the coolant pump. Program operation cannot be continued from the stopping point. This will also stop any auxiliary axes' motion. RESET must be used at least twice to remove the alarms and start again.
- 3) **Alarm condition**
This can occur any time an alarm comes on during program operation. Since a program cannot be restarted until RESET is pressed, a program execution cannot be continued from the stopping point. Alarms can be caused by programming errors or machine faults. Use the Graphics simulation mode to test your program first for errors.
- 4) **Power-off**
This will stop all motors within one second but does not guarantee any conditions when the machine is powered-on again.

12.6 EMERGENCY STOP SWITCH

The EMERGENCY STOP switch is normally closed. If the switch opens or is broken, power to the servos will be removed instantly. This will also shut off the tool changer, spindle drive, and coolant pump. The EMERGENCY STOP switch will shut down motion even if the switch opens for as little 0.005 seconds.

Be careful of the fact that Parameter 57 contains a status switch that, if set, will cause the control to be powered down when EMERGENCY STOP is pressed.

You should not normally stop a tool change with EMERGENCY STOP as this will leave the tool changer in an abnormal position that takes special action to correct.

Part Program Storage and Edit

Note the tool changer alarms can be easily corrected by first correcting any mechanical problem, pressing RESET until the alarms are clear, selecting ZERO RETURN mode, and selecting AUTO ALL AXES.

If the shuttle should become jammed, the control will automatically come to an alarm state. To correct this, push the EMERGENCY STOP button and remove the cause of the jam. Push the RESET key to clear any alarms. Push the ZERO RETURN and the AUTO ALL AXES keys to reset the Z-axis and tool changer. Never put your hands near the tool changer when powered unless the **EMERGENCY STOP** button is pressed.

12.7 WORKING LIGHTS

Working lights allow the operator to easily monitor the status of a machine by observing the red and green beacons located directly on top of the control panel arm.

During normal operation the GREEN beacon will be steadily on.

The beacon will *flash* GREEN if:

- The operator selects FEEDHOLD or SINGLE BLOCK stop.
- If the control is in a M00, M01, M02, M30

The beacon will *flash* RED if:

- The control encounters an alarm.
- If the EMERGENCY STOP button is pressed.

If the control is in a RESET state the beacon will be off.

13. PART PROGRAM STORAGE AND EDIT

When using anything other than HELP or Messages function, alphanumeric key entries are displayed along the bottom line of the CRT. This is called the data entry line. When the line contains what you want to enter, press the WRITE, ALTER, or INSERT key as appropriate.

When the HELP display is selected, the alphanumeric keys are used to select one of the topics; so they are not displayed on the data entry line of the CRT.

When the Message function is selected, the cursor is positioned on the screen and you type directly into the display.

13.1 CREATING PROGRAMS

To create a new program, you must be in the PROGRAM DISPLAY and LIST PROG mode. Enter **0** (letter, not number) and a four digit program number and press SELECT PROG. The selected program is the "Main" program and is the one you will see on the MEM and EDIT modes. Press EDIT to show the new program. A new program will consist of only the **0nnnn** and an EOB (;). All further entries are made by typing a letter followed by a numeric value and pressing INSERT, ALTER, or WRITE. All items entered into a program are either addressed data (a letter of the alphabet followed by a number), a comment (text surrounded by parenthesis), or the End-Of-Block (EOB or ;).

The CURSOR **up** and **down** keys can be used to search for the entered value. Simply enter the value to search for on the bottom line and press the CURSOR **up** or **down** keys. The CURSOR **up** key will search for the entered item backwards to the start of the program. The CURSOR **down** key will search forward to the end of the program. Searching also works in MEM mode. If you enter a letter without a number, the search will stop on the first use of that letter with any value.

NOTE : When INSERT is pressed, the new data is put in after the highlighted data. The CURSOR **up**, **down**, **left**, and **right** keys are used to select the highlighted item. The PAGE UP and PAGE DOWN keys move farther distances and the HOME and END keys go to the start or end of the program. All of these keys work in EDIT, MEM, and MDI modes.

A comment can be edited without entering the entire comment again. Simply move the highlighted cursor to the character you wish to change, enter the new characters, and press ALTER, INSERT, or DELETE.

After creating a program, the name can be very easily changed by simply altering the **Oxxxx** on the first line. If the maximum number of programs are already present, the message "DIR FULL" will be displayed and the program cannot be created. The maximum number of programs in memory is 200.

13.2 EDITING PROGRAMS

The EDIT mode is used to make changes to a program already in memory. If a program does not exist yet, the LIST PROG mode is used to create it. A newly created program contains only the program **Oxxxx** name and an EOB.

To enter the EDIT mode, press the EDIT mode key. The screen will display the current program. If no program file exists, program 00000 will be displayed. To change a program name, move the cursor to the existing **Oxxxx**, type in the letter "O" followed by a four digit number, such as O1234, and press the ALTER key. The upper right hand screen will display the new program number. Your data will first appear in the lower left screen and will be input to the upper screen upon pressing INSERT, ALTER, or WRITE.

To enter a program from the keypad, type in the data you wish and press the INSERT key. More than one code, such as X, Y, and Z, can be entered before you press INSERT. After a program is entered, you may wish to change the data. Use the CURSOR keys to move the cursor to the word you wish to edit. Input your desired change in the lower left screen and then press INSERT, ALTER, or DELETE to alter the data. Use the UNDO button to reverse any changes. The UNDO button will work for the last nine entries.

The CURSOR **up** and **down** keys can be used to search for the entered value. Simply enter the value being searched for on the bottom line and press the CURSOR **up** or **down** keys. The CURSOR **up** key will search for the entered item backwards to the start of the program. The CURSOR **down** key will search forward to the end of the program. Searching also works in MEM mode. If you enter a letter without a number, the search will stop on the first use of that letter with any value.

You can change to a different program while in the EDIT mode by using the CURSOR **up** and **down** keys, enter **Oxxxx** on the input line and then press the CURSOR **up** and **down** keys or the F4 key. **Oxxxx** is the program you wish to change to.

As an option, the jog handle can be used to move the cursor during editing. Parameter 57 is used to enable this function. If enabled, the handle will act like the CURSOR **left** and **right** buttons.

Background editing is also possible with this machine and is now a standard feature. If background editing is available on your control, all of the above editing functions can be used while a program is running in MEM. Background edit is described in detail later on in this chapter.

Editing error messages:

Guarded Code	You tried to remove the Oxxxx from start of a program.
Bad Code	A line contained invalid data or comment over 80 characters.
Editing Error	Some previous edit was not completed; fix the problem or press UNDO.
Bad Name	Program name Oxxxx is invalid or missing.

Invalid Number	The number with an alphabet code was invalid.
Block Too Long	A block may only be 256 characters.
No Code	An insert was done without any data to insert.
Can't Undo	May only use undo for previous nine changes.
End Of Prg	End of prog EOB cannot be deleted.

13.3 BACKGROUND EDIT

As a standard feature, this machine is shipped with a BACKGROUND EDIT capability. With BACKGROUND EDIT, you may edit a program in memory while any other program is being run. BACKGROUND EDIT can be enabled and disabled by Parameter 57.

BACKGROUND EDIT is selected from MEM mode when in PROGRAM DISPLAY by typing **Oxxxx** for the program you want to edit and pressing F4. If you do not enter the **Oxxxx**, you will instead get the PROGRAM REVIEW display.

While in BACKGROUND EDIT, you may perform any of the operations available in the EDIT mode. The last five lines of the CRT will, however, display the status of the running program and the top line will show the name and line number of the running program.

Selecting any other display or pressing F4 will exit from BACKGROUND EDIT. In order to list the programs that are in memory, a display function is available to view the program memory list while a program is running. This display is called LIST. It is selected by pressing the LIST PROG button while a program is running. The display is just like the LIST PROG mode display but it does not allow any send, receive, copy, select, or erase functions.

The CYCLE START button may not be used while in BACKGROUND EDIT. If the program contains an M00 stop, you must exit BACKGROUND EDIT and then press CYCLE START to resume the program.

All of the changes made during BACKGROUND EDIT are saved in a different memory area until the running program stops. This means that you can even edit the program that is running, or any of its subprograms, and those changes will not effect the running program.

The first time you select a program for BACKGROUND EDIT, you will get the message PROG EXISTS if the program is already in memory or NEW PROG if it is not. The NEW PROG message means that the program is being created and will be initially empty. In either case, you will then be able to edit that program. The second time you select a program for BACKGROUND EDIT without stopping the running program, you will get the message SECOND EDIT.

When you are in BACKGROUND EDIT and the running program finishes, the display will automatically change to the PROGRAM DISPLAY and will show the program that just finished running. To continue editing your program, you must select it with LIST PROG and then display it in EDIT mode.

BACKGROUND EDIT is not available from MDI or from DNC operating modes

13.4 DELETING PROGRAMS

To delete an existing program, you must be in LIST PROG mode. The programs will be listed here by program number. Use the CURSOR up or down keys to highlight the program number, or type in the program number at the blinking cursor, then press the ERASE PROG key.

All the programs in the list may be deleted by selecting ALL at the end of the list and pressing the ERASE PROG key. Use caution when deleting single programs, and read all prompts, to ensure that ALL programs are not selected. The UNDO key will not recover programs that are deleted.

13.5 SPECIAL FUNCTION KEYS

The F1, F2, F3, and F4 buttons perform different functions depending on what display and mode is selected. The following is a quick summary of the **Fn** buttons:

- F1 In EDIT mode and PROGRAM DISPLAY, this will start a block definition.
In LIST PROG mode, F1 will duplicate a program already stored and give it a new name from the command line.

In OFFSET display, F1 will set the entered value into the offsets.
- F2 In EDIT mode, PROGRAM DISPLAY, this will end a block definition.
- F3 In EDIT and MDI modes, the F3 key will copy the highlighted circular help line into the data entry line at the bottom of the screen. This is useful when you want to use the solution developed for a circular motion. Push INSERT to add that circular motion command line to your program. In the calculator HELP function, this button copies the value in the calculator window to the highlighted data entry for Trig or Circular Help.
- F4 When in EDIT mode with no program running, entering **Oxxxx** in the input line and pressing F4 will change the program being edited to **Oxxxx**.

When in MEM mode and PROGRAM DISPLAY, F4 can be pressed to select either BACKGROUND EDIT or PROGRAM REVIEW. BACKGROUND EDIT is selected by entering the program number at the input line and pressing F4. BACKGROUND EDIT can only be selected when a program is running. PROGRAM REVIEW can be selected whether or not a program is running, simply by pressing F4. If a program is running, PROGRAM REVIEW will show the running program on the left half of the screen, and allows the operator to review the program on the right half of the screen.

In the calculator HELP function, F4 uses the highlighted Trig, Circular, or Milling data value to load, add, subtract, multiply or divide with the calculator.

13.6 THE UNDO KEY

A very powerful keyboard button available in this control is the UNDO button. When editing, this button will allow you to basically undo any changes or edits you have made but wish you hadn't. Any time you use the INSERT, ALTER, or DELETE buttons, the condition of the original block is saved and can be restored with the UNDO button. In fact, the previous nine changes can be undone in the opposite order that they were entered by pressing the UNDO button for each change that is to be backed out.

The UNDO button can be used in EDIT, BACKGROUND EDIT, and MDI. But if you change operating modes between EDIT and MDI, you cannot use the UNDO button as the list of saved data is cleared.

13.7 BLOCK OPERATIONS

Block operations can be performed on a group of one or more blocks of the program. These operations include BLOCK DUPLICATE, BLOCK MOVE, and BLOCK DELETE. Prior to a block being defined, the bottom right of the screen show how to define a block; the F1 key is pressed when the cursor is on the first line of the block and the F2 key is pressed when the cursor is on the last line of the block.

Once a block is defined, it is displayed in reverse video and the lower right of the screen shows how to manipulate the block; the INSERT key is used to duplicate the defined block wherever the cursor is positioned, the DELETE key is used to delete the block, the ALTER key is used to move the block, and the UNDO key cancels the block definition.

When a block is defined, the cursor is indicated by the ">" symbol and is always at the beginning of a line. When a block is copied or moved, the lines are added after the block with the cursor. Only whole command lines may be moved with the block functions.

Parts of programs can be copied from one program to another with the block copy feature. This is done by highlighting the section of code that is to be copied using the F1 and F2 keys. Once a section of code is highlighted, you then change to another program by selecting an existing program or create a new one. Cursor to the location that the previously defined block is to be inserted and press the INSERT or write key. A copy of the defined block will be inserted into the current program and the copied code segment becomes the currently-defined block. Press the UNDO key to exit the BLOCK COPY mode.

Blocks of code can be copied into an MDI program, but blocks of code cannot be copied from an MDI program into another program. You can always rename the MDI program and then copy its text to any other program in the manner described above.

14. PART PROGRAM INPUT / OUTPUT

14.1 DATA INPUT / OUTPUT (RS-232)

Programs are sent or received through the first RS-232 port located on the rear control box pendant side. Note that this is the top connector. All data sent or received is ASCII. In order to use this port, you will need to obtain a cable and connectors with the following wiring:

pin #1	Shield Ground	pin #2	TXD-Transmit Data
pin #3	RXD-Receive Data	pin #4	RTS (optional)
pin #5	CTS (optional)	pin #7	Signal Ground

All other pins are optional and are not usually used. The RS-232 connector is a DB-25 and is wired as a DTE. This means that we send data on the TXD wire and receive data on the RXD wire. If you do not understand this, your dealer will be glad to help. The simplest connection would be to an IBM PC that can be done with a standard cable made up of a DB-25 male on one end and a DB-25 female on the other. Pin 2 at one end is wired to pin 3 at the other end, pin 3 to pin 2 and pin 7 is wired to pin 7.

All RS-232 data is ASCII but the number of bits, parity and speed can be changed from settings. The number of data bits is selected with Setting 37 for either 7 or 8. Parity is selected with Setting 12 and is none, even, odd, or zero. Zero parity will always set the parity bit to 0. The data speed is selected with Setting 11.

Once the connection to your computer has been made and verified, go to the Setting page and set the baud rate, parity, number of stop bits, end of block (EOB) format, and leader parameters to match your requirements.

All programs sent to the control must begin with a line containing a single % and must end with a line containing a single %. All programs sent by the control will have these % symbols.

To receive a program, push the LIST PROG key. Move the cursor to the word ALL and push the RECV RS-232 key and the control will receive all main and sub programs until it receives a % sign indicating end of input. Please note that when using "ALL", all your programs must have an address **Oxxxx** to be filed. If you do not have a program number, type in the program number before you push RECV RS-232 and the program will be stored under that number. You can also select an existing program for input and it will be replaced. An ASCII EOF character (code 04) will also terminate input. The colon character (:) may be used in place of the 0 for a program name, but it is always displayed as 0.

When receiving RS-232 data, there is a status message at the bottom of the screen. It will update as follows:

WAITING	When you first press RECV RS-232.
LOADING XXX	When first % is received; if in XMODEM, XXX is the current block being loaded.
LOADING Onnnn	When program name is received.
RS232 DONE	When complete and last % is received.
RS232 ABORT	When anything causes abnormal stop.

There is a maximum of 200 programs stored in this control at a time.

To send a program, use the cursor as above to select the program and push the SEND RS-232 key. You can select "ALL" to send all of the programs in memory. A setting can be turned on to add spaces to the RS-232 output and improve the readability of your programs.

The synchronization protocol used to send data to slower computers is selected from the Setting 14. Setting 14 may be set to XON/XOFF, RTS/CTS, DC CODES, or XMODEM. Transmission can be stopped with either the XON/XOFF characters or the RTS/CTS wires.

Parameters, settings, offsets, and macro variables pages may also be sent individually via RS-232 by selecting the "LIST PROG" mode, selecting the desired display screen, and pushing the SEND key. They can be received by pushing the RECV key.

The EOB (semicolon) character is not normally sent by the RS-232 port. If it is received by the input port, it will cause a blank line in the program.

The format of data sent and received for parameters, settings, offsets, and macro variables is the following:

```
%  
N0 Vnnnnnn  
N1 Vnnnnnn  
N2 Vnnnnnn  
.  
.  
.  
%
```

The **N** number is the data number and **V** is the value. **N0** is a CRC code that is computed by the control prior to sending the data. The **N0** value is mandatory with parameters but is optional with settings and offsets. If you make a change to some saved data value and leave the old CRC, you will get an alarm when you try to load that data. With settings and offsets, you should delete the **N0** line if you make changes to the saved data.

NOTE: Data will be loaded even though an alarm has been generated.

Data that is received garbled is usually converted into a comment and stored into your program while an alarm is generated. In addition, any parity errors or framing errors will generate an alarm and they will also stop the receive operation.

At the end of a send or receive function, the bottom left corner of the display will show either: "RS232 DONE" for normal completion or "RS232 ABORT" if any errors cause it to stop. The actual errors are listed in the ALARM display.

The Haas CNC serial ports optionally support the full DC1, DC2, DC3, DC4 code sequence that is compatible with paper tape readers and punches. Setting 14 is used to select this mode of operation. Setting 14 can be set to "DC CODES". When this setting is selected, the following occurs:

- 1) When sending out of the serial port, a DC2 (0x12) will precede all other data. This code is used to turn on a paper punch.
- 2) When sending out of the serial port, a DC4 (0x14) will follow all other data. This code is used to turn off a paper tape punch.
- 3) When receiving from the serial port, a DC1 (0x11 Xon) is sent first. This code is used to turn on a paper reader.
- 4) When receiving from the serial port, a DC3 (0x13 Xoff) is sent after the last % is received. This code is used to turn off a paper tape reader.

Note that the Setting 14 selection XON/XOFF is similar to the "DC CODES" selection. Both of these settings use the DC1/DC3 XON/XOFF codes to start/stop the sender when data is received too fast. When DC CODES is selected for Setting 14 (synchronization), serial port #1 will transmit an XON (DC1) if a character has not been received for five (5) seconds.

XMODEM may also be selected in setting 14. It is a receiver-driven communications protocol that sends data in blocks of 128 bytes. Setting **synchronization** to XMODEM gives your RS-232 communication an added level of reliability because each block is checked for integrity. If the receiver determines that the most recently sent block is in error, it will request that the sender try to send the block again.

In order to use XMODEM, parity must be none, and RS-232 data bits must be set to 8. Also, the computer that is sending the data must be equipped with a communications package that supports the XMODEM protocol. It must be set to XMODEM to operate.

This version of XMODEM supports **checksum** verification only. Also, 512 bytes of memory must be available before using XMODEM with DNC.

WARNING! One of the biggest causes of electronic damage is a lack of a good earth ground on both the CNC and the computer that is connected by RS-232. A ground fault condition (i.e., a lack of good ground on both) will damage the CNC or the computer, or both.

Parameters, settings, macro variables and offsets may also be sent individually via RS-232 by selecting the LIST PROG mode, selecting the desired display screen, and pushing the SEND key. They can be received by pushing the RECEIVE key.

The settings that control RS-232 are:

11	BAUD RATE
12	PARITY
13	STOP BITS
14	SYNCHRONIZATION
24	LEADER TO PUNCH
25	EOB PATTERN
37	NUMBER DATA BITS

Port #2 on the side cabinet is dedicated to auxiliary axes communication. See Auxiliary Axis Control for further more information.

14.2 DATA INPUT / OUTPUT (OPTIONAL RS-422)

Programs may also be sent or received by the optional RS-422. It is sent through the top DB25 connector located on the rear control box pendant side. All data sent or received is ASCII. In order to use this port, you will need to obtain a cable and connectors with the following wiring:

pin #1	Shield Ground	pin #2	TXD-Transmit Data
pin #3	RXD-Receive Data	pin #4	RTS (optional)
pin #5	CTS (optional)	pin #7	Signal Ground
pin #13	CTS' (optional)	pin #14	TXD'-Transmit Data
pin #16	RXD-Receive Data	pin #19	RTS' (optional)

All other pins are optional and are not usually used. The serial port connector is a DB-25 and is wired as a DTE. This means that we send data on the TXD wire and receive data on the RXD wire. If you do not understand this, your dealer will be glad to help. The simplest connection would be to an IBM PC; that can be done with a standard cable made up of a DB-25 male on one end and a DB-25 female on the other. Pin 2 at one end is wired to pin 3 at the other end, and pin 3 to pin 2. Pin 14 at one end is wired to pin 16 at the other end, pin 16 to pin 14. Pin 7 is wired to pin 7.

RS-422 is controlled in exactly the same way as RS-232. All features, settings, and keys are the same. Refer to the previous section for detailed instructions on using RS-422.

14.3 DIRECT NUMERICAL CONTROL (DNC)

As a standard feature, this machine is shipped with a DNC capability. With DNC, there is no limit to the size of your CNC programs. The programs are directly executed by the control as they are sent over the RS-232 (or RS-422) interface. Note, that this is the first serial port or the top connector. Do not confuse DNC with RS-232 uploading and downloading.

If you wish to use DNC, it is enabled by Parameter 57 and Setting 55. When enabled, DNC is selected by pressing MDI a second time when already in MDI. DNC mode will not be enabled unless there is a minimum of 512 bytes of user memory available. When DNC is selected, the PROGRAM DISPLAY will show:

WAITING FOR DNC...

This means that no DNC data has been received yet and you may begin sending data. You must start sending the program to the control before the START button can be pushed. After the beginning of the program is seen by the control, the display will show part of the program and a message at the bottom, left of the CRT will show DNC PROG FOUND. After the program is found, you may push CYCLE START just like running any other program from Memory.

If you try to press START before receiving a program, you will get the message: NO DNC PROG YET. The reason for not allowing the command of START before receiving the DNC program is for safety. If the operation is allowed to start from a remote location, the operator may not be present to ensure that the machine is operating safely.

While a DNC program is executing, you are not allowed to change modes. You must first press RESET to stop the program.

When the end of the DNC program is received, the message DNC END FOUND is displayed. When the DNC program is finished running, the PROGRAM DISPLAY will show the last few lines of the program. You must press RESET or exit the DNC mode before you can run any other programs. If you try to press START before RESET of the previous DNC, you will get the message: RESET FIRST.

DNC supports DRIPMODE. The control will execute one block at a time from the RS-232 port. Each block entered will be executed immediately with no block lookahead buffering. The exception is that Cutter Compensation requires three blocks of motion commands to be buffered prior to a compensated block being executed.

There are several restrictions on what can be in a DNC program. An M98 Pnnn may not be programmed to jump to another part of your program. An M30 is not allowed as it is not possible to start over at the beginning. The program must begin with a % just like any other program sent over RS-232 and the program must end with a %. The data rate selected for the RS-232 port by settings must be fast enough to keep up with the rate of block execution of your program. If the data rate is too slow, the tool may be stopped in a cut when you might otherwise expect continuous cutter motion. The highest standard RS-232 data rate available is 38400 bits per second. As an option, 115200 bits per second is available. If the displays are turned off (M76) the maximum DNC rate is 150 blocks per second.

It is recommended that DNC be run with parity selected because an error in transmission will then be detected and will stop operation of the DNC program without crashing. The settings page is used to select parity. The recommended RS-232 settings for DNC are:

9600 or 19200 BITS PER SECOND
EVEN PARITY
2 STOP BITS
XON/XOFF

Full duplex communication during DNC is possible by using the G102 command to output axes coordinates back to the controlling computer. When DNC is running, BACKGROUND EDIT is not available.

14.4 FLOPPY OPERATION

The optional floppy drive system applies to all machines operating with revision 7.08 software or later.

All files must be on PC/MS DOS formatted 720K or 1.44M floppy disks and must reside in the root directory. Parameter 209 FLOPPY ENABLE must be 1.

NOTE: Use an empty (containing no other files) floppy disk for faster operation.

All programs must begin with a line containing a single % and must end with a line containing a single %. All programs saved by the control will have these % symbols.

Programs may be loaded and saved from the floppy disk. To **LOAD** a program, press the LIST PROG key with PRGM selected. Enter the floppy file name and press F3 and the control will receive all main and sub programs until it receives a % sign indicating end of input. Please note that when using "ALL", all your programs must have an address **Oxxxx** to be filed. An ASCII EOF character (code 04) will also terminate input. The colon character (:) may be used in place of the **O** for a program name, but it is always displayed as **O**.

When loading floppy data, there is a status message at the bottom of the screen. It will update as follows:

LOADING Onnnn	When program name is received.
FLOPPY DONE	When complete and last % is received.
FLOPPY ABORT	When anything causes abnormal stop.

There is a maximum of 200 programs stored in this control at a time.

To **SAVE** a program to floppy, press the LIST PROG key with PRGM selected. Enter the floppy file name, use the cursor as above to select the program, and press the F2 key. You can select "ALL" to send all of the programs in memory.

Parameters, Settings, Macro Variables, and Offsets may also be sent individually to the floppy disk by selecting the "LIST PROG" mode, entering the floppy file name, selecting the desired display screen (PARAM, SETNG, OFSET, or the Macro Variables page of CRNT CMDS), and pressing the F3 key. They can be received by pressing the F2 key.

If an EOB (semicolon) is loaded, it will cause a blank line in the program.

The format of data sent and received for settings, offsets, and parameters is the following:

```
%
NO Vnnnnnn
N1 Vnnnnnn
N2 Vnnnnnn
.
.
%
```

The **N** number is the data number and **V** is the value. **NO** is a CRC code that is computed by the control prior to sending the data. The **NO** value is mandatory with parameters but is optional with settings and offsets. If you make a change to some saved data value and leave the old CRC, you will get an alarm when you load that data. With settings and offsets, you should delete the **NO** line if you make changes to the saved data.

Data that is received garbled is usually converted into a comment and stored into your program while an alarm is generated. Errors generating an alarm may also stop the receive operation.

To get a **DIRECTORY LISTING**, select the PRGM/LIST PROG mode, and press F4. This will generate a Disk Directory Listing that will be saved in program 0xxxx where xxxx is defined in parameter 227. The default value is 8999.

At the end of a save or load function, the bottom left corner of the display will show either: "FLOPPY DONE" for normal completion or "FLOPPY ABORT" if any errors cause it to stop. The actual errors are listed in the ALARM display.

14.5 FLOPPY - DIRECT NUMERICAL CONTROL (DNC)

As a standard feature, this machine is shipped with a DNC capability. With DNC, there is no limit to the size of your CNC programs, except the space on the floppy disk. The programs are directly executed by the control as they are sent from the floppy drive.

If you wish to use DNC, it is enabled by Parameter 57 and Setting 55. When enabled, DNC is selected by entering the floppy file name, and pressing MDI a second time when already in MDI. (NOTE: Do not press MDI three consecutive times, or a "FLOPPY ABORT" will result). DNC mode will not be enabled unless there is a minimum of 512 bytes of user memory available. When DNC is selected, the PROGRAM DISPLAY will show:

WAITING FOR DNC...

This means that no DNC data has been received yet and you may begin sending data. You must start sending

Part Program Input / Output

the program to the control before the START button can be pushed. After the beginning of the program is seen by the control, the display will show part of the program and a message at the bottom, left of the CRT will show DNC PROG FOUND. After the program is found, you may push CYCLE START just like running any other program from Memory.

If you try to press START before receiving a program, you will get the message: NO DNC PROG YET. The reason for not allowing the command of START before receiving the DNC program is for safety. If the operation is allowed to start from a remote location, the operator may not be present to ensure that the machine is operating safely.

While a DNC program is executing, you are not allowed to change modes. You must first press RESET to stop the program.

When the end of the DNC program is received, the message DNC END FOUND is displayed. When the DNC program is finished running, the PROGRAM DISPLAY will show the last few lines of the program. You must press RESET or exit the DNC mode before you can run any other programs. If you try to press START before RESET of the previous DNC, you will get the message: RESET FIRST.

There are several restrictions on what can be in a DNC program. An M98 **Pnnn** may not be programmed to jump to another part of your program. An M30 is not allowed as it is not possible to start over at the beginning. The program must begin with a % just like any other program sent from the floppy disk and the program must end with a %.

When DNC is running, BACKGROUND EDIT is not available.

II. PROGRAMMING

The definition of a part program for any CNC consists of movements of the tool and speed changes to the tool RPM. It also contains auxiliary command functions such as tool changes, coolant on or off commands, or external **M** code commands.

Tool movements consist of rapid positioning commands, straight line movement of the tool at a controlled speed, and movement along an arc.

This machine has three (3) linear axes named **X**, **Y**, and **Z**. The **X**-axis moves the table left and right, the **Y**-axis moves it to and from the operator, and the **Z** moves the milling head up and down. The machine zero position is where the tool is at the right corner of the mill table farthest away from the front doors. Motion in the **X**-axis will move the table to the right for negative numbers and to the left for positive numbers. Motion in the **Y**-axis will move the table away from the operator for negative numbers and toward the operator for positive numbers. Motion in the **Z**-axis will move the tool down for negative numbers and up for positive numbers.

The optional fourth, or rotary, axis can be programmed for both rapid positioning commands and for feed commands either by itself or in conjunction with the other axes.

In addition to the above, there may be up to five, external, axes that can be programmed for rapid or feed motions, but one axis at a time only.

1. PROGRAM STRUCTURE

A CNC part program consists of one or more blocks of commands. When viewing the program, a block is the same as a line of text. Blocks shown on the CRT are always terminated by the ";" symbol which is called an EOB. Blocks are made up of alphabetical address codes and the "/" symbol. Address codes are always an alphabetical character followed by a numeric value. For instance, the specification of the position to move the **X**-axis would be a number preceded by the **X** symbol.

The "/" symbol, sometimes called a slash, is used to define an optional block. A block that contains this symbol can be optionally deleted with the BLKDEL button when running a program.

There is no positional requirement for the address codes. They may be placed in any order within the block. The following is a sample program as it would appear on the CRT. The words following the "." are not part of the program but are put here as further explanation.

This program will drill four holes and mill a two-inch hole in a four-inch square plate with **X** and **Y** zero at the center. The program with comment statements would appear like this.

%	:PROGRAM MUST BEGIN AND END WITH %
O1234 (OP1 SAMPLE MILL PART)	:PROGRAM # AND COMMENT STATEMENT
N1 (TOOL #1 IS A 1/2 INCH STUB DRILL)	:(*****) NOTES TO OPERATOR
N5 G40 G49 T#1 M06	:
N100 G00 X0 Y0 Z.5 G43 H1 M3 S1400 T2	:RAPID TO POS, OFFSET 1, SPIN FWD
N101 G01 Z.2 F30.	:FEED 30 INCH/MINUTE TO Z DEPTH
N102 G83 G98 Z-.625 R.03 Q.2 F5.	:PECK TO Z-.625 START .03 ABOVE
N103 X1.5 Y1.5	:DRILL ANOTHER HOLE AT NEW X,Y
N104 Y-1.5	:DRILL 3RD HOLE, PECK DEPTH IS .20
N105 X-1.5	:DRILL FOURTH HOLE

N106 Y1.5	:DRILL FIFTH HOLE
N107 G00 G80 Z.5	:CANCEL CANNED CYCLE
N108 T2 M06	:TOOL CHANGE TO TOOL #2
N2 (T #2 IS 5/8 90 DEG. COUNTERSINK)	:N### ARE LINE NUMBERS
N200 G00 X0 Y0 Z.5 G43 H2 M3 S500	:OFFSET 2, SPINDLE SPEED 500 RPM
N201 G01 Z.2 F30.	:FEED TO Z AT 30 INCH PER MINUTE
N202 G82 G98 Z-.27 R.0 F5.	:SPOT DRILL CYCLE, DRILL AT X0 Y0
N203 X1.5 Y1.5	:SEC HOLE R=START PLANE ABOVE ZERO
N204 Y-1.5	:3RD HOLE G98=RETURN TO INIT POINT
N205 X-1.5	:FOURTH HOLE
N206 Y1.5	:FIFTH HOLE
N207 G00 G80 Z.5	:RAPID TO Z.5
N208 G28 X0 Y0 Z2.0	:ZERO RETURN AFTER MOVE TO X0, Y0
N209 T#3 M	:TOOL CHANGE
N3 (TOOL #3 IS A 1/2 END MILL)	:N #'S ARE FOR YOUR CONVENIENCE
(SET DIAMETER VALUE TOOL #3)	:COMMENTS ARE IGNORED BY CONTROL
N300 G00 X0 Y0 Z.5 G43 H3 M3 S1000	:G43 = OFFSET Z IN MINUS DIRECTION
N301 G01 Z.2 F30.	:G01 CAN BE SPECIFIED AS G1
N302 Z-.625 F5.	:FEED TO DEPTH
N303 G01 G41 X-1.00	:COMPENSATE CUTTER LEFT OF LINE
N304 G03 I1.0 D1	:CUT CIRCLE CCW WITH TOOL DIA D1
N305 G00 G40 X00	:RAPID TO CENTER, G40 CANCELS COMP
N306 G00 Z.5	:RAPID OUT OF PART
N307 G28	:ZERO RETURN, Z GOES FIRST THAN X,Y
M30	:RESET PROGRAM TO BEGINNING
%	:END OF TAPE

Please note that each tool has some slight variations. This is done to show the flexibility of the control. For example, to change tools, all that is needed is an M06 even without a G28 in the previous line. Also, a G28 can be specified as G28 X0 Y0 Z0 or simply as G28. A "T" command can be put in with the M06 or it can be specified earlier in the program. This gives the maximum compatibility with other controls.

More than one program can be stored in the memory of the CNC. Every program stored has an **O** address code to define the number of that program. Those numbers are used to identify the program for selection as the main program being run or as a subprogram called from a main program.

2. ALPHABETICAL ADDRESS CODES

The following is a list of the Address Codes used in programming the CNC.

A FOURTH AXIS ROTARY MOTION

The **A** address character is used to specify motion for the optional fourth, **A**, axis. It specifies an angle in degrees for the rotary axis. It is always followed by a signed number and up to three fractional decimal positions. If no decimal point is entered, the last digit is assumed to be 1/1000 degrees. The smallest magnitude is 0.001 degrees, the most negative value is -8380.000 degrees, and the largest number is 8380.000 degrees.

B FIFTH AXIS ROTARY MOTION

The **B** address character is used to specify motion for the optional fifth, **B**, axis. It specifies an angle in degrees for the rotary axis. It is always followed by a signed number and up to three fractional decimal positions. If no decimal point is entered, the last digit is assumed to be 1/1000 degrees. The smallest

magnitude is 0.001 degrees, the most negative value is -8380.000 degrees, and the largest number is 8380.000 degrees.

C AUXILIARY EXTERNAL ROTARY AXIS

The **C** address character is used to specify motion for the optional external sixth, **C**, axis. It specifies an angle in degrees for the rotary axis. It is always followed by a signed number and up to three fractional decimal positions. If no decimal point is entered, the last digit is assumed to be 1/1000 degrees. The smallest magnitude is 0.001 degrees, the most negative value is -8380.000 degrees, and the largest number is 8380.000 degrees.

D TOOL DIAMETER SELECTION

The **D** address character is used to select the tool diameter or radius used for cutter compensation. The number following must be between 0 and 100. **D0** specifies that the tool size is zero and serves to cancel a previous **Dn**. Any other value of **D** selects the numbered entry from the tool diameter/radius list under the Offsets display.

E CONTOURING ACCURACY

The **E** address character is used, with **G187**, to select the accuracy required when cutting a corner during high speed machining operations. The range of values possible for the **E** code is 0.0001 to 0.25. Refer to the "Contouring Accuracy" section for more information.

F FEED RATE

The **F** address character is used to select the feed rate applied to any interpolation functions, including pocket milling and canned cycles. It is either in inches per minute with four fractional positions or mm per minute with three fractional positions.

G PREPARATORY FUNCTIONS (G CODES)

The **G** address character is used to specify the type of operation to occur in the block containing the **G** code. The **G** is followed by a two or three digit number between 0 and 187. Each **G** code defined in this control is part of a group of **G** codes. The Group 0 codes are non-modal; that is, they specify a function applicable to this block only and do not effect other blocks. The other groups are modal and the specification of one code in the group cancels the previous code applicable from that group. A modal **G** code applies to all subsequent blocks so those blocks do not need to re-specify the same **G** code. More than one **G** code can be placed in a block in order to specify all of the setup conditions for an operation. See "G Codes" section.

H TOOL LENGTH OFFSET SELECTION

The **H** address character is used to select the tool length offset entry from the offsets memory. The **H** is followed by a two digit number between 0 and 100. **H0** will cause no offset to be used and **Hn** will use the tool length entry **n** from the Offsets display. Note that **G49** is the default condition disabling tool length offsets; so you must also select either **G43** or **G44** for tool offsets to work. The **TOOL OFFSET MESUR** button will enter a value into the offsets to correspond to the use of **G43**.

I CANNED CYCLE AND CIRCULAR OPTIONAL DATA

The **I** address character is used to specify data used for some canned cycles and circular motions. It is either in inches with four fractional positions or mm with three fractional positions. It is followed by a signed number in inches between -8380.000 and 8380.000 for inches or between -83800.00 and 83800.00 for metric.

J CANNED CYCLE AND CIRCULAR OPTIONAL DATA

The **J** address character is used to specify data used for some canned cycles and circular motions. It is formatted just like the **I** data.

K CANNED CYCLE AND CIRCULAR OPTIONAL DATA

The **K** address character is used to specify data used for some canned cycles and circular motions. It is formatted just like the **I** data.

L LOOP COUNT FOR REPEATED CYCLES

The **L** address character is used to specify a repetition count for some canned cycles and auxiliary functions. It is followed by an unsigned number between 0 and 32767.

M M CODE MISCELLANEOUS FUNCTIONS

The **M** address character is used to specify an **M** code for a block. These codes are used to control miscellaneous machine functions. Note that only one **M** code is allowed per block of the CNC program and all **M** codes are performed at the end of the block. See the "M Codes" section for a detailed list of **M** codes.

N NUMBER OF BLOCK

The **N** address character is entirely optional. It can be used to identify or number each block of a program. It is followed by a number between 0 and 99999. The M97 functions may reference an **N** line number.

O PROGRAM NUMBER/NAME

The **O** address character is used to identify a program. It is followed by a number between 0 and 9999. A program saved in memory always has a **O** identification in the first block; it cannot be deleted. Altering the **O** in the first block causes the program to be renamed. An **O** can be placed in other blocks of a program but will have no effect and can be confusing to the reader. A colon (:) may be used in the place of **O**, but is always displayed as "0".

P DELAY TIME OR PROGRAM NUMBER

The **P** address character is used to enter either a time in seconds or a program number for a subroutine call. If it is used as a time (for a G04 dwell) or a program name (for a M98), the value may be either a positive number without decimal point up to 9999. If it is used as a time, it may be a positive decimal with fraction between 0.001 and 1000.0.

Q CANNED CYCLE OPTIONAL DATA

The **Q** address character is used in canned cycles and is followed by a signed number in inches between -8380.000 and 8380.000 for inches or between -83800.00 and 83800.00 for metric.

R CANNED CYCLE AND CIRCULAR OPTIONAL DATA

The **R** address character is used in canned cycles and circular interpolation. It is either in **inches** with four fractional positions or **mm** with three fractional positions. It is followed by a signed number in inches between -8380.000 and 8380.000 for inches or between -83800.00 and 83800.00 for metric. It is usually used to define the reference plane for canned cycles.

S SPINDLE SPEED COMMAND

The **S** address character is used to specify the spindle speed in conjunction with M41 and M42. The **S** is followed by an unsigned number between 1 - 99999. The **S** command does not turn the spindle on or off; it only sets the desired speed. If a gear change is required in order to set the commanded speed, this command will cause a gear change to occur even if the spindle is stopped. If the spindle is running, a gear change operation will occur and the spindle will continue running at the new speed.

T TOOL SELECTION CODE

The **T** address character is used to select the tool for the next tool change. The number following must be a positive number between 1 and the number in Parameter 65. It does not cause the tool change operation to occur. The **Tn** may be placed in the same block that starts the tool change (M6 or M16) or in any previous block.

U AUXILIARY EXTERNAL LINEAR AXIS

The **U** address character is used to specify motion for the optional external linear, **U**, axis. It specifies a position of motion in inches. It is always followed by a signed number and up to four fractional decimal positions. If no decimal point is entered, the last digit is assumed to be 1/10000 inches. The smallest magnitude is 0.0001 inches, the most negative value is -838.0000 inches, and the largest number is 838.0000 inches.

V AUXILIARY EXTERNAL LINEAR AXIS

The **V** address character is used to specify motion for the optional external linear, **V**, axis. It specifies a position of motion in inches. It is always followed by a signed number and up to four fractional decimal positions. If no decimal point is entered, the last digit is assumed to be 1/10000 inches. The smallest magnitude is 0.0001 inches, the most negative value is -838.0000 inches, and the largest number is 838.0000 inches.

W AUXILIARY EXTERNAL LINEAR AXIS

The **W** address character is used to specify motion for the optional external linear, **W**, axis. It specifies a position of motion in inches. It is always followed by a signed number and up to four fractional decimal positions. If no decimal point is entered, the last digit is assumed to be 1/10000 inches. The smallest magnitude is 0.0001 inches, the most negative value is -838.0000 inches, and the largest number is 838.0000 inches.

X LINEAR X-AXIS MOTION

The **X** address character is used to specify motion for the X-axis. It specifies a position or distance along the X-axis. It is either in **inches** with four fractional positions or **mm** with three fractional positions. It is followed by a signed number in inches between -8380.000 and 8380.000 for inches or between -83800.00 and 83800.00 for metric. If no decimal point is entered, the last digit is assumed to be 1/10000 inches or 1/1000 mm.

Y LINEAR Y-AXIS MOTION

The **Y** address character is used to specify motion for the Y-axis. It specifies a position or distance along the Y-axis. It is either in **inches** with four fractional positions or **mm** with three fractional positions. It is followed by a signed number in inches between -8380.000 and 8380.000 for inches or between -83800.00 and 83800.00 for metric. If no decimal point is entered, the last digit is assumed to be 1/10000 inches or 1/1000 mm.

Z LINEAR Z-AXIS MOTION

The Z address character is used to specify motion for the Z-axis. It specifies a position or distance along the Z-axis. It is either in **inches** with four fractional positions or **mm** with three fractional positions. It is followed by a signed number in inches between -8380.000 and 8380.000 for inches or between -83800.00 and 83800.00 for metric. If no decimal point is entered, the last digit is assumed to be 1/10000 inches or 1/1000 mm.

3. PREPARATORY FUNCTIONS (G CODES)

The following is a summary of the G codes. A "*" indicates the default within each group, if there is one:

Code:	Group:	Function:	Description On Page:
G00	*01	Rapid Motion	43
G01	01	Linear Interpolation Motion	43
G02	01	CW Interpolation Motion	43
G03	01	CCW Interpolation Motion	45
G04	00	Dwell	45
G09	00	Exact Stop	45
G10	00	Programmable Offset Setting	45
G12	00	CW Circular Pock Milling (Yasnac)	46
G13	00	CCW Circular Pock Milling (Yasnac)	46
G17	*02	XY Plane Selection	47
G18	02	ZX Plane Selection	48
G19	02	YZ Plane Selection	48
G20	06	Inch Programming Selection	48
G21	06	Metric Programming Selection	48
G28	00	Return To Machine Zero	49
G29	00	Move to Location Through G29 Reference	49
G31	00	Skip Function	49
G35	00	Automatic Tool Diameter Measurement	49
G36	00	Automatic Work Offset Measurement	50
G37	00	Automatic Tool Length Measurement	49
G40	*07	Cutter Comp Cancel	51
G41	07	Cutter Compensation Left	51
G42	07	Cutter Compensation Right	51
G43	08	Tool Length Compensation +	51
G44	08	Tool Length Compensation -	51
G47	00	Engraving	51
G49	*08	G43/G44 Cancel	51
G50	11	G51 Cancel	53
G51	11	Scaling	54
G52	12	Select Work Coordinate System G52 (Yasnac)	58
G52	00	Set Local Coordinate System (Fanuc)	58
G52	00	Set Local Coordinate System (HAAS)	58
G53	00	Non-Modal Machine Coordinate Selection	58
G54	*12	Select Work Coordinate System 1	58
G55	12	Select Work Coordinate System 2	58
G56	12	Select Work Coordinate System 3	58
G57	12	Select Work Coordinate System 4	58
G58	12	Select Work Coordinate System 5	58
G59	12	Select Work Coordinate System 6	58
G60	00	Unidirectional Positioning	58
G61	13	Exact Stop Modal	59
G64	*13	G61 Cancel	59
G65	00	Macro Subroutine Call	139
G68	16	Rotation	55
G69	16	G68 Cancel	58
G70	00	Bolt Hole Circle (Yasnac)	59
G71	00	Bolt Hole Arc (Yasnac)	59
G72	00	Bolt Holes Along an Angle (Yasnac)	59

G CODES

G73	09	High Speed Peck Drill Canned Cycle	63
G74	09	Reverse Tap Canned Cycle	64
G76	09	Fine Boring Canned Cycle	65
G77	09	Back Bore Canned Cycle	65
G80	*09	Canned Cycle Cancel	66
G81	09	Drill Canned Cycle	66
G82	09	Spot Drill Canned Cycle	67
G83	09	Peck Drill Canned Cycle	68
G84	09	Tapping Canned Cycle	70
G85	09	Boring Canned Cycle	70
G86	09	Bore/Stop Canned Cycle	71
G87	09	Bore/Manual Retract Canned Cycle	72
G88	09	Bore/Dwell Canned Cycle	72
G89	09	Bore Canned Cycle	73
G90	*03	Absolute	74
G91	03	Incremental	74
G92	00	Set Work Coordinates - FANUC or HAAS	74
G92	00	Set Work Coordinates - YASNAC	74
G98	*10	Initial Point Return	74
G99	10	R Plane Return	74
G100	00	Disable Mirror Image	75
G101	00	Enable Mirror Image	75
G102	00	Programmable Output To RS-232	75
G103	00	Block Lookahead Limit	76
G110	12	Select Work Coordinate System 7	76
G111	12	Select Work Coordinate System 8	76
G112	12	Select Work Coordinate System 9	76
G113	12	Select Work Coordinate System 10	76
G114	12	Select Work Coordinate System 11	76
G115	12	Select Work Coordinate System 12	76
G116	12	Select Work Coordinate System 13	76
G117	12	Select Work Coordinate System 14	76
G118	12	Select Work Coordinate System 15	76
G119	12	Select Work Coordinate System 16	76
G120	12	Select Work Coordinate System 17	76
G121	12	Select Work Coordinate System 18	76
G122	12	Select Work Coordinate System 19	76
G123	12	Select Work Coordinate System 20	76
G124	12	Select Work Coordinate System 21	76
G125	12	Select Work Coordinate System 22	76
G126	12	Select Work Coordinate System 23	76
G127	12	Select Work Coordinate System 24	76
G128	12	Select Work Coordinate System 25	76
G129	12	Select Work Coordinate System 26	76
G136	00	Automatic Work Offset Center Measurement	50
G150	00	General Purpose Pocket Milling	77
G187	00	Accuracy Control for High Speed Machining	78

Each **G** code defined in this control is part of a group of **G** codes. The Group 0 codes are non-modal; that is, they specify a function applicable to this block only and do not affect other blocks. The other groups are modal and the specification of one code in the group cancels the previous code applicable from that group. A modal **G** code applies to all subsequent blocks so those blocks do not need to re-specify the same **G** code.

There is also one case where the Group 01 **G** codes will cancel the Group 9 (canned cycles) codes. If a canned cycle is active (G73 through G89), the use of G00 or G01 will cancel the canned cycle.

3.1 RAPID POSITION COMMANDS

G00 RAPID MOTION POSITIONING

GROUP 01

- X Optional X-axis motion command
- Y Optional Y-axis motion command
- Z Optional Z-axis motion command
- A Optional A axis motion command

This **G** code is used to cause a rapid traverse of the three or four axes of the machine. The auxiliary axes **B**, **C**, **U**, **V**, and **W** can also be moved with a G00. This **G** code is modal so that a previous block with G00 causes all following blocks to be rapid motions until another Group 01 code is specified. The rapid traverse rate is dependent on the maximum speed possible for each axis independently as modified by the RAPID override keys.

Generally, rapid motions will not be in straight lines. All of the axes specified are moved at the same time but will not necessarily complete their motions at the same time. The control will wait until all motions are complete. Only the axes specified are moved and the incremental or absolute modal conditions (G90 or G91) will change how those values are interpreted. Parameter 57 can change how closely the machine waits for a precise stop before and after a rapid move.

3.2 INTERPOLATION COMMANDS

G01 LINEAR INTERPOLATION MOTION

GROUP 01

- F Feed rate in inches (mm) per minute
- X Optional X-axis motion command
- Y Optional Y-axis motion command
- Z Optional Z-axis motion command
- A Optional A axis motion command

This **G** code provides for straight line (linear) motion from point to point. Motion can occur in 1, 2 or 3 dimensions. All axes will start and finish motion at the same time. The rotary axis may also be commanded and this will provide a helical motion. The speed of all axes is controlled so that the feed rate specified is achieved along the actual path. Rotary axis speed is dependent on the rotary axis diameter setting (Setting 34) and will provide a helical motion. The **F** command is modal and may be specified in a previous block. Only the axes specified are moved and the incremental or absolute modal conditions (G90 or G91) will change how those values are interpreted. The auxiliary axes **B**, **C**, **U**, **V**, and **W** can also be moved with a G01 but only one axis is moved at a time.

G02 CW CIRCULAR INTERPOLATION MOTION

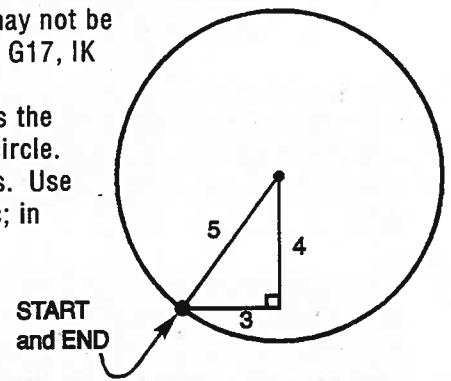
GROUP 01

- F Feed rate in inches (mm) per minute
- I Optional distance along X-axis to center of circle
- J Optional distance along Y-axis to center of circle

K	Optional distance along Z-axis to center of circle
R	Optional radius of circle
X	Optional X-axis motion command
Y	Optional Y-axis motion command
Z	Optional Z-axis motion command
A	Optional A axis motion command

This G code is used to specify a clockwise circular motion of two of the linear axes. Circular motion is possible in any two of X, Y, and Z axes as selected by G17, G18, and G19. The X, Y, and Z are used to specify the end point of the motion that can use either absolute (G90) or incremental (G91) motion. If any of the X, Y, or Z for the selected plane is not specified, the endpoint of the arc is the same as the starting point for that axis. There are two ways to specify the center of the circular motion; the first uses I, J, or K to specify the distance from the starting point to the center of the arc; the second uses R to specify the radius of the arc. These are further described below:

I, J, K: When I, J, or K are used to specify the center of the arc, R may not be used. Only the I, J, or K specific to the selected plane (IJ for G17, IK for G18, JK for G19) are allowed. If only one of the I, J, K is specified, the others are assumed to be zero. The I, J, or K is the signed distance from the starting point to the center of the circle. Small errors in these values are tolerated up to 0.0010 inches. Use of I, J, or K is the only way to cut a complete 360 degree arc; in this case, the starting point is the same as the ending point and no X, Y, or Z is needed.

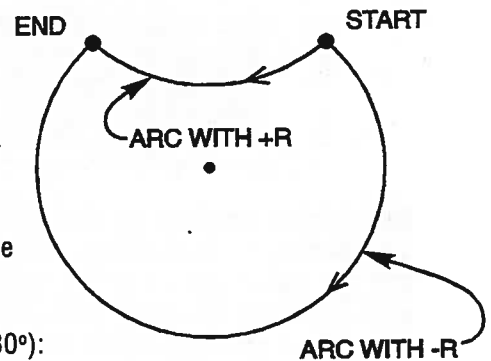


To cut a complete circle of 360 degrees (360°), you do not need to specify an ending point X, Y, or Z; just program I, J, or K to define the center of the circle. The following line will cut a complete circle:

G02 I3.0 J4.0 (Assumes G17; XY plane)

In cases where you are cutting less than a complete circle, it is much easier to use R instead of I, J, K.

R: When R is used to specify the center of the circle, a complete 360 degree arc is not possible. X, Y, or Z is required to specify an endpoint different from the starting point. R is the distance from the starting point to the center of the circle. With a positive R, the control will generate a path of 180 degrees or less; to generate an angle of over 180 degrees, specify a negative R. Small errors in this value are tolerated up to 0.0010 inches.



The following line will cut an arc less than 180 degrees (180°):

G01 X3.0 Y4.0
G02 X-3.0 R5.0

and the following line will cut an arc of more than 180 degrees (180°):

G01 X3.0 Y4.0
G02 X-3.0 R-5.0

G03 CCW CIRCULAR INTERPOLATION MOTION**GROUP 01**

G03 will generate counterclockwise circular motion but is otherwise the same as G02.

HELICAL

A helical motion is possible with G02 or G03 by programming the linear axis that is not in the selected plane. This third axis will be interpolated along the specified axis in a linear manner while the other two axes will be moved in the circular motion. The speed of each axis will be controlled so that the helical rate matches the programmed feed rate.

The length of the third axis motion may not be greater than the length of the motion of the two axes for the circular motion. This means that for a complete revolution around a one-inch diameter, the circumference will be 3.1416 and the third axis motion may not be more than 3.1416 inches.

3.3 MISCELLANEOUS G CODES**G04 DWELL****GROUP 00**

P The dwell time in seconds or milliseconds

G04 is used to cause a delay or dwell in the program. The block containing G04 will delay for the time specified in the P code. If the P has no fraction part, the delay is in milliseconds (0.001 seconds); otherwise the delay is in seconds.

G09 EXACT STOP**GROUP 00**

The G09 code is used to specify exact stop. It is not modal and does not affect the following blocks. Rapid and interpolated moves will decelerate to an exact stop before another block is processed. In exact stop, moves will take a longer time and continuous cutter motion will not occur. This may cause deeper cutting where the tool stops.

3.4 PROGRAMMABLE OFFSET SETTING**G10 PROGRAMMABLE SETTING OF TOOL OFFSETS****GROUP 00**

- L Selection of length, length wear, diameter, diameter wear, or work coordinates.
- P Selection of offset number.
- R Offset value or increment for length and diameter.
- X Optional X-axis zero location.
- Y Optional Y-axis zero location.
- Z Optional Z-axis zero location.
- A Optional A-axis zero location.

G10 can be used to change the tool length and work offsets from inside of a program. The following codes are used for selection of offsets:

- L2 Work coordinate origin for G52 and G54-G59
- L10 Length offset amount (for H code)
- L1 or L11 Tool wear offset amount (for H code)
- L12 Diameter offset amount (for D code)
- L13 Diameter wear offset amount (for D code)
- L20 Auxiliary work coordinate origin for G110-G129

The P code is used to index the appropriate offsets.

P1-P100	Used to reference D or H code offsets,	L10-L13
P0	G52 references work coordinate	L2
P1-P6	G54-G59 references work coordinates	L2
P1-P20	G110-G129 references auxiliary coordinates	L20

The R, X, Y, Z, and A codes are signed numbers with fractions in inches (or MM). The R, X, Y, Z, and A values are absolute or incremental, depending on the current G90/G91 modal value.

G10 Examples:

G10	L2	P1	G91	X6.0		{Move coordinate G54 6.0 to the right.};
G10	L20	P2	G90	X10.	Y8.	{Set work coordinate G111 to X10.0 ,Y8.0};
G10	L10	G90	P5	R2.5		{Set offset for Tool #5 to 2.5.};
G10	L12	G90	P5	R.375		{Set diameter for Tool #5 to 3/8ths.};

3.5 CIRCULAR POCKET MILLING

There are two G codes that will provide for pocket milling of a circular shape. They are different only in which direction of rotation is used.

G12 CIRCULAR POCKET MILLING CLOCKWISE

GROUP 00

- *D Tool Radius Or Diameter Selection
- I Radius Of First Circle (Or Finish If No K)
- K Radius Of Finished Circle (If Specified)
- L Loop count for repeating deeper cuts
- Q Radius Increment (Must Be Used With K)
- F Feed Rate in inches (mm) per minute
- Z Z depth of cut or increment

***In order to get the exact programmed circle diameter, the control uses the selected D code tool size. If this compensation is not desired, program D0.**

This G Code implies the use of G42.

The tool must be positioned at the center of the circle either in a previous block or in this block using X and Y. The cut is performed entirely with circular motions of varying radius. G12 belongs to Group zero and thus is non-modal. If G91 (incremental) is specified and an L count is included, the Z increment is repeated L times at the F feed rate. If no K is specified, the center of the cut is removed completely.

G13 CIRCULAR POCKET MILLING COUNTERCLOCKWISE

GROUP 00

This G Code implies the use of G41 and is otherwise similar to G12. G13 belongs to Group 00 and thus is non-modal.

%

```

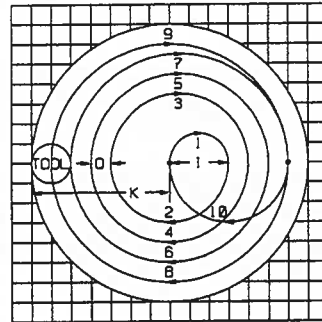
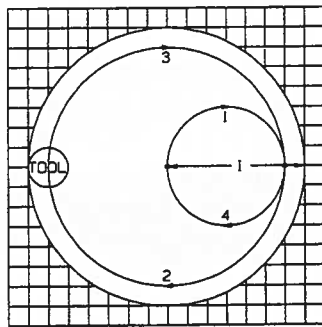
O0100 (SAMPLE G12 AND G13)
(OFFSET D01 SET TO APPROX. TOOL SIZE)
(TOOL MUST BE MORE THAN 0.3 IN DIAM.)
G54 G00 G90 Z-1. X0. Y0.
S2000 M03
G12 I1.5 F10. Z-1.2 D01
G28

```

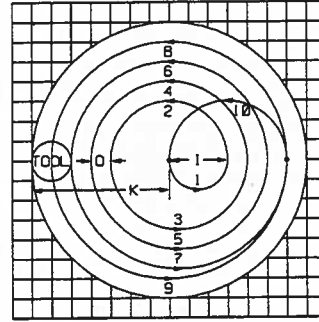
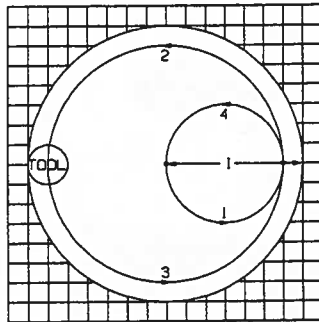
G55 Z-1. X0. Y0.
 G12 I0.3 K1.5 Q0.3 F10. Z-1.2 D01
 G28
 G56 Z-1. X0. Y0.
 G13 I1.5 F10. Z-1.2 D01
 G28
 G57 Z-1. X0. Y0.
 G13 I0.3 K1.5 Q0.3 F10. Z-1.2 D01
 G28 M30
 %

CIRCULAR POCKET MILLING

G12



G13



I ONLY

I, K, AND Q

3.6 CIRCULAR PLANE SELECTION

The plane used for circular motions must be comprised of two of X, Y, or Z. There are three G codes used to select the plane; G17 for XY, G18 for XZ, and G19 for YZ.

G17 XY PLANE SELECTION

GROUP 02

The G17 code is used to select the XY plane for circular motion. It is modal and applies to all following circular motions until another Group 02 is found. In this plane, circular motion is defined as clockwise for the operator looking down onto the X-Y table from above. This is the motion of the tool relative to the table.

G18 ZX PLANE SELECTION**GROUP 02**

The G18 code is used to select the ZX plane for circular motion. It is modal and applies to all following circular motions until another Group 02 is found. In the X-Z plane (G18), circular motion is defined as clockwise for the operator looking towards the rear of the machine from the front control panel.

G19 YZ PLANE SELECTION**GROUP 02**

The G19 code is used to select the YZ plane for circular motion. It is modal and applies to all following circular motions until another Group 02 is found. In the Y-Z plane (G19), circular motion is defined as clockwise for the operator looking across the table from the side of the machine the control panel is mounted.

Plane selection applies also to G12 and G13 circular pocket milling which must always be in the X-Y plane (G17).

The default plane selection when the machine is powered on is G17 for the X-Y plane. This means that a circular motion in the plane of the X-Y table may be programmed without first selecting G17.

A helical motion is possible with G02 or G03 by programming the linear axis which is not in the selected plane. This third axis will be interpolated along the specified axis in a linear manner while the other two axes will be moved in the circular motion. The speed of each axis will be controlled so that the helical rate matches the programmed feed rate.

If cutter radius compensation is selected (G41 or G42), you may only use the X-Y plane for circular motions (G17). Cutter radius compensation is only available in the X and Y axes.

3.7 INCH / METRIC SELECTION (G20, G21)**G20 INCH PROGRAMMING SELECTION****GROUP 06****G21 METRIC PROGRAMMING SELECTION****GROUP 06**

Selection between inch and metric programming can only be done from the Setting 9. Inch programming allows displacements up to 838 inches and a resolution of 0.0001 inches. Metric programming uses millimeter units with a maximum displacement of 8380 mm and a resolution of 0.001 mm.

When in metric, the feed rate is also defined as millimeters per minute with a range of 1000. to 0.001 mm/min.

When jogging in metric, the speeds and units on the keypad are interpreted as mm/min but the value used is ten times larger than shown on the keypad.

The optional fourth or fifth axis programming is not effected by the selection of metric. It is always programmed in degrees. The auxiliary C axis is also always in degrees.

Changing the setting from inches to metric or back again will change the content of any programs already stored in memory. You must reload your programs with metric values after changing this setting.

The standard G codes G20 and G21 are sometimes used to select between inch and metric BUT, in this control, the G20 (inch) and G21 (mm) codes can only be used to ensure that the inch/metric setting is set correctly for that program.

3.8 REFERENCE POINT DEFINITION AND RETURN**G28 RETURN TO MACHINE ZERO, SET OPTIONAL G29 REFERENCE POINT****GROUP 00**

The G28 code is used to return to the machine zero position on all axes. If an X, Y, Z, or A axis is specified on the same block, only those axes will move and return to the machines' zero point. If X, Y, Z, or A specifies a different location than the current position, then the movement to machine zero will be through the specified point. This point is called the G29 reference point and is saved for use in G29. If no X, Y, Z, or A is specified, all axes will be moved directly to machine zero. Any auxiliary axes (B, C,...) are returned to home after the X, Y, Z, and A axes. G28 also cancels tool length offsets.

G29 MOVE TO LOCATION THROUGH G29 REFERENCE POINT**GROUP 00**

The G29 code is used to move the axes to a commanded X, Y, Z, or A position. The axes that are selected in this block are moved to the G29 reference point saved or recorded in G28 and then moved to X,Y,Z or A specified in the G29 command point. The positions are interpreted in the current coordinate system.

3.9 SKIP FUNCTION (G31)**G31 SKIP FUNCTION****GROUP 00**

- F Feed rate in inches (mm) per minute
- X Optional X-axis motion command
- Y Optional Y-axis motion command
- Z Optional Z-axis motion command
- A Optional A-axis motion command

The skip function is a non-modal operation that causes a linear move to the specified X, Y, Z, and A position. It applies only to the block in which G31 is specified. A feed rate must be defined previously or in this block. The specified move is started and continues until the end point or the skip signal. The skip signal is a discrete input that usually indicates that the end of travel has been reached; this is usually a probe. Cutter compensation may not be active during a skip function. M78 or M79 may be used to test if the skip signal was received.

An M75 can be used to mark the probed point as the reference point for G35 or G136.

3.10 AUTOMATIC TOOL MEASUREMENT (G35, G37)**G35 AUTOMATIC TOOL DIAMETER MEASUREMENT****GROUP 00****G37 AUTOMATIC TOOL LENGTH MEASUREMENT****GROUP 00**

- F Feed rate in inches (mm) per minute
- D Tool diameter offset number (G35)
- H Tool offset number (G37)
- Z Required Z-axis offset

The automatic tool length measurement operation (G37) is a non-modal operation that causes a linear move of the Z-axis until the skip signal is received or the end of Z travel limits. A nonzero H code must be active, G43 or G44 must be active, a Z value must be specified, and a feed rate must be defined. No X, Y, or A code is allowed. When the move is terminated, the specified Z and the final Z positions are used to set the specified (Hnn) tool offset. The active coordinate system is taken into account.

The coordinate system (G54..G59, G110..G129) and tool length offset (H01..H100) may be selected in this block or in a previous block. The end point of the Z move is controlled only by the maximum travel limits defined for the machine.

The resulting tool offset value is such that a subsequent move to the Z value specified in the G37 will move the tool to the position where the skip signal was sensed. The skip signal is a discrete input that usually indicates that the end of travel has been reached; this is sometimes a probe. Cutter compensation may not be active during a skip function. M78 or M79 may be used to test if the skip signal was received. The resulting tool offset is the offset between the work zero and the point where the probe is touched.

The automatic tool diameter measurement function (G35) is used to set the tool diameter (or radius) using two different probe passes; one on each side of the tool. The first point is set with a G31 block using an M75 and the second point is set with the G35 block. The distance between these two points is set into the Dnn value active. A nonzero D code must be selected. Setting 63 is used to reduce this measurement by the width of the tool probe.

3.11 AUTOMATIC WORK OFFSET MEASUREMENT (G36 , G136)

G36 AUTOMATIC WORK OFFSET CENTER MEASUREMENT
G136 AUTOMATIC WORK OFFSET CENTER MEASUREMENT

GROUP 00
GROUP 00

F Feed rate in inches (mm) per minute
 I Optional offset distance along X-axis
 J Optional offset distance along Y-axis
 K Optional offset distance along Z-axis
 X Optional X-axis motion command
 Y Optional Y-axis motion command
 Z Optional Z-axis motion command
 A Optional A-axis motion command

The automatic work offset measurement operation is a non-modal operation that causes a linear move of the X, Y, Z, and A axes until the skip signal is received or the end of the programmed motion. The X, Y, Z, and A axes are moved to the programmed position in a linear move but will stop early if the skip signal is received. Tool offsets must not be active when this function is performed. M78 or M79 may be used to test if the skip signal was received. The currently active work coordinate system is set for each axis that is programmed. The point where the skip signal is received becomes the work zero position. The work coordinate system may be selected in this block or in a previous block.

The points probed are offset by the values in Settings 59 through 62.

A G36 will set the work coordinates to the point where the probe is hit. The G136 will set the work coordinates to a point at the center of a line between the probed point and the point set with M75. This allows the center of a part to be found using two separated probed points.

Note that the X, Y, Z, or A programmed into this block are interpreted in the coordinate system that is about to be set. Thus, the end point of the move will be interpreted in the old work coordinate value. For this reason, it is easier to program these moves as incremental (G91).

If an I, J, or K is specified, the appropriate axis work offset is shifted by the amount in the I, J, or K. This allows the work offset to be shifted some distance away from where the probe actually hits.

3.12 CUTTER COMPENSATION

G40 CUTTER COMP CANCEL **GROUP 07**

G40 will cancel the G41 or G42 cutter compensation. Programming a D00 will also cancel cutter compensation.

G41 CUTTER COMPENSATION LEFT **GROUP 07**

G41 will select cutter compensation left; that is the tool is moved to the left of the programmed path to compensate for the size of the tool. A **Dnn** must also be programmed to select the correct tool size from compensation memory. If compensation memory contains a negative value for cutter size, cutter compensation will operate as though G42 was specified.

G42 CUTTER COMPENSATION RIGHT **GROUP 07**

G42 will select cutter compensation right; that is the tool is moved to the right of the programmed path to compensate for the size of the tool. A **Dnn** must also be programmed to select the correct tool size from compensation memory. If compensation memory contains a negative value for cutter size, cutter compensation will operate as though G41 was specified.

Refer to the "Cutter Compensation" section for more information.

3.13 TOOL LENGTH COMPENSATION

G43 TOOL LENGTH COMPENSATION + (PLUS) **GROUP 08**

This code selects tool length compensation in a positive direction. That is; the tool length offsets are added to the commanded axis positions. A nonzero **Hnn** must be programmed to select the correct entry from offsets memory. The automatically entered offsets using the TOOL OFFSET MESUR key assume that G43 is being used.

G44 TOOL LENGTH COMPENSATION - (MINUS) **GROUP 08**

This code selects tool length compensation in a negative direction. That is; the tool length offsets are subtracted from the commanded axis positions. A nonzero **Hnn** must be programmed to select the correct entry from offsets memory.

G49 G43/G44 CANCEL **GROUP 08**

This **G** code cancels tool length compensation. Putting in a H0 will also cancel tool length compensation. G28, M30, and RESET will also cancel tool length compensation.

3.14 ENGRAVING

G47 ENGRAVING **GROUP 00**

- E = Plunge rate (units/min)
- F = Engraving feed rate (units/min)
- I = Angle of rotation (-360. to +360.), default is 0.
- J = Scaling factor in inches (minimum = 0.001 inches), default is 1.0 inch

P	=	1 for Sequential Serial Number Engraving 0 for Literal String Engraving
R	=	Return plane
X	=	X start of engraving
Y	=	Y start of engraving
Z	=	Depth of cut

The text to engrave should be in the form of a comment on the same line as G47, with either a P1 or P0 before it. P1 selects Sequential Serial Number Engraving and P0 selects Literal String Engraving.

SEQUENTIAL SERIAL NUMBER ENGRAVING

This method is used to engrave numbers on a series of parts, with the number being incremented by one each time. The '#' symbol is used to select the number of digits in the serial number. For example:


```
G47 P1 (####)
```

will limit the serial number to four digits.

The initial serial number can be either programmed or set manually. If it is programmed, for example:

```
G47 P1 (1234)
```

will set the initial serial number to "1234".

The initial serial number can also be set manually into a macro variable. The "MACROS" option does not have to be enabled to do this. Macro variables are temporary storage locations for numbers. Macro variable #599 is used to hold the initial serial number to be engraved. To set this variable, go to the CURNT COMDS page and press the PAGE DOWN key until the "Macro Variables" page appears. Then type in "599" and press the  (DOWN ARROW). Now enter the desired initial serial number at the cursor and press the WRITE key. For example, when macro variable #599 is set to "1234",

```
G47 P1 (####)
```

will produce this:

```
1234
```

If the number in macro variable #599 has more characters than specified in the format string, only the quantity specified will be printed. For example, if #599 is set to "12345" and only four places are specified in the format string, only "2345" will be engraved.

LITERAL STRING ENGRAVING

This method is used to engrave desired text on a part. The characters available for engraving are:

```
A..Z  
a..z  
0..9,  
! " # & ' ( ) * + , / : ; < = > ? [ \ ] ^ { }
```

However, programs downloaded through the serial port or the floppy drive can take advantage of characters not available on the mill keypad.

For Literal String Engraving, the text should be in the form of a comment on the same line as the P0 statement.

For example:

```
G47 P0 (ENGRAVE THIS)
```

will produce

ENGRAVE THIS

Note that P0 is used, instead of P1, for Literal String Engraving.

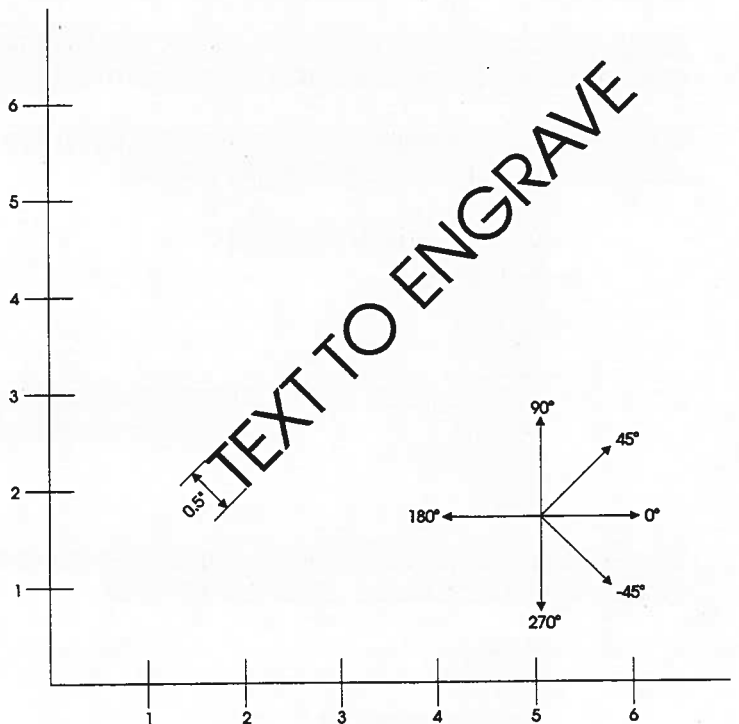
EXAMPLE

This example will create the figure below.

```
G47 P0 X2.0 Y2.0 I45. J.5 R.05 Z-.005 F15.0 E10.0 (TEXT TO ENGRAVE)
```

In this example:

- G47 P0 select Literal String Engraving
- X2.0 Y2.0 select 2.0, 2.0 as the starting point for the text.
- I45. places the text at a positive 45 degree angle
- J.5 sets the text height to 0.5 inch
- R.05 commands the cutter to retract to .05 inches (mm) above the cutting plane after engraving
- Z-.005 selects a .005 inch (mm) deep cut
- F15.0 selects an engraving feed rate of 15 units/min
- E10.0 commands the cutter to plunge at a rate of 10 units/min



3.15 COORDINATE ROTATION AND SCALING

This control function is optional. If you would like further information on installing this feature please call Haas Automation or your dealer for more information.

G50 CANCEL SCALING

GROUP 11

G code 50 cancels scaling on all axes. Any axis scaled by a previous G51 command is no longer in effect.

G51 SCALING

GROUP 11

X optional center of scaling for the X axis.
 Y optional center of scaling for the Y axis.
 Z optional center of scaling for the Z axis.
 P optional scaling factor for all axes. Three-place decimal .001 to 8383.000.

G51 [X...] [Y...] [Z...] [P...]

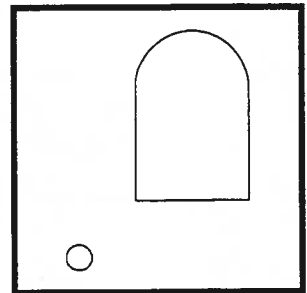
When scaling is invoked, all subsequent X, Y, Z, I, J, K, or R values pertaining to machine motion are multiplied by a scaling factor and are offset relative to a scaling center.

G51 is modal and modifies appropriate positional values in the blocks following the G51 command. It does not change or modify values in the block from which it is called. Axes X, Y, and Z are all scaled when the P code is used. If the P code is not used, the scaling factor currently in Setting 71 is used. The default scaling factor in Setting 71 is 1.0. A scaling factor of 1.0 means that no scaling is done. That is, all values are multiplied by 1.0 before being interpreted by the control.

A scaling center is always used by the control in determining the scaled position. If any scaling center is not specified in the G51 command block, then the current work coordinate position is used as the scaling center.

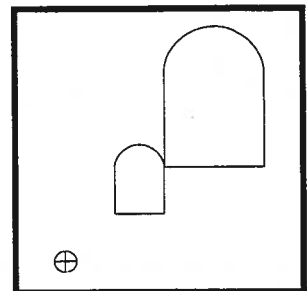
The following programs illustrate how scaling is performed when different scaling centers are used. All three examples call subroutine O0001 which follows.

```
O0001 (GOTHIC WINDOW) ;
F20. S500 ;
G00 X1. Y1. ;
G01 X2. ;
Y2. ;
G03 X1. R0.5      O = Work coordinate origin
G01 Y1. ;         + = Center of scaling
G00 X0 Y0 ;
M99 ;
```



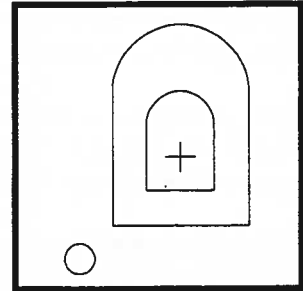
The first example illustrates how the control uses the current work coordinate location as a scaling center. Here, it is X0 Y0 Z0.

```
O0010 ;
G59 ;
G00 G90 X0 Y0 Z0 ;
M98 P1 ;
G51 P2. (scaling center is X0 Y0 Z0) ;
M98 P1 ;
M30 ;
```



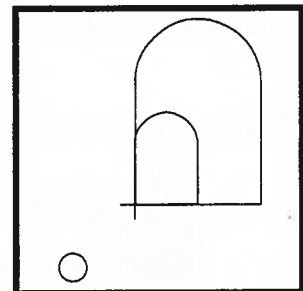
The next example specifies the center of the window as the scaling center.

```
O0011 ;
G59 ;
G00 G90 X0 Y0 Z0 ;
M98 P1 ;
G51 X1.5 Y1.5 P2. ;
M98 P1 ;
M30 ;
```



The last example illustrates how scaling can be placed at the edge of tool paths as if the part was being set against locating pins.

```
O0012 ;
G59 ;
G00 G90 X0 Y0 Z0 ;
M98 P1 ;
G51 X1.0 Y1.0 P2. ;
M98 P1 ;
M30 ;
```



If macros are enabled, G65 arguments are not affected.

Tool offsets and cutter compensation values are not affected by scaling.

The stored program is not changed by G51, so that program lines displayed by the control will not reflect actual machine positions. Position displays WILL reflect the proper scaled values.

Scaling does not affect canned cycle Z axis movements such as clearance planes and incremental values.

The final results of scaling are rounded to the lowest fractional value of the variable being scaled.

G68 ROTATION

GROUP 16

```
[ G17 | G18 | G19 ] G68 [a...] [b...] [R...];
```

- G17,G18,G19 optional plane of rotation, default is current.
- a optional center of rotation for the first axis of the selected plane.
- b optional center of rotation for the second axis of the selected plane.
- R optional angle of rotation specified in degrees.
Three-place decimal -360.000 to 360.000.

In the above example, 'a' and 'b' correspond to the axes of the current rotation plane. If G17 is the current rotation plane, then 'a' is X and 'b' is Y.

When rotation is invoked, all subsequent X, Y, Z, I, J, and K values are rotated through a specified rotation angle R using a center of rotation.

G68 is modal and modifies appropriate positional values in the blocks following the G68 command. Values in the block containing G68 are not rotated. For subsequent blocks, only the values in the plane of rotation are rotated. Thus, if G17 is the current plane of rotation, only X and Y values are affected.

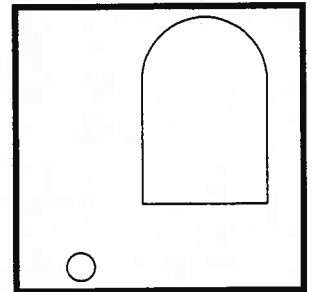
For a positive angle, the rotation is counterclockwise. If the angle of rotation - the R code - is not specified in the G68 command block, then the angle of rotation is taken from Setting 72. The default rotation angle in Setting 72 is 0.0 degrees.

A center of rotation is always used by the control to determine the positional values passed to the control after rotation. If any axis' center of rotation is not specified, then the current location of the work coordinate is used as the center of rotation.

In G90 mode (absolute), the rotation angle takes on the value specified in R. When Setting 73 (G68 INCREMENTAL R) is set to ON, then the rotational value can be incremented on each call to G68. In G91 mode (Incremental), the rotation angle is incremented by the value in R. Each G68 command block, when in G91 mode, will increment the rotation angle by the value specified in R. Angles are modulo 360, so that when an angle is incremented past 360 degrees, the angle will become an equivalent value between 0 and 360 degrees. The rotational angle is set to zero upon cycle start, or it can be set explicitly by using a G68 block in the G90 mode.

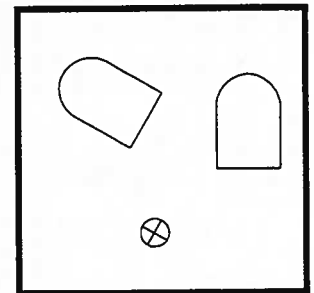
The following examples illustrate rotation using G68.

```
O0001 (GOTHIC WINDOW) ;
F20. S500 ;
G00 X1. Y1. ;
G01 X2. ;
Y2. ;
G03 X1. R0.5 ;      O = Work coordinate origin
G01 Y1. ;          + = Center of rotation
M99 ;
```



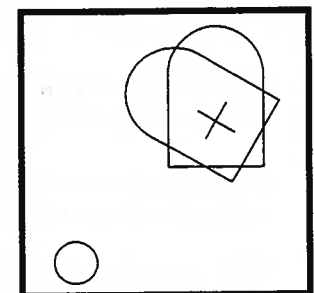
The first example illustrates how the control uses the current work coordinate location as a rotation center. Here, it is X0 Y0 Z0.

```
O0002 ;
G59 ;
G00 G90 X0 Y0 Z0 ;
M98 P1 ;
G90 G00 X0 Y0 ;
G68 R60. ;
M98 P1 ;
G69 G90 G00 X0 Y0 ;
M30 ;
```



The next example specifies the center of the window as the rotation center.

```
O0003 ;
G59 ;
G00 G90 X0 Y0 Z0 ;
M98 P1 ;
G00 G90 X0 Y0 Z0 ;
G68 X1.5 Y1.5 R60. ;
M98 P1 ;
G69 G90 G00 X0 Y0 ;
M30 ;
```

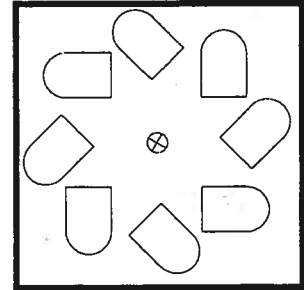


This example shows how the G91 mode can be used to rotate patterns about a center. This is often useful for making parts that are symmetric about a regular polygon.

```

O0004 ;
G59 ;
G00 G90 X0 Y0 Z0 ;
M98 P10 L8 (SUBROUTINE O0010) ;
M30 ;

O0010 ;
G91 G68 R45. ;
G90 M98 P1 ;
G69 ;
G90 G00 X0 Y0 ;
M99 ;
    
```



Do not change the plane of rotation while G68 is in effect.

ROTATION WITH SCALING

If scaling and rotation is used simultaneously, it is recommended that scaling is turned on prior to rotation, and that separate blocks be used. Use the following template when doing this.

```

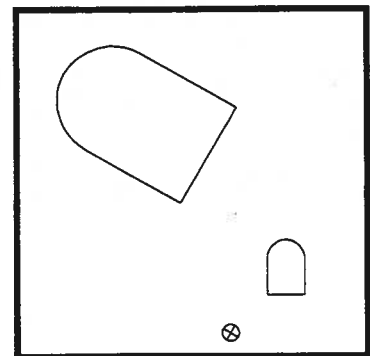
G51 ..... (SCALING) ;
...
G68 ..... (ROTATION) ;
.
. program.
.
G69 ..... (ROTATION OFF) ;
...
G50 ..... (SCALING OFF) ;
    
```

When rotating after scaling, any center specified as the center of rotation will be scaled. Any angle specified in the G68 block is NOT scaled. The control applies scaling and then rotation to any block with motion commands.

Below is an example of a program that has been scaled and rotated.

```

O0004 ;
G59 ;
G00 G90 X0 Y0 Z0 ;
M98 P1 ;
G90 G00 X0 Y0 ;
G51 P3.0 ;
G68 R60. ;
M98 P1 ;
G69 G51 G90 G00 X0 Y0 ;
M30 ;
    
```



ROTATION WITH CUTTER COMPENSATION

Cutter compensation should be turned on after the rotation and scaling commands are issued. Compensation should also be turned off prior to turning rotation or scaling off.

G69 CANCEL G68 ROTATION**GROUP 16**

G code 69 cancels any rotation specified previously.

3.16 WORK COORDINATE SYSTEM SELECTION

Note: The G52 command works differently depending on the value of Setting 33. This setting selects the FANUC, HAAS, or YASNAC style of coordinates, which are listed below:

G52 SELECT WORK COORDINATE SYSTEM G52 YASNAC**GROUP 12**

This code selects the G52 work coordinate system. The G52 works the same as G54 etc., except that the G52 system will be set by a G92 code as well as from the offsets display. YASNAC compatible.

G52 SET LOCAL COORDINATE SYSTEM FANUC**GROUP 00**

This code sets the origin of the local (child) coordinate system to the command location, relative to the current work system origin. G52 is a non-modal, no motion code. The G52 coordinate system will stay in effect for all work systems until it is canceled. The G52 is canceled when RESET is pressed and at the end of a program. It is also canceled during a program by M30, G52, X0 Y0 Z0, or by a G92 command.

G52 SET LOCAL COORDINATE SYSTEM HAAS**GROUP 00**

This code acts the same as in the Fanuc control except that G52 is not cleared at power-up, RESET, or when an M30 is performed.

G53 NON-MODAL MACHINE COORDINATE SELECTION**GROUP 00**

This code temporarily cancels work coordinates offset and uses the machine coordinate system. It is non-modal; so the next block will revert to whatever conditions were previously selected.

G54-59 SELECT COORDINATE SYSTEM #1 - #6**GROUP 12**

These codes select one of the six user coordinate systems stored within the offsets memory. All subsequent references to axes' positions will be interpreted in the new coordinate system. Work coordinate system offsets are entered from the Offsets display page.

3.17 MORE MISCELLANEOUS G CODES**G60 UNI-DIRECTIONAL POSITIONING****GROUP 00**

This G code is used to provide positioning always from the plus direction. In older systems it was used to reduce backlash and is not recommended for use with this control. It is provided only for compatibility. It is non-modal so does not effect the following blocks. Setting 35 controls the distance an axis is positioned past the point prior to reversing for an approach in the plus direction.

G61 EXACT STOP MODAL**GROUP 13**

The G61 code is used to specify exact stop. It is modal and thus affects the following blocks. Rapid and interpolated moves will decelerate to an exact stop before another block is processed. In exact stop, moves will take a longer time and continuous cutter motion will not occur.

This may cause deeper cutting where the tool stops.

G64 G61 CANCEL (SELECT NORMAL CUTTING MODE)**GROUP 13**

The G64 code is used to cancel exact stop. It is modal and thus affects the following blocks. Rapid and interpolated moves will not decelerate to an exact stop before another block is processed. Rapid blocks will decelerate to within the distance specified in Parameters 101-104 before another block is processed and interpolated motion will not decelerate at all before the next block is processed.

3.18 BOLT HOLE PATTERNS

There are three **G** codes that provide patterns usually used for bolt holes. These are G70, G71, and G72. They are normally used with one of the Group 09 canned cycles.

G70 BOLT HOLE CIRCLE**GROUP 00**

- I Radius (Minus Reverses Direction)
- J Starting angle (0 to 360.0 degrees CCW from horizontal)
- L Number of holes evenly spaced around the circle

This **G** code must be used with one of the canned cycles G73, G74, G76, G77, Or G81-G89. The tool must be positioned at the center of the circle either in a previous block or in the G70 block. G70 belongs to Group zero and thus is non-modal. For a G70 to work correctly, a canned cycle should be active so that at each of the positions, some type of drill or tap cycle is performed.

G71 BOLT HOLE ARC**GROUP 00**

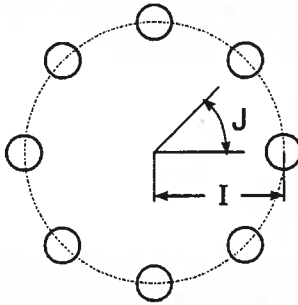
- I Radius
- J Starting angle (Degrees CCW from horizontal)
- K Angular spacing of holes (+ or -)
- L Number of holes

This **G** code is similar to G70 except that it is not limited at one complete circle. G71 belongs to Group zero and thus is non-modal. For a G71 to work correctly, a canned cycle should be active so that at each of the positions, some type of drill or tap cycle is performed.

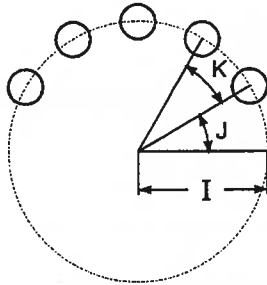
G72 BOLT HOLES ALONG AN ANGLE**GROUP 00**

- I Distance between holes (Minus will reverse direction)
- J Angle of line (Degrees CCW from horizontal)
- L Number of holes

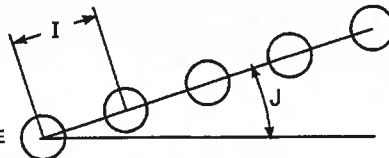
This **G** code drills **L** holes in a straight line at the specified angle. It operates similarly to G70 and G71. G72 belongs to Group zero and thus is non-modal. For a G72 to work correctly, a canned cycle should be active so that at each of the positions, some type of drill or tap cycle is performed.

G70
BOLT HOLE CIRCLE

- I = Radius of the bolt circle
 J = Starting angle from three o'clock position
 L = Number of holes (evenly spaced)

G71
BOLT HOLE ARC

- I = Radius of the bolt circle
 J = Starting angle from three o'clock position
 K = Angular spacing between holes
 L = Number of holes (evenly spaced)

G72
BOLT HOLES
ALONG AN ANGLE

- I = Distance between holes
 J = Angle from the three o'clock position
 L = Number of holes

3.19 CANNED CYCLES

A canned cycle is used to simplify programming of a part. Canned cycles are defined for most common Z-axis repetitive operation such as drilling, tapping, and boring. Once selected a canned cycle is active until canceled with G80. When active, the canned cycle is executed every time an X or Y-axis motion is programmed. Those Y-Y motions are executed as feed commands (G01) and the canned cycle operation is performed after the X-Y mode. There are six operations involved in every canned cycle:

- 1) positioning of X and Y axes (and optional A),
- 2) rapid traverse to R plane,
- 3) drilling,
- 4) operation at bottom of hole,
- 5) retraction to R plane,
- 6) rapid traverse up to initial point.

A canned cycle is presently limited to operations in the Z-axis. That is, only the G17 plane is allowed. This means that the canned cycle will be executed in the Z-axis whenever a new position is selected in the X or Y axes.

The following is a summary of the canned cycles defined for the VF Series Mill:

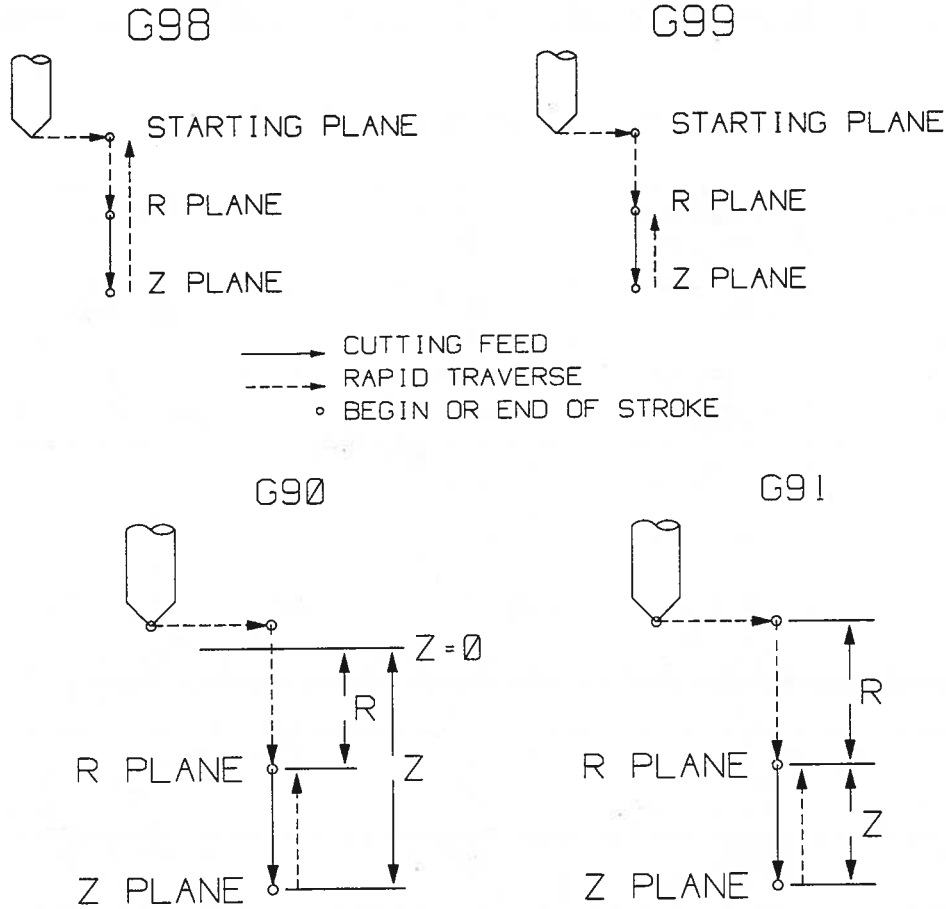
G code	Z Drilling direction	Operation at bottom of hole	Retraction Z direction	Application
G73	intermittent feed	none	rapid	high speed peck drilling
G74	feed	spindle CW	feed	left hand tapping
G76	feed then stop	orient spindle	rapid	fine boring
G81	feed	none	rapid	spot drilling
G82	feed	dwel	rapid	counter boring
G83	intermittent feed	none	rapid	peck drilling
G84	feed	spindle CCW	feed	tapping cycle
G85	feed	none	feed	boring cycle
G86	feed	spindle stop	rapid	boring cycle
G87	feed	spindle stop	manual/rapid	back boring
G88	feed	dwel, then spindle stop	manual/rapid	boring cycle
G89	feed	dwel	feed	boring cycle

G98 and G99 are modal commands that change the way the canned cycles operate. When G98 is active, the Z-axis will be returned to the same position as at the start of the canned cycle when it completes. When G99 is active, the Z-axis will be returned to the R point when the canned cycle completes.

If a canned cycle is defined in a block without an X or Y motion, there are two common actions taken by other controls; some will execute the canned cycle at that time and some will not. With the VF Series Mill, these two options are selectable from Setting 28. In addition to this, if a canned cycle is defined without an X or Y and a loop count of 0 (L0), the cycle will not be performed initially. The operation of a canned cycle will vary according to whether incremental (G91) or absolute (G90) is active.

Incremental motion in a canned cycle is often useful as a loop (L) count can be used to repeat the operation with an incremental X or Y move between each cycle.

The positioning of the X-Y axis prior to a canned cycle is normally a rapid move and that move does not exact stop prior to plunging the Z-axis to the R depth. This may cause a crash with a close tolerance fixture. Setting 57 can be used to select exact stop of these X-Y moves.



The G80 code is used to cancel a canned cycle. In addition to this, a G00 or G01 code will also cancel any active canned cycle. Once a canned cycle is defined, that operation is performed at every X-Y position subsequently listed in a block. Some of the canned cycle numerical values can also be changed after the canned cycle is defined. The most important of these are the R plane value and the Z depth value.

If these are listed in a block with an X-Y, the X-Y move is done and all subsequent canned cycles are performed with the new R or Z value.

Changes to the G98/G99 selection can also be made after the canned cycle is active. If changed, the new G98/G99 value will change all subsequent canned cycle.

G73 HIGH SPEED PECK DRILLING CANNED CYCLE

GROUP 09

- F Feed Rate in inches (mm) per minute
- I Optional size of first cutting depth
- J Optional amount to reduce cutting depth each pass
- K Optional minimum depth of cut
- L Number of repeats
- Q The cut-in value, always incremental
- R Position of the R plane
- X Optional X-axis motion command
- Y Optional Y-axis motion command
- Z Position of bottom of hole

This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X or Y will cause this canned cycle to be executed. This cycle is a high speed peck cycle where the retract distance is set by Setting 22.

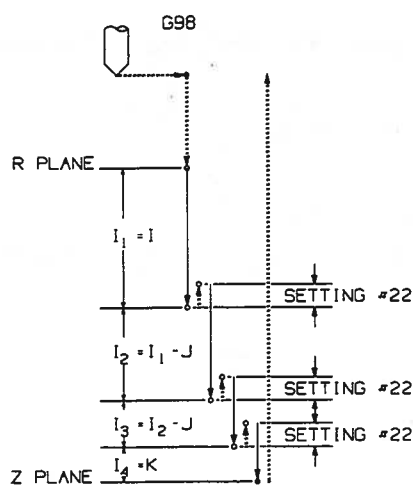
If I, J, and K are specified, a different operating mode is selected. The first pass will cut in by I, each succeeding cut will be reduced by amount J, and the minimum cutting depth is K.

If K and Q are both specified, a different operating mode is selected for this canned cycle. In this mode, the tool is returned to the R plane after a number of passes totals up to the K amount. This allows much faster drilling than G83 but still returns to the R plane occasionally to clear chips.

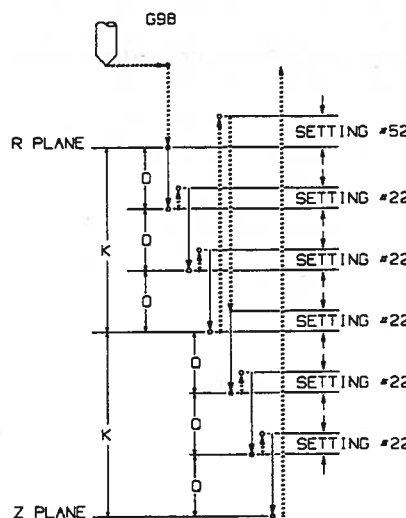
I, J, K, and Q are always positive numbers.

Setting 52 also changes the way G73 works when it returns to the R plane. Most programmers set the R plane well above the cut to ensure that the chip clear motion actually allows the chips to get out of the hole but this causes a wasted motion when first drilling through this "empty" space. If Setting 52 is set to the distance required to clear chips, the R plane can be put much closer to the part being drilled. When the clear move to R occurs, the Z will be moved above R by this setting.

G73 PECK DRILLING CANNED CYCLE WITH I, J, AND K OPTIONS



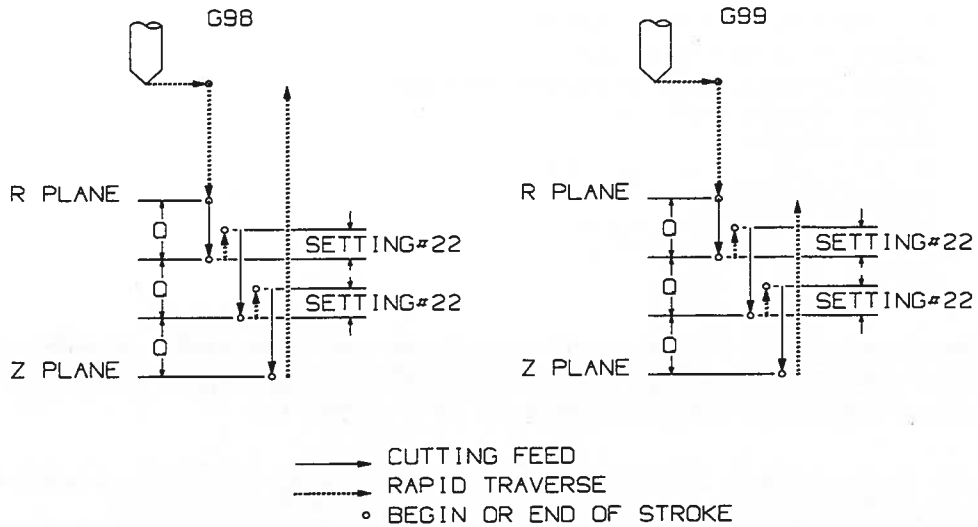
G73 PECK DRILLING CANNED CYCLE WITH K AND Q OPTIONS



— CUTTING FEED
 - - - - - RAPID TRAVERSE
 • BEGIN OR END OF STROKE

— CUTTING FEED
 - - - - - RAPID TRAVERSE
 • BEGIN OR END OF STROKE

G73 PECK DRILLING CANNED CYCLE



G74 REVERSE TAP CANNED CYCLE

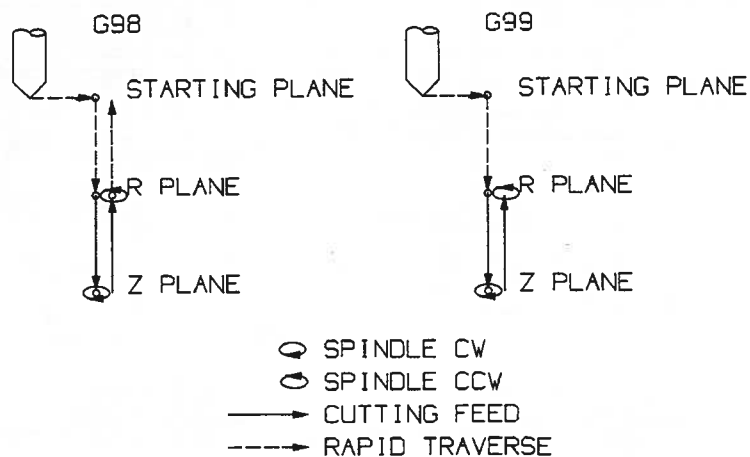
GROUP 09

- F Feed Rate in inches (mm) per minute
- L Number of repeats
- R Position of the R plane
- X Optional X-axis motion command
- Y Optional Y-axis motion command
- Z Position of bottom of hole

This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X or Y will cause this canned cycle to be executed. Note that operation of this cycle is different if the rigid tapping option is installed and selected (See Section 7.2). When rigid tapping is used, the ratio between the feed rate and spindle speed must be precisely the thread pitch being cut.

You do not need to start the spindle CCW before this canned cycle. The control does this automatically.

G74 REVERSE TAP CANNED CYCLE



G76 FINE BORING CANNED CYCLE

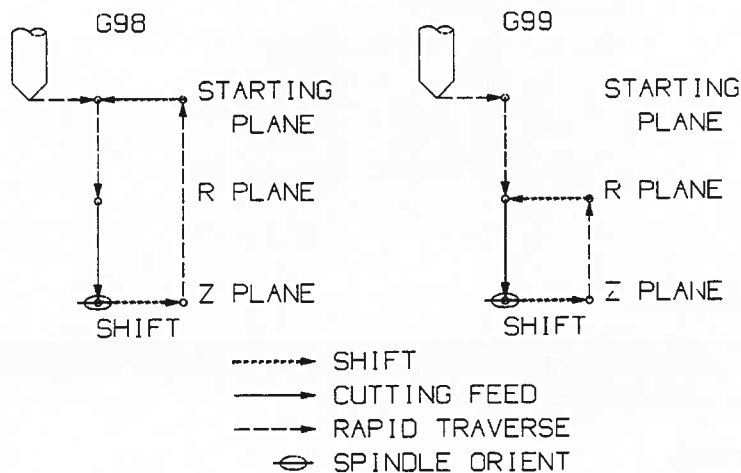
GROUP 09

- F Feed Rate in inches (mm) per minute
- I Optional shift value, if Q is not specified.
- J Optional shift value, if Q is not specified.
- L Number of repeats
- P The dwell time at the bottom of the hole
- Q The shift value, always incremental
- R Position of the R plane
- X Optional X-axis motion command
- Y Optional Y-axis motion command
- Z Position of bottom of hole

This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X and/or Y will cause this canned cycle to be executed. This cycle will shift the X and/or Y-axis prior to retracting in order to clear the tool while exiting the part. This shift direction is set by Setting 27.

The Q value shift direction is set by setting 27. If Q is not specified, the optional I and J values are used to determine the shift direction and distance.

G76 FINE BORING CANNED CYCLE



G77 BACK BORE CANNED CYCLE

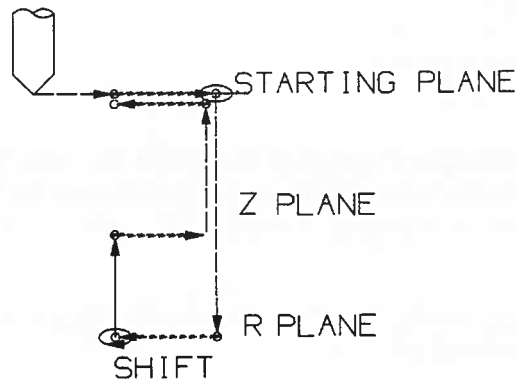
GROUP 09

- F Feed Rate in inches (mm) per minute
- I Optional shift value, if Q is not specified.
- J Optional shift value, if Q is not specified.
- L Number of repeats
- Q The shift value, always incremental
- R Position of the R plane
- X Optional X-axis motion command
- Y Optional Y-axis motion command
- Z Position of bottom of hole

This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X or Y will cause this canned cycle to be executed.

This cycle will shift the X and/or Y axis prior to and after cutting in order to clear the tool while entering and exiting the part. The Q value shift direction is set by setting 27. If Q is not specified, the optional I and J values are used to determine the shift direction and distance.

G77 BACK BORE CANNED CYCLE



- > SHIFT
- > CUTTING FEED
- > RAPID TRAVERSE
- ⊖ SPINDLE ORIENT
- ⊙ SPINDLE CW

G80 CANNED CYCLE CANCEL**GROUP 09**

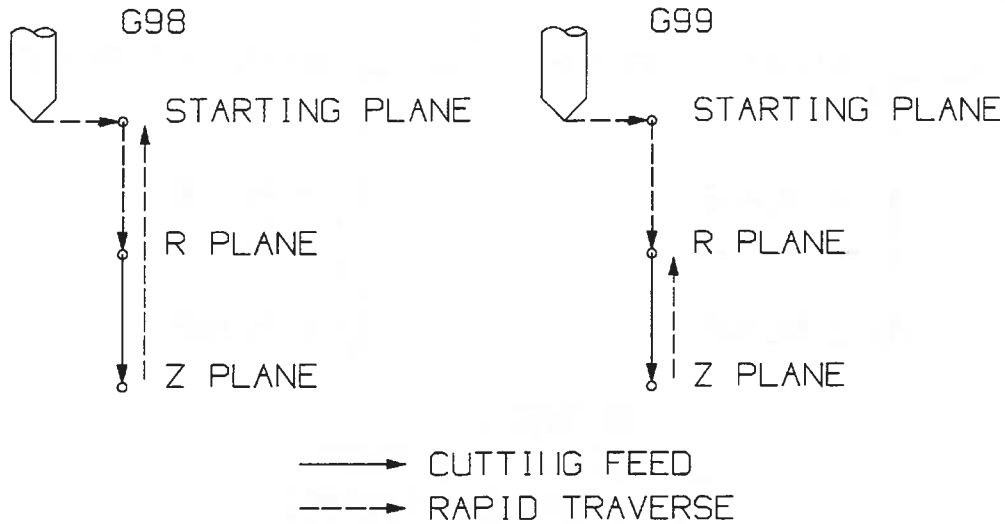
This G code is modal in that it deactivates all canned cycles until a new one is selected. Note that use of G00 or G01 will also cancel a canned cycle.

G81 DRILL CANNED CYCLE**GROUP 09**

- F Feed Rate in inches (mm) per minute
- L Number of repeats
- R Position of the R plane
- X Optional X-axis motion command
- Y Optional Y-axis motion command
- Z Position of bottom of hole

This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X or Y will cause this canned cycle to be executed.

G81 DRILL CANNED CYCLE



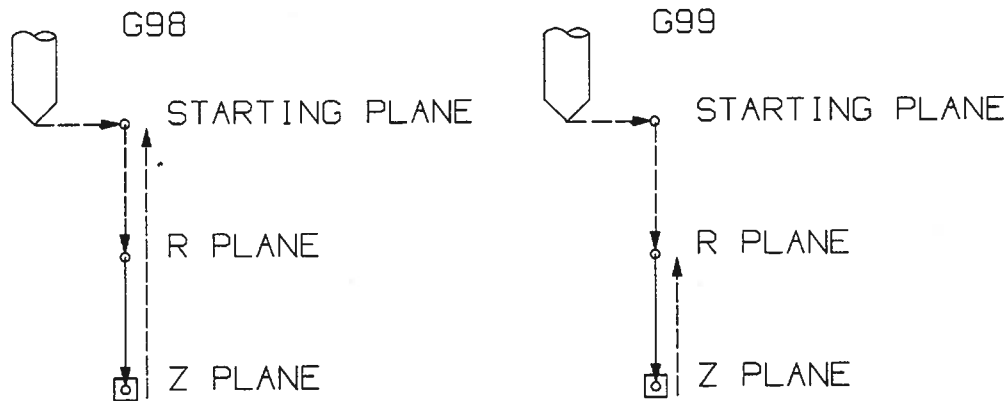
G82 SPOT DRILL CANNED CYCLE

GROUP 09

- F Feed Rate in inches (mm) per minute
- L Number of repeats
- P The dwell time at the bottom of the hole
- R Position of the R plane
- X Optional X-axis motion command
- Y Optional Y-axis motion command
- Z Position of bottom of hole

This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X or Y will cause this canned cycle to be executed.

G82 SPOT DRILL CANNED CYCLE



◻ DWELL

————→ CUTTING FEED

- - - - -→ RAPID TRAVERSE

◦ BEGIN OR END OF STROKE

G83 PECK DRILLING CANNED CYCLE

GROUP 09

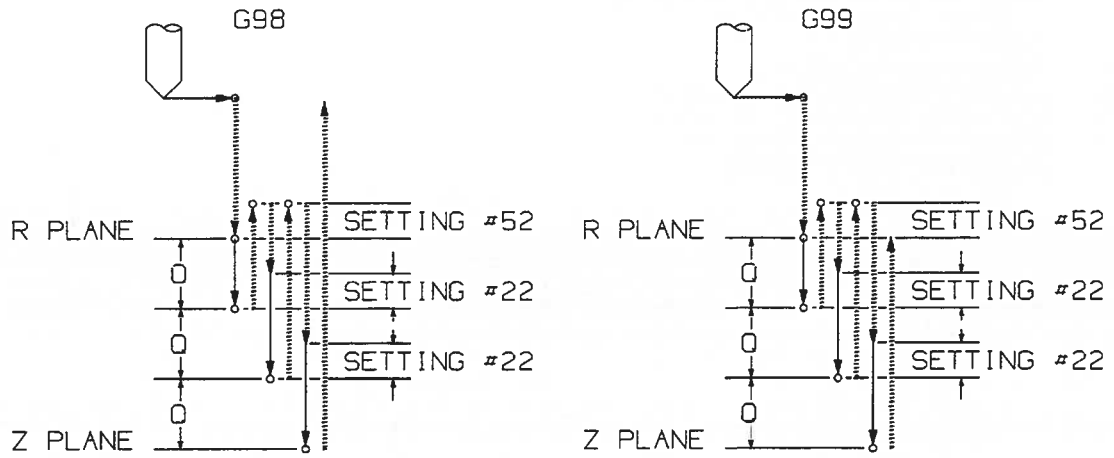
- F Feed Rate in inches (mm) per minute
- I Optional size of first cutting depth
- J Optional amount to reduce cutting depth each pass
- K Optional minimum depth of cut
- L Number of repeats
- Q The cut-in value, always incremental
- R Position of the R plane
- X Optional X-axis motion command
- Y Optional Y-axis motion command
- Z Position of bottom of hole

This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X or Y will cause this canned cycle to be executed.

If I, J, and K are specified, a different operating mode is selected. The first pass will cut in by I, each succeeding cut will be reduced by amount J, and the minimum cutting depth is K.

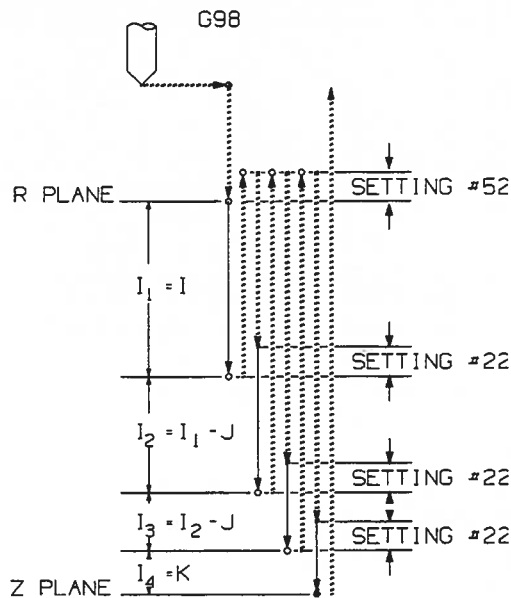
Setting 52 also changes the way G83 works when it returns to the R plane. Most programmers set the R plane well above the cut to insure that the chip clear motion actually allows the chips to get out of the hole but this causes a wasted motion when first drilling through this "empty" space. If Setting 52 is set to the distance required to clear chips, the R plane can be put much closer to the part being drilled. When the clear move to R occurs, the Z will be moved above R by this setting.

G83 PECK DRILLING CANNED CYCLE



→ CUTTING FEED
 → RAPID TRAVERSE
 ◦ BEGIN OR END OF STROKE

G83 PECK DRILLING CANNED CYCLE WITH I, J, AND K OPTIONS



→ CUTTING FEED
 → RAPID TRAVERSE
 ◦ BEGIN OR END OF STROKE

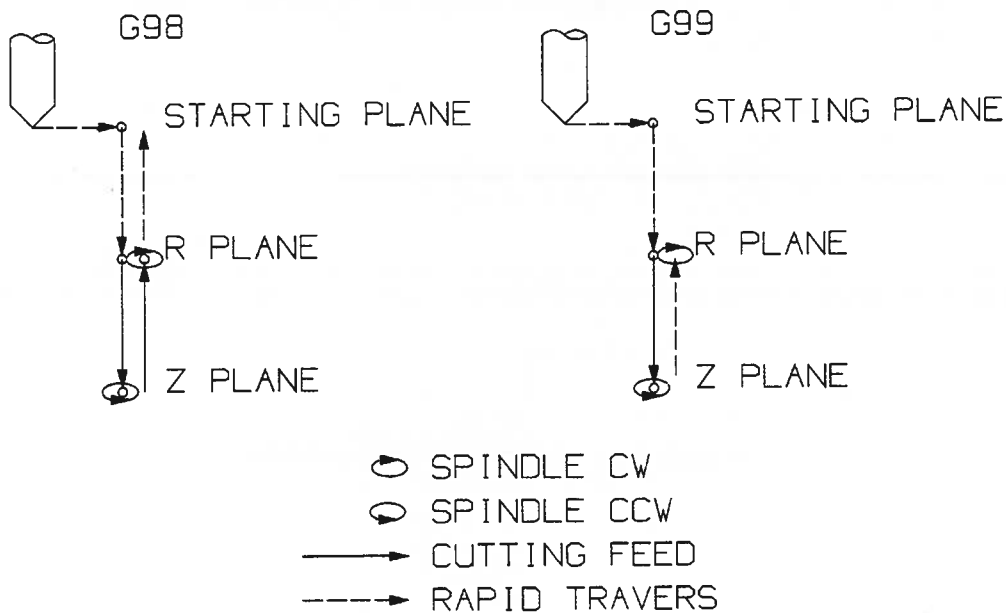
G84 TAPPING CANNED CYCLE
GROUP 09

- F Feed Rate in inches (mm) per minute
- L Number of repeats
- R Position of the R plane
- X Optional X-axis motion command
- Y Optional Y-axis motion command
- Z Position of bottom of hole

This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X or Y will cause this canned cycle to be executed. Note that operation of this cycle is different if the rigid tapping option is installed and selected (See Section 7.2). When rigid tapping is used, the ratio between the feed rate and spindle speed must be precisely the thread pitch being cut.

You do not need to start the spindle CW before this canned cycle. The control does this automatically.

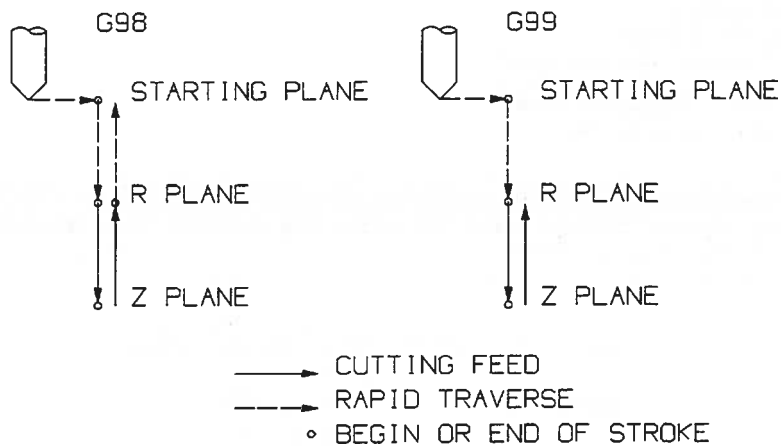
G84 TAPPING CANNED CYCLE


G85 BORING CANNED CYCLE
GROUP 09

- F Feed Rate in inches (mm) per minute
- L Number of repeats
- R Position of the R plane
- X Optional X-axis motion command
- Y Optional Y-axis motion command
- Z Position of bottom of hole

This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X or Y will cause this canned cycle to be executed.

G85 BORING CANNED CYCLE



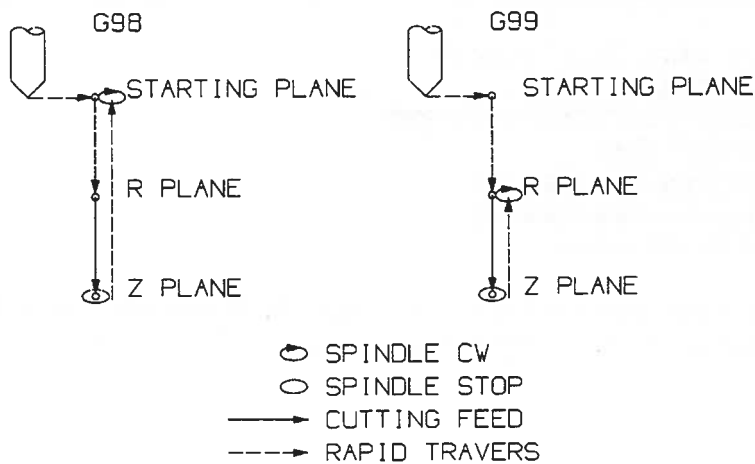
G86 BORE AND STOP CANNED CYCLE

GROUP 09

- F Feed Rate in inches (mm) per minute
- L Number of repeats
- R Position of the R plane
- X Optional X-axis motion command
- Y Optional Y-axis motion command
- Z Position of bottom of hole

This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X or Y will cause this canned cycle to be executed.

G86 BORE AND STOP CANNED CYCLE

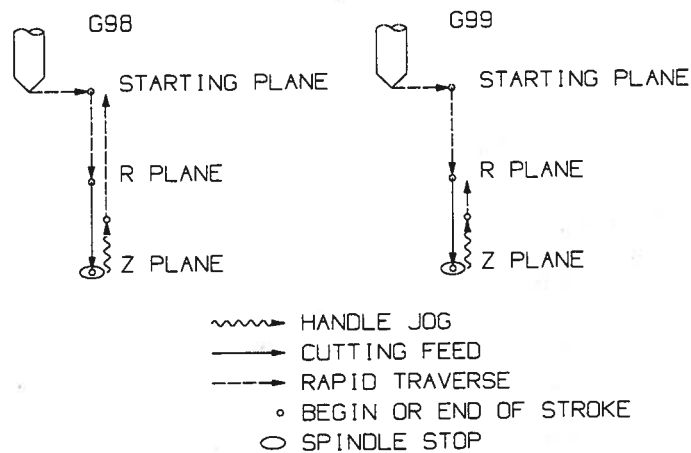


G87 BORE AND MANUAL RETRACT CANNED CYCLE**GROUP 09**

- F Feed Rate in inches (mm) per minute
- L Number of repeats
- R Position of the R plane
- X Optional X-axis motion command
- Y Optional Y-axis motion command
- Z Position of bottom of hole

This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X or Y will cause this canned cycle to be executed.

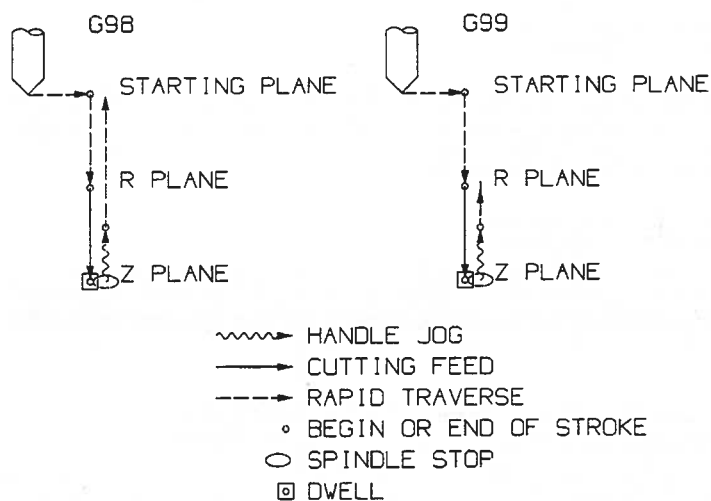
G87 BORE AND RETRACT CANNED CYCLE

**G88 BORE AND DWELL CANNED CYCLE****GROUP 09**

- F Feed Rate in inches (mm) per minute
- L Number of repeats
- P The dwell time at the bottom of the hole
- R Position of the R plane
- X Optional X-axis motion command
- Y Optional Y-axis motion command
- Z Position of bottom of hole

This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X or Y will cause this canned cycle to be executed.

G88 BORE AND DWELL CANNED CYCLE



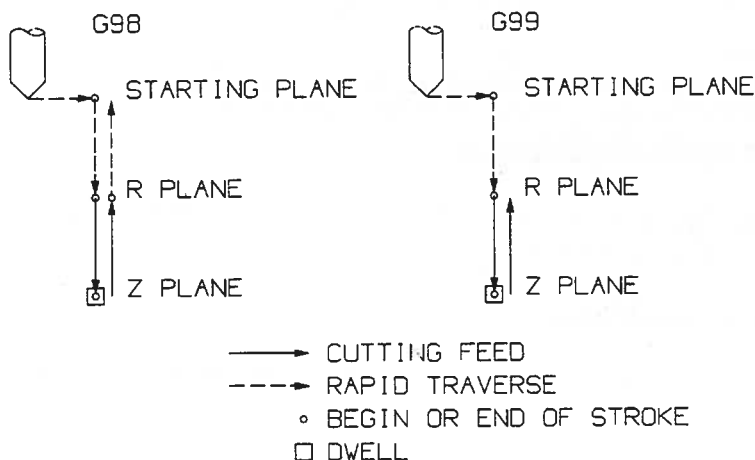
G89 BORE CANNED CYCLE

GROUP 09

- F Feed Rate in inches (mm) per minute
- L Number of repeats
- P The dwell time at the bottom of the hole
- R Position of the R plane
- X Optional X-axis motion command
- Y Optional Y-axis motion command
- Z Position of bottom of hole

This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X or Y will cause this canned cycle to be executed.

G89 BORE CANNED CYCLE



3.20 ABSOLUTE / INCREMENTAL SELECTION

G90 ABSOLUTE POSITION COMMANDS

GROUP 03

This code is modal and changes the way axis motion commands are interpreted. G90 makes all subsequent commands absolute positions within the selected user coordinate system. Each axis which is moved will be placed at the position coded in the command block.

G91 INCREMENTAL POSITION COMMANDS

GROUP 03

This code is modal and changes the way axis motion commands are interpreted. G91 makes all subsequent commands incremental. Each axis which is moved will be moved by the amount coded in the command block.

3.21 MORE WORK COORDINATE SELECTION

This command works differently depending on the value of Setting 33. That setting selects FANUC, HAAS, or YASNAC style of coordinates. This command does not move any of the axis; it only changes the values stored as user work offsets.

G92 SET WORK COORDINATE SYSTEMS SHIFT VALUE - FANUC OR HAAS

GROUP 00

A G92 command effectively shifts all work coordinate systems (G54-59, G110-129) so that the command position becomes the the current position in the active work system. G92 is a non-modal, non-motion code.

A G92 command cancels any G52 in effect for the command axes. Example: G92 X1.4 cancels the G52 for the X-axis. The other axes are not affected.

The G92 shift value is displayed at the bottom of the work offsets page and may be cleared there if necessary. It is also cleared automatically after power up when the POWER UP/RESTART key is pressed, and any time ZERO RET is used to AUTO ALL AXES or ZERO SINGLE AXIS.

G92 SET WORK COORDINATE SYSTEMS SHIFT VALUE - YASNAC

GROUP 00

A G92 command sets the G52 work coordinate system so that the command position becomes the current position in the active work system. The G52 work system then automatically becomes active until another work system is selected. G92 is a non-modal, non-motion code.

3.22 CANNED CYCLE AUXILIARY FUNCTIONS

G98 CANNED CYCLE INITIAL POINT RETURN

GROUP 10

This G code is modal and changes the way canned cycles operate. With G98, the canned cycle will return to the initial starting point of the canned cycle when it completes.

G99 CANNED CYCLE R PLANE RETURN

GROUP 10

This G code is modal and changes the way canned cycles operate. With G99, the canned cycle will return to the R plane when the canned cycle completes.

3.23 PROGRAMMABLE MIRROR IMAGE (G100, G101)

G100 DISABLE MIRROR IMAGE	GROUP 00
G101 ENABLE MIRROR IMAGE	GROUP 00

- X Optional X-axis command
- Y Optional Y-axis command
- Z Optional Z-axis command
- A Optional A-axis command

Programmable mirror image can be turned on or off individually for any of the four axes. The two G codes (G100 and G101) are non-modal but the mirror image status of each axis is modal. The bottom of the CRT will indicate when an axis is mirrored. These G codes should be used in a command block without any other G codes and will not cause any axis motion. G101 will turn on mirror image for any axis listed in that block. G100 will turn off mirror image for any axis listed in the block. The actual value given for the X, Y, Z, or A code has no effect.

When using Cutter Compensation with Mirror Imaging, follow this guideline: After turning Mirror Imaging ON or OFF with a G100 or G101, the next motion block should be to a different work coordinate position than the first one. The following code is an example:

Incorrect:	Correct:
G41 X1.0 Y1.0	G41 X1.0 Y1.0
G01 X2.0 Y2.0	G01 X2.0 Y2.0
G101 X0	G101 X0
G00 Z1.0	G00 Z1.0
G00 X2.0 Y2.0	G00 X1.0
	G00 X2.0 Y2.0
G40	G40

The mirror function can change the direction of motion along any of the axes. If any one of these are selected, the display will show the status. Mirror image will reflect programmed motion around your work coordinate zero point. Be careful that mirror of only one of X or Y will cause the cutter to move along the opposite side of a cut. In addition, if mirror is selected for only one axis of a circular motion plane, circular motion G02 and G03 are reversed and left side and right side cutter compensation G41 and G42 are reversed. Settings 45 through 48 are used to select mirror image.

3.24 PROGRAMMABLE OUTPUT TO RS-232 (G102)

G102 PROGRAMMABLE OUTPUT TO RS-232	GROUP 00
---	-----------------

- X Optional X-axis command
- Y Optional Y-axis command
- Z Optional Z-axis command
- A Optional A-axis command

Programmable output to the RS-232 port allows the current work coordinates of the four axes to be output. This G code (G102) is non-modal so only effects the block in which it is programmed. This G code should be used in a command block without any other G codes and will not cause any axis motion. The actual value given for the X, Y, Z, or A code has no effect. One complete line of text is sent to the first RS-232 port (same one used for upload, download, and DNC). Each axis listed in the G102 command block is output to the RS-

232 port in the same format as values are displayed in a program.

Optional spaces (Setting 41) and EOB control (Setting 25) are applied. The values sent out are always the current axes positions referenced to the current work coordinate system.

Digitizing of a part is possible using this G code and a program which steps over a part in X-Y and probes downward in Z with a G31. When the probe hits, the next block could be a G102 to send the X, Y, Z position out to a computer which could store the coordinates as a digitized part.

3.25 BLOCK LOOKAHEAD LIMIT FUNCTION (G103)

G103 BLOCK LOOKAHEAD LIMIT

GROUP 00

P = 0-15 Max. number of blocks the control will look ahead

G103 [P..]

"Block Lookahead" is a term used to describe what the control is doing in the background during machine motion. A motion block may take several seconds to execute. The control can take advantage of this by preparing additional blocks of the program ahead of time. Time is saved while the current block is executing and the next block has already been interpreted and prepared by the continuous, uninterrupted motion between consecutive blocks. Block lookahead is also important for obtaining information necessary for predicting compensated positions for cutter compensation.

When G103 P0 is programmed, block limiting is disabled. Block limiting is also disabled if G103 appears in a block without a P address code.

When G103 Pn is programmed, lookahead is limited to n blocks.

At this time G103 cannot be used if cutter compensation, G41 or G42, is in effect. Alarm 371 is generated if you attempt to do so.

G103 is also useful for debugging programs using macros. Macro expressions are executed at lookahead time. By inserting a G103 P1 into the program, macro expressions will be performed one block ahead of the current executing block.

G103 is not a FANUC compatible command.

3.26 MORE WORK COORDINATE SELECTION

G110-G129 COORDINATE SYSTEM #7-26

GROUP 12

These codes select one of the additional 20 user coordinate systems stored within the offsets memory. All subsequent references to axes positions will be interpreted in the new coordinate system. Operation of G110 to G129 are the same as G54 to G59.

3.27 GENERAL PURPOSE POCKET MILLING FUNCTION

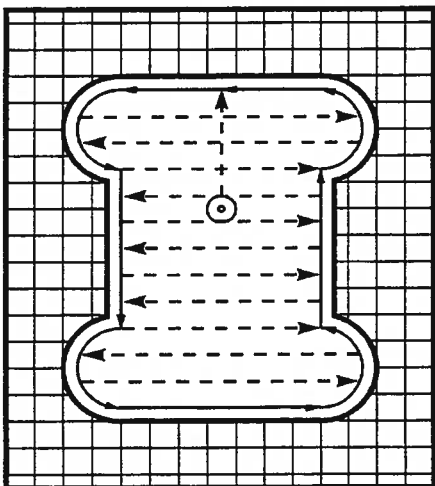
G150 GENERAL PURPOSE POCKET MILLING

GROUP 00

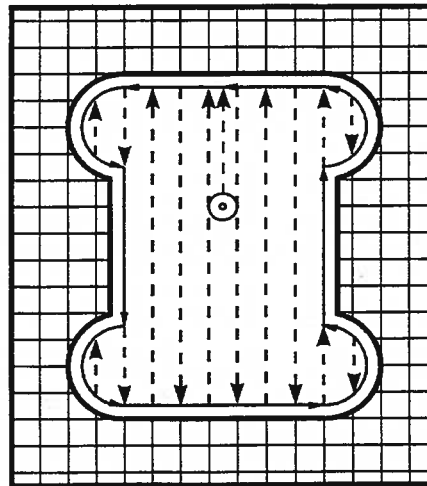
- D Cutter size selection
- F Feed rate
- I X-axis cut increment
- J Y-axis cut increment
- K Finishing cut allowance
- L Optional repetition count
- P Subroutine number defining outside of shape
- Q Incremental Z-axis cut depth per pass, positive values only (> 0)
- R R plane position
- S Optional spindle speed
- X X position of starting hole
- Y Y position of starting hole
- Z Final depth of the pocket

This G code provides for general purpose pocket milling. The shape of the pocket to be cut must be defined by a series of motions within a subroutine. A series of motions in either the X or Y-axis will be used to cut out the specified shape followed by a finishing pass to clean up the outer edge. One of either I or J must be specified. If I is used, the pocket is cut from a series of strokes in the Y-axis. If J is used, the pocket is cut from a series of strokes in the X-axis. I and J must be positive numbers. The finishing pass is of width K and K must be a positive number. There is no finishing pass in the Z depth.

G150 ADVANCED POCKET MILLING



J > 0



I > 0

⊙ = **STARTING HOLE**

Multiple passes over the area can be selected to control the depth of the cut. At least one pass is made over the pocket and multiple passes are made after feeding down by Q amount until the Z depth is reached. Q must be positive. If an L count is specified, the entire block is repeated and an incremental X or Y (G91) will reposition the pocket.

The subroutine must define a closed area by a series of G01, G02, or G03 motions in X and Y and must end with an M99. G codes G90 and G91 can also be used in the subroutine to select absolute or incremental. Any codes other than G, I, J, R, X, or Y are ignored in the subroutine. This subroutine must consist of less than 20 strokes.

Pocket milling should begin from a hole which has been previously drilled to the Z depth in order to clear the tool on entry to the pocket. The G150 block must specify this hole location with X and Y. The first motion in the subroutine should move from this clear hole to the starting point of the block shape. The final motion in the subroutine should return to the same point as the starting motion of the subroutine, ie., in the example on the previous page, the start point of the G150 line is X3.25 Y4.5 and the first move of the of the sub is Y7.0. Therefore the end of the sub must return to X3.25 Y7.0.

If a K is specified, the finishing pass is taken along the outside edge but is done at the full pocket depth and the previous cuts will cut inside the programmed pocket size by K.

```
00100 (G150 POCKET EXAMPLE)
G58 G00 G90 X3.25 Y4.5 (STARTING HOLE POSITION)
T1 M06 (T1 CUTS ENTRY FOR END MILL)
G83 R-1. Q0.5 Z-2. F20.
T2 M06 (END MILL T2 CUTS POCKET)
(0.4 DIA CUTTER, TWO PASSES TO Z DEPTH)
(LEAVE 0.1 FOR FINISH PASS)
G150 G41 F15. D01 J0.35 K0.1 Q0.5 R-1. X3.25 Y4.5 Z-2. P200
G40 G28
M30
```

```
00200 (G150 POCKET SUBROUTINE)
G01 Y7.
X1.5
G03 Y5.25 R0.875
G01 Y2.25
G03 Y0.5 R0.875
G01 X5.
G03 Y2.25 R0.875
G01 Y5.25
G03 Y7. R0.875
G01 X3.25
M99 (RETURN FROM SUBROUTINE)
```

3.28 IN POSITION ACCURACY (G 187)

G187 ACCURACY CONTROL

GROUP 00

The G 187 code is used to select the accuracy with which corners are machined. The form for using G 187 is G 187 Ennnn, where nnnn is the desired accuracy. Refer to "Contouring Accuracy" for more information.

4. MISCELLANEOUS FUNCTIONS (M FUNCTIONS)**4.1 M CODE SUMMARY**

Only one **M** code may be programmed per block of a program. All **M** codes are effective or cause an action to occur at the end of the block and only one **M** code is allowed in each block.

M00	Stop Program
M01	Optional Program Stop
M02	Program End
M03	Spindle Forward
M04	Spindle Reverse
M05	Spindle Stop
M06	Tool Change
M08	Coolant On
M09	Coolant Off
M10	Engage 4th Axis Brake
M11	Release 4th Axis Brake
M12	Engage 5th Axis Brake
M13	Release 5th Axis Brake
M16	Tool Change (same as M06)
M19	Orient Spindle
M21-M24	Optional Pulsed User M Function with Fin
M30	Prog End and Rewind
M31	Chip Conveyor Forward
M32	Chip Conveyor Reverse
M33	Chip Conveyor Stop
M34	Increment Coolant Spigot Position
M35	Decrement Coolant Spigot Position
M36	Pallet Rotate
M39	Rotate Tool Turret
M41	Low Gear Override
M42	High Gear Override
M51-M54	Optional User M turn ON
M61-M64	Optional User M turn OFF
M75	Set G35 or G136 reference point
M76	Disable Displays
M77	Enable Displays
M78	Alarm if skip signal found
M79	Alarm if skip signal not found
M82	Tool Unclamp
M86	Tool Clamp
M88	Thru Spindle Coolant ON
M89	Thru Spindle Coolant OFF
M96	Conditional Local Branch when Discrete Input Signal is 0.
M97	Local Sub-Program Call
M98	Sub Program Call
M99	Sub Program Return Or Loop

4.2 M CODE DETAILED DESCRIPTION**M00 STOP PROGRAM**

The M00 code is used to stop a program. It also stops the spindle and turns off the coolant and stops interpretation lookahead processing. The program pointer will advance to the next block and stop. A cycle start will continue program operation from the next block. If the Through the Spindle Coolant option is ON, M00 will shut it off.

M01 OPTIONAL PROGRAM STOP

The M01 code is identical to M00 except that it only stops if OPTIONAL STOP is turned on from the front panel. A cycle start will continue program operation from the next block. If the Through the Spindle Coolant option is ON, M01 will shut it off.

M02 PROGRAM END

The M02 code will stop program operation the same as M00 but does not advance the program pointer to the next block.

M03 SPINDLE FORWARD

The M03 code will start the spindle moving in a clockwise direction at whatever speed was previously set. The block will delay until the spindle reaches about 90% of commanded speed.

M04 SPINDLE REVERSE

The M04 code will start the spindle moving in a counterclockwise direction at whatever speed was previously set. The block will delay until the spindle reaches about 90% of commanded speed.

M05 SPINDLE STOP

The M05 code is used to stop the spindle. The block is delayed until the spindle slows below 10 RPM.

M06 TOOL CHANGE

The M06 code is used to initiate a tool change. The previously selected tool (Tn) is put into the spindle. If the spindle was running, it will be stopped. No previous axis commands are required before the tool change unless there is a problem with tool/part/fixture clearance. The Z-axis will automatically move up to the machine zero position and the selected tool will be put into the spindle. The Z-axis is left at machine zero. The spindle will not be started again after the tool change but the Snnnn speed and gear will be unchanged. The Tnn must be in the same block or in a previous block. The coolant pump will be turned off during a tool change.

When the Through the Spindle Coolant (TSC) is ON, M06 will orient the spindle and move the Z-axis to tool change position, turn off the TSC pump, purge the coolant from the drawbar, then perform a tool change. TSC will remain OFF until an M88 is called.

M08 COOLANT ON

The M08 code will turn on the coolant supply. Note that the M code is performed at the end of a block; so that if a motion is commanded in the same block, the coolant is turned on after the motion. The low coolant status is only checked at the start of a program so a low coolant condition will not stop a program which is already running.

M09 COOLANT OFF

The M09 code will turn off the coolant supply.

M10 ENGAGE 4TH AXIS BRAKE

The M10 code is used to apply the optional brake to the 4th axis. It is only used when M11 is used to release the brake.

M11 RELEASE 4TH AXIS BRAKE

The M11 code will "pre-release" the 4th axis brake. This is useful to prevent the delay otherwise occurring when a 4th axis is used with a brake and a motion is commanded in that axis. It is not required but, without a prior M11, there will be a delay in motion in order to release the air.

M12 RELEASE 5TH AXIS BRAKE

This will activate relay M27 (fifth axis brake) and leave it active. Use M13 to engage the fifth axis brake.

M13 ENGAGE 5TH AXIS BRAKE

This will deactivate relay M27 (fifth axis brake). Use M12 to release the fifth axis brake.

M16 TOOL CHANGE

The M16 code is used to initiate a tool change. In the present machine configuration, M16 works exactly like M06.

M19 ORIENT SPINDLE

The M19 code is used to orient the spindle to a fixed position. This command leaves the spindle in that position and locked by a pin. The next spindle motion command (**Snnnn**, M3, M4, M41, or M42) will release the pin and unlock the spindle.

M21-M24 OPTIONAL USER M

The M21 through M24 codes are optional for user interfaces. They will activate one of relays 25 through 28, wait for the M-fin signal, release the relay, and wait for the M-fin signal to cease. The RESET button will terminate any operation that is hung-up waiting for M-fin. When Through the Spindle Coolant is enabled, M24 is disabled.

When the MCD RELAY BRD is installed and the bit (Parameter 209) is set to 1, M21-M28 become available on a secondary board.

M30 PROG END AND REWIND

The M30 code is used to stop a program. It also stops the spindle and turns off the coolant. The program pointer will be reset to the first block of the program and stop. The parts counters displayed on the Current Commands display are also incremented. M30 will also cancel tool length offsets. When the Through the Spindle Coolant (TSC) option is ON, M30 will shut it OFF, and then perform an M30 operation.

M31 CHIP CONVEYOR FORWARD

M31 starts the chip conveyor motor in the forward direction. The forward direction is defined as the direction that the conveyor must move to transport chips out of the work cell. If the conveyor motor is on, then the conveyor will be stopped and restarted in the forward direction.

M32 CHIP CONVEYOR REVERSE

M32 starts the chip conveyor motor in the reverse direction. The reverse direction is defined as the direction opposite of forward. If the conveyor motor is on, then the conveyor will be stopped and restarted in the reverse direction.

M33 CHIP CONVEYOR STOP

M33 Stops Conveyor motion.

M34 INCREMENT COOLANT SPIGOT POSITION

M34 Increments the current spigot position one place. Incrementing the spigot position causes the spigot to advance one place from the home position. The home position is designated as zero. If the current home position is designated as 5 and M34 is executed, then the current spigot position will advance to position 6. The spigot home position for a horizontal mill places the spigot at the most positive Z axis location the spigot can attain. Incrementing the spigot then lowers the coolant stream direction.

M35 DECREMENT COOLANT SPIGOT POSITION

M35 decrements the coolant spigot position one place. Decrementing the spigot position causes the spigot to move toward the spigot home position. The home position is designated as zero. If the current spigot position is 5 and M35 is executed, then the current spigot position will move to 4. The spigot home position for a horizontal mill places the spigot at the most positive Z axis location. Decrementing the spigot will raise the coolant stream direction.

M36 PALLET ROTATE

This is used on Horizontal mills only. This allows the pallet to rotate on the single pallet when the pallet part ready button on the front control panel is lit.

M39 ROTATE TOOL TURRET

The M39 code is used to rotate the tool turret without performing a tool change. This is not normally required but is useful for diagnostic purposes or to recover from a tool changer crash. Remember that the pocket facing the spindle must always be empty for a tool change. This M code may be useful to move an empty pocket to face the spindle.

M41 LOW GEAR OVERRIDE

The M41 code is used to override the spindle gear implied by the Snnn command. With M41, the spindle gear will always be low. If the speed commanded is above the low gear limit, the spindle speed will be the low gear limit. This M code does not turn the spindle on or off. If the spindle was turning before this command, it will be started again. If it was stopped before this command it will be left off. M41 is ignored if there is no gear box.

M42 HIGH GEAR OVERRIDE

The M42 code is used to override the spindle gear implied by the Snnn command. With M42, the spindle gear will always be high. Note that this may reduce the torque at the tool. This M code does not turn the spindle on or off. If the spindle was turning before this command, it will be started again. If it was stopped before this command it will be left off. M42 is ignored if there is no gear box.

M51-M54 OPTIONAL USER M ON

The M51 thru M54 codes are optional for user interfaces. They will activate one of relays 17 through 24 and leave it active. These are the same relays used for M21-M28. Use M61-M68 to turn these off. The RESET button will turn off all of these relays. When Through the Spindle Coolant is enabled, M54 is disabled.

M61-M64 OPTIONAL USER M OFF

The M61 thru M68 codes are optional for user interfaces. They will deactivate one of relays 17 through 24. These are the same relays used for M21-M28. When Through the Spindle Coolant is enabled, M64 is disabled.

M75 SET G35 OR G136 REFERENCE POINT

This code is used to set the reference point used for G35 and G136. It must be used after a motion which is terminated with the skip function.

M76 DISABLE DISPLAYS

This code is used to disable the updating of the screen displays during high speed machining.

M77 ENABLE DISPLAYS

This code is used to enable the updating of the screen displays at the end of high speed machining.

M78 ALARM IF SKIP SIGNAL FOUND

This code is used to generate an alarm if the previous skip function actually got the skip signal. This is usually used when a skip signal is not expected and may indicate a probe crash. This code can be placed in a block with the skip function or in any subsequent block. The skip functions are G31, G36, and G37.

M79 ALARM IF SKIP SIGNAL NOT FOUND

This code is used to generate an alarm if the previous skip function did not actually get the skip signal. This is usually done when the absence of the skip signal means a positioning error of a probe. This code can be placed in a block with the skip function or in any subsequent block. The skip functions are G31, G36, and G37.

M82 TOOL UNCLAMP

This code is used to release the tool from the spindle. It is not normally needed as tool change operations do this automatically and a manual TOOL RELEASE button is available to the operator. THIS M CODE IS NOT RECOMMENDED FOR USE AS THE TOOL WILL BE DROPPED FROM THE SPINDLE AND MAY DAMAGE THE TOOL, THE MACHINE, OR YOUR SETUP.

M86 TOOL CLAMP

This code will clamp a tool into the spindle. It is not normally needed as tool change operations do this automatically and a manual TOOL RELEASE button is available to the operator.

M88 THRU SPINDLE COOLANT ON

This code is used to turn on the Through the Spindle Coolant (TSC) option. When M88 is called, the spindle will stop, then the TSC pump will turn on, and the spindle will restart.

M89 THRU SPINDLE COOLANT OFF

This code is used to turn off the Through the Spindle Coolant (TSC) option. When M89 is called, the spindle will stop, the TSC pump will shut off.

M96 CONDITIONAL LOCAL BRANCH WHEN DISCRETE INPUT SIGNAL IS 0

- P Block to branch to when conditional test succeeds
- Q Discrete input to test, 0..31

This code is used to test a discrete input for 0 status. When this block is executed and the input signal specified by Q is 0, a branch to the block specified by P is performed. A Pnnnn code is required and must match a line number within the same program. The Q value must be in the range of 0 to 31. These correspond to the discrete inputs found on the diagnostic display page with the upper left being input 0 and the lower right being 31. Q is not required within the M96 block. The last specified Q will be used. This command stops the lookahead queue until the test is made at runtime. Since the lookahead queue is exhausted, M96 cannot be executed when cutter compensation is invoked. M96 cannot be executed from a main DNC program. If you wish to use M96 in DNC, it must be in a resident subroutine called from the DNC program.

The following is an M96 example:

```
N05 M96 P5 Q8 (TEST INPUT DOOR S, UNTIL CLOSED);  
N10 (START OF SOME PROGRAM LOOP);
```

```
.(PROGRAM THAT MACHINES PART);
```

```
N85 M21 (EXECUTE AN EXTERNAL USER FUNCTION)  
N90 M96 P10 Q27 (LOOP TO N10 IF SPARE INPUT IS 0);  
M95 M30 (IF SPARE INPUT IS 1 THEN END PROGRAM);
```

M97 LOCAL SUB-PROGRAM CALL

This code is used to call a subroutine referenced by a line N number within the same program. A Pnnnn code is required and must match a line number within the same program. This is useful for simple subroutines within a program and does not require the complication of a separate program. The subroutine must still be ended with an M99. An L count on the M97 block will repeat the subroutine call that number of times.

M98 SUB PROGRAM CALL

This code is used to call a subroutine. The **Pnnnn** code is the number of the program being called. The **Pnnnn** code must be in the same block. The program by the same number must already be loaded into memory and it must contain an M99 to return to the main program. An **L** count can be put on the line containing the M98 and will cause the subroutine to be called **L** times before continuing to the next block.

M99 SUB PROGRAM RETURN OR LOOP

This code is used to return to the main program from a subroutine or macro. It will also cause the main program to loop back to the beginning without stopping if it is used in other than a subprogram without a **P** code. If an M99 **Pnnnn** is used, it will cause a jump to the line containing **Nnnnn** of the same number.

M99 **Pnnnn** in the HAAS control varies from that seen in FANUC compatible controls. In FANUC compatible controls M99 **Pnnnn** will return to the calling program and resume execution at block **N** specified in **Pnnnn**. For the HAAS control, M99 will NOT return to the calling program, but instead will jump to block **N** specified in **Pnnnn** in the current program.

You can simulate FANUC behavior by using the following code.

calling program:	HAAS	FANUC
	00001	00001

	N50 M98 P2	N50 M98 P2
	N51 M99 P100	...
	...	N100 (continue here)
	N100 (continue here)	...
	...	M30
	M30	
subroutine:	00002	00002
	M99	M99 P100

If you have macros, you can use a global variable and specify a block to jump to by adding **#nnn=dddd** in the subroutine and then using **M99 P#nnn** after the subroutine call. There are many ways to jump conditionally after a M99 return when using macros.

5. SUBROUTINES

One of the more important programming features of a CNC is called subroutines. Subroutines allow the CNC programmer to define a series of commands which might be repeated several times in a program and, instead of repeating them many times, they can be "called". A subroutine call is done with M97 or M98 and a Pnnnn. The P code is the same as the O number of the subroutine to be called.

It is important to note that there is little difference between the main program and the subroutines. In the LIST PROG display, they all appear as numbered programs. When starting execution of a program, the LIST PROG display is used to select the MAIN program and any subroutines used are called from within the main program.

Local subroutines can be used with the M97. This can be even easier to use than the M98 because the subroutine is part of a single main program without the need to define a different Onnnn program. With local subroutines, you can code an M30 for the end of your main program followed by a line number and a subroutine that ends with an M99.

The subroutine call causes the blocks in the subroutine to be executed just as if they were included in the main program. In order to return control to the main program, subroutines must end with an M99.

Another very important feature of subroutines is that the M98 "call" block may also include an L or repeat count. If there is an L, the subroutine call is repeated that number of times before the main program continues with the next block.

The most common use of subroutines is in the definition of a series of holes which must be first center drilled, peck drilled, tapped, and chamfered. If a subroutine is defined that consists only of the X-Y position of the holes, the main program can call that subroutine after defining a canned cycle to do each of the operations. Thus, the X-Y positions can be used several times and need not be repeated for each tool. An example follows:

```
00100 (MAIN PROGRAM FOR EXAMPLE OF SUBROUTINES) ;
G54 G00 G90 X0. Y0. ;
T01 M06 (CENTER DRILL) ;
G81 R0.2 Z-0.1 F20. L0 (NO OPERATION HERE, JUST DEFINE CANNED CYCLE) ;
S2000 M03 ;
M98 P0200 (CENTER DRILL EACH HOLE) ;
T02 M06 (PECK DRILL) ;
G83 R0.2 Z-1. F10. L0 (NO OPERATION HERE, JUST DEFINE CANNED CYCLE) ;
S1000 M03 ;
M98 P0200 (PECK DRILL EACH HOLE) ;
T03 M06 (TAP IN FLOATING HOLDER OR HARD TAP) ;
G84 R0.2 Z-1. F10. L0 (NO OPERATION HERE, JUST DEFINE CANNED CYCLE) ;
S200 (1/4-20) ;
M98 P0200 (TAP EACH HOLE) ;
T04 M06 (CHAMFER) ;
G81 R0.2 Z-0.1 F20. L0 (NO OPERATION HERE, JUST DEFINE CANNED CYCLE) ;
S2000 M03 ;
M98 P0200 (CHAMFER EACH HOLE) ;
G28 M30 (END OF MAIN PROGRAM) ;

00200 (SUBROUTINE EXAMPLE LISTING ALL HOLE POSITIONS) ;
X0. Y0. ;
X1. Y0. ;
X2. Y0. ;
```

X0. Y1. ;
X1. Y1. ;
X2. Y1. ;
X0. Y2. ;
X1. Y2. ;
X2. Y2. ;
M99 (END OF SUBROUTINE) ;

O0300 (EXAMPLE USING A LOCAL SUBROUTINE)
G54 G00 G90 X0. Y0. ;
G81 R0.2 Z-0.1 F20 L0 (NO OPERATION HERE, JUST DEFINE CANNED CYCLE);
S2000 M03;
M97 P0500 (CENTER DRILL EACH HOLE);
T02 M06 (PECK DRILL);
G83 R0.2 Z-1. F10. L0 (NO OPERATION HERE, JUST DEFINE CANNED CYCLE);
S1000 M03;
M97 P0500 (PECK DRILL EACH HOLE);
G28 M30 (END OF MAIN PROGRAM);

N0500 (LOCAL SUBROUTINE EXAMPLE LISTING ALL HOLE POSITIONS);
X0. Y0. ;
X1. Y0. ;
X2. Y0. ;
X0. Y1. ;
X1. Y1. ;
X2. Y1. ;
X0. Y2. ;
X1. Y2. ;
X2. Y2. ;
M99 (END OF SUBROUTINE);

6. TOOL FUNCTIONS (TN)

The **Tnn** code is used to select 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** and **M16** are used to start a tool change operation. The **Tnn** does not need to be in a block prior to the **M06** or **M16**; it can be in the same block.

NOTE: There is no **X** or **Y** motion required prior to performing a tool change and it would waste time in most cases to return **X** or **Y** to the home position. However, if your work piece or fixture is quite large, you may need to position **X** or **Y** prior to a tool change in order to prevent a crash between the tools and your fixture.

Other controls may require the programmer to position the **Z**-axis to machine zero prior to a tool change but this is not required with this control. You may command a tool change with **X**, **Y**, and **Z** in any position and the control will bring the **Z** up to the machine zero position prior to starting the tool change. The control will move the **Z** to a position above machine zero during a tool change but will never move below machine zero. At the end of a tool change, the **Z**-axis will be at machine zero.

The tool changer is an all electric fixed shuttle type. Tools are always loaded through the spindle and should never be installed directly in the carousel in order to avoid crashes. The pocket open to the spindle must always be empty in the retracted position.

The tool holders used are #40 taper, V flange, commonly called "CT 40". The tool changer is manufactured to hold either BT40 or CAT40 tools. They are NOT interchangeable.

For machines equipped for CAT40 tools, use a "45 degree, P40T type 1, inch threads" pull stud built to JMTBA standard "MAS 403-1982". This pull stud is characterized by a long shaft and a 45 degree shoulder under the head. Do not use the short shaft or pull studs with a sharp right angle (90 degree) head as they will not work and will cause serious damage.

For machines equipped for BT40 tools, use only HAAS pull studs (PN: 22-7165).

Tool holders and pull studs must be in good condition and tighten together with wrenches or they may stick in the spindle taper. Clean the tool tapers with a lightly oiled rag to leave a film to prevent rusting. Tools that make a loud bang when being released indicate a problem and should be checked before serious damage to the shuttle occurs. When the TOOL RELEASE button is pressed the tool should be pushed out of the spindle by a small amount (approximately .07 Inch). This is an indication that the pull stud is correctly touching the release mechanism.

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

After POWER UP/RESTART and ZERO RET, the control will insure that the tool changer is in a normal position. To load a new tool, select MDI mode, put the tool in the spindle with the TOOL RELEASE button and then push the ATC FWD or ATC REV and the machine will put the tool in the carousel. Use the CURNT COMDS display to see what tool is currently in the spindle.

NOTE: Tool changer goes to tool #1 first, then to tool designated in Setting 81.

To manually select another tool, use the "ATC FWD" or "ATC REV" key while in the MDI mode. To select a tool other than an adjacent one, enter the "T" number first. That is: "T8" "ATC FWD" will place tool 8 in the spindle.

If the shuttle should become jammed, the control will automatically come to an alarm state. To correct this, push the EMERGENCY STOP button and remove the cause of the jam. Push the RESET key to clear any alarms. Push the ZERO RET and the AUTO ALL AXES keys to reset the Z-axis and tool changer. **CAUTION!** Never put your hands near the tool changer when powered unless the EMERGENCY STOP button is pressed first.

The tool changer is protected by fuse FU5, located on the POWER PCB. It might be blown by an overload or jam of the tool changer. Operation of the tool changer can also be interrupted by problems with the tool clamp/unclamp and the spindle orientation mechanism.

There are some other M codes which will also cause tool operations to occur:

- M19 Will orient the tool for special user functions
- M39 Will rotate the tool turret without changing tools
- M82 Will unclamp the tool (be careful, it will fall!)
- M86 Will clamp the tool

7. SPINDLE SPEED FUNCTIONS

7.1 SPINDLE SPEED COMMANDS

Spindle speed functions are controlled primarily by the **S** address code. The **S** address specifies RPM in integer values from 1 to maximum spindle speed (Parameter 131). **NOT TO BE CHANGED BY USER!**

Speeds from S1 to the Parameter 142 value (usually 1200) will automatically select low gear and speeds above Parameter 142 will select high gear. Two **M** codes, M41 and M42 can be used to override the gear selection. M41 for low gear and M42 for high gear. Low gear operation above S1250 is not recommended. High gear operation below S100 may lack torque or speed accuracy.

If there is no gear box in your machine (VF-0) the gear box is disabled by parameters, it is always in high gear, and M41 and M42 commands are ignored.

Three **M** codes are used to start and stop the spindle. M03 starts the spindle clockwise, M04 starts the spindle counterclockwise, and M05 stops the spindle.

Note that only one **M** code is allowed in a block. This means that if you wish to override the gear with M41 or M42, you must put the **Snnnn** and M41 (or M42) in one block and the M03 (or M04) in the next block. The **Snnn** should always be in the same block as the M41 or M42 as an unneeded double gear change might otherwise be performed.

7.2 RIGID TAPPING CONTROL OF SPINDLE

Rigid tapping is an option which can be installed on this machine at the factory. It is enabled with the Parameter 57 "Rigid Tap" flag. When installed and enabled, it changes the way G74 and G84 work and a floating tap holder is not needed for these **G** codes. In addition, if the "REPT RIG TAP" flag in Parameter 57 is set, every repetition of a tapping operation will control the orientation of the spindle so that the tapping is repeatable.

Rigid tapping allows the use of a tap without a floating tap holder. Pitch control is within 0.0005 inch. Bottom depth control is +/-0.020 inch and repeatability is +/-0.005 inch. Rigid tapping will operate from 100 to 2000 RPM and up to 100 inches per minute feed. Bottom depth control is better at lower speeds and low gear. Thread pitch can be from 5 to 100 TPI.

The pitch of a tapped hole is defined by the ratio between the feed rate and spindle speed. When rigid tapping is selected, these two must be set exactly. An encoder mounted with the spindle tracks the position of the spindle and the Z-axis is moved precisely to match the pitch of the thread. If the repeatable option is selected, a position pulse from the encoder is used to synchronize the starting of the Z motion with the position of the spindle.

Note that with G74 and G84, you do not ever need to use M03, M04, or M05. These canned cycles start and stop the spindle automatically. This applies to using normal or rigid tapping.

The rigid tapping option requires additional hardware be added to the spindle head assembly and a different electronics card put into the control. When this option is installed, the second page of diagnostic data will show the actual spindle speed. See "Tapping with the VF Series CNC Mill" for more information.

8. PROGRAMMING EXAMPLES

8.1 G81 DRILLING CANNED CYCLE

FORMAT:

G81 Z- ___ F ___ R ___

Z = Position of the bottom of the hole being drilled.

F = Feed rate in inches per minute.

R = Reference plane, or a position placed above Z0.

NOTE: The Z, F, and R codes are required data for all canned cycles.

NOTE: The optional X and Y can be included in the canned cycle line. In most cases, this would be the location of the first hole to be drilled.

The following is the program to drill through the aluminum plate in Figure 8-1:

```

T1 M06
G00 G90 G54 X1.125 Y-1.875 S2500 M03
G43 H01 Z.1
G81 Z-.35 F15. R.1
X2.0
X3.0 Y-3.0
X4.0 Y-5.625
X5.250 Y-1.375
G00 G80 Z1.0
G28
M30
    
```

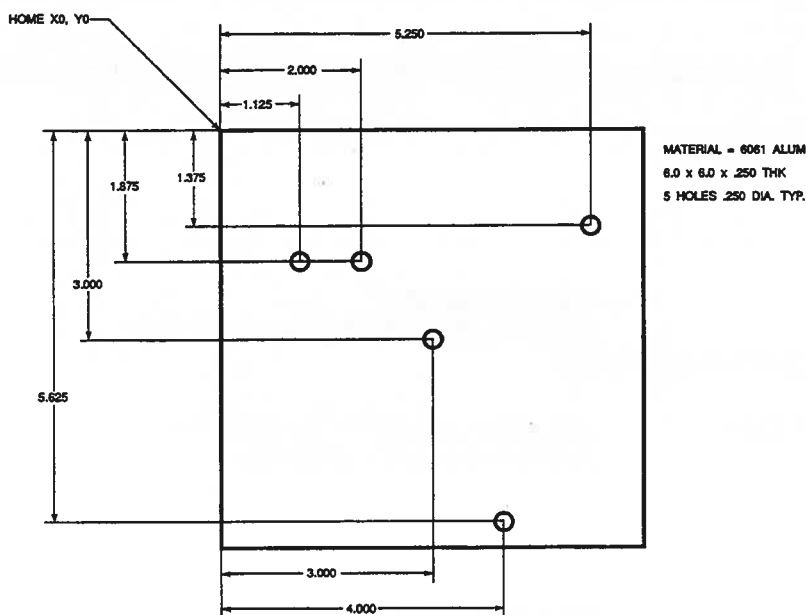


Fig. 8-1 Canned cycle programming example using aluminum block.

8.2 G82, G83, G84 CANNED CYCLES**G82 FORMAT :**

G82 Z - ____ F ____ R ____ P ____;

These are the required codes for **spot drilling**.

P = Dwelling time in milliseconds at the bottom of the Z-axis move.

EX. P300 is approximately 1/3 second.

G83 FORMAT -

G83 Z - ____ F ____ R ____ Q ____;

These are the required codes for **peck drilling**.

Q = Incremental pecking amount in minus Z direction.

EX. Q.200 in a G83 line will peck toward the specified Z depth, taking .200 per peck until final depth is reached.

G84 FORMAT -

G84 Z - ____ F ____ R ____

These are the codes required for **tapping**.

No new codes to review.

NOTE: The most important item to be aware of during tapping is the speed and feed calculation.

FEED FORMULA: Spindle Speed/Threads per inch on the tap = Feed Rate in inches/min.

CANNED CYCLE PROGRAM**(HELPFUL NOTES)**

```

%
O1234                                (Exercise program)
T1 M06                                (TOOL #1 IS A .5 X 90 degree spot drill)
G00 G90 G54 X.565 Y-1.875 S1275 M03
G43 H01 Z.1 M08
G82 Z-.175 F10. R.1 P300              >(90 degree spot drill, the depth is)
X1.115 Y-2.750                        (half of the chamfer diameter)
X3.365 Y-2.875
X4.188 Y-3.313
X5.0 Y-4.0
G00 G80 Z1.0 M09
T2 M06                                (Tool #2 IS A .3125 stub drill)
G00 G90 G54 X.565 Y-1.875 S2500 M03
G43 H02 Z.1 M08

```

```

G83 Z-.620 F15. R.1 Q.175      >(Drill point is 1/3 of the drill diameter.)
X1.115 Y-2.750
X3.365 Y-2.875
X4.188 Y-3.313
X5.0 Y-4.0
G00 G80 Z1.0 M09
T3 M06                          (Tool #3 is a 3/8-16 tap)
G00 G90 G54 X.565 Y-1.875 S900 M03
G43 H03 Z.2 M08
G84 Z-.600 F56.25 R.2          >(900 RPM divided by 16 TPI = 56.25 IPM)
X1.115 Y-2.750
X3.365 Y-2.875
X4.188 Y-3.313
X5.0 Y-4.0
G00 G80 Z1.0 M09
G28 G91 Y0 Z0
M30
%
```

8.3 SUBPROGRAMS AND CANNED CYCLES

After reviewing the canned cycle, we can get a good idea of the amount of lines of code required to produce the five holes. The best way to conserve on program space and programming time is to use a subprogram. We can do this by grouping the X and Y locations of the holes into a separate program and then calling up this program when we need to tell a canned cycle the X,Y coordinates.

Instead of writing the X,Y locations once for each tool, we can write the X,Y locations once for any number of tools.

The canned cycle program that we reviewed on the previous page could use some constructive rearranging.

<pre> % O1234 (Example program) T1 M06 G00 G90 G54 X.565 Y-1.875 S1275 M03 G43 H01 Z.1 M08 G82 Z-.175 F10. R.1 P300 M98 P1000 G00 G80 Z1.0 M09 T2 M06 G00 G90 G54 X.565 Y-1.875 S2500 M03 G43 H02 Z.1 M08 G83 Z-.620 F15. R.1 Q.175 M98 P1000 G00 G80 Z1.0 M09 T3 M06 G00 G90 G54 X.565 Y-1.875 S900 M03 G43 H03 Z.2 M08 G84 Z-.600 F56.25 R.2 M98 P1000 G00 G80 Z1.0 M09 G28 G91 Y0 Z0 M30 %</pre>	<pre> % O1000 (X,Y LOC. SUB) X 1.115 Y-2.750 X 3.365 Y-2.875 X 4.188 Y-3.313 X 5.0 Y-4.0 M99 %</pre>
---	--

8.4 SUBPROGRAMS WITH MULTIPLE FIXTURES

So far we have learned that using subprograms with canned cycles can save programming time and help reduce coordinate input error. Let's take this one step further. There are six vises mounted on the table. Each of these vises will use a new X,Y zero. They will be called up in the program as G54 through G59. The machine will have to be told where each of the vises is located on the table. By using an edge finder or an indicator, the zero point on each part can be established. Use the part zero set key in the work coordinate offset page to record each X,Y location. Once the X,Y zero position for each vise is in the offset page, the programming can begin.

By looking at the next page, we can get a good idea of 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.

%	%
O2000	O3000
T1 M06	X0 Y0
G00 G90 G54 X0 Y0 S1500 M03	G83 Z-1.0 F15. R.1 Q.2
G43 H01 Z.1 M08	G00 G80 Z.2
M98 P3000	M99
G55	%
M98 P3000	
G56	
M98 P3000	
G57	
M98 P3000	
G58	
M98 P3000	
G59	
M98 P3000	
G00 Z1.0 M09	
G28 G91 Y0 Z0	
M30	
%	

Figure 8-2 represents a multiple-fixture setup. Each vise will have an absolute zero once it is specified in the program. This is done by using G54 through G59 and G110 through G129, a total of 26 possible.

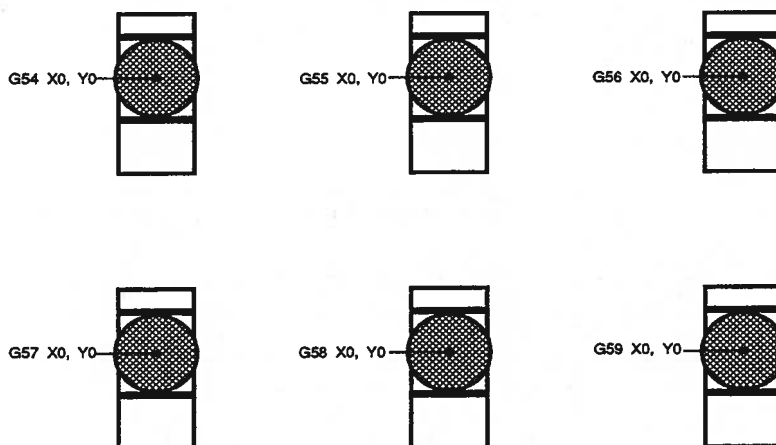


Fig.8-2 Multiple fixture setup.

8.5 LOOPING CANNED CYCLES

Below is an example of a program using a drilling canned cycle that is incrementally looped. Compare the grid plate drawing in Figure 8-3 to the program below.

```

%
O3400                                (Drilling grid plate)
T1 M06
G00 G90 G54 X1.0 Y-1.0 S2500 M03
G43 H01 Z.1 M08
G81 Z-1.5 F15. R.1
G91 X1.0 L9                            (Or stay in G91 and repeat Y-1.0)
G90 Y-2.0
G91 X-1.0 L9
G90 Y-3.0
G91 X1.0 L9
G90 Y-4.0
G91 X-1.0 L9
G90 Y-5.0
G91 X1.0 L9
G90 Y-6.0
G91 X-1.0 L9
G90 Y-7.0
G91 X1.0 L9
G90 Y-8.0
G91 X-1.0 L9
G90 Y-9.0
G91 X1.0 L9
G90 Y-10.0
G91 X-1.0 L9
G00 G90 G80 Z1.0 M09
G28 G91 Y0 Z0
M30
%
```

NOTE: The sequence of drilling used here is designed to save time and to follow the shortest path from hole to hole.

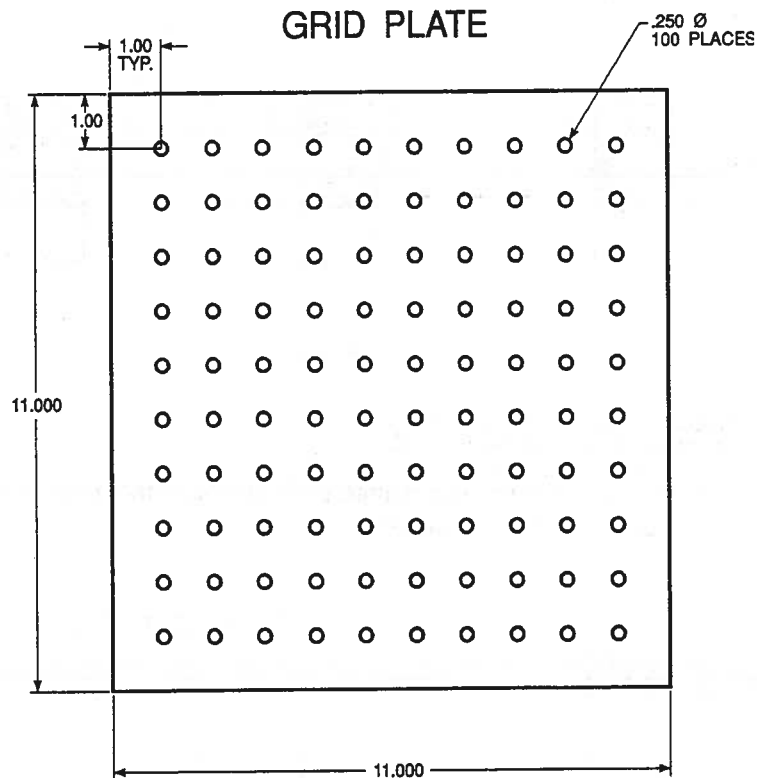


Fig. 8-3 Grid plate for multiple-fixture subprogram exercise.

8.6 MODIFYING CANNED CYCLES

In this section we will cover canned cycles that have to be customized in order to make the programming of difficult parts easier. In result, making the machining process more efficient.

Using G98 and G99 to clear clamps:

For example, we have a square part being held to the table with one inch tall table clamps. We need to write a program to clear the table clamps.

```

%
O4500
T1 M06
G00 G90 G54 X1.0 Y-1.0 S3500 M03
G43 H01 Z1.125 M08
G81 G99 Z-1.500 F20. R.05
X2.0 G98 ( Will return to starting point after executing cycle )
X6.0 G99 ( Will return to reference plane after executing cycle )

```

```
X8.0  
X10.0  
X12.0 G98  
X16.0 G99  
X18.0 G98  
G00 G80 Z2.0 M09  
G28 G91 Y0 Z0  
M30  
%
```

X,Y Plane Obstacle Avoidance In A Canned Cycle:

So far we have learned how G98 and G99 can be used to avoid an obstacle in the Z-axis. There is also a way to avoid an obstacle in the X,Y plane during a canned cycle by placing an L0 in a canned cycle line, we can tell the control to make an X,Y move without executing the Z-axis canned operation.

For example, we have a six inch square aluminum block, with a one inch by one inch deep flange on each side. The print calls for two holes centered on each side of the flange. We need to write a program to avoid each of the corners on the block.

```
%  
O4600  
(X0,Y0 is at the top left corner)  
(Z0 is at the top of the part)  
T1 M06  
G00 G90 G54 X2.0 Y-.5 S3500 M03  
G43 H01 Z-.9 M08  
G81 Z-2.0 F15. R-.9  
X4.0  
X5.5 L0 (angular corner avoidance)  
Y-2.0  
Y-4.0  
Y-5.5 L0  
X4.0  
X2.0  
X.5 L0  
Y-4.0  
Y-2.0  
G00 G80 Z1.0 M09  
G28 G91 Y0 Z0  
M30  
%
```

8.7 SPECIAL CANNED CYCLES

In this section, we will cover the special canned cycles that the Haas control offers. These canned cycles are used in conjunction with other drilling, boring, and tapping cycles.

G70 = BOLT HOLE CIRCLE

G71 = BOLT HOLE ARC

G72 = BOLT HOLES ALONG AN ANGLE

The sample program below will show the format for using a G70 to drill a three inch diameter bolt hole pattern combined with a G81 drilling canned cycle.

```
%
O5000
T1 M06
G00 G90 G54 X0 Y0 S1500 M03
G43 H01 Z.1 M08
G70 I1.5 J0 L8 G81 Z-1.0 F15. R.1
G00 G80 Z1.0 M09
G28 G91 Y0 Z0
M30
%
```

RULES FOR BOLT PATTERN CANNED CYCLES:

1. The tool must be placed at the center of the bolt pattern before the canned cycle execution. The center is usually X0, Y0.
2. The J code is the angular starting position and is always 0 to 360 degrees counterclockwise from the three o'clock position.

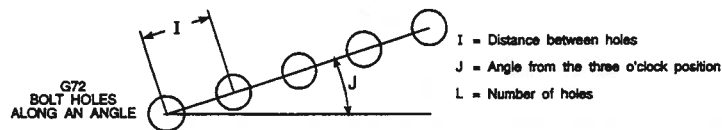
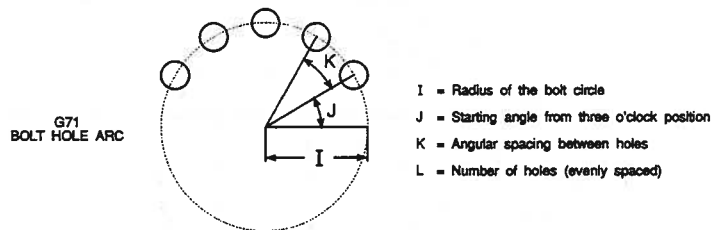
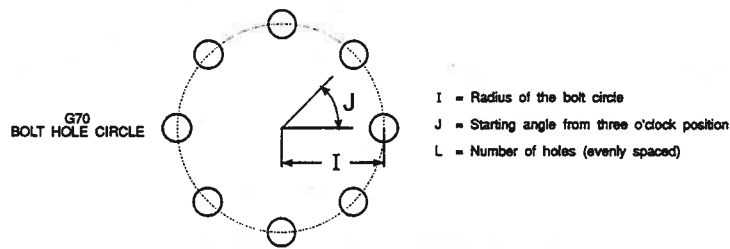
In the case of conflicting address codes, You can specify a drilling cycle prior to the block that invokes the special canned cycle. For instance:

```
%
O5000
T1 M06
G00 G90 G54 X0 Y0 S1500 M03
G43 H01 Z.1 M08
G83 R.1 Z-1.0 I.25 J.03 K.15 F15. L0 (L0 PREVENTS DRILLING AT CENTER)
G70 I1.5 J0 L8
G00 G80 Z1. M09
G28 G91 Y0 Z0
M30
%
```

In the case of conflicting address codes, You can specify a drilling cycle prior to the block that invokes the special canned cycle. For instance:

```
%
O5000
T1 M06
G00 G90 G54 X0 Y0 S1500 M03
G43 H01 Z.1 M08
G83 R.1 Z-1.0 I.25 J.03 K.15 F15. L0 (L0 PREVENTS DRILLING AT CENTER)
G70 I1.5 J0 L8
G00 G80 Z1. M09
G28 G91 Y0 Z0
M30
%
```

[See following page for illustrations of G70, G71, and G72.]



8.8 CIRCULAR INTERPOLATION AND CUTTER COMPENSATION

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

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°), an I or a J with a value must be specified. The circle section illustration below will describe the different sections of a circle.

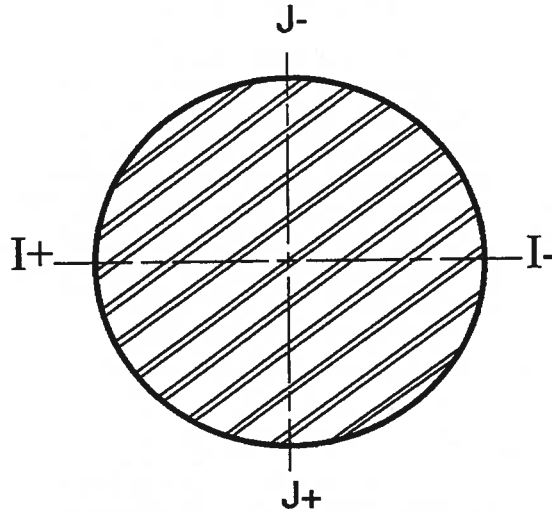
By using cutter compensation in this section we, the programmers, will be able to shift a cutter by the amount of the cutter radius and be able to program a profile or a contour to the exact print dimensions. By shifting the cutter radius, the programming time and the likelihood of calculation error is reduced.

Before we get into circular interpolation and how it is used, below are a few rules about cutter compensation that have to be closely followed in order to perform successful machining operations. Always refer to these rules when programming!

- 1) Cutter compensation must be turned ON during a G00 or G01 X,Y move that is equal to or greater than the cutter radius, or the amount being compensated for.
- 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.

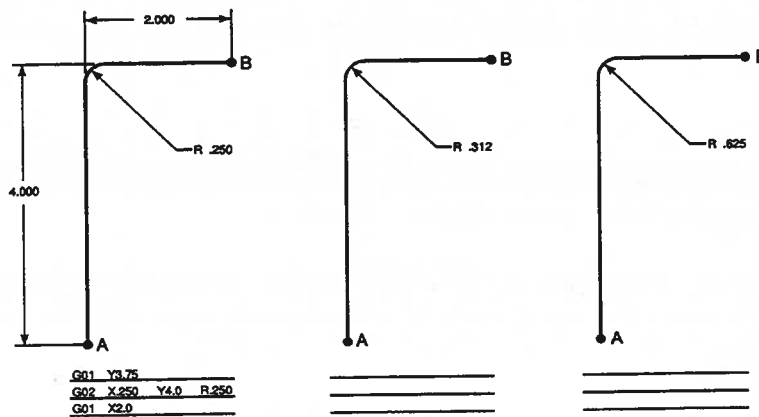
- 3) In most machines, during cutter compensation, a linear X,Y move that is smaller than the cutter radius may not work. (Setting 58 - set to Fanuc - for positive results.)
- 4) Cutter compensation cannot be turned ON or OFF in a G02 or G03 arc movement.

CIRCLE SECTIONS



DO THE FOLLOWING EXERCISE FOR PRACTICE.

WRITE THE CODE NECESSARY TO GET FROM POINT A TO POINT B



FIRST MOVE: Overall dimension and subtract the given radius.

SECOND MOVE: Add radius to current X and Y values. Always program the ending points in circular interpolation.

THIRD MOVE: Linear move to the next point.

WRITE A PROGRAM TO PRODUCE THE PART SHOWN BELOW.

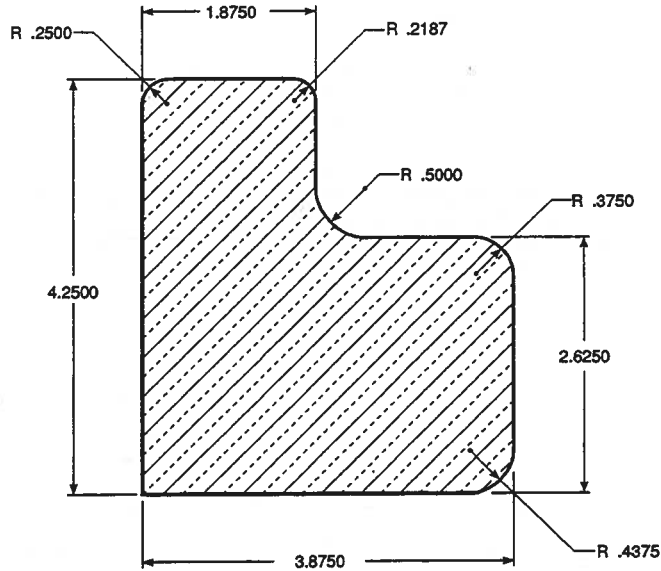


Fig. 8-4 Part for G02, G03, and R practice.

The correct program for Figure 8-4 is as follows:

```

%
O600
T1 M06 (Tool is a .250 diameter two-flute end mill)
G00 G90 G54 X-.2 Y-.2 S5000 M03
G43 H01 Z.1 M08
Z-1.0
G41 D01 X0 (Turn ON cutter compensation, .200 move)
G01 Y4.0 F25.
G02 X.250 Y4.250 R.250
G01 X1.6562
G02 X1.875 Y4.0313 R.2187
G01 Y3.125
G03 X2.375 Y2.625 R.500
G01 X3.5
G02 X3.875 Y2.25 R.375
G01 Y.4375
G02 X3.4375 Y0 R.4375
G01 X-.1
G00 G40 X-.3 (Turn OFF cutter compensation, .200 move)
Z1.0 M09
G28 G91 Y0 Z0
M30
%
```

The following illustration shows how the tool path is calculated for the cutter compensation.

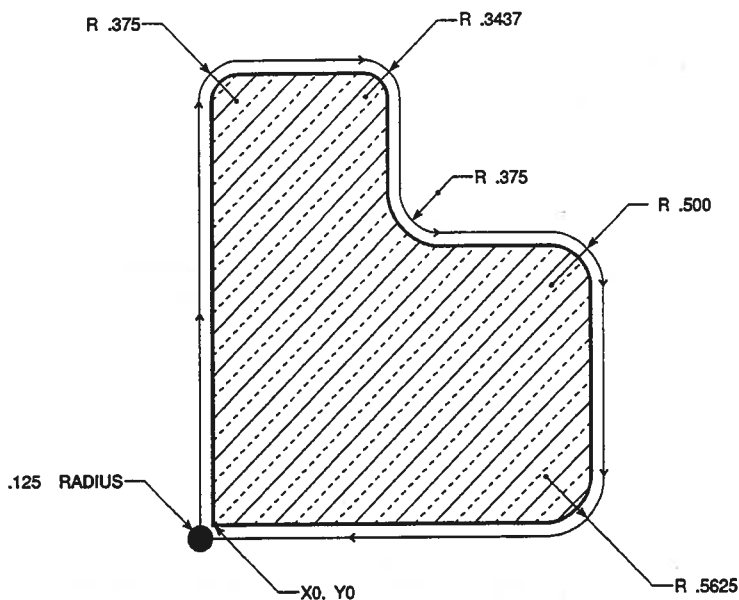


Fig. 8-5 Programming exercise showing tool path.

This program uses no cutter compensation. Tool path is programmed to centerline of the cutter. This is also the way the control calculates for cutter compensation. Note the machine tool path in Figure 8-5.

```

%
O6100
T1 M06
G00 G90 G54 X-.125 Y-.2 S5000 M03
G43 H01 Z.1 M08
G01 Z-1.0 F50.
Y4.125 F25.
G02 X.250 Y4.375 R.375
G01 X1.6562
G02 X2.0 Y4.0313 R.3437
G01 Y3.125
G03 X2.375 Y2.750 R.375
G01 X3.5
G02 X4.0 Y2.25 R.5
G01 Y.4375
G02 X3.4375 Y-.125 R.5625
G01 X-.2
G00 Z1.0 M09
G28 G91 Y0 Z0
M30
%
```

8.9 PROGRAMMING USING I AND J

Most contour machining will use a radius value **R** for circular interpolation moves less than 360°. An **I** and **J** can also be used in the place of **R**, but this can be more confusing at times. The **I** and **J** are signed distances from the starting point to the center of the circle.

Referring back to the previous section, we can see the program below using the **R** or the **I** and **J**:

Using R:	Using I and J:
G01 Y3.750	G01 Y3.750
G02 X.250 Y4.0 R.250	G02 X.250 Y4.0 I.250 JO
G01 X2.0	G01 X2.0

Note the +1.250 move and compare it with the "Circle Sections" illustration shown earlier.

NOTE: The G02 or G03 line will always need the X,Y end points, whenever R is used or I and J.

Programming a complete 360° circle can only be done by using an **I** or a **J**. For example: we have a one inch diameter hole and want to open it up to one and a half inches. The cutter is .750 diameter. If we took the finished hole diameter and subtracted the cutter diameter, we would have .750 left over — .375 each side.

The start move would be to position to the center of the hole — most likely X0,Y0. The first cutting move would be to move .375 in any direction. X+, X-, Y+, Y-. Let's go with the X+ direction.

```
G01 X.375
G03 I-.375 (An I or J will always be a radial value.)
G01 X0
```

NOTE: The start point is also the end point for a 360° move.

NOTE: If we decided to make the first move in the Y+ direction, the G03 line would contain a J-.375 move.

8.10 CIRCULAR POCKET MILLING

The Haas control has included in its software a Yasnac style circular pocket milling program (G12 clockwise circular pocket, G13 counterclockwise pocket). These G codes imply the use of cutter compensation, i.e., a G41 or G42 is not required to be stated in the program line. However, a D ___ offset number for cutter radius or diameter is required for the ability to adjust the circle diameter.

In this section, we will cover the G12 and G13 format, as well as the different ways these programs can be written for many various applications:

SINGLE PASS: Using **I** only.

APPLICATIONS: One-pass counterboring; rough and finish pocketing of smaller holes; I.D. keyway cutting; "O"-ring grooves.

MULTIPLE PASS: Using **I**, **K**, and **Q**.

APPLICATIONS: Multiple-pass counterboring, rough and finish pocketing of large holes with cutter overlap.

MULTIPLE Z DEPTH PASS: Using I only, or, I, K, and Q. (G91 and L)

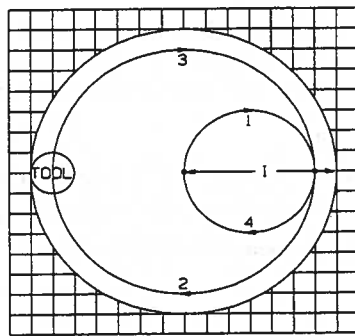
APPLICATIONS: Deep rough and finish pocketing; incremental Z depth stepping.

NOTE: The tool must be positioned at the center of the circle, either in a previous block or in the G12/G13 line by using X and Y.

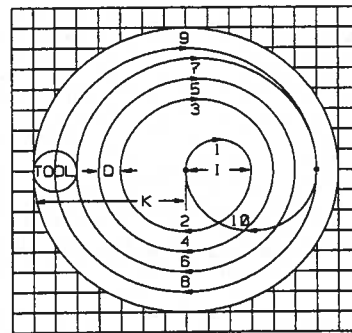
The figure below shows the tool path during the G12 and G13 cycles. One uses I only and the other uses I, K, and Q.

Circular Pocket Milling

G12

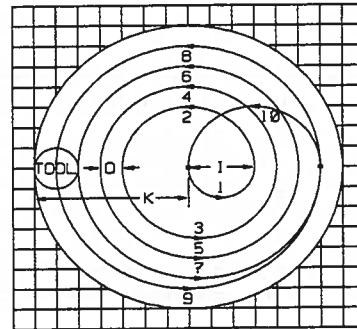
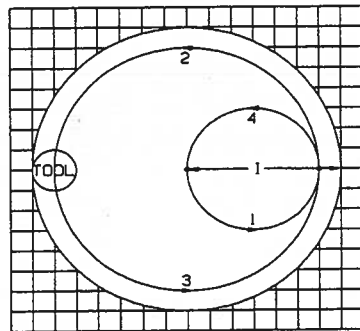


I ONLY



I, K, and Q

G13



PROGRAM LINE REQUIREMENTS:

- Z = Depth of cut or increment
- F = Feed Rate.
- I = Radius of first circle
- K = Radius of finished circle
- Q = Radius increment or cutter overlap
- D = Tool geometry offset number
- L = Loop count for incremental Z depth stepping

(Finished radius if no K specified)
 (If using I, K, and Q)
 (Must use with K)
 (Not required)
 (Optional)

EXAMPLE G13 single-pass using I only:

```

%
O2000                                (.500 entered in the Radius/Diameter offset column)
T1 M06                                (Tool #1 is a .500 diameter end mill)
G00 G90 G54 X0 Y0 S4000 M03
G43 H01 Z.1 M08
G13 Z-1.0 F20. I.500 D01              (Will complete a one-inch diameter hole one-inch deep)
G00 Z1.0 M09
G28 G91 Y0 Z0
M30
%
```

EXAMPLE G13 multiple-pass using I, K, and Q:

```

%
O3000                                (.500 entered in the Radius/Diameter offset column)
T1 M06                                (Tool #1 is a .500 diameter end mill)
G00 G90 G54 X0 Y0 S4000 M03
G43 H01 Z.1 M08
G13 Z-1.0 F20. D01 I.400 K1.5 Q.400
G00 Z1.0 M09
G28 G91 Y0 Z0
M30
%
```

This example will complete a three-inch diameter hole, one inch deep, with a cutter overlap of .100 thousandths of an inch.

EXAMPLE G13 Multiple-pass using I, K, Q, L, and G91:

```

%
O4000                                (.500 entered in the Radius/Diameter offset column)
T1 M06                                (Tool #1 is a .500 diameter end mill)
G00 G90 G54 X0 Y0 S4000 M03
G43 H01 Z.1 M08
G01 Z0 F10.
G13 G91 Z-.5 F20. D01 I.400 K2.0 Q.400 L4
G00 G90 Z1.0 M09
G28 G91 Y0 Z0
M30
%
```

The previous program uses G91 and an L count of 4. This cycle will execute a total of four times. The Z depth increment is .500. This is multiplied by the L count, making the total depth of this hole 2.000.

The G91 and L count can also be used in G13 "I only" line.

NOTE: If the geometry column has a value inserted within, the circular pocket milling cycle will automatically read the data, regardless of the D01 being present or not. The only effective way to cancel the cutter compensation for the pocket milling is to insert a D00 in the program line. This will bypass the value in the geometry column.

8.11 GENERAL PURPOSE POCKET MILLING

The general purpose pocket milling program is has been included in the Haas control. This program is used to mill irregular shapes and is capable of leaving islands and bosses within a contour. With the G150, there is a main program for technical input and a subprogram for contour definition.

G150 FORMAT EXAMPLE:

```
%
O4500
T1 M06
G00 G90 G54 X0 Y0 S3500 M03
G43 H01 Z.1 M08
G150 X__ Y__ Z__ F__ R__ Q__ I__ OR J__ K__ P4600 D__ G41 OR G42
G00 Z1.0 M09
G28 G40 G91 Y0 Z0
M30
%
```

```
%
O4600
G01 X__ Y__
X__
Y__
X__ Y__
M99
%
```

PROGRAM LINE REQUIREMENTS:

X = X-axis position of the starting hole
 Y = Y-axis position of the starting hole
 Z = Final depth of the hole
 F = Feed rate
 R = Reference plane
 Q = Incremental Z-axis cut depth per pass
 I = X-axis cut increment
 J = Y-axis cut increment
 K = finish cut allowance
 P = Subprogram number
 D = Geometry offset number
 G41 or G42 = Cutter compensation turn **ON**

The shape of the pocket to be cut must be defined by a series of motions within a subprogram. One of either I or J must be specified. If I is used, the pocket is cut from a series of strokes in the X-axis. If J is used, the pocket is cut from a series of strokes in the Y-axis. The value entered with the I or J will be the shift amount or cutter overlap. The K amount is the finishing allowance for the walls of the pocket.

The subprogram must define a closed area by a series of G01, G02, or G03 motions on X- and Y-axes, and must end with an M99. The only other codes that can be used in the subprogram are: G90, G91, I, J, R, X, and Y. Any other codes are ignored. This subprogram must not exceed 20 strokes.

NOTE: When defining the contour in the subprogram, the idea to keep in mind is to only connect the contour — not to return to the starting point.

G150 EXAMPLES -

4.0 x 4.0 x .500 DP. SQUARE POCKET:

```
%
O1000
T1 M06 (Tool #1 is a .500 diameter end mill)
G00 G90 G54 S2000 M03
G43 H01 Z.1 M08
G01 Z0 F10.
G150 X0 Y0 Z-.5 F10. R.1 Q.25 I.4 K.01 P500 D01 G41
G00 Z1.0 M09
G40 G28 G91 Y0 Z0
M30
%
```

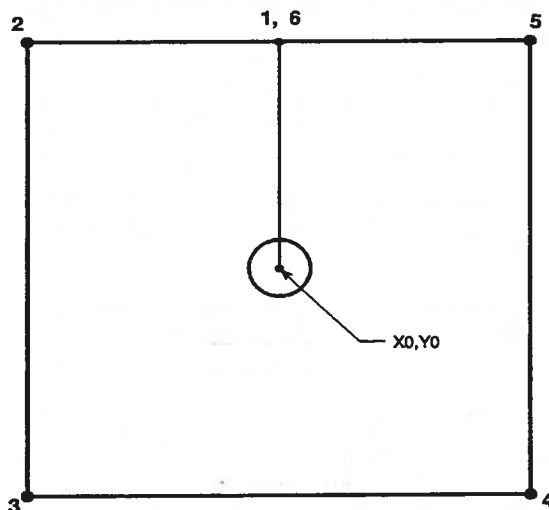


Fig. 8-6 Pocket milling exercise for G150 operation.

ABSOLUTE SUBPROGRAM:

```
%
O0500
G01 Y2.0
X-2.0
Y-2.0
X2.0
Y2.0
X0
M99
%
```

INCREMENTAL SUBPROGRAM:

```

%
O0500
G01 G91 Y2.0
X-2.0
Y-4.0
X4.0
Y4.0
X-2.0
G90
M99
%
```

SQUARE ISLAND:

```

%
O1000
T1 M06 (Tool is a .500 diameter end mill)
G00 G90 G54 S2500 M03
G43 H01 Z.1 M08
G01 Z0 F10.
G150 X1.0 Y1.0 Z-.5 F15. R.1 Q.25 I.4 K.01 P500 D01 G41
G00 Z1.0 M09
G40 G28 G91 Y0 Z0
M30
%
```

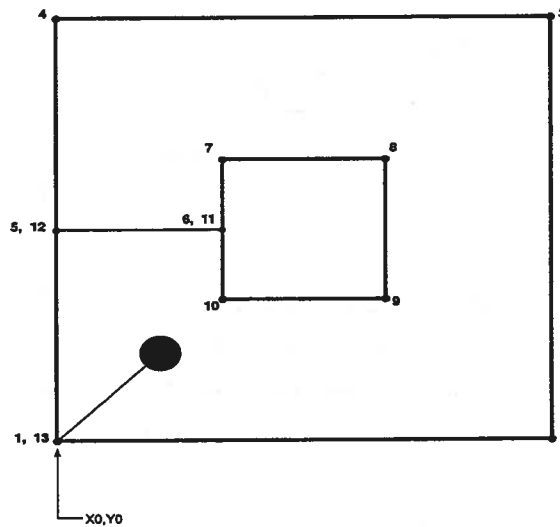


Fig. 8-7 Square island programming exercise using G150.

```

%
O0500
G01 X0 Y0
X6.0
Y6.0
X0
```

Y3.0
X2.0
Y4.0
X4.0
Y2.0
X2.0
Y3.0
X0
Y0
M99
%

ROUND ISLAND:

%
O1000
T1 M06 (Tool is a .500 diameter end mill)
G00 G90 G54 S2500 M03
G43 H01 Z.1 M08
G01 Z0 F10.
G150 X1.0 Y1.0 Z-.5 F15. R.1 Q.25 I.4 K.01 P500 D01 G41
G00 Z1.0 M09
G40 G28 G91 Y0 Z0
M30
%

%
O0500
G01 X0 Y0
X6.0
Y6.0
X0
Y3.0
X2.0
G02 I1.0
G01 X0
Y0
M99
%

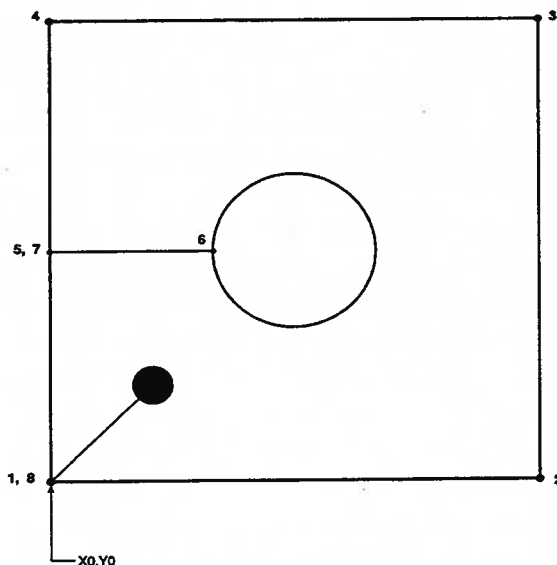


Fig. 8-8 Round island programming exercise using G150.

8.12 PROGRAMMABLE MIRROR IMAGE

Programmable mirror image can be turned on or off individually for any of the four axes. The two codes are non-modal, but the mirror status of each axis is modal. The bottom of the CRT screen will indicate when an axis is mirrored. These codes should be used in a command block without any other G codes and will not cause any axis movement. G101 will turn off mirror image for any axis listed in that block. The actual value given for the X, Y, Z, or A code has no effect and should be entered as zero value.

Example:

G101 X0 = Will turn on mirror image for the X-axis.
G100 X0 = Will turn off mirror image for the X-axis.

Most mirror image applications would consist of irregular pockets and contours and would most likely be set up in subprograms for convenience.

NOTE: The first pocket or contour will need to be run before the mirror image function can duplicate the geometry. After completion of the first item, a Z-axis clearance move should be made. Then, the mirror image should be turned on with an axis specification. The following line needs the coordinates of the starting location of the original pocket. The following line will feed to the required Z-axis depth, the next line would contain a subprogram call or a contour definition, and last, a positive Z-axis clearance move.

The pockets should be arranged around a given origin, usually described as X0,Y0.

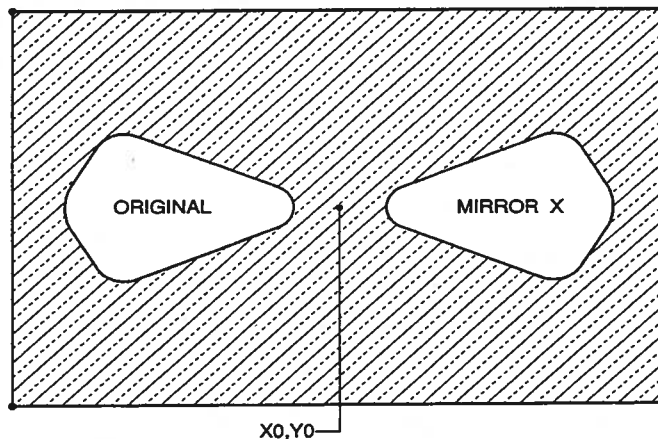


Fig. 8-9 Mirror image and pocket milling exercise.

NOTE: When milling a shape with X-Y motions, turning on MIRROR IMAGE for just one of the X and Y will change climb milling to conventional milling and/or conventional milling to climb milling. As a result, you may not get the type of cut or finish that was desired. Mirror image of both X and Y will eliminate this problem.

PROGRAM CODE FOR MIRROR IMAGE IN X-AXIS:

```

%
O3600                                     (Mirror image X-axis)
T1 M06                                    (Tool #1 is a .250 diameter end mill)
G00 G90 G54 X-.4653 Y.052 S5000 M03
G43 H01 Z.1 M08
G01 Z-.25 F5.
F20.
M98 P3601
G00 Z.1
G101 X0.
X-.4653 Y.052
G01 Z-.25 F5.
F20.
M98 P3601
G00 Z.1
G100 X0.
G28 G91 Y0 Z0
M30
%

```

```

%
O3601                                     (Contour subprogram)
G01 X-1.2153 Y.552
G03 X-1.3059 Y.528 R.0625
G01 X-1.5559 Y.028
G03 X-1.5559 Y-.028 R.0625
G01 X-1.3059 Y-.528
G03 X-1.2153 Y-.552 R.0625
G01 X-.4653 Y-.052
G03 X-.4653 Y.052 R.0625
M99
%

```

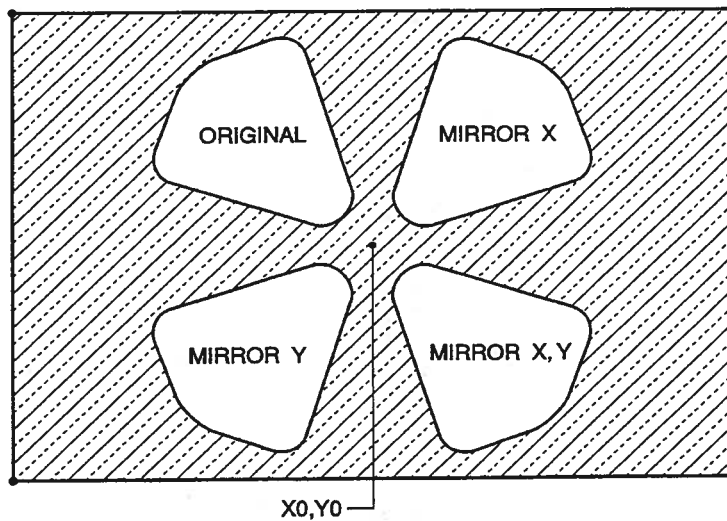


Fig. 8-10 Mirror image (X, Y, and X-Y) and pocket milling exercise.

PROGRAM CODE FOR MIRROR IMAGE IN THE X, Y, AND XY AXES:

```

%
O3700                                     (Mirror image X, Y, and XY axes)
T1 M06                                     (Tool #1 is a .250 diameter end mill)
G00 G90 G54 X-.2923 Y.3658 S5000 M03
G43 H01 Z.1 M08
G01 Z-.25 F5.
F20.
M98 P3701
G00 Z.1
G101 X0.                                   (Turn on mirror image X-axis)
X-.2923 Y.3658                             (Position to original coordinates)
G01 Z-.25 F5.                             (Feed to Z depth)
F20.                                       (Pocket feed rate)
M98 P3701                                   (Pocket contour subprogram call)
G00 Z.1                                     (Part clearance)
G100 X0.                                   (Cancel mirror image X-axis)
G101 Y0.                                   (Turn on mirror image Y-axis)
X-.2923 Y.3658
G01 Z-.25 F5.
F20.
M98 P3701
G00 Z.1
G100 Y0.                                   (Cancel mirror image Y-axis)
G101 X0. Y0.                               (Turn on mirror image X and Y axes)
X-.2923 Y.3658
G01 Z-.25 F5.
F20.
M98 P3701
G00 Z.1
G100 X0. Y0.                               (Cancel mirror image X and Y axes)
G28 G91 Y0 Z0
M30

```

```
O3701 (Contour subprogram)
G01 X-.469 Y1.2497
G03 X-.5501 Y1.2967 R.0625
G01 X-1.0804 Y1.12
G03 X-1.12 Y1.0804 R.0625
G01 X-1.2967 Y.5501
G03 X-1.2497 Y.469 R.0625
G01 X-.3658 Y.2923
G03 X-.2923 Y.3658 R.0625
M99
%
```

8.13 THREAD MILLING

We will use the following example and go through the thread milling procedures step-by-step to get the desired result:

DATA:

- I.D. Thread milling a 1.5 x 8 TPI hole.
- Using .750 diameter x 1.0 thread hob.
- Take the hole diameter 1.500.
- Subtract cutter diameter .750 = .750 Then divide by 2 = .375.

STEP 1: Within this space we need to turn on cutter compensation and ramp on to the circle to be machined.

STEP 2: Perform complete circle while simultaneously moving in the Z-axis the amount of one full pitch of the thread. This is called **helical interpolation**.

STEP 3: Ramp off the circle and turn off the cutter compensation.

NOTE: Always climb cut the cutter.

I.D. will be G03; **O.D.** will be G02. An **I.D.** right hand thread will move **up** in the Z-axis by the amount of one thread pitch. An **O.D.** right hand thread will move **down** in the Z-axis by the amount of one thread pitch.

PITCH = 1.0/Threads per inch

Example 1.0 divided by 8 TPI = .125

Cutter compensation cannot be turned off or on during an arc movement. A linear turn on and turn off movement must be made, either in the X- or Y-axis. This move will be the maximum compensation amount that can be adjusted.

I.D. THREAD MILLING

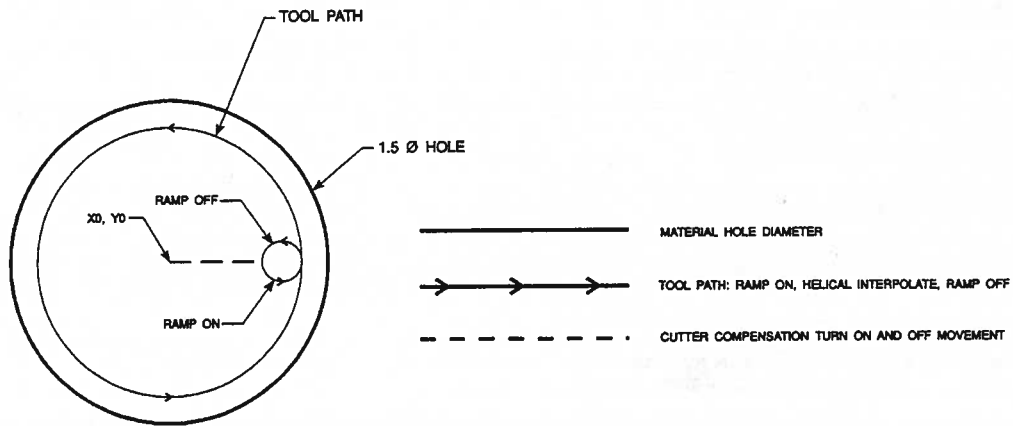


Fig. 8-11 Thread milling exercise

The code for Figure 8-11 is as follows:

```

%
O2300                                     (Thread milling 1.5 diameter x 8 TPI)
(X0, Y0 is at the center of the hole)
(Z0 is at the top of the part)
(Using .5 thick material)
G00 G90 G54 X0 Y0 S400 M03
G43 H01 Z.1 M08
Z-.6
G01 G41 D01 X.175 F25.
G03 X.375 R.100 F7.
G03 I-.375 Z-.475
G03 X.175 R.100
G01 G40 X0 Y0
G00 Z1.0 M09
G28 G91 Y0 Z0
M30
%
```

NOTE: Maximum cutter compensation adjustability is .175, which is more than enough for this application.

Start with zero in the diameter offset column and enter a negative number to increase the thread diameter.

O.D. THREAD MILLING -

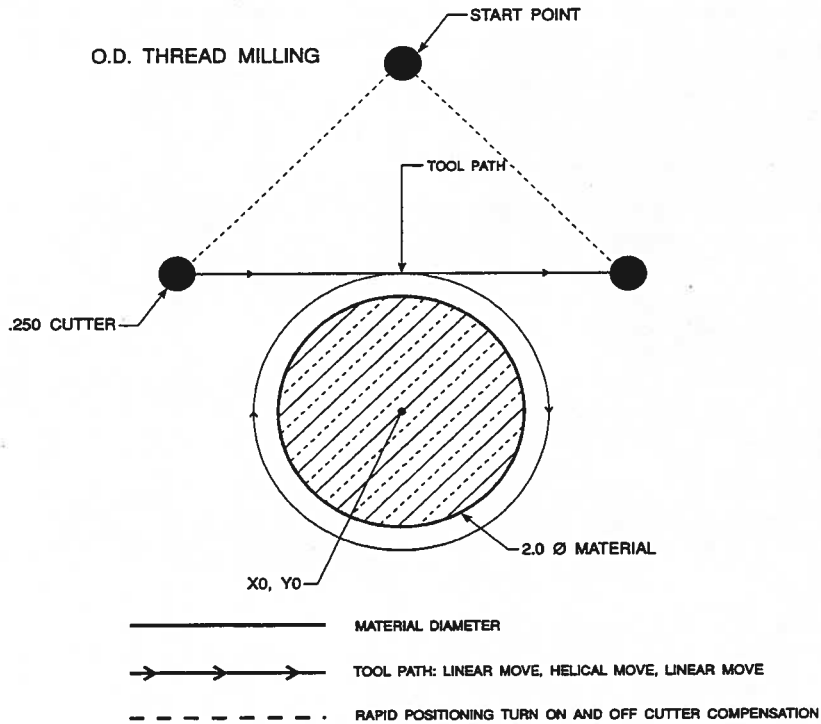


Figure 8-12 O.D. thread milling exercise.

The code for Figure 8-12 is as follows:

```

%
O2400                                     (Thread milling a 2.0 diameter post x 16 TPI)
(X0,Y0 is at the center of the post)
(Z0 is at the top of the part)
(Post height is 1.125 inch)
G00 G90 G54 X0 Y2.0 S2000 M03
G43 H01 Z.1 M08
Z-1.0
G41 D01 X-1.5 Y1.125                       (Turning on cutter compensation.)
G01 X0. F15.                               (Linear interpolation onto the post.)
G02 J-1.125 Z-1.0625                       (360° helical circle; negative Z move.)
G01 X1.5                                   (Linear interpolation off the post.)
G00 G40 X0 Y2.0                             (Turning off cutter compensation.)
Z1.0 M09
G28 G91 Y0 Z0
M30
%
```

NOTE: A cutter compensation turn on move can consist of any X or Y move from any position just as long as the move is greater than the amount being compensated for. The same rule applies for turning off cutter compensation.

8.14 SINGLE-POINT THREAD MILLING

Using the following data, we will write a program for single-point thread milling procedures:

DATA:

- 2.500 Ø hole
- Diameter of cutter (Subtract .750): 1.75
- Radial value (Divide by 2): .875
- Thread pitch: .0833 (12 TPI)
- Part thickness: 1.00

```

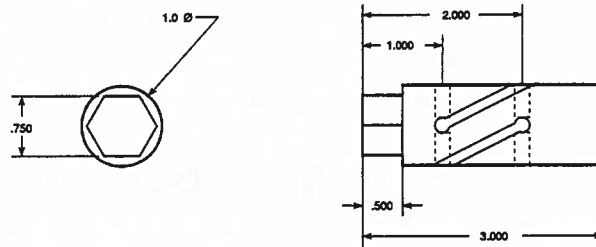
%
O1000                                (Main program)
(X0,Y0 is at the center of the hole)
(Z0 is at the top of the part)
T1 M06                                (Tool #1 is a .750 diameter single-point thread tool)
G00 G90 G54 X0 Y0 S2500 M03
G43 H01 Z.1 M08
G01 Z-1.083 F35.
G01 X.875 F15.                        (Radial value)
M98 P1001 L14                         (Multiply .0833 pitch x 14 passes = 1.1662 = total in Z-axis)
G00 G90 Z1.0 M09
G28 G91 Y0 Z0
M30
%
```

```

%
O1001                                (Helical subprogram)
G91
G03 I-.875 Z.0833
M99
%
```

8.15 FOURTH AXIS PROGRAMMING

The following is an example of fourth axis programming. Refer to Fig. 8-13.



EIGHT (8) HOLES CHAMFERED (.2 Ø)
 EIGHT (8) HOLES .1875 Ø, 90° SPACING, .4 DP.
 FOUR (4) CHANNELS, .125 WIDE



Fig. 8-13 Fourth axis programming exercise.

PROGRAM CODE FOR FIG. 8-13:

```

%
O1234
(Material is 1.0 Ø round stock x 3.0 L)
(Set material in collet to protrude 2.25.)
(Set fixture parallel with the table T-slots on the right side.)
(X0 is the front of the material.)
(Y0 is the centerline of the spindle and material.)
(Z0 is the top of the part.)
T1 M06
G00 G90 G54 X.250 Y-.500 A0 S4500 M03
G43 H01 Z.1 M08
M98 P1235 L6
G00 G90 Z.1 M09
T2 M06
G00 G90 G54 X1.0 Y0 A0 S5000 M03
G43 H02 Z.1 M08
G82 Z-.1 F10. R.1 P300
X2.0
A90.
X1.0
A180.
X2.0
A270.
    
```

(Fourth axis program using a Haas Servo 5C)

(Tool #1 is a .500 end mill to mill hex.)

(Tool #2 is a .375 Ø NC spot drill.)

PROG. EXAMPLES

```
X1.0
G00 G80 Z.1 M09
T3 M06
G00 G90 G54 X1.0 Y0 A0 S5000 M03
G43 H03 Z.1 M08
G83 Z-1.125 F12. R.1 Q.25
```

(Tool #3 is a .1875 Ø stub twist drill.)

```
X2.0
A90.
X1.0
G00 G80 Z.1 M09
```

(Tool #4 is a .125 Ø end mill.)

```
T4 M06
G00 G90 G54 X1.0 Y0 A0 S5000 M03
G43 H04 Z.1 M08
M98 P1236
G00 G90 Z.1 M09
G28 G91 Y0 Z0
M30
%
```

```
%
O1235
G01 Z-.125 F50.
Y.5 F35.
G00 Z.1
G91 Y-1.0 A60.
G90
M99
%
```

(Subprogram to mill hex.)

The following subprogram can be written in absolute or incremental programming. Examine each program and determine which style would be faster and easier to understand and to program in the future.

ABSOLUTE:

```
%
O1236 (Subprogram to mill channels.)
G01 Z-.25 F15.
X2.0 A90.
G00 Z.1
A180.
G01 Z-.25
X1.0 A90.
G00 Z.1
A180.
G01 Z-.25
X2.0 A270.
G00 Z.1
A360.
G01 Z-.25
X1.0 A270.
G00 Z.1
M99
%
```

INCREMENTAL:

```
%
O1236 (Subprogram to mill channels.)
G91
G01 Z-.35 F15.
X1.0 A90.
G00 Z.35
A90.
G01 Z-.35
X-1.0 A-90.
G00 Z.35
A90.
G01 Z-.35
X1.0 A90.
G00 Z.35
A90.
G01 Z-.35
X-1.0 A-90.
G00 G90 Z.1
M99
%
```

8.16 FORMULAS**TAPPING -**

STANDARD thread formula:

Revolutions per minute (RPM) divided by threads per inch (TPI) = Feed rate in inches per minute
$$\text{RPM/TPI} = F$$

METRIC thread formula:

Pitch (P) multiplied by .03937 = _____ multiplied by RPM = Feed rate in inches per minute
$$(P \times .03937) \times \text{RPM} = F$$
SPEED AND FEEDS -

S.F.M. (Surface Feet per Minute):

.262 multiplied by the cutter diameter multiplied by the RPM = SFM
$$.262 \times \text{Cutter Diameter} \times \text{RPM} = \text{SFM}$$

R.P.M. (Revolutions Per Minute):

3.82 multiplied by the recommended SFM divided by the cutter diameter = RPM
$$(3.82 \times \text{SFM}) / \text{Cutter Diameter} = \text{RPM}$$

I.P.M. (Inch Per Minute):

Feed per tooth multiplied by the number of cutter teeth multiplied by the RPM = Feed rate in inches per minute.

$$(\text{Feed/tooth} \times n) \times \text{RPM} = \text{IPM or F}$$

CUBIC INCH PER MINUTE:

Effective diameter of cut multiplied by the depth of cut multiplied by the inch per minute feed rate = cubic inch per minute.

$$(E \text{ Diameter} \times d) \times \text{IPM} = \text{CIPM}$$

9. CUTTER COMPENSATION DESCRIPTION

Cutter compensation is a method of shifting the tool path so that the actual finished cut is moved to either the left or right of the programmed path. Normally cutter compensation is programmed to shift by exactly the radius of the tool so that the finished cut matches the programmed path. The Offset display page is used to enter the amount for the tool to be shifted. The offset can be entered as either diameter or radius for both a geometry and a wear value. The effective value is the sum of the geometry and wear value. Setting 40 is used to select either diameter or radius. If diameter is specified, the shift amount is half of the value entered. Cutter radius compensation is only available in the X-Y-axis (G17).

9.1 GENERAL DESCRIPTION OF CUTTER COMPENSATION

G41 will select cutter compensation left; that is, the tool is moved to the left of the programmed path to compensate for the size of the tool. A **Dnn** must also be programmed to select the correct tool size from compensation memory. If compensation memory contains a negative value for cutter size, cutter compensation will operate as though G42 was specified. Cutter path compensation in this machine applies only to motion in the X and Y axes.

G42 will select cutter compensation right; that is, the tool is moved to the right of the programmed path to compensate for the size of the tool. A **Dnn** must also be programmed to select the correct tool size from compensation memory. If compensation memory contains a negative value for cutter size, cutter compensation will operate as though G41 was specified.

The code G40 will cancel cutter compensation and is the default condition when a machine is powered-on. When canceled, 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.

If cutter radius compensation is selected (G41 or G42), you may only use the X-Y plane for circular motions (G17). Cutter radius compensation is only available in the X and Y axes.

There is a simple rule about cutter compensation which helps to understand the motions the control uses to compensate for tool size. The control operates on one motion block at a time. It will look ahead, however, to check the next two blocks containing X or Y motions. The interference checks are performed on these three motions. Setting 58 controls how this part of cutter compensation works. It can be set to Fanuc or Yasnac.

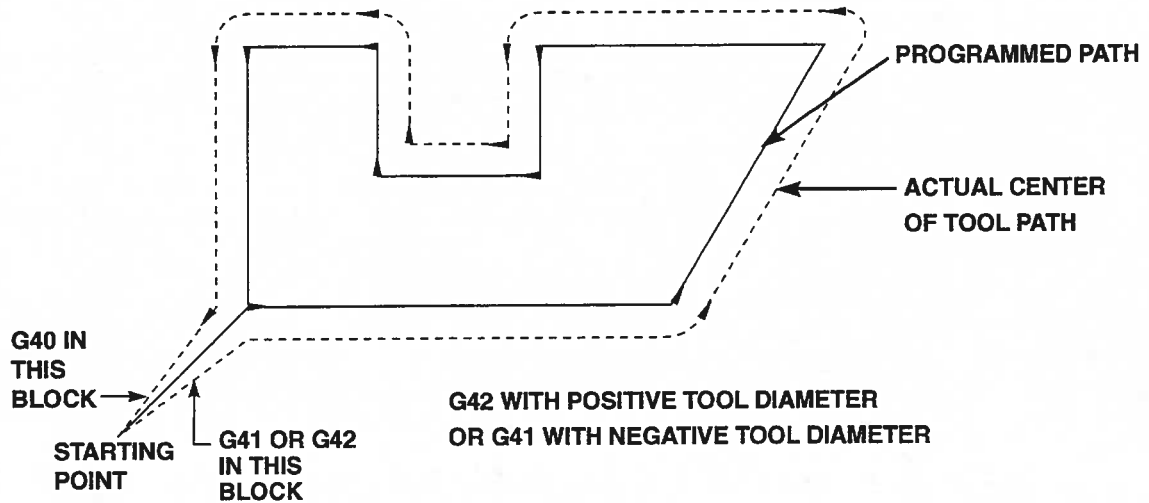
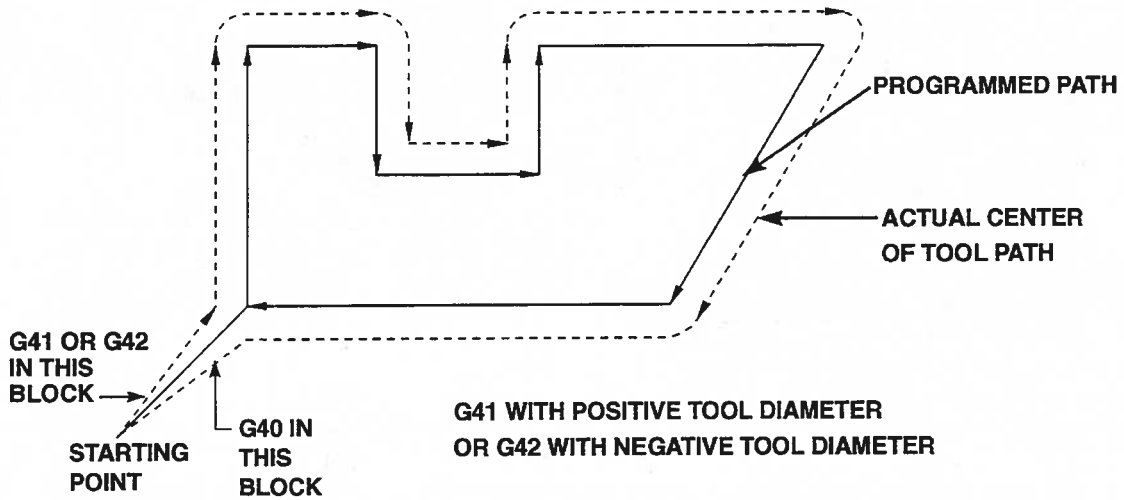
When Setting 58 is set to Yasnac, the control must be able to position the tool edge along all of the programmed cuts without overcutting the next two motions. All outside angles are joined by a circular motion.

When Setting 58 is set to Fanuc, the control does not require that the tool cutting edge be placed along all programmed cuts. Overcutting, however, is still prevented and, if overcutting cannot be prevented, an alarm will still occur. Outside angles less than or equal to 270 degrees are joined by a square corner and outside angles of more than 270 degrees are joined by an extra linear motion.

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

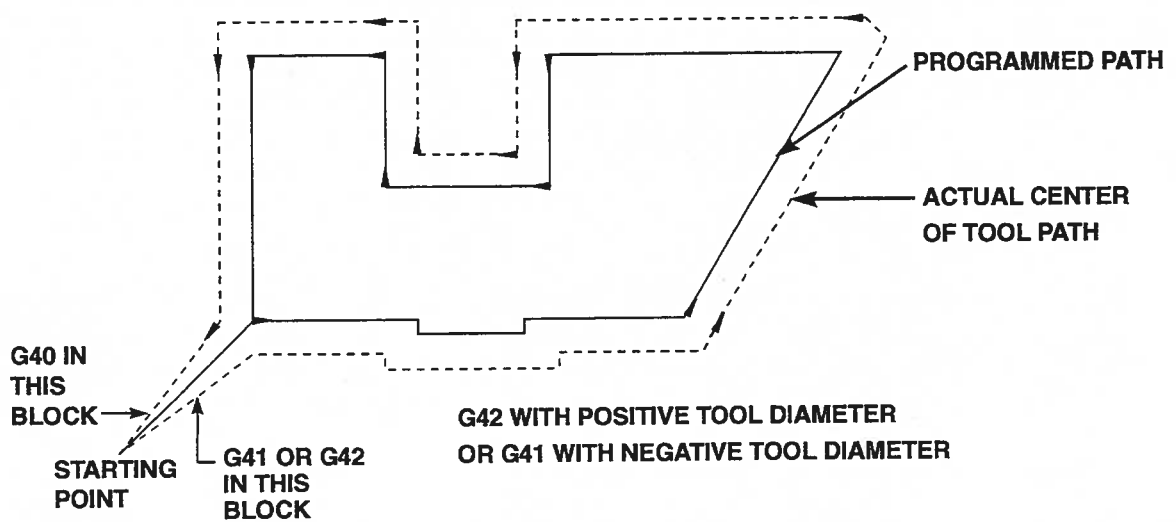
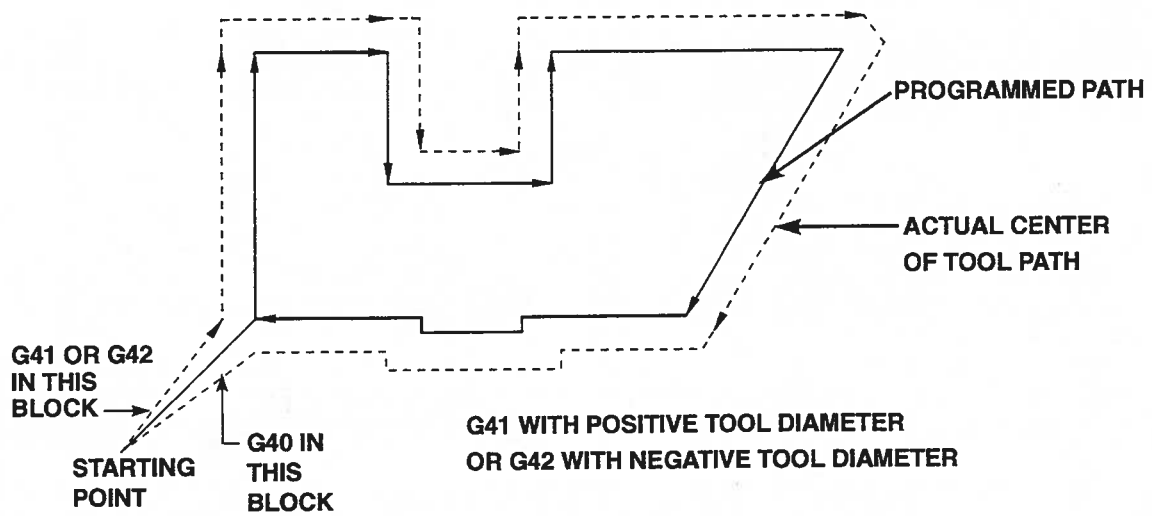
CUTTER COMPENSATION

(Yasnac style)



CUTTER COMPENSATION

(Fanuc style)



9.2 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 three type of moves. In a block that turns on cutter compensation, the starting position of the move is the same as the programmed position but the ending position of the move will be offset by the cutter compensation size. In a block that turns off cutter compensation, the starting point is offset and the ending point is not offset. Similarly, when a block changes from left to right compensation, the starting point is shifted in one direction and the ending point is shifted in the other direction. The result of all of this is that the tool is moved 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 enter cutter compensation, a nonzero D code must be specified and either G41 or G42 specified. To exit from cutter compensation, you may either specify D0 or G40 or both.

You should always turn off cutter compensation in a move which 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 on or off cutter compensation during a circular move (G02 or G03); otherwise an alarm will be generated.

An offset selection of D00 will use zero as the offset size and have the same effect as turning off cutter compensation. If a new value from offset memory is selected while cutter compensation is active, the starting point of a move will reflect the old value and the ending point will reflect the new value. This will also have the effect of shifting the motion to something other than what was intended by the programmer. You cannot change the offset code or side during a circular motion block.

When turning on cutter compensation in a move followed by a second move at an angle of less than 90 degrees, there are two common ways of computing the first motion. They are called cutter compensation type A and B. Type A will not stay on the programmed side of the first cut but will go directly to the starting point for the second cut. Type B will remain clear of the first line and follow it with the same motions as described in 10.1 to position for the second cut. Types A and B are selected with Setting 43.

Setting 58 also changes the way the entry and exit to cutter compensation works. There is still a type A or B but the type of moves used to clear the tool from the beginning of the cut change as described in the previous section. The following two diagrams describe how this works.

9.3 FEED ADJUSTMENTS IN CUTTER COMPENSATIONS

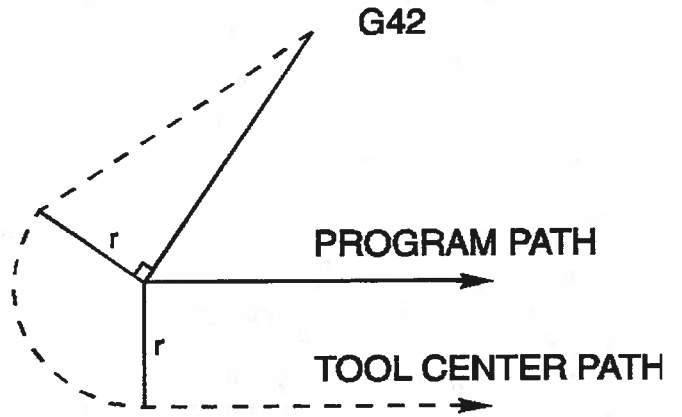
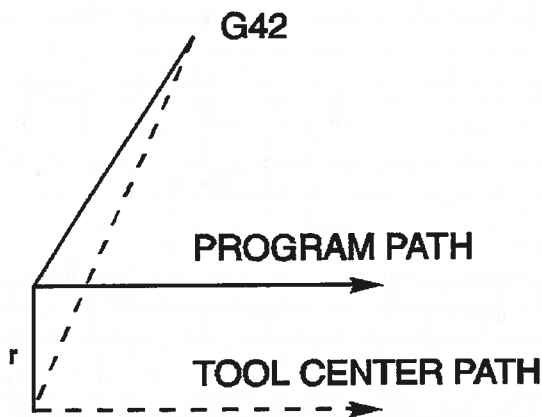
When using cutter compensation in circular moves, there is the possibility of speed adjustments to what has been programmed. If the intended finish cut is on the outside of a circular motion, the tool should be slowed down to ensure that the surface feed does not exceed what was intended by the programmer. 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 inside of circular motion, there is no speedup adjustment made to the feed rate.

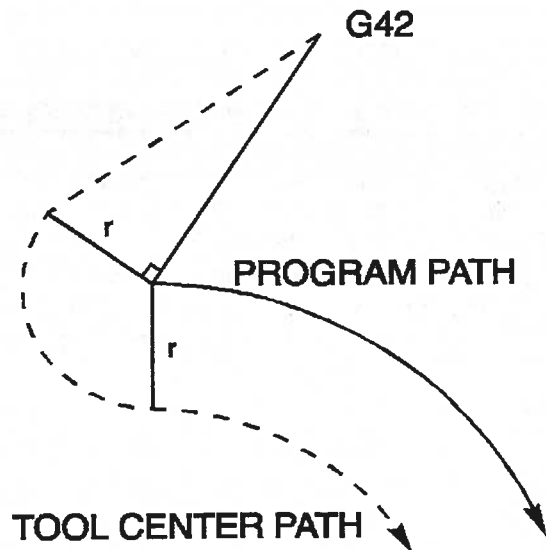
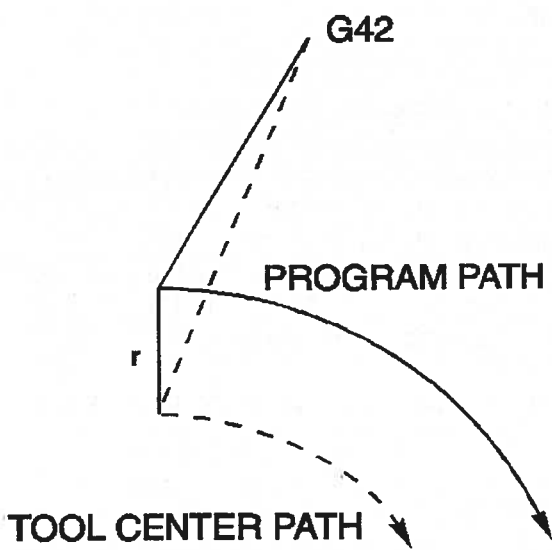
CUTTER COMPENSATION ENTRY (Yasnac)

TYPE A

TYPE B



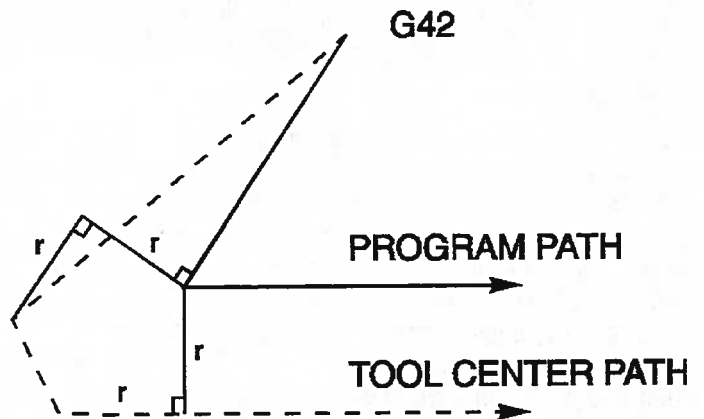
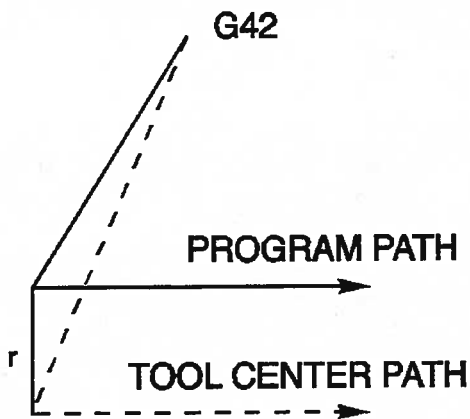
r = TOOL RADIUS



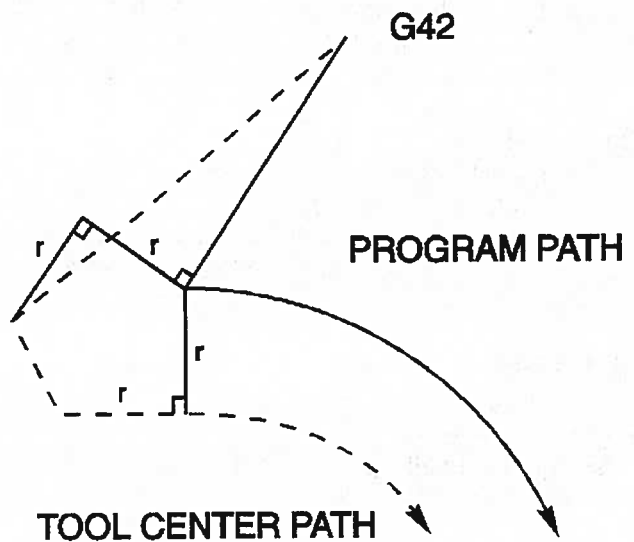
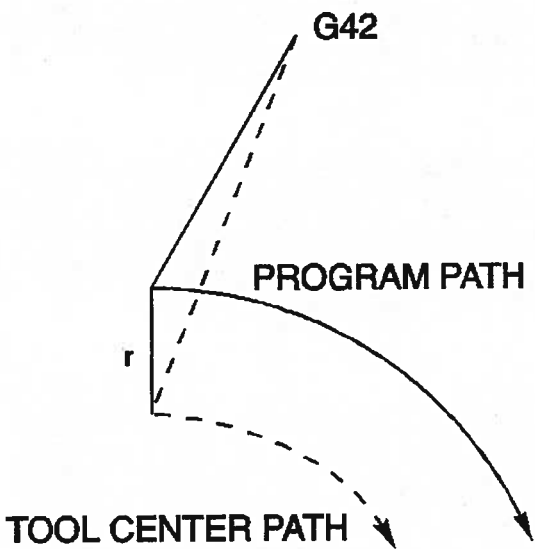
CUTTER COMPENSATION ENTRY (Fanuc style)

TYPE A

TYPE B



$r = \text{TOOL RADIUS}$



10. QUICKCODE

10.1 INTRODUCTION

This programming option can be activated by contacting your local HAAS dealer.

QUICKCODE is an innovative new way to program CNC machines. It combines the simplicity and flexibility of G code programming with English descriptive sentences to enable even beginning programmers to construct most 2 dimensional parts. Experienced programmers will also love the speed they can now enter programs manually. This is possible because with one button push you can replace a hundred individual keystrokes or more. Often used code can be "canned" into the program and called back with one key stroke. And what if you don't like the way Quick Code is programmed? Simple! You can change it to suit your needs or programming tastes. Make it complex or simple as you like.

Background

When NC machines were first introduced they had very limited or no memory at all. They were often run from tapes and instructions needed to be as concise as possible. In order to accomplish this a sort of cryptive language evolved which we called G code programming. A command to "TURN OFF COOLANT" which requires 16 letters and spaces is reduced to "M09" which takes only 3 characters. This made tape lengths and memory requirements manageable to say the least. As it evolved, hundreds of instructions and canned cycles were encrypted into G and M code programming. For an experienced programmer, the G codes are actually very easy to use but the learning process requires constant referring back to the manual to figure out which code to use to accomplish the task. And even the most experienced programmers have to admit that every once in a while you forget to put the right "I,K,Q or P's" into say a G83 drilling cycle. Quick Code eliminates this tedium. Simply cursor over to the drill cycle you want and press the write button and all the code you need to drill the hole is inserted with default values for all necessary "I,K,Q,P's". And you can edit those values to suit your individual needs.

How It Works

Quick Code reverses the G code encryption confusion. On the right side of the screen you have English commands that describe the operation to perform. By selecting the operation and with one button push, the code is inserted in your program on the left side of the screen. A program is constructed by selecting English commands that are then changed over to machine language or G codes. In doing this you will learn quickly the G code format without studying any manual. Another feature is the ability to scan through a program and Quick code will tell you what all the G and M codes mean, a great help in learning the code.

An Open System

One of the neatest features of Quick Code is that it is adaptable to the way you program. Everybody programs a little differently and have special preferences, such as, do you put the "T" command on the same line as the tool change or before. With Quick Code you can edit the program so that any English command you desire can be matched with any G code to be inserted. Because of this open format we are hoping that you can show us innovative new ways to program complex parts.

What it is Not

Quick Code is not a CAD/CAM package for generating complex moves on 3 dimensional parts. With most CAM packages you have to draw a drawing much like you would in AUTO CAD and then indicate the moves around the drawing and finally generate the code through the post processor. Not a simple task. Nor is it a conversational program with icons where you are asked to fill in the blanks. The difference with these packages is that they require training and much like learning a second language you have to have the time and determination to learn them. They have a tremendous amount of power but you don't always need it. Quick Code is a bridge between high end CAD/CAM and slow and cumbersome G code programming. It is our expectation that it can be used by anyone with very minimal training. For most simple parts we believe that Quick Code is an ideal choice.

Conversational QuickCode

A new feature at HAAS, "Conversational" QuickCode, makes programming with QuickCode even easier. This feature can be used to "prompt" the operator for the information necessary to create a program. Refer to the "Conversational QuickCode" section for a description of how to use this feature.

10.2 QUICKCODE TERMINOLOGY

Before describing the QUICKCODE environment you need to know the terms listed below. Following this brief list is an illustration of the QUICKCODE display and how the terms are related to the display.

- EDIT WINDOW** Portion of the display that shows the currently edited program.

- ITEM** A line of text representing code that can be added to the edit window when it is selected.

- GROUP** A list of items that usually have something in common so that they can be grouped together.

- GROUP WINDOW** Portion of the display which presents a list of groups and items.

- HELP WINDOW** Portion of the display which presents user created help, address code help, and warning messages.

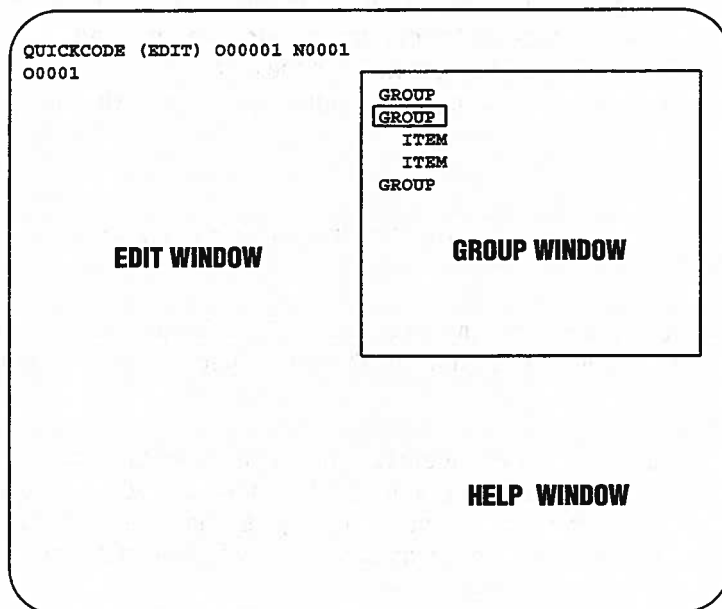


Figure 10-1. The QUICKCODE display.

10.3 USAGE AND FEATURES

ACCESSING QUICKCODE

Before QUICKCODE can be used, the bit labeled ENA CONVERS in parameter 57 must be set to 1. When this bit is set to 0, you will not be able to access the QUICKCODE screen. Enter QUICKCODE by selecting edit mode and then pressing the PRGRM/CONVRS key twice. The first press of the PRGRM/CONVRS key enters the standard editor, whereas the second press of this key will enter the 80 column format of the QUICKCODE screen. Each additional press of this key will switch between standard edit mode and QUICKCODE mode.

THE EDIT WINDOW

The QUICKCODE edit window is exactly the same as the standard editor that you are used to on the HAAS control. Each time that you select a group item, as described below, the edit window will be updated to show you what code has been added to the currently edited program. You have access to all of the edit functions with the exception of the jog handle and the block copy function keys. In the standard editor, you can use the jog handle to traverse program text quickly. While in QUICKCODE, the jog handle is reserved to maneuver the group list. You can still cursor through the program text by using the cursor keys provided on the center of the keypad. You are also restricted from using the block copy keys while in QUICKCODE. For this, you can always switch back to standard edit mode by pressing the PROGRM/CONVRS key. At this point you have access to the jog handle, for long comments, and the block copy functions. QUICKCODE is not available while in BACKGROUND EDIT MODE.

THE GROUP WINDOW

The group window displays a list of groups that are defined in the QUICKCODE source file. The groups can be moved through for selection by turning the jog handle in the plus, clockwise, direction. For each jog handle click in the plus direction, the group window cursor will advance to the next group. In this manner you can move through every group in the list. When the last group is highlighted, the next plus click will move the cursor to the first group in the list. To view and cursor through items within a group, turn the jog handle in the minus, counter clockwise, direction. As long as you turn the jog handle in the minus direction the cursor will advance through, display and highlight items in the current group. By turning the jog handle one click clockwise, the group item list will be closed and additional plus clicks will continue to traverse the group list.

THE HELP WINDOW

The help window is just below the group window. It is used to display QUICKCODE source file help, address code help, and warning messages to the user.

The QUICKCODE source file can contain comments that will not be placed into the edit window. These comments will be displayed on the first five lines of the help window. These comments are typically used for explaining item code and usage.

As the user cursors through a program, each address code that is highlighted will be interpreted and a short description of its usage is displayed in the help window. This address code help is as accurate as possible. Since the program is not being interpreted sequentially as it is when a program is run, full interpretation cannot take place. When the context of an address code cannot be fully determined, the most likely usage is displayed.

Sometimes during editing we can determine if a run time error will occur without actually running the program. For instance we can tell if multiple codes from one G code group are on a line. In this case QUICKCODE will display a highlighted warning message to the user indicating that there is a problem. This is found on the last line of the help window.

SPECIAL KEYS

QUICKCODE makes use of the jog handle to select from the group list and group items. This is described in the group window section above. QUICKCODE action takes place when the WRITE key is pressed. If there is text on the input line, normal text insertion takes place when the WRITE key is pressed. When the input line is blank, pressing the WRITE key will cause QUICKCODE to take the following action:

-If the currently highlighted item is designed as a help only item, the edit window is not modified.

-If numeric program code is found associated with the highlighted item, the edit window cursor is moved to the end of the current edit block and the associated code is inserted after that block. The edit cursor is left at the end of the last QUICKCODE block that was inserted.

10.4 CONVERSATIONAL QUICKCODE

Conversational QuickCode is used to "prompt" the operator for the information necessary to create a program. The "prompting comments" are created by placing a '?' as the first character of a comment in the QuickCode source file (O9999). A comment is any text, up to 34 characters, that is contained in parentheses. When a program is being written using QuickCode, the prompting comments will appear on the screen, requiring a response from the operator. The numeric value entered by the operator will be assigned to the G-code item that immediately precedes the prompting comment in the source file. For example, the following line of code:

```
G1 F25. (?WHAT IS THE FEED RATE ?) X2. Y5. ;
```

will produce the following prompt:

```
WHAT IS THE FEED RATE ?
```

with the default feed rate value displayed below the prompt, as shown:

```
WHAT IS THE FEED RATE ?
```

```
F25. 
```

The operator must then enter a numeric value and press the WRITE key to change the feed rate, or simply press the WRITE key to accept the default feed rate. The control will wait for an operator response before writing the block to the edit file. Unacceptable responses, such as those containing too many digits or an unnecessary decimal, will cause the control to flash an error message and wait for another response.

Once the operator has input a value for all of the 'variable' G-code items in a block of code, the entire (revised) block is displayed on the input line, as shown:

```
CORRECT (Y/N) ?
```

```
G1 F25. X2. Y5. ; 
```

The operator then must enter 'Y' to accept the block, or 'N' to cancel it. If it is accepted, the block is written to the edit file, and the QuickCode processing resumes with the next block (if there is one). If it is not accepted, the prompting process is repeated for the same block.

Press the UNDO or RESET keys to exit the current block at any time while in Conversational QuickCode.

In machines shipped after April 11, 1996 (with software version 9.06 or later), the QuickCode source file has been modified to include prompting comments.

10.5 A SAMPLE QUICKCODE SESSION

The following illustrates how QUICKCODE can be used to build a program. A program will be built to spot, drill, and tap 5 holes on a circular bolt hole pattern. We will assume that tool 1 is a spot drill, tool 2 is a drill for a 10-32 tap, and tool 3 is the tap. Before you proceed, make certain that QUICKCODE is enabled in parameter 57. ENA CONVERS should be set to 1. You will also need program O9999 in the control.

The jog handle is an integral part of using QUICKCODE and is used quite often. For brevity we use JHCW to mean jog handle clockwise and JHCCW to mean jog handle counter clockwise. For instance, seeing JHCW means that you should turn the jog handle in a clockwise direction.

SELECT THE PROGRAM

QUICKCODE will not generate a program for you. So the first thing you must do is to create a program. Create a program with the following.

- 1.) Press LIST PROG.
- 2.) Type O0005 (or another convenient program number)
- 3.) Press WRITE.

This creates a program in the usual manner. Proceed to edit the program by pressing EDIT. The control will switch to the PROGRAM display and you will see the program number and semicolon in the top left of the screen. Now press the PRGRM/CONVRS key to enter QUICKCODE.

The following screen is presented.

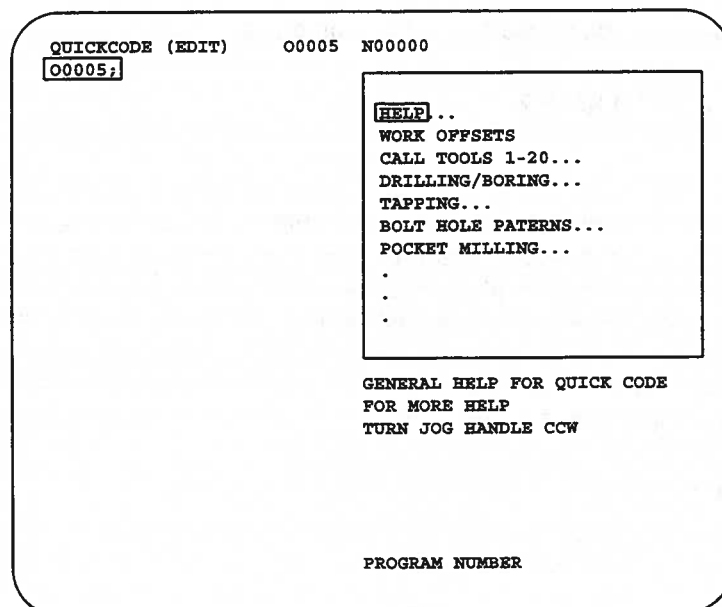


Figure 10-2. Empty Program.

SELECT A WORK COORDINATE SYSTEM

- 1.) JHCW until the group titled WORK OFFSETS in the group window is highlighted.
- 2.) JHCCW one click. The items belonging to WORK OFFSETS will appear and the first item, G54, is highlighted.
- 3.) Press the WRITE key. This will copy the code associated with the G54 item into your program.

The following figure shows what the screen should look like. Note that the cursor has been moved to the end of the last block copied into your program. This is where the next block of code will be entered. Also note that the code that was added, is displayed just below the group window. When the QUICKCODE source file is constructed properly, you will see the code that will be added to your program just below the group window. This can be helpful in determining which item in a group you want.

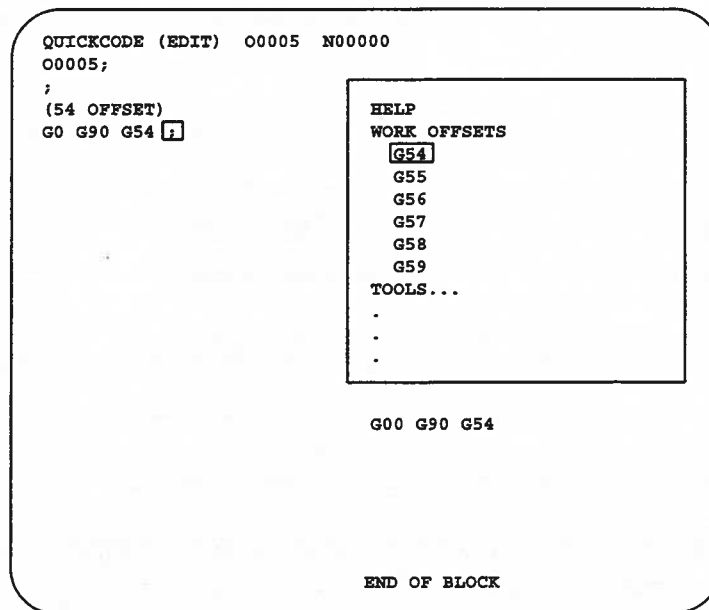


Figure 10-3. Program with G54 added.

- CALL TOOL 1

- 1.) JHCW and highlight the group titled TOOLS.
- 2.) JHCCW one click. CALL TOOL 1 will be highlighted.
- 3.) Press the WRITE key to insert the code for calling tool 1 into your program. Your program will look like this.

```

QUICKCODE (EDIT) 00005 N00000
;
(CALL TOOL 1)
G80 M09;
M06 T1
G0 X0 Y0 S1250 M03
G43 Z1. H01 M08 [F]

```

```

HELP...
WORK OFFSETS
TOOLS...
CALL TOOL 1
CALL TOOL 2
CALL TOOL 3
CALL TOOL 4
CALL TOOL 5
.
.
.

```

```

G80 M09
M06 T1
G00 X0 Y0 S1250 M03
G43 Z1. H01 M09

END OF BLOCK

```

Figure 10-4. Program with CALL TOOL 1 added.

Remember that after pressing WRITE you can always edit the program to make minor adjustments to code that QUICKCODE inserts into your program. You do not have to leave the QUICKCODE display to do this. But you must remember to reposition the cursor back to the block where you want to add the next item. QUICKCODE will automatically seek the end of the current block that the cursor is on, so there is no need to cursor to the end of the block.

We will assume that the material is aluminum and we will not have to edit the inserted code. We also assume that the work coordinate zero for G54 is at the center of the bolt hole pattern. The QUICKCODE source file was set up with average spindle speeds in mind. You could have different QUICKCODE files for different materials. By changing parameter 228 you can quickly change the file that QUICKCODE works with.

- INVOKE THE SPOT DRILLING CANNED CYCLE

- 1.) JHCW and highlight the group titled DRILLING / BORING.
- 2.) JHCCW one click. SPOT DRILL will be highlighted.
- 3.) Press the WRITE key to insert the code for setting up spot drilling.

Note that QUICKCODE added a block to actually execute a spot drill at the location X0 Y0. This would normally be edited to the first location you want to drill. You could also add more X and Y drill locations if needed.

Your program will look like this.

```

QUICKCODE (EDIT) O0005 N00000
;
GO X0 Y0 S1250 M03
G43 Z1. H01 M08
;
(SPOT DRILL)
G82 G98 Z-0.1 R.05 F3.5 P0.2 L0
X0 Y0 [ ]
    
```

```

HELP...
WORK OFFSETS
TOOLS...
DRILLING AND BORING...
[SPOT DRILL]
DRILL
PECK DRILL
.
.
.
    
```

```

G82 G98 Z-0.1 5.05 F3.5 P0.2 L0
P=DWELL TIME AT Z DEPTH 1.=1SEC
X0 Y0
ENTER DRILL LOCATIONS
    
```

Figure 10-5. Program with spot drilling invoked.

For this example we do not want to drill a hole at X0 Y0. The bolt pattern will do all the work. At this time delete X0 Y0 from the edit window.

- EXECUTE A CIRCULAR BOLT HOLE PATTERN

- 1.) JHCW and highlight the group titled BOLT HOLE PATTERNS.
- 2.) JHCCW one click. BOLT CIRCLE will be highlighted.
- 3.) Press the WRITE key to insert the code for executing a bolt hole circle.

Here QUICKCODE inserted a call to G70 to execute a bolt hole pattern. In this particular instance L7 was placed into your program as a default meaning that 7 holes on a circle are to be drilled. We want to drill 5 holes. So here you will have to edit the program so that L5 will be on the G70 line.

I1.5 in the code indicates a bolt circle radius of 1.5 and a diameter of 3.0. If needed, you might also change the specified radius of the bolt circle.

Since this is the last line in your program, after editing you will not have to cursor down to prepare for your next item selection.

Your program will look like this.

```

QUICKCODE (EDIT) 00005 N00000
(SPOT DRILL);
G82 G98 Z-0.1 R.05 F3.5 P0.2 L0;
;
(BOLT CIRCLE)
G70 I1.5 J90 L5 [ ]

```

```

HELP...
WORK OFFSETS
TOOLS...
DRILLING AND BORING...
TAPPING
BOLT HOLE PATTERNS
  BOLT CIRCLE
  BOLT HOLE ARC
  BOLT HOLES ALONG AN ANGLE
  CIRCULAR POCKET MILLING
.
.
.

```

```

G70 I1.5 L7
DRILL 7 HOLES EXACTLY SPACED
ON A 3.0 DIA. B.C WITH THE
FIRST HOLE IN THE 4+ DIRECTION

```

Figure 10-6. Program with a bolt hole circle added.

By now you should have a good idea of how your program changes after selecting a group item and pressing WRITE. To save space we will not show you each display as a selection is made. Instead we will list the remaining actions needed to finish drilling and tapping the 5 holes. The remaining selections are very similar to what we have already done.

- CALL TOOL 2

- 1.) JHCW and highlight the group titled TOOLS.
- 2.) JHCCW two clicks. CALL TOOL 2 will be highlighted.
- 3.) Press the WRITE key.

- INVOKE A DRILLING CANNED CYCLE

- 1.) JHCW and highlight the group titled DRILLING / BORING.
- 2.) JHCCW two clicks. DRILL will be highlighted.
- 3.) Press the WRITE key.

For most drilling cycles you will have to edit the generated program to specify the depth of drilling. And since we are doing a bolt hole pattern, don't forget to remove the X0 Y0 supplied as a sample drill location.

- EXECUTE A CIRCULAR BOLT HOLE PATTERN

- 1.) JHCW and highlight the group titled BOLT HOLE PATTERNS.
- 2.) JHCCW one click. BOLT CIRCLE will be highlighted.
- 3.) Press the WRITE key.

And now for tool 3, the 10-32 tap.

- CALL TOOL 3

- 1.) JHCW and highlight the group titled TOOLS.
- 2.) JHCCW three clicks. CALL TOOL 3 will be highlighted.
- 3.) Press the WRITE key.

- INVOKE A TAPPING CYCLE

- 1.) JHCW and highlight the group titled TAPPING.
- 2.) JHCCW one click. 10-32 TAP will be highlighted.
- 3.) Press the WRITE key.

Note that for all tapping you will never have to calculate the spindle speed and feed again. Only the depth of the tap will have to be edited, and perhaps your X and Y coordinates.

Remove the supplied X0 Y0 to prepare for the bolt hole pattern as done above.

- EXECUTE A CIRCULAR BOLT HOLE PATTERN

- 1.) JHCW and highlight the group titled BOLT HOLE PATTERNS.
- 2.) JHCCW one click. BOLT CIRCLE will be highlighted.
- 3.) Press the WRITE key.

At this point you may decide to move the table forward to remove the part. You can do this in your program with the following.

- HOME THE Y AND Z AXES

- 1.) JHCW and highlight the group titled MISC. INSTRUCTIONS.
- 2.) JHCCW until HOME Y AND Z AXIS is highlighted.
- 3.) Press WRITE.

And finally, you can properly terminate your program with:

- PROGRAM RESTART

- 1.) JHCCW until END PROGRAM/RESET TO BEGINNING is highlighted.
- 2.) Press WRITE.

You now have a ready to run program. You should always verify everything in graphics to make certain that you have not forgotten any steps. Although this looks like a lot of steps, it is actually very easy once you become familiar with the QUICKCODE environment. The above program can be generated in less than a minute.

10.6 THE QUICKCODE SOURCE FILE

All of the text seen in the group window, all of the code associated with items of groups, and much of the help text observed in the help window is contained in a G code program. This program is called the QUICKCODE source file. With this design, the user can modify QUICKCODE and tailor it to his specific needs. You can add or change groups and items. The user can develop his own QUICKCODE file, or program, by editing this file. Dealers can develop new applications and distribute them to their customers. The ability to edit the source file makes QUICKCODE an extremely flexible tool.

SOURCE FILE PROGRAM DESIGNATION

Program number 9999 is the default QUICKCODE source file. Every HAAS control equipped with QUICKCODE comes with a sample 09999 program installed. The default program number can be changed by changing parameter 228. If the file number in parameter 228 is not found in the control, the message FILE NOT FOUND is displayed and you will not be able to enter the QUICKCODE screen. The source file must be formatted as defined below. If program 09999 is not formatted in the appropriate manner then you may not see all, or any, of the defined QUICKCODE groups. You can use the following skeleton as a start for defining a QUICKCODE source file.

```
%
09999 (QUICKCODE - HAAS AUTOMATION INC)
()
(ADD ANY COMMENTS HERE THAT PERTAIN TO
(THE ENTIRE SOURCE FILE. FOR INSTANCE)
(YOU CAN RECORD WHO MADE THE FILE, THE
(DATE AND TIME OF THE LAST CHANGE, A )
(VERSION NUMBER, OR ANYTHING ELSE YOU )
(WANT. ALL COMMENTS PRIOR TO THE FIRST)
(GROUP ARE NOT SEEN BY THE USER.    )
()
(QUICKCODE GROUP DEFINITIONS FOLLOW)
.
.
.
()
(END OF QUICKCODE)
%
```

DEFINING A GROUP IN THE GROUP LIST (*)

To define a group that will show up in the group window, simply enter a comment where the first character is an asterisk. For instance if you want five groups to show up in the group window, then you would include the following five lines in the QUICKCODE source file.

```
(*GROUP1)  
(*GROUP2)  
(*GROUP3)  
(*GROUP4)  
(*GROUP5)
```

Of course, you can use any descriptive title for the group that is appropriate to what the group will contain. Group titles can be up to 35 characters long. Any additional characters beyond 35 will not be displayed.

GROUP HELP

The first five comments after the group definition will be displayed in the help window. These comments can be used to explain what is contained in the group. For example:

```
(*HELP)  
(THIS GROUP CONTAINS HELP ON HOW TO)  
(USE QUICKCODE. WHEN THIS GROUP IS)  
(HIGHLIGHTED, TURN THE JOG HANDLE IN)  
(THE MINUS DIRECTION FOR MORE HELP.)
```

Additional comments beyond five lines are not displayed by QUICKCODE. This is a method of documenting the source file for the developer of the QUICKCODE file. Documenting comments can also be hidden in the source file by placing an empty comment after group help comments. In the following example only the first two comments are displayed in the help window.

```
(*HELP)  
(ONLY THE FIRST TWO COMMENTS ARE)  
(DISPLAYED IN THE HELP WINDOW.)  
( )  
(THIS COMMENT IS NOT DISPLAYED)
```

If more than five lines are required to comment on a group, then you can use several groups to display 5, 10 or 15 lines of help. With this method you can add any amount of information you want about the group that is desired.

GROUP CODE

What happens when a group definition is highlighted and the user presses the WRITE key? If there is a G code after the group definition and before any other group or item definitions, then that G code will be inserted into the program that is being developed. Groups do not have to contain items for generating G code. A group title can stand alone as a code generating entity. The following group definition would add a G28 M30 to the program being developed when WRITE is pressed.

```
(*END OF PROGRAM)  
(THIS RETURNS ALL AXES TO MACHINE)
```



```
(ZERO AND ENDS PROGRAM EXECUTION)
(G28 M30)
G28 M30
```

Note that the user will not see what G code is generated until the WRITE key is pressed and the code is inserted into the program. For this reason you may want to place the code that is to be generated in a help comment as is done above.

QUICKCODE can also generate comments in the program being generated. Any comments following an empty comment will be added to the currently edited program. In fact all code following an empty comment is inserted into the program until another empty comment is encountered or until a group or item definition is encountered. The empty comment must be the first code in the block. Any code in the same block as the empty comment is not entered into the program. In the following example, only the code in blocks between the empty comment blocks are added to the program being generated.

```
(*GENERATES COMMENTS AND CODE)
(THIS IS NOT ADDED TO PROGRAM)
)(THIS IS NOT ADDED TO PROGRAM)
(THESE COMMENTS WILL BE ADDED TO THE)
(PROGRAM WHEN THIS GROUP IS)
(HIGHLIGHTED AND WRITE IS PRESSED)
G0 G90 G54 (THIS CODE IS ADDED)
()
(THESE COMMENTS ARE NOT ADDED TO THE)
(PROGRAM BEING GENERATED)
```

DEFINING AN ITEM BELONGING TO A GROUP (**)

To define an item belonging to a group simply enter a comment after a group definition where the first two characters of the comment are asterisks. For instance, the following code generates a group with four subordinate items.

```
(*GROUP)
(**ITEM1)
(**ITEM2)
(**ITEM3)
(**ITEM4)
```

With the above QUICKCODE source file, only one group is displayed in the group window when the jog handle is turned clockwise. When the jog handle is turned counter clockwise, the five items are displayed and traversed. The item titles are indented one space so that you can differentiate items from groups. Only 34 characters of the item definition comment are displayed in the group window. Additional characters are ignored. The group that items belong to will always be displayed on the screen. The only limit to the number of items in a group is the amount of control memory available.

ITEM HELP

Item help works the same way as group help. The first four comments after the item definition are displayed in the help window. If more than four lines are required, it is recommended that prior items contain the desired comments. In this case instructions would have to be added to indicate which item generates the G code.

For example:

```
(*GROUP)
(**HELP FOR THE FOLLOWING ITEM)
(THESE LINES OF CODE ARE HELP)
(COMMENTS THAT REQUIRE MORE THAN)
(FIVE LINES OF COMMENTARY)
(THIS IS THE LAST LINE OF THIS ITEM)
(**ITEM THAT GENERATES CODE)
(AND HERE WE FINISH THE COMMENTARY)
(FOR CODE GENERATED BY THIS ITEM)
GO G90 G01 F30
()(*****
(*NEXT GROUP)
```

Although the above example is somewhat awkward, it does provide a method that will satisfy unusual cases. The line with all of the asterisks is legal. It is not inserted into the current program when the WRITE key is pressed. It is used to visually separate groups.

ITEM CODE

Code generated by group items is formatted in the same manner as group code is formatted. Refer to the section on group code for an explanation of how code is generated.

10.7 A SAMPLE QUICKCODE SOURCE FILE

After developing or modifying a QUICKCODE file, it is recommended that you save an off-line copy in a computer. You can keep comments in the QUICKCODE source file prior to the first group indicating what version the file is and how it differs from other versions. Maintain this program as you would any other G code program in your control with a proper backup scheme. **Remember! This file operates the QUICKCODE feature in your HAAS machine.**

A sample QUICKCODE source file can be found on the floppy that comes with the control. It contains many examples of how QUICKCODE can be used.

11. MACROS

11.1 INTRODUCTION

This control function is optional. If you would like further information on installing this feature please call Haas Automation or your dealer for more information.

This is an introduction to macros as implemented on the HAAS CNC controls. MACROS adds capabilities and flexibility to standard G-code programming that allow the programmer to better define a tool path in a quicker and more natural way. With few exceptions, MACROS, as implemented on the HAAS controls, is compatible with FANUC 10M and 15M controls. Macro features not included in the current release are listed at the end of the manual. Programmers already familiar with macro programming will want to review this section in order to avoid unnecessary work.

In traditional CNC programming, a program consists of subroutines that CANNOT be changed or altered except by editing individual values with an editor. MACROS allows the capability to program subroutines where the tool path or location of the tool path is changed, depending on the values contained within variables set by the programmer. These variables can be passed to the subroutine as parameters, or the values can reside in what are called *global variables*.

What this all means is that a programmer can create a collection of subroutines that have been fully debugged. These programs can be used as high level tools that can enhance programmer and machinist productivity. MACROS is not intended to replace modern CAD/CAM software, but it can and has improved machine productivity for those who use it.

Here are a few examples of the applications for MACROS. Rather than give macro code here, we will outline the general applications that MACROS can be used for.

TOOLS FOR IMMEDIATE, ON-TABLE FIXTURING

Many setup procedures can be semi-automated to assist the machinist. Tools can be reserved for immediate situations that were not anticipated during tool design. For instance, suppose a company uses a standard clamp with a standard bolt hole pattern. If it is discovered, after setup, that a fixture will need an additional clamp and if macro subroutine 2000 has been programmed for drilling the bolt pattern of the clamp, then the following two-step procedure is all that is needed for adding the clamp to the fixture.

1) Determine X, Y, and Z coordinates and angle where the clamp is to be placed by jogging the machine to the proposed clamp position and reading the position coordinates from the machine display.

2) Execute the following command in MDI mode:

```
G65 P2000 X??? Y??? Z??? A??? ;
```

where ??? are the values determined in step 1.

Here, macro 2000 (not shown) takes care of all the work since it was designed to drill the clamp bolt hole pattern at the specified angle of A. Essentially, the machinist has created his own custom canned cycle.

SIMPLE PATTERNS THAT ARE REPEATED OVER AND OVER AGAIN IN THE SHOP

Patterns that recur over and over again can be parameterized and kept around for easy, immediate use. For example:

- 1) Bolt hole patterns.
- 2) Slotting.
- 3) Angular patterns, 5 holes at 30 degrees 1 inch apart.
- 4) Specialty milling such as soft jaws.
- 5) Matrix Patterns, 12 across and 15 down.
- 6) Flycutting a surface, 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.

PROBING

Probing enhances the capabilities of the machine in many ways. Below is just a hint of the possibilities.

- 1) Profiling of a part to determine unknown dimensions for later machining.
- 2) Tool calibration for offset and wear values.
- 3) Inspection prior to machining to determine material allowance on castings.
- 4) Inspection after machining to determine parallelism and flatness values as well as location.

Macros allow less experienced personnel to operate the machine. Conditions can be detected and custom operator messages or alarms can be displayed on the console to notify the operator.

11.2 MACRO SUBROUTINE CALL (G65)

G65 is the command that calls a subroutine with the ability to pass arguments to it. The format follows.

`[N#####] G65 P##### [L####] [arguments] ;`

Anything enclosed in brackets is optional. This should not be confused with expression brackets that are explained below. The G65 command requires a P address parameter corresponding to any program number currently in memory. When the optional L address is used the macro call is repeated the specified number of times.

In Example 1, subroutine 1000 is called once with no parameters passed to the routine. G65 calls are similar to, but not the same as, M98 calls. Up to four G65 calls can be made at the same time, (Nesting four deep).

Example 1:

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

In Example 2, subroutine 9010 is designed to drill a sequence of holes along a line whose slope is determined by the X and Y arguments that are passed to it in the G65 command line. The Z drill depth is passed as Z, the feed rate is passed as F, and the number of holes to be drilled is passed as T. The line of holes is drilled

starting from the current tool position when the macro subroutine is called.

Example 2:	G00 G90 X1.0 Y1.0 Z.05 ;	(Position tool)
	G65 P9010 X.5 Y.25 Z.05 F10. T10 ;	(Call 9010)
	G28 M30 ;	
	O9010 ;	(Diagonal hole pattern)
	F#9 ;	(F=Feed rate)
	WHILE [#20>0] D01	(Repeat T times)
	G91 G81 Z#26 ;	(Drill To Z depth)
	#20=#20-1 ;	(Decrement counter)
	IF [#20 EQ 0] GOT05 ;	(All holes drilled)
	G00 X#24 Y#25 ;	(Move along slope)
	N5 END1 ;	
	M99 ;	(Return to caller)

11.3 ALIASING

Aliasing is a means of assigning a G code to a G65 P##### sequence. For instance, in Example 2 it would be easier if one could write:

```
G93 X.5 Y.25 Z.05 F10. T10 ;
```

Here, we have substituted G93, an unused G code, for G65 P9010. In order for the above block to work we must set the parameter associated with subroutine 9010 to 93.

Program numbers 9010 through 9019 are reserved for G code aliasing. The following table lists which HAAS parameters are reserved for macro subroutine aliasing.

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

G00, G65, G66, and G67 can not be aliased. All other codes between 1 and 255 can be used for aliasing.

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

11.4 MACRO ARGUMENTS

The arguments in a G65 statement are a means of sending values to and setting the local variables of a called macro subroutine.

In Example 2 above, the arguments X and Y are passed to the macro subroutine local variables. Local variable #24 is associated with X and is set to 0.5. Similarly, Local variable #25 is associated with Y and is set to 0.25.

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

Alphabetic Addressing

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

Alternate Alphabetic Addressing

Address:	A	B	C	I	J	K	I	J	K	I	J
Variable:	1	2	3	4	5	6	7	8	9	10	11
Address:	K	I	J	K	I	J	K	I	J	K	I
Variable:	12	13	14	15	16	17	18	19	20	21	22
Address:	J	K	I	J	K	I	J	K	I	J	K
Variable:	23	24	25	26	27	28	29	30	31	32	33

Arguments accept any floating point value to four decimal places. If you are in metric, the control will assume thousandths (.000). In Example 3 below, local variable #7 will receive .0004.

If a decimal is not included in an argument value, such as:

```
G65 P9910 A1 B2 C3
```

The values are passed to macro subroutines according to the following table:

Integer Argument Passing (no decimal point)

Address:	A	B	C	D	E	F	G	H	I	J	K
Variable:	.001	.001	.001	1.	1.	1.	-	1.	.0001	.0001	.0001
Address:	L	M	N	O	P	Q	R	S	T	U	V
Variable:	1.	1.	-	-	-	.0001	.0001	1.	1.	.0001	.0001
Address:	W	X	Y	Z							
Variable:	.0001	.0001	.0001	.0001							

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

Example 3: G65 P2000 I1 J2 K3 I4 J5 K6 ;

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

11.5 MACRO CONSTANTS

Constants are floating point values placed in a macro expression. They can be combined with addresses A...Z or they can stand alone when used within an expression. Examples of constants are .0001, 5.3 or -10.

11.6 MACRO VARIABLES

There are three categories of macro variables: *system* variables, *global* variables, and *local* variables.

VARIABLE USAGE

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

Variables are decimal values that are represented as floating point numbers. If a variable has never been used, it can take on a special "undefined" value. This indicates that it has not been used. A variable can be set to undefined with the special variable #0. #0 has the value of undefined or 0.0 depending on the context it is used in. More about this later. Indirect references to variables can be accomplished by enclosing the variable number in brackets.

#[<expression>]

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

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

This sets the variable #3 to the value 6.5.

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

In the block

```
N1 G0 G90 X1.0 Y0 ;
```

the variables can be set to the following values:

```
#7=0;
#11=90;
#1=1.0;
#2=0.0;
```

and the block replaced by:

```
N1 G#7 G#11 X#1 Y#2 ;
```

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

LOCAL VARIABLES

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

Local Variables and Corresponding Address

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

Note that variables 10, 12, 14..16 and 27..33 do not have corresponding address arguments. They can be set if a sufficient number of I, J and K arguments are used as indicated above in the section about arguments.

Once in the macro subroutine, the local variables can be read and modified by referencing the variable numbers 1..33.

When the L argument is used to do multiple repetitions of a macro subroutine, the arguments are set only on the first repetition. This means that if local variables 1..33 are modified in the first repetition, 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 subroutine via an M98 does not nest the local variables. Any local variables referenced in a subroutine called by an M98 are the same variables and values that existed prior to the M98 call.

GLOBAL VARIABLES

Global variables are variables that are accessible at all times. There is only one copy of each global variable. Global variables occur in two ranges: 100..199 and 500..599. The global variables remain in memory when power is turned off. They are not cleared as in the FANUC controls.

SYSTEM VARIABLES

System variables give the programmer the ability to interact with a variety of control parameters and settings. By setting a system variable, the function of the control can be modified or altered. By reading a system variable, a program can modify its behavior based on the value in the variable. Some system variables have a READ ONLY status. This means that they can not be modified by the programmer. A brief table of currently implemented system variables follows with an explanation of their use.

VARIABLES**USAGE**

#1000-#1031	32 x 1-BIT DISCRETE INPUTS
#1100-#1131	32 x 1-BIT DISCRETE OUTPUTS
#2000-#2999	TOOL OFFSETS
#3000	PROGRAMMABLE ALARM WITH MESSAGE
#3001	MILLISECOND TIMER
#3002	HOUR TIMER
#3004	OVERRIDE CONTROL
#3006	PROGRAMMABLE STOP WITH MESSAGE
#4001-#4021	LAST BLOCK GROUP CODES
#4101-#4126	LAST BLOCK ADDRESS DATA

NOTE: Mapping of 4101 to 4126 is the same as the alphabetic addressing of "Macro Arguments" section. For example, 4111 holds H code value and 4107 holds D code value.

#5001-#5005	LAST BLOCK TARGET POSITION
#5021-#5025	CURRENT MACHINE COORD POSITION
#5041-#5045	CURRENT WORK COORD POSITION
#5061-#5065	CURRENT SKIP SIGNAL POSITION
#5081-#5085	TOOL LENGTH COMPENSATION
#5221-#5225	G54 OFFSET VALUES
#5241-#5245	G55 " "
#5261-#5265	G56 " "
#5281-#5285	G57 " "
#5301-#5305	G58 " "
#5321-#5325	G59 " "
#7001-#7005	G110 ADDITIONAL OFFSET VALUES
#7021-#7025	G111 " " "
#7381-#7385	G129 " " "

11.7 SYSTEM VARIABLES IN-DEPTH

This section fully describes system variables.

1-BIT DISCRETE INPUTS

For a complete description of discrete inputs, refer to the "Technical Reference" section. Inputs designated as "spare" can be connected to external devices and used by the programmer.

#1000-#1020	Reserved for HAAS Controller use.
#1021	Spare
#1022	Spare
#1023	Spare
#1024-#1028	Reserved for HAAS Controller use.
#1029	Skip signal
#1030	Spare
#1031	Spare

1-BIT DISCRETE OUTPUTS

#1124 Some HAAS controls have eight discrete outputs designated as SPARE. These extra relays usually must be specially ordered. The user can actuate these spare output relays by

reading or writing to variables #1124-#1131. An assignment of "1" sets the relay, whereas a reference reads the relay. For example:

```
#1124=1;           (Turns #1124 relay on)
#101=#3001+1000;  (101 is second from now)
WHILE [[#101 GT #3001] AND [#1125 EQ 0]] D01
END1              (Wait here 1 second or until relay #1125 goes high)
#1124=0;          (Turns #1124 relay off)
```

TOOL OFFSETS

HAAS macros have been implemented with FANUC control memory C option in mind. This means that each tool offset has a length (H) and radius (D) along with associated wear values.

#2001-#2100	H geometry offsets (1-100) for length.
#2200-#2300	H geometry wear (1-100) for length.
#2401-#2500	D geometry offsets (1-100) for diameter.
#2601-#2700	D geometry wear (1-100) for diameter.

PROGRAMMABLE MESSAGES

#3000 ALARMS can be programmed. A programmable alarm will act just like HAAS internal alarms. An alarm is generated by setting the macro variable #3000 to a number between 1 and 999.

#3000= 15 (MESSAGE PLACED INTO ALARM LIST) ;

When this is done, ALARM flashes in the lower right hand corner of the display and the text in the next comment is placed into the alarm list. The alarm number (in this example, 15) is added to 1000 and used as an alarm number. If an alarm is generated in this manner all motion stops and the program must be reset to continue. Programmable alarms can always be identified in alarm history because the alarm numbers range between 1000 and 1999.

The first 34 characters of the comment will be used for the alarm message.

TIMERS

HAAS macros supports access to two timers. These timers can be set to a value by assigning a number to the respective variable. A program can then later read the variable and determine the time passed since the timer was set. Timers can be used to emulate dwell cycles, determine part to part time or wherever time dependent behavior is desired.

#3001 MILLISECOND TIMER - The millisecond timer is updated every 20 milliseconds and thus activities can be timed with an accuracy of only 20 milliseconds. At POWER ON, the millisecond timer is reset. The timer has a limit of 497 days. The whole number returned after accessing #3001 represents the number of milliseconds.

#3002 HOUR TIMER - The hour timer is similar to the millisecond timer except that the number returned after accessing #3002 is in hours. The hour and millisecond timers are independent of each other and can be set separately.

SYSTEM OVERRIDES

#3004 Variable #3004 is a bitmapped variable that overrides specific control features during runtime.

The first bit disallows FEED HOLD from the keypad. If you do not want feed hold to be executed during any section of code, then bracket that code with assignments to variable #3004. Assigning "1" to #3004 disables the console's feed hold button. Assigning "0" to #3004 re-enables the FEED HOLD button. For example:

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

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

#3004	MAP	FEED HOLD
0	000	E
1	001	D
2	010	E
3	011	D
4	100	E
5	101	D
6	110	E
7	111	D

PROGRAMMABLE STOP

#3006 Stops can be programmed. A programmable stop acts like an M00. In the following example, when the assignment statement is executed, the first 15 characters of the comment are displayed in the messaging area on the lower left part of the screen above the command input line. The control stops and waits for a cycle start from the operator. Upon cycle start, operation continues with the next block after the assignment statement.

IF [#1 EQ #0] THEN #3006=101(ARG.A REQUIRED);

LAST BLOCK (MODAL) GROUP CODES

#4001-#4021 The grouping of G codes permits more efficient processing. G codes with similar functions are usually under the same group. For instance, G90 and G91 are under group 3. Variables have been set aside to store the last or default G code issued for any of 21 groups. By reading the group code, a macro program can change its behavior based on the contents of the group code. If 4003 contains 91, then a macro program could determine that all moves should be incremental rather than absolute. There is no associated variable for group zero, group zero G codes are NON-modal.

LAST BLOCK (MODAL) ADDRESS DATA

#4101-#4126 Address codes A..Z (excluding G) are also maintained as modal values. The modal information represented by the last block interpreted by the lookahead process is contained in variables 4101 through 4126. The numeric mapping of variable numbers

to alphabetic addresses corresponds to the mapping under alphabetic addresses. For instance, the value of the previously interpreted **D** address is found in #4107 and the last interpreted **I** value is #4104.

LAST TARGET POSITION

#5001-#5005 The final programmed point, target position, for the most recent motion block can be accessed through variables #5001-#5005, X, Y, Z and A respectively. Values are given in the current work coordinate system and can be used while the machine is in motion.

CURRENT MACHINE COORD POSITION

#5021-#5025 The current position in machine coordinates can be obtained through #5021-#5025, X, Y, Z, and A respectively. The values CANNOT be read while the machine is in motion. #5023 (Z) represents the value after tool length compensation has been applied.

CURRENT WORK COORD POSITION

#5041-#5045 The current position in the current work coordinates can be obtained through #5041, 5044, X, Y, Z and A respectively. The values can NOT be read while the machine is in motion. #5043 (Z) represents the value after tool length compensation has been applied.

CURRENT SKIP SIGNAL POSITION

#5061-#5065 The position where the last skip signal was triggered can be obtained through #5061 #5064, X, Y, Z and A respectively. Values are given in the current work coordinate system and can be used while the machine is in motion. #5063 (Z) represents the value after tool length compensation has been applied.

TOOL LENGTH COMPENSATION

#5081-#5085 The current total tool length compensation that is being applied to the tool is returned. This includes tool length offset referenced by the current modal value set in H (#4008) plus the wear value.

OFFSETS

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

#5201-#5205	G52 X, Y, Z, A, B OFFSET VALUES
#5221-#5225	G54 " " " " " " "
#5241-#5245	G55 " " " " " " "
#5261-#5265	G56 " " " " " " "
#5281-#5285	G57 " " " " " " "
#5301-#5305	G58 " " " " " " "
#5321-#5325	G59 " " " " " " "

#7001-#7005	ADDITIONAL X, Y, Z, A, B OFFSET VALUES
#7021-#7025	" " " " " " " "
" "	" " " " " " " "
#7381-#7385	ADDITIONAL X, Y, Z, A, B OFFSET VALUES

11.8 ADDRESS CONSTANT SUBSTITUTION

The usual method of setting control addresses A..Z is by appending a constant to the address. For instance,

G01 X1.5 Y3.7 F20. ;

sets addresses G, X, Y and F to 1, 1.5, 3.7 and 20.0 respectively and thus instructs the control to move linearly, G01, to position X=1.5 Y=3.7 at a feed rate of 20 inches per minute. Macro syntax allows the constants to be replaced with any variable or expression in any section of code (i.e., you do not have to be in a macro subroutine).

The previous statement can be replaced by the following code:

```
#1=1;
#2=.5;
#3=3.7;
#4=20;
G#1 X[#1+#2] Y#3 F#4 ;
```

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

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

If the value of the variable does not agree with the range of the address, then the usual control alarm will result. For instance, the following code would result in a range error alarm because tool diameter numbers range from 0..50.

```
#1=75;
D#1;
```

When a variable or expression is used in place of an address constant, then the floating point value is rounded to the least significant digit. If #1=.123456, then G1X#1 would move the machine tool to .1235 on the X axis. If the control is in the metric mode, the machine would be moved to .123 on the X axis.

When an UNDEFINED variable is used to replace an address constant, then that address reference is ignored. For example, if #1 is undefined then the block

G00 X1.0 Y#1 ;

becomes

G00 X1.0.

No Y movement takes place.

11.9 MACRO STATEMENTS

Macro statements are lines of code that allow the programmer to manipulate the control with features similar to any standard programming language. Included are functions, operators, conditional and arithmetic expressions, assignment statements, and control statements.

Functions and operators are used in expressions to modify variables or values. The operators are essential to expressions while functions make the programmer's job easier.

FUNCTIONS

Functions are built-in routines that the programmer has available to use. All functions have the form **<function_name>[argument]**. Functions can be passed any expression as arguments. Functions return floating point decimal values. The function provided with the HAAS control are as follows:

FUNCTION	ARGUMENT	RETURNS	NOTES
SIN[]	Degrees	Decimal	Sine
COS[]	Degrees	Decimal	Cosine
TAN[]	Degrees	Decimal	Tangent
ATAN[]	Decimal	Degrees	Arctangent Same as FANUC ATAN[]/[1]
SQRT[]	Decimal	Decimal	Square root
ABS[]	Decimal	Decimal	Absolute value
ROUND[]	Decimal	Decimal	Round off a decimal
FIX[]	Decimal	Integer	Truncate fraction
ACOS[]	Degrees	Decimal	Arccosine
ASIN[]	Degrees	Decimal	Arcsine
#[]	Integer	Integer	Variable Indirection
DPRNT[]	ASCII text		External Output

NOTES ON FUNCTIONS

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

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

When round is used in an address expression, then the argument of round is rounded to the addresses significant precision. For *metric* and *angle* dimensions, three-place precision is the default. For *inch*, four-place precision is the default. Integral addresses such as D, T and H are rounded normally.

```
#1= 1.00333 ;
GO X[ #1 + #1 ] ;
    (Table moves to 1.0067) ;
GO X[ ROUND[ #1 ] + ROUND[ #1 ] ] ;
    (Table moves to 1.0066) ;
GO A[ #1 + #1 ] ;
```

```

      (Axis moves to 1.007) ;
GO A[ ROUND[ #1 ] + ROUND[ #1 ] ] ;
      (Axis moves to 1.006) ;
D[1.67]      (Diameter 2 is made current) ;

```

OPERATORS

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

ARITHMETIC OPERATORS

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

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

LOGICAL OPERATORS

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

OR - logically OR two values together
 XOR - Exclusively OR two values together
 AND - Logically AND two values together

Examples:

```

#1=1.0;      0000 0001
#2=2.0;      0000 0010
#3=#1 OR #2  0000 0011

```

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

```

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

```

Here control will transfer to block 1 GOTO1 because #1 GT 3.0 evaluates to 1.0 and #2 LT 10 evaluates to 1.0, thus 1.0 AND 1.0 is 1.0 (TRUE) and the GOTO occurs.

As can be seen from the previous examples, CARE must be taken when using logical operators so that the desired result is achieved.

BOOLEAN OPERATORS

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

EQ - Equal to
NE - Not Equal to
GT - Greater Than
LT - Less Than
GE - Greater than or Equal to
LE - Less Than or Equal to

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

Example

IF [#1 EQ 0.0] GOTO100;

WHILE [#101 LT 10] DO1;
#1=[1.0 LT 5.0];
IF [#1 AND #2 EQ #3] GOTO1 Description

Jump to block 100 if variable #1 equals 0.0.

While variable #101 is less than 10 then repeat loop DO1..END1.
Variable #1 is set to 1.0 (TRUE).

If variable #1 logically ANDed with variable #2 is equal to the value in #3 then control is transferred to block 1.

EXPRESSIONS

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

CONDITIONAL EXPRESSIONS

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

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

N50 M99 P10 ;

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

N50 [#100 LT 10] M99 P10 ;

In this case, the branch occurs only when #100 is less than 10, otherwise processing continues with the next

program line in sequence. In the above, the conditional M99 can be replaced with

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

ARITHMETIC EXPRESSIONS

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

Examples of Arithmetic expressions: #101=#145*#30;

```
#1=#1+1;
```

```
X[#105+COS[#101]];
```

```
#[#2000+#13]=0;
```

ASSIGNMENT STATEMENTS

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

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

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

```
O0300 (Initialize an array of variables) ;
N1 IF [#2 NE #0] GOTO2 (B=base variable) ;
#3000=1(BASE VARIABLE NOT GIVEN) ;
N2 IF [#19 NE #0] GOTO3 (S=size of array);
#3000=2(SIZE OF ARRAY NOT GIVEN) ;
WHILE [#19 GT 0] DO1 ;
#19=#19-1 (DECREMENT COUNT) ;
#[#2+#19]=#22 (V=value to set array to) ;
END1 ;
M99 ;
```

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

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

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

CONTROL STATEMENTS

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

UNCONDITIONAL BRANCH (GOTONNN AND M99 PNNNN)

In the HAAS control, there are two methods of branching unconditionally. An unconditional branch will

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

COMPUTED BRANCH (GOTO#n AND GOTO[expression])

Computed branching allows the program to transfer control to another block in the same subprogram. The block can be computed on the fly, as in the case of the GOTO[expression] form, or the block can be passed in through a local variable, as in the GOTO#n form.

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

The following code skeleton could be developed to make a program that serializes parts:

```
O9200 (Engrave digit at current location.)
;
(D=DECIMAL DIGIT TO ENGRAVE) ;
IF [[#7 NE #0] AND [#7 GE 0] AND [#7 LE 9]] GOTO99
#3000=1 (INVALID DIGIT)
;
N99
#7=FIX[#7]      (TRUNCATE ANY FRACTIONAL PART) ;
;
GOTO#7         (NOW ENGRAVE THE DIGIT) ;
;
N0             (DO DIGIT ZERO)
...
M99
;
N1             (DO DIGIT ONE)
;
M99
;
N2             (DO DIGIT TWO)
;
;
...
;
(etc.,...)
```

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

```
G65 P9200 D5;
```

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

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

```
...
M99
N2 (1030=1, 1031=0)
```

```
...
M99
N3 (1030=1, 1031=1)
```

```
...
M99
```

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

CONDITIONAL BRANCH (IF AND M99 PNNNN)

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

IF [<conditional expression>] GOTO n

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

```
IF [#1 NE 0.0] GOTO5;
could also be:
```

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

In this statement, if the variable #1 contains anything but 0.0, or the undefined value #0, then branching to block 5 will occur; otherwise, the next block will be executed. If portability to a control other than HAAS is desired, then it is recommended that the conditional operators be used.

In the HAAS control, a conditional expression can also be used with the M99 Pnnnn format, providing that macros have been enabled. An example is as follows:

```
G0 X0 Y0 [#1EQ#2] M99 P5;
```

Here, the conditional is for the M99 portion of the statement only. The machine tool is instructed to X0, Y0 whether or not the expression evaluates to TRUE or FALSE. Only the branch, M99, is executed based on the value of the expression. It is recommended that the IF GOTO version is used if portability is desired.

CONDITIONAL EXECUTION (IF THEN)

Execution of control statements can also be achieved by using the IF THEN construct. The format is:

```
IF [<conditional expression>] THEN <statement> ;
```

This format is traditionally used for conditional assignment statements such as:

```
IF [#590 GT 100] THEN #590=0.0 ;
```

Here, variable #590 is set to zero when the value of #590 exceeds 100.0. In the HAAS control, if a conditional

evaluates to FALSE (0.0), then the remainder of the IF block is ignored. This means that control statements can also be conditioned so that we could write something like:

```
IF [#1 NE #0] THEN G1 X#24 Y#26 F#9 ;
```

This executes a linear motion only if variable #1 has been assigned a value. You might try something like this:

```
IF [#1 GE 180] THEN #101=0.0 M99 ;
```

This says that if variable #1 (address A) is greater than or equal to 180, then set variable #101 to zero and return from the subroutine.

Here is an example of an IF statement that branches if a variable has been initialized to contain any value. Otherwise, processing will continue and an alarm will be generated. Remember, when an alarm is generated, program execution is halted.

```
N1 IF [#9NE#0] GOTO3 (TEST FOR VALUE IN F) ;
N2 #3000=11(NO FEED RATE) ;
N3 (CONTINUE) ;
```

ITERATION/LOOPING (WHILE DO END)

Essential to all programming languages is the ability to execute a sequence of statements a given number of times or to loop through a sequence of statements until a condition is met. Traditional G coding allows this with the use of the L address. A subroutine can be executed any number of times by using the L address.

```
M98 P2000 L5 ;
```

This is limited since you can not terminate execution of the subroutine on condition. Macros allows more flexibility with the WHILE-DO-END construct. The syntax is as follows:

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

This executes the statements between DOn and ENDn as long as the conditional expression evaluates to TRUE. The brackets in the expression are necessary. If the expression evaluates to FALSE, then the block after ENDn is executed next. WHILE can be abbreviated to WH. The DOn-ENDn portion of the statement is a matched pair. The value of n is 1..3. This means that there can be no more than three nested loops per subroutine. A nest is basically a loop within a loop. A good example of how nesting of WHILE loops can be used is in defining a matrix.

```
#101= 3 ;
#102= 4 ;
G0 X#101 Y4. ;
F2.5 ;
WH [ #101 GT 0 ] DO1 ;
  #102= 4 ;
  WH [ #102 GT 0 ] DO2 ;
    G81 X#101 Y#102 Z-0.5 ;
    #102= #102 - 1 ;
  END2 ;
  #101= #101 - 1 ;
END1 ;
```

```
;
M30 ;
```

This program drills a 3 x 4 matrix hole pattern.

Although nesting of WHILE statements can only be nested to three levels, there really is no limit since each subroutine can have up to three levels of nesting. If there ever is a need to nest to a level greater than 3, then the segment containing the three lowest levels of nesting can be made into a subroutine thus overcoming the limitation.

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

```
#3001=0 (WAIT 500 MILLISECONDS) ;
WH [#3001 LT 500] D01 ;
END1 ;
```

<other statements>

```
#3001=0 (WAIT 300 MILLISECONDS) ;
WH [#3001 LT 150] D01 ;
END1 ;
```

This is valid code.

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

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

```
D01 ;
<statements>
END1 ;
```

executes until the RESET key is pressed.

CAUTION! The following code can be confusing:

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

In the above, an alarm results indicating no "then" was found, here "then" refers to the D01. Change D01 (zero) to D01 (letter O).

11.10 COMMUNICATION WITH EXTERNAL DEVICES - DPRNT[]

Macros allow additional capabilities to communicate with peripheral devices. One can do digitizing of parts, provide runtime inspection reports, or synchronize controls with user provided devices. The commands provided for this are POPEN, DPRNT[] and PCLOS.

COMMUNICATION PREPARATORY COMMANDS

POPEN and PCLOS are not required on the HAAS mill. It has been included so that programs from different controls can be ported to the HAAS. On some controls POPEN is required prior to using a DPRNT statement. POPEN prepares the device on the serial port by sending it a DC2 code. PCLOS terminates communication with external devices by sending it a DC4 code.

FORMATTED OUTPUT

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

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

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

```
[00] [54] [45] [36] /* not legal formats */
```

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

A carriage return is sent out after every DPRNT block.

DPRNT[] EXAMPLES

```
#1= 1.5436 ;
DPRNT[X#1[44]*Z#1[03]*T#1[40]] ;
      outputs:  X1.5436 Z 1.544 T 1
DPRNT[***MEASURED*INSIDE*DIAMETER***] ;
      outputs:  MEASURED INSIDE DIAMETER
```

```
DPRNT[ ] ;
      outputs:  (no text, only a carriage return)
```

```
#1=123.456789 ;
DPRNT[X-#1[25]] ;
      outputs:  X-123.45679 ;
```

11.11 RUNTIME EXECUTION

DPRNT statements are executed at block interpretation time. This means that the programmer must be careful about where the DPRNT statements appear in the program, particularly if the intent is to print out positional information. Generally, a program is interpreted many blocks ahead in order to prevent the machine from pausing between movements.

G103 is useful for limiting lookahead. If you wanted to limit lookahead interpretation to one block, you would include the following command at the beginning of your program: (This actually results in a two block lookahead.)

G103 P1 ;

To cancel the lookahead limit, then issue a G103 P0 ;. G103 can not be used when cutter compensation is active.

11.12 OPERATION NOTES

This section explains the additional screens and operator actions that come with macros.

Macro variables can be saved and restored to RS-232 or the optional floppy, much like parameters, settings, and offsets. Refer to the "Part Program Input / Output" section for RS-232 sending and receiving of macro variables, or the "Floppy Operation" section for sending and receiving them with this method.

VARIABLE DISPLAY PAGE

The macro variables are displayed and can be modified through the current commands display. The variable display is located after the operation timers display. To get to this page, press CURNT COMDS and use the page up/down key.

As the control interprets a program, the variable changes are displayed on the variable display page and results can be viewed.

Pages contain up to 32 variables and the display can be "paged" by pressing the left/right arrow keys.

Setting of a variable is accomplished by entering a value and then pressing the WRITE key. The variable that is highlighted on the screen is the variable that is affected.

Searching for a variable can be done by entering the variable number and pressing the up/down arrow. The page will change to the one that contains that variable and the entered variable will become the highlighted item.

The variables displayed represent the values of the variables at program interpretation time. At times, this may be up to 15 blocks ahead of the actual machine activity. Debugging of programs can be made easier by inserting a G103 at the beginning of a program to limit block buffering and then removing the G103 block after debugging is completed.

EDITING

For the most part, the editing of macro programs from the control is the same as before. There are a few peculiarities to be aware of.

Editing macro statements is more open than previously. For instance, it is possible to place a floating point constant within a standard G-code block, but it doesn't make much sense, and the control will raise an alarm at runtime. For all instances of improperly structured or improperly placed macro statements, the control will raise an appropriate alarm. Most of these alarms have been put off until runtime so that operator editing can be more flexible. Be careful when editing expressions. Brackets must be balanced and you will not receive an alarm until runtime.

The DPRNT[] function can be edited much like a comment. You can delete it or move it as a whole item, or you can edit individual items within the brackets. Variable references and format expressions must be altered as a whole entity. If you wanted to change [24] to [44], place the cursor so that [24] is highlighted, enter [44] and press the write key. Remember, you can use the crank handle to maneuver through long DPRNT[] expressions.

Addresses with expressions can be somewhat confusing. In this case, the alphabetic address stands alone. For instance, the following block contains an address expression in X:

```
G1 G90 X [ COS[ 90 ] ] Y3.0 (CORRECT) ;
```

Here, the X and brackets stand alone and are individually editable items. It is possible, through editing, to delete the entire expression and replace it with a floating point constant.

```
G1 G90 X 0 Y3.0 (!!! WRONG !!!) ;
```

The above block will result in an alarm at runtime. The correct form looks as follows:

```
G1 G90 X0 Y3.0 (CORRECT) ;
```

Note that the zero is attached to X. REMEMBER when you see an alpha character standing alone it is an address expression.

11.13 FANUC-STYLE MACRO FEATURES NOT INCLUDED IN HAAS CNC CONTROL

This section lists the FANUC macro features that have not yet been implemented.

<p>M ALIASING P ADDRESS IN G65 G66 MODAL G66.1 MODAL G67 MODAL CANCEL M98 M98 M98 SKIP/N #3007 #3011 #3012 #3901 #3902 #4201-#4320 #5101-#5106</p>	<p>REPLACE G65 Pnnnn WITH Mnn PROGS 9020-9029. NEED TO ALLOW P ADDRESS CALL IN EVERY MOTION BLOCK CALL IN EVERY BLOCK</p>
<p>ADDITIONAL OFFSETS NAMES FOR VARIABLES ATAN []/[] BIN [] BCD [] FUP [] LN [] EXP [] ADP [] BPRNT []</p>	<p>ALIASING, T CODE PROG 9000, VAR #149, ENABLE BIT ALIASING, S CODE PROG 9029, VAR #147, ENABLE BIT ALIASING, B CODE PROG 9028, VAR #146, ENABLE BIT N=1..9 MIRROR IMAGE ON FLAG EACH AXIS YEAR/MONTH/DAY HOUR/MINUTE/SECOND TOTAL NUMBER OF PARTS REQUIRED NUMBER OF PARTS CURRENT BLOCK MODAL DATA CURRENT SERVO DEVIATION</p> <p>G54.1P## FORMAT FOR DISPLAY PURPOSES ARCTANGENT, FANUC VERSION CONVERSION FROM BCD TO BIN CONVERSION FROM BIN TO BCD TRUNCATE FRACTION CEILING NATURAL LOGARITHM BASE E EXPONENTIATION RE-SCALE VAR TO WHOLE NUMBER</p>

The following can be used as an alternative methods for achieving the same results for a few unimplemented FANUC macro features.

GOTO-nnnn

Searching for a block to jump in the negative direction, i.e. backwards through a program), is not necessary if you use unique N address codes.

A block search is made starting from the current block being interpreted. When the end of the program is reached, searching continues from the top of the program until the current block is encountered.

III. MACHINE OPERATION

1. WORK COORDINATE SYSTEM

These machines have three linear axes named **X**, **Y**, and **Z**. The **X**-axis moves the table left and right, the **Y**-axis moves it to and from the operator and the **Z** moves the milling head up and down. The machine zero position is the upper right corner of the mill table. All moves from this point are in a negative machine direction. If a rotary table is connected, an additional **A**-axis work offset is provided.

The work offset display is found on the offset display by pushing the PAGE UP key. You can display and manually enter work offsets from here. The work coordinate systems on a control with a fifth axis have all been expanded to accommodate **B**, the fifth axis. Work coordinate offsets can be set for the **B** axis in the offset display. Note that the auxiliary axes **C**, **U**, **V**, and **W** do not have any offsets; they are always programmed in machine coordinates.

The Home or Machine-Zero position is **X0**, **Y0**, **Z0**. Travel of these axes is limited in the negative direction by stored stroke limits defined in the parameters. Travel in the positive direction for the **X** and **Y** axes is limited simply to values less than zero. Positive travel for the **Z**-axis is limited to the highest position used for tool changing (about **Z4.5**). In addition, positive travel for all axes is limited by the home switch which acts as a limit switch.

Before a tool can machine your part, the control must know where your part is. The work coordinate system tells the control the distance from the work zero point of your part to the machine zero position. The work zero point of the part is decided by the programmer and usually is the common point where all print dimensions are referenced from. The machine zero position is fixed by the machine on power up and does not change. The operator must determine this distance and enter the value.

This control automatically chooses the **G54** system on power up. If you do not wish to use this system, zero out the values in the **G54 X**, **Y**, and **Z** or select another work offset.

The **G54** through **G59** or **G110** through **G129** offsets can be set by using the PART ZERO SET key. Position the axes to the work zero point of your part. Using the cursor, select the proper axis and work number. Press the PART ZERO SET key and the current machine position will be automatically stored in that address. This will work with only the work zero offsets display selected. Note that entering a nonzero **Z** work offset will interfere with the operation of an automatically entered tool length offset.

Work coordinate numbers are usually entered as positive numbers, except when Parameter 57, bit "Neg. Work Offset" is set to 1. In this case, the work coordinate numbers are entered as negative numbers.

Work coordinates are entered into the table as a number only, that is, to enter an **X** value of **X2.00** into **G54** you would cursor over to the column and enter the number 2.0 only.

The mirror function can change the direction of motion along any of the axes. If any one of these are selected, the display will show the status. Mirror image will reflect programmed motion around your work coordinate zero point. Be careful that mirror of only one of **X** or **Y** will cause the cutter to move along the opposite side of a cut. In addition, if mirror is selected for only one axis of a circular motion plane, circular motion **G02** and **G03** are reversed and left side and right side cutter compensation **G41** and **G42** are reversed. Settings 45 through 48 are used to select mirror image.

See the sections on **G52**, **G92**, and Setting 33 for more on work coordinate systems.

Offsets can be sent and received with the RS-232 (or optional RS-422) port. See the "Data Input / Output" section for information on how to do this.

2. SETUP PROCEDURES

The following sequence of operations is strongly recommended for setting up this machine:

- 1) Load a program into memory. This is either manually entered or downloaded from a CAM package via the RS-232 (or optional RS-422) interface.
- 2) Determine the tools needed and get them ready.
- 3) Use a vise or fixture to hold the part and mount it on the mill.
- 4) Locate the X and Y zero points of your program on the part. Usually these points will coincide with the print reference point where dimensioning begins and needs to be clearly indicated by the programmer. Use an edge finder or indicator to locate this point with the handle function. After locating the programmed zero point, push the display offset key and page down until the work coordinate page appears. Use the cursor to get to work set G54 X. Push the PART ZERO SET button and the X-axis machine value will be stored at this location. Cursor to the G54 Y location and repeat the above. You have now told the machine where part zero is located. Usually Z and A values will not have to be set and should be zero.
- 5) Remove any tools from changer and MDI a T1 M6 command to install tool #1 into spindle (it should be empty). Put your tool #1 into the spindle using the TOOL RELEASE button. Push the OFSET key and page down to get to the tool offset page and cursor to tool #1. Do not install any tools directly into the carousel. Use MDI or ATC FWD/REV to retrieve tools.
- 6) Push the Z- JOG key until you are close to the top of your part. (The top of your part should be Z0). Use the Handle to accurately position the tool edge to Z0. Push the TOOL OFSET MESUR key and the Z machine value will be stored in tool offset #1. Note that this automatic offset measurement works with G43 only and the work Z offset must be zero.
- 7) Push the NEXT TOOL key and the Z-axis will retract to tool change and tool #2 (empty) will be installed in the spindle. Put your tool #2 into the spindle and jog to Z zero as you did for tool #1. The cursor will automatically be on offset #2. Push TOOL OFSET MESUR.
- 8) Repeat this procedure until all tools are measured and installed.
- 9) MDI a T1 M6 to return to tool #1.
- 10) You are now ready to run your program.
- 11) Please note that in order to load and measure all of your tools, you do not need to use any buttons other than JOG, TOOL OFSET MESUR, and NEXT TOOL. Also note that this automatic offset measurement works with G43 only.

3. DRY RUN OPERATION

The DRY RUN function is used to check a program quickly without actually cutting parts. DRY RUN is selected by pressing the DRY RUN button while in MEM or MDI mode. When in DRY RUN, all rapids and feeds are run at the DRY RUN speed selected from the JOG speed buttons. The bottom of the screen will display the rate as 100, 10, 1.0 or 0.1 inches per minute.

DRY RUN cannot be turned on while a program is running. It can only be turned on or off when a program has completely finished or is reset. The first push of the DRY RUN button turns on this function and the second push will turn it off again. DRY RUN will still make all of the requested tool changes. The speed used in DRY RUN can be changed at any time and the operator can then check that the motions that are programmed are exactly what were intended. Note that Graphics mode is just as useful and may be even safer since it does not begin moving the machine before the program is checked.

4. DISPLAYS

You can select any of the following displays using the eight DISPLAY select buttons:

PRGRM / CONVRS	To show or edit the program selected OR to select conversational mode (QuickCode)
POSIT	To show the axes positions.
OFFSET	To show or enter working offsets.
CURNT COMDS	To show current commands and times.
ALARM / MESGS	To show alarms and user messages.
PARAM / DGNOS	To show parameters and diagnostic data.
SETNG / GRAPH	To show or enter settings OR to select graphics simulation mode.
HELP / CALC	To show the help data and calculator.

In addition, when a program is running, you may press LIST PROG to select a list of the programs in memory. This is helpful in determining which programs can be edited in BACKGROUND EDIT, which is selected from the PROGRAM DISPLAY.

The CRT will ALWAYS show some of the current conditions selected in the control. These are fixed status displays that describe the condition of the machine. The following conditions are displayed on the screen:

The selected display in the top left corner,
 The selected mode in parentheses,
 The presently selected program in the top right corner,
 The most recent line number in the top right corner,
 Up to 18 lines of variable display data,
 Any of the following conditions that apply:

SPIND %	If spindle speed override is active.
FEED %	If feed rate override is active.
RAPID %	If rapid override is active.
SINGBK	If SINGLE BLOCK is turned on.
DRYRUN	If DRY RUN is selected.
OPTSTP	If OPTIONAL STOP is turned on.
BLKDEL	If BLOCK DELETE is turned on.
FEED HOLD	If a FEED HOLD is active.
DOOR HOLD	If an open door has stopped program.

ALARM	Blinking in lower right corner when alarm occurs.
BUF	When next block is ready in continuous path.
FEED	When a feed motion is in progress.
XYZA-MIR	When these axes are set to mirror image.
TOOL UNCLP	Reverse video when the tool is unclamped.
RUNNING	When a program is running.
DWELL	When a G04 is being performed.
SINGBK STOP	When a program is stopped in SINGLE BLOCK.

The following error messages are received when the wrong button is pressed.

FUNCTION LOCKED	Function attempted is locked from settings.
SERVO IS ON!	Parameter change was made with servo on. This is dangerous!
WRONG MODE	Function requested is available only in another mode.
WAIT OR RESET	Cannot perform requested function until program finishes or you reset.
DISABLED AXIS	Cannot jog a disabled axis.
PLEASE WAIT	Wait until spindle is stopped.
SENDING...	RS-232 output is in process.
WAITING...	Waiting for RS-232 input.
LOADING...	Got some RS-232 input and loading it.
RS-232 ABORT	RS-232 was aborted by operator action.
RS-232 ERROR	RS-232 error (shown in alarms).
RS-232 DONE	RS-232 operation is complete.
BAD NAME	Name entered is not Onnnn .
ALARM ON	Cannot start an operation until alarms are reset.
PROGRAM END	Cannot remove last EOB in program.
INVALID NUMBER	Number entered is invalid.
DIVIDE BY ZERO	An attempt to divide by zero in calculator mode.
SPINDLE IN USE	Spindle is controlled by program - manual controls not available.
BLOCK TOO LONG	Block being edited would be too long.
MEMORY LOCKED	Memory lock is set in settings.
MEMORY FULL	Memory space is full.
GUARDED CODE	Cannot remove Onnnn at start of program.
BAD CODE	Code entered is not understood.
SERVO IS OFF	When servos are off , you cannot start a program.
NOT IN DRYRUN	The function requested applies to DRY RUN but not in that mode.
NO INPUT	Cannot alter unless enter something first.
ONE PROG ONLY	Program name being selected cannot be ALL.
DIR FULL	Maximum number of programs exceeded.
PROG EXISTS	Cannot receive RS-232 an existing program.
MACRO LOCKED	Macros 9000 to 9099 are locked by setting.
PROG NOT FOUND	Requested program not in memory.
PROG READY	has been received and is ready to run.
END FOUND	End of program has been received.
NO PROG YET	Cannot Cycle Start until receives a program.
NO ZERO X	Cannot run machine until search for zero is complete on X-axis.
NO ZERO Y	Cannot run machine until search for zero is complete on Y-axis.
NO ZERO Z	Cannot run machine until search for zero is complete on Z-axis.
NO ZERO A	Cannot run machine until search for zero is complete on A-axis.
NOT AVAILABLE	Function requested is not available at that time.
DISPLAYS OFF	Indicates that M76 was used to turn off displays.
AUX AXIS BUSY	One or more auxiliary axes are busy in an operation.
LOW AIR PR	Air pressure is low.

And the following responses, only when in graphics mode:

M30 FOUND	End of program found and execution stopped.
LINEAR	A linear motion is being performed.
RAPID	A rapid motion is being performed.
CIRCULAR	A circular motion is being performed.

In addition, the CRT display can show one of the following eight types of data in the 18 lines of variable display:

PROGRAM DISPLAYS:

The PROGRAM DISPLAY is used to show your program while in either MEM, EDIT, or MDI modes.

POSITION DISPLAY:

The POSITION DISPLAY is used to select the X, Y, Z, or A axes positions in any of several coordinate systems. The PAGE UP and PAGE DOWN keys select between these.

OFFSETS DISPLAY:

The OFFSETS DISPLAY is used to enter and display tool length offsets, tool radius offsets, and work offsets. The PAGE UP and PAGE DOWN keys select between these.

CURRENT COMMANDS DISPLAY:

The CURRENT COMMANDS DISPLAY is used to display the Program Command Check, the Current Commands, Running Timers, Tool Life Timers, and Tool Load Monitor. The PAGE UP and PAGE DOWN keys select between these.

ALARMS / MESSAGES DISPLAYS:

The ALARMS/MESSAGES DISPLAY is used to display alarms and to enter and display user messages. The second push of the ALARM button will select messages display. The CURSOR up and down buttons will display additional alarms if there is more than will fit on one page.

PARAMETERS / DIAGNOSTICS DISPLAYS:

The PARAMETERS DISPLAY shows all of the machine dependent control parameters and the Diagnostic data. The second push of the PARAM DGNOS button will select the diagnostic display. The PAGE UP and PAGE DOWN buttons will select additional data for display.

SETTINGS / GRAPHICS DISPLAYS:

The SETTINGS DISPLAY is used to display and change user controlled parameters. The second push of the SETNG / GRAPH button will select the Graphics display. The cursor and PAGE UP and PAGE DOWN buttons will select additional settings.

HELP / CALCULATOR DISPLAYS:

The HELP DISPLAY shows a mini-manual on the CRT along with a directory of available help information. Each alphabet button will select a different topic within the HELP display. The second push of the HELP button will select the Calculator display. The PAGE UP and PAGE DOWN buttons will select different calculator functions.

4.1 PROGRAM DISPLAYS

The PROGRAM DISPLAY is used to show a program being edited in EDIT mode or a program being run in MEM. In MEM mode, there is also a PROGRAM REVIEW display available.

The PROGRAM DISPLAY uses 18 lines of the text display area of the CRT to show the command blocks of a CNC program. The display is 40 positions wide and blocks that are longer than 40 positions are continued on the next line of the display.

The PROGRAM REVIEW function is available whenever a program is being run. This allows you to review the program that is running. This is selected by pressing F4 while in MEM mode and PROGRAM DISPLAY. The screen is changed to an 80 column display with the normal MEM display on the left and PROGRAM REVIEW on the right. The CURSOR and PAGE UP and PAGE DOWN keys can be used to change the display on the right to a different part of the program. The display on the left will show the progress of the running program. To exit PROGRAM REVIEW, select any other display.

While you are running a program, the BACKGROUND EDIT function is available as a standard feature. BACKGROUND EDIT allows you to edit any named program in memory while any program is being run in memory. BACKGROUND EDIT is selected from MEM mode in PROG display by entering 0nnnn with the program number and pressing F4. The display will change to the selected program while still running the first program. BACKGROUND EDIT is enabled by parameters if it is available in your machine.

4.2 POSITION DISPLAYS

The following are the five position displays in this control:

HOME PAGE

This display shows the four displays simultaneously in small characters. The other displays are shown in large characters. The PAGE UP and PAGE DOWN keys will change displays. The last display selected will be shown in CURNT COMDS and SETNG / GRAPH displays when they are selected.

OPERATOR DISPLAY

This display is for the operator/setup person to use as desired, and is not used by the control for any positioning functions. In JOG mode, and with this display selected, the ORIGIN button can be used to set the zero position. This display will then show position relative to the selected zero position.

WORK DISPLAY

This display shows how far the tool is away from the X, Y and Z zero of the programmed part. On power up, it will display the value in work offset G54 automatically. It can only be changed by G54 through G59, G110 through G129, or by a G92 command. The machine uses this coordinate system to run the part.

MACHINE DISPLAY

This display is the machine coordinate system that is automatically set on power up and the first ZERO RET. It cannot be changed by the operator or any work coordinate systems, and will always show the distance from machine zero. It can be used by a non-modal G53 command.

DISTANCE TO GO

This display is an incremental display that shows the travel distance remaining before the axes stop. When in the ZERO RET mode, this display shows a diagnostic value. When in JOG mode, this display shows the total distance jogged. In rigid tapping, this number decreases to zero at the bottom of the hole and then increases again as the reverse stroke occurs.

4.3 TOOL OFFSETS DISPLAY

Tool length and part zero offsets are displayed and can be entered in the Tool Offsets display. There are 100 H and D codes possible. The same offset number for both Z-axis offset (H1) and tool diameter compensation (D1) can be used because the offset contains separate values for each. Work offsets can also be specified for the fifth axis B address, in the work offset display. If the fifth axis is enabled (Setting 78), then additional data fields are made available for the fifth axis.

PAGE DOWN in the OFSET display will go through all 100 possible tool offsets and then change to the work zero offsets. PAGE UP will go directly to work zero offsets. Work zero offsets may be entered and displayed

from this page or using the PART ZERO SET key. Refer to the "Setup Procedures" section for more information on using this key.

When entering offsets, pressing F1 will set the value to your entry. Pressing WRITE will cause the new value to be added to the old value. This allows small adjustments to be made to the offsets. Note that entering a negative entry and pressing WRITE will decrease the value of the offset.

There is a geometry and wear value with each offset and these are added together by the control during operation. The initial value is entered into the geometry column by the setup person. During operation, the operator makes minor wear changes in the wear column. This method allows the operator to see actual tool wear by limiting it to the wear column. The geometry values can also be entered automatically by using the TOOL OFFSET MESUR key during setup procedures. Note that this automatic offset measurement works with G43 only. Refer to the "Setup Procedures" section for more information on using the TOOL OFFSET MESUR key.

The function of the offset display page has been modified slightly to accommodate multiple axes. Only the axes that are enabled are displayed on the work offset display. For example, if the fourth axis is enabled (Setting 30), and the fifth axis is not, then the display will show X,Y,Z,A. In this case, all axes can be displayed on the entire display. If only the fifth axis is enabled, then the display will show X,Y,Z,B. Since the fourth axis is disabled, there is no need to display it.

If both the **A** and **B** axes are enabled, then the last column of the offsets display serves a dual purpose. Either **A** or **B** can be accessed in the last column. By using the **left** or **right** arrow keys, the screen cursor can be moved into the axis field that is to be set. When the rightmost field is highlighted by the cursor and the top of the column indicates **A**, then the values in this column represent **A** axis offsets. If you press the **right** arrow key once, the cursor stays in the same place, but the column and its values will change to **B** axis parameters. You can view and modify work offsets in this manner using the **left** and **right** arrow keys when both axes are enabled.

Setting 15, "H & T Code Agreement", may be used to force the spindle tool number and offset number to be equal; otherwise an alarm will occur. This is the preferred setting as it avoids crashes.

Tool diameter may be entered as either radius or diameter. Setting 40 is used to select between these. The value used for cutter compensation is the sum of the geometry and wear values.

Cutter compensation is controlled by G41 and G42 and the selected tool diameter (D value). Positive values for cutter compensation work normally. Negative values for cutter compensation cause the opposite side cutter compensation to be used. This means that G41 with a negative number will be the same as G42 with the same but positive number.

Offsets may be sent and received with the RS-232 port. Refer to the "Data Input / Output" section for a description of how to do this.

When the COOLANT SPGT of Parameter 57 is enabled, the CLNT POS column of the offset display is accessible. The spigot can be positioned when an M08 is encountered in a program. The current **H** code is used by the M08 to determine where to position the coolant spigot. If offset #5 (H05) has **10** entered under the CLNT POS column, the spigot will be moved to position 10. When the cursor is positioned on the CLNT POS column, the current spigot position will be shown at the bottom of the display. If the spigot has not been zeroed, the position will be blank.

The permissible values that can be entered into this column are controlled by Parameter 206, SPIGOT POSITIONS. Entering a value of zero (0) indicates that the spigot will not be moved when an M08 code sequence is encountered.

The following code sequence demonstrates how the spigot can be commanded:

```

00001          (Sample coolant positioning)
G90 G54
T1 H1 M06
M08          (Moves spigot to H1 position)
.
.
T2 H2M06
M08          (Moves spigot to H2 position)
.
.
H42M08        (Position to H42 position)
              (Note that the length and radius offsets should be the same as Tool 2)
.
.
M30

```

4.4 CURRENT COMMANDS DISPLAY

The following are the six current command displays in this control:

**Program Command Check (Home Page),
Current Display Command,
Operation Timers,
Macro Variables,
Tool Life Timers,
and Tool Load Monitor.**

The PAGE UP and PAGE DOWN keys are used to select among these displays.

PROGRAM COMMAND CHECK DISPLAY

This display, which is the Home Page for the Current Commands Display, shows a current overview of the important commands. It shows the programmed spindle speed (**Snnnnn**), the spindle speed command to the spindle drive (**CMDnnnnn**), and if Parameter 278 bit ACT DISPLAY is set to 1, the actual encoder spindle speed (**ACTnnnnn**). In addition, this display shows the CW, CCW, or stopped command being sent to the spindle and the current gear position.

This display also shows the position of the axes. The coordinates displayed (operator, work, machine, or distance to go) are selected using the cursor **up** and **down** keys.

If the spigot is enabled, the current spigot position will be displayed.

CURRENT DISPLAY COMMAND

This display shows all of the alphabetical address codes (i.e. G, M, S) and their current value. These values may not be changed in this display. The default value is shown for the address codes that are not being used in the current program.

MACRO VARIABLES DISPLAY

This display shows a list of the macro variables and their present values. As the control interprets a program, the variable changes are displayed on this page and the results can be viewed. The variables may be modified in this display. For more information on this display, refer to the "Macros" section of this manual.

OPERATION TIMERS DISPLAY

This display shows the current power-on time, cycle start time, and the feed time. These times may be reset to zero by using the cursor **up** and **down** keys to highlight the desired title and pressing the ORIGIN button.

Listed below these times are two M30 counters that are used for counting completed parts. They may be set to zero independently to provide for the number of parts per shift and total parts. Both counters are increased when an M30 is operated.

TOOL LIFE DISPLAY

This display shows the time the tool is in feed, the time the tool is selected, and the number of times the tool has been used. This information can be used to assist in predicting tool life. The values in this display can be reset to zero using the Cursor and ORIGIN buttons. This is done by putting the cursor on the title line, and pressing ORIGIN to zero all of the data in that column.

This display may also be used to generate an alarm when a tool has been used a specific number of times. The last column is labeled ALARM, and if the number for a tool is not zero, an alarm will be generated when that count is reached. Alarm 362 is generated when the count is reached, and may be cleared with RESET.

TOOL LOAD MONITOR AND DISPLAY

With the tool load display, the operator can enter the maximum load that is expected for each tool, and when this load is exceeded in a feed, a certain action will be taken. This display provides for the entry of this alarm point and also displays the largest load that tool has seen in any previous feed.

The tool load monitor function operates whenever the machine is in a feed operation (G01, G02, or G03). The values entered into the tool load display are checked against the actual spindle motor load. If the limit is exceeded, the tool overload action specified in Setting 84 (alarm, feedhold, beep, or pause) will be taken. If "alarm" is selected and the limit is exceeded, Alarm 174, "Tool Load Exceeded", will be generated. This alarm will stop the axis motors and the spindle motor, turn off the coolant, and disable the servos.

4.5 ALARMS / MESSAGES DISPLAY

The **ALARMS DISPLAY** can be selected at any time by pressing the ALARM MESSAGES button. When there are no alarms, the display will show NO ALARM. If there are any alarms, they will be listed with the most recent alarm at the bottom of the list. The CURSOR and PAGE UP and PAGE DOWN buttons can be used to move through a large number of alarms. The CURSOR **right** and **left** buttons can be used to turn on and off the ALARM history display.

The **MESSAGE DISPLAY** can be selected at any time by pressing the ALARM MESSAGES button a second time. This is an operator message display and has no other effect on operation of the control. Any message can be typed into the message display and called up later.

You may leave an electronic note to yourself or anyone else by using this feature. The note may be for the operator to change tools after running a number of parts or it may be a diary for machine maintenance

intervals that are performed. Data is automatically stored and maintained even in a power off state. The message display page will be come up during power up if there are no alarms present.

To enter messages, press the ALARM MESSAGES button twice. You may now enter data by simply typing directly onto the screen. The cancel and space keys can be used to remove existing messages. The DELETE button can be used to remove an entire line.

4.6 PARAMETER / DIAGNOSTIC DISPLAY

The **PARAMETER DISPLAY** can be selected at any time by pressing the PARAM DGNOS button. Changes to parameters can be made when in any mode except when running a program. The **CURSOR up** and **down** buttons move to different parameters and the **PAGE UP** and **PAGE DOWN** buttons move through groups of parameters. Parameters 1, 15, 29, 43, and 57 are displayed as a single page of discrete flags. Selecting among the flags is done with the **CURSOR left** and **right** buttons. It is recommended that parameters not be changed with the servos on. Changing parameters with the servos on causes a warning message and beep on the front panel. The **EMERGENCY STOP** button can be used to turn off the servos.

For machines with the fifth axis option, the parameter display organization has been modified to accommodate the extra axis parameters. Parameter numbers have remained the same as in a four-axis control. Additional parameters have been added for the fifth axis.

The parameters have been reorganized, so that logically-associated parameters are grouped together. These logical groupings are placed together into contiguous screens called pages. The most commonly changed parameters have been placed at the beginning of the page list. A list of the parameter pages and the order of succession in the control are given below.

PAGE TITLE	DATA DESCRIPTION
COMMON SWITCH	Non-axis bit switches.
COMMON PAGE1	First page of non-axis parameters.
COMMON PAGE2	Second page of non-axis parameters.
COMMON PAGE3	Third page of non-axis parameters.
MACRO M CALL	Parameters that alias M codes to subroutines.
MACRO G CALL	Parameters that alias G codes to macros.
X BIT SWITCH	Bit switches for the X axis.
X PARAMETERA	First page of X axis parameters.
X PARAMETERB	Second page of X axis parameters.
Y BIT SWITCH	Bit switches for the Y axis.
Y PARAMETERA	First page of Y axis parameters.
Y PARAMETERB	Second page of Y axis parameters.
Z BIT SWITCH	Bit switches for the Z axis.
Z PARAMETERA	First page of Z axis parameters.
Z PARAMETERB	Second page of Z axis parameters.
A BIT SWITCH	Bit switches for the A axis.
A PARAMETERA	First page of A axis parameters.
B BIT SWITCH	Bit switches for the B axis.
B PARAMETERA	First page of B axis parameters.
B PARAMETERB	Second page of B axis parameters.
X SCREW COMP	X axis screw compensation value.
Y SCREW COMP	Y axis screw compensation value.
Z SCREW COMP	Z axis screw compensation value.

The HOME key displays the first parameter page "COMMON SWTCH". Pressing the PAGE DOWN key will display the next page of parameters in the above list. The END key displays the last parameter page "B PARAMETERB". Pressing the PAGE UP key will display the preceding page of parameters in the above list. All other features on the parameters display have remained the same. So, if you are unfamiliar with the new format of the parameters, you can still search by parameter number. Enter the number of the parameter you want to see or view and press the **up** or **down** arrow key. The page that the parameter is on will be displayed and the parameter being searched for will be highlighted. Refer to the "Parameters" section for more information.

The **DIAGNOSTIC DATA DISPLAY** can be selected at any time by pressing the PARAM DGNOS button a second time. There are two pages of diagnostic data and the PAGE UP and PAGE DOWN buttons are used to select between them. After this, the current run time and the number of tool changes is displayed.

A five-axis control has additional diagnostic data to be aware of:

The first page of diagnostic data has two discrete outputs that control the rotary axes brakes; they are labeled "A BRAK" and "B BRAK". When motion is commanded to either the **A** or **B** axis, the brake for that axis must first be released. To release a brake on the HAAS control, a relay is activated. These two outputs represent the activation of the brake relays. If the **A** or **B** axis brakes are disengaged, then these outputs will read high (logical 1). Normally these bits will read low (logical 0).

The second page of diagnostic data show the status of inputs from the motor interface board. Additional inputs for the expanded motor interface board are listed under "INPUTS4". These bits are monitored by the control to determine if the interface for the **B** axis is working correctly. Refer to the "Technical Reference" section of this manual for a description of the diagnostic page inputs.

4.7 SETTING / GRAPHIC DISPLAY FUNCTION

The **SETTINGS DISPLAY** can be selected at any time by pressing the SETNG GRAPH button. When the settings are displayed, changes can be made to any of the settings. There are some special functions in the settings; refer to the "Settings" section for a more detailed description.

The **GRAPHICS FUNCTION** is a visual dry run of your part program without the need to move the axes and risk tool damage from programming errors. This function is far more powerful than using the DRY RUN mode because all of your work offsets, tool offsets, and travel limits can be checked before any attempt is made to move the machine. The risk of a crash during setup is greatly reduced.

To run a program in Graphics, you must be in either MEM or MDI mode.

After loading the program into memory, select MEM (or MDI) and press the SETNG/GRAPH key twice to select the Graphics Simulation mode. This function operates the same as if running a program on the machine except no physical machine action occurs.

The graphics screen is composed of the following areas:

DISPLAY TITLE AREA

The title area is on the top left line of the screen and indicates the display (GRAPHICS), the mode you are in (MEM or MDI), the program number, and the current program line being executed. It is the same as the top line of all displays.

KEY HELP AREA

The right side of the top line is the function key help area. Function keys that are currently available are displayed here with a brief description of their usage.

LOCATOR WINDOW

The lower right part of the screen has two functions: it can display the whole table area and indicate where the tool is currently located during simulation, or it can be used to display four lines of the program that is being executed. The F4 key can be used to toggle between these two modes.

TOOL PATH WINDOW

In the center of the display is a large window that represents a look down perspective of the X-Y axis. It displays tool paths during a graphics simulation of a CNC program. Rapid moves are displayed as coarse dotted lines, while feed motion is displayed as fine continuous lines. The rapid path can be disabled by Setting 4. The places where a drill can or canned cycle can be executed are marked with an X. The drill mark can be disabled by Setting 5.

The tool path window can be scaled. After running a program, you can scale any portion of the tool path by pressing F2 and then using the PAGE DOWN key and the ARROW keys to select the portion of the tool path that you want to see enlarged. During this process, a reticle will appear within the TOOL PATH window and the Locator window indicating what the TOOL PATH window will represent when the zoom process is complete. The KEY HELP AREA will flash, indicating that the user is rescaling the view. The locator window always portrays the entire table with an outline of where the TOOL PATH window is zoomed to. The PAGE UP key unzooms the reticle one step. After sizing or moving the reticle, pressing the WRITE key will complete the zoom process and re-scale the TOOL PATH window. Pressing F2 and then the HOME key will expand the TOOL PATH window to cover the entire table. After the TOOL PATH window is re-scaled, the TOOL PATH window is cleared and you must rerun the program, or a portion of it, to see the tool path. The tool path is not retained in the control.

The scale and position of the TOOL PATH window is saved in Settings 65 through 68. Any scaling performed on the TOOL PATH window is retained. You can leave graphics to edit your program and when you return, your previous scaling is still in effect.

Z AXIS WINDOW

A long window on the rightmost part of the screen shows the location of the Z-axis and indicates spindle movement. A horizontal line in the top part of this window represents the tool change position.

CONTROL STATUS

The lower left portion of the screen displays control status. It is the same as the last four lines of all other displays.

POSITION WINDOW

The location of all enabled axes can be viewed in this window. By default it is OFF. This window can be opened by pressing the F3 key. Additional presses of the F3 key will display the various position formats that the control keeps track of. This window also displays the current scale of the tool path window and the current simulated tool number. The value represented by the vertical dimension of the Tool Path window is labeled Y-SIZE. At power-on, this will be the full Y-axis table travel. When you zoom into a table area, this value will become smaller, indicating that you are viewing a smaller portion of the table. In addition to the above, a perspective 3D graphics view is also selected by Setting 3.

To exit the Graphic mode, select any other display or mode. When you exit Graphics, the graphics image is lost and must be built again by running the program.

4.8 HELP / CALCULATOR FUNCTION

The **HELP FUNCTION** is selected by pressing the HELP display button. This will bring a mini-manual up on the CRT. There are 26 topic areas selectable with the A-Z keys. This also includes a directory of the topics. The areas covered are:

- A MACHINE INSTALLATION
- B START/RUN//BGEDIT/POWER DOWN
- C G/M/S/T COMMAND CODES
- D RETURN TO THIS DIRECTORY
- E EDITING PROGRAMS
- F SETTING PAGE
- G SPECIAL G CODES
- H TROUBLE SHOOTING
- I MDI / MANUAL DATA INPUT
- J JOGGING / HANDLE FUNCTION
- K CRT DISPLAY / KEYBOARD
- L ALARMS / MESSAGES
- M MAINTENANCE REQUIREMENTS
- N SET UP PROCEDURES
- O OVERRIDES: FEED/SPIN/COOLANT
- P PARAMETERS / DIAGNOSTICS
- Q POSITION DISPLAYS
- R RECV / SEND PROGRAMS
- S SAMPLE PROGRAM
- T TOOL OFS/TOOL LIFE/LOAD
- U GRAPHIC FUNCTION
- V TOOL CHANGER
- W WORK COORDINATES
- X CREATING PROGRAMS
- Y SPECIAL FUNCTIONS
- Z ZERO RETURN

When the HELP display is selected, the alphanumeric keys are used to select one of the above topics; so they are not displayed on the data entry line of the CRT.

The **CALCULATOR FUNCTION** is selected by pressing the HELP key a second time. There are three calculator pages: Trig Help, Circular Interpolation Help, and Milling/Tapping Help. All of these have a simple calculator and an equation solver. Trig, Circular, and Milling Help are selected using the PAGE UP and PAGE DOWN keys. The **Fn** keys also allow data to be moved from other displays to/from the calculator.

All of the Calculator Help functions have a calculator for simple add, subtract, multiply, and divide operations. When one of these functions (Trig, Circular, or Milling) is selected, a calculator window appears in the upper left corner of the screen, and below it the possible operations (**LOAD** + - * and /). **LOAD** is initially highlighted, and the other options can be selected with the left and right cursor arrows. Numbers are entered by typing them in at the cursor in the lower left corner of the screen and pressing the WRITE key. When a number is entered and **LOAD** is selected, that number will be entered into the calculator window directly. When a number is entered when one of the other functions (+ - * /) is selected, that calculation will be performed with the number just entered and any number that was already in the calculator window.

TRIGONOMETRY HELP FUNCTION

The Trig Help page will help you solve a triangular problem. You enter the lengths and the angles of a triangle and when enough data has been entered, the control will solve for the triangle and display the rest of the

values. Use the **CURSOR up** and **down** buttons to select the value to be entered with **WRITE**. For inputs that have more than one solution, entering the last data value a second time will cause the next possible solution to be displayed. The **F3** and **F4** buttons perform special data import and export functions:

F3 In **EDIT** and **MDI** modes the **F3** key will copy the highlighted circular help line into the data entry line at the bottom of the screen. This is useful when you want to use the solution developed for a circular motion. Push **INSERT** to add that circular motion command line to your program.

In the Calculator Help function, this button copies the value in the calculator window to the highlighted data entry for Trig or Circular Help.

F4 In the Calculator Help function, this button uses the highlighted Trig data value to load, add, subtract, multiply, or divide with the calculator.

CIRCULAR INTERPOLATION HELP

The Circular Help page will help you solve a circle problem. You enter the center, radius, angles, start and end points and when enough data has been entered, the control will solve for the circular motion and display the rest of the values. In addition, it will list the four ways that such a move could be programmed with a **G02** or **G03**. Those four lines can be selected using the **CURSOR up** or **down** buttons, and the **F3** button will import the highlighted line into a program you are editing. Use the **CURSOR up** and **down** buttons to select the value to be entered with **WRITE**.

For inputs that have more than one solution, entering the last data value a second time will cause the next possible solution to be displayed. The **CW/CCW** entry is changed to the other value by pressing **WRITE**. The **F3** and **F4** buttons perform special data import and export functions:

F3 The **F3** key will copy the highlighted circular help line into the data entry line at the bottom of the screen. This is useful in **EDIT** or **MDI** modes where you can then press **INSERT** to add that circular motion command line to your program.

F4 In the Calculator Help function, this button uses the highlighted circular data value to load, add, subtract, multiply, or divide with the calculator.

MILLING/TAPPING HELP

The Milling/Tapping Help page will help you solve three equations relating to milling and tapping. They are:

- 1) $SFM = (CUTTER\ DIAMETER\ IN.) * RPM * 3.14159 / 12$
- 2) $(CHIP\ LOAD\ IN.) = (FEED\ IN./MIN.) / RPM / \#FLUTES$
- 3) $(FEED\ IN./MIN.) = RPM / (THREAD\ PITCH)$

With all three equations, you may enter all but one of the values and the control will compute the remaining value and display it. Note that the **RPM** value for equations 1 and 2 are the same entry.

When Metric units are selected, the units displayed change to millimeters, mm per minute, threads per mm, and meters, respectively.

The **F3** and **F4** buttons perform special data import and export functions:

F3 The **F3** key will copy the highlighted circular help line into the data entry line at the bottom of the screen.

This is useful in **EDIT** or **MDI** modes where you can then push **INSERT** to add that circular motion command line to your program.

F4 In the **Calculator Help** function, this button uses the highlighted milling or tapping data value to load, add, subtract, multiply, or divide with the calculator.

5. TRAVEL LIMITS

Travel limits in this machine are defined by a limit switch in the positive direction and by stroke limits set by parameter in the negative direction. Prior to establishing the home positions with the **AUTO POWER UP** or **AUTO ALL AXES** buttons, there are no travel limits and the user must be careful not to run the table into the stops and damage the screws or way covers.

In machines built after October 1990, there are hard limits built into the axes so that you cannot damage the way covers.

Prior to establishing the home positions (**AUTO POWER UP** or **AUTO ALL AXES**), jogging is normally not allowed. Setting 53 can be turned on to allow jogging prior to zero return but this defeats the travel limits and you may damage the machine running the axes into the stops.

Note that all motion is in a negative direction from machine zero except for the **Z**-axis that can move about 4.5 inches up from machine zero. There are no travel limits for the fourth, **A**, axis. Travel limits for any auxiliary axes are set into those single axis controls.

When jogging, an attempt to move past the travel limits will not cause an alarm but the axis will stop at the limit. The **JOG** handle inputs may be ignored in this case.

When running a program, an attempt to move outside of the travel limits will cause an alarm prior to starting the motion and the program will stop. An exception is a circular motion which starts and ends inside of the travel limits but moves outside of the limits during the motion. This will cause an alarm to occur part way through the motion.

Travel limits apply even when running a program in **Graphics** mode. An alarm is generated and the program will stop.

6. FEED/RAPID/SPINDLE OVERRIDES

The feed rate can be varied from 10% to 200% of the programmed value while in operation. This is done with the feed rate +10%, -10% and 100% buttons. The **FEED RATE** override is ineffective during **G74** and **G84** tapping cycles. **FEED RATE** override does not change the speed of any auxiliary axes.

During manual jogging, the feed rate override will adjust the rates selected from the keypad. This allows for fine control of the jog speed. The spindle speed can also be varied using the **SPINDLE** override as above and is also ineffective for **G74** and **G84**. In the **SINGLE BLOCK** mode, the spindle may be stopped. It will automatically start up upon continuing the program.

Rapid moves (G00) may be limited to 5, 25, or 50 % of maximum. If the 100% rapid is too fast, it may be set to 50% of maximum by Setting 10.

In the Setting page, it is possible to disable the override keys so that the operator cannot select them. This is Setting 19, 20 and 21.

The FEED HOLD button acts as an override button as it sets the rapid and feed rates to zero when it is pressed. The CYCLE START must be pressed to proceed after a FEED HOLD. When in a FEED HOLD, the bottom left of the screen will indicate this. The door switch on the enclosure also has a similar result but it will display "Door Hold" when the door is opened. When the door is closed, machine operation will continue normally. Door hold can be prevented with Setting 51. Door Hold and FEED HOLD do not stop any auxiliary axes.

When Parameter 57 flag DOOR STOP SP is set to 1, the door switch will stop the servos and the spindle. In addition, the override setting does not work, and manual tool changes cannot be done with the door open.

There is also an override function for the coolant supply. This is done from the Setting 32. The "NORMAL" setting checks the low coolant alarm and turns the pump on and off with M codes. The "OFF" setting ignores the coolant alarm but will alarm if an attempt is made to turn the coolant on. The "IGNORE" setting is used to ignore all coolant commands and the low coolant alarm.

At any time a program is running, the operator may override the coolant setting by pressing the MDI Coolant button. The pump will remain either on or off until the next M command or operator action.

Overrides can now be reset through to defaults upon processing M30 and/or RESET. This feature is selected by **Setting #83**.

7. WHAT TO DO WHEN ALARMS OCCUR

7.1 ALARMS

Any time an alarm is present, the lower right hand corner will have a blinking "ALARM". Push the ALARM display key to view the current alarm. All alarms are displayed with a reference number and a complete description. If the RESET key is pressed, one alarm will be removed from the list of alarms. If there are more than 18 alarms, only the last 18 are displayed and the RESET must be used to see the rest. The presence of any alarm will prevent the operator from starting a program.

Note that tool changer alarms can be easily corrected by first correcting any mechanical problem, pressing RESET until the alarms are clear, selecting ZERO RET mode, and selecting AUTO ALL AXES. Some messages are displayed while editing to tell the operator what is wrong but these are not alarms. See the editing topic for those errors.

7.2 ALARM LIST

The following alarm list shows the alarm numbers, the text displayed along with the alarm, and a detailed description of the alarm, what can cause it, when it can happen, and how to correct it.

Alarm number and text:	Possible causes:
102 Servos Off	Indicates that the servo motors are off, the tool changer is disabled, the coolant pump is off, and the spindle motor is stopped. Caused by EMERGENCY STOP, motor faults, tool changer problems, or power fail.
103 X Servo Error Too Large	Too much load or speed on X-axis motor. The difference between the motor position and the commanded position has exceeded a parameter. The motor may also be stalled, disconnected, or the driver failed. The servos will be turned off and a RESET must be done to restart. This alarm can be caused by problems with the driver, motor, or the slide being run into the mechanical stops.
104 Y Servo Error Too Large	same as 103.
105 Z Servo Error Too Large	same as 103.
106 A Servo Error Too Large	same as 103.
107 Emergency Off	EMERGENCY STOP button was pressed. Servos are also turned off. After the E-STOP is released, the RESET button must be pressed at least twice to correct this; once to clear the E-STOP alarm and once to clear the Servo Off alarm.
108 X Servo Overload	Excessive load on X-axis motor. This can occur if the load on the motor over a period of several seconds or even minutes is large enough to exceed the continuous rating of the motor. The servos will be turned off when this occurs. This can be caused by running into the mechanical stops but not much past them. It can also be caused by anything that causes a very high load on the motors.
109 Y Servo Overload	same as 108.
110 Z Servo Overload	same as 108.
111 A Servo Overload	same as 108.

ALARMS

- | | | |
|-----|---------------------------|---|
| 112 | No Interrupt | Electronics fault. Call your dealer. |
| 113 | Shuttle In Fault | Tool changer not completely to right. During a tool changer operation the tool in/out shuttle failed to get to the in position. Parameters 62 and 63 can adjust the time-out times. This alarm can be caused by anything that jams the motion of the slide or by the presence of a tool in the pocket facing the spindle. A loss of power to the tool changer can also cause this, so check CB5 and relays 1-8, 2-1, and 2-2. |
| 114 | Shuttle Out Fault | Tool changer not completely to left. During a tool change operation the tool in/out shuttle failed to get to the out position. Parameters 62 and 63 can adjust the time-out times. This alarm can be caused by anything that jams the motion of the slide or by the presence of a tool in the pocket facing the spindle. A loss of power to the tool changer can also cause this, so check CB5 and relays 1-8, 2-1, and 2-2. |
| 115 | Turret Rotate Fault | Tool carousel motor not in position. During a tool changer operation the tool turret failed to start moving or failed to stop at the right position. Parameters 60 and 61 can adjust the time-out times. This alarm can be caused by anything that jams the rotation of the turret. A loss of power to the tool changer can also cause this, so check CB5 and relays 1-8, 2-3, and 2-4. |
| 116 | Spindle Orientation Fault | Spindle did not orient correctly. During a spindle orientation function, the spindle is rotated until the lock pin drops in; but the lock pin never dropped. Parameters 66, 70, 73, and 74 can adjust the time-out times. This can be caused by a trip of circuit breaker CB4, a lack of air pressure, or too much friction with the orientation pin. |
| 117 | Spindle High Gear Fault | For Vertical Mills only. Gearbox did not shift into high gear. During a change to high gear, the spindle is rotated slowly while air pressure is used to move the gears but the high gear sensor was not detected in time. Parameters 67, 70 and 75 can adjust the time-out times. Check the air pressure, the solenoids circuit breaker CB4, and the spindle drive. |
| 118 | Spindle Low Gear Fault | For Vertical Mills only. Gearbox did not shift into low gear. During a change to low gear, the spindle is rotated slowly while air pressure is used to move the gears but the high gear sensor was not detected in time. Parameters 67, 70 and 75 can adjust the time-out times. Check the air pressure, the solenoids circuit breaker CB4, and the spindle drive. |
| 119 | Over Voltage | Incoming line voltage is above maximum (about 255 volts when wired for 240 or 235 when wired for 208). The servos will be turned off and the spindle, tool changer, and coolant pump will stop. If this condition remains for 4.5 minutes, an automatic shutdown will begin. |
| 120 | Low Air Pressure | Air pressure dropped below 80 PSI for a period defined by Parameter 76. The "Low Air Pr" alarm will appear on the screen as soon as the pressure gets low, and this alarm appears after [Parameter 76] has elapsed. Check your incoming air pressure for at least 100 PSI and ensure that the regulator is set at 85 PSI. |

- | | | |
|-----|----------------------------|--|
| 121 | Low Lub or
Low Pressure | Way lube is low or empty or there is no lube pressure or too high a pressure. Check tank at rear of mill and below control cabinet. Also check connector P5 on the side of the control cabinet. Check that the lube lines are not blocked. |
| 122 | Control Overheat | The control internal temperature is above 150 degrees F. This can be caused by almost anything in the control overheating. But is usually caused by overheat of the two regen resistors for servos and spindle drive. This alarm will also turn off the servos, spindle drive, coolant pump, and tool changer. One common cause of this overheat condition is an input line voltage too high. If this condition remains for 4.5 minutes, an automatic shutdown will begin. |
| 123 | Spindle Drive Fault | Overheat or failure of spindle drive or motor. The exact cause is indicated in the LED window of the spindle drive inside the control cabinet. This can be caused by a stalled motor, shorted motor, overvoltage, undervoltage, overcurrent, overheat of motor, or drive failure. |
| 124 | Low Battery | Memory batteries need replacing within 30 days. This alarm is only generated at power on and indicates that the 3.3 volt Lithium battery is below 2.5 volts. If this is not corrected within about 30 days, you may lose your stored programs, parameters, offsets, and settings. |
| 125 | Shuttle fault | Tool shuttle not initialized at power on, CYCLE START or spindle motion command. This means that the tool shuttle was not fully retracted to the Out position. |
| 126 | Gear Fault | Gearshifter is out of position when a command is given to rotate the spindle. This means that the two speed gear box is not in either high or low gear but is somewhere in between. Check the air pressure, the solenoids circuit breaker CB4, and the spindle drive. |
| 127 | No Turret Mark | Tool carousel motor not in position. The turret motor only stops in one position indicated by a switch and cam on the Geneva mechanism. This alarm is only generated at power-on. The AUTO ALL AXES button will correct this but be sure that the pocket facing the spindle afterwards does not contain a tool. |
| 128 | Tool In Turret | Pocket opposite spindle has tool in it. Future option not yet implemented. |
| 129 | M Fin Fault | M-Fin was active at power on. Check the wiring to your M code interfaces. This test is only performed at power-on. |
| 130 | Tool Unclamped | When orienting the spindle, commanding a speed change; commanding a gear change or commanding Through the Spindle Coolant. During POWER UP the tool release piston is energized. This is a possible fault in the air solenoids, relays on the IO Assembly, the draw bar assembly, or wiring. |
| 131 | Tool Not Clamped | When clamping or powering up the machine, the Tool Release Piston is not HOME. This is a possible fault in the air solenoids, relays on the IO Assembly, the drawbar assembly, or wiring. |
| 132 | Power Down Failure | Machine did not turn off when an automatic power- down was commanded. Check wiring to POWIF card on power supply assembly, relays on the IO assembly, and the main contactor K1. |

ALARMS

- | | | |
|-----|-------------------------------|--|
| 133 | Spindle Locked | Shot pin did not release. This is detected when spindle motion is commanded. Check the solenoid that controls the air to the lock, relay 2-8, the wiring to the sense switch, and the switch. |
| 134 | Tool Clamp Fault | When UNCLAMPING the Tool did not release from spindle when commanded. Check air pressure and solenoid circuit breaker CB4. Can also be caused by misadjustment of drawbar assembly. |
| 135 | X Motor Over Heat | Servo motor overheat. The temperature sensor in the motor indicates over 150 degrees F. This can be caused by an extended overload of the motor such as leaving the slide at the stops for several minutes. |
| 136 | Y Motor Over Heat | same as 135. |
| 137 | Z Motor Over Heat | same as 135. |
| 138 | A Motor Over Heat | same as 135. |
| 139 | X Motor Z Fault | Encoder marker pulse count failure. This alarm usually indicates that the encoder has been damaged and encoder position data is unreliable. This can also be caused by loose connectors at P1-P4. |
| 140 | Y Motor Z Fault | same as 139. |
| 141 | Z Motor Z Fault | same as 139. |
| 142 | A Motor Z Fault | same as 139. |
| 143 | Spindle Not Locked | Shot pin not fully engaged when a tool change operation is being performed. Check air pressure and solenoid circuit breaker CB4. This can also be caused by a fault in the sense switch that detects the position of the lock pin. |
| 144 | Time-out-
Call Your Dealer | Time allocated for use prior to payment exceeded. Call your dealer. |
| 145 | X Limit Switch | Axis hit limit switch or switch disconnected. This is not normally possible as the stored stroke limits will stop the slides before they hit the limit switches. Check the wiring to the limit switches and connector P5 at the side of the main cabinet. Can also be caused by a loose encoder shaft at the back of the motor or coupling of motor to the screw. |
| 146 | Y Limit Switch | same as 145 |
| 147 | Z Limit Switch | same as 145 |
| 148 | A Limit Switch | Normally disabled for rotary axis. |
| 149 | Spindle Turning | Spindle not at zero speed for tool change. A signal from the spindle drive indicating that the spindle drive is stopped is not present while a tool change operation is going on. |
| 150 | Z and Tool
Interlocked | Tool changer not at home and Z is not either at machine home or above. If RESET, E-STOP, or POWER OFF occurs during tool change, Z-axis motion and tool changer motion may not be safe. Check the position of the tool changer and remove the tool if possible. Re-initialize with the AUTO ALL AXES button but be sure that the pocket facing the spindle afterwards does not contain a tool. |

- | | | |
|-----|-----------------------------|---|
| 151 | Low Thru Spindle Coolant | For machines with Through the Spindle Coolant only. This alarm will shut off the spindle, feed, and pump all at once. Check for low coolant tank level, any filter or intake strainer clogging, or for any kinked or clogged coolant lines. |
| 152 | Self Test Fail | Control has detected an electronics fault. All motors and solenoids are shut down. This is most likely caused by a fault of the processor board stack at the top left of the control. Call your dealer. |
| 153 | X-axis Z Ch Missing | Broken wires or encoder contamination. All servos are turned off. This can also be caused by loose connectors at P1-P4. |
| 154 | Y-axis Z Ch Missing | same as 153. |
| 155 | Z-axis Z Ch Missing | same as 153. |
| 156 | A-axis Z Ch Missing | same as 153. |
| 157 | Motor Interface PCB Failure | Internal circuit board problem. The MOTIF PCB in the processor stack is tested at power-on. Call your dealer. |
| 158 | Video/Keyboard PCB Failure | Internal circuit board problem. The VIDEO PCB in the processor stack is tested at power-on. This could also be caused by a short in the front panel membrane keypad. Call your dealer. |
| 159 | Keyboard Failure | Keyboard shorted or button pressed at power on. A power-on test of the membrane keypad has found a shorted button. It can also be caused by a short in the cable from the main cabinet or by holding a switch down during power-on. |
| 160 | Low Voltage | The line voltage to control is too low. This alarm occurs when the AC line voltage drops below 190 when wired for 230 volts or drops below 165 when wired for 208 volts. |
| 161 | X-Axis Drive Fault | Current in X servo motor beyond limit. Possibly caused by a stalled or overloaded motor. The servos are turned off. This can be caused by running a short distance into a mechanical stop. It can also be caused by a short in the motor or a short of one motor lead to ground. |
| 162 | Y-Axis Drive Fault | same as 161. |
| 163 | Z-Axis Drive Fault | same as 161. |
| 164 | A-Axis Drive Fault | same as 161. |
| 165 | X Zero Ret Margin Too Small | This alarm will occur if the home/limit switches move or are misadjusted. This alarm indicates that the zero return position may not be consistent from one zero return to the next. The encoder Z channel signal must occur between 1/8 and 7/8 revolution of where the home switch releases. This will not turn the servos off but will stop the zero return operation. |

ALARMS

166	Y Zero Ret Margin Too Small	Same as 165.
167	Z Zero Ret Margin Too Small	Same as 165.
168	A Zero Ret Margin Too Small	Not normally enabled for A-axis.
169	Spindle Direction Fault	Problem with rigid tapping hardware. The spindle started turning in the wrong direction.
170	Phase Loss L1-L2	Problem with incoming line voltage between legs L1 and L2. This usually indicates that there was a transient loss of input power to the machine.
171	Phase Loss L2-L3	Problem with incoming line voltage between legs L2 and L3.
172	Phase Loss L3-L1	Problem with incoming line voltage between legs L3 and L1.
173	Spindle Ref Signal Missing	The Z channel pulse from the spindle encoder is missing for hard tapping synchronization.
174	Tool Load Exceeded	The tool load monitor option is selected and the maximum load for a tool was exceeded in a feed. This alarm can only occur if the tool load monitor function is installed in your machine.
175	Ground Fault Detected	A ground fault condition was detected in the 115V AC supply. This can be caused by a short to ground in any of the servo motors, the tool change motors, the fans, or the oil pump.
176	Over Heat Shutdown	An overheat condition persisted for 4.5 minutes and caused an automatic shutdown.
177	Over Voltage Shutdown	An overvoltage condition persisted for 4.5 minutes and caused an automatic shutdown.
178	Divide by Zero	Software Error; Call your dealer.
179	Low Pressure Transmission Oil	Spindle coolant oil is low or low pressure condition in lines.
180	Pallet Not Clamped	Used on horizontal mills only.
182	X Cable Fault	Cable from X-axis encoder does not have valid differential signals.
183	Y Cable Fault	Same as 182.
184	Z Cable Fault	Same as 182.
185	A Cable Fault	Same as 182.
186	Spindle Not Turning	Status from spindle drive indicates error.
187	B Servo Error Too Large	Same as 103.
188	B Servo Overload	Same as 108.

189	B Motor Overheat	Same as 135.
190	B Motor Z Fault	Same as 139.
191	B Limit Switch	Same as 148.
192	B Axis Z Ch Missing	Same as 153.
193	B Axis Drive Fault	Same as 161.
194	B Zero Ret Margin Too Small	Same as 168.
195	B Cable Fault	Same as 182.
196	Coolant Spigot Failure	Spigot failed to achieve commanded location after two (2) attempts.
197	100 Hours Unpaid Bill	Call your dealer.
198	Precharge Failure	During TSC operation, the precharge failed for greater than 0.1 seconds. It will shut off the feed, spindle and pump all at once. If received, check all air lines and the air supply pressure.
199	Negative RPM	A negative spindle RPM was sensed.
201	Parameter CRC Error	Parameters lost maybe by low battery. Check for a low battery and low battery alarm.
202	Setting CRC Error	Settings lost maybe by low battery. Check for a low battery and low battery alarm.
203	Lead Screw CRC Error	Lead screw compensation tables lost maybe by low battery. Check for CRC Error low battery and low battery alarm.
204	Offset CRC Error	Offsets lost maybe by low battery. Check for a low battery and low battery alarm.
205	Programs CRC Error	Users program lost maybe by low battery. Check for a low battery and low battery alarm.
206	Internal Program Error	Software Error; Call your dealer.
207	Queue Advance Error	Software Error; Call your dealer.
208	Queue Allocation Error	Software Error; Call your dealer.
209	Queue Cutter Comp Error	Software Error; Call your dealer.
210	Insufficient Memory	Not enough memory to store users program. Check the space available in the LIST PROG mode and possibly delete some programs.
211	Odd Prog Block	Software Error; Call your dealer.
212	Program Integrity Error	Software Error; Call your dealer.

ALARMS

213	EPROM CRC Error	Electronics fault; Call your dealer.
214	No. of Programs Changed	Indicates that the number of programs disagrees with the internal variable that keeps count of the loaded programs. Call your dealer.
215	Free Memory PTR Changed	Indicates the amount of memory used by the programs counted in the system disagrees with the variable that points to free memory. Call your dealer.
216	EPROM Speed Failure	Indicates that an EPROM internal driver has weakened so that data read from that EPROM may be unreliable. Call your dealer.
217	X Axis Phasing Error	Error occurred in phasing initialization of brushless motor. This can be caused by a bad encoder, or a cabling error.
218	Y Axis Phasing Error	Same as above.
219	Z Axis Phasing Error	Same as above.
220	A Axis Phasing Error	Same as above.
221	B Axis Phasing Error	Same as above.
222	C Axis Phasing Error	Same as above.
224	X Transition Fault	Illegal transition of count pulses in X axis. This alarm usually indicates that the encoder has been damaged and encoder position data is unreliable. This can also be caused by loose connectors at the MOCON PCB.
225	Y Transition Fault	Same as above.
226	Z Transition Fault	Same as above.
227	A Transition Fault	Same as above.
228	B Transition Fault	Same as above.
229	C Transition Fault	Same as above.
231	Jog Handle Transition Fault	Same as 223.
232	Spindle Transition Fault	Same as above.
233	Jog Handle Cable Fault	Cable from jog handle encoder does not have valid differential signals.
234	Spindle Enc. Cable Fault	Cable from spindle encoder does not have valid differential signals.
235	Spindle Z Fault	Same as 139.
240	Empty Prog or No EOB	Software Error; Call your dealer.
241	Invalid Code	RS-232 load bad. Data was stored as comment. Check the program being received.
242	No End	Check input file for a number that has too many digits
243	Bad Number	Data entered is not a number.
244	Missing)	Comment must end with a ") ".

245	Unknown Code	Check input line or data from RS-232. This alarm can occur while editing data into a program or loading from RS-232.
246	String Too Long	Input line is too long. The data entry line must be shortened.
247	Cursor Data Base Error	Software Error; Call your dealer.
248	Number Range Error	Number entry is out of range.
249	Prog Data Begins Odd	Software Error; Call your dealer.
250	Program Data Error	Same as 249.
251	Prog Data Struct Error	Same as 249.
252	Memory Overflow	Same as 249.
257	Program Data Error	Same as 249.
258	Invalid DPRNT Format	Macro DPRNT statement not structured properly.
259	Bad Language Version	Call your dealer.
260	Bad Language CRC	Indicates FLASH memory has been changed. Call your dealer.
261	Rotary CRC Error	Rotary table saved parameters (used by Settings 30, 78) have a CRC error. Indicates a loss of save memory - call your dealer.
262	Parameter CRC Missing	RS-232 or floppy read of parameter had no CRC when loading from floppy or RS-232.
263	Lead Screw CRC Missing	Lead screw compensation tables have no CRC when loading from floppy or RS-232.
264	Rotary CRC Missing	Rotary table parameters have no CRC when loading from floppy or RS-232.
265	Macro Variable File CRC Error	Macro variable file has a CRC error. Indicates a loss of save memory. Call your dealer.
302	Invalid R In G02 or G03	Check your geometry with the HELP page. R must be less than or equal to half the distance from start to end within an accuracy of 0.0010 inches.
303	Invalid X, Y, or Z In G02 or G03	Check your geometry with the HELP page.
304	Invalid I, J, or K In G02 or G03	Check your geometry with the HELP page. Radius at start must match radius at end of arc within 0.0010 inches.
305	Invalid Q In Canned Cycle	Q in a canned cycle must be greater than zero.
306	Invalid I, J, K, or Q In Canned Cycle	I, J, K, and Q in a canned cycle must be greater than zero.
307	Subroutine Nesting Too Deep	Subprogram nesting is limited to nine levels. Simplify your program.

ALARMS

309	Exceeded Max Feed Rate	Use a lower feed rate.
310	Invalid G Code	G code not defined and is not a macro call.
311	Unknown Code	Possible corruption of memory by low battery. Call your dealer.
312	Program End	End of subroutine reached before M99. Need an M99 to return from subroutine.
313	No P Code In M97, M98, or G65	Must put subprogram number in P code.
314	Subprogram or Macro Not In Memory	Check that a subroutine is in memory or that a macro is defined.
315	Invalid P Code In M97, M98 or M99	The P code must be the name of a program stored in memory without a decimal point for M98 and must be a valid N number for M99.
316	X Over Travel Range	X-axis will exceed stored stroke limits. This is a parameter in negative direction and is machine zero in the positive direction. This will only occur during the operation of a user's program.
317	Y Over Travel Range	same as 316.
318	Z Over Travel Range	same as 316.
319	A Over Travel Range	Not normally possible with A-axis.
320	No Feed Rate Specified	Must have a valid F code for interpolation functions.
321	Auto Off Alarm	A fault turned off the servos automatically; occurs in debug mode only.
322	Sub Prog Without M99	Add an M99 code to the end of program called as a subroutine.
324	Delay time Range Error	P code in G04 is over 1000.0 or over 9999.
325	Queue Full	Control problem; call your dealer.
326	G04 Without P Code	Put a Pn.n for seconds or a Pn for milliseconds.
327	No Loop For M Code Except M97, M98	L code not used here. Remove L Code.
328	Invalid Tool Number	Tool number must be between 1 and 16.
329	Undefined M Code	That M code is not defined and is not a macro call.
330	Undefined Macro Call	Macro name O90nn not in memory. A macro call definition is in parameters and was accessed by user program but that macro was not loaded into memory.
331	Range Error	Number too large.
332	H and T Not Matched	This alarm is generated when Setting 15 is turned ON and an H code number in a running program does not match the tool number in the

		spindle. Correct the Hn codes, select the right tool, or turn off Setting 15.
333	X-axis Disabled	Parameters have disabled this axis. Not normally possible in VF Series CNC Mill.
334	Y-axis Disabled	same as 333.
335	Z-axis Disabled	same as 333.
336	A-axis Disabled	Parameters have disabled this axis. Must enable A-axis to program it or remove programming of A-axis. The A-axis can be disabled permanently by Parameter 43 or temporarily by Setting 30.
337	Line Referenced By P, Not Found	Subprogram is not in memory, or P code is incorrect.
338	Invalid IJK and XYZ in G02 or G03	There is a problem with circle definition; check your geometry.
339	Multiple Codes	Only one M, X, Y, Z, A, Q , etc. allowed in any block or two G codes in the same group.
340	Cutter Comp Begin With G02 or G03	Select cutter comp earlier.
341	Cutter Comp End With G02 or G03	Disable cutter comp later.
342	Cutter Comp Path Too Small	Geometry not possible. Check your geometry with the HELP page.
343	Display Queue Record Full	A block exists that is too long for displaying queue. Shorten title block.
344	Cutter Comp With G18 and G19	Cutter comp only allowed in XY plane (G17).
345	Diff Step Ratio On G17 Plane	Parameters 5 and 19 must be same value.
346	Diff Step Ratio On G18 Plane	Parameters 5 and 33 must be same value.
347	Diff Step Ratio On G19 Plane	Parameters 19 and 33 must be same value.
348	Illegal Spiral Motion	Linear axis path is too long. For helical motions, the linear path must not be more than the length of the circular component.
349	Prog Stop W/O Cancel Cutter Comp	Information message only. Fix or ignore.
350	Cutter Comp Look Ahead Error	There are too many non-movement blocks between motions when cutter comp is being used. Remove some intervening blocks.

ALARMS

351	Invalid P Code	In a block with G103 (Block Lookahead Limit), a value between 0 and 15 must be used for the P code.
352	Aux Axis Power Off	Aux B, C, U, V, or W axis indicate servo off. Check auxiliary axes. Status from control was OFF.
353	Aux Axis No Home	A ZERO RET has not been done yet on the aux axes. Check auxiliary axes. Status from control was LOSS.
354	Aux Axis Disconnected	Aux axes not responding. Check auxiliary axes and RS-232 connections.
355	Aux Axis Position Mismatch	Mismatch between machine and aux axes position. Check aux axes and interfaces. Make sure no manual inputs occur to aux axes.
356	Aux Axis Travel Limit	Aux axes are attempting to travel past their limits.
357	Aux Axis Disabled	Aux axes are disabled.
358	Multiple Aux Axis	Can only move one auxiliary axis at a time.
359	Invalid I, J, or K In G12 or G13	Check your geometry with the HELP page.
360	Tool Changer Disabled	Check Parameter 57. Not a normal condition for VF Series CNC Mill.
361	Gear Change Disabled	Check Parameter 57. Not a normal condition for VF Series CNC Mill.
362	Tool Usage Alarm	Tool life limit was reached. To continue, reset the usage count in the Current Commands display and press RESET.
363	Coolant Locked Off	Override is off and program tried to turn on coolant.
364	No Circ Interp Aux Axis	Only rapid or feed is allowed with aux axes.
367	Cutter Comp Interference	G01 cannot be done with tool size.
368	Groove Too Small	Tool too big to enter cut.
369	Tool Too Big	Use a smaller tool for cut.
370	Pocket Definition Error	Check geometry for G150.
371	Invalid I, J, K, OR Q	Check G150.
372	Tool Change In Canned Cycle	Tool change not allowed while canned cycle is active.
373	Invalid Code in DNC	A code found in a DNC program could not be interpreted because of restrictions to DNC.

374	Missing XYZA in G31 or G36	G31 skip function requires an X , Y , Z , or A move.
375	Missing Z or H in G37	G37 auto offset skip function requires H code, Z value, and tool offset enabled. X , Y , and A values not allowed.
376	No Cutter Comp In Skip	Skip G31 and G37 functions cannot be used with cutter compensation.
377	No Skip in Graph/Sim	Graphics mode cannot simulate skip function.
378	Skip Signal Found	Skip signal check code was included but skip was found when it was not expected.
379	Skip Signal Not Found	Skip signal check code was included but skip was not found when it was expected.
380	X, Y, A, or G49 Not Allowed in G37	G37 may only specify Z-axis and must have tool offset defined.
381	G43 or G44 Not Allowed in G36 or G136	Auto work offset probing must be done without tool offset.
382	D Code Required in G35	A Dnn code is required in G35 in order to store the measured tool diameter.
383	Inch Is Not Selected	G20 was specified but settings have selected metric input.
384	Metric Is Not Selected	G21 was specified but settings have selected inches.
385	Invalid L, P, or R Code In G10	G10 was used to changes offsets but L , P , or R code is missing or invalid.
386	Invalid Address Format	An address A...Z was used improperly.
387	Cutter Comp Not Allowed With G103	If block buffering has been limited, Cutter comp cannot be used.
388	Cutter Comp Not Allowed With G10	Coordinates cannot be altered while cutter comp is active. Move G10 outside of cutter comp enablement.
389	G17, G18, G19 Illegal in G68	Planes of rotation cannot be changed while rotation is enabled.
390	No Spindle Speed	S code has not been encountered. Add an S code.
391	Feature Disabled	An attempt was made to use a control feature not enabled by a parameter bit. Set the parameter bit to 1.
392	B Axis Disabled	Same as 336.
393	Invalid Motion In G74 or G84	Rigid Tapping can only be in the Z minus G74 or G84 direction. Make sure that the distance from the initial position to the commanded Z depth is in the minus direction.

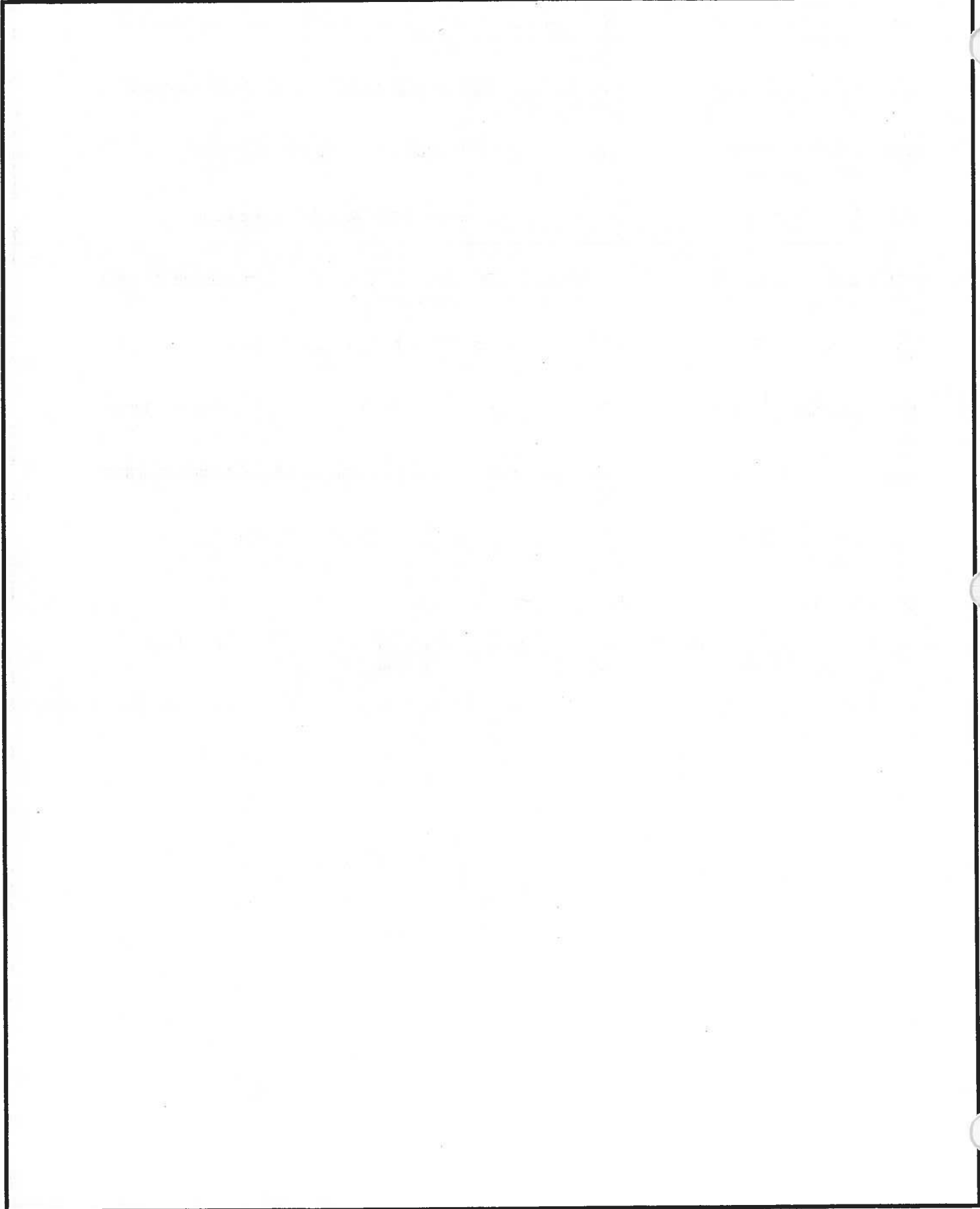
ALARMS

394	B Over Travel Range	Same as 316.
398	Aux Axis Servo Off	Aux. axis servo shut off due to a fault.
403	RS-232 Too Many Progs	Cannot have more than 100 programs in memory.
404	RS-232 No Program Name	Need name in programs when receiving ALL; otherwise has no way to store them.
405	RS-232 Illegal Prog Name	Check files being loaded. Program name must be Onnnn and must be at beginning of a block.
406	RS-232 Missing Code	A receive found bad data. Check your program. The program will be stored but the bad data is turned into a comment.
407	RS-232 Invalid Code	Check your program. The program will be stored but the bad data is turned into a comment.
408	RS-232 Number Range Error	Check your program. The program will be stored but the bad data is turned into a comment.
409	RS-232 Invalid N Code	Bad Parameter or Setting data. User was loading settings or parameters and something was wrong with the data.
410	RS-232 Invalid V Code	Bad parameter or setting data. User was loading settings or parameters and something was wrong with the data.
411	RS-232 Empty Program	Check your program. Between % and % there was no program found.
412	RS-232 Unexpected End of Input	Check Your Program. An ASCII EOF code was found in the input data before program receive was complete. This is a decimal code 26.
413	RS-232 Insufficient Memory	Program received doesn't fit. Check the space available in the LIST PROG mode and possibly delete some programs.
414	RS-232 Buffer Overflow	Data sent too fast to CNC. This alarm is not normally possible as this control can keep up with even 38400 bits per second.
415	RS-232 Overrun	Data sent too fast to CNC. This alarm is not normally possible as this control can keep up with as much as 38400 bits per second.
416	RS-232 Parity Error	Data received by CNC has bad parity. Check parity settings, number of data bits and speed. Also check your wiring.
417	RS-232 Framing Error	Data received was garbled and proper framing bits were not found. One or more characters of the data will be lost. Check parity settings, number of data bits and speed.
418	RS-232 Break	Break condition while receiving. The sending device set the line to a break condition. This might also be caused by a simple break in the cable.
419	Invalid Function For DNC	A code found on input of a DNC program could not be interpreted.

420	Program Number Mismatch	The O code in the program being loaded did not match the O code entered at the keyboard. Warning only.
430	Floppy Unexpected End of Input	Check your program. An ASCII EOF code was found in the input data before program receive was complete. This is a decimal code 26.
431	Floppy No Prog Name	Need name in programs when receiving ALL; otherwise has no way to store them.
432	Floppy Illegal Prog Name	Check files being loaded. Program must be Onnnn and must be at the beginning of a block.
433	Floppy Empty Prog Name	Check your program. Between % and % there was no program found.
434	Floppy Insufficient Memory	Program received doesn't fit. Check the space available in the LIST PROG mode and possibly delete some programs.
435	Floppy Abort	Could not read disk.
436	Floppy File Not Found	Could not find floppy file.
501	Too Many Assignments In One Block	Only one assignment "=" is allowed per block. Divide block in error into multiple blocks.
502	[Or = Not First Term In Expressn	An expression element was found where it was not preceded by "[" or "=", that start expressions.
503	Illegal Macro Variable Reference	A macro variable number was used that is not supported by this control, use another variable.
504	Unbalanced Paren. In Expression	Unbalanced brackets, "[" or "]", were found in an expression. Add or delete a bracket.
505	Value Stack Error	The macro expression value stack pointer is in error. Call your dealer.
506	Operand Stack Error	The macro expression operand stack pointer is in error. Call your dealer.
507	Too Few Operands On Stack	An expression operand found too few operands on the expression stack. Call your dealer.
508	Division By Zero	A division in a macro expression attempted to divide by zero. Re-configure expression.
509	Illegal Macro Variable Use	See "Macros" section for valid variables.
510	Illegal Operator or Function Use	See "Macros" section for valid operators.
511	Unbalanced Right Brackets	Number of right brackets not equal to the number of left brackets.

512	Illegal Assignment Use	Attempted to write to a read-only macro variable.
513	Var. Ref. Not Allowed With N Or O	Alphabetic addresses N and O cannot be combined with macro variables. Do not declare N#1, etc.
514	Illegal Macro Address Reference	A macro variable was used incorrectly with an alpha address. Same as 513.
515	Too Many Conditionals In a Block	Only one conditional expression is allowed in any WHILE or IF-THEN block.
516	Illegal Conditional Or No Then	A conditional expression was found outside of an IF-THEN, WHILE, or M99 block.
517	Exprsn. Not Allowed With N Or O	A macro expression cannot be concatenated to N or O. Do not declare O[#1], etc.
518	Illegal Macro Exprsn Reference	An alpha address with expression, such as A[#1+#2], evaluated incorrectly. Same as 517.
519	Term Expected	In the evaluation of a macro expression an operand was expected and not found.
520	Operator Expected	In the evaluation of a macro expression an operator was expected and not found.
521	Illegal Functional Parameter	An illegal value was passed to a function, such as SQRT[or ASIN[.
522	Illegal Assignment Var Or Value	A variable was referenced for writing. The variable referenced is read only.
523	Conditional Reqd Prior To THEN	THEN was encountered and a conditional statement was not processed in the same block.
524	END Found With No Matching DO	An END was encountered without encountering a previous matching DO. DO-END numbers must agree.
525	Var. Ref. Illegal During Movement	Variable cannot be read during axis movement.
526	Command Found On DO/END Line	A G-code command was found on a WHILE-DO or END macro block. Move the G-code to a separate block.
527	= Not Expected Or THEN Required	Only one Assignment is allowed per block, or a THEN statement is missing.
528	Parameter Precedes G65	On G65 lines all parameters must follow the G65 G-code. Place parameters after G65.
529	Illegal G65 Parameter	The addresses G, L, N, O, and P cannot be used to pass parameters.

530	Too Many I, J, or K's In G65	Only 10 occurrences of I, J, or K can occur in a G65 subroutine call. Reduce the I, J, or K count.
531	Macro Nesting Too Deep	Only four levels of macro nesting can occur. Reduce the amount of nested G65 calls.
532	Unknown Code In Pocket Pattern	Macro syntax is not allowed in a pocket pattern subroutine.
533	Macro Variable Undefined	A conditional expression evaluated to an UNDEFINED value, i.e. #0. Return True or False.
534	DO Or END Already In Use	Multiple use of a DO that has not been closed by and END in the same subroutine. Use another DO number.
535	Illegal DPRNT Statement	A DPRNT statement has been formatted improperly, or DPRNT does not begin block.
536	Command Found On DPRNT Line	A G-code was included on a DPRNT block. Make two separate blocks.
537	RS-232 Abort On DPRNT	While a DPRNT statement was executing, the RS-232 communications failed.
538	Matching END Not Found	A WHILE-DO statement does not contain a matching END statement. Add the proper END statement.
539	Illegal Goto	Expression after "GOTO" not valid.
613	Command Not Allowed In Cutter Comp.	A command (M96, for example) in the highlighted block cannot be executed while cutter comp. is invoked.



8. PARAMETERS

Parameters are seldom-modified values that change the operation of the machine. These include servo motor types, gear ratios, speeds, stored stroke limits, lead screw compensations, motor control delays and macro call selections. These are all rarely changed by the user and should be protected from being changed by the parameter lock setting. If you need to change parameters, contact HAAS or your dealer. Parameters are protected from being changed by Setting 7.

The Settings page lists some parameters that the user may need to change during normal operation and these are simply called "Settings". Under normal conditions, the parameter displays should not be modified. A complete list of the parameters is provided here.

There are 226 parameters in this control. The first 56 apply to the individual servo axes, 14 each. The first 14 of these will be described. The other axes parameters (15 through 56) are identical in function.

8.1 PARAMETER LIST

Parameter 1	<p>X SWITCHES Parameter 1 is a collection of single-bit flags used to turn servo related functions on and off. The left and right cursor arrows are used to select the function being changed. All values are 0 or 1 only. The function names are:</p>	
	REV ENCODER	Used to reverse the direction of encoder data.
	REV POWER	Used to reverse direction of power to motor.
	REV PHASING	Used to reverse motor phasing.
	DISABLED	Used to disable any axis.
	Z CH ONLY	With A only, indicates that no home switch.
	AIR BRAKE	With A only, indicates that air brake is used.
	DISABLE Z T	Disables encoder Z test (for testing only).
	SERVO HIST	Graph of servo error (for diagnostics only).
	INV HOME SW	Inverted home switch (N.C. switch).
	INV Z CH	Inverted Z channel (normally high).
	CIRC. WRAP.	(Future Option - Not Yet Implemented) With A only, causes 360 wrap to return to 0.
	NO I IN BRAK	With A only, removes I feedback when brake is active.
	LOW PASS +1X	Adds 1 term to low pass filter.
	LOW PASS +2X	Adds two terms to low pass filter.
	OVER TEMP NC	Selects a normally closed overheat sensor in motor.
	CABLE TEST	Enables test of encoder signals and cabling.
	Z TEST HIST	History plot of Z channel test data.
	SCALE FACT/X	If set to 1, the scale ratio is interpreted as divided by X; where X depends on bits SCALE/X LO and SCALE/X HI.
	INVIS AXIS	Used to create an invisible axis.
	ROT ALM LMSW	Rotary alarms at the limit switch.
	ROT TRVL LIM	Rotary travel limits are used.
	UNDEFINED	
	UNDEFINED	
	UNDEFINED	
	TORQUE ONLY	For HAAS only.
	3 EREV/MREV	For HAAS only.
	2 EREV/MREV	For HAAS only.
	UNDEFINED	
	BRUSH MOTOR	Enables the brushless motor option.

LINEAR DISPL This bit changes the display from degrees to inches (or millimeters) on the A and B axes.

SCALE/X LO With SCALE/X HI bit, determines the scale factor used in bit SCALE FACT/X,

SCALE/X HI With SCALE/X LO bit, determines the scale factor used in bit SCALE FACT/X. See below:

HI	LO	
0	0	3
0	1	5
1	0	7
1	1	9

Parameter	2	X	P GAIN Proportional gain in servo loop.
Parameter	3	X	D GAIN Derivative gain in servo loop.
Parameter	4	X	I GAIN Integral gain in servo loop.
Parameter	5	X	RATIO (STEPS/UNIT) The number of steps of the encoder per unit of travel. Encoder steps supply four times their line count per revolution. Thus a 2000 line encoder and a 6mm pitch screw give: $2000 \times 4 \times 25.4 / 6 = 33867$
Parameter	6	X	MAX TRAVEL (STEPS) Max negative direction of travel from machine zero in encoder steps. Does not apply to A-axis. Thus a 20 inch travel and 2000 line encoder and 6 mm pitch screw give: $20.0 \times 33867 = 677340$
Parameter	7	X	ACCELERATION Maximum acceleration of axis in steps per second per second.
Parameter	8	X	MAX SPEED Max speed for this axis in steps per second.
Parameter	9	X	MAX ERROR Max error allowed in servo loop before alarm is generated. Units are encoder steps.
Parameter	10	X	FUSE LEVEL Fuse level in % of max power to motor. Applies only when motor in motion.
Parameter	11	X	BACK EMF Back EMF of motor in volts per 1000 RPM times 10. Thus a 63 volt/KRPM motor gives 630.
Parameter	12	X	STEPS/REVOLUTION Encoder steps per revolution of motor. Thus a 2000 line encoder gives: $2000 \times 4 = 8000$.

Parameter	13	X	BACKLASH Backlash correction in encoder steps.
Parameter	14	X	DEAD ZONE Dead zone correction for driver electronics. Units are 0.0000001 seconds.
Parameter	15	Y	SWITCHES See Parameter 1 for description.
Parameter	16	Y	P GAIN See Parameter 2 for description.
Parameter	17	Y	D GAIN See Parameter 3 for description.
Parameter	18	Y	I GAIN See Parameter 4 for description.
Parameter	19	Y	RATIO (STEPS/UNIT) See Parameter 5 for description.
Parameter	20	Y	MAX TRAVEL (STEPS) See Parameter 6 for description.
Parameter	21	Y	ACCELERATION See Parameter 7 for description.
Parameter	22	Y	MAX SPEED See Parameter 8 for description.
Parameter	23	Y	MAX ERROR See Parameter 9 for description.
Parameter	24	Y	FUSE LEVEL See Parameter 10 for description.
Parameter	25	Y	BACK EMF See Parameter 11 for description.
Parameter	26	Y	STEPS/REVOLUTION See Parameter 12 for description.
Parameter	27	Y	BACKLASH See Parameter 13 for description.
Parameter	28	Y	DEAD ZONE See Parameter 14 for description.
Parameter	29	Z	SWITCHES See Parameter 1 for description.
Parameter	30	Z	P GAIN See Parameter 2 for description.

PARAMETERS

Parameter	31	Z	D GAIN See Parameter 3 for description.
Parameter	32	Z	I GAIN See Parameter 4 for description.
Parameter	33	Z	RATIO (STEPS/UNIT) See Parameter 5 for description.
Parameter	34	Z	MAX TRAVEL (STEPS) See Parameter 6 for description.
Parameter	35	Z	ACCELERATION See Parameter 7 for description.
Parameter	36	Z	MAX SPEED See Parameter 8 for description.
Parameter	37	Z	MAX ERROR See Parameter 9 for description.
Parameter	38	Z	FUSE LEVEL See Parameter 10 for description.
Parameter	39	Z	BACK EMF See Parameter 11 for description.
Parameter	40	Z	STEPS/REVOLUTION See Parameter 12 for description.
Parameter	41	Z	BACKLASH See Parameter 13 for description.
Parameter	42	Z	DEAD ZONE See Parameter 14 for description.
Parameter	43	A	SWITCHES See Parameter 1 for description AND make sure that this parameter is set to enable the fourth axis before you try to enable the fourth axis from settings.
Parameter	44	A	P GAIN See Parameter 2 for description.
Parameter	45	A	D GAIN See Parameter 3 for description.
Parameter	46	A	I GAIN See Parameter 4 for description.
Parameter	47	A	RATIO (STEPS/UNIT) See Parameter 5 for description.
Parameter	48	A	MAX TRAVEL (STEPS) See Parameter 6 for description.

Parameter	49	A	ACCELERATION See Parameter 7 for description.
Parameter	50	A	MAX SPEED See Parameter 8 for description.
Parameter	51	A	MAX ERROR See Parameter 9 for description.
Parameter	52	A	FUSE LEVEL See Parameter 10 for description.
Parameter	53	A	BACK EMF See Parameter 11 for description.
Parameter	54	A	STEPS/REVOLUTION See Parameter 12 for description
Parameter	55	A	BACKLASH See Parameter 13 for description.
Parameter	56	A	DEAD ZONE See Parameter 14 for description.

Parameters 57 through 128 are used to control other machine dependent functions. They are:

Parameter	57	COMMON SWITCH 1 Parameter 57 is a collection of general purpose single bit flags used to turn some functions on and off. The left and right cursor arrows are used to select the function being changed. All values are 0 or 1 only. The function names are:
REV CRANK		Reverses direction of jog handle.
DISABLE T.C.		Disables tool changer operations.
DISABLE G.B.		Disables gear box functions.
POF AT E-STP		Causes power off at EMERGENCY STOP.
RIGID TAP		Indicates hardware option for rigid tap.
REV SPIN ENC		Reverses sense direction of spindle encoder.
REPT RIG TAP		Selects repeatable rigid tapping.
EX ST MD CHG		Selects exact stop in moves when mode changes.
SAFETY CIRC.		This enables safety hardware, if machine is so equipped.
SP DR LIN AC		Selects linear deceleration for rigid tapping. 0 is quadratic.
PH LOSS DET		When enabled, will detect a phase loss.
COOLANT SPGT		Enables coolant spigot control and display.
OVER T IS NC		Selects control over temp sensor as N.C.
SKIP OVERSHT		Causes Skip (G31) to act like Fanuc and overshoot sense point.
NONINV SP ST		Non-inverted spindle stopped status.
SP LOAD MONI		Spindle load monitor option is enabled.
SP TEMP MONI		Spindle temperature monitor option is enabled.
ENA ROT & SC		Enables rotation and scaling.
ENABLE DNC		Enables DNC selection from MDI.
ENABLE BGEDT		Enables BACKGROUND EDIT mode.
ENA GRND FLT		Enables ground fault detector.
KEYBD SHIFT		Enables use of keyboard with shift functions.
ENABLE MACRO		Enables macro functions.

INVERT SKIP		Invert sense of skip to active low=closed.
HANDLE CURSR		Enable use of jog handle to move cursor.
NEG WORK OFS		Selects use of work offsets in negative direction.
SPIN COOLANT		Enables spindle low oil pressure detection.
ENA CONVERSE		Enables conversational programming.
OILER ON/OFF		Enables oiler power when servos or spindle is in motion.
NC OVER VOLT		Inverts sense of over voltage signal.
ALT CHAR SET		Enables alternate character set on CRT.
DOOR STOP SP		Enables functions to stop spindle and manual ops at door switch.
Parameter	58	<p>LEAD COMPENS SHIFT</p> <p>Shift factor when applying lead screw compensation. Lead screw compensation is based on a table of 256 offsets; each +/-127 encoder steps. A single entry in the table applies over a distance equal to two raised to this parameter power encoder steps.</p>
Parameter	59	<p>MAX FEED RATE (INCH)</p> <p>Maximum feed rate in inches per minute.</p>
Parameter	60	<p>TURRET START DELAY</p> <p>Maximum delay allowed in start of tool turret. Units are milliseconds. After this time, an alarm is generated.</p>
Parameter	61	<p>TURRET STOP DELAY</p> <p>Maximum delay allowed in motion of tool turret. Units are milliseconds. After this time, an alarm is generated.</p>
Parameter	62	<p>SHUTTLE START DELAY</p> <p>Maximum delay allowed in start of tool shuttle. Units are milliseconds. After this time, an alarm is generated.</p>
Parameter	63	<p>SHUTTLE STOP DELAY</p> <p>Maximum delay allowed in motion of tool shuttle. Units are milliseconds. After this time, an alarm is generated.</p>
Parameter	64	<p>Z TOOL CHANGE OFFSET</p> <p>On Vertical mills: For Z-axis; displacement from home switch to tool change position and machine zero. About 4.6 inches so for a 2000 line encoder this gives:</p> $4.6 \times 33867 = 155788$ <p>On Horizontal mills, this parameter is not used. It should be set to zero.</p>
Parameter	65	<p>NUMBER OF TOOLS</p> <p>Number of tool positions in tool changer. This number must be 20 for the present VMC configuration.</p>
Parameter	66	<p>SPINDLE ORI DELAY</p> <p>Maximum delay allowed when orienting spindle. Units are milliseconds. After this time, an alarm is generated.</p>
Parameter	67	<p>GEAR CHANGE DELAY</p> <p>Maximum delay allowed when changing gears. Units are milliseconds. After this time, an alarm is generated.</p>

Parameter	68	DRAW BAR MAX DELAY Maximum delay allowed when clamping and unclamping tool. Units are milliseconds. After this, time an alarm is generated.
Parameter	69	A AIR BRAKE DELAY Delay provided for air to release from brake on A-axis prior to moving. Units are milliseconds.
Parameter	70	MIN SPIN DELAY TIME Minimum delay time in program after commanding new spindle speed and before proceeding. Units are milliseconds.
Parameter	71	DRAW BAR OFFSET Offset provided in motion of Z-axis to accommodate the tool pushing out of the spindle when unclamping tool. Units are encoder steps.
Parameter	72	DRAW BAR Z VEL UNCL Speed of motion in Z-axis to accommodate tool pushing out of the spindle when unclamping tool. Units are encoder steps per second.
Parameter	73	SP HIGH G/MIN SPEED Command speed used to rotate spindle motor when orienting spindle in high gear. Units are 5000/256 RPM.
Parameter	74	SP LOW G/MIN SPEED Command speed used to rotate spindle motor when orienting spindle in low gear. Units are 5000/256 RPM.
Parameter	75	GEAR CHANGE SPEED Command speed used to rotate spindle motor when changing gears. Units are 5000/256RPM.
Parameter	76	LOW AIR DELAY Delay allowed after sensing low air pressure before alarm is generated. Alarm skipped if air pressure returns before delay. Units are 1/50 seconds.
Parameter	77	SP LOCK SETTLE TIME Required time in milliseconds that the spindle lock must be in place and stable before spindle orientation is considered complete.
Parameter	78	GEAR CH REV TIME Time in milliseconds before motor direction is reversed while in a gear change.
Parameter	79	SPINDLE STEPS/REV Sets the number of encoder steps per revolution of the spindle. Applies only to rigid tapping option.
Parameter	80	MAX SPIN DELAY TIME The maximum delay time control will wait for spindle to get to commanded speed or to get to zero speed. Units are milliseconds.
Parameter	81	M MACRO CALL O9000 M code that will call O9000. Zero causes no call.

PARAMETERS

Parameter	82	M MACRO CALL 09001	same as 81
Parameter	83	M MACRO CALL 09002	same as 81
Parameter	84	M MACRO CALL 09003	same as 81
Parameter	85	M MACRO CALL 09004	same as 81
Parameter	86	M MACRO CALL 09005	same as 81
Parameter	87	M MACRO CALL 09006	same as 81
Parameter	88	M MACRO CALL 09007	same as 81
Parameter	89	M MACRO CALL 09008	same as 81
Parameter	90	M MACRO CALL 09009	same as 81
Parameter	91	G MACRO CALL 09010	G code that will call 09010. Zero causes no call.
Parameter	92	G MACRO CALL 09011	same as 91
Parameter	93	G MACRO CALL 09012	same as 91
Parameter	94	G MACRO CALL 09013	same as 91
Parameter	95	G MACRO CALL 09014	same as 91
Parameter	96	G MACRO CALL 09015	same as 91
Parameter	97	G MACRO CALL 09016	same as 91
Parameter	98	G MACRO CALL 09017	same as 91
Parameter	99	G MACRO CALL 09018	same as 91
Parameter	100	G MACRO CALL 09019	same as 91
Parameter	101	IN POSITION LIMIT X	How close motor must be to endpoint before any move is considered complete when not in exact stop (G09 or G61). Units are encoder steps.
Parameter	102	IN POSITION LIMIT Y	Same definition as Parameter 101.
Parameter	103	IN POSITION LIMIT Z	Same definition as Parameter 101.
Parameter	104	IN POSITION LIMIT A	Same definition as Parameter 101.
Parameter	105	X MAX CURRENT	Fuse level in % of max power to motor. Applies only when motor is stopped.
Parameter	106	Y MAX CURRENT	Same definition as Parameter 105.
Parameter	107	Z MAX CURRENT	Same definition as Parameter 105.
Parameter	108	A MAX CURRENT	Same definition as Parameter 105.
Parameter	109	D*D GAIN FOR X	Second derivative gain in servo loop.
Parameter	110	D*D GAIN FOR Y	Second derivative gain in servo loop.
Parameter	111	D*D GAIN FOR Z	Second derivative gain in servo loop.

Parameter	112	D*D GAIN FOR A Second derivative gain in servo loop.
Parameter	113	X ACC/DEC T CONST Exponential acceleration time constant. Units are 1/10000 seconds. This parameter provides for a constant ratio between profiling lag and servo velocity. It is also the ratio between velocity and acceleration. In conjunction with Parameter 7, it defines the speed above which exponential accel/decel is not provided. Thus if Parameter 7 is 1200000 steps/sec/sec and this parameter is 750 (0.075 seconds); the maximum velocity for accurate interpolation should be: $1200000 \times 0.075 = 90000 \text{ steps/second}$ <p>For a 2000 line encoder and 6 mm screw, this would be $60 \times 90000 / 33867 = 159$ inches min</p>
Parameter	114	Y ACC/DEC T CONST Same definition as Parameter 113
Parameter	115	Z ACC/DEC T CONST Same definition as Parameter 113
Parameter	116	A ACC/DEC T CONST Same definition as Parameter 113
Parameter	117	LUB CYCLE TIME If this is set nonzero, it is the cycle time for the lube pump and the Lube pressure switch option is checked for cycling in this time. It is in units of 1/50 seconds.
Parameter	118	SPINDLE REV TIME Time in milliseconds to reverse spindle motor.
Parameter	119	SPINDLE DECEL DELAY Time in milliseconds to decelerate spindle motor.
Parameter	120	SPINDLE ACC/DECEL Accel/decel time constant in steps/ms/ms for spindle motor.
Parameter	121	X PHASE OFFSET The motor phase offset for X motor. This is arbitrary units.
Parameter	122	Y PHASE OFFSET See Parameter 121 for description.
Parameter	123	Z PHASE OFFSET See Parameter 121 for description.
Parameter	124	A PHASE OFFSET See Parameter 121 for description.
Parameter	125	X GRID OFFSET This parameter shifts the effective position of the encoder Z pulse. It can correct for a positioning error of the motor or home switch.

Parameter	126	Y GRID OFFSET See Parameter 125 for description.
Parameter	127	Z GRID OFFSET See Parameter 125 for description.
Parameter	128	A GRID OFFSET See Parameter 125 for description.
Parameter	129	GEAR CH SETTLE TIME Gear change settle time. This is the number of one millisecond samples that the gear status must be stable before considered in gear.
Parameter	130	GEAR STROKE DELAY This parameter controls the delay time to the gear change solenoids when performing a gear change.
Parameter	131	MAX SPINDLE RPM This is the maximum RPM available to the spindle. When this speed is programmed, the D-to-A output will be +10V and the spindle drive must be calibrated to provide this.
Parameter	132	SPIN. Y TEMP. COEF. This parameter controls the amount of correction to the Y-axis in response to heating of the spindle head. It is 10 times the number of encoder steps per degree F.
Parameter	133	SPIN. Z TEMP. COEF. This parameter controls the amount of correction to the Z-axis in response to heating of the spindle head. It is 10 times the number of encoder steps per degree F.
Parameter	134	X EXACT STOP DIST.
Parameter	135	Y EXACT STOP DIST.
Parameter	136	Z EXACT STOP DIST.
Parameter	137	A EXACT STOP DIST. These parameters control how close each axis must be to its end point when exact stop is programmed. They apply only in G09 and G64. They are in units of encoder steps. A value of 34 would give $34/33867 = 0.001$ inch.
Parameter	138	X FRICTION FACTOR
Parameter	139	Y FRICTION FACTOR
Parameter	140	Z FRICTION FACTOR
Parameter	141	A FRICTION FACTOR These parameters compensate for friction on each of the four axes. The units are in 0.004V.
Parameter	142	HIGH/LOW GEAR CHANG This parameter sets the spindle speed at which an automatic gear change is performed. Below this parameter, low gear is the default; above this, high gear is the default.
Parameter	143	DRAW BAR Z VEL CLMP This parameter sets the speed of the Z-axis motion that compensates for tool motion during tool clamping. Units are in encoder steps per second.

Parameter	144	RIG TAP FINISH DIST This parameter sets the finish tolerance for determining the end point of a rigid tapping operation.
Parameter	145	X ACCEL FEED FORWARD This parameter sets the feed forward gain for the X-axis servo. It has no units.
Parameter	146	Y ACCEL FEED FORWARD Same as Parameter 145.
Parameter	147	Z ACCEL FEED FORWARD Same as Parameter 145.
Parameter	148	A ACCEL FEED FORWARD Same as Parameter 145.
Parameter	149	PRE-CHARGE DELAY This parameter sets the delay time from pre-charge to tool release. Units are milliseconds.
Parameter	150	MAX SP RPM LOW GEAR Max spindle RPM in low gear.
Parameter	151	B SWITCHES See Parameter 1 for description.
Parameter	152	B P GAIN See Parameter 2 for description.
Parameter	153	B D GAIN See Parameter 3 for description.
Parameter	154	B I GAIN See Parameter 4 for description.
Parameter	155	B RATIO (STEPS/UNIT) See Parameter 5 for description.
Parameter	156	B MAX TRAVEL (STEPS) See Parameter 6 for description.
Parameter	157	B ACCELERATION See Parameter 7 for description.
Parameter	158	B MAX SPEED See Parameter 8 for description.
Parameter	159	B MAX ERROR See Parameter 9 for description.
Parameter	160	B FUSE LEVEL See Parameter 10 for description.
Parameter	161	B BACK EMF See Parameter 11 for description.

PARAMETERS

Parameter	162	B STEPS/REVOLUTION See Parameter 12 for description.
Parameter	163	B BACKLASH See Parameter 13 for description.
Parameter	164	B DEAD ZONE See Parameter 14 for description.
Parameter	165	IN POSITION LIMIT B Same definition as Parameter 101.
Parameter	166	B MAX CURRENT Same definition as Parameter 105.
Parameter	167	D*D GAIN FOR B Second derivative gain in servo loop.
Parameter	168	B ACC/DEC T CONST Same definition as Parameter 113.
Parameter	169	B PHASE OFFSET See Parameter 121 for description.
Parameter	170	B GRID OFFSET See Parameter 125 for description.
Parameter	171	B EXACT STOP DIST. See Parameters 134 for description.
Parameter	172	B FRICTION FACTOR See Parameter 138 for description.
Parameter	173	B ACCEL FEED FORWARD Same description as Parameter 145.
Parameter	174	SPINDLE B TEMP. COEF. This parameter controls the amount of correction to the B-axis in response to heating of the spindle head. It is 10 times the number of encoder steps per degree F.
Parameter	175	B AIR BRAKE DELAY Delay provided for air to release from brake on B-axis prior to moving. Units are milliseconds.
Parameter	201	X SPINDLE TEMP. COEF. This parameter controls the amount of correction to the X-axis in response to heating of the spindle head. It is 10 times the number of encoder steps per degree F.
Parameter	202	X AIR BRAKE DELAY This parameter is not used.
Parameter	203	Y AIR BRAKE DELAY This parameter is not used.

Parameter	204	Z AIR BRAKE DELAY This parameter is not used.
Parameter	205	A SPINDLE TEMP. COEF. This parameter controls the amount of correction to the A-axis in response to heating of the spindle head. It is 10 times the number of encoder steps per degree F.
Parameter	206	SPIGOT POSITIONS Maximum number of spigot positions.
Parameter	207	SPIGOT TIMEOUT (MS) Maximum timeout allowed for spigot to traverse one spigot location.
Parameter	208	SPIN. FAN OFF DELAY Delay for turning the spindle fan off after the spindle has been turned off.
Parameter	209	COMMON SWITCH 2 Parameter 209 is a collection of general purpose single bit flags used to turn some functions on and off. The left and right cursor arrows are used to select the function being changed. All values are 0 or 1 only. The function names are:
HORIZONTAL		When set to (1), the control identifies the machine as a horizontal mill. The control will then make the necessary adjustments, such as enabling the horizontal tool changer.
RST STOPS T.C. M21-28 @ 540 ENA CONVEYOR		Tool changer can be stopped with RESET button. When enabled (1), M21-M28 is installed at cable 540. Enables chip conveyor, if machine is so equipped.
50% RPD KBD		When (1) the control will support the new style keyboards with the 50% rapid traverse key. For controls without a 50% rapid keypad set this bit to (0).
FRONT DOOR		When enabled the control will look for an additional door switch and will generate an operator message.
TC Z NO HOME		In Horizontal mills only. This bit prevents Z-axis motion to machine zero prior to a tool change.
M36 AUTO MOT		In Horizontal only. When set to (1), an M36 rotates the A-axis after the PART READY button is pressed.
AUX AXIS TC		In Horizontal mills only. When enabled, means the tool changer carousel is driven by an aux. axis.
SPIGOT KEY INV		When (1) the commands to the conveyor motor are reversed so that forward becomes reverse. If the conveyor is wired incorrectly, this bit can be set so that the conveyor runs in the proper direction.
T SUBROUTINE SPIN Y ENCDR REV CONVEYOR		Reserved for future use. For Lathe only. When enabled, spindle encoder input is to the Y-axis. Reverses the direction of the chip conveyor.
M27-M28 CONVYR		Usually the chip conveyor motor and direction relays are attached to the user relays M21 M22. When this bit is set, the control expects to see the conveyor hooked up to M27 and M28.

PARAMETERS

LOPH A ONLY	When (0) three discrete inputs are used to detect power phase loss. When (1) only LOPH A is used to detect phase loss.
GREEN BEACON	When (1) user relay M25 is used to flash a beacon. If the control is in a reset state, the beacon will be off. If the control is running normally, the beacon will be steadily on. If the control is in a M00, M01, M02, M30 feedhold, or single block state, then the beacon will flash.
RED BEACON	When (1) user relay M26 is used to flash a beacon. The beacon flashes if the control is experiencing an alarm or emergency stop condition.
CONVY DR OVRD	When (1) the conveyor will continue to run with the door open. When (0) the conveyor will stop when the door is open, but will resume when the door is closed. For safety it is recommended that the bit be set to (0).
DSBL CLNT IN	If set to 1 low coolant input will not be used.
DSC INP PR	Discrete pallet rotate/part ready; inputs enabled if set to 1.
RMT TOOLS RLS	If set to 1, allows use of remote tool release button on spindle head.
FLOPPY ENABL	If set to 1, enables the optional floppy drive.
TCR KEYPAD	If set to 1, enables tool changer restore button on keypad.
MCD RLY BRD	If set to 1, adds 8 additional relays, for a total of 40. These additional relays (M21-M28) become available on a secondary board, and are shown on the discrete outputs page.
TSC ENABLE	When set to 1, "DSBL CLNT IN" bit is ignored, and TSC will operate. When set to zero, the control functions normally.
AUX JOG NACC	Does not allow accumulation on auxiliary axis jog.
ALISM PRGRST	Alias M codes during program restart.
DSBL JOG TST	Disables the encoder test for the jog handle.
AIR DR @ M24	Used on horizontal mills only.
UNDEFINED	
P RDY @ Y160	Used on horizontal mills only.
SPNDL NOWAIT	When (1), the machine will not wait for the spindle to come up to speed immediately after an M03 or M04 command. Instead, it will check and/or wait for the spindle to come up to speed immediately before the next interpolated motion is initiated. This bit does not affect rigid tapping or the TSC option.
Parameter 210	X TOOL CHANGE OFFSET This parameter is not used.
Parameter 211	Y TOOL CHANGE OFFSET On Horizontal mills: For Y-axis; displacement from the home position to tool change position. On Vertical mills, this parameter is not used.
Parameter 212	A TOOL CHANGE OFFSET This parameter is not used.
Parameter 213	B TOOL CHANGE OFFSET This parameter is not used.

Parameter 214	RESERVED
Parameter 215	CAROUSEL OFFSET Parameter used to align tool 1 of tool changing carousel precisely. Units are encoder steps.
Parameter 216	CNVYR RELAY DELAY Delay time in 1/50 seconds required on conveyor relays before another action can be commanded. Default is 5.
Parameter 217	CNVYR IGNORE OC TIM Amount of time in 1/50 seconds before overcurrent is checked after conveyor motor is turned on. Default is 50.
Parameter 218	CONVYR RETRY REV TIM Amount of time that the conveyor is reversed in 1/50 seconds after overcurrent is sensed. Default is 200.
Parameter 219	CONVYR RETRY LIMIT Number of times that the conveyor will cycle through the reverse/forward sequencing when an overcurrent is sensed before the conveyor will shut down. An overcurrent is sensed when chips jam the conveyor. By reversing and then forwarding the conveyor, the chip jam may be broken. Default is 3.
Parameter 220	CONVYR RETRY TIMEOUT Amount of time in 1/50 seconds between consecutive overcurrents in which the overcurrents is considered another retry. If this amount of time passes between overcurrents then the retry count is set to (0). Default is 1500, 30 minutes.
Parameter 221	MAX TIME NO DISPLAY The maximum time (in 1/50 sec.) between screen updates.
Parameter 222	ROTARY AXIS INCRMNT For Horizontal mills only. This parameter sets the degrees of rotation of the A-axis at an M36 or Pallet Rotate.
Parameter 223	AIR TC DOOR DELAY For Horizontal mills only. This parameter sets the delay to open the tool changer door (in milliseconds). If the tool changer does not have a pneumatic door, this parameter is set to zero.
Parameter 224	ROT AXIS ZERO OFFSET This parameter shifts the zero point of A for a wheel fixture or tombstone.
Parameter 225	MAX ROT AXIS ALLOW For Horizontal mills with a wheel fixture only. This parameter sets the maximum rotation (in degrees) allowed before stopping at front door.
Parameter 226	RESERVED
Parameter 227	FLOPPY DIR NAME This parameter sets the program numbers to store in the floppy directory.

Parameter 228	QUICKCODE FILE This parameter set the program numbers to store in the Quickcode definition.
Parameter 229	X LEAD COMP 10E9 This parameter sets the X-axis lead screw compensation signed parts per billion.
Parameter 230	Y LEAD COMP 10E9 This parameter sets the Y-axis lead screw compensation signed parts per billion.
Parameter 231	Z LEAD COMP 10E9 This parameter sets the Z-axis lead screw compensation signed parts per billion.
Parameter 232	A LEAD COMP 10E9 This parameter sets the A-axis lead screw compensation signed parts per billion.
Parameter 233	B LEAD COMP 10E9 This parameter sets the B-axis lead screw compensation signed parts per billion.
Parameter 235	TSC PISTON SEAT With the TSC option, the amount of time given for the piston to seat during system start-up. The default is 500 milliseconds.
Parameter 236	TSC CLNT STABILIZE With the TSC option, the amount of time given for the coolant to stabilize during system start-up. The default is 1000 milliseconds (1200 msec for VF-6/8).
Parameter 237	TSC CLNT LINE PURGE The amount of time given for the coolant to purge when the TSC system is shut off. The default is 2500 milliseconds.
Parameter 238	MAX TSC SPINDLE RPM When Through the Spindle Coolant (TSC) is enabled and in use, this parameter limits the maximum spindle speed.
Parameter 239	SPNDL ENC STEPS/REV This parameter sets the number of encoder steps per revolution of the spindle encoder.
Parameter 240	C AXIS MAX TRAVEL This parameter sets the C-axis maximum travel in the positive direction.
Parameter 241	U AXIS MAX TRAVEL This parameter sets the U-axis maximum travel in the positive direction.
Parameter 242	V AXIS MAX TRAVEL This parameter sets the V-axis maximum travel in the positive direction.
Parameter 243	W AXIS MAX TRAVEL This parameter sets the W-axis maximum travel in the positive direction.
Parameter 244	C AXIS MIN TRAVEL This parameter sets the C-axis minimum travel in the negative direction.

Parameter 245	U AXIS MIN TRAVEL This parameter sets the U-axis minimum travel in the negative direction.
Parameter 246	V AXIS MIN TRAVEL This parameter sets the V-axis minimum travel in the negative direction.
Parameter 247	W AXIS MIN TRAVEL This parameter sets the W-axis minimum travel in the negative direction.
Parameter 256	PALLET LOCK INPUT Used in horizontal mills only.
Parameter 278	COMMON SWITCH 3 Parameter 278 is a collection of general purpose single bit flags used to turn some functions on and off. The left and right cursor arrows are used to select the function being changed. All values are 0 or 1 only. The function names are:
INVERT G.B.	This bit allows an alternate gearbox configuration. It inverts the sense of the gearbox inputs.
DPR SERIAL	Causes the main serial inputs/outputs to go through the floppy video board.
CK PALLET IN	This bit is used on horizontal mills only.
CK HIDN VAR	This bit is used on horizontal mills only.
ACT DISPLAY	When set to 1, displays the actual spindle speed on the Current Commands display page.

8.2 LEAD SCREW COMPENSATION

Separate lead screw compensation is provided for each of the X, Y, and Z axes. The operator-entered compensation values are spaced at 0.5 inch intervals within the machine coordinate system. The compensation values are entered in inches with a resolution of 0.0001 inch. The operator entered values are used to interpolate into a table of 256 entries. The spacing between two entries in the table of 256 is defined by Parameter 58. The entered values are limited to +/-127 encoder steps; so the limit in inches is dependent on Parameters 5, 19, and 33.

Note that the first entry corresponds to machine position zero and subsequent entries are for increasingly negative positions in the machine coordinate system. The user should not ever need to adjust the lead screw compensation tables.

9. SETTINGS

The setting pages contain parameters that the user may need to change and that control machine operation. Most settings can be changed by the operator. The settings are preceded by a short description on the left and the value on the right. In general, settings allow the operator or setup person to lock out or turn on specific functions.

Machines with software version 4.4 or later have the settings organized into pages of functionally similar groupings. This will make it easier for the user to remember where the settings are located and reduces the amount of time spent maneuvering through the settings display. The list below is separated into page groups with the page title as the heading.

Use the vertical cursor keys to move to the desired setting. Depending on the setting, you may change it by entering a new number or, if the setting has specific values, press the horizontal cursor keys to display the choices. Press the WRITE key to enter or change the value. The message near the top of the screen tells you how to change the selected setting.

The serial number is Setting 26 on this page and is protected from user change. If you need to change this setting, contact HAAS or your dealer.

One of the more commonly adjusted settings will be number 34, the "Rotary Axis Diameter". This setting is used to control the surface feed rate when the fourth axis is used in a cutting feed. Feeding with the X, Y, or Z and the A axes assumes that the linear motion is along the axis of the rotary motion. When this is true and the diameter setting is correct, the programmed surface feed rate will be correct for helical cuts. In addition, feeds of just the A-axis depend on this setting to determine the correct angular rate.

If you have a fourth axis 5C, fifth axis, or rotary table, it may be disabled from the setting page and removed from the machine. Do not connect or disconnect any cables with the control on. If you do not disable the fourth axis when it is disconnected, you will get an alarm.

Settings may be sent and received with the RS-232 port. See the "Data Input / Output" section for a description of how to do this.

The settings are listed here with a description of each. The page title will precede each page of settings and the settings will appear in order as shown on the screen.

Page Name:	Setting No:	Description:	Range of Value:
GENERAL			
	26	SERIAL NUMBER	1000 to 99999
	82	LANGUAGE	English, German, French, Italian, Spanish
	1	AUTO POWER OFF TIMER	0 to 9999 minutes
	81	TOOL AT POWER DOWN	0 to 20
	9	DIMENSIONING	INCH or METRIC
	77	SCALE INTEGER F	ON or OFF
	33	COORDINATE SYSTEM	FANUC, YASNAC, or HAAS
	53	JOG W/O ZERO RETURN	ON or OFF
	40	TOOL OFFSET MEASURE	RADIUS or DIAMETER
	64	T. OFS MEAS USES WORK	ON or OFF

PROGRAM 1

2	POWER OFF AT M30	ON or OFF
31	RESET PROGRAM POINTER	ON or OFF
36	PROGRAM RESTART	ON or OFF
39	BEEP AT M30	ON or OFF
51	DOOR HOLD OVERRIDE	ON or OFF
56	M30 RESTORE DEFAULT G	ON or OFF
59	PROBE OFFSET X+	-30.0000 to +30.0000 inches
60	PROBE OFFSET X-	-30.0000 to +30.0000 inches
61	PROBE OFFSET Y+	-30.0000 to +30.0000 inches
62	PROBE OFFSET Y-	-30.0000 to +30.0000 inches
63	TOOL PROBE WIDTH	-30.0000 to +30.0000 inches
71	DEFAULT G51 SCALING	.001 to 8380.000
72	DEFAULT G68 ROTATION	0.0000 to 360.0000
73	G68 INCREMENTAL ANGLE	ON or OFF

PROGRAM 2

30	4TH AXIS ENABLE	OFF, S5C, HA5C, 5CN, 7RT, 9RT, SRT, 11RT, 160, 210, 310, 450
34	ROTARY AXIS DIAMETER	0 to 29.9999 inches
78	5TH AXIS ENABLE	OFF, S5C, HA5C, 5CN, 7RT, 9RT, SRT, 11RT, 160, 210, 310, 450
79	FIFTH AXIS DIAMETER	0 to 29.9999 inches
38	AUX AXIS NUMBER	0 to 5
22	CAN CYCLE DELTA Z	0 to 29.9999 inches
28	CAN CYCLE ACT W/O X/Y	ON or OFF
52	G83 RETRACT ABOVE R	0.0 to 9.9999 inches
57	EXACT STOP CANNED X-Y	ON or OFF
43	CUTTER COMP TYPE	A or B
44	MIN F IN RADIUS CC %	1 to 100
58	CUTTER COMPENSATION	FANUC or YASNAC
85	IN POSITION ACCURACY	0 to 0.25

PROGRAM 3

15	H & T CODE AGREEMENT	ON or OFF
27	G76/G77 SHIFT DIR.	X+, X-, Y+, or Y-
29	G91 NON-MODAL	ON or OFF
32	COOLANT OVERRIDE	NORMAL, OFF, or IGNORE
35	G60 OFFSET	0 to 0.9999 inches
42	M00 AFTER M06	ON or OFF
49	SKIP SAME TOOL CHANGE	ON or OFF
86	M39 LOCKOUT	ON or OFF
45	MIRROR IMAGE X-AXIS	ON or OFF
46	MIRROR IMAGE Y-AXIS	ON or OFF
47	MIRROR IMAGE Z-AXIS	ON or OFF
48	MIRROR IMAGE A-AXIS	ON or OFF
80	MIRROR IMAGE B-AXIS	ON or OFF

RS-232 PORTS

11	BAUD RATE SELECT	50, 110, 200, 300, 600, 1200, 2400, 4800, 7200, 9600, 19200, 38400, or 115200 (optional)
12	PARITY SELECT	NONE, ODD, EVEN, ZERO
13	STOP BIT	1 or 2
14	SYNCHRONIZATION	XON/XOFF, RTS/CTS, DC CODES, or XMODEM
37	RS-232 DATA BITS	7 or 8

SETTINGS

24	LEADER TO PUNCH	NONE, BLANK, or NUL CHAR
25	EOB PATTERN	CR LF, LF ONLY, CR ONLY, or LF CR CR
41	ADD SPACES RS232 OUT	ON or OFF
50	AUX AXIS SYNC	XON/XOFF, RTS/CTS, DC CODES, or XMODEM
54	AUX AXIS BAUD RATE	50, 110, 200, 300, 600, 1200, 2400, 4800, 7200, 9600, 19200, or 38400
69	DPRNT LEADING SPACES	ON or OFF
70	DPRNT OPEN/CLOS DCODE	ON or OFF

CONTROL PANEL

6	FRONT PANEL LOCK	ON or OFF
55	ENABLE DNC FROM MDI	ON or OFF
76	TOOL RELEASE LOCK OUT	ON or OFF
16	DRY RUN LOCK OUT	ON or OFF
17	OPT STOP LOCK OUT	ON or OFF
18	BLOCK DELETE LOCK OUT	ON or OFF
10	LIMIT RAPID AT 50%	ON or OFF
103	CYC START/FH SAME KEY	ON or OFF
104	JOG HANDL TO SNGL BLK	ON or OFF
84	TOOL OVERLOAD ACTION:	ALARM, FEEDHOLD, BEEP, PAUSE

EDITING

7	PARAMETER LOCK	ON or OFF
8	MEMORY PROTECT	ON or OFF
23	9xxxx PROGS EDIT LOCK	ON or OFF
74	9xxxx PROGS TRACE	ON or OFF
75	9xxxx PROGS SINGLE BLK	ON or OFF

GRAPHICS

3	3D GRAPHICS	ON or OFF
4	GRAPHICS RAPID PATH	ON or OFF
5	GRAPHICS DRILL POINT	ON or OFF
65	GRAPH SCALE (HEIGHT)	0 to 16.250
66	GRAPHICS X OFFSET	0 to 30
67	GRAPHICS Y OFFSET	0 to 24
68	GRAPHICS Z OFFSET	0 to 16

OVERRIDES

19	FEEDRATE OVERRIDE LOCK	ON or OFF
20	SPINDLE OVERRIDE LOCK	ON or OFF
21	RAPID OVERRIDE LOCK	ON or OFF
87	M06 RESET OVERRIDES	ON or OFF
83	M30 RESET OVERRIDE	ON or OFF
88	RESET RESET OVERRIDE	ON or OFF

The following is a detailed description of each of the settings:

1 AUTO POWER OFF TIMER

This is a numeric setting. When it is set to a number other than zero, the machine will be automatically turned off after that many minutes of idle operation. This will not occur while a program is running and will not occur while the operator is pressing any keys. The auto off sequence gives the operator a 30 second warning and pressing any key will interrupt the sequence.

2 POWER OFF AT M30

This is an On/Off setting. If it is set to ON, the machine will begin an automatic power down when an M30 ends a program. The auto off sequence gives the operator a 30 second warning and pressing any key will interrupt the sequence.

3 3D GRAPHICS

This is an On/Off setting. When it is off, graphics are displayed in two parts on the screen; the top 2/3 of the screen is an X-Y look down and the bottom 1/3 of the screen is an X-Z look-across. When it is turned on, graphics are shown from a 3-D point of view.

4 GRAPHICS RAPID PATH

This is an On/Off setting. It changes what is displayed in graphics. When it is off, the rapid motions do not leave a trail. When it is on, rapid motions leave a dashed line on the screen.

5 GRAPHICS DRILL POINT

This is an On/Off setting. It changes what is displayed in graphics. When it is off, nothing is added to the graphics display. When it is on, any motion in the Z-axis will leave an X mark on the screen.

6 FRONT PANEL LOCK

This is an On/Off setting. When it is off, the machine operates normally. When it is on, the spindle CW and CCW buttons are disabled.

7 PARAMETER LOCK

This is an On/Off setting. When it is off, parameters can be changed. When it is on, parameter changes are locked out. When the control is turned on, this setting is set to On.

8 MEMORY PROTECT

This is an On/Off setting. When it is off, memory can be edited. When it is on, memory edit functions are locked out.

9 DIMENSIONING

This is an Inch/Metric setting. When it is set to Inch, the programmed units for X, Y, and Z are inches to 0.0001. When it is set to Metric, programmed units are millimeters to 0.001. Note: Changing this setting will not automatically translate a program already stored in memory. You must reload your programs for the new units. When set to Inch, the Group 6 default G Code is G20. When set to Metric, the default G Code is G21.

10 LIMIT RAPID AT 50%

This is an On/Off setting. When it is off, the highest rapid speed of 100% is available normally. When it is on, the highest rapid rate is limited to 50% of maximum. When you press the 100% button, the display will indicate a 50% rapid override. When this settings is turned on, the rapid override will not automatically change from 100% to 50%; you must press the 100% override buttons to get 50%. If the machine is turned on after this setting is turned on, the override will automatically be to 50%.

11 BAUD RATE SELECT

This setting allows the operator to change the serial data rate for the first serial port. This applies to program, settings, offsets, and parameters upload and download and to DNC functions.

12 PARITY SELECT

This setting allows the setting of parity for the first serial port. The possible values are: NONE, ODD, EVEN, ZERO. When set to none, no parity bit is added to the serial data. When set to zero, a 0 bit is added in the place of parity. Even and odd work like normal parity functions. Make sure you know what your system needs.

13 STOP BIT

This setting changes the number of stop bits for the first serial port. It can be selected to be 1 or 2.

14 SYNCHRONIZATION

This changes the synchronization protocol between sender and receiver for the first serial port. When set to RTS/CTS, the signal wires in the serial data cable are used to tell the sender to temporarily stop sending data while the receiver catches up. When it is set to XON/XOFF, those ASCII character codes are used by the receiver to tell the sender to temporarily stop. XON/XOFF is the most common setting.

DC CODES is like XON/XOFF but the paper tape punch or reader start/stop codes are sent. XMODEM is a receiver-driven communications protocol that sends data in blocks of 128 bytes. XMODEM gives the RS-232 communication added reliability because each block is checked for integrity. Refer to "Data Input/Output" section for more information.

15 H & T CODE AGREEMENT

This is an On/Off setting. When it is **off**, no special functions occur. When it is set to **on**, a check is made to ensure that the H offset code matches the tool presently in the spindle. This check can help to prevent crashes.

16 DRY RUN LOCK OUT

This is an On/Off setting. When it is **off**, the machine operates normally. When it is **on**, the DRY RUN function cannot be turned on.

17 OPT STOP LOCK OUT

This is an On/Off setting. When it is **off**, the machine operates normally. When it is **on**, the OPTIONAL STOP function cannot be turned on.

18 BLOCK DELETE LOCK OUT

This is an On/Off setting. When it is **off**, the machine operates normally. When it is **on**, the BLOCK DELETE function cannot be turned on.

19 FEED RATE OVERRIDE LOCK

This is an On/Off setting. When it is **off**, the machine operates normally. When it is **on**, the feed rate override buttons are locked out.

20 SPINDLE OVERRIDE LOCK

This is an On/Off setting. When it is **off**, the machine operates normally. When it is **on**, the spindle speed override buttons are locked out.

21 RAPID OVERRIDE LOCK

This is an On/Off setting. When it is **off**, the machine operates normally. When it is **on**, the rapid speed override buttons are locked out.

22 CAN CYCLE DELTA Z

This is a decimal numeric entry. It must be in the range of 0.0 to 29.9999 inches. This setting specifies the delta Z used in the G73 canned cycle when the Z-axis is moved up to clear chips.

23 9XXX PROGS EDIT LOCK

This is an On/Off setting. When it is **off**, the machine operates normally. When it is **on**, the 9000 series programs (usually macro programs) are invisible to the operator and cannot be uploaded or download. They also cannot be listed, edited, or deleted.

24 LEADER TO PUNCH

This setting is used to control the leader sent to a paper tape punch device connected to the first RS-232 port. The values that can be selected are: NONE, BLANK, or NUL CHAR. None causes no extra data to be sent as a leader. Blank causes two feet of blanks to be punched at the start of a program and one foot of blanks at the end. Null causes the same thing as blanks but uses the ASCII code null which is all zero.

25 EOB PATTERN

This setting controls what is sent out and expected as input to represent the EOB (end of block) on serial port one. The possible selections are: CR LF, LF only, CR only, or LF CR CR.

26 SERIAL NUMBER

This is a numeric entry. It is the serial number of your machine. It cannot be changed.

27 G76/G77 SHIFT DIR.

This setting controls the shift direction used to clear a boring tool during a G76 or G77 canned cycle. The possible selections are: X+, X-, Y+, or Y-.

28 CAN CYCLE ACT W/O X/Y

This is an On/Off setting. When it is **off**, an initial canned cycle definition without an **X** or **Y** motion will not cause the canned cycle to be executed. When it is **on**, the initial canned cycle definition will cause one cycle to be executed even if there is no **X** or **Y** motion in that command block. Note that if an **L0** is in that block, it will never execute the canned cycle on the definition line.

29 G91 NON-MODAL

This is an On/Off setting. When it is **off**, the machine operates normally. When it is **on**, G91 is not modal and applies only to the command block on which it occurs.

30 4TH AXIS ENABLE

When it is **off**, the fourth axis is disabled and no commands can be sent to that axis. When **off** is not selected, the selected rotary table type parameters are called up. A change to rotary parameters is saved under the selected table type for later recall. In order to change this setting the servos must be turned off.

31 RESET PROGRAM POINTER

This is an On/Off setting. When it is **off**, the RESET button will not change the execution program pointer. When it is **on**, a RESET will change the program execution pointer to the beginning of the program.

32 COOLANT OVERRIDE

This setting controls how the coolant pump operates. The possible selections are: NORMAL, OFF, or IGNORE. The "NORMAL" setting checks the low coolant alarm, allows the operator to turn the pump on and off, and turns the pump on and off with **M** codes. The "OFF" setting ignores the coolant alarm and will alarm if an attempt is made to turn the coolant on manually or from a program. The "IGNORE" setting is used to ignore all coolant commands and the low coolant alarm but the pump can be turned on manually.

33 COORDINATE SYSTEM

This setting changes the way the G92/G52 offset system works. It can be set to FANUC, HAAS, or YASNAC. The Fanuc and HAAS class of controls uses the G52 offset differently than the Yasnac class of controls. In a Fanuc or HAAS control, G52 sets a local coordinate system. In a Yasnac control, the G52 stands on its own as another work offset.

34 4TH AXIS DIAMETER

This is a numeric entry. It is used to set the angular feed rate of the A-axis. It must be in the range of 0.0 to 29.9999 inches. Since the feed rate specified in a program is always inches per minute (or mm per minute), the control must know the diameter of the part being worked in the A-axis in order to compute the angular

feed rate. When this setting is set correctly, the surface feed rate on a rotary cut will be exactly the feed rate programmed into the control.

35 G60 OFFSET

This is a numeric entry in the range of 0.0 to 0.9999 inches. It is used to specify the amount of overshoot when unidirectional positioning (G60) is programmed.

36 PROGRAM RESTART

This is an On/Off setting. When it is **off**, starting a program from anywhere other than the beginning may produce inconsistent results. When it is **on**, starting a program from the middle causes the entire program to be scanned to ensure that the correct tools, offsets, G codes, and axes positions are set correctly before starting at the block where the cursor is positioned.

37 RS-232 DATA BITS

This setting can be selected to be either 7 or 8. It is used to change the number of data bits for serial port one. Normally, seven data bits should be used. Some computers require eight. Note that parity is added to this count.

38 AUX AXIS NUMBER

This is a numeric entry between 0 and 5. It is used to select the number of external auxiliary axes added to the system. If it is set to 0, there are no auxiliary axes. If it is set to 1, there is a B-axis. If it is set to 2, there are B and C axes.

39 BEEP AT M30

This is an On/Off setting. When it is **off**, nothing is changed. When it is **on**, a program ending in an M30 will cause the keyboard beeper to sound until another keyboard key is pressed.

40 TOOL OFFSET MEASURE

This setting selects how tool size is specified for cutter compensation. It can be set to either Radius or Diameter. The value in the tool offset tables is used differently depending on this setting. In addition, the label on the offsets page changes to indicate how offsets should be entered.

41 ADD SPACES RS232 OUT

This is an On/Off setting. When it is **off**, programs sent out the serial port have no spaces and are difficult to read. When it is **on**, spaces are added between address codes when a program is sent out RS-232 serial port one. This can make program much easier to read.

42 M00 AFTER M06

This is an On/Off setting. When it is **off**, tool changes occur normally. When it is **on**, a program stop will occur after a tool change and M00 AFTER M06 is displayed as a message at the bottom left. This affects only programmed tool changes.

43 CUTTER COMP TYPE

This setting controls how an entry to cutter compensation occurs. It can be selected to be A or B. It affects only the first stroke that begins cutter compensation and changes the way the tool is cleared from the part being cut.

44 MIN F IN RADIUS CC %

This setting is a numeric entry between 1 and 100. It affects the feed rate when cutter compensation moves the tool towards the inside of a circular cut. In order to maintain a constant surface feed rate, such a cut will be slowed down. This setting specifies the minimum feed rate as a percentage of the programmed feed rate.

45 MIRROR IMAGE X-AXIS

46 MIRROR IMAGE Y-AXIS

47 MIRROR IMAGE Z-AXIS**48 MIRROR IMAGE A-AXIS**

These are On/Off settings. When it is **off**, axes motions occur normally. When it is **on**, the specific axis motion is mirrored (or reversed) around the work zero point.

49 SKIP SAME TOOL CHANGE

This is an On/Off setting. When it is **off**, an M16 will always cause a tool change sequence to occur; even if the same tool is put back into the spindle. When it is **on**, a tool change to the same tool as is in the spindle will cause no action.

50 AUX AXIS SYNC

This changes the synchronization protocol between sender and receiver for the second serial port. When set to RTS/CTS, the signal wires in the serial data cable are used to tell the sender to temporarily stop sending data while the receiver catches up. When it is set to XON/XOFF, those ASCII character codes are used by the receiver to tell the sender to temporarily stop. XON/XOFF is the most common setting. Make sure that the Haas servo control is set to the same condition.

DC CODES is like XON/XOFF but the paper tape punch or reader start/stop codes are sent. XMODEM is a receiver-driven communications protocol that sends data in blocks of 128 bytes. XMODEM gives the RS-232 communication added reliability because each block is checked for integrity. Refer to "Data Input/Output" section for more information.

51 DOOR HOLD OVERRIDE

This is an On/Off setting. When it is **off**, a program cannot be started when the doors are open and opening the doors will cause a running program to stop just like in FEED HOLD. When it is **on**, and Parameter 57 bits DOOR STOP SP and SAFETY CIRC are set to zero, the door condition is ignored. When the control is turned on, this setting is set to Off.

52 G83 RETRACT ABOVE R

This is a numeric entry in the range of 0.0 to 9.9999 inches. This setting changes the way G73 works when it returns to the **R** plane. Most programmers set the **R** plane well above the cut to ensure that the chip clear motion actually allows the chips to get out of the hole but this causes a wasted motion when first drilling through this "empty" space. If Setting 52 is set to the distance required to clear chips, the **R** plane can be put much closer to the part being drilled. When the clear move to **R** occurs, the **Z** will be moved above **R** by this setting.

53 JOG W/O ZERO RETURN

This is an On/Off setting. When it is **off**, jogging of an axis is inhibited until the zero return operation is completed. When it is **ON**, jogging of an axis is allowed prior to the zero return. The **ON** condition can be dangerous in that an axis can be run into the stops, however, the maximum speed allowed is one inch per minute or 0.0010 inches per handle increment. When the control is turned on, this setting is set to **OFF**.

54 AUX AXIS BAUD RATE

This setting allows the operator to change the serial data rate for the second serial port. This applies to the interface with the optional **B**, **C**, **U**, **V**, and **W** axes. The possible values include: 50, 110, 200, 300, 600, 1200, 2400, 4800, 7200, 9600, 19200, 38400. Note that 4800 is standard in Haas servo controls and this should be set to the same value.

55 ENABLE DNC FROM MDI

This is an On/Off setting. When it is **off**, DNC cannot be selected. When it is turned **on**, DNC is selected by pressing MDI while already in MDI. The DNC option must be enabled in the control.

56 M30 RESTORE DEFAULT G

This is an On/Off setting. When it is **off**, no change to the modal **G** codes occurs at the end of a program

(normally M30). When it is **on**, an M30 will reset all of the modal **G** codes to their defaults. If this setting is **on**, RESET will also reset defaults.

57 EXACT STOP CANNED X-Y

This is an On/Off setting. When it is **off**, the rapid X-Y motion associated with a canned cycle may not get exact stop; according to other conditions. When it is **on**, the X-Y motion always gets exact stop. This will make canned cycles slower but less likely to run into a close tolerance fixture.

58 CUTTER COMPENSATION

This setting controls the type of cutter compensation used in the control. The types are similar to the method of cutter compensation available in those classes of controls.

59 PROBE OFFSET X+

60 PROBE OFFSET X-

61 PROBE OFFSET Y+

62 PROBE OFFSET Y-

Settings 59 through 62 are used to define the displacement and size of the spindle probe. These numbers only apply to the probing option. These four numbers specify the travel distance in four directions from where the probe is triggered to where the actual sensed surface is located. They are used by G31, G36, G136, and M75. They can be both positive and negative numbers. If the probe width were 0.23 inches in diameter and the probe was set exactly at the center of the spindle, these four settings would all be 0.115 inches.

63 TOOL PROBE WIDTH

This setting is used to specify the width of the probe that is used to test tool diameter. This setting only applies to the probing option. It is used by G35.

64 T. OFS MEAS USES WORK

This is an on/off setting. It changes the way the TOOL OFFSET MESUR button works. When this is ON, the entered tool offset will be relative to the currently selected work coordinate Z offset. When it is OFF, the tool offset equals the Z machine position.

65 GRAPH SCALE (HEIGHT)

This setting specifies what amount of the work area the height of the graphics screen indicates in inches. A specific scale can be set by using the following formula. Its default is the Total Y travel.

$$\begin{aligned} \text{Total Y travel} &= \text{Parameter 20} / \text{Parameter 19} \quad (16.25 \text{ VF-0 through VF-2}) \\ \text{Scale} &= \text{Total Y travel} / \text{Setting 65} \end{aligned}$$

66 GRAPHICS X OFFSET

This setting locates the right side of the scaling window relative to the machine X zero position, (See "Displays" section). Its default is zero.

67 GRAPHICS Y OFFSET

This setting locates the top of the scaling window relative to the machine Y zero position, (See "Displays" section). Its default is zero.

68 GRAPHICS Z OFFSET

Reserved for future use.

69 DPRNT LEADING SPACES

This is an on/off setting. It suppresses leading spaces that are generated by a macro DPRNT format statement. In a DPRNT statement the format specifies the number of characters printed to the serial port for the whole portion of a variable. If the number is smaller than the space allowed for, then leading spaces are sent to the serial port. When this setting is OFF, then no leading spaces are generated. The following example illustrates control behavior when this setting is OFF or ON.

```
#1= 3.0 ;
G0 G90 X#1 ;
DPRNT[X#1[44]] ;
```

```
Setting 69: OFF      ON
OUTPUT: X3.0000    X 3.0000
```

The default value is OFF.

70 DPRNT OPEN/CLOS DCODE

This is an on/off setting. It controls whether the POPEN and PCLOS statements in macros send DC control codes to the serial port. When the setting is ON, these statements will send DC control codes. When it is OFF, the control codes are suppressed. Its default value is ON.

71 DEFAULT G51 SCALING

Specifies the scaling for a G51 command when the P address is not contained in the same block. It must be in the range of .001 to 8380.000. This settings default is 1.000.

72 DEFAULT G68 ROTATION

Specifies the rotation, in degrees, for a G68 command when the R address is not contained in the same block. It must be in the range of 0.0000 to 360.0000. This settings default is 0.0000.

73 G68 INCREMENTAL ANGLE

This is an on/off setting. It is a switch that allows the internal variable that controls rotation to be incremented for each call to a G68 command. When this switch is ON, and a G68 command is executed in the absolute mode (G91), then the value specified in the R address is added to the internal variable. Otherwise, the internal variable is set to the rotation value specified by R. The default setting is ON.

74 9XXXX PROGS TRACE

This setting, along with Setting 75, is useful for debugging CNC programs. When Setting 74 is set to ON, the control will display all blocks that are executed in programs that have an 0 number of 9000 or above. When the setting is OFF, the control will not display 9000 series blocks. The default setting is ON.

75 9XXX PROGS SINGLE BLK

When Setting 75 is set to ON and the control is operating in SINGLE BLOCK mode, then the control will stop at each block in a 9000 series program and wait for the operator to press CYCLE START. When Setting 75 is set to OFF, then all blocks in a 9000 series program are executed in a continuous manner even if SINGLE BLOCK is ON. The default setting is ON.

When Setting 74 and Setting 75 are both ON, the control acts normally. That is, all blocks executed are highlighted and displayed and when in single block mode there is a pause before each block is executed.

When Setting 74 and Setting 75 are both OFF, the control will execute 9000 series subroutines without displaying the blocks contained in that subroutine. If the control is in single block mode, no single block pause will occur within a 9000 series subroutine.

When Setting 75 is ON and Setting 74 is OFF, then 9000 series subroutines will be displayed as they are executed.

76 TOOL RELEASE LOCK OUT

When this setting is set to ON, the tool release key is disabled. The default setting is OFF.

77 SCALE INTEGER F

This setting aids those wishing to run programs developed on a control other than HAAS. It allows the operator to select how the control interprets an F address code that does not contain a decimal point, (It is recommended that the programmer always use a decimal point). The setting can be set to the following values:

DEFAULT	-	F12 is interpreted as	.0012	units/minute.
INTEGER	-	"	"	" 12.0 " "

.1	-	"	"	1.2	"	"
.01	-	"	"	.12	"	"
.001	-	"	"	.012	"	"
.0001	-	"	"	.0012	"	"

The default setting is DEFAULT.

78 FIFTH-AXIS ENABLE

When this setting is off, the fifth axis is disabled and no commands can be sent to that axis. When off is not selected, the selected rotary table type parameters are called up. A change to rotary parameters is saved under the selected table type for later recall. In order to change this setting the servos must be turned off.

79 FIFTH-AXIS DIAMETER

This is a numeric entry. It is used to set the angular feed rate of the B-axis. It must be in the range of 0.0 to 29.9999 inches. Since the feed rate specified in a program is always inches per minute (or mm per minute), the control must know the diameter of the part being worked in the B-axis in order to compute the angular feed rate. When this setting is set correctly, the surface feed rate on a rotary cut will be exactly the feed rate programmed into the control. The feed rate will be correct only as long as the axis remains orthogonal (at right angles) to the Y axis.

80 MIRROR IMAGE B-AXIS

This is an On/Off setting. When it is off, axes motions occur normally. When it is on, the B axis motion is mirrored (or reversed) around the work zero point.

81 TOOL AT POWER DOWN

When the POWER DOWN key is pressed, the control will change to the tool specified in this setting. If zero (0) is specified, no tool change occurs at power down. 1 is the default.

82 LANGUAGE

This setting allows the user to change between available languages. If the language selected does not reside in the control, NOT AVAILABLE will be displayed in the message area when that language is selected.

83 M30 /RESET OVERRIDES

When set, causes all three overridable machine controls to be returned or reset to their default values.

84 TOOL OVERLOAD ACTION

Causes the specified action to occur anytime a tool becomes overloaded.

85 IN POSITION ACCURACY

Defines the accuracy of corners within a selected tolerance. Initial default value is set to .1 inch. If this setting is zero, the control acts as if exact stop is commanded on each motion block.

86 M39 LOCKOUT

This is an on/off setting. Locks out the rotation of the tool changer.

87 M06 RESETS OVERRIDE

This is an on/off setting. When M06 is executed and this setting is on, any overrides are canceled and set to their programmed values.

88 RESET RESETS OVERRIDES

When the reset key is pressed and this is on, any overrides are cancelled and set to their programmed values.

103 CYC START/FH SAME KEY

This is an ON/OFF setting. When it is OFF, the machine operates normally. When it is ON, CYCLE START must be pressed and held to run a program. When CYCLE START is released, a FEED HOLD is generated.

This setting cannot be ON while Setting 104 is ON. When one of them is set to ON, the other will automatically turn OFF. This setting can be changed while running a program.

104 JOG HANDL TO SNGL BLK

This is an ON/OFF setting. When it is OFF, the machine operates normally. When it is ON, and SINGLE BLOCK is selected, the jog handle can be used to single step through a program.

Reversing the jog handle direction will generate a FEED HOLD. This can be useful when an unexpected long motion block is encountered.

CYCLE START must be used to begin running a program.

This setting cannot be ON while Setting 103 is ON. When one of them is set to ON, the other will automatically turn OFF. This setting can be changed while running a program.

10. SETTING UP A FOURTH AXIS

Installing a fourth axis in this machine requires that the servo control parameters be set for the type of fourth axis that you have. Any of the HAAS servo rotary products may be used but there are different parameter settings for each. If the fourth axis is purchased with the VF Series, the parameters will be set correctly for that indexer. If the indexer is purchased at a later time, select Setting 30 and push the right arrow cursor until the model of the fourth axis is shown. Then, push the WRITE button.

Once the parameters have been entered, be sure to turn parameter lock back on (Setting 7). The fourth axis can be disabled and re-enabled easily from the Setting 30. **NEVER CONNECT OR DISCONNECT THE FOURTH AXIS CABLE WHILE THE CONTROLLER IS POWERED ON.** When turning the control on after connecting the fourth axis, be sure to enable it from Setting 30 **BEFORE** doing an AUTO POWER UP or AUTO ALL AXES. When turning the control on after disconnecting the fourth axis, be sure to turn off Setting 30 **BEFORE** doing an AUTO POWER UP or AUTO ALL AXES.

Setting 34 is used to specify the diameter of the part being turned by the rotary axis. This parameter must be set close to the correct value in order to get the correct surface feed rate when an A-axis motion is involved in a cut.

If the fourth axis is being set up for the first time, you will not be able to enable the fourth axis from settings if Parameter 43 is set to disable the fourth axis. Make sure Parameter 43 is set correctly.

The tilting rotary table (TRT) is a little more complicated in its setup. One of the two axes may be the A-axis that can be controlled in simultaneous interpolated moves but the other axis must be setup as an auxiliary (B) axis and can only be used for positioning or single axis feeds; see "Auxiliary Axis Control" section. Depending on which axis may be used in cutting, you can exchange these two.

The following table summarizes the fourth axis options:

Product:	Max speed: (deg/sec)	Max torque: (ft - lb)	Comments:
HA5C	270	30	Quick and easy to use for light loads only.
HA5C2, HA5C3 HA5C4	200	30	HA5C2 fits along Y and X axes; HA5C3, HA5C4 only along X axis.
RT7*	80	100	Versatile table with air brake.
RT9*	80	125	For larger O.D. parts.
RT11*	60	150	Very difficult to fit into VF-0/1/2; okay for VF-3/VF-4.
SRT*	60	180	Difficult to fit in VF-0/1/2 (okay for VF-3 and VF-4), but includes air collet closer and brake.
TRT 160	60/80 (A/B)	60/100 (A/B)	Convenient compact size fits VF-1,2.
TRT 210	50/60 (A/B)	150/160 (A/B)	Fits VF-3 and VF-4 with some restrictions in tool length.
HRT160	80	100	Available with 5" spindle nose. For light loads.
HRT 210	60	160	Available with 6" spindle nose and pneumatic drawbar.

HRT 310	50	150	Large capacity. Well-suited to VF-3 and 4.
HRT 450	50	170	For very large or heavy parts. Large through-hole. Ideal for VF-6,8.

* No longer in production.

11. AUXILIARY AXIS CONTROL

Besides the five directly controlled axes in this control, up to four additional external positioning axes may be added. These axes may be commanded directly from the program using the **C**, **U**, **V**, and **W** axis codes. Commands to these axes are only allowed in a G00 or G01 block. Connection of these axes is done through the second RS-232 port to one or more HAAS single axis controls. In the Setting page, Setting 38 is used to select the number of auxiliary axes. The machine position display will show the present position of these axes.

Only one auxiliary axis is moved at a time. If a feed (G01) is programmed, the feed rate programmed in the CNC is sent to the auxiliary control without any changes. For a C-axis feed at F30.0, this means that the C-axis will move at 30 degrees per second. A G00 motion of an auxiliary axis will move at its maximum feed rate.

The FEED HOLD and RESET buttons will not stop an auxiliary axis. EMERGENCY STOP and SINGLE BLOCK will stop an auxiliary axis. When the CNC control is waiting for an auxiliary axis motion to complete, the bottom of the screen will display "C FIN". A failure in RS-232 communication with the auxiliary axes may cause this display to pause indefinitely. The RESET button will terminate any "hung-up" auxiliary axis communication.

If you add one auxiliary axis, setting number 38, then the auxiliary axis will be designated **C**. If you add two auxiliary axes they will be designated **C** and **U**; etc.

When interfacing to an auxiliary axis the Haas single axis servo control must have Parameter 21 set according to the following table.

Name in CNC:	Parameter 21:	Axis select
C	6	Z
U	1	U
V	2	V
W	3	W

Multiple auxiliary axes must be daisy chained through the second RS-232 port as described in the single axis control manual.

Auxiliary axes can be jogged from the CNC front panel using the jog handle. Each step must start and stop before another one can be commanded.

There are no work offsets for these axes so all commands are in the machine coordinate system. But if a displaced zero position has been entered into the HAAS servo control, that position will be used as zero. On power-up of the CNC, the auxiliary axes control will also be initialized and zero will be shifted by the value set into the single axis control. To set a displaced zero, you must jog the single axis control to a new zero position and then press and hold the CLEAR key on the single axis control. This must only be done when the single axis control is otherwise idle.

Auxiliary axes communication is always seven data bits, even parity, two stop bits. The data rate is CNC Setting 54 and should be set to 4800. CNC Setting 50 must be set to XON/XOFF. Parameter 26 in the single axis control must be set to 5 for 4800 bit per second and Parameter 33 must be set to 1 for XON/XOFF. Parameter 12 in the single axis control should always be set to 3 or 4 to prevent circular wraparound.

The cable connecting the CNC to the single-axis control must be a DB-25 cable (male lead on both ends) and must wire at least pins 1, 2, 3, and 7 directly from the second (lower) serial port of the CNC to the upper connector of the servo control.

12. FIFTH AXIS CONTROL

This section describes the features of a HAAS five-axis control.

12.1 THE COORDINATE SYSTEM

The layout of the **A** and **B** axes on the HAAS five-axis control are depicted below. The **A** axis is rotary motion about the **X** axis, while the **B** axis determines rotary motion about the **Y** axis. The right hand rule can be used to determine axis rotation for the **A** and **B** axes. When placing the thumb of the right hand along the positive **X** axis, the fingers of the right hand will point in the direction of tool movement for a positive **A** axis command. Likewise, when placing the thumb of the right hand along the positive **Y** axis, the fingers of the right hand will point in the direction of tool movement for a positive **B** axis command. It is important to remember that the right hand rule determines direction of tool movement and not the table movement direction. For the right hand rule, the fingers will point opposite of the positive rotary table movement. Refer to Figures 12-1 and 12-2.

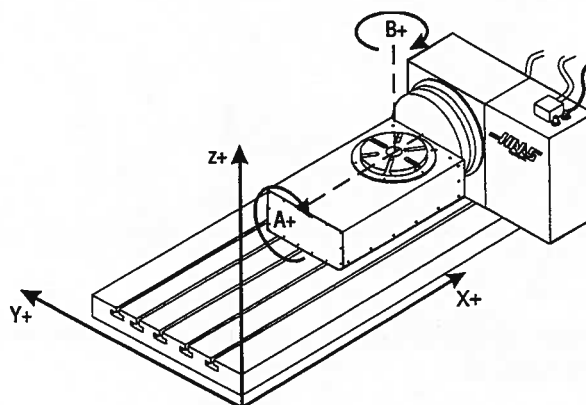


Fig. 12-1 Work coordinates (Positive direction).

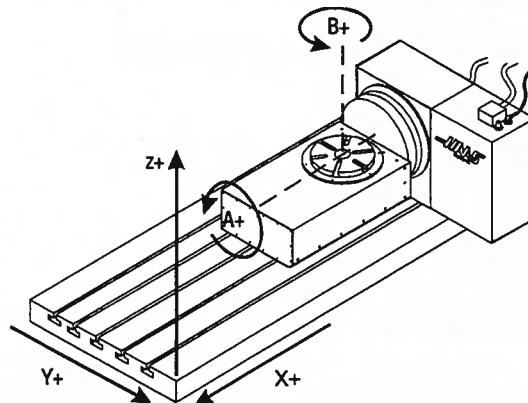


Fig. 12-2 Table movement (Positive command).

NOTE: Figures 12-1 and 12-2 represent one of many possible machine tool and table configurations. You may have different table movements for positive directions, depending on the equipment, parameter settings, or five-axis software being used.

12.2 ALPHABETICAL ADDRESS CODES

B Fifth axis rotary motion

The **B** address character is used to specify motion for the optional fifth, **B**, axis. It specifies an angle in degrees for the rotary axis. It is always followed by a signed number and up to three fractional decimal positions. If no decimal point is entered, the last digit is assumed to be 1/1000 degrees. The smallest magnitude is 0.001 degrees, the most negative value is -8380.000 degrees, and the largest number is 8380.000 degrees.

12.3 PREPARATORY FUNCTIONS (G CODES)

In a control configured with a fifth axis, all G codes that have an option for an **A** axis motion command can also simultaneously command fifth axis, **B**, motion. Of course, since address **B** is modal, it can be entered on any line. The following G codes are users of address **B**:

G00	G03	G29	G73	G77	G83	G86	G89	G101
G01	G10	G31	G74	G81	G84	G87	G92	G102
G02	G28	G36	G76	G82	G85	G88	G100	G136

12.4 INCH/METRIC SELECTION (G20,G21)

Fifth-axis programming is not effected by the selection of metric. It is always programmed in degrees.

12.5 WORK COORDINATE SYSTEMS

The work coordinate systems on a control with a fifth axis have all been expanded to accommodate **B**, the fifth axis. Work coordinate offsets can be set for the **B** axis in the offset display.

12.6 M CODE SUMMARY

Two M codes have been added to support a brake for the fifth axis.

M12	Apply fifth axis brake
M13	Release fifth axis brake

M CODE DETAILED DESCRIPTIONS

M12 Engages the fifth axis brake.

This M code waits until the M-fin signal is received through the secondary RS-232 port.

M13 Releases the fifth axis brake.

This code activates the fifth axis brake relay and waits for the M-fin signal, releases the relay upon receipt, and waits for the M-fin signal to cease. The RESET key will terminate any operation that is hung-up waiting for M-fin.

Both M12 and M13 are aliased by M10 and M11 on fourth-axis machines.

12.7 HANDLE/JOG

All aspects of handle jogging for the fifth axis work as they do for the other axes. The exception is the method of selecting jog between axis A and axis B.

By default the '+A' and '-A' keys, when pressed, will select the A axis for jogging. The display will show "JOGGING A AXIS HANDLE .01" while you are jogging the A axis. The B axis can be selected for jogging by pressing the 'shift' key, and then pressing either the '+A' or '-A' key. When this is done the control will switch to jogging the B axis and the display will change to "JOGGING B AXIS HANDLE .01".

The axis assigned to the '+A' and '-A' keys will remain selected for jogging even if the operating mode is changed or if the machine is turned off. The selected axis for '+A' and '-A' can be toggled by pressing the shift key prior to pressing the '+A' or '-A' keys.

12.8 SETTINGS

The fifth axis, or rotary table, may be disabled from the setting page and removed from the machine. Do not connect or disconnect any cables with the control on. If you do not disable the fifth axis when it is disconnected, you will get an alarm.

The settings listed here are settings that have been added to a fifth-axis control.

Number:	Name:	Range of value:
78	FIFTH-AXIS ENABLE	OFF, S5C, 7RT, 9RT, SRT, 11RT, HA5C, HRT 160, HRT 210, HRT 310
79	FIFTH-AXIS DIAMETER	0 to 29.9999 inches
80	MIRROR IMAGE B AXIS	ON or OFF

The following is a detailed description of each of the settings.

78 FIFTH-AXIS ENABLE

This is an on/off setting. When it is off, the fifth axis is disabled and no commands can be sent to that axis. When it is on, the selected rotary table type parameters are called up. A change to rotary parameters is saved under the selected table type for later recall.

79 FIFTH-AXIS DIAMETER

This is a numeric entry. It is used to set the angular feed rate of the **B**-axis. It must be in the range of 0.0 to 29.9999 inches. Since the feed rate specified in a program is always inches per minute (or mm per minute), the control must know the diameter of the part being worked in the **B**-axis in order to compute the angular feed rate. When this setting is set correctly, the surface feed rate on a rotary cut will be exactly the feed rate programmed into the control. The feed rate will be correct only as long as the axis remains orthogonal (at right angles to) to the **Y** axis.

80 MIRROR IMAGE B-AXIS

This is an On/Off settings. When it is off, axes motions occur normally. When it is on, the **B** axis motion is mirrored (or reversed) around the work zero point.

12.9 TOOL OFFSETS

Work offsets can be specified for the fifth axis **B** address.

The work offset display is found on the offset display page. If the fifth axis is enabled, setting 78, then additional data fields are made available for the fifth axis.

The function of the offset display page has been modified slightly to accommodate multiple axes. Only the axes that are enabled are displayed on the work offset display. For example, if the fourth axis is enabled, setting 30, and the fifth axis is not, then display will show X,Y,Z,A. In this case, all axes can be displayed on the entire display. If only the fifth axis is enabled, then the display will show X,Y,Z,B. Since the fourth axis is disabled, there is no need to display it.

If both **A** and **B** axes are enabled, then the last column of the offsets display serves a dual purpose. Either **A** or **B** can be accessed in the last column. By using the left or right arrow keys, the screen cursor can be moved into the axis field that is to be set. When the right most field is highlighted by the cursor and the top of the column indicates **A**, then the values in this column represent **A** axis offsets. If you press the right arrow key once, the cursor stays in the same place, but the column and its values will change to **B** axis parameters. You can view and modify work offsets in this manner using the left and right arrow keys when both axes are enabled.

12.10 PARAMETER DISPLAY

The Parameter Display organization has been modified to accommodate the extra axis parameters. Parameter numbers have remained the same as in a four-axis control. Additional parameters have been added for the fifth.

The parameters have been reorganized, so that logically-associated parameters are grouped together. These logical groupings are placed together into contiguous screens called pages. The most commonly changed parameters have been placed at the beginning of the page list. A list of the parameter pages and the order

of succession in the control are given below.

PAGE TITLE	DATA DESCRIPTION
COMMON SWITCH	Non-axis bit switches.
COMMON PAGE1	First page of non-axis parameters.
COMMON PAGE2	Second page of non-axis parameters.
COMMON PAGE3	Third page of non-axis parameters.
MACRO M CALL	Parameters that alias M codes to subroutines.
MACRO G CALL	Parameters that alias G codes to macros.
X BIT SWITCH	Bit switches for the X axis.
X PARAMETERA	First page of X axis parameters.
X PARAMETERB	Second page of X axis parameters.
Y BIT SWITCH	Bit switches for the Y axis.
Y PARAMETERA	First page of Y axis parameters.
Y PARAMETERB	Second page of Y axis parameters.
Z BIT SWITCH	Bit switches for the Z axis.
Z PARAMETERA	First page of Z axis parameters.
Z PARAMETERB	Second page of Z axis parameters.
A BIT SWITCH	Bit switches for the A axis.
A PARAMETERA	First page of A axis parameters.
B BIT SWITCH	Bit switches for the B axis.
B PARAMETERA	First page of B axis parameters.
B PARAMETERB	Second page of B axis parameters.
X SCREW COMP	X axis screw compensation value.
Y SCREW COMP	Y axis screw compensation value.
Z SCREW COMP	Z axis screw compensation value.

The HOME key displays the first parameter page "COMMON SWITCH". Pressing the PAGE DOWN key will display the next page of parameters in the above list. The END key displays the last parameter page "B PARAMETERB". Pressing the PAGE UP key will display the preceding page of parameters in the above list. All other features on the parameters display have remained the same. So, if you are unfamiliar with the new format of the parameters, you can still search by parameter number. Enter the number of the parameter you want to see or view and press the up or down arrow key. The page that the parameter is on will be displayed and the parameter being searched for will be highlighted.

12.11 DIAGNOSTIC DATA DISPLAY

A five-axis control has additional diagnostic data to be aware of.

The first page of diagnostic data has two discrete outputs that control the rotary axes brakes; they are labeled "4TH BK" and "5TH BK". When motion is commanded to either the A or B axis, the brake for that axis must first be released. To release a brake on the HAAS control, a relay is activated. These two outputs represent the activation of the brake relays. If the A or B axis brakes are disengaged, then these outputs will read high (logical 1). Normally these bits will read low (logical 0).

The second page of diagnostic data show the status of inputs from the motor interface board. Additional inputs for the expanded motor interface board are listed under "INPUTS4". These bits are monitored by the control to determine if the interface for the B axis is working correctly. Refer to the "Technical Reference" section for a description of the diagnostic page inputs.

12.12 PARAMETERS

Additional parameters specifying the fifth axis have been added to the parameter list. B axis parameter numbers range from 151 through 175. The parameter descriptions and their number assignments are listed below in the order that they are arranged on the B parameter pages. Refer to the Parameters List in the "Operation" section of this manual for more information.

PARAMETERS LISTED HERE IN ORDER OF APPEARANCE ON DISPLAY PAGES.

B BIT SWITCH

Parameter 151 B SWITCHES

B PARAMETERA

Parameter 152	B	P GAIN
Parameter 153	B	D GAIN
Parameter 154	B	I GAIN
Parameter 155	B	RATIO (STEPS/INCH)
Parameter 156	B	MAX TRAVEL (STEPS)
Parameter 157	B	ACCELERATION
Parameter 158	B	MAX SPEED
Parameter 159	B	MAX ERROR
Parameter 160	B	FUSE LEVEL
Parameter 161	B	BACK EMF
Parameter 162	B	STEPS/REVOLUTION
Parameter 163	B	BACKLASH
Parameter 164	B	DEAD ZONE
Parameter 165	B	IN POSITION LIMIT B
Parameter 166	B	HOLDING LIMIT B
Parameter 167	B	D*D GAIN FOR B

B PARAMETERB

Parameter 168	B	ACC/DEC T CONST
Parameter 169	B	BEMF BIAS
Parameter 170	B	GRID OFFSET
Parameter 171	B	EXACT STOP DIST.
Parameter 172	B	FRICTION FACTOR
Parameter 173	B	ACCEL FEED FORWARD
Parameter 174	B	SPINDLE TEMP. COEF.
Parameter 175	B	AIR BRAKE DELAY
Parameter 213	B	TOOL CHANGE OFFSET

12.13 SETTING UP A FIFTH AXIS

Installing a fifth axis is the same as installing a fourth axis. Refer to the "Setting Up A Fourth Axis" section of this manual for detailed instructions on fifth axis installation.

13. AUTOMATIC ACCELERATION/DECELERATION

This machine is not capable of instantly changing speed; it takes some nonzero time to accelerate and decelerate. Acceleration and deceleration in this machine have both a constant accel/decel mode and an exponential mode. Constant acceleration is used at the beginning of a rapid move and at the end of any move whose speed exceeds the exponential accel/decel time constant.

13.1 CONSTANT ACCELERATION

Constant acceleration is a type of motion when the amount of speed change over time is constant. This constant is set by Parameters 7, 21, 35, and 49. It has units of encoder increments per second per second.

Constant acceleration applies to the beginning of a rapid move so that the minimum time is spent getting up to rapid speed. It also applies to the end of rapid moves until the speed drops below the exponential accel/decel time constant. That change occurs at about 100 inches per minute in the following example.

13.2 EXPONENTIAL ACCELERATION

Exponential acceleration/deceleration is a type of motion where the speed is proportional to the distance remaining in a programmed travel. The exponential accel/decel time constant is set by Parameters 113, 114, 115, and 116. It has units of 0.0001 seconds. The speed limit at which exponential accel/decel is not available is defined by the relationship between Parameters 7 and 113 (for the X-axis). Thus, if Parameter 7 is 8000000 steps/sec/sec and Parameter 113 is 375 (0.0375 seconds); the maximum velocity for exponential accel/decel would be:

$$\begin{aligned} 8000000 \times 0.0375 &= 300000 \text{ steps/second} \\ \text{For a 8192 line encoder and 6 mm screw, this would be:} \\ 60 \times 300000 / 138718 &= 129.7 \text{ inches/minute} \end{aligned}$$

13.3 ACCELERATION IN FEED MOTIONS

In the normal feed cutting mode, with G64 active, giving continuous cutter motion, deceleration of the axes in motion begins at some distance away from the end point. If lookahead has buffered another motion, the acceleration for that motion will begin at the same instant. This means that two motions, at right angles to each other, will not produce a perfectly square corner. The corner will be rounded. In addition, two motions which smoothly blend one into the other will not cause the tool to pause.

If you use cutter compensation to cut an outside corner, there will be no rounding if the cutter compensation amount is close to the actual tool size. This is because the tool is moved beyond the end of the first programmed stroke before it is moved to the beginning of the second stroke. Note that in this machine, using the default parameter settings, rapid and feed moves will both be blended to provide continuous cutter path and rounded corners. Unless you specify exact stop, the following rapid or feed block will be started slightly before the completion of the previous block.

The end of a feed move is delayed until the following error is below an amount set in Setting 85. If Setting 85 is set to 0.1, with Parameter 113 set to 375 (0.0375 seconds), this means that the highest feed rate which will give continuous cutter motion is:

$$(0.1) \times 60 / 0.0375 = 160 \text{ inches per minute}$$

13.4 ACCELERATION IN RAPID MOVES

Rapid moves have a slightly different operation when continuous cutter mode is active. Acceleration for the next motion is started when the axes being moved are all within the "In Position Limit" Parameters 101, 102, 103, and 104. These parameters have units of encoder steps. Rapid moves will also decelerate at the constant accel/decel limit until the speed drops below that for exponential accel/decel. An example of the "In Position Limit" values follows. If Parameter 101 (for X) is 8000 and Parameter 5 is 138718, a rapid move of will proceed to the next block when the X-axis is within a distance of:

$$8000 / 138718 = 0.0577 \text{ inches}$$

To prevent the rounding of corners, you can specify exact stop either with G09 (non-modal) or with G61 (modal). When either of these is active in a motion, all of the axes are brought to an exact stop, at zero speed, before the next motion is started.

Note that in this machine, using the default parameter settings, rapid and feed moves will both be blended to provide continuous cutter path and rounded corners. Unless you specify exact stop, the following rapid or feed block will be started slightly before the completion of the previous block.

13.5 ACCELERATION/DECELERATION IN CIRCULAR MOVES

The tool path in a circular move (G02 or G03) is not changed by the exponential acceleration/deceleration so that there is no error introduced in the radius of the cut unless the speed exceeds that for exponential accel/decel (see example above giving 100 inches per minute). However, the actual radius of a circular move will always be slightly smaller than the programmed value. The amount of change can be computed by the following equation:

$$Ra = \text{SQRT}(R * R - L * L)$$

Where Ra is the actual radius,
R is the programmed radius, and
L is the accel/decel lag in feed motion.

The lag amount is computed by:

$$L = (\text{Par. 113}) * (\text{feed in/min}) / 600000$$

As an example; if Par 113 is 375 (0.0375 sec) and the feed is 30 inches per minute and the programmed diameter is two inches, the actual radius will be:

$$L = 375 * 30 / 600000 = 0.0187 \text{ inches}$$

and

$$Ra = \text{SQRT}(1 - 0.000351) = 0.999824$$

or an error of 176 millionth's of an inch. This is an upper bound on the accuracy of this cut and many other factors could contribute additional errors.

13.6 FANUC 6M, 10M, AND 15M COMPATIBILITY

Parameter 57 may be used to change the rapid accel/decel mode to one closer to that of the 10M and 15M controls. This is done with the flag called "EX ST MD CHG". This means "exact stop in mode change" and, if this flag is set to 1, will cause an exact stop at both the beginning and end of any rapid move. Thus continuous cutter motion is provided only for a feed motion followed by another feed motion. When this flag is set, the exact stop codes G09 and G61 will still provide an exact stop between two feed motions.

Setting 33 controls how the G52 and G92 codes work. These are different between Fanuc class controls and Yasnac class controls. To operate like a Fanuc control, Setting 33 should be set to FANUC.

Setting 58 controls how cutter compensation goes around outside corners. This motion is different between Fanuc class controls and Yasnac class controls. To operate like a Fanuc control, Setting 58 should be set to FANUC.

NOTE: The Haas CNC Control is compatible with many other controls; it is not identical in performance to any single control.

14. HIGH SPEED MACHINING

High speed machining with the Haas CNC is possible up to 300 inches per minute and up to 1000 blocks per second. 500 blocks per second is standard and 1000 blocks per second is a purchased option. No special programming or setup is required to get these speeds.

There are a few points to consider at high feed rates.

- 1) At higher feed rates interpolation accuracy will be reduced.
- 2) Above 150 inches per minute in position limit may cause pauses between blocks (1995 or earlier controls)
- 3) The higher the resolution you specify on your CAD/CAM the longer the program and the slower the speed
- 4) When using DNC you must have a very fast computer and excellent DNC software

In most CNC controls you have to set the maximum feedrate according to the most accurate detail you were machining. Just as in a car you cannot make a right turn at 100 MPH so to the CNC control must slow down in the corners in order to give the accuracy you require. Unfortunately the corners often dictate the maximum feed rate for the entire program when that may not be desired. The HAAS control uses a special function called Contouring Accuracy (G187) which allows you to specify the required accuracy. If the programmed feedrate is above what the control can accurately machine the control will automatically slow the feedrate to obtain the required accuracy. In straight line moves the feed rate will be at the programmed rate. See Contouring accuracy in the next section.

When operating at feed rates above 150 inches per minute the following error of the servos may exceed the maximum error allowed (in position limit) and the machine will pause until it "catches up". The machine will not execute the next block until the servos have reduced the error to the parameter value. This will manifest itself by jerky motions while operating. This default parameter value corresponds to 0.06 inches. If you operate faster than 150 inches per minute and the strokes are well blended (no sharp corners), these parameters (101, 102, 103,104) can be changed to a larger value. The default value is 2000 for brush and 8000 for brushless. This does not apply controls made in 1996 or later.

When machining a sculptured mold you obviously would like to have the smoothest surface possible. The higher the resolution you select (the distance between machined points) the more accurate the final part. The downside to this is that program length is longer and feedrates slower. Beware of the relationship between stroke length , feed rate, and block per second. If you have 1000 blocks per second and each block is 0.001 inches in length, this is 1 inch per second and 60 inches per minute. Trying to program more than 60 inches per minute in this case will not go any faster; you cannot get more than 1000 blocks per second. Similarly a resolution of .0001 will limit your feedrate to 6 IPM.

The best way to machine a part is from CNC memory. The HAAS control can be purchased with up to 8 megabytes of ram. This eliminates any communications problems and offers the highest blocks per second possible. If you chose to run the HAAS control directly from your computer you need to have need make sure that your computer can send data fast enough. A general purpose Windows based communication program will not work reliably. The serial port and wiring from your computer must be of the highest quality. Beware, that when running DNC, computer serial ports are not able to send data to the CNC fast enough to get the blocks per second rate which you require. The fastest serial rate of this control is 38400 bits per second (115200 via RS432 in 1996) This translates to 3840 characters per second. If a typical block consists of about 20 characters, it is not possible to run DNC at more than 190 blocks per second. If you are not experienced with high-speed DNC get expert help from your dealer or have him recommend someone local with hands on experience.

In order to get maximum performance from large memory programs the following hints will help.

- 1) Eliminate all unnecessary codes such as line numbers, comments, redundant G codes, etc.
- 2) Eliminate all leading and trailing zeros, i.e. make X002.0000 to X2.
- 3) Incremental (G91) programming utilizes the least data i.e., (G91 X1 Y1 Vs G90 X2.0001 Y2.0001)

14.1 CONTOURING ACCURACY

When milling a part it is desirable to machine it at the fastest feedrate possible and yet obtain the highest accuracy. No machine can instantly accelerate or decelerate and this leads to positioning errors when axis directions change. The higher the feedrate and the sharper the turn the more pronounced the effect. A high feedrate selected for straight line milling will cause corner rounding at intersections when you actually want a sharp corner. Similarly a high feed milling cut blending into a radius will cause the radius to be smaller than programmed because of servo lag. To alleviate this the HAAS control uses a special function called Contouring Accuracy (G187) so the operator can select the required accuracy.

When using this function the control will move up to the programmed feedrate in straight line moves and slow down at intersections or radii to obtain the required accuracy you want. This accuracy is measured in true three dimensional motion. It uses setting 85 to define a default value and G187 to program a new value directly from your program. The amount of slow down depends on the accuracy specified and on how well one stroke blends into the next. If two strokes blend into each other exactly (in one line), there is never any slow down.

Programming G187 is as follows:

```
G187 E0.01      (to set value)
G187            (to revert to setting 85 value)
```

The first line will set the required accuracy to 0.01 inches. G187 must be programmed on a line by itself. If there is no E code, the accuracy reverts to setting 85. If MM programming is active, the units are millimeters. The range of values possible are 0.0001 to 0.25 inches and 0.001 to 2.5 mm.

The most important thing to remember with Contouring Accuracy is that normal, well blended strokes should not get a slowdown in feedrate. Only the sharp corners need this in order to achieve the programmed accuracy. However, if you set the accuracy to extremely small values (0.0005 and smaller) you may effect such a slow feedrate that the machine appears to pause when you don't think it should. Even well blended strokes can have a slight error.

When roughing out a pocket the default setting (85) should be used for maximum speed then on the final clean up passes you can specify a higher accuracy. A simple test part of milling a rectangle and varying the values for Contouring Accuracy will demonstrate the principles outlined above.

15. TAPPING WITH THE VF SERIES CNC MILL

Making tapped holes with the VF Series CNC Mill can be made with several devices. Threads may be generated with a tap held in a rigid tool holder (called rigid tapping), a floating tap holder, a reversing tapping head, or helical thread milling. Each method has distinct advantages.

Tapping is done using canned cycles. You must select the tapping RPM and, using the pitch (threads per inch), calculate the feed rate that is entered in the F command. The HELP/CALC page will compute these numbers for you.

15.1 RIGID TAPPING

Rigid Tapping eliminates the cost of special tap holders since taps can be held in drill collet holders. The spindle is accurately synchronized with the Z-axis feed, thereby producing threads as accurately as a lead screw tapper. No side forces are generated on the flanks of the threads and tighter thread tolerances are produced. Rigid tapping also eliminates the pullout and distortion of the first thread that occurs on all spring compression/tension devices and tapping heads. While this is not usually a problem on medium to coarse threads, small diameter, fine pitch or soft material tapped holes can have their last thread damaged when the tap pops out of the hole. You can also re-tap a hole without cross-threading provided the tap and Z depths have not been changed. Rigid tapping is used with canned cycles G74 and G84 and is an extra cost factory installed option. Example:

```
N100 G84 Z-1. R.3 F37.5 (for a 20 pitch thread at 750 RPM)
```

A word of caution on Rigid Tapping: As the term implies, the tap is rigidly held in place. This requires that runout be less than .001 TIR or the tap will generate an oversized thread. This problem can be minimized by using a small diameter drill extension to provide flex, a radial floating tool holder, or specially-designed chucks for holding taps, because tap shanks are not common collet sizes.

15.2 USING FLOATING TAP HOLDERS

Floating tap holders are probably the most common method of tapping holes. The tap is held in a quick change holder that can float up and down slightly. This is done to allow the tap to follow the hole it is tapping and compensate for differences in the accel and decel of the spindle versus the feed of the Z-axis. Upon reaching the bottom of the hole, the feed stops and the spindle reverses, if you watch closely, you will see the tap pull the floating holder out slightly. Upon reversal, the tap will be pushed back into the holder.

If the holder is pulled out or pushed in to its mechanical limits while tapping, you can break the tap, damage the threaded part, or pull the tap completely out of the holder. Carefully watch for this condition when setting up a job because it usually becomes a problem after the job has been running a while. Also tapping of diameters less than 5/16 of an inch while below 1201 rpm (low gear shift point par 142) should be done in high gear. Spindle reversal is quicker in high gear and will minimize tap pullout. This is done by putting an M42 code with the speed command such as: M42 S900. Tapping is done with G74 and G84 cycles which automatically reverse the spindle at Z depth. The feed rate can be calculated by using the HELP display and paging down to the tapping calculator and inputting your speed and tap pitch into the control to obtain your feed rate that is then input to the F command of the cycle. Example:

```
N100 G84 Z-1.0 R.3 F46.875 (for 32 pitch tap at 1500 RPM)
```

15.3 AUTOREVERSING TAPPING HEADS

Auto reversing tapping heads eliminate the need for the spindle to reverse at bottom and provide for high production rates. The reversing function of the tapping head requires an arm to prevent the body from rotating. This must be considered when changing tools so as not to interfere with operation. The tool block on the VF Series CNC Mill will accommodate the Tapmatic series of heads. Several sizes are available and should be chosen dependent on tap size. Choose the head specifically for NC use as they have a 1:1 feed rate. Manual types have a faster withdraw rate that leads to clatter on the up stroke. A disadvantage to these types of heads is that when the inevitable crash occurs, you can destroy an expensive device.

Use the G85 or G89 (dwells at bottom) cycle when using a tapping head. Example:

```
N100 G98 G85 Z-1.0 R0.25 F46.875
```

15.4 THREAD MILLING

Thread milling uses a cutter formed with the pitch of the thread to mill the thread. The cutters are solid carbide, fragile and expensive. Some companies sell replaceable insert holders that are more economical. Internal holes smaller than 3/8 inch may not be possible or practical. It does allow for making thread diameter compensation and external threads. For large threads, port threads, blind hole threads, thread milling can be the most economical method.

Thread milling is accomplished with Helical milling. Use a standard G02 or G03 move to create the circular move in X-Y and then insert a Z move on the same block corresponding to the thread pitch. The feed rate is selected as in standard milling practice. This will generate one turn of the thread. The multiple teeth of the cutter will generate the rest. A typical line would be as follows:

```
N100 G02 I-1.0 Z-.05 F5. (generates a one inch radius for 20 pitch thread)
```

16. PROGRAMMABLE COOLANT SPIGOT

The optional programmable coolant spigot allows the user to direct the coolant stream to the most optimum location in order to flush out chips from the cutting area. The direction of the coolant can be changed dynamically by the CNC program.

16.1 OPERATING THE COOLANT SPIGOT

If the spigot has been enabled, the spigot will home itself when the AUTO POWER UP key is pressed or when the user zeros all axes. The spigot will also search for home if it ever loses position. If after 3 contiguous searches for home the spigot has not found home, then Alarm, 193 COOLANT SPIGOT FAILURE, is generated.

16.2 OFFSET DISPLAY WITH SPIGOT ENABLED

When the spigot is enabled, an additional field can be accessed on the tool offset display. The left column indicates CLNT POS for coolant position. By default this column has all zeros for the coolant position. If the position is zero, then the spigot will not be moved when the H code for that tool offset is in effect and an M08 is encountered in the program. If a value is placed into the field, then the spigot will be moved to that value if the corresponding H code is in effect and M08 is executed.

16.3 SPIGOT PROGRAM COMMANDS

There are two ways that the spigot can be moved under program control. The first, as just discussed, is by entering positional values into the CLNT POS fields on the tool offset display page. Having positional values entered into the CLNT POS field does not mean that the spigot will move. Only when the M08 command is executed and the current H code has a value in its CLNT POS field, will the spigot move to the designated position.

The second method of moving the spigot is by programming M34 or M35.

M34 moves the spigot in a positive direction. If the spigot is at the end of travel then no spigot movement occurs. For example, if the current spigot position is at 8 and M34 is executed, then the spigot will move to position 9. On a vertical mill this would lower the coolant stream.

M35 moves the spigot in a negative direction. If the spigot is at home position then no spigot movement occurs. For example, if the current spigot position is at 8 and M35 is executed, then the spigot will move to position 7. On a vertical mill this would raise the coolant stream.

It is important to note that each programming method requires the operator to specifically program the spigot in reference to the particular tool being used, while also taking into consideration the tool length, width and size of the part.

16.4 SPIGOT PARAMETERS

The parameters that control the spigot are below. Refer to the "Parameters" section for a complete description of their usage.

SPIGOT POSITIONS	Parameter	206
SPIGOT TIMEOUT (MS)	Parameter	207

16.5 M CODE SUMMARY

M34 Coolant Spigot Increment Position
M35 Coolant Spigot Decrement Position

16.6 M CODE DETAILED DESCRIPTION

M34 Increment Coolant Spigot Position

M34 increments the current spigot position one place. Incrementing the spigot position causes the spigot to advance away from the spigot home position. The home position is designated as zero. If the current spigot position is 5 and M34 is executed, then the current spigot position will advance to position 6. The Spigot home position for a horizontal mill places the spigot at the most positive Z axis location that the spigot can attain. Incrementing the spigot then lowers the coolant stream direction.

M35 Decrement Coolant Spigot Position

M35 Decrements the current spigot position one place. Decrementing the spigot position causes the spigot to move toward the spigot home position. The home position is designated as zero. If the current spigot position is 5 and M35 is executed, then the current spigot position will move to 4. The Spigot home position for a horizontal mill places the spigot at the most positive Z axis location that the spigot can attain. Decrementing the spigot will raise the coolant stream direction.

17. AUTOMATIC CHIP CONVEYOR

The automatic chip conveyor assists the user in removal of chips for jobs with heavy material removal. When running, the chip conveyor will sense conveyor motor overcurrent and reverse direction momentarily, thus attempting to free up chip jams. This procedure will be repeated until chips are cleared or conveyor retry limit (Parameter 219) is reached. If the chip conveyor is running and the door is opened, the chip conveyor will stop, thus adding a degree of safety to conveyor operation.

17.1 CONVEYOR KEYBOARD COMMANDS

The conveyor can be started at anytime from the keyboard. The conveyor can be enabled in either direction by pressing the CNVYR FWD or CNVYR REV and stopped by pressing the CONVYR STOP key. The conveyor will also stop by pressing the RESET key.

17.2 CONVEYOR PROGRAM COMMANDS

Use M codes M31, M32 and M33 to control the conveyor from within a program or in MDI. M31 commands the conveyor forward, M32 commands the conveyor in reverse and M33 stops the conveyor. Refer to the "M Codes" section for a more detailed description.

17.3 CONVEYOR PARAMETERS

The parameters that control the conveyor are below.

CNVYR RELAY DELAY	Parameter 216
CNVYR IGNORE OC TIM	Parameter 217
CNVYR RETRY REV TIM	Parameter 218
CNVYR RETRY LIMIT	Parameter 219
CNVYR RETRY TIMEOUT	Parameter 220

A complete description of conveyor parameters is given in the Parameters Section of the Users Manual.

17.4 M CODE SUMMARY

M31 Chip Conveyor Forward
M32 Chip Conveyor Backward
M33 Chip Conveyor Stop

17.5 M CODE DETAILED DESCRIPTION

M31 Chip Conveyor Forward

M31 starts the chip conveyor motor in the forward direction. The forward direction is defined as the direction that the conveyor must move to transport chips out of the work cell. If the conveyor motor is on, then the conveyor will be stopped and restarted in the forward direction.

M32 Chip Conveyor Backward

M32 starts the chip conveyor motor in the reverse direction. The reverse direction is defined as the direction opposite of forward. If the conveyor motor is on, then the conveyor will be stopped and restarted in the reverse direction.

M33 Chip Conveyor Stop

M33 Stops Conveyor motion.

18. THROUGH THE SPINDLE COOLANT (TSC)

The HAAS Through The Spindle Coolant (TSC) option includes an auxiliary coolant pump that is used to supply high pressure coolant to the cutting tool.

When TSC is commanded (M88), the air purge system is activated and the tool release piston moves down to engage the drawbar, forming a seal.

NOTE: the wear surfaces of the seal are engaged only when TSC is in use.

Air flows through the spindle for approximately 1/2 second before the auxiliary pump turns on and begins sending coolant to the spindle. A one (1) second delay is provided to allow the coolant flow to stabilize and coolant pressure to be verified by the control before proceeding.

When the coolant system is turned off (M89), the spindle is stopped, the pump is shut off, and air flows through the spindle for 2-1/2 seconds to purge leftover coolant.

18.1 OPERATION**GENERAL WARNINGS**

Proper tooling, with a through-hole, must be in place before using the TSC system. Use a "45 Degree, P40T Type 1, inch threads" pull stud built to JMTBA standard "MAS 403-1982". BT tooling may also be used with this option. Contact the tool manufacturer for further information.

The maximum spindle speed when using the TSC system is 7500 RPM.

Coolant will be used more quickly when the TSC system is in use. Make sure to keep the coolant level up and to check the level more frequently. The spindle will shut off automatically if the coolant level gets too low.

NOTE: Follow the tool manufacturer's instructions for speed and feed rate restrictions.

18.2 PROGRAM COMMANDS

Through The Spindle Coolant is controlled mainly by two M Codes, M88 and M89; and three Parameters, 235, 236, and 237 (See the "Parameters" section for a description of these). M88 is used to turn the system on, and M89 is used to turn it off. The AUX CLNT button on the control panel may also be used to control TSC. Pressing this button while in MDI mode will turn on the Through The Spindle Coolant system, and pressing it again will shut off the system.

When Through The Spindle Coolant is on, the RESET and EMERGENCY STOP buttons on the control panel perform slightly different functions. When RESET is pressed, the feed and spindle are shut off, as in a normal reset. However, with TSC on, the control also waits for the spindle to stop, turns off the TSC coolant pump, waits for [Parameter 237] for the coolant to purge, and then turns off the precharge. When EMERGENCY STOP is pressed, the feed, spindle, and pump are shut off, as in a normal E-Stop. However, with TSC turned on, the control will wait for [Parameter 237] for the coolant to purge, and then shut off the precharge.

18.3 TSC PARAMETERS

The following parameters (and bits) apply only to the Through The Spindle Coolant system:

WARNING! These are factory preset parameters. Changing them may void the warranty.

- | | |
|----------------------|----------------------------|
| Parameter 235 | TSC PISTON SEAT |
| Parameter 236 | TSC CLNT STABILIZE |
| Parameter 237 | TSC CLNT LINE PURGE |
| Parameter 209 | COMMON SWITCH 2 |

A new bit, "TSC ENABLE" was added to **Parameter 209**.

A complete description of TSC parameters is given in the "Parameters" section of this manual.

18.4 M CODE SUMMARY

The following M codes apply only to the Through The Spindle Coolant system:

- M88 Thru Spindle Clnt ON**
- M89 Thru Spindle Clnt OFF**

The following pre-existing M codes perform a slightly different function when Through the Spindle Coolant is turned ON:

- M00 Stop Prog**
- M01 Optional Stop**
- M06 Tool Change**
- M24 Option User M**
- M30 Prog End Rew**
- M54 Set User M**
- M64 Clear User M**

18.5 M CODE DETAILED DESCRIPTION

- M00 Stop Prog**
- M01 Optional Stop**

When TSC is ON, M00 and M01 will shut it OFF, as in M89.

M06 Tool Change

When TSC is ON, M06 causes the following operations to be performed:

- Orient the spindle and move Z-axis to tool change position
- Turn OFF the TSC pump
- Wait for [Parameter 237] for coolant to purge
- Perform tool change
- Turn off precharge

TSC then remains OFF until M88 is used.

- M24 Option User M**
- M54 Set User M**
- M64 Clear User M**

When TSC is enabled, these M codes are disabled.

FEATURES

M30 Prog End Rew

When TSC is ON, M30 will shut it OFF as in M89, then perform an M30 operation.

M88 Thru Spindle Clnt ON

M88 performs the following operations:

- Stop the spindle
- Activate precharge
- Wait for [Parameter 235] for piston to seat
- Turn on the TSC pump
- Wait for [Parameter 236] for coolant to stabilize
- Restart spindle

M89 Thru Spindle Clnt OFF

M89 performs the following operations:

- Stop the spindle
- Turn off the TSC pump
- Wait for [Parameter 237] for coolant to purge
- Turn off precharge

18.6 ALARM DESCRIPTION**151 Low Tool Coolant**

This alarm will shut off the spindle, feed and pump all at once. If this alarm is received, check for low coolant tank level, any filter or intake strainer clogging, or for any kinked or clogged coolant lines. Also, ensure that the TSC coolant pump turns on when TSC is commanded. If not, check the coolant pump circuit breaker.

198 Precharge Failure

This alarm is received if the precharge fails for greater than 0.1 seconds. It will shut off the feed, spindle and pump all at once. If received, check all air lines and the air supply pressure. Regulated air pressure above 85 psi may cause this alarm.

18.7 SAMPLE PROGRAM**IMPORTANT!**

Note that the M88 command appears before the spindle speed command in this program. This is a good programming practice because the M88 command will stop the spindle, start TSC, then restart the spindle, slowing the cycle time.

```

T1 M6;                               (TSC Coolant Through Drill)
G90 G54 G00 X0 Y0;
G43 H06 Z.5;
M88;                                  (Turn TSC on)
S4400 M3;
G81 Z-2.25 F44. R.03;
M89 G80;                              (Turn TSC off)
G91 G28 Z0;
M30;

```

HAAS FOURTH AND FIFTH AXIS BRUSH ROTARY PRODUCT PARAMETERS FOR REV 8 BRUSH MILL SOFTWARE

ENCODER LINES	S5C (1000)	7RT (1000)	7RT/APL (2000)	9RT (1000)	SRT (1000)	11RT (1000)	HA5C (2000)	HRT160 (2000)	HRT210 (2000)	HRT310 (2000)	HRT310H (2000)	5CN (2000)	HRT450 (2000)
REV ENCODER	1	1	1	1	1	1	1	1	1	1	1	1	1
REV POWER	0	0	0	0	0	0	0	0	0	0	0	0	0
REV PHASING	0	0	0	0	0	0	0	0	0	0	0	0	0
DISABLED	0	0	0	0	0	0	0	0	0	0	0	0	0
Z CH ONLY	1	0	0	0	0	0	0	0	0	0	0	0	0
AIR BRAKE	0	1	0	1	1	1	1	1	1	1	1	1	1
DISABLE Z T	0	0	0	0	0	0	0	0	0	0	0	0	0
SERVO HIST	0	0	0	0	0	0	0	0	0	0	0	0	0
INV HOME SW	0	1	0	1	1	1	1	1	1	1	1	1	1
INV Z CH	0	0	0	0	0	0	0	0	0	0	0	0	0
CIRC. WRAP.	0	0	0	0	0	0	0	0	0	0	0	0	0
NO I IN BRK.	0	1	0	1	1	1	1	1	1	1	1	1	1
LOW PASS+1X	1	1	1	1	1	1	1	1	1	1	1	1	1
LOW PASS+2X	0	0	0	0	0	0	0	0	0	0	0	0	0
OVER TEMP NC	1	1	1	1	1	1	1	1	1	1	1	1	1
CABLE TEST	1	1	1	1	1	1	1	1	1	1	1	1	1
Z TEST HIST	0	0	0	0	0	0	0	0	0	0	0	0	0
SCALE FACT/X	0	0	0	0	0	0	0	0	0	0	0	0	0
IRVIS AXIS	0	0	0	0	0	0	0	0	0	0	0	0	0
ROT ALM LMSW	0	0	0	0	0	0	0	0	0	0	0	0	0
ROT TRVL LIM	0	0	0	0	0	0	0	0	0	0	0	0	0
TORQUE ONLY	0	0	0	0	0	0	0	0	0	0	0	0	0
3 ERV/MREV	0	0	0	0	0	0	0	0	0	0	0	0	0
2 ERV/MREV	0	0	0	0	0	0	0	0	0	0	0	0	0
BRUSH MOTOR	0	0	0	0	0	0	0	0	0	0	0	0	0
LINEAR DISPL	0	0	1	0	0	0	0	0	0	0	0	0	0
SCALE/X LO	0	0	0	0	0	0	0	0	0	0	0	0	0
SCALE/X HI	0	0	0	0	0	0	0	0	0	0	0	0	0
P GAIN	8	8	2	8	8	8	5	4	6	6	4	6	6
D GAIN	150	200	150	200	250	250	120	100	150	150	80	150	150
I GAIN	64	64	64	64	64	64	48	64	64	64	64	48	64
RATIO (STEPS/UNIT)	800	1600	19098	2000	2000	2000	4000	2800	4000	3200	3200	4000	4800
MAX TRAVEL (STEPS)	480000	480000	827000	480000	480000	480000	960000	480000	480000	480000	480000	960000	480000
ACCELERATION	600000	480000	1377938	480000	480000	480000	1000000	672000	600000	320000	240000	960000	480000
MAX SPEED	240000	128000	373333	160000	120000	120000	360000	224000	240000	160000	120000	256000	240000
MAX ERROR	2000	2000	2000	2000	2000	2000	4000	4000	4000	4000	4000	4000	4000
FUSE LEVEL	25	30	35	30	35	35	30	35	35	35	35	35	40
BACK EMP	210	370	410	370	410	410	370	370	410	410	550	410	410
STEPS/REVOLUTION	4000	4000	8000	4000	4000	4000	8000	8000	8000	8000	8000	8000	8000
RACKLASH	0	6	6	6	6	6	20	15	15	15	15	30	15
DEAD ZONE	0	0	0	0	0	0	0	0	0	0	0	0	0
IN POSITION LIMIT	1000	1000	1000	1000	1000	1000	1000	1000	1000	1000	1000	1000	1000
HOLDING LIMIT	5	5	5	5	5	5	5	5	5	5	5	5	5
D+D GAIN	0	0	0	0	0	0	0	0	0	0	0	0	0
ACC/DEC T CONST	375	375	375	375	375	375	375	375	375	375	375	375	375
BEAP BIAS	15	15	15	15	15	15	15	15	15	15	15	15	15
GRID OFFSET	0	0	0	0	0	0	0	0	0	0	0	0	0
GRAT STOP DIST.	3	3	3	3	3	3	6	3	3	3	3	6	3
FRICITION FACTOR	0	0	0	0	0	0	0	0	0	0	0	0	0
ACCEL FEED FORWARD	125000	125000	125000	125000	125000	125000	125000	125000	125000	125000	125000	125000	125000
SPIEDLE TEMP. CORP.	0	0	0	0	0	0	0	0	0	0	0	0	0
AIR BRAKE DELAY	0	1000	1000	1000	1000	1000	100	1000	1000	1900	1900	100	1000
TOOL CHANGE OFFSET	0	0	0	0	0	0	0	0	0	0	0	0	0
LEAD COMP 10E9	0	0	0	0	0	0	0	0	0	0	0	0	0

HAAS FOURTH AND FIFTH AXIS BRUSH (EXCEPT WHERE NOTED) ROTARY PRODUCT PARAMETERS FOR REV 9 BRUSHLESS MILL SOFTWARE

SSC 7RT 9RT SRT 11RT HA5C HRT160 HRT210 HRT310 310/HS-1R HRT310HA 5CN HRT450

ENCODER LINES	(1000)	(1000)	(1000)	(1000)	(1000)	(2000)	(2000)	(2000)	(2000)	(2000)	(2000)	(2000)	(2000)
REV ENCODER	1	1	1	1	1	1	1	1	1	1	1	1	1
REV POWER	1	1	1	1	1	1	1	1	1	1	1	1	1
REV PHASING	0	0	0	0	0	0	0	0	0	0	0	0	0
DISABLED	0	0	0	0	0	0	0	0	0	0	0	0	0
Z CH ONLY	1	0	0	0	0	0	0	0	0	0	0	0	0
AIR BRAKE	0	1	1	1	1	1	1	1	1	1	1	1	1
DISABLE Z T	0	0	0	0	0	0	0	0	0	0	0	0	0
SERVO HST	0	0	0	0	0	0	0	0	0	0	0	0	0
INV HOME SW	0	1	1	1	1	1	1	1	1	1	1	1	1
INV Z CH	1	1	1	1	1	1	1	1	1	1	1	1	1
CIRC. WRAP.	0	0	0	0	0	0	0	0	0	0	0	0	0
NO I IN BRAK	0	1	1	1	1	1	1	1	1	1	1	1	1
LOW PASS+1X	1	1	1	1	1	1	1	1	1	1	1	1	1
LOW PASS+2X	0	0	0	0	0	0	0	0	0	0	0	0	0
OVER TEMP MC	1	1	1	1	1	1	1	1	1	1	1	1	1
CABLE TEST	1	1	1	1	1	1	1	1	1	1	1	1	1
Z TEST HST	0	0	0	0	0	0	0	0	0	0	0	0	0
SCALE FACT/X	0	0	0	0	0	0	0	0	0	0	0	0	0
INVIS AXIS	0	0	0	0	0	0	0	0	0	0	0	0	0
ROT ALM LMSW	0	0	0	0	0	0	0	0	0	0	0	0	0
ROT TRVL LIM	0	0	0	0	0	0	0	0	0	0	0	0	0
TORQUE ONLY	0	0	0	0	0	0	0	0	0	0	0	0	0
3 REVV/REV	0	0	0	0	0	0	0	0	0	0	0	0	0
2 REVV/REV	0	0	0	0	0	0	0	0	0	0	0	0	0
BRUSH MOTOR	1	1	1	1	1	1	1	1	1	1	1	1	1
LINEAR DISPL	0	0	0	0	0	0	0	0	0	0	0	0	0
SCALE/X LO	0	0	0	0	0	0	0	0	0	0	0	0	0
SCALE/X HI	0	0	0	0	0	0	0	0	0	0	0	0	0
P GAIN	250	250	250	250	250	96	96	128	128	128	128	128	128
D GAIN	15000	20000	20000	25000	25000	7500	7500	10000	10000	10000	10000	10000	10000
I GAIN	64	64	64	64	64	48	48	64	64	64	64	64	64
RATIO (STEPS/UNIT)	800	1600	2000	2000	1600	4000	2800	4000	3200	3200	4000	4800	4800
MAX TRAVEL (STEPS)	480000	480000	480000	480000	480000	960000	480000	480000	480000	480000	2254518	960000	480000
ACCELERATION	600000	480000	480000	480000	480000	1500000	672000	600000	320000	240000	600000	600000	480000
MAX SPEED	240000	128000	160000	120000	120000	360000	224000	240000	160000	120000	1000000	256000	240000
MAX ERROR	2000	2000	2000	2000	2000	4000	4000	4000	4000	4000	16384	4000	4000
FUSE LEVEL	300000	250000	250000	800000	800000	250000	250000	800000	800000	800000	3000000	800000	800000
BACK EMP	210	370	410	410	410	370	370	410	410	550	550	410	410
STEPS/REVOLUTION	4000	4000	4000	4000	4000	8000	8000	8000	8000	8000	32768	8000	8000
BACKLASH	0	6	6	6	6	20	15	15	15	6	48	30	15
DEAD ZONE	0	0	0	0	0	0	0	0	0	0	0	0	0
INV POSITION LIMIT	1000	1000	1000	1000	1000	1000	1000	1000	1000	1000	1000	1000	1000
MAX CURRENT	2240	1774	1774	3276	3276	1774	1774	3276	3276	3276	4095	3276	3276
D-D GAIN	-2000	-2000	-2000	-2000	-2000	-2000	-2000	-2000	-2000	-2000	-1000	-2000	-2000
ACC/DEC T CONST	375	375	375	375	375	375	375	375	375	375	375	375	375
PHASE OFFSET	0	0	0	0	0	0	0	0	0	0	0	0	0
GRID OFFSET	0	0	0	0	0	0	0	0	0	0	0	0	0
EXACT STOP DIST.	3	3	3	3	3	6	3	3	3	3	12	6	3
FRICITION COMPENSAT.	0	0	0	0	0	0	0	0	0	0	0	0	0
ACCEL FEED FORWARD	-125000	125000	125000	125000	125000	125000	125000	125000	125000	125000	500000	125000	125000
SPINDLE TEMP. COMP.	0	0	0	0	0	0	0	0	0	0	0	0	0
AIR BRAKE DELAY	100	1000	1000	1000	1000	100	1000	1000	1900	1200	800	100	1000
TOOL CHANGE OFFSET	0	0	0	0	0	0	0	0	0	0	0	0	0
LEAD COMP 1029	0	0	0	0	0	0	0	0	0	0	0	0	0

Options installed:
 Voltage: 208/230
 Macros

Code to turn on Macros: 64216

(PARAMETERS)
 (VER 9.22N p 96-12-20)
 (S/N 9345)
 (6 32 99 12)

Parameter 1 X SWITCH A (32771)

REV ENCODER	1	INV HOME SW	0	Z TEST HIST	0	TORQUE ONLY	0
REV POWER	1	INV Z CH	0	SCALE FACT/X	0	3 EREV/MREV	0
REV PHASING	0	CIRC. WRAP.	0	INVIS AXIS	0	2 EREV/MREV	0
DISABLED	0	NO I IN BRAK	0	ROT ALM LMSW	0	NON MUX PHAS	0
Z CH ONLY	0	LOW PASS+1X	0	ROT TRVL LIM	0	BRUSH MOTOR	0
AIR BRAKE	0	LOW PASS+2X	0	LIN SCALE EN	0	LINEAR DISPL	0
DISABLE Z T	0	OVER TEMP NC	0	INVRT LN SCL	0	SCALE/X LO	0
SERVO HIST	0	CABLE TEST	1	DSBL LS ZTST	0	SCALE/X HI	0

Parameter 266 X SWITCH B (0)

LIN SCALE EN	0	RESERVED	0	RESERVED	0	RESERVED	0
INVRT LN SCL	0	RESERVED	0	RESERVED	0	RESERVED	0
DSBL LS ZTST	0	RESERVED	0	RESERVED	0	RESERVED	0
RESERVED	0	RESERVED	0	RESERVED	0	RESERVED	0
RESERVED	0	RESERVED	0	RESERVED	0	RESERVED	0
RESERVED	0	RESERVED	0	RESERVED	0	RESERVED	0
RESERVED	0	RESERVED	0	RESERVED	0	RESERVED	0
RESERVED	0	RESERVED	0	RESERVED	0	RESERVED	0
RESERVED	0	RESERVED	0	RESERVED	0	RESERVED	0

2 X P GAIN	96	101 X IN POSITION LIMIT	8000
3 X D GAIN	6000	105 X MAX CURRENT	4095
4 X I GAIN	64	109 X D*D GAIN	1000
5 X RATIO (STEPS/UNIT)	138718	113 X ACC/DEC T CONST	375
6 X MAX TRAVEL (STEPS)	4161540	121 X PHASE OFFSET	0
7 X ACCELERATION	8000000	125 X GRID OFFSET	5638
8 X MAX SPEED	1638400	134 X EXACT STOP DIST.	180
9 X MAX ERROR	16384	138 X FRICTION COMPENSAT.	0
10 X FUSE LEVEL	3000000	145 X ACCEL FEED FORWARD	500000
11 X BACK EMF	550	201 X SPINDLE TEMP. COEF.	0
12 X STEPS/REVOLUTION	32768	202 X AIR BRAKE DELAY	0
13 X BACKLASH	3	210 X TOOL CHANGE OFFSET	0
14 X DEAD ZONE	0	229 X LEAD COMP 10E9	-75000

Parameter 15 Y SWITCH A (32768)

REV ENCODER	0	INV HOME SW	0	Z TEST HIST	0	TORQUE ONLY	0
REV POWER	0	INV Z CH	0	SCALE FACT/X	0	3 EREV/MREV	0
REV PHASING	0	CIRC. WRAP.	0	INVIS AXIS	0	2 EREV/MREV	0
DISABLED	0	NO I IN BRAK	0	ROT ALM LMSW	0	NON MUX PHAS	0
Z CH ONLY	0	LOW PASS+1X	0	ROT TRVL LIM	0	BRUSH MOTOR	0
AIR BRAKE	0	LOW PASS+2X	0	LIN SCALE EN	0	LINEAR DISPL	0
DISABLE Z T	0	OVER TEMP NC	0	INVRT LN SCL	0	SCALE/X LO	0
SERVO HIST	0	CABLE TEST	1	DSBL LS ZTST	0	SCALE/X HI	0

Parameter 267 Y SWITCH B (0)

LIN SCALE EN	0	RESERVED	0	RESERVED	0	RESERVED	0
INVRT LN SCL	0	RESERVED	0	RESERVED	0	RESERVED	0
DSBL LS ZTST	0	RESERVED	0	RESERVED	0	RESERVED	0
RESERVED	0	RESERVED	0	RESERVED	0	RESERVED	0
RESERVED	0	RESERVED	0	RESERVED	0	RESERVED	0
RESERVED	0	RESERVED	0	RESERVED	0	RESERVED	0
RESERVED	0	RESERVED	0	RESERVED	0	RESERVED	0
RESERVED	0	RESERVED	0	RESERVED	0	RESERVED	0
RESERVED	0	RESERVED	0	RESERVED	0	RESERVED	0

16 Y P GAIN		96		102 Y IN POSITION LIMIT		8000	
17 Y D GAIN		6000		106 Y MAX CURRENT		4095	
18 Y I GAIN		64		110 Y D*D GAIN		1000	
19 Y RATIO (STEPS/UNIT)		138718		114 Y ACC/DEC T CONST		375	
20 Y MAX TRAVEL (STEPS)		2254168		122 Y PHASE OFFSET		0	
21 Y ACCELERATION		8000000		126 Y GRID OFFSET		-2341	
22 Y MAX SPEED		1638400		135 Y EXACT STOP DIST.		180	
23 Y MAX ERROR		16384		139 Y FRICTION COMPENSAT.		0	
24 Y FUSE LEVEL		3000000		146 Y ACCEL FEED FORWARD		500000	
25 Y BACK EMF		550		132 Y SPINDLE TEMP. COEF.		0	
26 Y STEPS/REVOLUTION		32768		203 Y AIR BRAKE DELAY		0	
27 Y BACKLASH		36		211 Y TOOL CHANGE OFFSET		0	
28 Y DEAD ZONE		0		230 Y LEAD COMP 10E9		0	

Parameter 29 Z SWITCH A (32771)

REV ENCODER	1	INV HOME SW	0	Z TEST HIST	0	TORQUE ONLY	0
REV POWER	1	INV Z CH	0	SCALE FACT/X	0	3 EREV/MREV	0
REV PHASING	0	CIRC. WRAP.	0	INVIS AXIS	0	2 EREV/MREV	0
DISABLED	0	NO I IN BRAK	0	ROT ALM LMSW	0	NON MUX PHAS	0
Z CH ONLY	0	LOW PASS+1X	0	ROT TRVL LIM	0	BRUSH MOTOR	0
AIR BRAKE	0	LOW PASS+2X	0	LIN SCALE EN	0	LINEAR DISPL	0
DISABLE Z T	0	OVER TEMP NC	0	INVRT LN SCL	0	SCALE/X LO	0
SERVO HIST	0	CABLE TEST	1	DSBL LS ZTST	0	SCALE/X HI	0

Parameter 268 Z SWITCH B (0)

LIN SCALE EN	0	RESERVED	0	RESERVED	0	RESERVED	0
INVRT LN SCL	0	RESERVED	0	RESERVED	0	RESERVED	0
DSBL LS ZTST	0	RESERVED	0	RESERVED	0	RESERVED	0
RESERVED	0	RESERVED	0	RESERVED	0	RESERVED	0
RESERVED	0	RESERVED	0	RESERVED	0	RESERVED	0
RESERVED	0	RESERVED	0	RESERVED	0	RESERVED	0
RESERVED	0	RESERVED	0	RESERVED	0	RESERVED	0
RESERVED	0	RESERVED	0	RESERVED	0	RESERVED	0
RESERVED	0	RESERVED	0	RESERVED	0	RESERVED	0

30 Z P GAIN		96		103 Z IN POSITION LIMIT		8000	
31 Z D GAIN		6000		107 Z MAX CURRENT		4095	
32 Z I GAIN		64		111 Z D*D GAIN		1000	
33 Z RATIO (STEPS/UNIT)		138718		115 Z ACC/DEC T CONST		375	
34 Z MAX TRAVEL (STEPS)		2254168		123 Z PHASE OFFSET		0	
35 Z ACCELERATION		8000000		127 Z GRID OFFSET		-20210	
36 Z MAX SPEED		1638400		136 Z EXACT STOP DIST.		180	
37 Z MAX ERROR		16384		140 Z FRICTION COMPENSAT.		0	
38 Z FUSE LEVEL		3000000		147 Z ACCEL FEED FORWARD		500000	
39 Z BACK EMF		550		133 Z SPINDLE TEMP. COEF.		0	
40 Z STEPS/REVOLUTION		32768		204 Z AIR BRAKE DELAY		0	
41 Z BACKLASH		12		64 Z TOOL CHANGE OFFSET		595591	
42 Z DEAD ZONE		0		231 Z LEAD COMP 10E9		0	



152 B P GAIN	250	165 B IN POSITION LIMIT	1000
153 B D GAIN	20000	166 B MAX CURRENT	1774
154 B I GAIN	64	167 B D*D GAIN	-2000
155 B RATIO (STEPS/UNIT)	1600	168 B ACC/DEC T CONST	375
156 B MAX TRAVEL (STEPS)	480000	169 B PHASE OFFSET	0
157 B ACCELERATION	480000	170 B GRID OFFSET	0
158 B MAX SPEED	128000	171 B EXACT STOP DIST.	3
159 B MAX ERROR	2000	172 B FRICTION COMPENSAT.	0
160 B FUSE LEVEL	250000	173 B ACCEL FEED FORWARD	125000
161 B BACK EMF	370	174 B SPINDLE TEMP. COEF.	0
162 B STEPS/REVOLUTION	4000	175 B AIR BRAKE DELAY	1000
163 B BACKLASH	6	213 B TOOL CHANGE OFFSET	0
164 B DEAD ZONE	0	233 B LEAD COMP 10E9	0

Parameter 57 COMMON SWITCH 1 (-1183017339)

REV CRANK	1	SAFETY CIRC.	0	SP TEMP MONI	0	HANDLE CURSR	1
DISABLE T.C	0	SP DR LIN AC	1	ENA ROT & SC	0	NEG WORK OFS	0
DISABLE G.B	1	PH LOSS DET	1	ENABLE DNC	1	SPIN COOLANT	0
POF AT E-STP	0	COOLANT SPGT	0	ENABLE BGEDT	1	ENA CONVERSE	1
RIGID TAP	0	OVER T IS NC	1	ENA GRND FLT	1	OILER ON/OFF	1
REV SPIN ENC	0	SKIP OVERSHT	0	KEYBD SHIFT	1	NC OVER VOLT	1
REPT RIG TAP	0	NONINV SP ST	0	ENABLE MACRO	1	UNUSED	0
EX ST MD CHG	1	SP LOAD MONI	1	INVERT SKIP	0	DOOR STOP SP	1

Parameter 209 COMMON SWITCH 2 (-2141724144)

HORIZONTAL	0	AUX AXIS TC	0	RED BEACON	1	TSC ENABLE	0
RST STOPS TC	0	SPGT KEY INV	1	CNVY DR OVRD	1	AUX JOG NACC	0
UNUSED	0	T SUBROUTINE	0	DSBL CLNT IN	1	ALISM PRGRST	0
ENA CONVEYOR	0	SPIN Y ENCDR	0	DSC INP PR	0	DSBL JOG TST	0
50% RPD KBD	1	REV CONVEYOR	0	RMT TOOL RLS	1	AIR DR @ M24	0
FRONT DOOR	0	M27-28 CNVYR	1	FLOPPY ENABL	0	UNDEFINED	0
TC Z NO HOME	0	LOPH A ONLY	1	TCR KEYPAD	1	P RDY @ Y160	0
M36 AUTO MOT	0	GREEN BEACON	1	MCD RLY BRD	0	SPNDL NOWAIT	1

58 LEAD COMPENS SHIFT	14	120 SPINDLE ACC/DECEL	25
59 MAX FEEDRATE (INCH)	300	129 GEAR CH SETTLE TIME	50
60 TURRET START DELAY	300	130 GEAR STROKE DELAY	1000
61 TURRET STOP DELAY	1500	131 MAX SPINDLE RPM	7500
62 SHUTTLE START DELAY	400	142 HIGH/LOW GEAR CHANG	1501
63 SHUTTLE STOP DELAY	2000	143 DRAW BAR Z VEL CLMP	120000
65 NUMBER OF TOOLS	20	144 RIG TAP FINISH DIST	692
66 SPINDLE ORI DELAY	5000	149 PRE-CHARGE DELAY	200
67 GEAR CHANGE DELAY	9000	150 MAX SP RPM LOW GEAR	1829
68 DRAW BAR MAX DELAY	5000	206 SPIGOT POSITIONS	14
70 MIN SPIN DELAY TIME	200	207 SPIGOT TIMEOUT (MS)	2000
71 DRAW BAR OFFSET	12000	208 SPIN. FAN OFF DELAY	6000
72 DRAW BAR Z VEL UNCL	120000	215 CAROUSEL OFFSET	0
73 SP HIGH G/MIN SPEED	30	216 CNVYR RELAY DELAY	50
74 SP LOW G/MIN SPEED	80	217 CNVYR IGNORE OC TIM	50
75 GEAR CHANGE SPEED	48	218 CNVYR RETRY REV TIM	200
76 LOW AIR DELAY	1500	219 CNVYR RETRY LIMIT	3
77 SP LOCK SETTLE TIME	200	220 CNVYR RETRY TIMEOUT	30000
78 GEAR CH REV TIME	500	221 MAX TIME NO DISPLAY	10
79 SPINDLE STEPS/REV	2000	222 ROTARY AXIS INCRMNT	0
80 MAX SPIN DELAY TIME	5000	223 AIR TC DOOR DELAY	0
81 M MACRO CALL 09000	0	224 ROT AXIS ZERO OFFSET	0
82 M MACRO CALL 09001	0	225 MAX ROT AXIS ALLOW	0



83	M	MACRO CALL	09002	0	226	EDITOR CLIPBOARD	8998
84	M	MACRO CALL	09003	0	227	FLOPPY DIR NAME	8999
85	M	MACRO CALL	09004	0	228	QUICKCODE FILE	9999
86	M	MACRO CALL	09005	0	235	TSC PISTON SEAT	500
87	M	MACRO CALL	09006	0	236	TSC LOW PR FLT	1000
88	M	MACRO CALL	09007	0	237	TSC CLNT LINE PURGE	2500
89	M	MACRO CALL	09008	0	238	TSC MAX SPINDLE RPM	7500
90	M	MACRO CALL	09009	0	239	SPNDL ENC STEPS/REV	4000
91	G	MACRO CALL	09010	0	240	C AXIS MAX TRAVEL	1999999
92	G	MACRO CALL	09011	0	241	U AXIS MAX TRAVEL	1999999
93	G	MACRO CALL	09012	0	242	V AXIS MAX TRAVEL	1999999
94	G	MACRO CALL	09013	0	243	W AXIS MAX TRAVEL	1999999
95	G	MACRO CALL	09014	0	244	C AXIS MIN TRAVEL	-1999999
96	G	MACRO CALL	09015	0	245	U AXIS MIN TRAVEL	-1999999
97	G	MACRO CALL	09016	0	246	V AXIS MIN TRAVEL	-1999999
98	G	MACRO CALL	09017	0	247	W AXIS MIN TRAVEL	-1999999
99	G	MACRO CALL	09018	0	255	CONVEYOR TIMEOUT	30
100	G	MACRO CALL	09019	0	256	PALLET LOCK INPUT	0
117		LUB CYCLE TIME	108000		257	SPINDL ORIENT OFFSET	0
118		SPINDLE REV TIME	30		258	LS PER INCH	12700
119		SPINDLE DECEL DELAY	35		259	LS PER REV	25000

Parameter 278 COMMON SWITCH 3 (0)

INVERT G.B.	0	RESERVED	0	RESERVED	0	RESERVED	0
DPR SERIAL	0	CNCR SPINDLE	0	RESERVED	0	RESERVED	0
CK PALLET IN	0	RESERVED	0	RESERVED	0	RESERVED	0
CK HIDDN VAR	0	RESERVED	0	RESERVED	0	RESERVED	0
DISPLAY ACT	0	RESERVED	0	RESERVED	0	RESERVED	0
TSC PRG ENBL	0	RESERVED	0	RESERVED	0	RESERVED	0
RESERVED	0	RESERVED	0	RESERVED	0	RESERVED	0
SPND DRV LCK	0	RESERVED	0	RESERVED	0	RESERVED	0

X AXIS LEAD SCREW COMPENSATION (INCHES):
ALL VALUES ARE ZERO

Y AXIS LEAD SCREW COMPENSATION (INCHES):
ALL VALUES ARE ZERO

Z AXIS LEAD SCREW COMPENSATION (INCHES):
ALL VALUES ARE ZERO

ROTARY PRODUCT PARAMETERS ARE LISTED ON A SEPARATE DATA SHEET

1	AUTO POWER OFF TIMER	0	46	MIRROR IMAGE Y AXIS	OFF
2	POWER OFF AT M30	OFF	47	MIRROR IMAGE Z AXIS	OFF
3	3D GRAPHICS	OFF	48	MIRROR IMAGE A AXIS	OFF
4	GRAPHICS RAPID PATH	ON	49	SKIP SAME TOOL CHANGE	ON
5	GRAPHICS DRILL POINT	ON	50	AUX AXIS SYNC	XON/XOFF
6	FRONT PANEL LOCK	OFF	51	DOOR HOLD OVERRIDE	OFF
7	PARAMETER LOCK	OFF	52	G83 RETRACT ABOVE R	0.0000
8	MEMORY PROTECT	OFF	53	JOG W/O ZERO RETURN	OFF
9	DIMENSIONING	INCH	54	AUX AXIS BAUD RATE	4800
10	LIMIT RAPID AT 50%	OFF	55	ENABLE DNC FROM MDI	OFF
11	BAUD RATE SELECT	9600	56	M30 RESTORE DEFAULT G	OFF
12	PARITY SELECT	EVEN	57	EXACT STOP CANNED X-Y	OFF
13	STOP BIT	1	58	CUTTER COMPENSATION	FANUC
14	SYNCHRONIZATION	XON/XOFF	59	PROBE OFFSET X+	0.0000
15	H & T CODE AGREEMENT	ON	60	PROBE OFFSET X-	0.0000
16	DRY RUN LOCK OUT	OFF	61	PROBE OFFSET Y+	0.0000
17	OPT STOP LOCK OUT	OFF	62	PROBE OFFSET Y-	0.0000
18	BLOCK DELETE LOCK OUT	OFF	63	TOOL PROBE WIDTH	0.0000
19	FEEDRATE OVERRIDE LOCK	OFF	64	T OFS MEAS USES WORK	ON
20	SPINDLE OVERRIDE LOCK	OFF	65	GRAPH SCALE (HEIGHT)	18.576
21	RAPID OVERRIDE LOCK	OFF	66	GRAPHICS X OFFSET	0.0000
22	CAN CYCLE DELTA Z	0.0100	67	GRAPHICS Y OFFSET	0.0000
23	9xxx PROGS EDIT LOCK	OFF	68	GRAPHICS Z OFFSET	0.0000
24	LEADER TO PUNCH	NONE	69	DPRNT LEADING SPACES	OFF
25	EOB PATTERN	CR LF	70	DPRNT OPEN/CLOS DCODE	ON
26	SERIAL NUMBER	9345	71	DEFAULT G51 SCALING	1.000
27	G76/G77 SHIFT DIR.	X-	72	DEFAULT G68 ROTATION	0.000
28	CAN CYCLE ACT W/O X/Y	ON	73	G68 INCREMENTAL ANGLE	OFF
29	G91 NON-MODAL	OFF	74	9xxx PROGS TRACE	ON
30	4TH AXIS ENABLE	OFF	75	9xxx PROGS SINGLE BLK	ON
31	RESET PROGRAM POINTER	ON	76	TOOL RELEASE LOCK OUT	OFF
32	COOLANT OVERRIDE	NORMAL	77	SCALE INTEGER F	DEFAULT
33	COORDINATE SYSTEM	YASNAC	78	5TH AXIS ENABLE	OFF
34	4TH AXIS DIAMETER	1.0000	79	5TH AXIS DIAMETER	1.0000
35	G60 OFFSET	0.3000	80	MIRROR IMAGE B AXIS	OFF
36	PROGRAM RESTART	OFF	81	TOOL AT POWERDOWN	1
37	RS-232 DATA BITS	7	82	LANGUAGE	ENGLISH
38	AUX AXIS NUMBER	0	83	M30 RESETS OVERRIDE	OFF
39	BEEP AT M30	OFF	84	TOOL OVERLOAD ACTION	ALARM
40	TOOL OFFSET MEASURE	RADIUS	85	IN-POSITION ACCURACY	0.0500
41	ADD SPACES RS232 OUT	ON	86	M39 LOCKOUT	OFF
42	M00 AFTER M06	OFF	87	M06 RESETS OVERRIDE	OFF
43	CUTTER COMP TYPE	A	88	RESET RESETS OVERRIDE	OFF
44	MIN F IN RADIUS CC %	50	103	CYC START/FH SAME KEY	OFF
45	MIRROR IMAGE X AXIS	OFF	104	JOG HANDL TO SNGL BLK	OFF

CHAPTER IV.

MAINTENANCE SCHEDULE AND LUBRICATION CHART FOR THE VF-SERIES MILL

The following is a list of required regular maintenance for the HAAS VF Series Vertical Machining Centers. Listed are the frequency of service, capacities, and type of fluids required. These required specifications must be followed in order to keep your machine in good working order and protect your warranty.

MAINTENANCE SCHEDULE

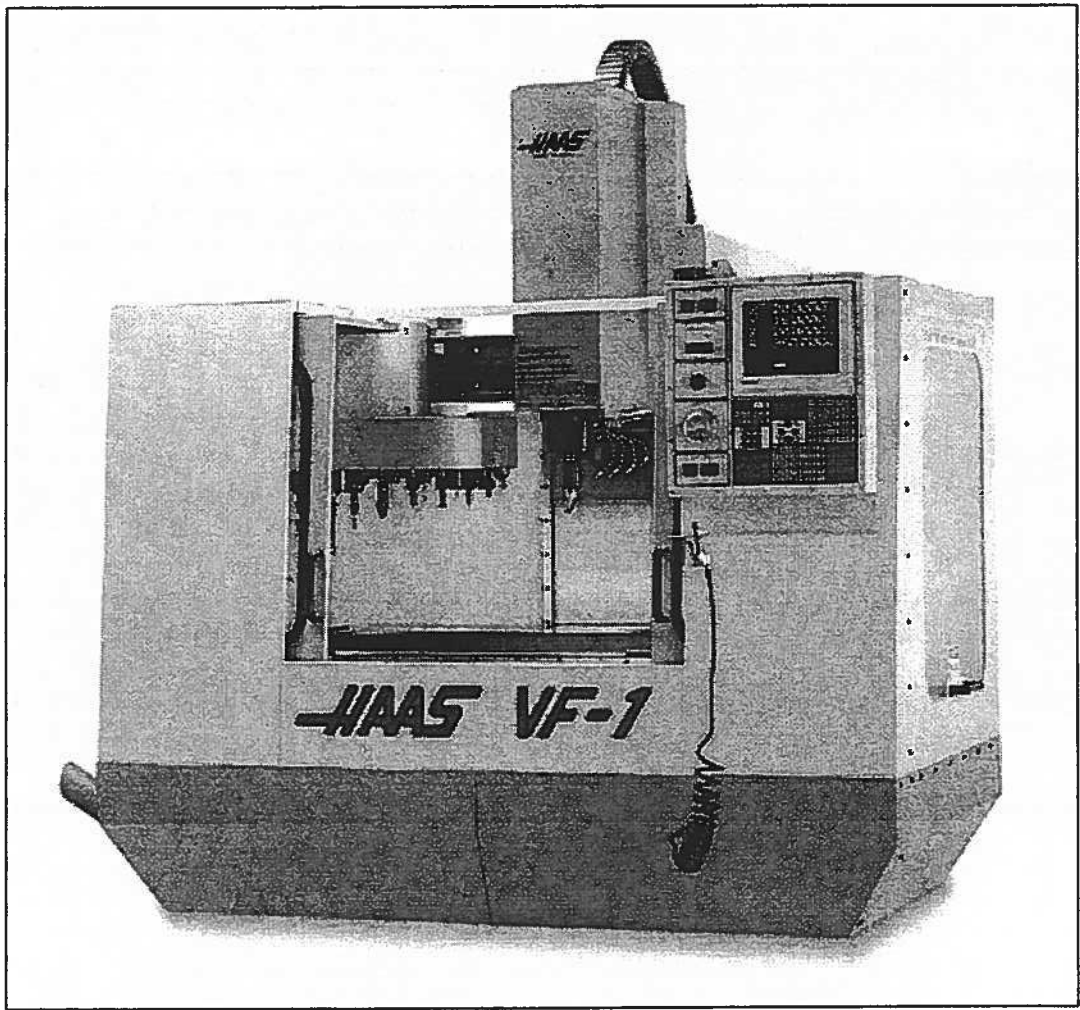
DAILY	<ul style="list-style-type: none"> ✓ Check coolant level. ✓ Check way lube lubrication tank level. ✓ Clean chips from way covers and bottom pan. ✓ Clean chips from tool changer. ✓ Wipe spindle taper with a clean cloth rag and apply light oil.
WEEKLY	<ul style="list-style-type: none"> ✓ Check oil level in gear box. Remove inspection cover beneath spindle head. Add oil slowly from top until oil begins dripping from overflow tube at bottom of sump tank. ✓ Check Through the Spindle Coolant (TSC) filters. Clean or replace element if needed. ✓ Check automatic dump air line's water trap for proper operation. ✓ On machines with the TSC option, clean the chip basket on the coolant tank. Remove the tank cover and remove any sediment inside the tank. Be careful to disconnect the coolant pump from the controller and POWER OFF the control before working on the coolant tank. Do this MONTHLY for machines without the TSC option. ✓ Check air gauge/regulator for 85 psi. ✓ For machines with the TSC option, place a dab of grease on the V-flange of tools. Do this MONTHLY for machines without the TSC option. ✓ Check aluminum air filters on control heat exchanger and at top of spindle motor. ✓ Clean exterior surfaces with mild cleaner. DO NOT use solvents. ✓ Check the hydraulic counterbalance pressure according to the machine's specifications.
MONTHLY	<ul style="list-style-type: none"> ✓ Check oil level in gear box by adding oil until it begins dripping from the drain tube. ✓ Inspect way covers for proper operation and lubricate with light oil, if necessary. ✓ Dump the oil drain bucket. ✓ Place a dab of grease on the outside edge of the Geneva wheel star and guide rails of the tool changer and run through all tools.
SIX MONTHS	<ul style="list-style-type: none"> ✓ Replace coolant and thoroughly clean the coolant tank. ✓ Check all hoses and lubrication lines for cracking. ✓ On machines with TSC option, check drawbar height every 6 months or after 1000 hrs of use. Replace coolant tip if drawbar height is more than .010" out of spec.
ANNUALLY	<ul style="list-style-type: none"> ✓ With the air pressure OFF, disassemble and clean the small filter at end of lubricator (left side of machine). ✓ Replace the gearbox oil. Drain the oil from the bottom of the gearbox. Remove inspection cover beneath spindle head. Add oil slowly from top until oil begins dripping from overflow tube at bottom of sump tank. ✓ Check oil filter and clean out residue at bottom of filter.

LUBRICATION CHART

SYSTEM:	WAY LUBE AND PNEUMATICS	TRANSMISSION	COOLANT TANK
LOCATION	Under the control panel at the rear of the machine	Above the spindle head	Side of machine
DESCRIPTION	Piston pump with 30-minute cycle time. Pump is only on when spindle is turning or when axis is moving.		
LUBRICATES	Linear guides and ball nuts	Transmission only	
QUANTITY	1-QT. Tank	2-QT. Tank	40 gallons
LUBRICANT	Mobil Vactra #2	Mobil DTE 25	Water soluble, synthetic

CHAPTER V.

INSTALLATION MANUAL FOR THE VF-SERIES MILL



Contents

INSTALLING THE VMC	247
DIMENSIONS	247
SERVICE REQUIREMENTS	248
ELECTRICITY REQUIREMENTS	248
AIR REQUIREMENTS	248
MOVING THE CRATE	249
UNPACKING THE VMC	249
UNCRATING	249
SETTING IN PLACE	250
INITIAL SETUP	251
AIR CONNECTION	251
OPTIONAL CHIP CONVEYOR INSTALLATION	252
ELECTRICAL CONNECTIONS	253
MACHINE START UP	254
COOLANT TANK INSTALLATION	256
OPTIONAL THROUGH THE SPINDLE COOLANT START-UP	257
LEVELING THE VMC	257
ROUGH LEVELING	257
FINE LEVELING	259
APPENDIX A: PREPARATION FOR SHIPPING	260
LOCK UP THE MACHINE FOR SHIPPING	261
DISCONNECT SERVICES	261
SECURE THE FRONT	262
MOVE ONTO THE PALLET	262
COMPLETE CRATING FOR THE VMC	264
APPENDIX B: SHIPPING THE MACHINE WITHOUT A PALLET	265
PROCEDURE	265
VMC CABLE LIFT INSTRUCTIONS	267

INSTALLING THE VMC

GENERAL WARNINGS

It is the shop operator's responsibility to make sure that everyone who is involved in installing and operating the Vertical Machining Center (VMC) is thoroughly acquainted with the installation, operating, and safety instructions provided with the machine **BEFORE** they perform any actual work.

NOTE: The ultimate responsibility for safety rests with the shop operator and the individuals who work with the machine.

The electrical power must meet the specifications in this manual. Attempting to run the VMC from any other source can cause severe damage and void the warranty.

DO NOT press POWER UP/RESTART on the control panel until after the installation is complete.

DO NOT attempt to operate the machine before all of the installation instructions have been completed.

Observe all of the warnings and cautions in the "Installation" section of the operation manual.

Move the crated VMC with care — position forklift forks only as specified by the marks on the crate and the instructions in this manual.

NEVER service the VMC with the power connected.

NEVER leave the control boxes open when the power is on. A deadly high voltage may be present in **ALL** areas of the control boxes.

IMPORTANT!!! Once the machine is installed, the control cabinet must be locked and the key given to the shop manager.

DIMENSIONS

OPERATING	VF-0	VF-1	VF-2	VF-3	VF-4	VF-6	VF-8
Height (in.)	110	110	110	110	110	130	130
Width (in.)	98	98	98	126	126	166	166
Depth (in.)	137	137	137	157	157	160	176
Weight (lb.)	6,500	6,500	7,000	13,540	14,000	21,000	24,000

NOTE: The *operating dimensions* are the maximum dimensions of the machine during operation. These are the dimensions of the machine with the control panel door completely open, the spindle head at it's highest point, the control at it's most forward position, and with the discharge tube installed.

OUT OF CRATE	VF-0/OE/1/2	VF-3/4	VF-6	VF-8
Height (in.)	91	102	107	107
Width (in.)	88	125	156	156
Depth (in.)	93	96	96	117

SERVICE REQUIREMENTS**ELECTRICITY REQUIREMENTS**

The VMC runs on a three-phase, 195-260 V, 50- or 60-Hz supply. The power required is 40 amps and should not fluctuate more than $\pm 5\%$. A high voltage (360-480 V) option is also available. The power required for this option is 20 amps, which should not fluctuate more than $\pm 5\%$.

If the service run from the electrical panel is less than 120', use No. 12 wire; if the run is longer, use No. 10 wire.

WARNING!!

A separate earth, or cold-water-pipe, ground is required (a conduit-type ground is not sufficient).

Do not connect to 480 V, unless the machine includes the high voltage option!

WARNING!

Maximum voltage leg-to-leg or leg-to-ground should not exceed 260 volts (504 volts for high voltage option)!

WARNING! The power must be turned off at the panel during electrical service. **Under no circumstances should this machine be supplied more than 260 V at 60 Hz or 240 V at 50 Hz (504 V at 60 Hz or 480 V at 50 Hz for high voltage option).** Use of voltages lower than the specified minimums reduces the torque of the spindle motor at high speeds. **Use of single-phase power is not possible with this machine.**

AIR REQUIREMENTS

The VMC requires a minimum of 100 PSI at 4 cfm at the input to the pressure regulator on the back of the machine. This should be supplied by at least a one horsepower compressor, with a minimum 20-gallon tank, that turns on when the pressure drops to 100 PSI.

The air must be supplied through a minimum 3/8" hose and fittings must be at least 1/4" NPT.

NOTE: Excessive oil and water in the air supply will cause the machine to malfunction. The air filter/regulator has an automatic bowl dump that should be empty before starting the machine. This must be checked for proper operation monthly. Also, excessive contaminants in the air line may clog the dump valve and cause air and/or water to pass into the machine.

NOTE: The nipple between the air filter/regulator and the Bijur oil lubricator (See illustration in "Air Connection" section) reservoir tank below the control box on the back of the machine is for the optional rotary table. **DO NOT** use this as a connection for an auxiliary air line. Auxiliary connections should be made on the left side of the air filter/regulator.

WARNING!

When the machine is operating and the pressure gauge (on the machine regulator) drops by more than 10 psi during tool changes, insufficient air is being supplied to the machine.

TOOLS REQUIRED

Precision bubble level, such as one calibrated to show 0.0005 inch per 10"
 Test indicator (0.0005)
 9/16" hex wrench
 Two 3/4" hex wrenches (one open end or box and one ratchet)
 Claw hammer

For VF-0/1/2: A forklift capable of lifting more than 7,500 pounds, with forks at least 5' long by 6" wide.
For VF-3/4: A forklift capable of lifting more than 15,000 pounds, with forks at least 7' long by 6" wide.
For VF-6: A forklift capable of lifting more than 21,000 pounds, with forks at least 7' long by 6" wide.
For VF-8: A forklift capable of lifting more than 24,000 pounds, with forks at least 8' long by 6" wide.

MATERIALS REQUIRED

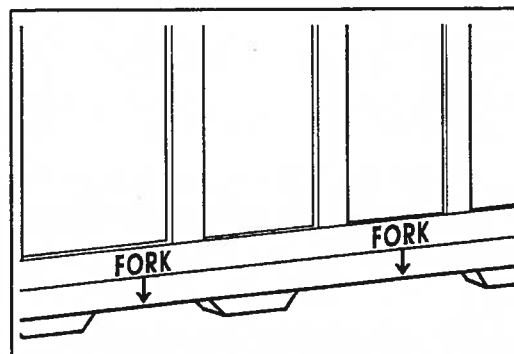
Wire and air hose or piping as specified in the Service Requirements section
 35-45 gallons of water-soluble synthetic, or cutting oil coolant
 A small amount of grease
 Way lube for the lubricator (Vactra #2)

CAUTION!

THE VMC CRATE CAN ONLY BE MOVED WITH A FORKLIFT.

MOVING THE CRATE

CAUTION! The fork positions are marked on the crate. (Also, note that there are three skids at each side of the pallet. The heavy part of the machine [the back] is positioned over the two skids that are closest together.) If the fork positions are ignored, there is a good chance that the retaining bolts will be sheared off by the forks and also that the machine will tip over when it is picked up.



UNPACKING THE VMC

UNCRATING

NOTE: Unless you are certain that you will not be shipping the machine, the crate and packing materials should be stored for reuse. Be careful not to damage the crate and the other packing materials.

CAUTION! Use extreme caution not to damage the pressurized hydraulic counterbalance when removing the machine from it's container.

1. Pry off the clips around the top of the crate with a claw hammer and remove the top panel.
2. Pry off all but one clip at each corner of the crate.
3. Using a 9/16" wrench, remove the lag bolts around the bottom of the crate.
4. Pry off the last clip at each corner and remove the side panels.

CAUTION!

The side panels are heavy — be careful that they do not drop on your feet or tip over on you.

5. Pull the plastic cover off the machine, fold it, and set it aside to store with the crate.

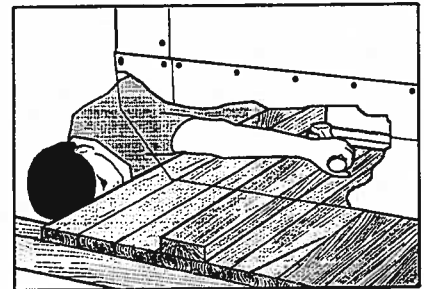
CAUTION!

Do not put undue pressure on the top of the machine as you remove the plastic.

6. Lift out the large box (containing the coolant tank) and the tool box on the left side of the machine. Remove the cleats that held them in place.

7. Remove the touch-up paint that is taped to the gutter of the table.

8. On the **VF-6/8**, use wrenches to remove the nuts (under the pallet) from the retaining bolts at the two rear corners of the machine. (A ratchet on top and a wrench on the bottom is the easiest way to do this.) At the two front corners, remove the bolts from the bottom of the pallet. Leave all four front corners, remove the bolts from the bottom of the pallet. Leave all four front bolts in the holes after the nuts are removed. On the **VF-0/1/2/3/4**, simply loosen the retaining bolts and leave them in the holes.



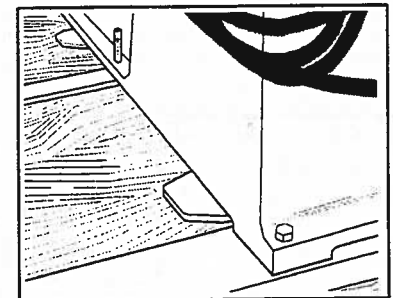
SETTING IN PLACE

CAUTION! Keep in mind when moving the VMC that much of its weight is concentrated in the column at the back. Therefore, when lifting a **VF-0/1/2** from the side, it is important that the forks of the forklift be positioned as close to the back of the machine as possible without being on the pads.

CAUTION! Do not lift the machine any farther than necessary off the floor when moving it, and move as slowly and cautiously as possible. Dropping the machine, even from a height of a few inches, can cause injury, result in expensive repairs, and void the warranty.

WARNING!

For the VF-0/1/2: The only acceptable way to move a **VF-0/1/2** is to pick it up from the **SIDE** with a forklift. The forklift must be capable of lifting more than 7,500 pounds. The forks must be at least 5' long by 6" wide and they must be set as far apart as possible without being on the pads. The forks must be positioned all the way to the back of the VMC and they must extend at least 3" past the far side of the machine base. Also, there must be about approximately 6" clearance between the forklift and the side of the machine.

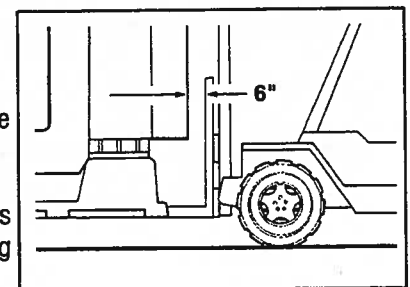


For the VF-3/4/6/8: The **VF-3/4/6/8** must be lifted from the **BACK** of the machine. The forklift must be capable of lifting more than 15,000 lbs. for the **VF-3/4**, 21,000 lbs. for the **VF-6**, and 24,000 lbs. for the **VF-8**. The forks must be at least 7' long for **VF-3/4** (8' long for the **VF-6/8**) by 6" wide and they must be set as far apart as possible without being on the pads. Also, there must be about approximately 6" clearance between the forklift and the back of the machine.

ATTEMPTING TO MOVE THE MACHINE ANY OTHER WAY MAY VOID THE WARRANTY.

CAUTION! Use caution when lifting the machine with a forklift, not to damage the sheet metal aprons with the forks.

1. Lift the machine until the bolts clear the pallet. Pull the bolts out of the holes in the machine. Be sure to remove the plastic bolt covers that the retaining bolts came with (not with **VF-6/8**).



VF-3/4/6/8

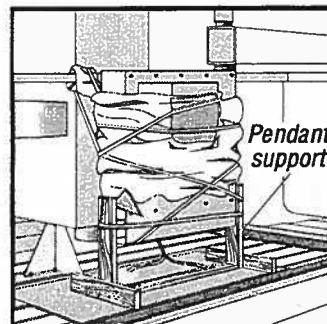
2. Get four leveling screws from the tool box and screw them into the holes at the four corners, hex-head end up, until they extend about an inch out of the bottom of the machine. If a screw is excessively hard to turn, remove it, dress the threads in the hole with a $\frac{3}{4}$ -10 tap, and inspect the screw. If the screw has dings, dress the threads with a 60° V file. (You must have good control over these screws because they are used to precision level the machine.)

3. Move the machine to where it will be located. Take four pads out of the tool box, grease the dimple in each pad, and locate them under the leveling screws at the four corners. Then lower the machine.

4. Remove all banding and packing material around the control panel and the doors.

5. On the VF-6/8, remove the pendant support.

6. Remove the control arm shipping brace. On the VF-3/4, swing the control arm into position and bolt it to the support on the top front of the machine enclosure. On the VF-6/8, swing the control arm to the proper position.



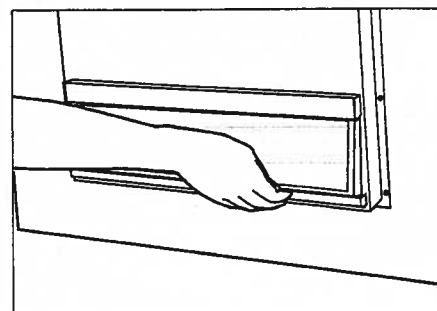
INITIAL SETUP

WARNING!

At this point, there should be NO electrical connection to the machine. The electrical panel should be closed and the three latches on the door 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 switch 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, you must exercise extreme caution when you are working in the panel.

1. Set the main switch at the upper right of the electrical panel on the back of the machine to OFF.
2. Using the screwdriver from the tool kit, unlock the three latches on the panel door and open the door.
3. The manuals and air filter screen are located at the bottom left of the panel.
4. Install the air filter toward the bottom on the outside of the panel door (push it up into the top mounting bracket and then push in the bottom edge).

5. Take sufficient time to check all the components and connectors associated with the circuit boards. With the power off, push on them gently to make sure that they are seated in their sockets. Look for any cables that have become disconnected, look for any signs of damage and loose parts in the bottom of the panel box. If there are any signs that the machine had a rough ride, be extremely careful in powering up the machine (be ready to shut it off IMMEDIATELY). Or if there are obvious problems, call the factory BEFORE proceeding.



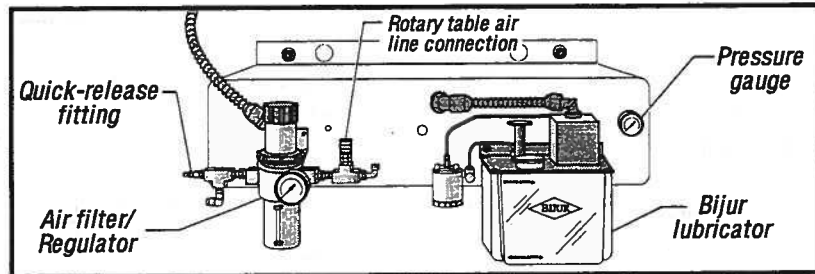
AIR CONNECTION

CAUTION!

Working with the air service required for VMC can be hazardous. Make sure that pressure has been removed from the air line before you connect it to the machine, disconnect it from the machine, or service parts of the air system on the machine.

INSTALLATION

1. When the machine leaves the factory, the air filter is empty, and air lubricator and the lubricator reservoir tank are full. However, they should be checked and serviced if required before compressed air is supplied to the machine.



2. With the pressure off in the air line, connect the air supply to the quick-release fitting next to the air filter/regulator (below the electrical panel). If the fitting supplied is not compatible, simply replace it.

3. Start the compressor, set it between 100 and 150 PSI. Set the regulator on the machine to 85 to 90 PSI.

4. Prime the Bijur lubricator to make sure it is working. To prime the lubrication system, pull up on the handle on top of the reservoir tank. **CAUTION! NEVER** push down on the primer handle! It gradually returns to the down position by itself, and the corresponding pressure increase can be seen on the pressure gauge.

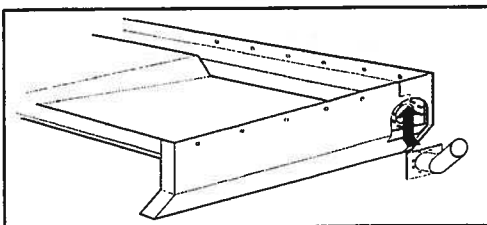
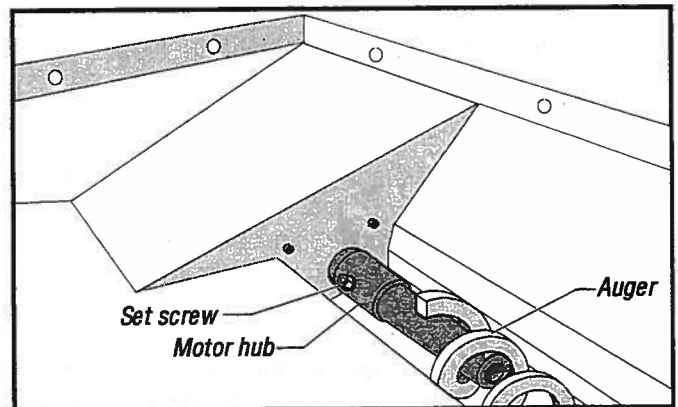
NOTE: Depending on the position of the cam that drives it, the lubrication system may not activate until a few minutes after the machine is started. However, if there is a problem with the system, an alarm will stop the machine.

OPTIONAL CHIP CONVEYOR INSTALLATION

1. Unpack the conveyor and discharge tube.

2. Slide the conveyor into the discharge tube opening and then slip opposite end onto motor hub. Fasten to motor hub with the 5/16-18 x 2½" bolt.

3. Install gasket and slide the discharge tube up and onto studs. Attach the eight nuts with locking washers and tighten uniformly.



4. After machine start-up, check the operation of the conveyor to ensure the direction of rotation will move the chips toward the discharge tube. If the conveyor is turning so that the chips are not being moved toward the discharge tube, change the bit switch in PARAM 209 from 1 to 0 or 0 to 1 to establish a new forward direction.

MAINTENANCE

During normal operation, most chips are discharged from the machine at the discharge tube. However, very small chips may flow through the drain and collect in the coolant tank strainer and pan coolant drain (under the pan). To prevent drain blockage, clean this trap regularly. Should the drain become clogged and cause coolant to collect in the machine's pan, stop the machine, loosen the chips blocking the drain, and allow the coolant to drain. Empty the coolant tank strainer, then resume operation.

ELECTRICAL CONNECTIONS

NOTE: The machine must have air pressure at the air gauge, or a "Low Air Pressure" alarm will be present on power up.

CAUTION!

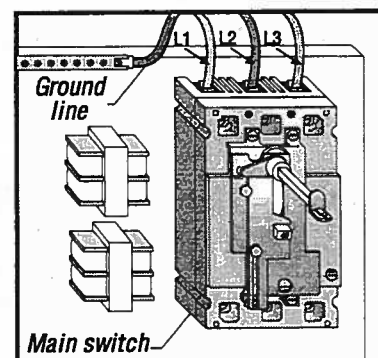
Working with the electrical services required for the VMC can be extremely hazardous. The electrical power must be off and steps must be taken to ensure that it will not be turned on while you are working with it. In most cases this means turning off a circuit breaker in a panel and then locking the panel door. However, if your connection is different or you are not sure how to do this, check with the appropriate personnel in your organization or otherwise obtain the necessary help BEFORE you continue.

WARNING!

The electrical panel should be closed and the three latches on the door 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.

1. Hook up the three power lines to the terminals on top of the main switch at upper right of electrical panel and the separate ground line to the ground bus to the left of the terminals. It is not necessary to be concerned with phase rotation (which wire is connected to L1, L2, and L3).

NOTE: Make sure that the service wires actually go into the terminal-block clamps. (It is easy to miss the clamp and tighten the screw. The connection looks fine but the machine runs intermittently or has other problems, such as servo overloads.) To check, simply pull on the wires after the screws are tightened.



2. After the line voltage is connected to the machine, make sure that main circuit breaker (at top-right of rear cabinet) is OFF (rotate the shaft that connects to the breaker counterclockwise until it snaps OFF). Turn ON the power at the source. Using an accurate digital voltmeter and appropriate safety procedures, measure the voltage between all three pair phases at the main circuit breaker and write down the readings. The voltage must be between 195 and 260 volts (360 and 480 volts for high voltage option).

NOTE: Wide voltage fluctuations are common in many industrial areas; you need to know the minimum and maximum voltage which will be supplied to the machine while it is in operation. U.S. National Electrical Code specifies that machines should operate with a variation of +5% to -5% around an average supply voltage. If problems with the line voltage occur, or low line voltage is suspected, an external transformer may be required. If you suspect voltage problems, the voltage should be checked every hour or two during a typical day to make sure that it does not fluctuate more than +5% or -5% from an average.

CAUTION!

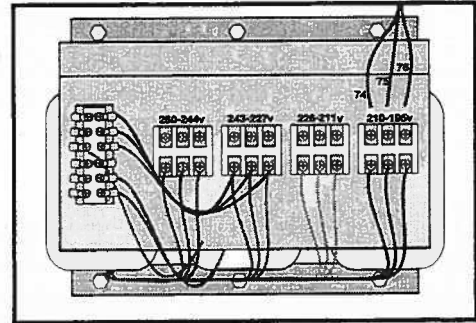
Make sure that the main breaker is set to OFF and the power is off at your supply panel BEFORE you change the transformer connections. Make sure that all three black wires are moved to the correct terminal block and that they are tight.

INSTALLATION

3. Check the connections on the transformer at the bottom-right corner of the rear cabinet. The three black wires labeled 74, 75, and 76 must be moved to the terminal block triple which corresponds to the average voltage measured in **step 2** above. There are four positions for the input power for the 260 volt transformer and five positions for the 480 volt transformer. The input voltage range for each terminal block is as follows:

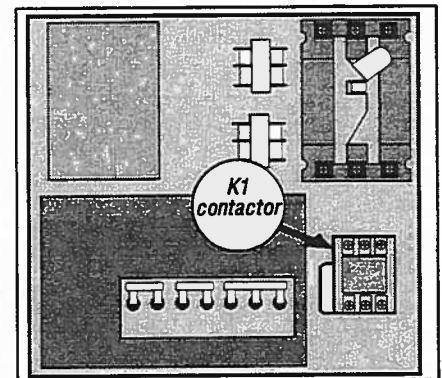
Right to left:	260 V	480 V
	195 to 210	353 to 376
	211 to 226	377 to 400
	227 to 243	401 to 425
	244 to 260	426 to 451
		452 to 480*

* 480 V transformer has additional terminal block



4. Set the main switch to ON (rotate the shaft that engages the handle on the panel door clockwise until it snaps into the ON position). Check for evidence of problems, such as the smell of overheating components or smoke. If such problems are indicated, set the main switch to OFF immediately and call the factory before proceeding.

5. After the power is on, measure the voltage across the upper terminals on the contactor K1 (located below the main circuit breaker). It should be the same as the measurements where the input power connects to the main breaker. If there are any problems, call the factory.



6. Check the DC voltage displayed in the second page of Diagnostic data on the CRT. It is labeled DC BUS. This voltage must be between 150 and 175 volts. If the voltage is outside these limits, turn off the power and recheck the 2 and 3. If the voltage is still incorrect, turn off the power and call the factory.

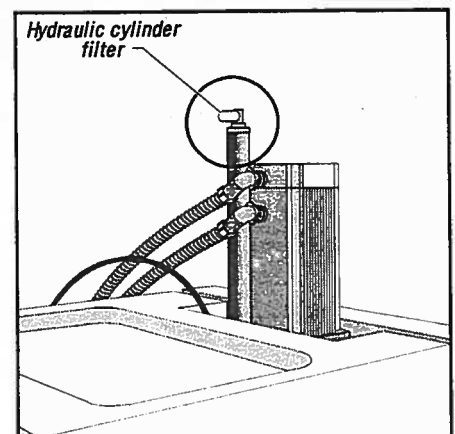
7. Turn off the power (rotate the shaft that engages the handle on the panel door counterclockwise until it snaps into the OFF position). Also, set the main switch handle on the panel door to OFF. (Both the handle and the switch must be set to OFF before the door can be closed). Close the door, lock the latches, and turn the power back on.

8. Remove the key from the control cabinet and give it to the shop manager.

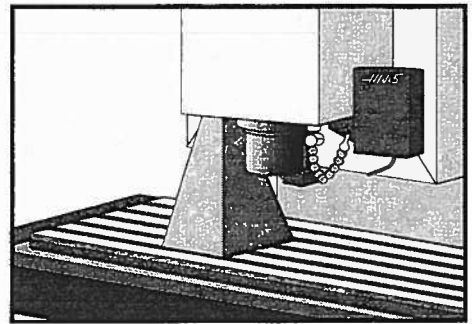
MACHINE START UP

1. Check if the filter is placed in the counterbalance cylinder. If it is not, remove the plug located at the top of the counterbalance cylinder and install and position the filter as shown in the figure at right. **CAUTION!** This filter must be installed prior to machine operation in order to prevent damage to the counterbalance system.

2. On the VF-0/1/2/3/4: before POWERING UP machine, remove the tie wrap that holds the flex tube in place (see figure on next page).



3. Loosen the four SHCS (three screws are in the table, and one is in the spindle head) holding the shipping bracket under the spindle head, and remove the bracket.



4. With the main switch on the electrical panel set to ON, press and release POWER ON at the upper left of the control panel. You will hear a click in the back of the machine and the fans will energize. (If you don't hear these sounds, the machine is not getting power and, with all necessary safety precautions, you should check the connections to the electrical panel.) After a few seconds, the display will appear on the screen.

CAUTION! DO NOT press POWER UP/RESTART on the control panel while the shipping bracket is under the spindle. Also, do not touch the X, Y, or Z buttons or the jog handle while the shipping bracket is located under the spindle.

5. Press and release SETNG GRAPH. Then page down to the last page (press and release PAGE DOWN several times). Cursor to Setting 53, JOG W/O ZERO RETURN (with the cursor down key). Press and release the cursor right key and then press and release the WRITE key at the extreme lower right of the control panel to turn this setting on. Turning on JOG W/O ZERO RETURN bypasses the zero return interlock.

NOTE: This setting, like many others, resets to OFF when the machine is powered up. This prevents the machine from operating until a zero return has been executed — the machine control cannot determine position until it has been set by a zero return routine. For this reason, it is important that you execute a zero return immediately each time the machine is started for normal operation BUT NOT FOR THIS START-UP ROUTINE.

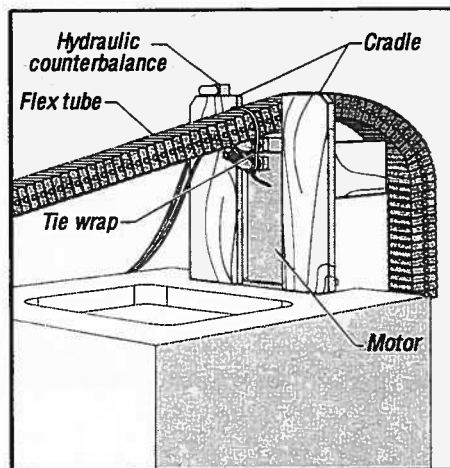
6. Press and release the RESET button twice, or until you have no alarms, to turn the servos on. (The message "ALARM" appears at the lower right of the screen if one or more alarms are in effect.)

NOTE: If any alarms are present and cannot be cleared with the RESET button, press and release the ALARM MESSAGES button for more information on the alarms. If you are unable to clear the alarms, write down the alarm numbers and call the factory.

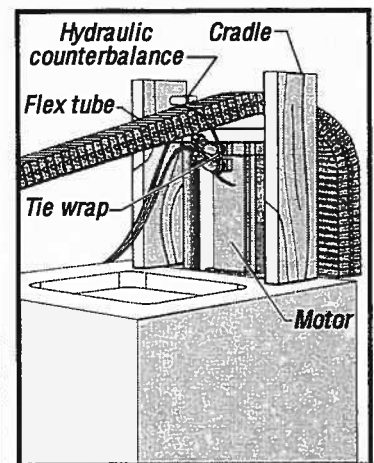
7. Press and release the HANDLE JOG button and check the screen for the "JOGGING Z AXIS HANDLE .001" message. Verify that the head will travel SLOWLY (not more than 0.001 inch per impulse — the ".001" part of the Z-axis message). If the message does not read .001, press and release the .001 button next to the HANDLE JOG button.

NOTE: The upper numbers on the buttons next to HANDLE JOG are for the jog handle use, and the lower numbers the jog speed in inches per minute when using the JOG buttons on the keypad. If the "JOGGING _" message shows the X- or Y-axis instead of Z, press and release the +Z button.

8. For the VF-0/1/2/3/4, jog the Z-axis to the top of its travel, and remove the flex tube cradle (as shown).



Flex Tube Cradle for VF-3/4



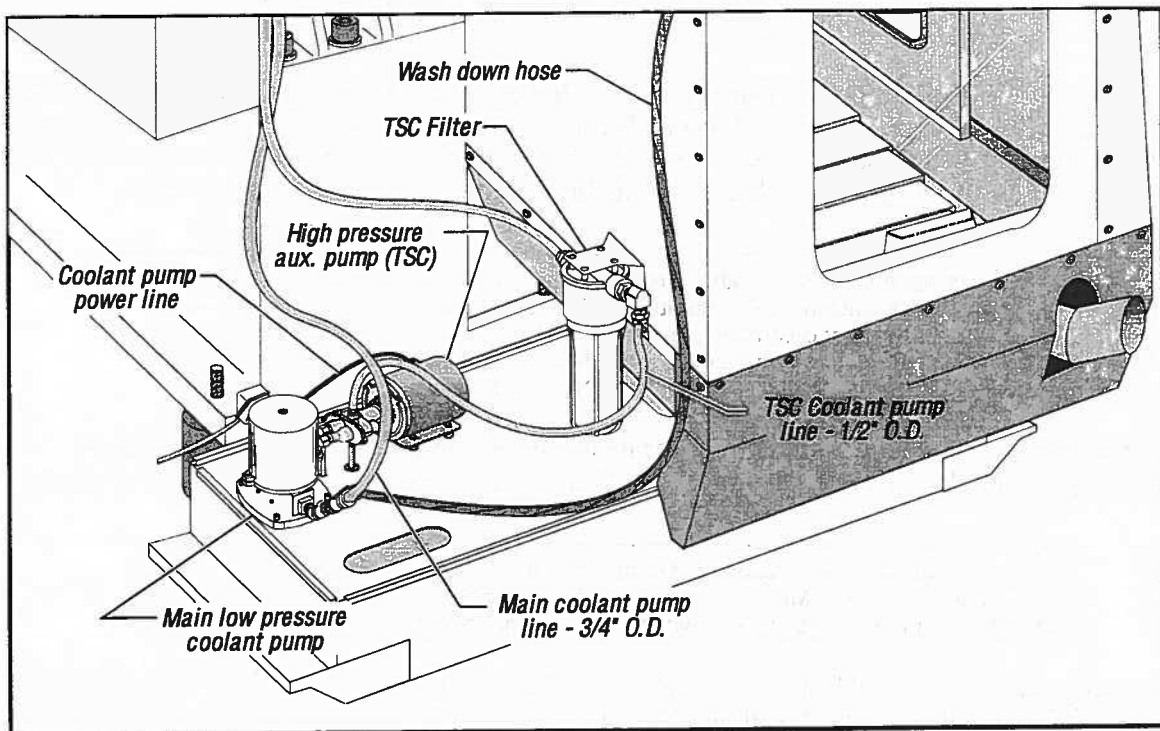
Flex Tube Cradle for VF-0/1/2

9. Once you are certain that the Z-axis is working correctly (that it operates smoothly and there are no strange noises, etc.), make sure that all alarms are clear — check for the “ALARM” message at the lower right of the screen. Next, close the doors and press and release the ZERO RETURN button followed by the AUTO ALL AXES button. The Z-axis moves up slowly. Then, after it has reached its home position, the X- and Y-axes move to their home positions.

CAUTION!

If you hear any strange noises, hit the EMERGENCY STOP button immediately and call the factory.

IMPORTANT!! To verify correct hydraulic counterbalance pressure, jog the head to the top of its travel and check pressure with those printed on tank.

COOLANT TANK INSTALLATION

1. Position the coolant tank under the left side of the machine as shown.
2. Connect the main coolant line (3/4" O.D.) to the main pump.
3. Connect the main pump power line to the outlet on the right side of the electrical panel.
4. If machine includes Through the Spindle Coolant option, attach the 1/2" O.D. coolant line to the TSC pump.

NOTE: The TSC power line is hard wired to the main pump power line.

5. Fill coolant tank with coolant (approximately 40 gallons).

NOTE: Before operating the coolant system, ensure the drain is positioned half way over tank strainer.

OPTIONAL THROUGH THE SPINDLE COOLANT STARTUP

NOTE: When machine is ready to operate, with coolant in the coolant tank, the Through the Spindle Coolant (TSC) pump must be primed according to the following procedure. This procedure is to be performed by authorized personnel only.

1. Set Parameter 236 to 12000 (msec).
2. With no tool in the spindle, switch to MDI mode.
3. Press the AUX CLNT button to turn on TSC. Wait for coolant to flow from the spindle.
4. Press the AUX CLNT button again to turn off TSC.
5. Turn the TSC system on and off twice more using the AUX CLNT button.
6. Reset Parameter 236 to 750 (msec).

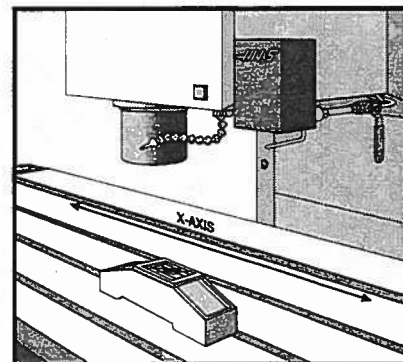
LEVELING THE VMC

Leveling of the machine is required to obtain the correct right angle geometry of the VMC's X, Y, and Z axes. Incorrect level will result in out-of-round circle milling and incorrect linear interpolation.

Leveling is done in two steps: **rough leveling** to ensure the machine is level for coolant and oil drainage, and **fine leveling** for axes' geometry. Finally, the spindle sweep is checked. Leveling is done without removing any covers.

NOTE: Many factors can affect a machine's ability to remain level — the rigidity of the floor, the stability of the support under the floor, trains or trucks passing nearby, seismic activity, and so on. Therefore, until your experience shows how often re-leveling is required, you should check the machine's level frequently after it is installed.

Use a precision bubble level with each division equal to **0.0005** inch per **10** inches, or **.05** mm per meter, or **10** seconds per division. Before starting, check the accuracy of your level. Set it on the table on the X-axis and record the reading. Then turn it **180°** and the reading should be the same. If it is not, the level is out of calibration and should be adjusted before you continue.

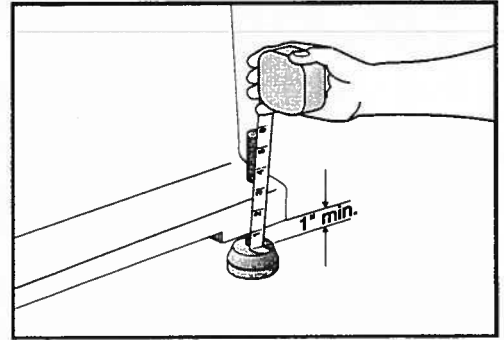


ROUGH LEVELING

NOTE: Be sure to have removed the plastic bolt covers from the retaining bolts before continuing.

NOTE: For the VF-0/1/2, it may be necessary to pull the coolant tank toward the back of the machine to access the leveling screws.

1. Screw the four leveling screws at the corners through the base (use the 3/8" allen wrench in the tool box) until the base is 2½" to 3" above the floor. That translates into a minimum of one inch of the leveling screw extending out of the bottom of the base of the machine, or one inch between the pads and the casting. Turn each screw until the tension is about the same as the tension on the other screws (it takes the same effort to turn each screw with the allen wrench).



2. Install the two center leveling screws, ensuring that they do not touch the floor.

3. Use HANDLE JOG set for 0.01 on the X- and Y- axes for the leveling procedure. (See the "Introduction" section for details on selecting jog rate and axis.) This provides a good rate of travel as you manually move the table.

4. Using the jog handle, center the table under the spindle. You do not need to move the table while rough-leveling the machine.

5. Place the level parallel to the X-axis (side-to-side) on the table and observe the bubble. If the bubble is centered, the table is level on this axis. If the bubble is off to the left of the level, it means that the left side of the table is high. And, conversely, if the bubble is off to the right, it means that the right side of the table is high.

NOTE: Each time you read the level, make sure that the bubble has steadied before you take the reading.

6. Turn the screws on the low side of the machine clockwise (screw them in) a little at a time and check the level until the bubble is centered.

NOTE: In most cases it is better to raise a side or corner than it is to lower it — when you lower a machine there is a greater risk of running out of adjustment.

7. Repeat the previous steps with the level on the Y-axis (front-to-back).

8. Continue this process until the machine is level on both axes.

NOTE: If the level is off on both axes, it indicates that one corner of the machine is high or low.

9. As the process continues, the leveling screws are turned in smaller increments — 1/4 turn, 1/8 turn, and smaller. Also, as the machine is leveled, make sure that the tension continues to be equal on the screws at all four corners.

NOTE: The following procedure for fine leveling the machine must be performed exactly as noted to ensure machine will meet all quality standards for machining operations. Failure to follow these guidelines will prevent the machine from being truly leveled and result in poor machining finishes.

FINE LEVELING

10. With the table centered, place the bubble level in the center of the table parallel to the X-axis. Using the jog handle, move the Y-axis, stopping at the front, middle, and back of the travels. The objective is to adjust the level to make the Y-axis guides parallel. The bubble level must indicate the same reading at each position (front, middle, back). Note the movement of the bubble and if the table is at front or back of travel. If the bubble moves, for example, to the right and the table is at the front of the travel, lower the right front corner adjustment screw slightly. Repeat the procedure until you get the bubble steady from front to back. This is the only leveling adjustment that can be done.

The following procedure is simply a check of machine level. If it does not meet specifications, then you must repeat this operation. Do not adjust the middle screws at this point.

IMPORTANT!!!

Refer to the Machine Inspection Report that accompanies your machine. Check your results with those of the report under the Table Travel Flatness verification. By duplicating these results exactly, you will obtain the same alignment specifications that were achieved at the factory.

11. Place a **0.0005** test indicator in the spindle and sweep a **10" diameter** circle on the table. (See the Machine Inspection Report in the manual for the results of this test at the factory.) Grease the dimple in each of the two remaining pads, locate them under the middle leveling screws, and use these screws to compensate for any error. If there is no error, tighten the screws evenly until they contact the pads.

APPENDIX A:

PREPARATION FOR SHIPPING

NOTE: See the illustrations in the "Installation" section and in Appendix B for additional information.

CAUTIONS!

1. All of the warnings and cautions in the previous sections of this manual apply to preparation for shipping — review them carefully.
2. Once the VMC is crated, the crate must be covered with waterproof tarpaulins any time there is a chance it can be exposed to rain, splashing from wet roads, or other conditions where it could get wet.
3. Parts of an unprotected VMC can be damaged by heavy tarps, indiscriminate use of tie-downs, etc. Therefore, unless a machine is being moved a short distance by professional machinery movers, it must be prepared for shipping and crated according to the following outlines.

TOOLS REQUIRED

Staple gun

Claw hammer

9/16" hex wrench

Two 3/4" wrenches (one open end or box and one ratchet)

For VF-0/1/2: A forklift capable of lifting more than 7,500 pounds, with forks at least 5' long by 6" wide.

For VF-3/4: A forklift capable of lifting more than 15,000 pounds, with forks at least 7' long by 6" wide.

For VF-3/4: A forklift capable of lifting more than 21,000 pounds, with forks at least 7' long by 6" wide.

For VF-8: A forklift capable of lifting more than 24,000 pounds, with forks at least 8' long by 6" wide.

MATERIALS REQUIRED

One crate (including pallet and 40 corner clips)

3/8" x 3" lag bolts (18)

3/8" washers (18)

6-mil plastic sheet

6-mil plastic cover

Coolant box (cardboard)

Wire ties

Foam pads

Fiber banding

1/2-13 x 10" bolts (4)

1/2" washers (4)

1/2-13 Nylock nuts (4)

16d nails as required

CLEANING

Clean the machine as necessary before preparing it for shipping.
(See the cleaning notes in the Maintenance Briefs section.)

LOCK UP THE MACHINE FOR SHIPPING

1. Remove any tools from the tool carousel and spindle.
2. Clean the interior of the enclosure, including the windows, as necessary. After cleaning, spray all metal parts around the way covers, table, spindle (including up into the spindle), with rust preventive oil. (See the cautionary notes on cleaning the machine in the "Maintenance" section.)
3. Use the jog handle to center the table under the spindle.

NEVER PUT ANY SUPPORT ON THE SPINDLE!! THIS WILL DESTROY THE SPINDLE BEARING!!!

4. Set the shipping bracket on the table, then use the JOG HANDLE to bring the head down until it rests on the bracket. Attach the shipping bracket to the table and head with the four SHCS.
5. Press EMERGENCY STOP and the machine is locked up for shipping.
6. Power down (press the POWER OFF button at upper left of control).
7. Remove the discharge tube. Remove the chip conveyor from the motor hub. Pack them both for shipping.
8. Tape the touch-up paint to the gutter of the table in the front of the machine. Place all loose items in the drain bucket. These items include the control cabinet key, tool kit, washdown kit, brush, leveling feet, and the leveling screws.

DISCONNECT SERVICES**CAUTION!**

Working with the electrical and air services required for VMC can be extremely hazardous. Make sure that pressure has been removed from the air line before you disconnect it from the machine. Similarly, the electrical power must be off and steps must be taken to ensure that it will not be turned back on while you are working with it. In most cases this means turning off a circuit breaker in a panel and then locking the panel door. However, if your connection is different or you are not sure how to do this, check with the appropriate personnel in your organization or otherwise obtain the necessary help BEFORE you continue.

1. Disconnect the air supply line at the pressure regulator (below the electrical panel on the back of the machine).
2. Turn off the electrical power to the machine at its source (see the caution at the beginning of this section).
3. Set the main switch on the machine (upper right of the electrical panel on the back of the machine) to OFF.
4. Release the three latches along the edge of the panel door and open the door.
5. Using a screwdriver to release the terminals, disconnect the three electrical leads to the terminals behind the main switch and the adjacent ground-bus connection. Pull the wires out of the top of the panel box and secure them safely.
6. Remove the air-filter screen mounted toward the bottom on the outside of the panel door (lift it up and then pull the bottom edge out). Clean it with compressed air if necessary.
7. Place the air-filter screen, along with the manuals and other documentation, inside the panel at the bottom left.

INSTALLATION

8. Set the main switch to OFF (rotate the shaft that engages the handle on the panel door counterclockwise as far as it will go and then let it snap into the OFF position). Also, set the main switch handle on the panel door to OFF. (Both the handle and the switch must be set to OFF before the door can be closed). Close the door and lock the latches.

SECURE THE FRONT

CAUTION! No heavy or abrasive ties, chains, wraps, or bindings should be placed in contact with painted or plastic parts of the machine. Damage caused by ignoring this caution is NOT covered by the warranty.

1. Make sure that the machine is locked up and remove any loose material from the table area.
2. Place a pad between the doors, pull them closed, and secure them with two wire ties through the handles.
3. Wrap some padded packing material around the pendant (control box) and secure with some heavy-duty tape. Also, place a couple inches thickness of foam or other packing material on the enclosure behind the control panel.
4. For the VF-3/4, remove the support bolt from the control arm, and swing the arm back and into shipping position. Put the shipping brace into place and secure with two bolts.

MOVE ONTO THE PALLET

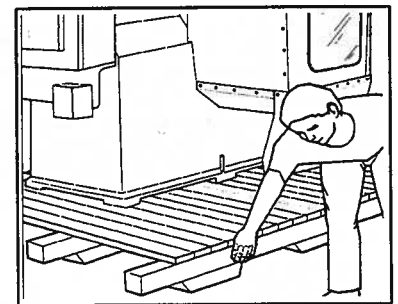
NOTE: If you did not store your shipping materials, pallets and crates are available from Haas Automation. Contact the factory for information.

CAUTION! Keep in mind when moving the VMC that much of its weight is concentrated in the column at the back. Therefore, when lifting a VF-0/1/2 from the side, it is important that the forks of the forklift be positioned as close to the back of the machine as possible without being on the pads.

CAUTION! Do not lift the machine any farther than necessary off the floor when moving it, and move as slowly and cautiously as possible. Dropping the machine, even from a height of a few inches, can cause injury, result in expensive repairs, and void the warranty.

WARNING!

For the VF-0/1/2: The only acceptable way to move a VF-0/1/2 is to pick it up from the **SIDE** with a forklift. The forklift must be capable of lifting more than 7,500 pounds. The forks must be at least 5' long by 6" wide and they must be set as far apart as possible without being on the pads. The forks must be positioned all the way to the back of the VMC and they must extend at least 3" past the far side of the machine base. Also, there must be approximately 6" clearance between the forklift and the side of the machine.



For the VF-3/4/6/8: The VF-3/4/6/8 must be lifted from the **BACK** of the machine. The forklift must be capable of lifting more than 15,000 lbs. for the VF-3/4, 21,000 lbs. for the VF-6, and 24,000 lbs. for the VF-8. The forks must be at least 7' long for VF-3/4 (8' long for the VF-6/8) by 6" wide and they must be set as far apart as possible without being on the pads. Also, there must be approximately 6" clearance between the forklift and the back of the machine.

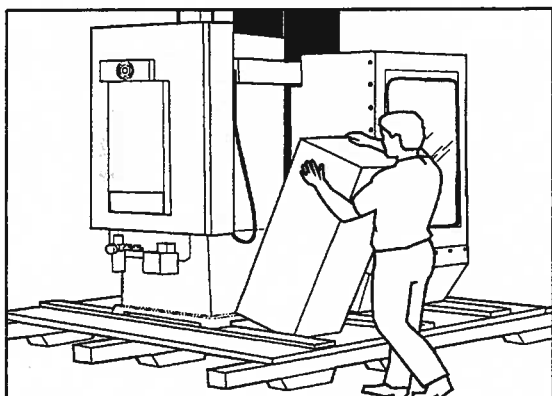
ATTEMPTING TO MOVE THE MACHINE ANY OTHER WAY MAY VOID THE WARRANTY.

CAUTION! Use caution when lifting the machine with a forklift not to damage the sheet metal aprons with the forks.

1. Lift the machine a few inches off the floor.
2. Cover the pallet with a 6-mil thick contiguous plastic sheet.

NOTE: This and the other plastic that is added later provide a shield against moisture while the machine is in transit. **HOWEVER**, the plastic wrapping is not 100 percent waterproof and the crate must be covered with waterproof tarpaulins any time there is a chance it can be exposed to rain, splashing from wet roads, or other conditions where it could get wet.

3. Move the machine over the pallet and align the leveling screws with the holes in the pallet. Note that the pallet center support is offset to one side. When the machine is placed on the pallet, the rear of the machine must be on this side of the pallet.
4. Remove the leveling screws at all four corners of the base. **For the VF-0/1/2/3/4:** Insert a plastic bolt sleeve, then a retaining bolt with washer, into each leveling screw hole. Tighten each bolt into the nut plate below the pallet. **For the VF-6/8:** For the front two holes, place a retaining bolt with washer up through the bottom of the pallet and through the base, and tighten. For the rear two holes, place a retaining bolt from the top, through the base and the plate below the pallet. Place a nut on each bolt and tighten to the plate.
5. Drain and clean the coolant system. Pack it in the box it came in and slip the box, heavy end down, into the space on the left side of the machine between the panel at the back of the machine and the enclosure.

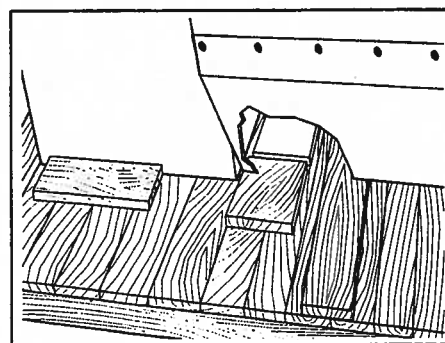


WARNING!

The box must be positioned with the heavy end down. Then slip packing material between the box and the machine at all points where there is contact. And finally, nail a 12", two-by-four cleat to the pallet with 16d nails to hold the box in place.

6. Put the leveling screws, pads, allen wrench, and screwdriver in the tool box and position it between the coolant box and the support strip toward the front of the machine. Then nail a 12", two-by-four cleat to the pallet with 16d nails to hold the tool box in place.

7. Clean the drip pan as necessary and slide it under the machine. If the machine is furnished with the drain bucket, place it at the rear of the machine, under the air/lube panel, and tie into place.



COMPLETE CRATING FOR THE VMC

1. Cover the machine with a sheet of 6-mil thick plastic. Then carefully roll the edges of the bottom plastic and the cover together and staple them to the pallet. This forms a moisture resistant cocoon around the machine. HOWEVER, as noted earlier, the plastic wrapping is not 100% waterproof and the crate must be covered with waterproof tarpaulins any time there is a chance it can be exposed to rain, splashing from wet roads, or other conditions where it could get wet.
2. Set the four sides of the crate around the machine. Note that the front and back of the pallet are shorter than the sides. Therefore, the side panels have more reinforcing strips than the front and back. Also, if you are reusing a crate, make sure that the fork location marks are in the correct location (see earlier comments).
3. Position one of the sides (longer panels) on the edge of the pallet and secure it with five 3/8" lag bolts.
4. Position an adjoining side and secure it in place with a corner clip (use a claw hammer to snap the clip into place). Then secure it with four lag bolts.
5. Repeat the above two steps for the remaining two sides--clip the corner and secure the side. (Remember to use four lag bolts for the short side and five lag bolts for the long side.)
6. Complete clipping the four corners. There should be a total of five clips at each corner.
7. Place the top panel on top of the crate and clip each side. Use five clips per side.
8. If the fork locations are not painted on the sides, add these marks (refer to earlier figure).

CAUTION!

The fork positions marked on the sides of the crate are the key to safely moving the crated VMC. If the fork positions are ignored, there is a good chance that the retaining bolts will be sheared off by the forks and also that the machine will tip over when it is picked up.

This completes packing the VMC for shipping. If you have questions, contact your dealer or the Haas Automation factory.

APPENDIX B:

SHIPPING THE MACHINE WITHOUT A PALLET

To ship a VMC a short distance, it may not be necessary to mount it on a pallet and crate if you employ professional machinery movers. Nevertheless, you must make specific preparations for moving it and certain precautions are necessary.

NOTE: We strongly recommend using an air-ride suspension truck to minimize jarring the machine. It is well worth the few extra dollars involved. And make sure to ask specifically for air-ride suspension – not all riggers have trucks of this type.

CAUTIONS!

1. All of the warnings and cautions in the previous sections of this manual apply to shipping without a pallet — review them carefully.
2. Parts of an unprotected VMC can be damaged by heavy tarps, indiscriminate use of tie-downs, etc. Therefore, it must be shipped in accordance with the following outlines.

TOOLS REQUIRED

9/16" Hex Wrench

3/4" Wrench (Open-end or Box-end)

For VF-0/1/2: A forklift capable of lifting more than 7,500 pounds, with forks at least 5' long by 6" wide.

For VF-3/4: A forklift capable of lifting more than 15,000 pounds, with forks at least 7' long by 6" wide.

For VF-6: A forklift capable of lifting more than 21,000 pounds, with forks at least 7' long by 6" wide.

For VF-8: A forklift capable of lifting more than 24,000 pounds, with forks at least 8' long by 6" wide.

MATERIALS REQUIRED

Four metal tie-downs

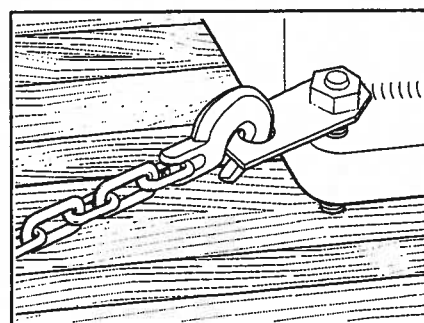
Four 3/4-11 nuts (in addition to the four nuts that come with the machine)

Two 4 x 4's at least 5' long

PROCEDURE

1. Prepare the machine as described in Appendix A up to the point of putting it on a pallet. Make sure the spindle is on tool number one. Replace and secure the shipping bracket -- even for the shortest moves. Tie-wrap the doors together and stow the air filter from the control panel door (on the back of the machine) inside the panel.

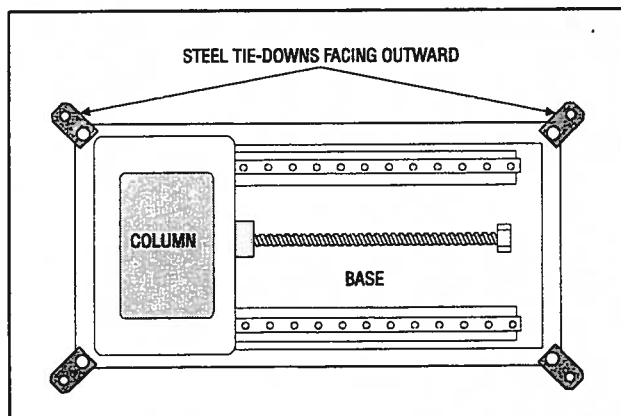
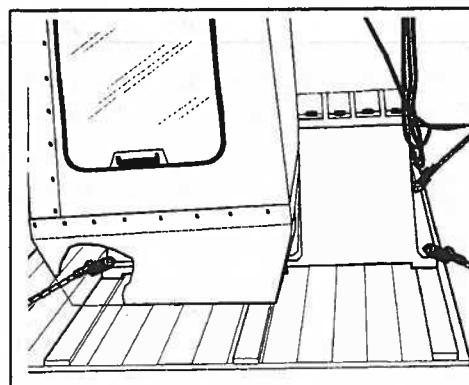
2. To transport the machine without a pallet, you need the four metal tie-downs supplied in your tool box. (Additional tie-downs can be purchased from Haas if you have lost yours.) After raising the machine with a forklift, turn the four corner levelling adjustment screws clockwise, with the 9/16" hex wrench, until they protrude at least 1/2" through the bottom of the machine. Tighten the 3/4-11 nuts until the levelling screws are secured in the casting. Place a metal tie-down on each of the four levelling screws (the small hole in the tie-down slips over the top of the screw and rests on the top of the 3/4-11 nut). Position the tie-downs so they are at 45° angles to the front, back, and sides of the machine and secure them on the screws with additional 3/4-11 nuts.



INSTALLATION

3. Place two 4 x 4's that are least 5' long on the truck. Lower the VMC onto the truck and align the two pieces of lumber under the leveling screws at the corners of the machine. Remove the forklift and have the rigger secure the machine using the large holes in the four tie-downs as the attachment points for the riggers' chains. When the chains are tightened, the leveling screws dig into the 4 x 4's to help hold the machine in place.

CAUTION! Do not cross the chains.



4. Using the four corner tie-downs, the machine can be secured without dragging chains across its top. It is a fast, easy, and secure transportation method that protects against damage. If there is limited access, wrap perimeter of base above feet with chain tension as necessary. Be sure to explain to the rigger that this is how you want the machine to be moved **BEFORE** he does something else.

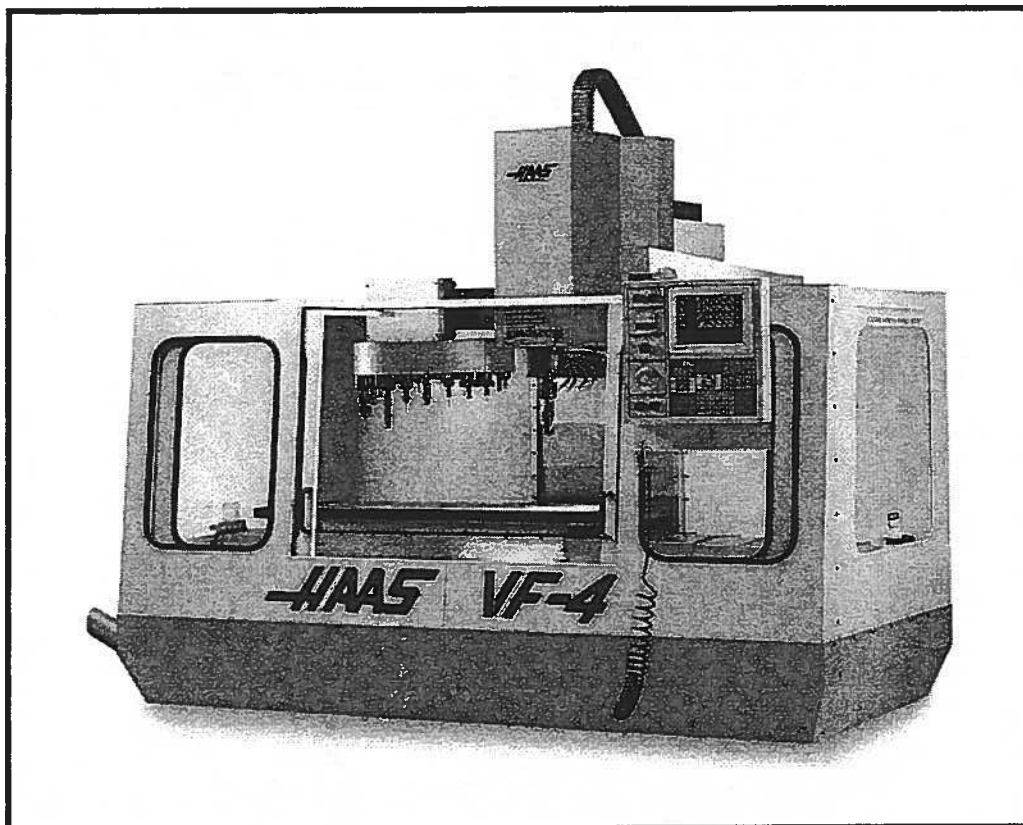
5. If you wish to tarp the machine, you should be aware of a few potential trouble spots. If a tarp is simply placed over the machine and tightened down, it can do considerable damage, such as bending the conduit carrier for the headstock, wearing paint off the machine, distorting the enclosure, and damaging the clear plastic parts. If you must use a tarp, make sure that it is supported off the machine by a wooden frame or other secure structural means.

6. If the fork locations are not painted on the sides, refer to the earlier figures and add them. **CAUTION!** These fork positions are very important, and if they are ignored, there is a good chance that the retaining bolts will be sheared off by the forks and that the machine will tip over when it is lifted.

This completes packing the VMC for shipping. If you have questions, contact your dealer or the Haas Automation factory.

VMC CABLE LIFT**INSTRUCTIONS**

**PLEASE READ THIS ADDENDUM IN ITS ENTIRETY
BEFORE ATTEMPTING TO LIFT VMC.**



It is MANDATORY that preparation and lifting of the VMC be performed by PROFESSIONAL RIGGERS and that this addendum be used as a reference guide only.

LIFT KITS for the HAAS VERTICAL MACHINING CENTERS

VF-0/1/2: (LIFTING SUPPLIES IN TOOLKIT 30-7900E)

- (1) 5/8-11 x 1 1/4" EYE BOLT
- (1) 5/8" FLAT WASHER
- (2) 3/4-10 x 1 1/4" EYE BOLTS
- (2) 3/4" FLAT WASHERS

VF-3/4: (LIFTING SUPPLIES IN TOOLKIT 30-9900E)

- (2) 90° LIFT BRACKETS
- (6) 1/2" FLAT WASHERS
- (2) 1/2-13 x 1 1/2" SHCS
- (2) 3/4-10 x 1 1/4" EYE BOLT
- (2) 3/4" FLAT WASHERS

VF-6/8: (LIFTING SUPPLIES IN TOOLKIT 30-7910E)

- (2) 1 1/8 - 7 x 2 3/4 EYEBOLT
- (1) BAG
- (1) INSTRUCTIONS

NOTE: PRIOR TO JAN. 1996 , 1- 8 EYE BOLT, HEX NUT (P/N 46-1696) AND HOLE PLUG (P/N 22-9918) ARE REQUIRED FOR THEOSE MACHINES WITH EYEBOLT THROUGH HOLES.

IMPORTANT!

EYEBOLT WORK LOAD RATINGS

EYEBOLT SIZE	RATING
1/2-13	2,800 lbs.
5/8-11	5,100 lbs.
3/4-10	7,000 lbs.
1 - 8	9,000 lbs.
1 1/8 - 7	12,000 lbs.

LIFTING PRECAUTIONS!

IMPORTANT: The chains attached to the top of the column are the main lifting points and should remain vertical to the floor throughout the lift.

The front chains attached to the base are to help stabilize the machine and should not be used to carry the main load.

When lifting the machine, slowly pull up on the chains until taut, then lift the machine off the floor, ensuring the machine remains balanced.

Before installing the lift kit or lifting the machine, ensure the machine is prepared for shipping as noted in the **PREPARATION FOR SHIPPING** section in the Installation Manual.

INSTALLING THE LIFT KIT

VF-0/1/2

1. Place a washer on the 5/8"-11 x 1-1/4" eye bolt and mount it to the front Y-axis way cover, as shown below. Hand tighten and orient the bolt **vertically** as shown.

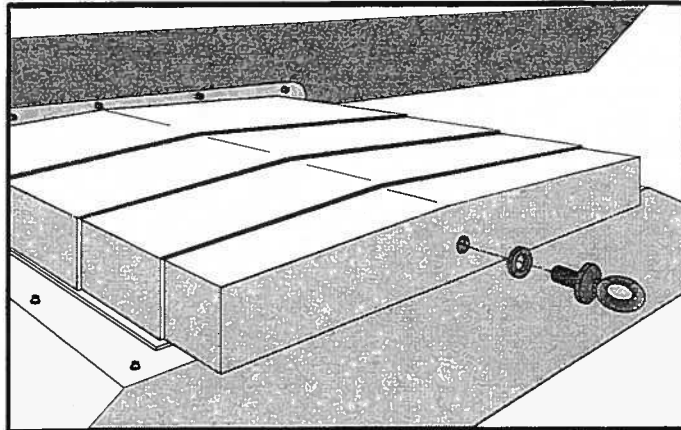


Fig. 1-1 Eyebolt mounting location for VF-0, VF-1, and VF-2.

2. Place a washer on each of the 5/8" x 1 1/4" eyebolts and mount to the top of the column as shown below. Tighten securely.

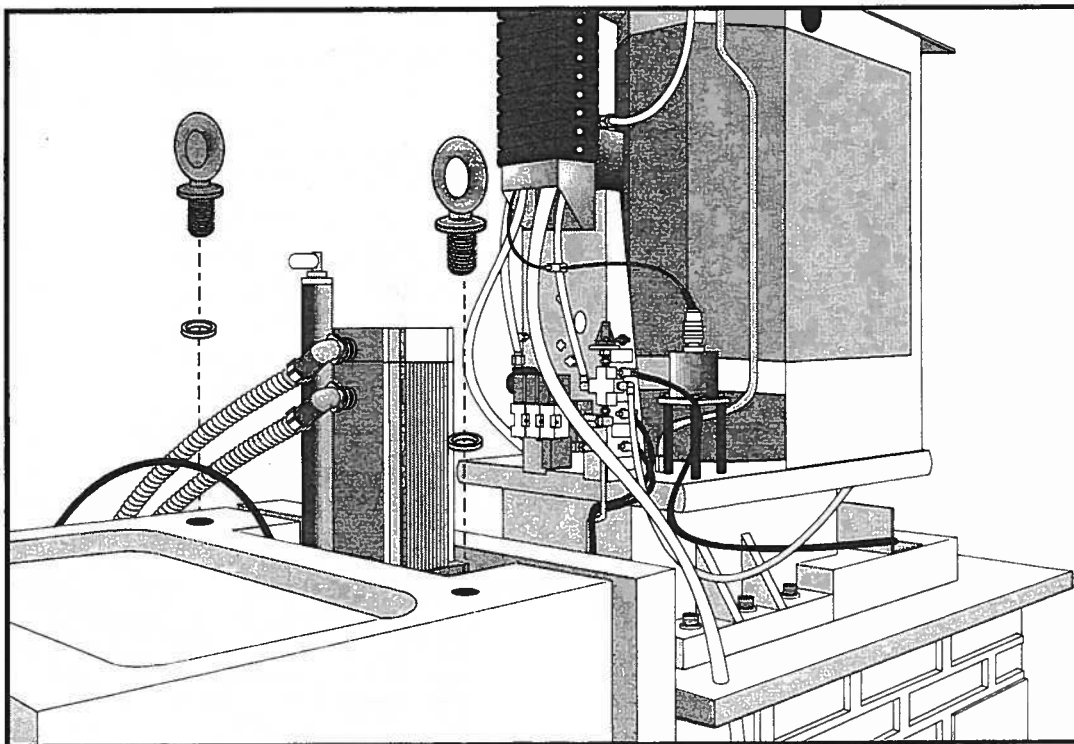


Fig. 1-2 VF-0/1/2 eyebolt locations.

INSTALLING THE LIFT KIT

VF-3/4

3. Mount the lifting brackets to the VMC's base casting as shown below. Tighten Securely.

NOTE: (2) WASHERS UNDER BRACKET
(1) ABOVE BRACKET

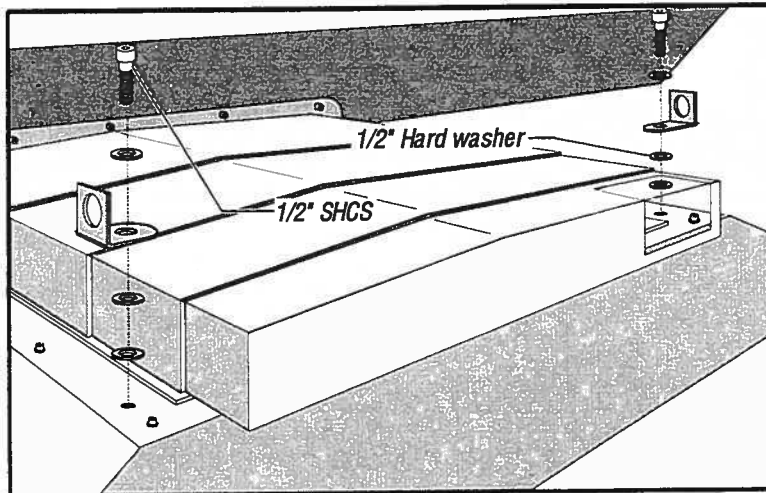


Fig. 1-3 Lifting bracket mounting location for VF-3 and VF-4.

4. Install the two (2) $\frac{3}{4}$ -10 x $1\frac{1}{4}$ " eyebolts to the column casting as shown below. Tighten securely.

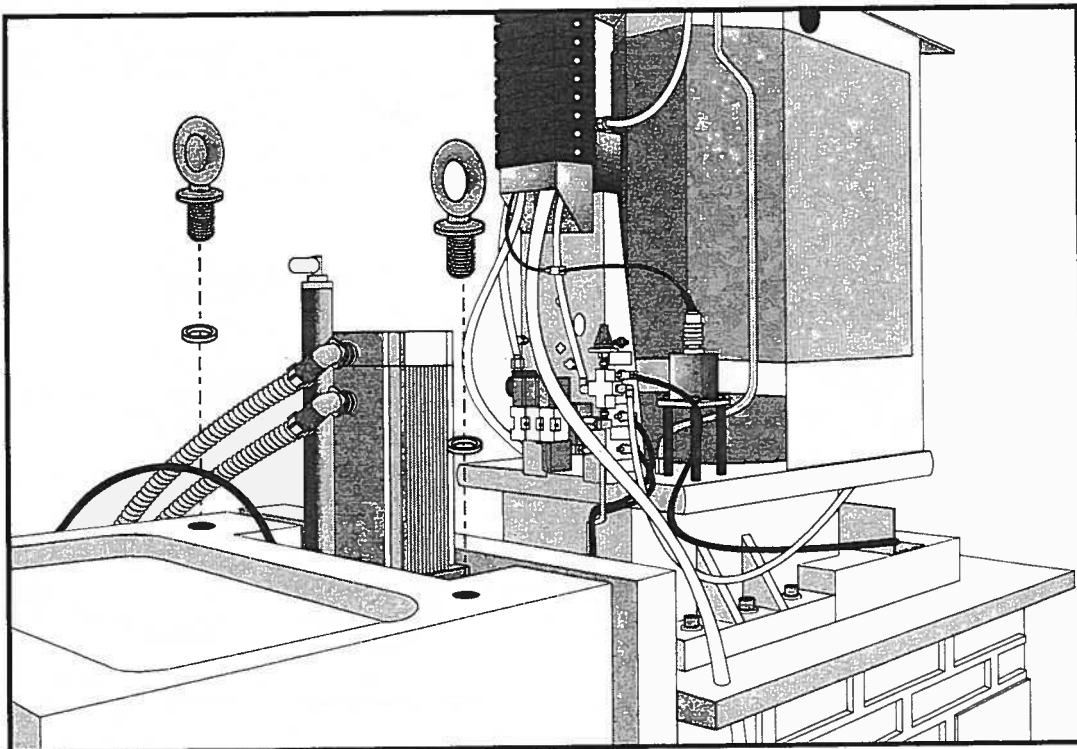


Fig. 1-4 VF-3/VF-4 eyebolt locations.

INSTALLING THE LIFT KIT

SINGLE LIFTING POINT VF-0/1/2/3/4

5. Attach the lifting chain and hooks to the lifting points as shown below, ensuring that the chain does not come in contact with the top door beams or rub against any part of the machine's enclosure.

A *minimum* chain height requirement is given, (see illustrations below), in order to avoid contact between the front chains and enclosure. The distance given is from the top of the column casting to the single lifting point.

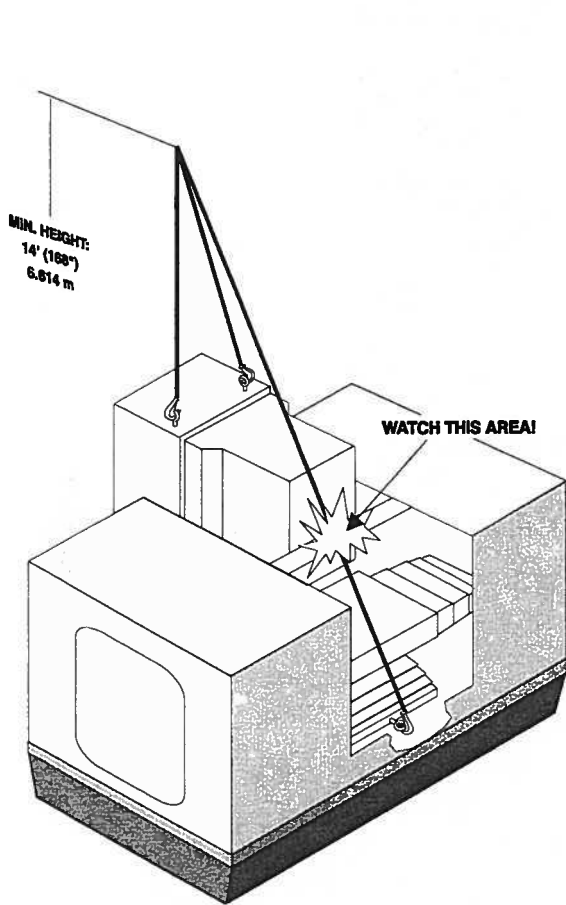


Fig. 1-5 VF-0/1/2 lifting positions.

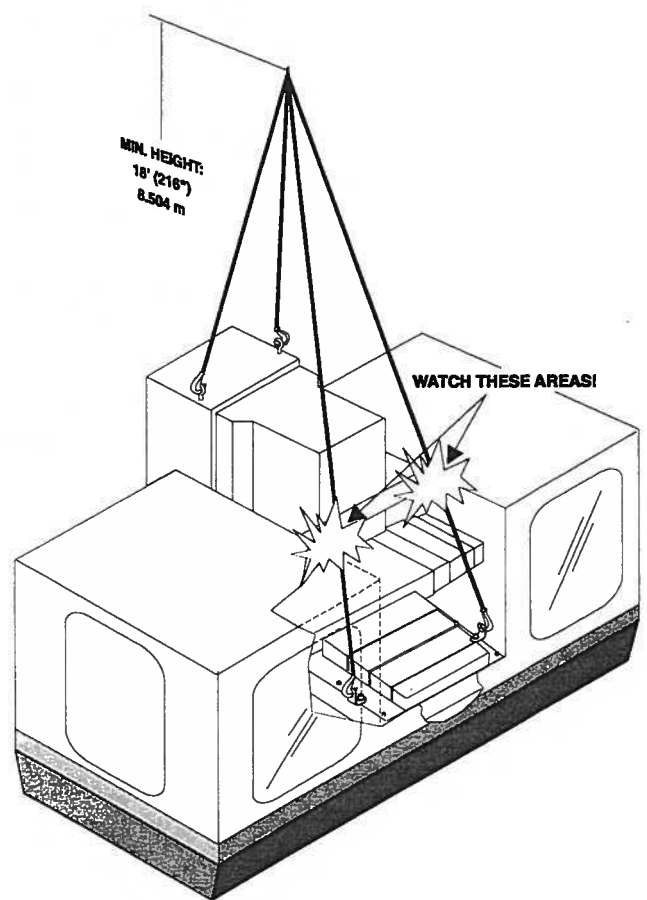


Fig. 1-6 VF-3/4 lifting positions.

INSTALLING THE LIFT KIT

VF-6 / 8

- * On machines with eyebolt through holes, remove the eyebolt plugs on the base casting.
- 6. Install the two 1 1/8" - 7 eyebolts to the base casting as shown below and tighten. Eyebolts should be facing towards the front of the machine when tightened.

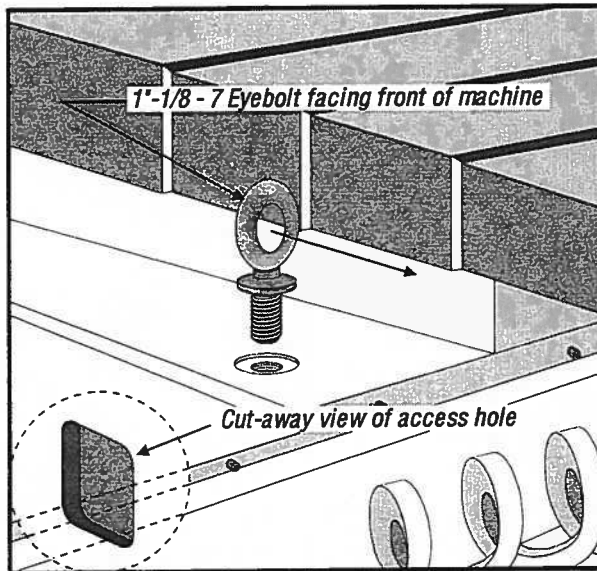


Fig. 1-7 Eyebolt and front access hole for VF-6 / 8.

- 7. These specifications are an approximation of the weight distribution and chain length needed to lift to VF-6.

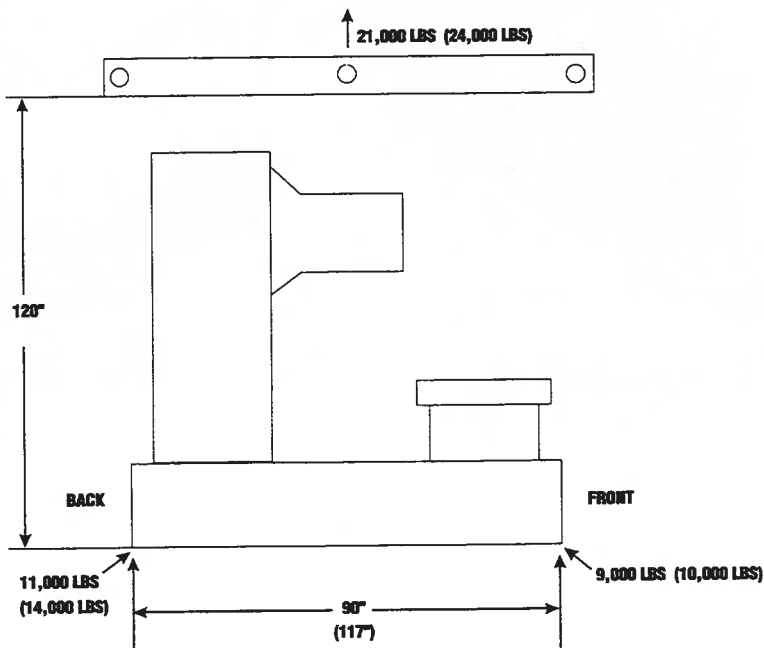


Fig. 1-8 VF-6 (VF-8) weight / height specifications.

INSTALLING THE LIFT KIT**LIFTING POINTS VF-6 / 8**

8. Attach the lifting chains and hooks to the lifting points, with the rigging beam, spreader bar, and spacer block in place. Position the spreader bar so the lifting chains do not come into contact with the waycovers.

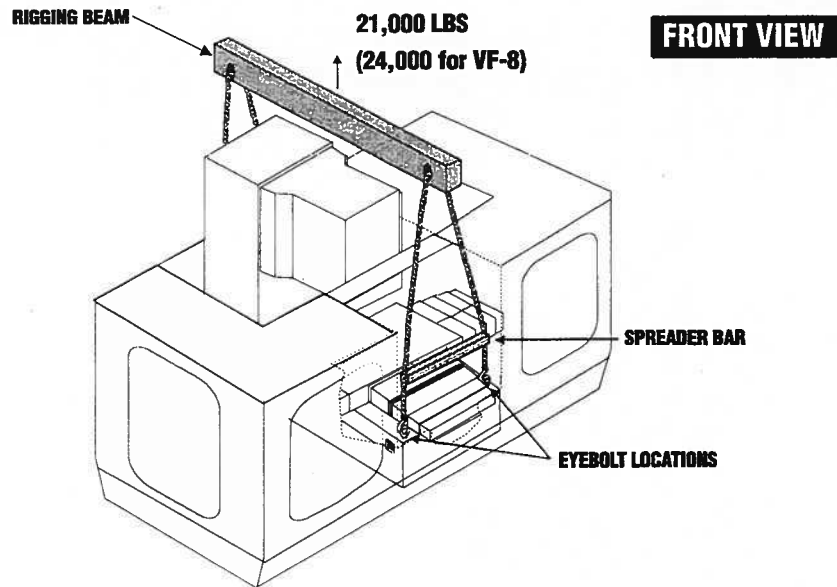


Fig. 1-9 VF-6/8 lifting positions.

9. Use the spacer block to keep the chains from damaging the rear conduit lines.

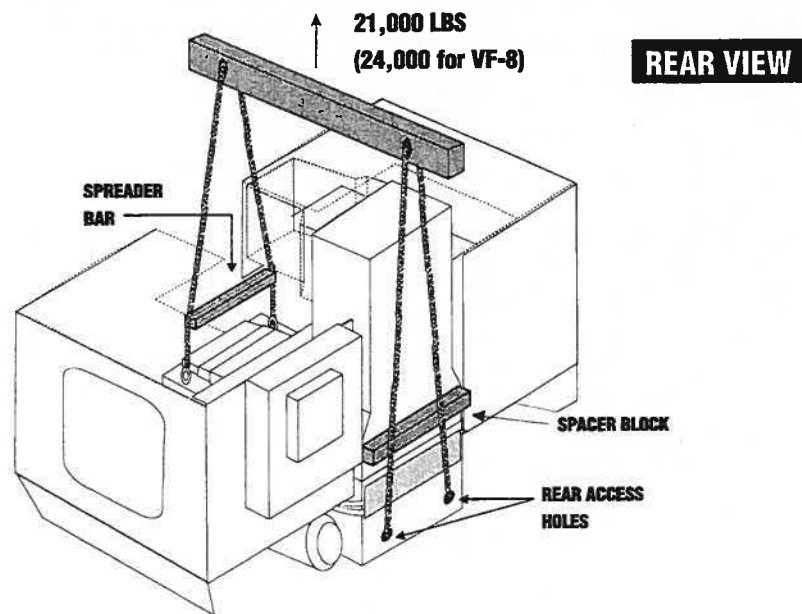


Fig. 1-10 VF-6/8 lifting positions.

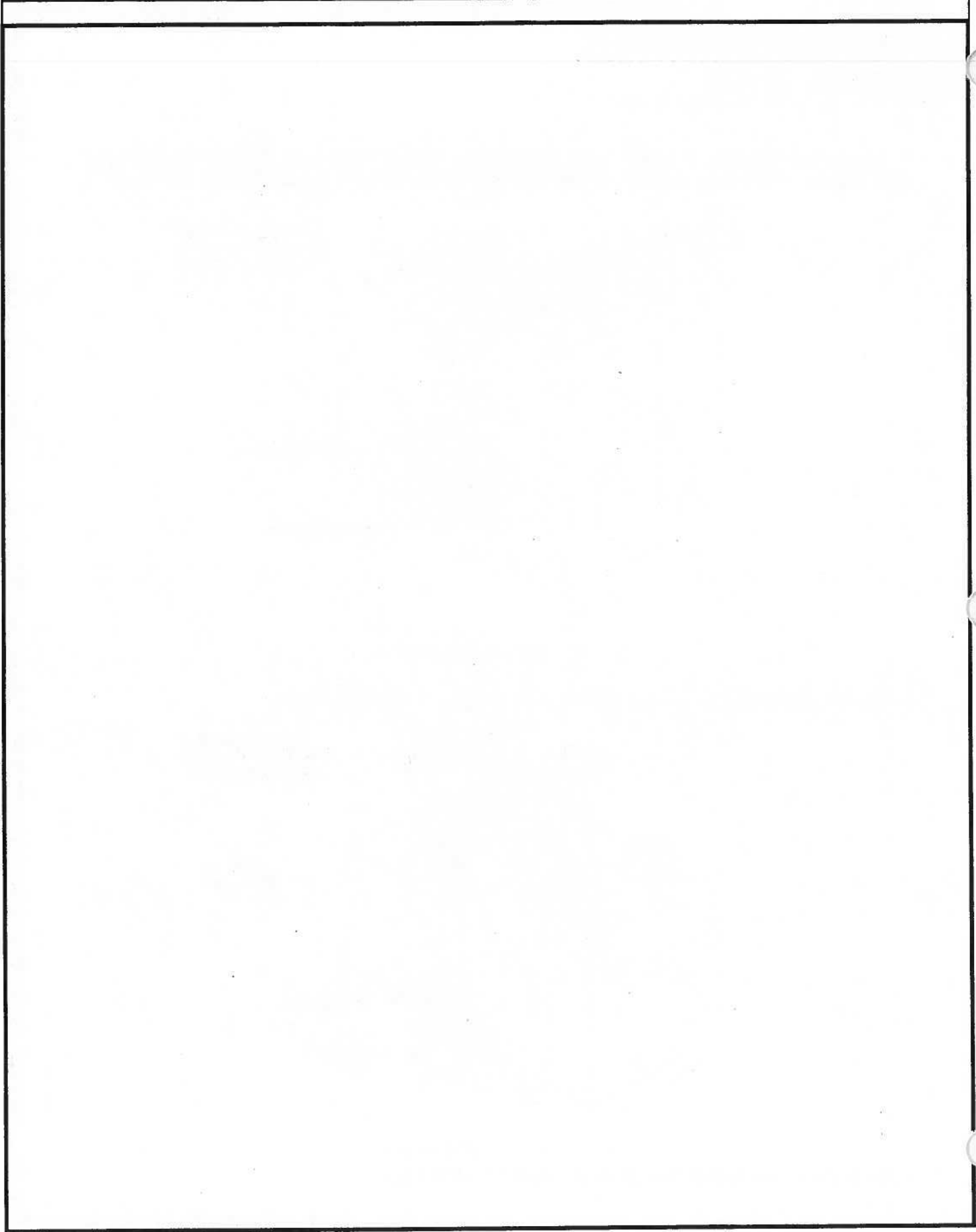
* On machines with eyebolt through holes, reinstall the hole plugs.

VF-SERIES

OPERATORS MANUAL



INSTALLATION



CHAPTER VI

TECHNICAL REFERENCE

1. TOOL CHANGER

The tool changer is an all electric fixed shuttle type. Tools are always loaded through the spindle and should never be installed directly in the carousel in order to avoid crashes. The pocket open to the spindle must always be empty in the retracted position. All wiring to the tool changer goes through connector P6 on the side of the control cabinet.

The tool holders used are CT #40 taper, V flange, commonly called "CT 40". Use A "45 Degree, P40T Type 1, inch threads" pull stud built to JMTBA standard "MAS 403-1982". This pull stud is characterized by a long shaft and a 45° shoulder under the head. Do not use the short shaft or pull studs with a sharp right angle (90°) head as they will not work and will cause serious damage.

Tool holders and pull studs must be in good condition and tightened together with wrenches or they may stick in the spindle taper. Clean the tool tapers with a lightly-oiled rag to leave a film to prevent rusting. Tools that make a loud bang when being released indicate a problem and should be checked before serious damage to the shuttle occurs. When the TOOL RELEASE button is pressed, the tool should be pushed out of the spindle by a small amount (approximately .07"). This is an indication that the pull stud is correctly touching the release mechanism.

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

If the shuttle should become jammed, the control will automatically come to an alarm state. To correct this, push the EMERGENCY STOP button and remove the cause of the jam. Push the RESET key to clear any alarms. Push the ZERO RETURN and the AUTO ALL AXES keys to reset the Z-axis and tool changer. Never put your hands near the tool changer when powered unless the EMERGENCY STOP button is pressed.

FU1 on the I/O PCB or the Power PCB is a fuse for the tool changer motors. It might be blown by an overload or jam of the tool changer. Operation of the tool changer can also be interrupted by problems with the tool clamp/unclamp and the spindle orientation mechanism. Problems with them can be caused by low air pressure or a blown solenoid circuit breaker CB4.

When a tool change operation is performed, the following sequence of events occurs:

- 1) Z axis moves up to machine zero,
- 2) If the spindle is turning, it is commanded to stop,
- 3) Spindle oriented to Tool Changer,
- 4) Turn TSC pump off, (optional)
- 5) Pre-charge is on,
- 6) Shuttle moves in to grab tool,
- 7) Tool unclamps,
- 8) Z axis moves up,
- 9) Tool Changer rotates,
- 10) Z axis moves down,
- 11) Tool clamps,
- 12) Pre-charge off,
- 13) Shuttle moves out.
- 14) TSC on (optional)

1.1 TOOL CHANGER LUBRICATION

Place a few drops of lubricating oil on the outside edge of the Geneva wheel star and guide rails of the tool changer and run through all tools.

1.2 SHUTTLE IN/OUT MOTOR

A DC brush motor is used to move the tool changer assembly towards and away from the spindle. This is called the shuttle. The motor is geared down to a low RPM and then connected to an arm that rotates through 180° and pushes the shuttle in and out.

NOTE: This motor should never be disassembled

1.3 TURRET ROTATION MOTOR

A DC brush motor is used to rotate the tool turret between tool changes. This motor is geared down to a low RPM and connected to a Geneva mechanism. One revolution of the Geneva mechanism moves the tool turret one tool position forward or backward.

NOTE: This motor should never be disassembled.

2. TOOL CLAMP/UNCLAMP

The tool holder draw bar is held clamped by spring pressure. Air pressure is used to release the tool clamp. When the tool is unclamped, air is directed down the center of the spindle to clear the taper of water, oil, or chips. Tool unclamp can be commanded from a program (but this is quite dangerous), from the keyboard, and from the button on the front of the spindle head. The two manual buttons only operate in MDI or JOG modes.

2.1 TOOL CLAMP/UNCLAMP AIR SOLENOIDS

A single solenoid controls the air pressure to release the tool clamp. This corresponds to relay K15. When the relay is activated, 115V AC is applied to the solenoid. This applies air pressure to release the tool. Relay K15 is on the I/O PCB. Circuit breaker CB4 will interrupt power to this solenoid.

2.2 TOOL CLAMP/UNCLAMP SENSE SWITCHES

There are two switches used to sense the position of the tool clamping mechanism. They are both normally closed and one will activate at the end of travel during unclamping and the other during clamping. When both switches are closed, it indicates that the draw bar is between positions.

A tool change operation will wait until the unclamped switch is sensed before the Z-axis pulls up from the tool. This prevents any possibility of breaking the tool changer or its support mounts.

The diagnostic display can be used to display the status of the relay outputs and the switch inputs.

The Precharge and Through the Spindle Coolant system applies low air pressure and releases the clamped switch.

2.3 REMOTE TOOL UNCLAMP SWITCH

The Remote Tool Unclamp switch is mounted on the front of the cover to the spindle head. It operates the same as the button on the keyboard. It must be held for ½ second before the tool will be released and the tool will remain released for ½ second after the button is released.

While the tool is unclamped, air is forced down the spindle to clear chips, oil, or coolant away from the tool holder.

3. SPINDLE OPERATION

Spindle speed is selectable from 1 to 7500 RPM (10,000 RPM for 10K spindles). When using the Through the Spindle Coolant option, the maximum spindle speed is 7500 RPM for all spindles. For two-speed machines, speeds at and below 1500 RPM automatically select low gear. Speeds at and above 1501 RPM automatically select high gear. For machines equipped with a 10K spindle, speeds at and below 2200 RPM automatically select low gear, and speeds above 2201 RPM automatically select high gear. Spindle speed accuracy is best at the higher speeds and in low gear.

The spindle is hardened and ground to the precise tool holder dimensions providing an excellent fit to the holder.

3.1 SPINDLE ORIENTATION

Orientation of the spindle is automatically performed for tool changes and can be programmed with M19. Orientation is performed by turning the spindle slowly until an air pressure driven pin drops into a detent and locks the spindle in place. This pin is located behind the spindle motor and above the gear box. If the spindle is oriented and locked, commanding spindle forward or reverse will release the lock.

3.2 SPINDLE ORIENTATION LUBRICATION

The spindle orientation mechanism does not require regular lubrication.

3.3 SPINDLE ORIENTATION AIR SOLENOID

A solenoid controls the air valve supplying pressure to the orientation lock pin. The diagnostic display can be used to display the status of the relay output and the switch inputs. Circuit breaker CB4 will interrupt power to this solenoid.

3.4 SPINDLE ORIENTATION SEQUENCE

When spindle orientation is commanded, the following sequence of operations occurs:

- 1) If the spindle is turning, it is commanded to stop,
- 2) Pause until spindle is stopped,
- 3) Spindle orientation speed is commanded forward,
- 4) Pause until spindle is at orientation speed,
- 5) Command spindle lock air solenoid active,
- 6) Pause until spindle locked status is active and stable,
- 7) If not locked after time-out time, alarm and stop.

4. SERVOS BRUSH / BRUSHLESS

4.1 SERVO ENCODERS (BRUSH)

Attached to each DC servo motor, there is an incremental encoder that is 2000 lines per revolution. These encoders also supply a Z channel pulse once per revolution. The encoders and Z channel are continuously monitored to ensure the number of pulses matches for each revolution of the motor. If the encoders become contaminated, these pulse counts will be wrong and an alarm will be generated. This ensures that the data from the encoders is reliable. There can never be a loss of servo position due to accumulated encoder errors. The alarms generated will indicate that either the Z pulse occurred and the encoder pulse was wrong or, after one and one half motor revolutions, the Z pulse did not occur.

Encoder faults can be caused by contamination of the encoder or by a wiring problem. If the encoder is contaminated, it must be replaced. Wiring problems may be a broken wire, shorted wire, or missing shield. All wires to the encoder are enclosed in their own shielded cable. In addition, all power wires to the motor are enclosed in a separately shielded cable. Failure of either of these shields may cause noise in the encoder circuits and result in the encoder fault alarms.

Never connect or disconnect the servo motor cables with the control powered as this will cause an apparent encoder fault.

The servo motor encoders are differential line drivers. This means that the A, B, and Z signals are transmitted to the control as signal pairs. A cable test is performed on these signals to ensure the differential pair are always present.

4.2 SERVO CHARACTERISTICS (BRUSH)

This machine is not capable of instantly changing speed. That is, it takes some non-zero time to accelerate and decelerate. Acceleration and deceleration in this machine have both a constant accel/decel mode and an exponential mode. Constant acceleration is used at the beginning of a rapid move and at the end of any move whose speed exceeds the exponential accel/decel time constant.

Constant acceleration is a type of motion when the amount of speed change over time is constant. This constant is set by Parameters 7, 21, 35, and 49. It has units of encoder increments per second per second.

Exponential acceleration and deceleration is a type of motion where the speed is proportional to the distance remaining in a programmed travel. The exponential accel/decel time constant is set by Parameters 113, 114, 115, and 116. It has units of 0.0001 seconds. The speed limit at which exponential accel/decel is not available is defined by the relationship between Parameters 7 and 113 (for the X-axis). Thus if Parameter 7 is 1200000 steps/sec/sec and Parameter 113 is 750 (0.075 seconds); the maximum velocity for accurate interpolation should be:

$$1200000 \times 0.075 = 90000 \text{ steps/second}$$

For a 2000 line encoder and 6 mm screw, this would be:

$$60 \times 90000 / 33867 = 159 \text{ inches/minute}$$

In the normal feed cutting mode, with G64 active, giving continuous cutter motion, deceleration of the axes in motion begins at some distance away from the end point. If look-ahead has provided another motion, the acceleration for that motion will begin at the same instant. This means that two motions, at right angles to each other, will not produce a perfectly square corner. The corner will be rounded. It also means that if the two motions are parallel or nearly parallel, there will be a smooth transition from one stroke to the next.

Rapid moves have a slightly different operation when continuous cutter mode is active. Acceleration for the next motion is started when the axes being moved all fall within the "In Position Limit" Parameters 101, 102, 103, and 104. These parameters have units of encoder steps. Rapid moves will also decelerate at the constant accel/decel limit until the speed drops below that for exponential accel/decel (see example above giving 159 inches per minute). Parameter 57 can be used to override this.

To prevent the rounding of corners, you can specify exact stop either with G09 (non-modal) or with G61 (modal). When either of these is active in a motion, all of the axes are brought to an exact stop, at zero speed, before the next motion is started.

The tool path in a circular move (G02 or G03) is not changed by the exponential acceleration/deceleration so there is no error introduced in the radius of the cut unless the speed exceeds that for exponential accel/decel (see example above giving 159 inches per minute).

4.3 SERVO DRIVE ASSEMBLY (BRUSH)

The servo drive assembly is on the left side of the main control cabinet and about halfway down. Never work on the servo drive assembly until the small red CHARGE light goes out. This light is at the top of the circuit card at the center of the assembly. Until this light goes out, there are dangerous voltages in the assembly EVEN WHEN POWER IS SHUT OFF. This assembly contains four servo drive cards, a Servo Distribution card, and a fan.

4.4 160 VOLT DC POWER SUPPLY (BRUSH)

The Servo Distribution card contains a DC power supply that produces an unregulated voltage between 145 and 175 volts. This is derived from the three-phase 115V AC coming from transformer T1. The nominal 160V DC is supplied to the four servo drive cards for the X, Y, Z, and A axes and to the tool changer. This supply is filtered by two capacitors in parallel for a total of 4000 Mfd. A soft charge-up of these capacitors is provided by a small resistor that is bypassed by a relay when the servos are on.

The negative side of the 160V power supply is always connected to chassis ground. This means that when the relays on SDIST are released, all DC power is disconnected and the drives are safe. This also includes the tool changer that uses the 160V buss to drive the tool changer motors.

The minimum DC buss voltage is 145V and anything lower will result in an alarm. The maximum voltage is 185V and anything above this will cause heating of the servo regen load resistor. Anything above 190V will cause an alarm.

4.5 SERVO COOLING FAN (BRUSH)

There is a cooling fan on the servo drive assembly to help cool the servo drive cards. It blows air up past the servo drive cards in order to support convection cooling. The fan power is supplied from SDIST by P7.

4.6 SERVO DISTRIBUTION PCB (SDIST)

The Servo Distribution PCB is used to provide the 160V DC buss for the servo drives, the low voltage AC power for the drives, and to monitor the supply voltage for the servos.

There are three pots on this card. They are:

- R2 This pot adjusts the buss voltage at which the regen load resistor is applied as a load to the power supply. This will consume any excess power caused by the regenerative effects of decelerating the servo motors. This should be set to turn on the load between 183 and 187V DC.
- R11 This pot adjusts the fraction of the buss voltage that is sent to the Motor Interface PCB A-to-D converter. This is a full scale 5V input and the program will interpret full scale as 200V on the buss.
- R15 This pot adjusts the voltage at which an overvoltage alarm discrete is generated. This should be set to alarm between 188 and 192V DC (about 265 AC).

The red "CHARGE" LED is also mounted on the SDIST PCB. It indicates that the supply capacitors still contain a charge. The discharge resistors provide a load through this LED. It will dim and appear off when the voltage is below 20 volts.

The connectors on the SDIST PCB are:

- P1 Low voltage AC power to X drive card (570)
- P2 Low voltage AC power to Y drive card (580)
- P3 Low voltage AC power to Z drive card (590)
- P4 Low voltage AC power to A drive card (600)
- P5 12V DC from power supply (860A)
- P7 115V AC to fan
- P8 160V DC supply to tool changer(80)
- P9 Voltage monitor to A-D (980)
- P10 Regen load resistor (920)
- P11 Relay #1 contacts from IOPCB (110)
- P12 Overvoltage status to IOPCB (970)
- P13 Ground fault detect signal to IOPCB (1060)
- TB1 Three phase 115V AC to SDIST
- TB2 +160V DC and return to each servo drive card

There are three fuses mounted on the SDIST PCB; FU1 and FU2 protect the primaries of the fan and transformers. They are ½ amp, 240V AC, AGC type. FU3 protects the regenerative load circuit from a short circuit.

4.7 SERVO DRIVE PCB'S (DRIVER)

The servo drive PCB's are H drive with PWM control. There are eight states used in the H drive providing free-wheeling current during PWM and very low current ripple. The PWM frequency is 16 kHz. All drive cards are current limited at 20 to 22 amps. They operate from a nominal supply voltage of 160 volts. The peak power output is thus about 3000 watts, or 4 H.P. The continuous power output is, however, limited by a microprocessor based fuse setting, overcurrent shutdown, and motor thermal protection. Short circuit protection is provided by the drive card and, if sustained for over 0.01 second, the microprocessor will shut the servo drives off and generate an alarm.

The motor output circuit is fuse protected at 20 amps but this will only blow if there is a drive failure as the current limit circuit is much faster than the fuses.

The PWM signal is provided by the Motor Interface PCB along with direction and H drive state control. The processor also monitors the overcurrent status from the drive card.

The connectors on the servo drive cards are:

- P8 160V DC from SDIST PCB
- P1 low voltage AC power from SDIST PCB
- P3 PWM and H drive control signals
- P2 Power connection to servo motor

There are three fuses on each servo drive card. One is in series with each leg of the servo motor. These fuses are type ABC and are rated at 20 amps, 200V DC. A third fuse on each driver card limits the plus (+) side of the power supplied to each card; this fuse is an ABC, 250V,

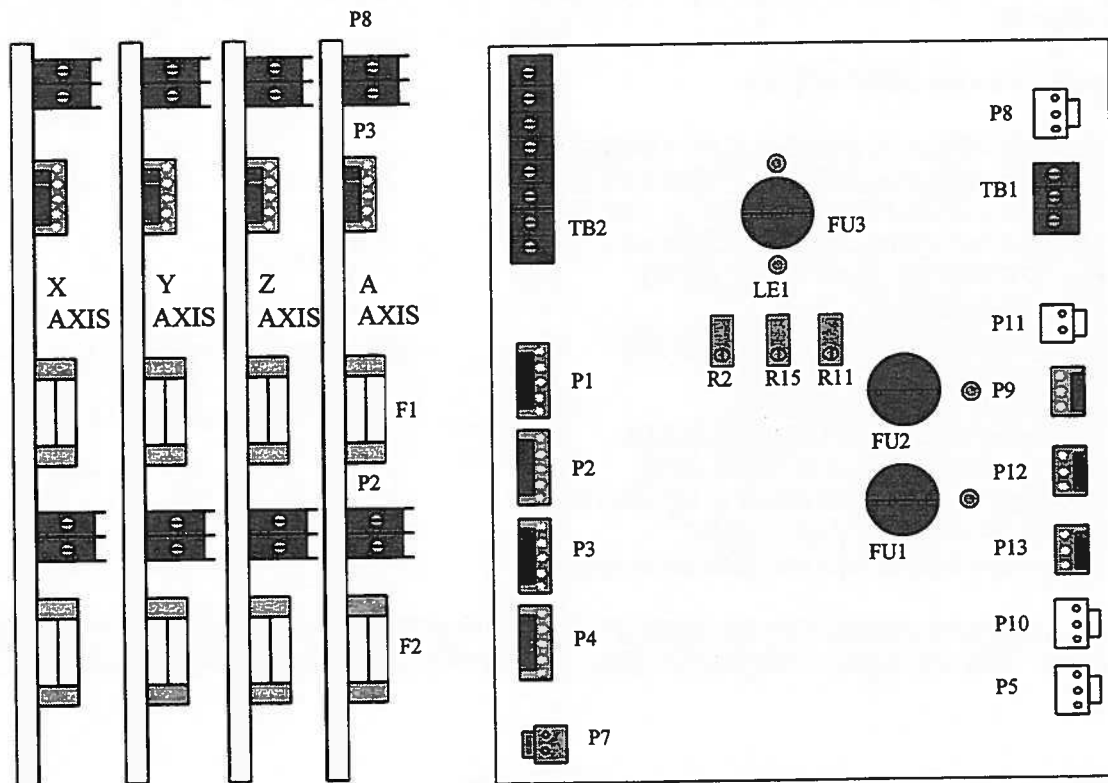


Fig. 4-1 Servo Drive Assembly

4.8 SERVO ENCODERS (BRUSHLESS)

Some Haas machines are equipped with brushless motors, which provides for better performance, and no maintenance. In addition to the performance differences, these machines differ from brush type machines, which have already been discussed, in the following areas:

The brushless motors have 8192 line encoders built in, which result in differences in acceleration parameters 7, 21, 35, 49 and 157. The exponential accel/decel time is set by parameters 115, 116 and 168. "In Position" parameters 101, 102, 103, 104 and 165 are also affected brushless motors.

The motor controller board has a dedicated processor which does all the servo control algorithm.

There is no servo distribution board anymore, therefore there is no CHARGE light present. Care should still be taken however, since there are high voltages present on the amplifiers, even when power is shut off. The high voltage comes from the spindle drive, which does have a CHARGE light.

The servo drive cards are replaced by Brushless Servo Amplifiers, and are controlled differently.

A low voltage power supply card is added to the servo drive assembly to supply the low voltage requirement to the amplifiers.

The CNC software is version 9.xx.

The user interface and motion profiling have not changed however, and the user should not see any functional differences between a brush type machine and a brushless machine.

4.9 SERVO CHARACTERISTICS (BRUSHLESS)

Servo characteristics are explained in detail in the previous chapter. The following is an example of how to achieve 130 inches/minute.

The exponential accel/decel time constant is set by Parameters 113, 114, 115, 116 and 168. It has units of 0.0001 seconds. The speed limit at which exponential accel/decel is not available is defined by the relationship between Parameters 7 and 113 (for the X-axis). Thus if Parameter 7 is 8000000 steps/sec/sec and Parameter 113 is 375 (0.0375 seconds); the maximum velocity for accurate interpolation should be:

$$8000000 \times 0.0375 = 300000 \text{ steps/second}$$

For a 8192 line encoder and 6 mm screw, this would be:

$$60 \times 300000 / 138718 = 130 \text{ inches/minute}$$

4.10 SERVO AMPLIFIERS (BRUSHLESS)

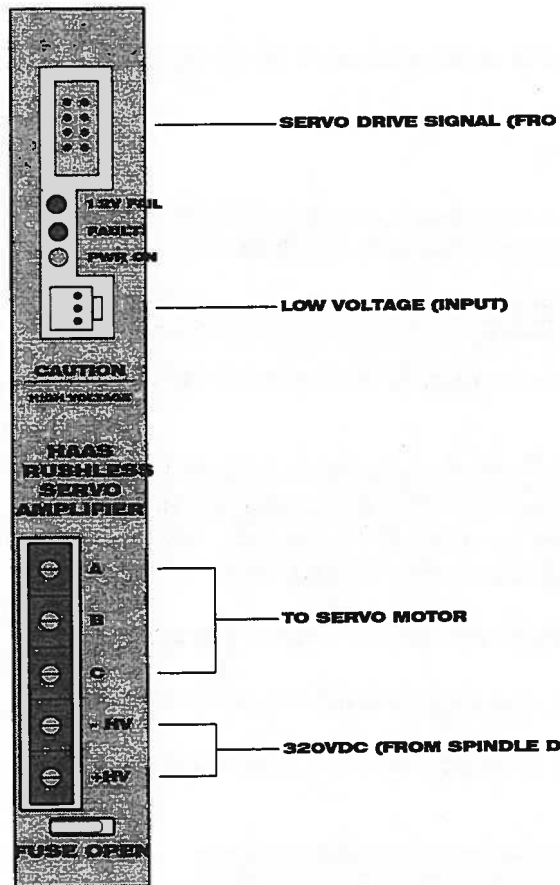
The brushless servo amplifier is a PWM based current source. The PWM outputs control the current to a three phase brushless motor. The PWM frequency is 16 KHz. The amplifiers are current limited to 30 amps peak. However there are fuse limits both in hardware and software to protect the amplifiers and motors from over current. The nominal voltage for these amplifiers is 320 volts. Therefore the peak power is about 9600 watts or 13 H.P. The amplifiers also have short circuit and over temperature and over heat protection.

There is a 10 amp supply fuse for failure protection. This fuse is relatively slow, therefore it can handle the 30 amp peak. Actual continuous current limit to the motor is controlled by software.

Commands to the amplifier are +/-5 volts current in two legs of the motor and a digital enable signal. A signal from the amplifier indicates drive fault or sustained high current in stalled motor.

The connectors on the amplifiers are:

- +H.V. + 320 volts DC
- H.V. 320 volts return
- A motor lead phase A
- B motor lead phase B
- C motor lead phase C
- J1 Three pin Molex connector used for +/-12 and GND.
- J2 Eight pin Molex connector used for input signals.



Brushless Amplifier

5. INPUT/OUTPUT ASSEMBLY

The IOPCB contains a circuit for electronically turning the tool changer power on and off. This prevents any arcing of the tool changer relays and increases their life tremendously. This includes an adjustable current limit to the tool changer. Potentiometer R45 adjusts the current limit to the tool changer motors. R45 should be set to limit current to between four and six amps.

The IOPCB also contains a circuit for sensing a ground fault condition of the servo power supply. If more than 0.5 amps is detected flowing through the grounding connection of the 160V DC buss, a ground fault alarm is generated and the control will turn off servos and stop.

Relay K6 is for the coolant pump 230V AC. It is a plug-in type and is double-pole. Relays K9 through K12 are also plug in types for controlling the tool changer.

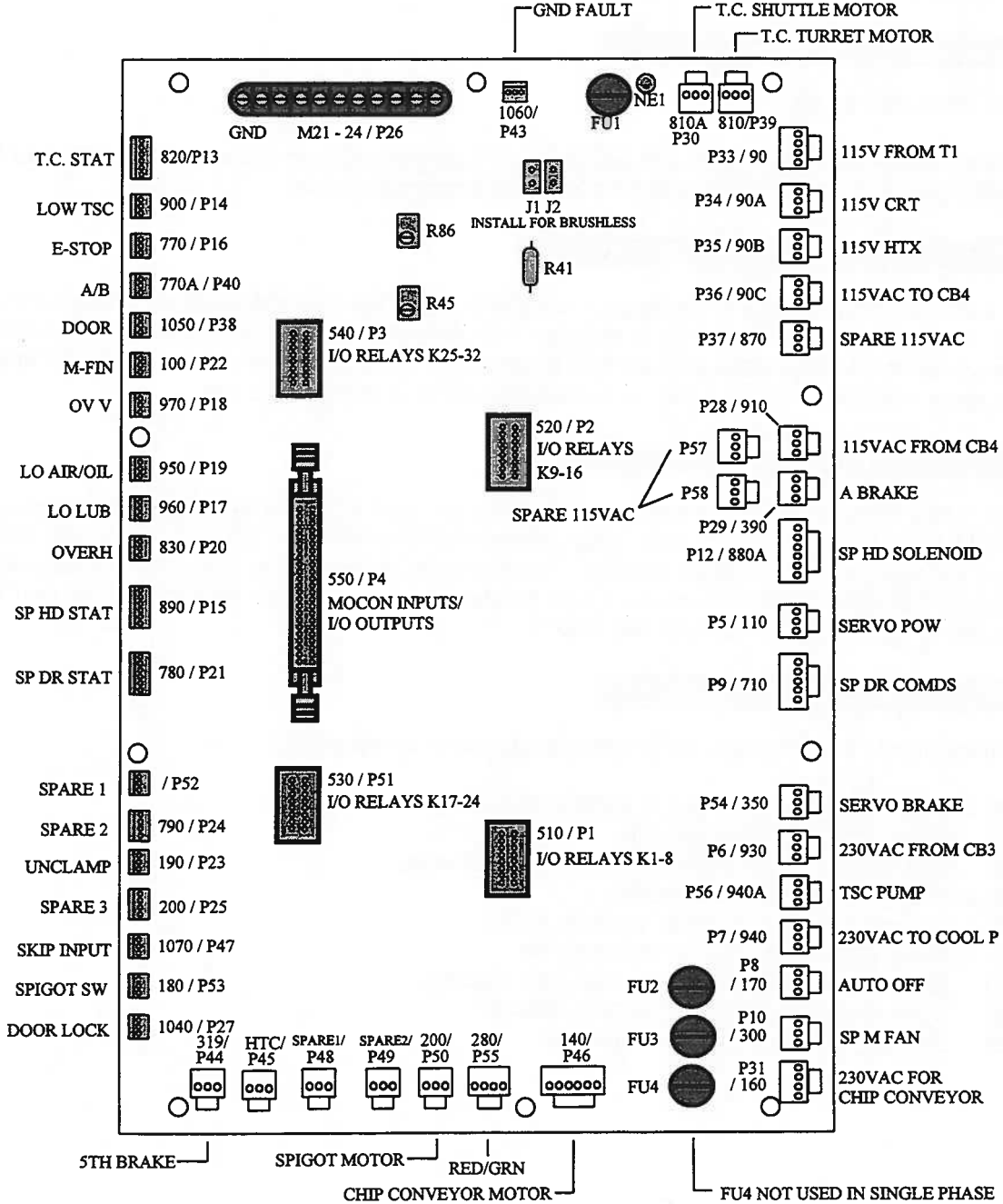
The Input/Output Assembly consists of a single printer circuit board called the IOPCB.

The connectors on the IOPCB are:

- P1 16-pin relay drivers from MOCON 1 to 8 (510)
- P2 16-pin relay drivers from MOCON 9 to 16 (520)
- P3 16-pin relay drivers from MOCON 17 to 24 (M21-M24) (540)
- P4 34-pin inputs to MOCON (550)
- P5 Servo power on relay 1-1 (110)
- P6 230V AC from CB3 (930)
- P7 230V AC to coolant pump (940)
- P8 Auto-off relay 1-7 (170)
- P9 Spindle drive commands (710)
- P10 Spindle fan and oil pump 115V AC (300)
- P12 115V AC to spindle head solenoids (880A)
- P13 Tool changer status inputs (820)
- P14 Low TSC(900)
- P15 Spindle head status inputs (890)
- P16 Emergency stop input (770)
- P17 Low Lube input (960)
- P18 Over Voltage Input (970)
- P19 Low Air Input (950)
- P20 Overheat input (830)
- P21 Spindle drive status inputs (780)
- P22 M-FIN input (100)
- P23 Remote Unclamp input (tool release) (190)
- P24 Spare 2 (790)
- P25 Spare 3 (200)
- P26 Spare terminals for M21 to M24
- P27 Door lock (1040)
- P28 115V AC from CB4 (910)
- P29 A-axis brake solenoid output (390)
- P30 Tool changer shuttle motor output (810A)
- P31 230 VAC for Chip Conveyor (160)
- P33 115V AC three-phase input from power supply assembly (90)
- P34 115V AC to CRT (90A)
- P35 115V AC to heat exchanger (90B)
- P36 115V AC to CB4 (90C)
- P37 115V AC spare (870)

- P38 Door open (1050)
- P39 Tool changer turret motor output (810)
- P40 (770A) A/B
- P43 Ground fault sense signal input (1060) Axis Brake
- P44 5TH axis brake (319)
- P45 HTC Shuttle
- P46 Chip Conveyor (140)
- P47 Skip input signal (1070)
- P48 spare 1
- P49 spare 2
- P50 Spigot Motor (200)
- P51 16 PIN Relay drivers 17-24 (530)
- P52 spare 1
- P53 Spigot Sense (180)
- P54 Servo Brake (350)
- P55 Red/green lights (280)
- P56 Thru spindle coolant pump(940A)
- P57 115V spare
- P58 115V spare

I/O illustration on next page



Input/Output Board

6. TWO-SPEED GEAR TRANSMISSION VF-1/2/3/4/6/8

The spindle head contains a two-speed gear transmission. The spindle motor is directly coupled to the transmission and the transmission is cog belt-coupled to the spindle.

6.1 GEAR BOX LUBRICATION

Gear Box: Mobil DTE 25 oil.

The spindle is air-pressurized and oil drip lubricated. The gear box uses an oil sump in the VF-1 thru 6 and is cooled by gear oil. The VF-0 does not have a gearbox and is air cooled.

6.2 GEAR BOX AIR SOLENOIDS

There is a double solenoid valve controlling air to the gear box. This solenoid sends air to select either the high gear or the low gear. When power is removed from the solenoids, the valve remains in its last state. Air is always required to ensure the gears are held in either high or low gear. Circuit breaker CB4 will interrupt power to these solenoids. Power is left on the solenoid which is commanded last.

6.3 GEAR BOX SENSE SWITCHES

On the VF-1 thru VF-6, there are two switches in the gear box used to sense the position of the gears. One switch indicates HIGH by opening and the other indicates LOW by opening. Between gears, both switches are closed indicating a between-gear condition. The diagnostic display shows the status of these switches and the CURNT COMDS display shows which gear is selected. If the switches indicate that the gear box is between gears, the display will indicate "No Gear".

6.4 GEAR CHANGE SEQUENCE

When a gear change is performed, the following sequence of events occurs:

- 1) If the spindle is turning, it is commanded to stop,
- 2) Pause until spindle is stopped,
- 3) Gear change spindle speed is commanded forward,
- 4) Pause until spindle is at speed,
- 5) Command high or low gear solenoid active,
- 6) Pause until in new gear or reversal time,
- 7) Alarm and stop if max gear change time elapsed,
- 8) If not in new gear, reverse spindle direction,
- 9) Turn off high and low gear solenoids

7. CONTROL PANEL

7.1 JOG HANDLE

The JOG handle is actually a 100-line-per-revolution encoder. We use 100 steps per revolution to move one of the servo axes. If no axis is selected for jogging, turning of the crank has no effect. When the axis being moved reaches its travel limits, the handle inputs will be ignored in the direction that would exceed the travel limits.

Parameter 57 can be used to reverse the direction of operation of the handle.

7.2 POWER ON/OFF SWITCHES

The POWER ON switch engages the main contactor. The on switch applies power to the contactor coil and the contactor thereafter maintains power to its coil. The POWER OFF switch interrupts power to the contactor coil and will always turn power off. POWER ON is a normally open switch and POWER OFF is normally closed. The maximum voltage on the POWER ON and POWER OFF switches is 24V AC and this voltage is present any time the main circuit breaker is on.

7.3 SPINDLE LOAD METER

The Load meter measures the load on the spindle motor as a percentage of the rated continuous power of the motor. There is a slight delay between a load and the actual reflection of the meter. The eighth A-to-D input also provides a measure of the spindle load for cutter wear detection. The second page of diagnostic data will display % of spindle load. The meter should agree with this display within 5%. The spindle drive display #7 should also agree with the load meter within 5%.

There are different types of spindle drive that are used in the control. They are all equivalent in performance but are adjusted differently.

7.4 EMERGENCY STOP SWITCH

The EMERGENCY STOP switch is normally closed. If the switch opens or is broken, power to the servos will be removed instantly. This will also shut off the tool changer, spindle drive, and coolant pump. The EMERGENCY STOP switch will shut down motion even if the switch opens for as little 0.005 seconds.

Be careful of the fact that Parameter 57 contains a status switch that, if set, will cause the control to be powered down when EMERGENCY STOP is pressed.

You should not normally stop a tool change with EMERGENCY STOP as this will leave the tool changer in an abnormal position that takes special action to correct.

Note that tool changer alarms can be easily corrected by first correcting any mechanical problem, pressing RESET until the alarms are clear, selecting ZERO RETURN mode, and selecting "AUTO ALL AXES".

If the shuttle should become jammed, the control will automatically come to an alarm state. To correct this, push the EMERGENCY STOP button and remove the cause of the jam. Push the RESET key to clear any alarms. Push the ZERO RETURN and the AUTO ALL AXES keys to reset the Z-axis and tool changer. Never put your hands near the tool changer when powered unless the EMERGENCY STOP button is pressed.

7.5 KEYBOARD BEEPER

There is a speaker inside the control panel that is used as an audible response to pressing keyboard buttons and as a warning beeper. The beeper is a one kHz signal that sounds for about 0.1 seconds when any keypad key, CYCLE START, or FEED HOLD is pressed. The beeper also sounds for longer periods when an auto-shut down is about to occur and when the "BEEP AT M30" setting is selected.

If the beeper is not audible when buttons are pressed, the problem could be in the keypad, keyboard interface PCB or in the speaker. Check that the problem occurs with more than one button and check that the speaker volume is not turned down.

8. MICROPROCESSOR ASSEMBLY

The microprocessor assembly is in the rear cabinet at the top left position. It contains three large boards. They are: microprocessor, the keyboard and the MOCON. All three boards of the processor assembly receive power from the low voltage power supply. The three PCB's are interconnected by a local buss on dual 50-pin connectors. At power-on of the control, some diagnostic tests are performed on the processor assembly and any problems found will generate alarms 157 or 158. In addition, while the control is operating, it continually tests itself and a self test failure will generate Alarm 152.

8.1 MICROPROCESSOR PCB (68EC030)

The Microprocessor PCB contains the 68EC030 processor running at 40 MHz, one 128K EPROM; between 256K and 8MB of CMOS RAM and between 512K and 1MB of FAST STATIC RAM. It also contains a dual serial port, a five year battery to backup RAM, buffering to the system buss, and eight system status LED's.

Two ports on this board are used to set the point at which an NMI* is generated during power down and the point at which RESET* is generated during power down.

The eight LED's are used to diagnose internal processor problems. As the system completes power up testing, the lights are turned on sequentially to indicate the completion of a step. The lights and meanings are:

+5V +5V logic power supply is present. (Normally On)

If this light does not come on, check the low voltage power supply and check that all three phases of 230V input power are present.

HALT Processor halted in catastrophic fault. (Normally Off)

If this light comes on, there is a serious problem with the processor PCB. Check that the EPROM is plugged in. Test the card with the buss connectors off.

POR Power-on-reset complete. (Normally On)

If this light does not come on, there is a serious problem with the processor PCB. Check that the EPROM is plugged in. Test the card with the buss connectors off.

SIO Serial I/O initialization complete. (Normally On)

If this light does not come on, there is a problem with the serial ports. Disconnect anything on the external RS-232 and test again.

MSG Power-on serial I/O message output complete. (Normally On)

If this light does not come on, there is a problem with serial I/O or interrupts. Disconnect anything on the external RS-232 and test again.

CRT CRT/VIDEO initialization complete. (Normally On)

If this light does not come on, there is a problem communicating with the VIDEO PCB. Check the buss connectors and ensure the VIDEO PCB is getting power.

PGM Program signature found in memory.(Normally On)

If this light does not come on, it means that the main CNC program package was not found in memory or that the auto-start switch was not set. Check that switch S1-1 is on and the EPROM is plugged in.

RUN PROGRAM RUNNING WITHOUT FAULT EXCEPTION.(Normally On)

If this light does not come on or goes out after coming on, there is a problem with the microprocessor or the software running in it. Check all of the buss connectors to the other two PCB's and ensure all three cards are getting power.

There 1 two-position DIP switch on the processor PCB labled S1. Switch S1-1 must be ON to auto-start the CNC operational program. If S1-1 is OFF, the PGM light will remain off.

Switch S2-1 is used to enable FLASH. If it is disabled it will not be possible to write to FLASH.

The processor connectors are:

- J1 Address buss
- J2 Data buss
- J4 Serial port #1 (for upload/download/DNC) (850)
- J5 Serial port #2 (for auxiliary 5th axis) (850A)
- J3 Power connector
- J6 Battery

8.2 MEMORY RETENTION BATTERY

The memory retention battery is initially soldered into the processor PCB. This is a 3.3V Lithium battery that maintains the contents of CMOS RAM during power off periods. Prior to this battery being unusable, an alarm will be generated indicating low battery. If the battery is replaced within 30 days, no data will be lost. The battery is not needed when the machine is powered on. Connector J6 on the processor PCB can be used to connect an external battery.

8.3 VIDEO KEYBOARD WITH FLOPPY

The VIDEO and KB PCB generates the video data signals for the monitor and the scanning signals for the keyboard. In addition, the keyboard beeper is generated on this board. There is a single jumper on this board used to select inverse video. The video PCB connectors are:

- P1 Power connector
- J3 Keyboard (700)
- J4 Address bus
- J5 Data
- J10 Floppy V+
- J11 SPARE
- J12 Floppy
- J13 Video (760)
- J14 RS422 B
- J15 RS422 A

8.4 MOTOR INTERFACE PCB (MOTIF)

The Motor Interface PCB provides all of the interfaces to motors and discrete inputs and outputs. It contains a single pot R54 to adjust the output of the D-A converter. The MOTIF PCB connectors are:

- P1 Data buss
- P2 X drive control and overcurrent sense (610)
- P3 Y drive control and overcurrent sense (620)
- P4 Z drive control and overcurrent sense (630)
- P5 A drive control and overcurrent sense (640)
- P6 X-axis encoder, Z, home, and overheat (660)
- P7 Y-axis encoder, Z, home, and overheat (670)
- P8 Z-axis encoder, Z, home, and overheat (680)
- P9 A-axis encoder, Z, home, and overheat (690)
- P10 32 discrete inputs (550)
- P11 Relay drives 1 to 8 (510)
- P12 Relay drives 9 to 16 (520)
- P13 Relay drives 17 to 24 (530)
- P14 Relay drives 25 to 32 (540)
- P15 Power connector (+5,+12+)
- P16 D-to-A output and -12V DC (720)
- P17 A-to-D inputs for DC buss voltage (980)
- P18 Jog Crank input and aux 1,2 (750)
- P19 Address buss
- P20 Spindle encoder inputs (1000)
- P21 A-to-D input for spindle temperature (1020)
- P22 A-to-D input for spindle load monitor (730B)
- P24 Home switch inputs X, Y, Z (990)

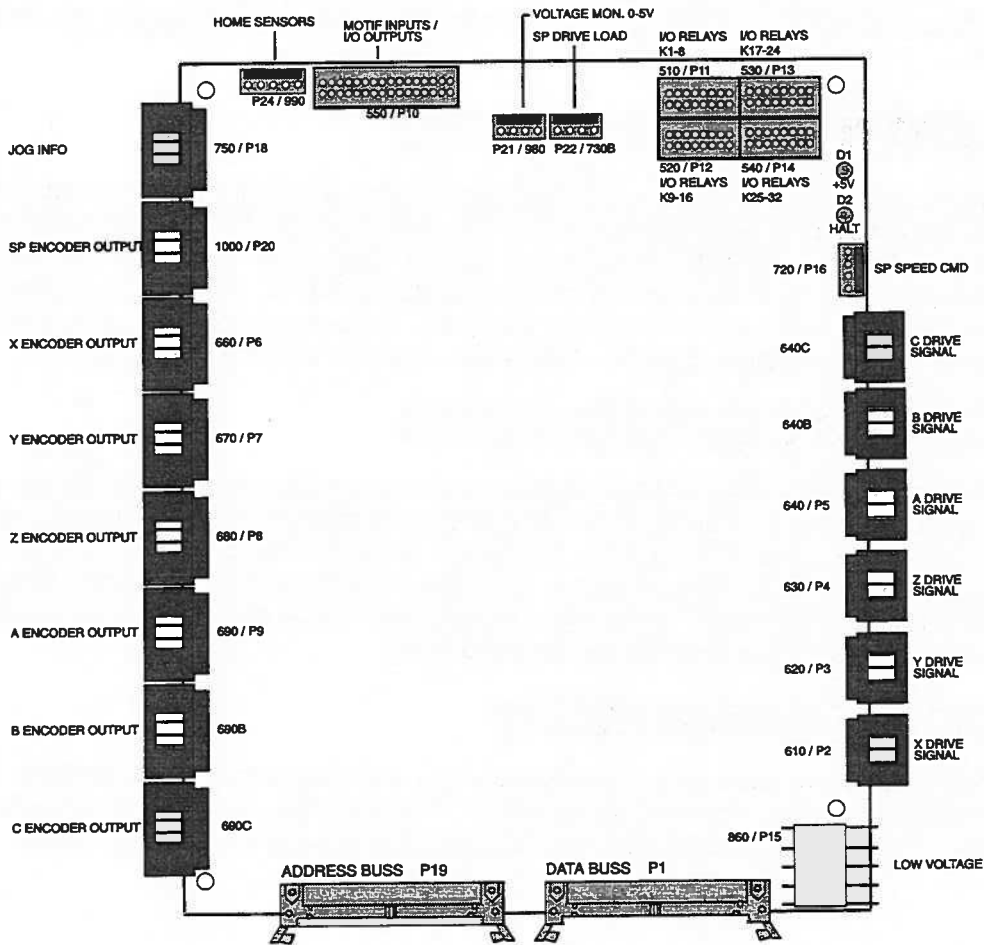
8.5 MOTOR CONTROLLER (MOCON) BRUSHLESS

The brushless machining centers are equipped with a microprocessor based brushless motor controller board (MOCON) that replaces the motor interface in the brush type controls. It runs in parallel with the main processor, receiving servo commands and closing the servo loop around the servo motors.

In addition to controlling the servos and detecting servo faults, the motor controller board, (MOCON), is also in charge of processing discrete inputs, driving the I/O board relays, commanding the spindle and processing the jog handle input. Another significant feature is that it controls 6 axes, so there is no need for an additional board for a 5 axis machine.

- P1 Data Bus
- P2 X amplifier control and fault sensing (610)
- P3 Y amplifier control and fault sensing (620)
- P4 Z amplifier control and fault sensing (630)
- P5 A amplifier control and fault sensing (640)
- P32 B amplifier control and fault sensing (640B)
- P33 C amplifier control and fault sensing (640C)
- P6 X encoder input (660)
- P7 Y encoder input (670)
- P8 Z encoder input (680)
- P9 A encoder input (690)
- P30 B encoder input (690B)
- P31 C encoder input (690C)
- P18 Jog encoder input (750)

- P20 Spindle encoder input (1000)
- P10 Inputs from I/O board (550)
- P11 I/O relays K1-8 (510)
- P12 I/O relays K9-16 (520)
- P13 I/O relays K17-24 (530)
- P14 I/O relays K25-32 (540)
- P15 Low Voltage Power (860)
- P16 Spindle command output (720)
- P19 Address bus
- P24 Axis home switches (990)



MOCON Board

9. SPINDLE DRIVE ASSEMBLY

The spindle drive is located in the main cabinet on the right side and halfway down. It has a blue cover on it. It operates from three-phase 200 to 240V AC. It has a 7.5 (or 10) H.P. continuous rating, and a 11.25 (or 15) H.P. one-minute rating. The spindle drive is protected by CB1 at 40 amps (20 for High Voltage option). Never work on the spindle drive until the small red CHARGE light goes out. Until this light goes out, there are dangerous voltages inside the drive, even when power is shut off.

For all other data on the spindle drive, refer to the supplied documentation for your drive.

10. RESISTOR ASSEMBLY

The Resistor Assembly is located on top of the control cabinet. It contains the servo and spindle drive regen load resistors.

10.1 SPINDLE DRIVE REGEN RESISTOR

A resistor bank is used by the spindle drive to dissipate excess power caused by the regenerative effects of decelerating the spindle motor. If the spindle motor is accelerated and decelerated again in rapid succession repeatedly, this resistor will get hot. In addition, if the line voltage into the control is above 255V, this resistor will begin to heat. This resistor is overtemp protected at 100° C. At that temperature, an alarm is generated and the control will begin an automatic shutdown. If the resistor is removed from the circuit, an alarm may subsequently occur because of an overvoltage condition inside the spindle drive.

10.2 SERVO DRIVE REGEN RESISTOR

A 25-ohm, 300-watt resistor is used by the servo drives to dissipate excess power caused by the regenerative effects of decelerating the servo motors. If the servo motors are accelerated and decelerated again in rapid succession repeatedly, this resistor will get hot. In addition, if the line voltage into the control is above 255V, this resistor will begin to heat. This resistor is overtemp protected at 100° C. At that temperature, an automatic control shutdown is begun. If that resistor is removed from the circuit, an alarm may subsequently occur because of an overvoltage condition for the servo buss.

10.3 OVERHEAT SENSE SWITCH

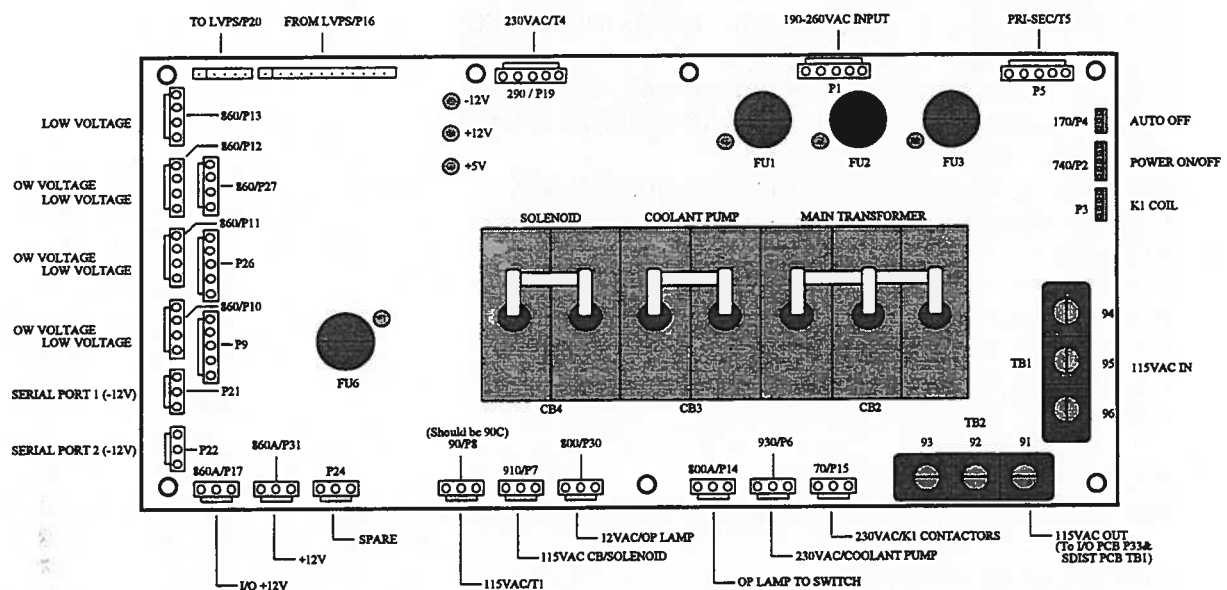
There is an overtemperature sense switch mounted near the above-mentioned regen resistors. This sensor is a normally-closed switch that opens at about 100° C. It will generate an alarm and all motion will stop. After four minutes of an overheat condition, an automatic shutdown will occur in the control.

11. POWER SUPPLY ASSEMBLY

All power to the control passes through the power supply assembly. It is located on the upper right corner of the control cabinet.

11.1 MAIN CIRCUIT BREAKER CB1

Circuit breaker CB1 is rated at 40 amps (20 for High Voltage option) and is used to protect the spindle drive and to shut off all power to the control. The locking On/Off handle on the outside of the control cabinet will shut this breaker off when it is unlocked. A trip of this breaker indicates a **SERIOUS** overload problem and should not be reset without investigating the cause of the trip. The full circuit breaker rating corresponds to as much as 15 horsepower.



Power Supply.

11.2 MAIN CONTACTOR K1

Main contactor K1 is used to turn the control on and off. The POWER ON switch applies power to the coil of K1 and after it is energized, an auxiliary switch on K1 continues to apply power to the coil. The POWER OFF switch on the front panel will always remove power from this contactor.

When the main contactor is off, the only power used by the control is supplied through two ½ amp fuses to the circuit that activates the contactor. An overvoltage or lightning strike will blow these fuses and shut off the main contactor.

The power to operate the main contactor is supplied from a 24V AC control transformer that is primary fused at ½ amp. This ensures that the only circuit powered when the machine is turned off is this transformer and only low voltage is present at the front panel on/off switches.

11.3 LOW VOLTAGE POWER SUPPLY

The low voltage power supply provides +5V DC, +12V DC, and -12V DC to all of the logic sections of the control. It operates from 115V AC nominal input power. It will continue to operate correctly over a 90V AC to 133V AC range.

11.4 POWER PCB (POWER)

The low voltage power distribution and high voltage fuses and circuit breakers are mounted on a circuit board called the POWER PCB. The following connectors are on it:

- P1 Five-pin brings 230V AC three ph from main breaker
- P2 On/Off connections to front panel (740)
- P3 Coil and aux connections to contactor K1
- P4 Auto-off connection to IOPCB (170)
- P5 Low voltage control transformer to power K1
- P6 230V AC from CB3 to coolant pump (930)
- P7 115V AC from CB4 to IOPCB for solenoids (910)
- P8 115V AC /T1 (90)
- P9 Tool changer fuse circuit from FU5 to IOPCB (840)
- P10 +5/+12/Gnd form low volt supply to logic boards (860)
- P11 +5/+12/Gnd form low volt supply to logic boards (860)
- P12 +5/+12/Gnd form low volt supply to logic boards (860)
- P13 +5/+12/Gnd form low volt supply to logic boards (860)
- P14 12V AC to operator's lamp (800A)
- P15 230V AC from contactor K1 for coolant pump (70)
- P16 Low voltage power from power supply
- P17 +12V DC to IOPCB (860A)
- P18 Not used
- P19 Connector to op. lamp transformer T4 (290)
- P20 115V AC to low voltage supply
- P21 -12V DC to processor PCB
- P22 -12V DC to MOTIF PCB

- P26 +12V DC option connector
- P27 +5/+12/Gnd form low volt supply to logic boards (860)

- P30 12V AC OP Lamp (800)
- P31 +12V (860A)

For older internal transformer with 208/230 taps:

- TB1 230V AC from contactor K1
- TB2 230V AC to T1 primary

11.5 POWER-UP LOW VOLTAGE CONTROL TRANSFORMER (T5)

The low voltage control transformer, T5, supplies power to the coil of the main contactor K1. It guarantees that the maximum voltage leaving the Power Supply assembly when power is off is 12V AC to earth ground. It is connected via P5 to the POWER PCB.

11.6 SECONDARY CIRCUIT BREAKERS

Three more circuit breakers are on the Power supply assembly.

CB2 controls the 115volt power from the main transformer to the servo transformers and, if tripped, will turn off the servo motors and air solenoids. CB2 could be blown by a severe servo overload.

CB3 controls the power to coolant pump only. It can be blown by an overload of the coolant pump motor or a short in the wiring to the motor.

CB4 controls the 115V AC to the air solenoids, 4th axis brake, and the oiler. It is never expected to trip. If it does trip, it is likely caused by a short circuit in the wiring on the I/O assembly or the wiring to the solenoids on the spindle head.

11.7 OPERATOR'S LAMP TRANSFORMER

Transformer T4 supplies low voltage to the operator's lamp. The primary is 115V AC and the secondary is 10V AC. The primary is protected at ½ amp by F6. It is connected to the POWER PCB by connector P19.

12. POWER TRANSFORMER ASSEMBLY (T1)

The power transformer assembly is used to convert three-phase 190/260V to three-phase 115V and is primarily used by the servo drives. The video monitor, solenoids, fans, and oiler also use 115V AC. This transformer's maximum input voltage is 260V @ 60 Hertz, and 240V @ 50 Hertz. It is located in the main cabinet in the lower right corner. It is rated at 12KVA and its primary is protected to 40 amps.

This transformer has four voltage connections that allow for a range of inputs from 195V to 260V. The transformer has an autotransformer primary to supply 240V, three-phase to the spindle drives other 240V applications.

- 195 to 210 right side
- 211 to 226 right center
- 227 to 243 left center
- 244 to 260 left side

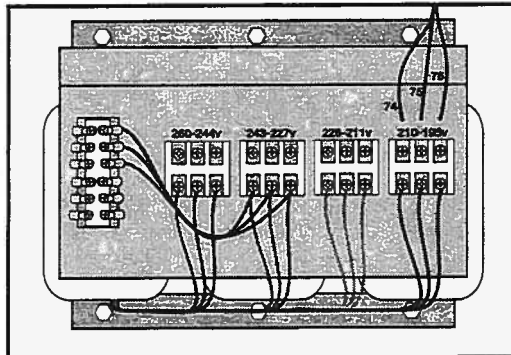


Fig. 12-1 Polyphase bank transformer.

12.1 PRIMARY CONNECTION TO T1

Input power to T1 is supplied through CB1, the 40 amp three-phase main circuit breaker. Three-phase 230 to T1 is connected to the first three terminals of TB10.

12.2 VOLTAGE SELECTION TAPS

There are four labeled plastic terminal blocks. Each block has three connections for wires labeled 74, 75, and 76. Follow the instructions printed on the transformer.

12.3 SECONDARY CONNECTION TO T1

The secondary output from T1 is 115V AC three-phase. CB2 protects the secondary of transformer T1 and is rated at 25 amps.

12.4 OPTIONAL 480 TRANSFORMER

Voltage Selection Taps for the 480 Transformer:

Right to left:

353 to 376
 377 to 400
 401 to 425
 426 to 451
 452 to 480*

* 480 V transformer has additional terminal block

13. FUSES

The servo drive (DRIVER) cards have three fuses on each of the X, Y, Z, and A PCB's (F1, F2, F3). If these fuses are ever blown, the associated motor will stop. This will only happen if there is a failure of the drive card and the user should never attempt to replace these fuses.

The POWER PCB contains three ½-amp fuses located at the top right (FU1, FU2, FU3). If the machine is subject to a severe overvoltage or a lightning strike, these fuses will blow and turn off all of the power. Replace these fuses only with the same type and ratings. The other two fuses protect the tool changer (FU5) and the operator's lamp (FU6).

On the servo drive assembly, there is a printed circuit board (SDIST) containing three one-amp fuses (FU1, FU2, FU3). Two of these fuses protect the contactor and small transformers. They are never expected to blow. The third fuse protects the regen load circuit load from shorts.

FUSE NAME	TYPE	RATING	VOLTAGE (amps)	LOCATION
FU1	AGC	½	250V	POWER pcb, upper right
FU2	AGC	½	250V	" " " "
FU3	AGC	½	250V	" " " "
LAMP	AGC	½	250V	" " lower left
FU1	AGC	½	250V	SDIST pcb, right center
FU2	AGC	½	250V	" " " "
FU3	AGC	5	250V	" " top center
F1	ABC	20	250V	SDRIVER pcb's (X, Y, Z, A)
F2	ABC	20	250V	" " " "
F3	ABC	10	250V	" " " "
FU1	ABC	5	250V	I/O PCB
FU2	ABC	5	250V	I/O PCB
FU3	ABC	5	250V	I/O PCB
FU4	ABC	5	250V	I/O PCB

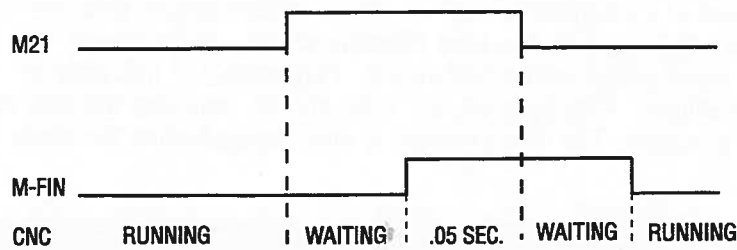
14. SPARE USER M CODE INTERFACE

The M code interface uses outputs M21-24 and one discrete input circuit. M codes M21 through M24 will activate relays labeled M21-24. These relay contacts are isolated from all other circuits and may switch up to 120V AC at one amp. The relays are SPDT.

WARNING! Power circuits and inductive loads must have snubber protection.

The M-FIN circuit is a normally open circuit that is made active by bringing it to ground. The one M-FIN applies to all eight of the user M codes.

The timing of a user M function must begin with all circuits inactive, that is, all circuits open. The timing is as follows:



The Diagnostic Data display page may be used to observe the state of these signals.

14.1 M FUNCTION RELAYS

The IOPCB contains position for four relays (M21-M24) and all are available to the user. In addition, M21 is already wired out to P12 at the side of the control cabinet. This is a four-pin DIN connector and includes the M-FIN signal.

14.2 M-FIN DISCRETE INPUT

The M-FIN discrete input is a low voltage circuit. When the circuit is open, there is +12V DC at this signal. When this line is brought to ground, there will be about 10 milliamps of current. M-FIN is discrete input #10 and is wired from input #10 on the Inputs PCB on the Input/Output Assembly. The return line for grounding the circuit should also be picked up from that PCB. For reliability, these two wires should be routed in a shielded cable where the shield is grounded at one end only. The diagnostic display will show this signal a "1" when the circuit is open and a "0" when this circuit is grounded.

14.3 TURNING M FUNCTIONS ON AND OFF

The eight optional M code relays can also be separately turned on and off using M codes M51-M54 and M61-M64. M51 to M54 will turn on one of the eight relays and M61 to M64 will turn the relays off. M51 and M61 correspond to M21, etc.

15. LUBRICATION SYSTEM

The lubrication system is a resistance type system which forces oil through metering units at each of the 16 lubricating points within the machine. The system uses one metering unit at each of the lubricating points: one for each linear guide pad, one for each lead screw and one for spindle lubrication. A single oil pump is used to lubricate the system. The pump is powered only when the spindle and/or an axis moves. Once powered the pump cycles approximately 3.0 cc of oil every 30 minutes throughout the oil lines to the lube points. Every lube point receives approximately 1/16 of oil. The control monitors this system through an internal level switch in the reservoir and an external pressure switch on the lube panel.

15.1 LOW LUBRICATION AND LOW PRESSURE SENSE SWITCHES

There is a low lube sense switch in the oil tank. When the oil is low, an alarm will be generated. This alarm will not occur until the end of a program is reached. There is also an lube pressure switch that senses the lube pressure. Parameter 117 controls the lube pressure check. If Parameter 117 is not zero, the lube pressure is checked for cycling high within that period. Parameter 117 has units of , 1/50 seconds; so 30 minutes gives a value of 90000. Parameter 57, bit "Oiler on/off", indicates the lube pump is only powered when the spindle fan is powered. The lube pressure is only checked when the pump is on.

16. SWITCHES

16.1 LAMP ON/OFF SWITCH

An on/off switch is supplied for the operator's lamp. It is located on the side of the control cabinet below all of the motor connectors.

16.2 DOOR OPEN SENSE SWITCH

The DOOR OPEN sense switch is a magnetic reed switch type and consists of two switches; one on each half of the enclosure front doors. These switches are normally closed and wired in series. When the doors open, one or both of these switches will open and the machine will stop with a "Door Hold" function. When the door is closed again, operation will continue normally.

The wiring for the door switches is wired in either of two paths. The first one applies to machines built before March 1990; this uses connector P5 at the side of the cabinet and routes the wires past the oiler and through the base of the mill. The second wiring routes through the front panel support arm and down through the top of the enclosure.

If the doors are open, you will not be able to start a program. Door Hold will not stop a tool change operation or a tapping operation, and will not turn off the coolant pump. Also, if the doors are open, the spindle speed will be limited to 750 RPM.

The Door Hold function can be temporarily disabled with by turning Setting 51 **on**, if Parameter 57 bits DOOR STOP SP and SAFETY CIRC are set to zero, but this setting will return to OFF when the control is turned off.

16.3 LIMIT SWITCHES

NOTE: There are fourteen (14) limit switches located on the VMC, and some are difficult to reach. Ensure the problem is the switch before beginning removal procedures. The following is a list of all switches, their general location, and a functional description:

CLAMP/UNCLAMP SWITCHES

[Tool Release Piston Assembly (2)]

There are two switches used to sense the position of the tool clamping mechanism. They are both normally closed and one will activate at the end of travel during unclamping and the other during clamping. When both switches are closed, it indicates that the draw bar is between positions.

A tool change operation will wait until the unclamped switch is sensed before the Z-axis pulls up from the tool. This prevents any possibility of breaking the tool changer or its support mounts.

The diagnostic display can be used to display the status of the relay outputs and the switch inputs.

SPINDLE ORIENT SWITCH

[Top rear of transmission]

A normally-open switch that is held closed is used to sense when the pin drops in to lock the spindle. When the pin drops the switch opens, indicating orientation is complete.

The normally-closed side of the same switch that is held open, is wired to the spindle drive and commands it into a "Coast Stop" condition. This is done to ensure the spindle motor is not powered when the pin is locking the spindle.

DOOR HOLD SWITCHES

[Top outer edges of door opening (2)]

The DOOR OPEN sense switch consists of two switches; one on each side of the enclosure front doors. These switches are normally closed and wired in series. When the doors open, one or both of these switches will open and the machine will stop with a "Door Hold" function. When the door is closed again, operation will continue normally.

If the doors are open, you will not be able to start a program. Door hold will not stop a tool change operation, will not turn off the spindle, and will not turn off the coolant pump.

The door hold function can be temporarily disabled with Setting 51, but this setting will return to OFF when the control is turned off.

X, Y, AND Z LIMIT SWITCHES

Prior to performing an POWER UP/RESTART or an AUTO ALL AXES operation, there are no travel limits. Thus, you can jog into the hard stops in either direction for X, Y, or Z. After a ZERO RETURN has been performed, the travel limits will operate unless an axis hits the limit switch. When the limit switch is hit, the zero returned condition is reset and an AUTO ALL AXES must be done again. This is to ensure that if you hit the limit switch, you can still move the servo back away from it.

The limit switches are normally closed. When a search for zero operation is being performed, the X, Y, and Z axes will move towards the limit switch unless it is already active (open); then they will move away from the switch until it closes again; then they will continue to move until the encoder Z channel is found. This position is machine zero.

Auto search for zero in the Z-axis is followed by a rapid move from the limit switch position down to the tool change position. This makes the Z-axis a little different from the other axes. The position found with the limit switch is not machine zero but is the position used to pull tools out of the spindle. Machine zero for Z is below this by Parameter 64. Be careful during the Z zero search and stay clear of that rapid move.

What Can Go Wrong With Limit Switches?

If the machine is operated without connector P5, a LOW LUBE and DOOR OPEN alarm will be generated. In addition, the Home search will not stop at the limit switch and will instead run into the physical stops on each axis.

If the switch is damaged and permanently open, the zero search for that axis will move in the negative direction at about 0.5 in/min until it reaches the physical travel stops at the opposite end of travel.

If the switch is damaged and permanently closed, the zero search for that axis will move at about 10 in/min in the positive direction until it reaches the physical stops.

If the switch opens or a wire breaks after the zero search completes, an alarm is generated, the servos are turned off, and all motion stops. The control will operate as though the zero search was never performed. The RESET can be used to turn servos on but you can jog that axis only slowly.

TOOL CHANGER POSITION SWITCHES

[Inside of Tool Carriage (2)]

GENEVA WHEEL POSITION MARK

The turret rotation mechanism has a switch mounted so that it is activated for about 30° of travel of the Geneva mechanism. When activated, this switch indicates that the turret is centered on a tool position. This switch is normally closed. The diagnostic display will show this status of this input switch as "TC MRK". A "1" indicates the Geneva wheel is in position.

TOOL #1 SENSE SWITCH

The tool rotation turret has a switch that is activated when tool one is in position or facing towards the spindle. At POWER ON this switch can indicate that tool #1 is in the spindle. If this switch is not active at power-on, the first tool change will rotate the turret until the switch engages and then move to the selected tool. The diagnostic display will show this status of this input switch as "TOOL #1". A "1" indicates that tool #1 is in position.

SHUTTLE IN/OUT SWITCHES

[Tool Changer Holding Plate (2)]

Two switches are used to sense the position of the tool changer shuttle and the arm that moves it. One switch is activated when the shuttle is moved full travel inward and one is activated when it is full travel outward. These switches are normally closed so that both will be closed between in and out. The diagnostic display will show this status of this input switch. A "1" indicates the associated switch is activated or open.

TRANSMISSION HIGH/LOW GEAR POSITION SWITCHES

[Bottom of Gearbox Assembly (2)]

On the VF-1, VF-2, VF-3 and VF-4, there are two switches in the gear box used to sense the position of the gears. One switch indicates HIGH by opening and the other indicates LOW by opening. Between gears, both switches are closed indicating a between-gear condition. The diagnostic display shows the status of these switches and the CURNT COMDS display shows which gear is selected. If the switches indicate that the gear box is between gears, the display will indicate "No Gear".

NOTE: The Transmission High/Low Gear Position Switches are located at the bottom of the Gearbox Assembly and are extremely difficult to reach. Removal of this assembly is necessary to replace these switches. See Mechanical Service, for Spindle Motor and Transmission removal.

17. HYDRAULIC COUNTERBALANCE

The spindle head weight is balanced by the upward pull of a hydraulic cylinder. The hydraulic oil forces the piston to retract into the cylinder body. The oil is then pressurized by a nitrogen reservoir. The system is self contained and passive (no pump is required to maintain the lift). Normal Z-Axis of the gas/oil counter balance has the initial pressure to balance the weight at full system volume, plus an additional 50-75 psi overcharge for longevity.

18. DIAGNOSTIC DATA

The ALARM MSGS display is the most important source of diagnostic data. At any time after the machine completes its power-up sequence, it will either perform a requested function or stop with an alarm. Refer to Section 2.5 for a complete list of alarms, their possible causes, and some corrective action.

If there is an electronics problem, the controller may not complete the power-up sequence and the CRT will remain blank. In this case, there are two sources of diagnostic data; these are the audible beeper and the LED's on the processor PCB. If the audible beeper is alternating a ½ second beep, there is a problem with the main control program stored in EPROM's on the processor PCB. If any of the processor electronics cannot be accessed correctly, the LED's on the processor PCB will or will not be lit.

If the machine powers up but has a fault in one of its power supplies, it may not be possible to flag an alarm condition. If this happens, all motors will be kept off and the top left corner of the CRT will have the message:

POWER FAILURE ALARM

and all other functions of the control will be locked out.

When the machine is operating normally, a second push of the PARAM/DGNOS key will select the diagnostics display page. The PAGE UP and PAGE DOWN keys are then used to select one of two different displays. These are for diagnostic purposes only and the user will not normally need them. The diagnostic data consists of 32 discrete input signals, 32 discrete output relays and several internal control signals. Each can have the value of 0 or 1. In addition, there are up to three analog data displays and an optional spindle RPM display. Their number and functions are:

19. DISCRETE INPUTS / OUTPUTS

#	Name	Description	#	Name	Description
1	TC IN	Tool Changer In	17	SP LOK	Spindle Locked
2	TC OUT	Tool Changer Out	18	SP FLT	Spindle Drive Fault
3	T ONE	At Tool One	19	SP ST*	Spindle Not Stopped
4	LO CNT	Low Coolant	20	SP AT*	Spindle Not At Speed
5	TC MRK	T.C. Geneva Mark	21	LO OIL	Spindle/GB coolant low
6	SP HIG	Spindle In High	22	spare1	
7	SP LOW	Spindle In Low	23	spare2	
8	EM STP	Emergency Stop	24	spare3	
9	DOOR S	Door Open Switch	25	UNCLA*	Remote tool unclamp
10	M-FIN*	Not M Func Finish	26	LOPHSE	Low voltage in phase 1
11	OVERV*	Not Over Voltage	27	spare4	
12	LO AIR	Low Air Pressure	28	spare5	
13	LO LUB	Low Lube Oil	29	GR FLT	Ground fault
14	OVRHT*	Not Over Heat	30	SKIP	Skip Signal
15	DB OPN	Tool Unclamped	31	SPIGOT	
16	DB CLS	Tool Clamped	32	CNVEYR	Conveyor overload

DISCRETE OUTPUTS

#	Name	Description	#	Name	Description
1	SRV PO	Servo Power On	17	SPGCW	Spigot clockwise
2	SP FOR	Spindle Forward	18	SPGCCW	Spigot counter/clockwise
3	SP REV	Spindle Reverse	19	SpareA	
4	SP RST	Spindle Reset	20	SpareB	
5	4TH BK	4th Axis Brk Rel	21	PRE-CH	Pre-charge
6	COOLNT	Coolant Pump	22	HTC SH	Horizontal T.C. shuttle
7	AUT OF	Auto Turn Off	23	5TH BK	5th Axis Brake
8	SP FAN	Spind Motor Fan	24	Y160	Door Lock
9	TC IN	Tool Changer In	25	M21	
10	TC OUT	Tool Changer Out	26	M22	
11	TC CW	Tool Changer CW	27	M23	
12	TC CCW	Tool Changer CCW	28	AUXCLT	Auxiliary Coolant
13	SP HIG	Spindle High Gear	29	GRNBCN	Red beacon worklight
14	SP LOW	Spindle Low Gear	30	RDBCN	Red beacon worklight
15	T UNCL	Tool Unclamped	31	CNVENA	Conveyor enabled
16	SP LOK	Spindle Lock Cmd	32	CNVREV	Conveyor reverse

The 38 inputs are numbered the same as the 38 connections on the inputs printed circuit board. The last eight outputs are reserved for expansion by HAAS.

The second page of diagnostic data is displayed using the PAGE UP and PAGE DOWN keys. It contains:

INPUTS 2

Name	Description	Name	Description
X Z CH	X-axis Z Channel	X DRVF	X-axis drive fault
Y Z CH	Y-Axis Z Channel	Y DRVF	Y-axis drive fault
Z Z CH	Z-axis Z Channel	Z DRVF	Z-axis drive fault
A Z CH	A-axis Z Channel	A DRVF	A-axis drive fault
B Z CH	B-axis Z Channel	B DRVF	B-axis drive fault
X HOME	X-axis Home/Lim Switch	X CABL	Broken cable to X encoder
Y HOME	Y-axis Home	Y CABL	Broken cable to Y encoder
Z HOME	Z-axis Home	Z CABL	Broken cable to Z encoder
A HOME	A-axis Home	A CABL	Broken cable to A encoder
B HOME	B-axis Home	B CABL	Broken cable to B encoder
X OVRH	X Motor OverTemp		
Y OVRH	Y Motor OverTemp		
Z OVRH	Z Motor OverTemp		
A OVRH	A Motor OverTemp		
B OVRH	B Motor OverTemp		

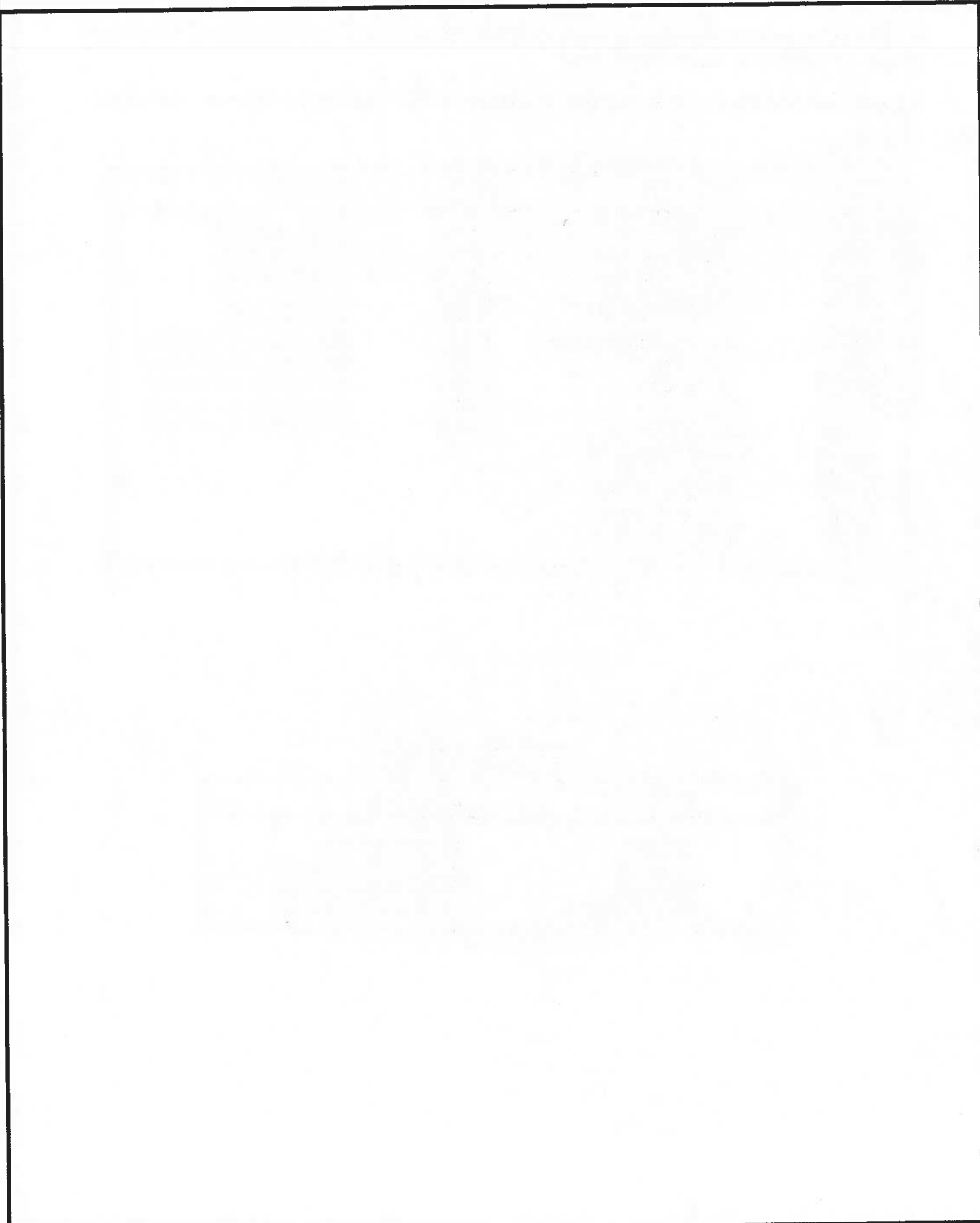
ANALOG DATA

Name	Description
SP LOAD	Spindle load in %
SP SPEED	Spindle RPM CW or CCW
RUN TIME	Total machine run time
TOOL CHANGES	Number of tool changes

VF-SERIES
OPERATORS MANUAL

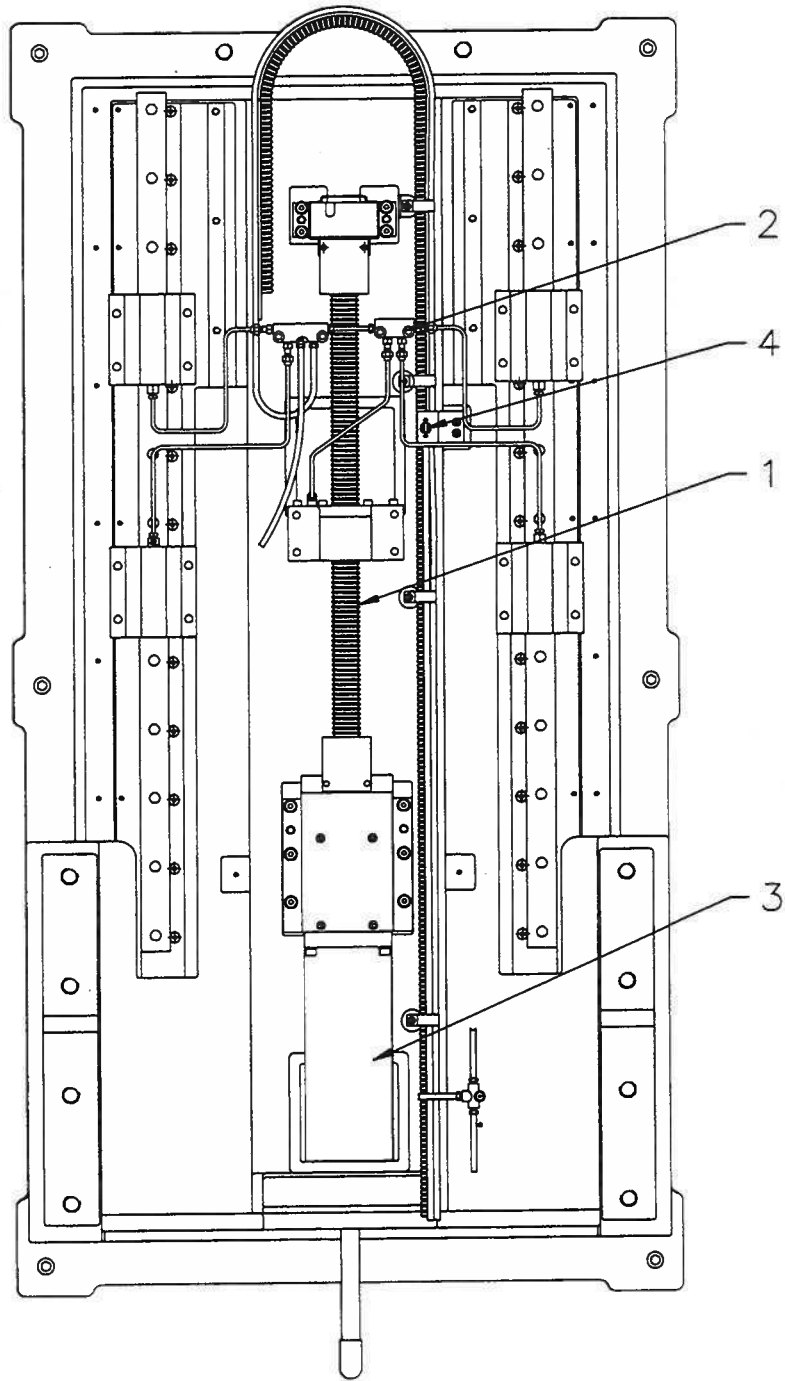


TECHNICAL REFERENCE



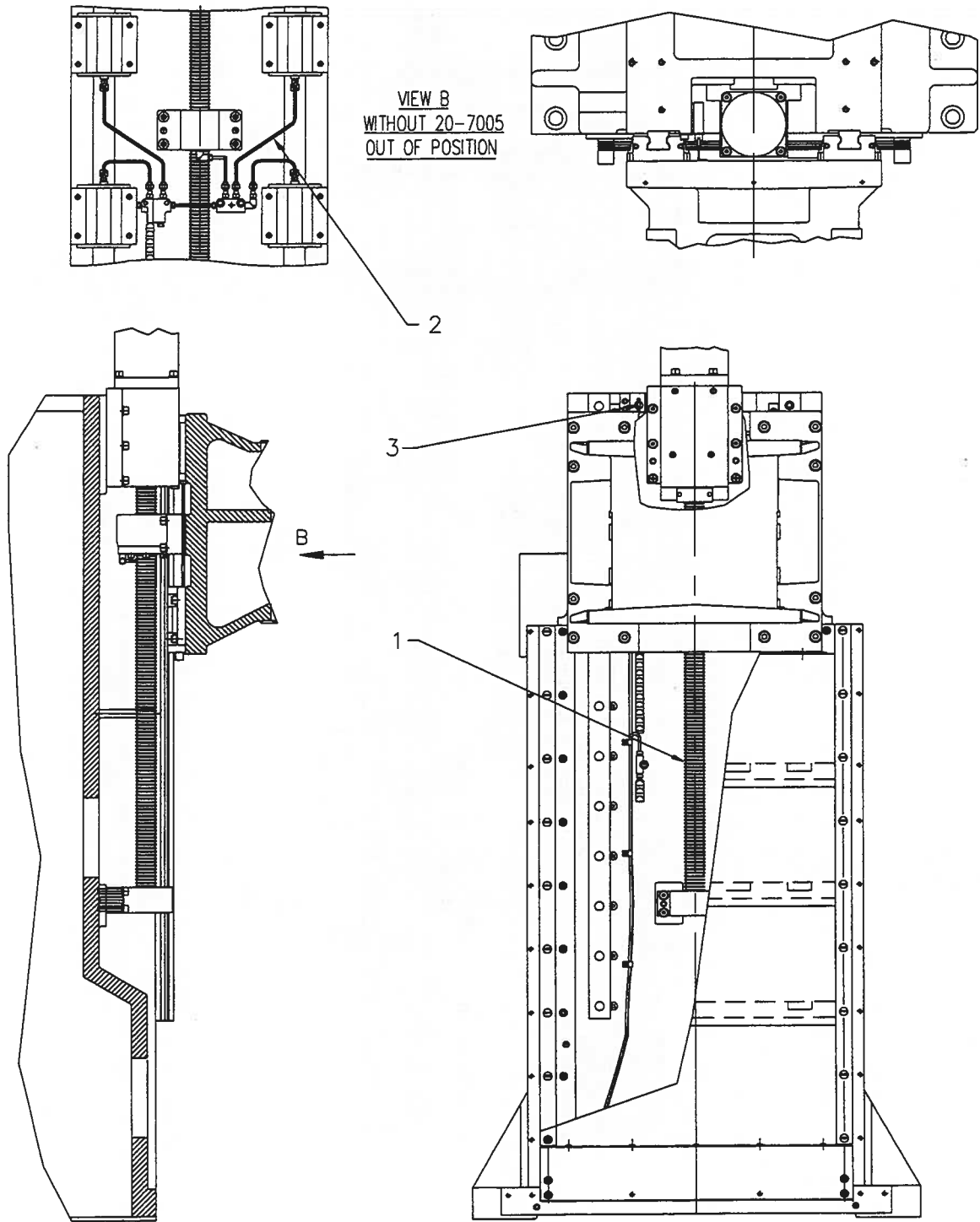
CHAPTER VII

ASSEMBLY DRAWINGS



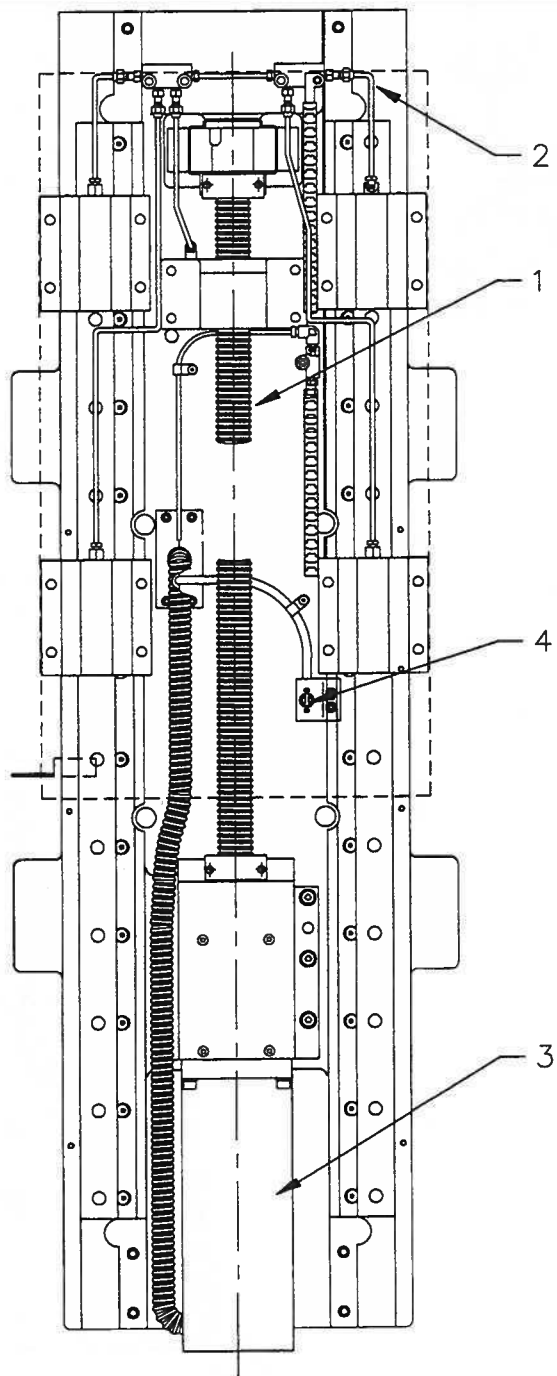
- 1 - 30-1100 - LEAD SCREW ASSEMBLY
- 2 - 30-7524 - BASE OIL LINE ASSEMBLY
- 3 - 32-1600 - Y AXIS MOTOR ASSEMBLY
- 4 - 32-2030 - TELEMECHANIQUE SWITCH ASSEMBLY

VF -1 BASE



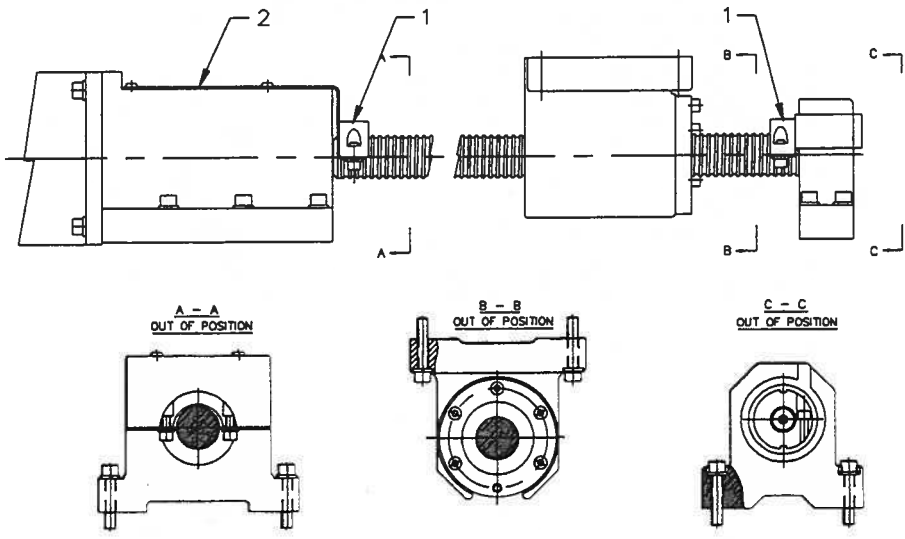
- 1 - 30-1100 - LEAD SCREW ASSEMBLY
- 2 - 30-7525 - COLUMN OIL LINE ASSEMBLY
- 3 - 32-2040 - TELEMECHANIQUE SWITCH ASSEMBLY

VF -1 COLUMN

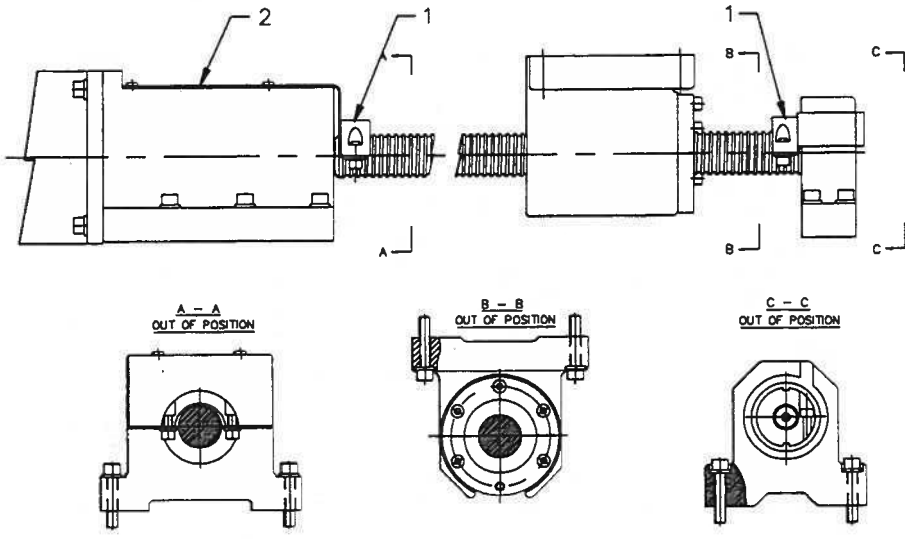


- 1 - 30-1100 - LEAD SCREW ASSEMBLY
- 2 - 30-7523 - SADDLE OIL LINE ASSEMBLY
- 3 - 32-1400 - X AXIS MOTOR ASSEMBLY
- 4 - 32-2050 - TELEMech. SWITCH ASSEMBLY

VF -1 SADDLE

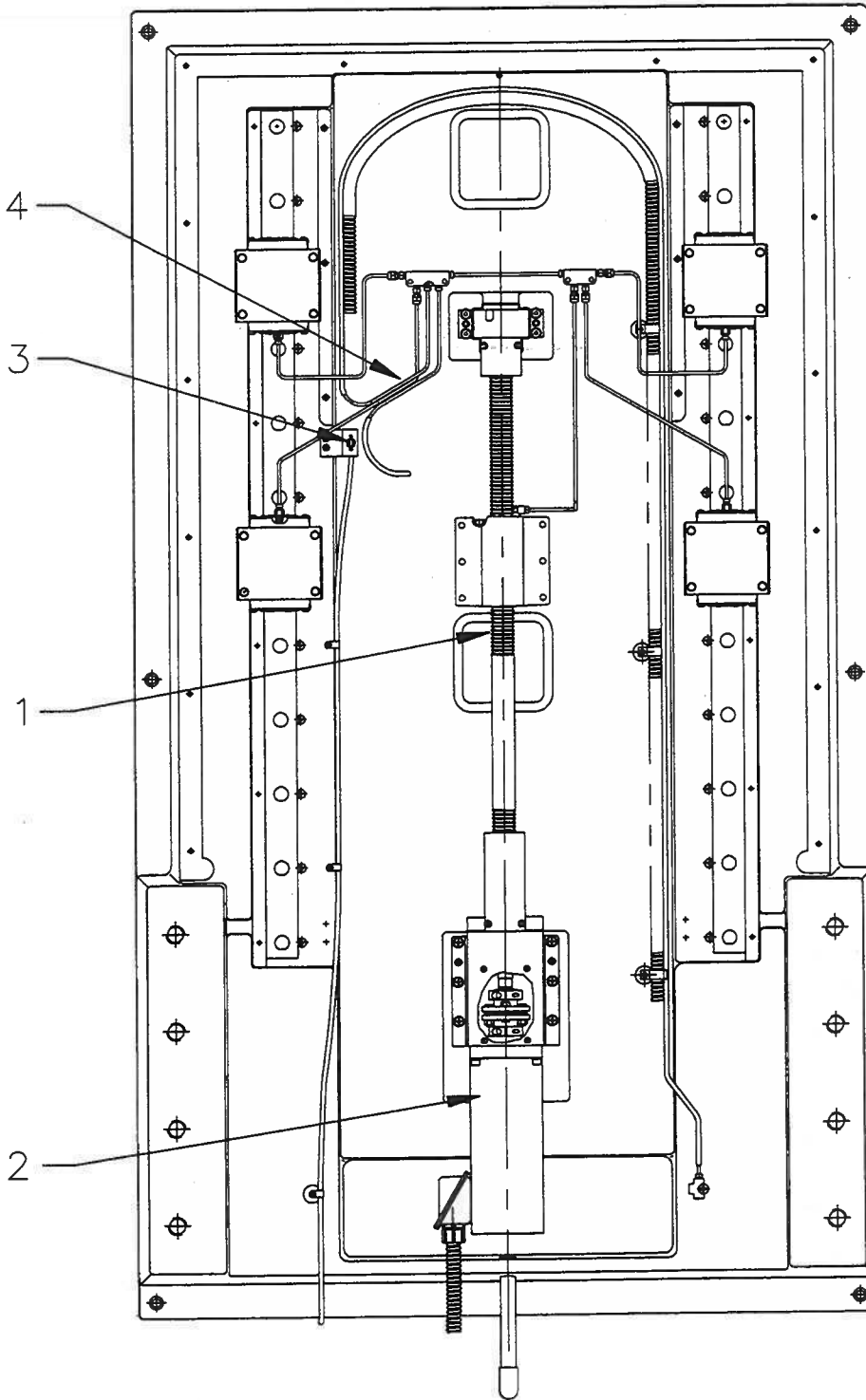


- 1 - 20-7185 - BUMPER
- 2 - 25-7042 - COVER PLATE, LEAD SCREW



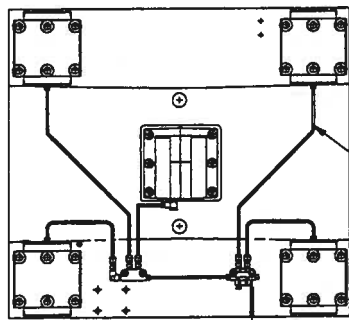
- 1 - 20-7186 - BUMPER
- 2 - 25-7042 - COVER PLATE, LEAD SCREW

VF -1 LEADSCREW

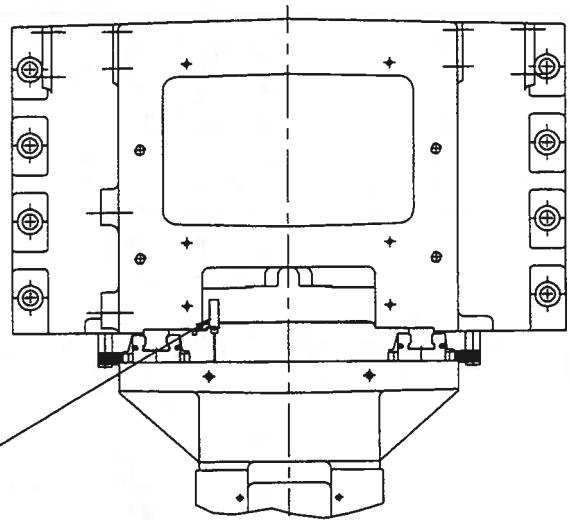


- 1 - 30-1210 - LEAD SCREW ASSEMBLY
- 2 - 32-1600 - MOTOR ASSEMBLY
- 3 - 32-2031 - TELEMECHANICS SWITCH ASSEMBLY
- 4 - 30-7526 - BASE OIL LINE ASSEMBLY

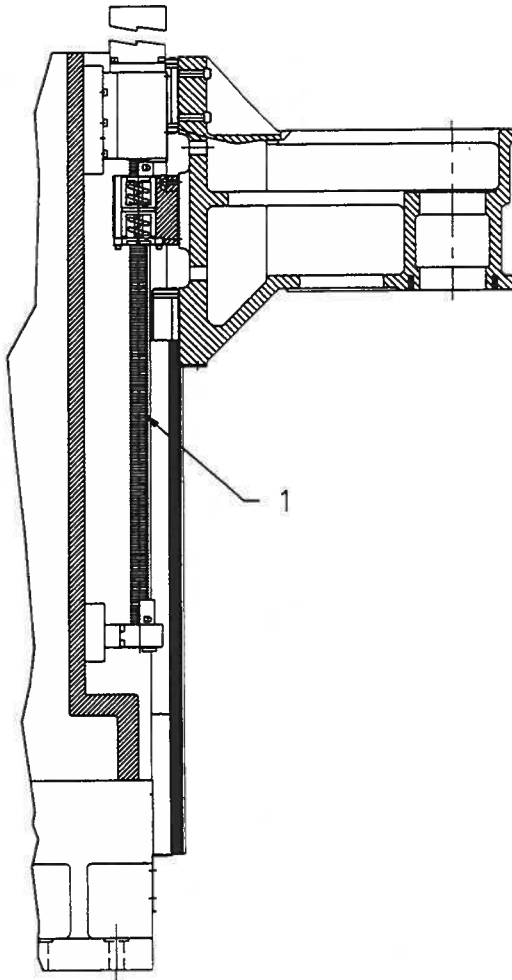
VF -3 BASE



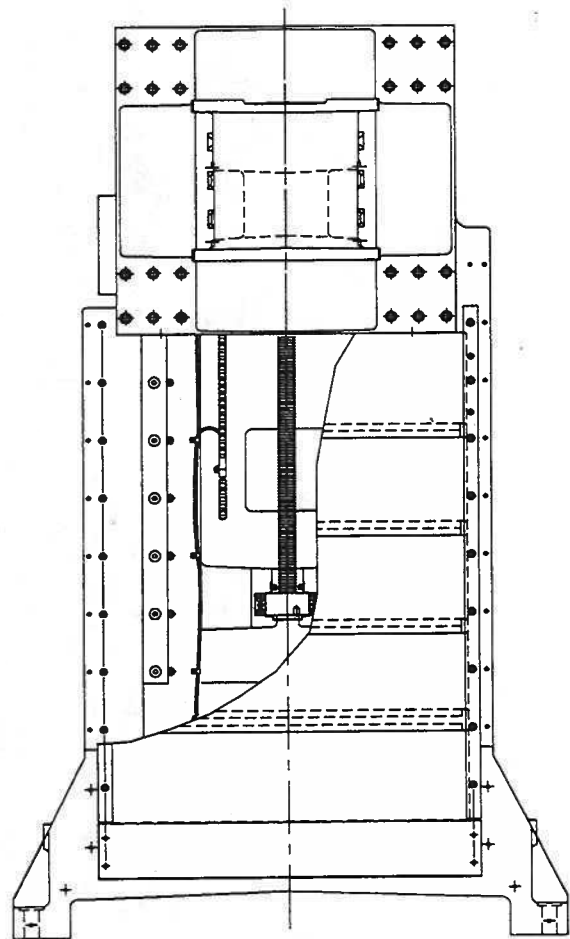
2



3

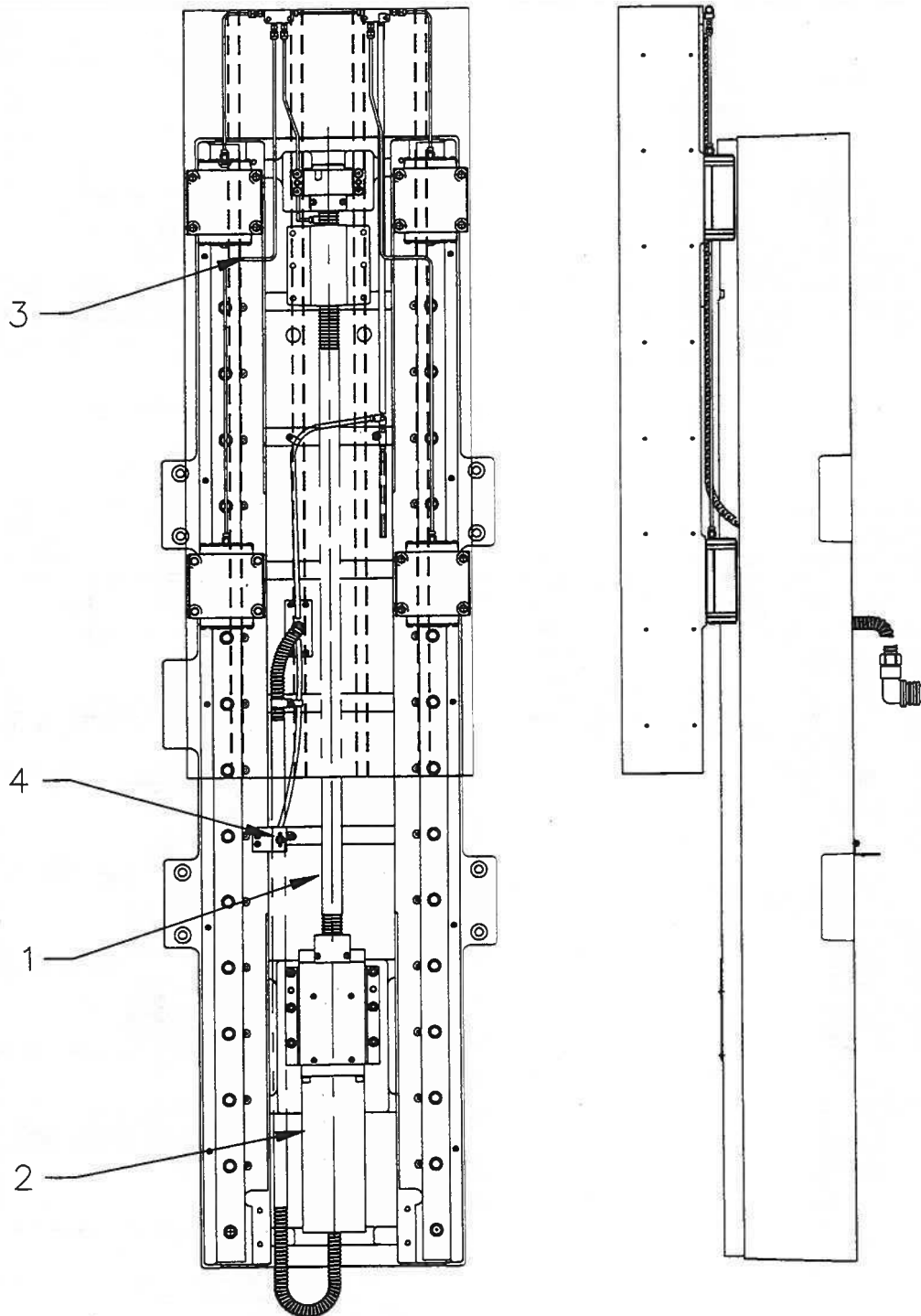


1



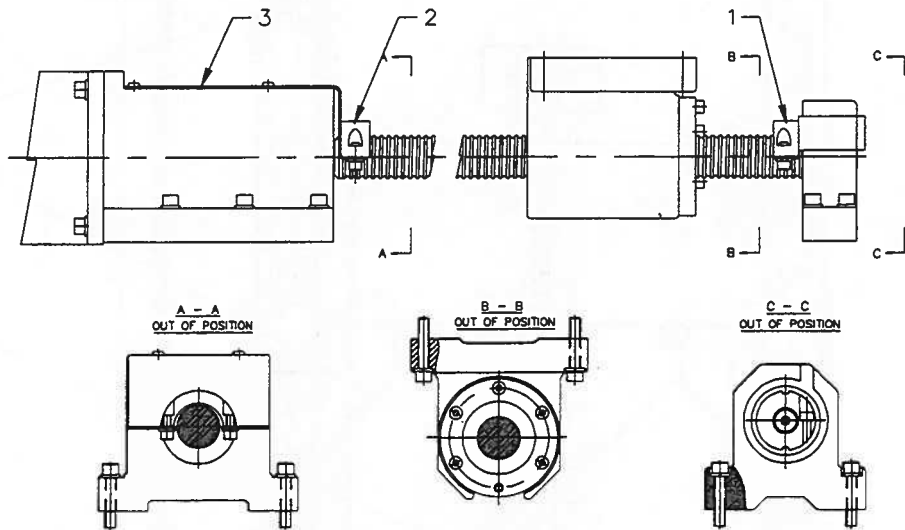
- 1 - 30-1210 - LEAD SCREW ASSEMBLY
2 - 32-7528 - COLUMN OIL LINE ASSEMBLY
3 - 32-2041 - TELEMECHANIQUE SWITCH ASSEMBLY

VF -3 COLUMN



- 1 - 30-1200 - LEAD SCREW ASSEMBLY
- 2 - 30-1401 - MOTOR ASSEMBLY
- 3 - 30-7527 - SADDLE OIL LINE ASSEMBLY
- 4 - 32-2050 - TELEMECHANIQUE SWITCH ASSEMBLY

VF-3 SADDLE

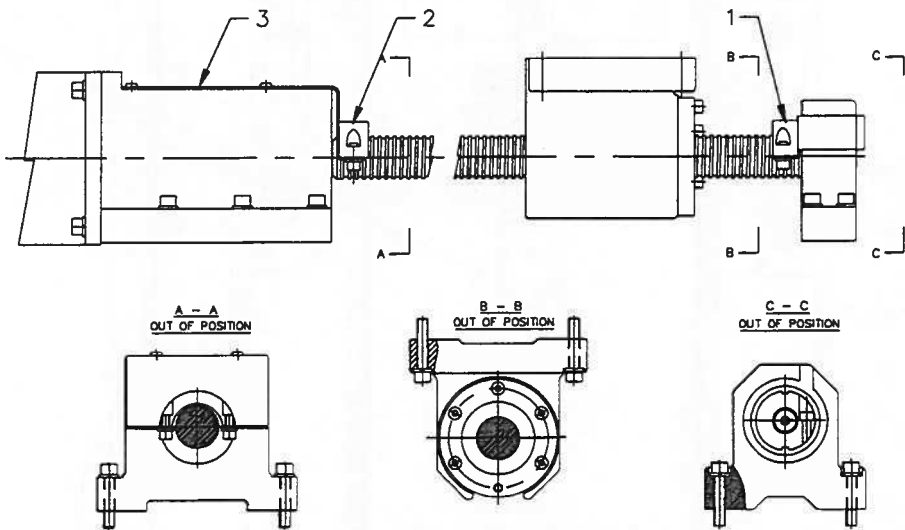


A - A
OUT OF POSITION

B - B
OUT OF POSITION

C - C
OUT OF POSITION

- 1 - 20-9057 - BUMPER
- 2 - 20-9058 - BUMPER
- 3 - 25-7042 - COVER PLATE, LEAD SCREW



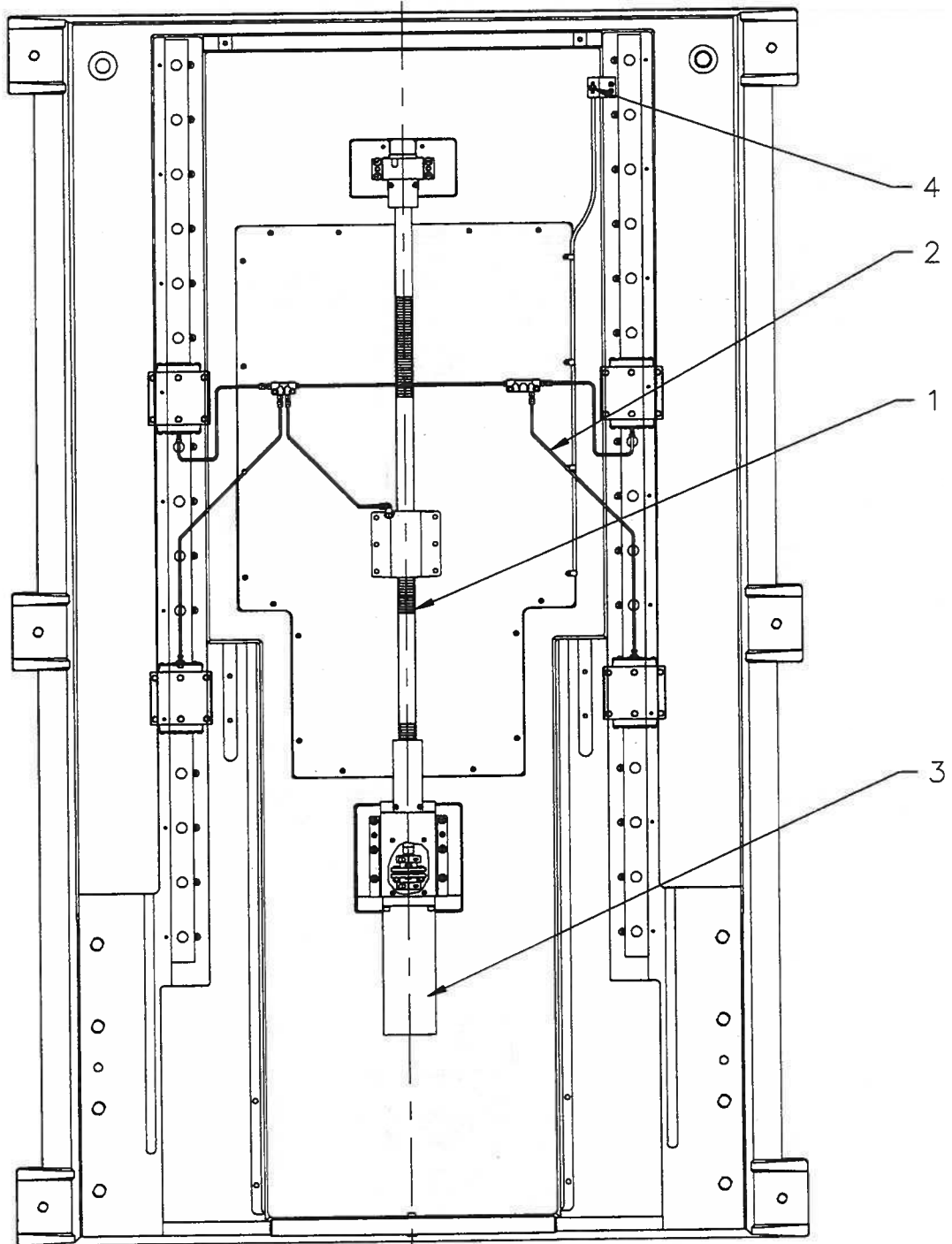
A - A
OUT OF POSITION

B - B
OUT OF POSITION

C - C
OUT OF POSITION

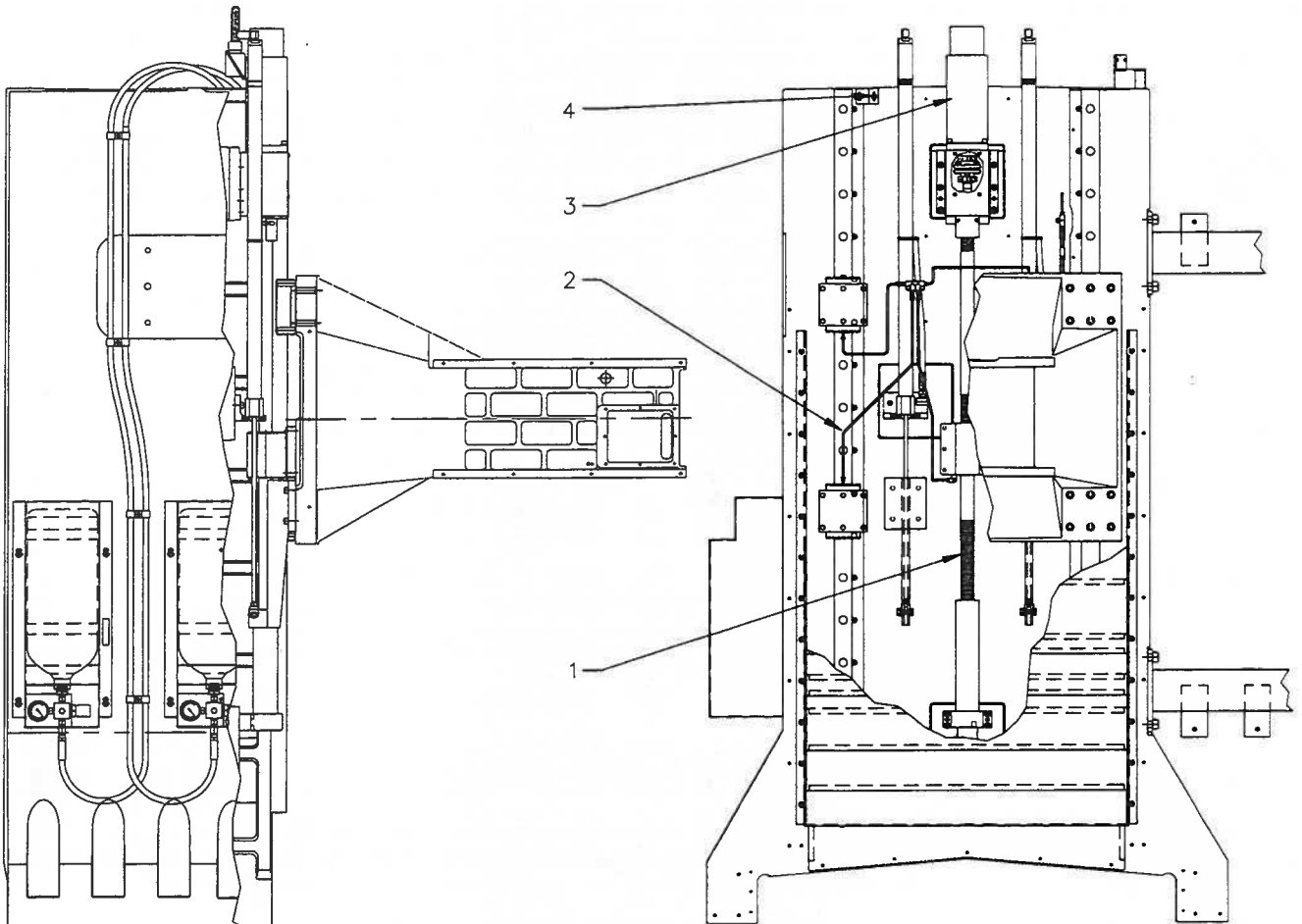
- 1 - 20-9095 - BUMPER
- 2 - 20-9096 - BUMPER
- 3 - 25-7042 - COVER PLATE, LEAD SCREW

VF -3 LEADSCREW



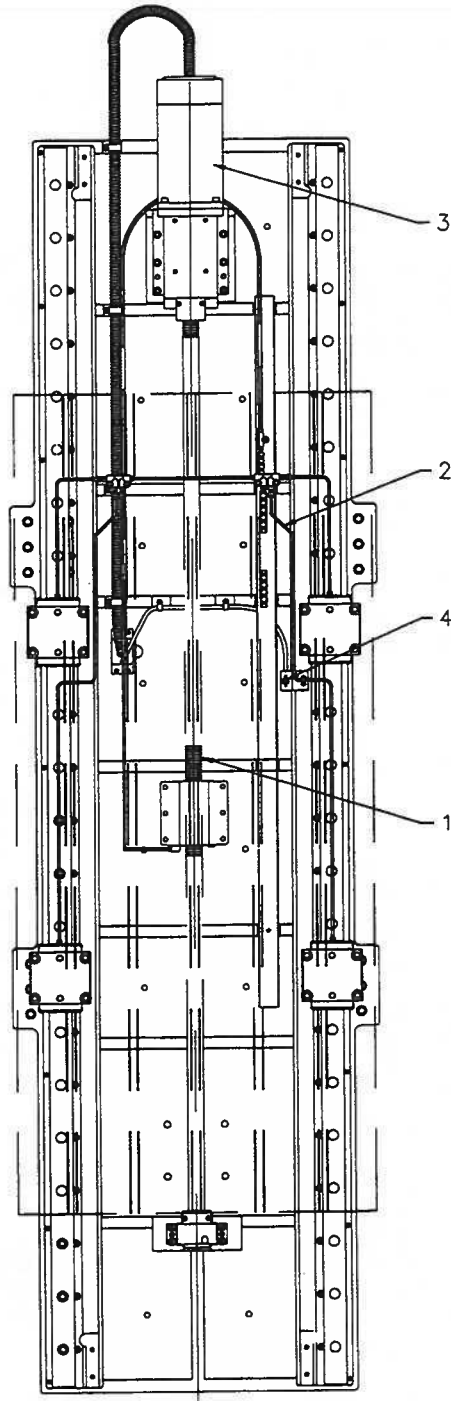
- 1 - 30-1200 - LEAD SCREW ASSEMBLY
- 2 - 30-7420 - BASE OIL LINE ASSEMBLY
- 3 - 32-1780 - Z AXIS MOTOR ASSEMBLY
- 4 - 32-5056 - LIMIT SWITCH ASSEMBLY

VF -6 BASE



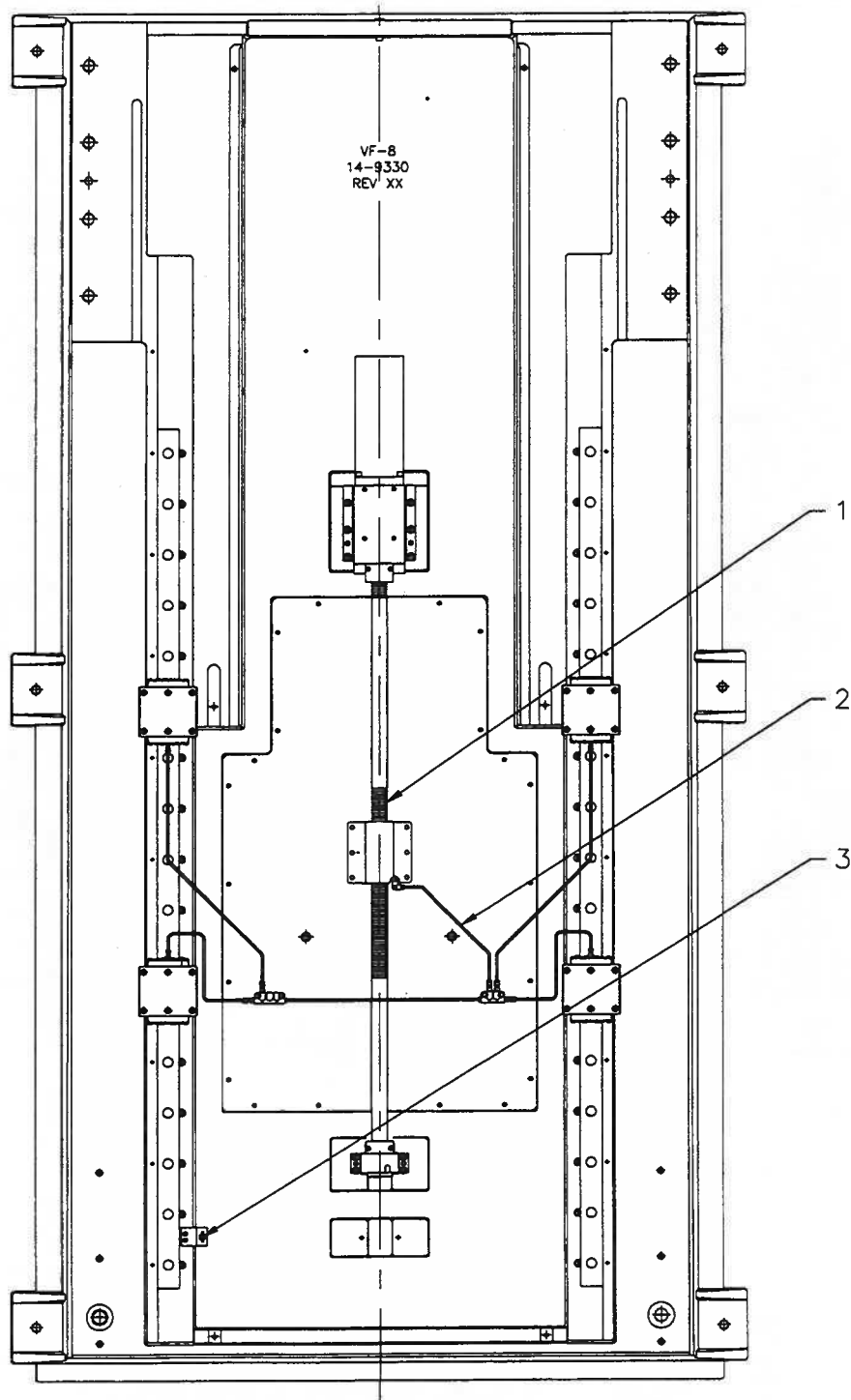
- 1 - 30-1200 - LEAD SCREW ASSEMBLY
- 2 - 30-7400 - COLUMN OIL LINE ASSEMBLY
- 3 - 32-1780 - Z AXIS MOTOR ASSEMBLY
- 4 - 32-2050 - TELEMECHANIQUE SWITCH ASSEMBLY

VF -6 COLUMN



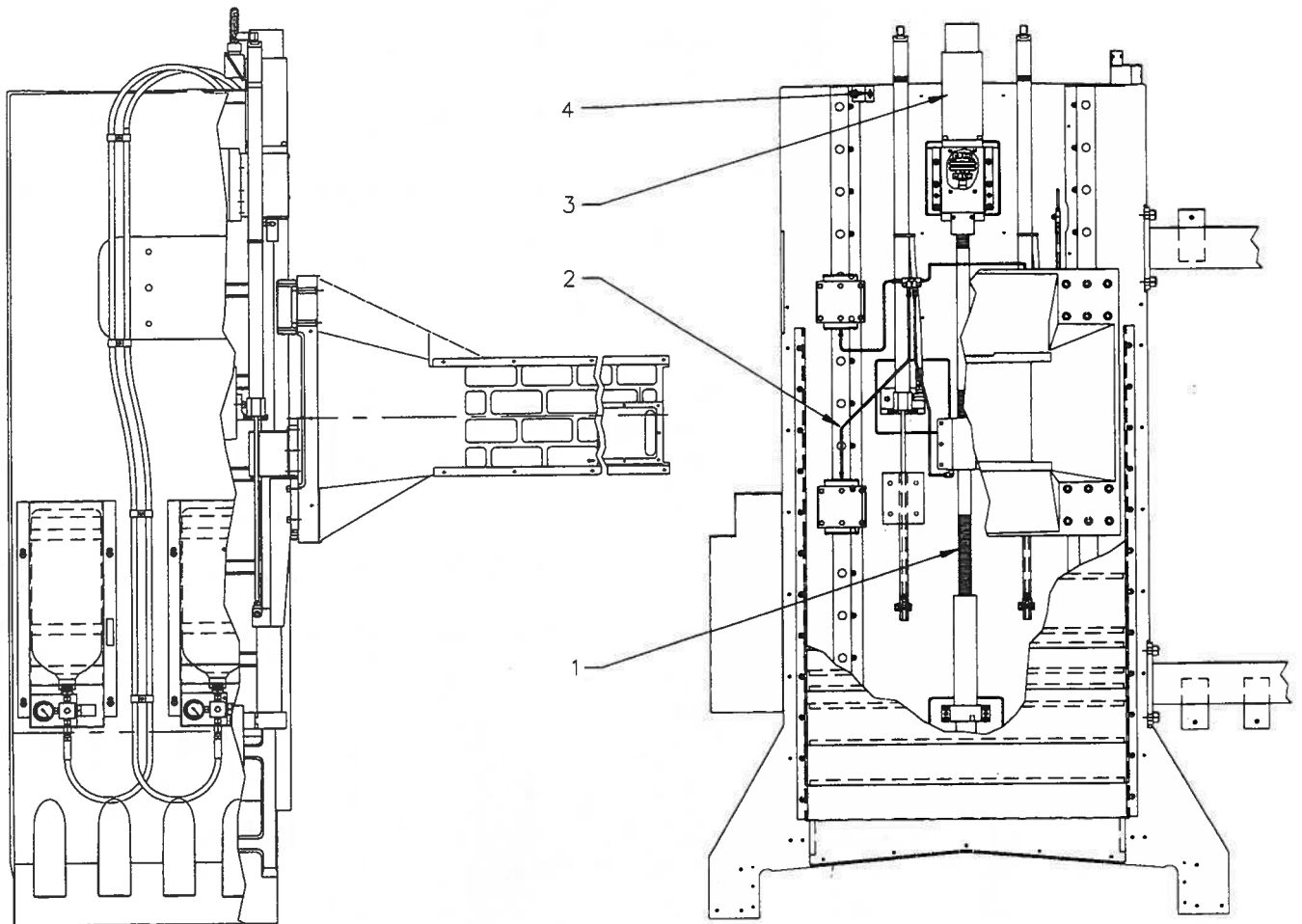
- 1 - 30-1240 - LEAD SCREW ASSEMBLY
- 2 - 30-7410 - SADDLE OIL LINE ASSEMBLY
- 3 - 32-1406 - X AXIS MOTOR ASSEMBLY
- 4 - 32-2051 - TELEMECHANIQUE SWITCH ASSEMBLY

VF -6 SADDLE



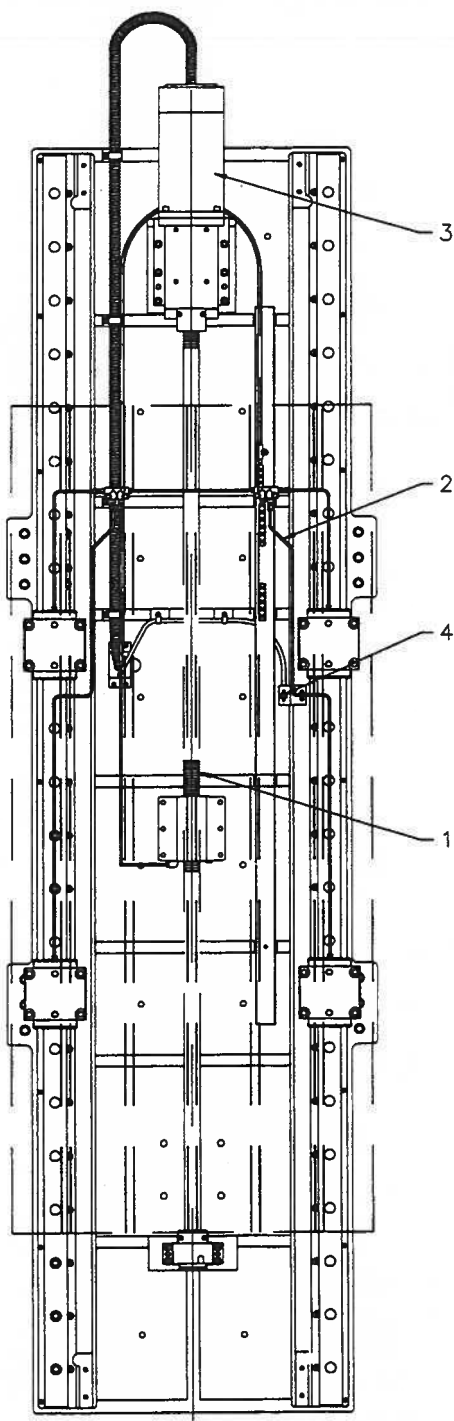
- 1 - 30-1200 - LEAD SCREW ASSEMBLY
- 2 - 30-7420 - BASE OIL LINE ASSEMBLY
- 3 - 32 - 5056 - LIMIT SWITCH ASSEMBLY

VF -8 BASE



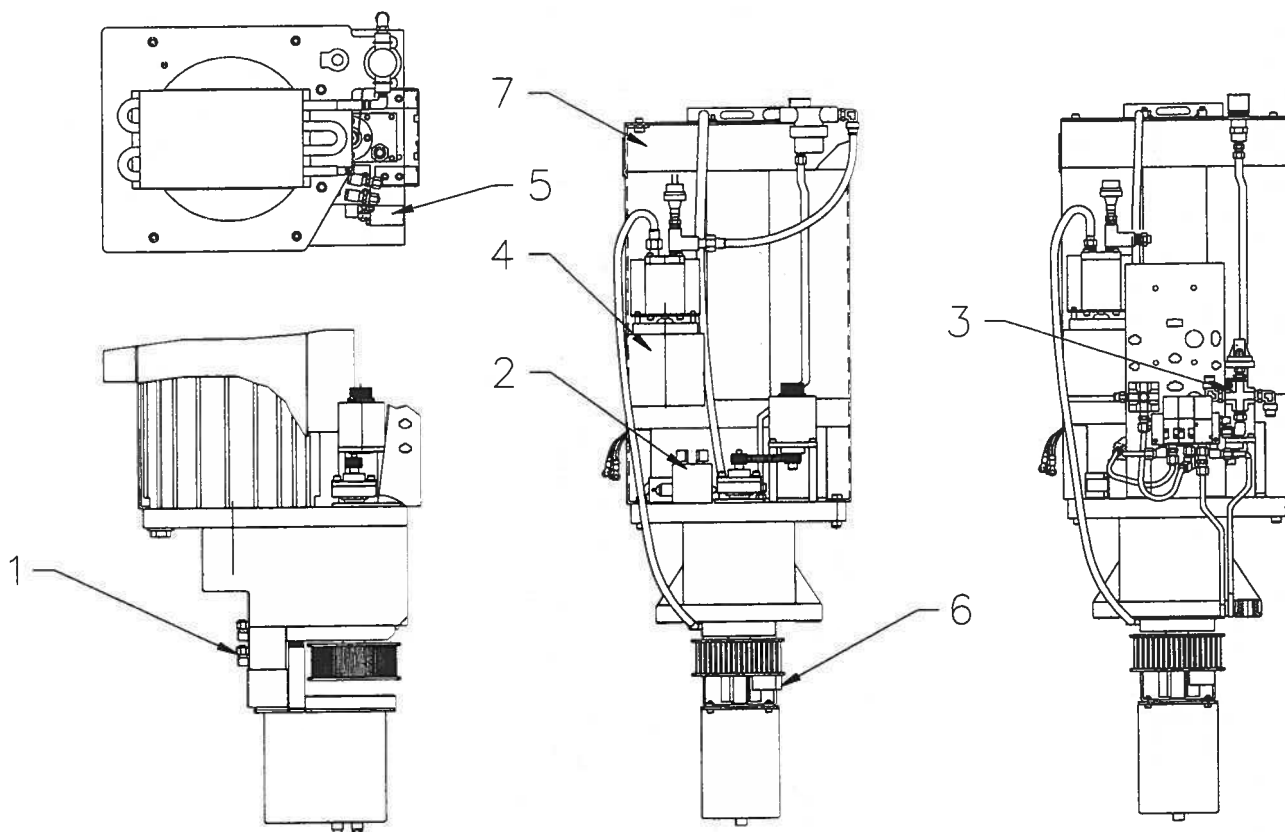
- 1 - 30-1200 - LEAD SCREW ASSEMBLY
- 2 - 30-7400 - COLUMN OIL LINE ASSEMBLY
- 3 - 32-1780 - Z AXIS MOTOR ASSEMBLY
- 4 - 32-2050 - TELEMECHANIQUE SWITCH ASSEMBLY

VF -8 COLUMN



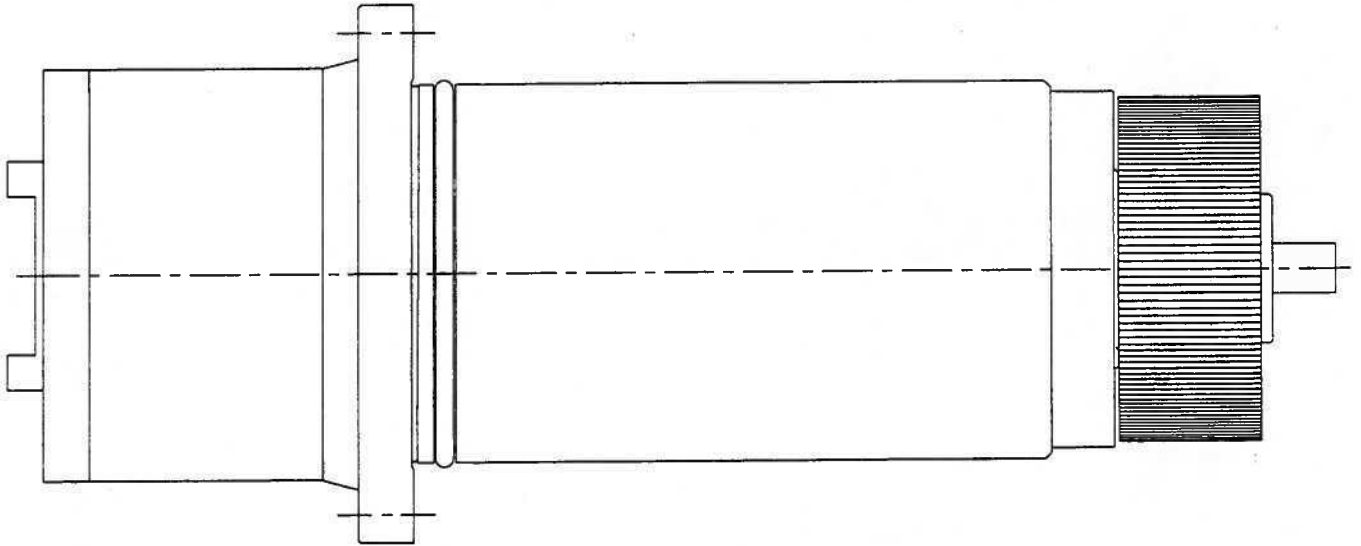
- 1 - 30-1240 - LEAD SCREW ASSEMBLY
- 2 - 30-7410 - SADDLE OIL LINE ASSEMBLY
- 3 - 32-1406 - X AXIS MOTOR ASSEMBLY
- 4 - 32-2051 - TELEMECHANICS SWITCH ASSEMBLY

VF -8 SADDLE

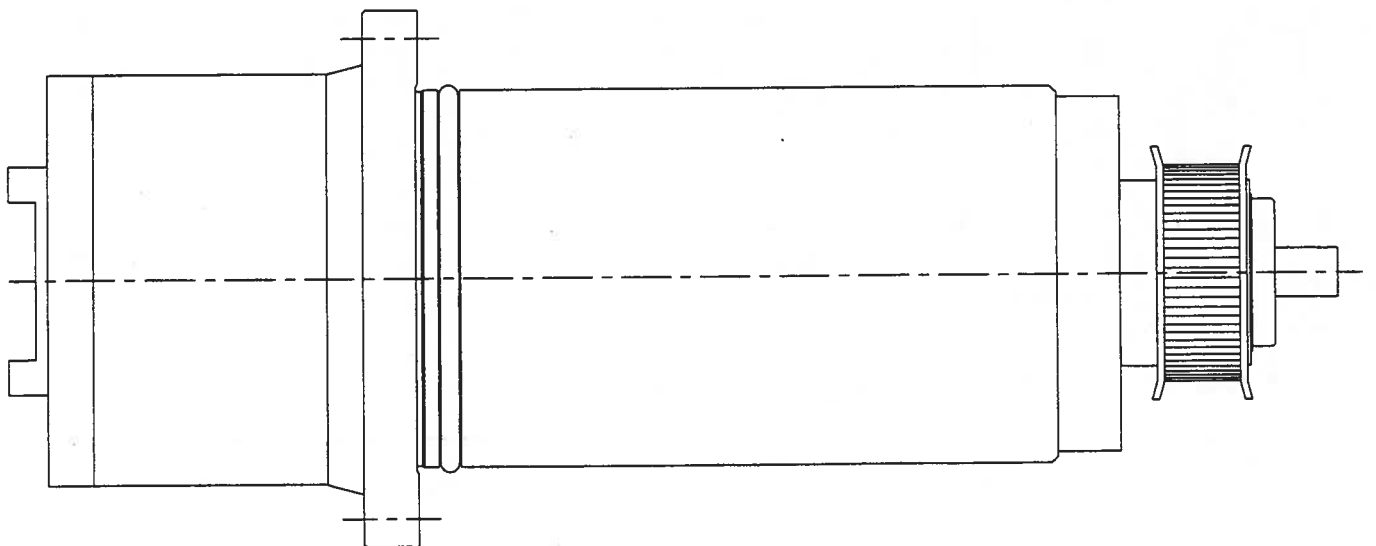


- 3130B - PISTON ASSEMBLY
- 3140A - SHOT PIN ASSEMBLY
- 3150C - AIR MANIFOLD ASSEMBLY
- 3260 - OIL GEAR PUMP ASSEMBLY
- 2010 - TELEMECHANIQUE SWITCH ASSEMBLY
- 2011 - TELEMECHANIQUE SWITCH ASSEMBLY
- 3035A - SPINDLE MOTOR FAN ASSEMBLY

VF -GEARBOX ASSEMBLY



VF -SERIES SPINDLE 7.5K / 10K



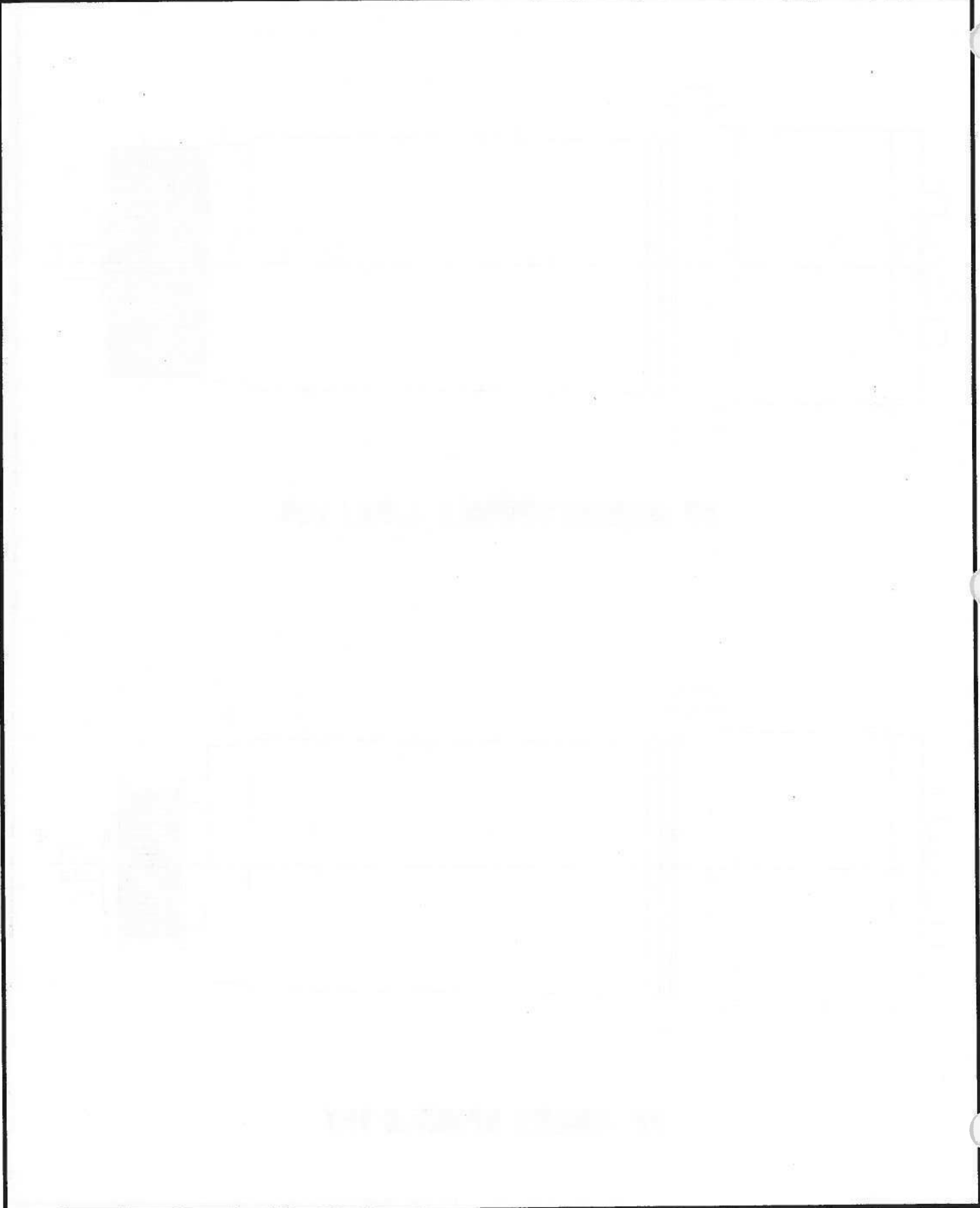
VF -SERIES SPINDLE 12K

VF-SERIES



ASSY. DRAWINGS

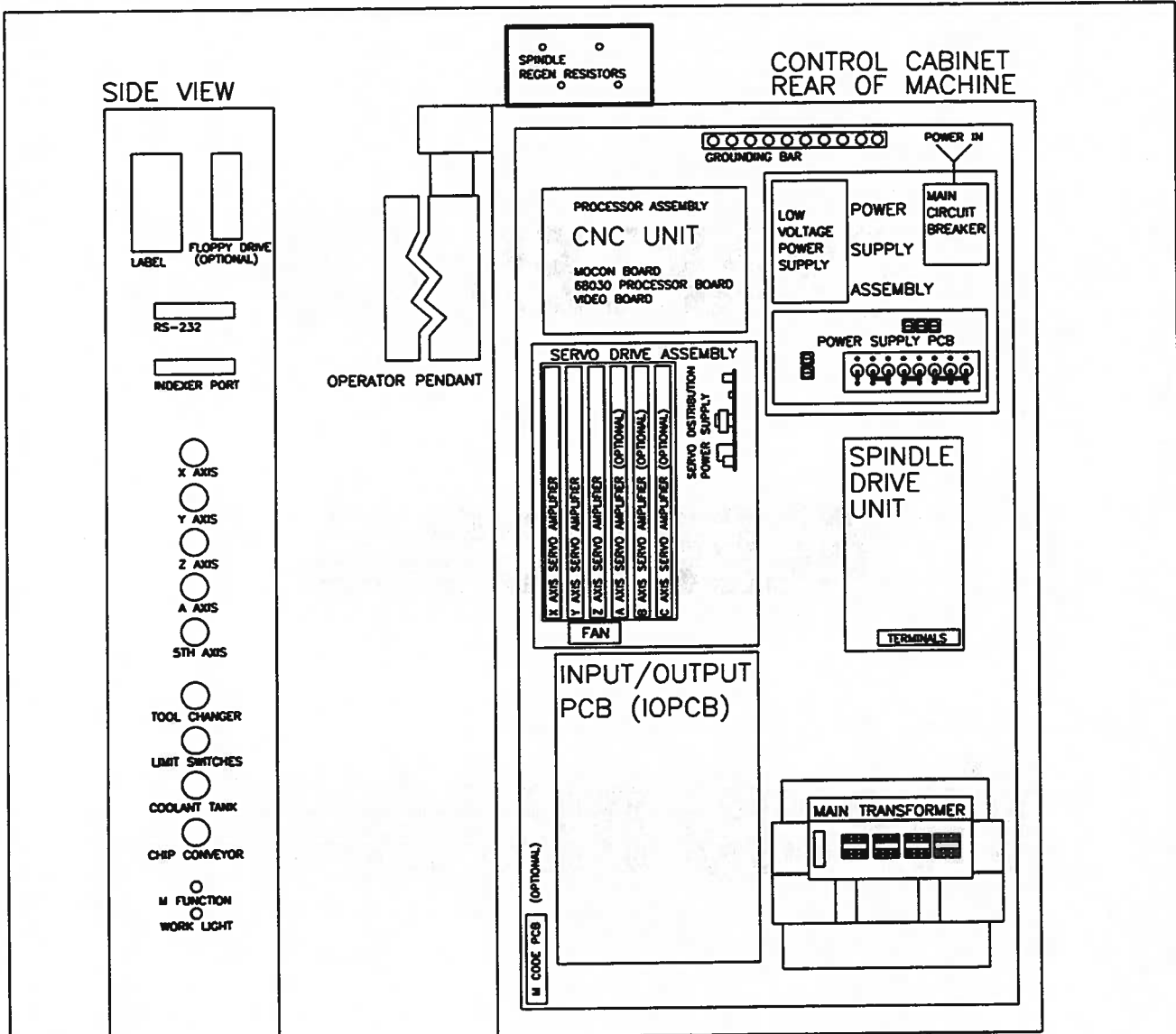
OPERATORS MANUAL



CHAPTER VIII

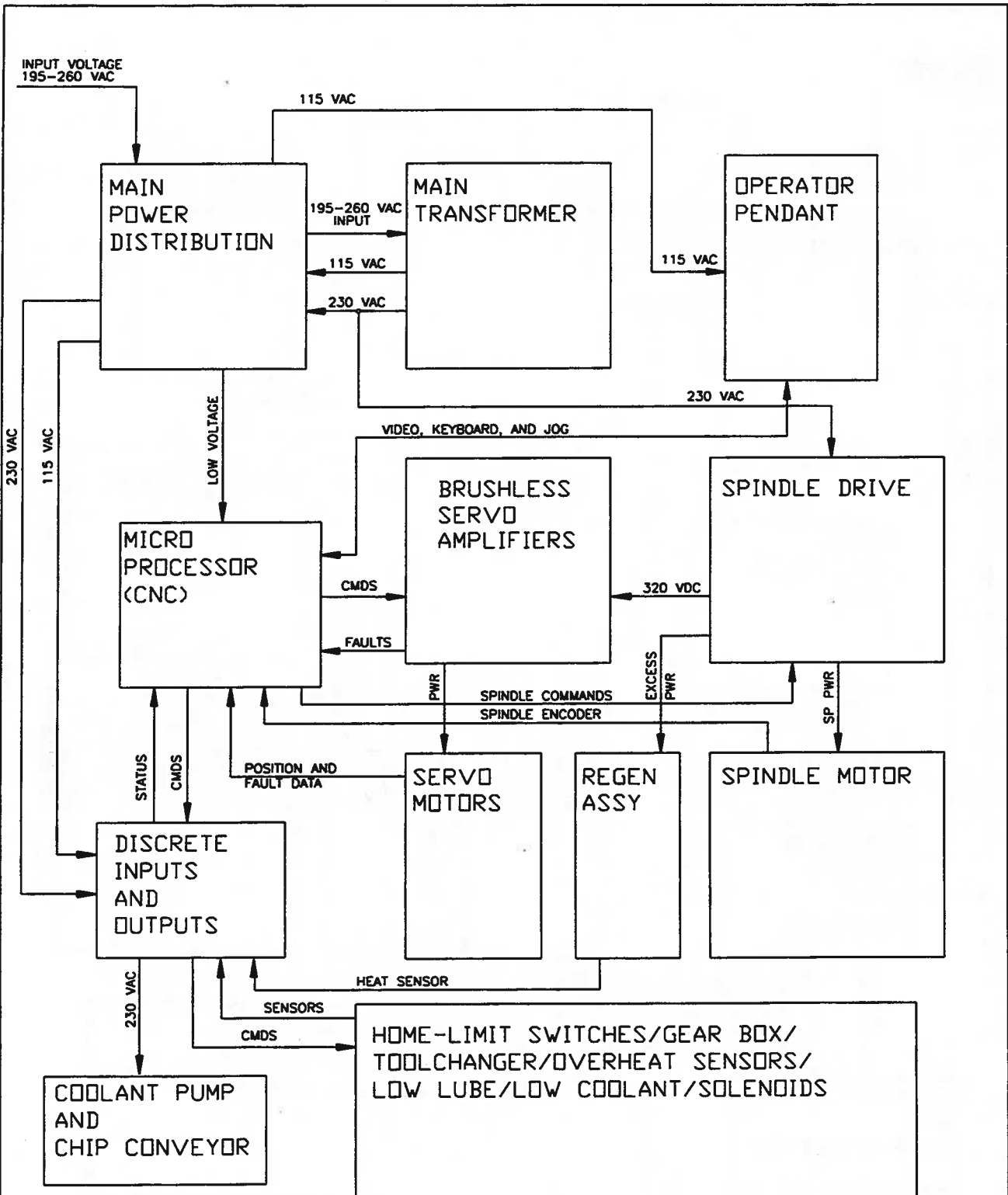
ELECTRICAL

WIRING DIAGRAMS



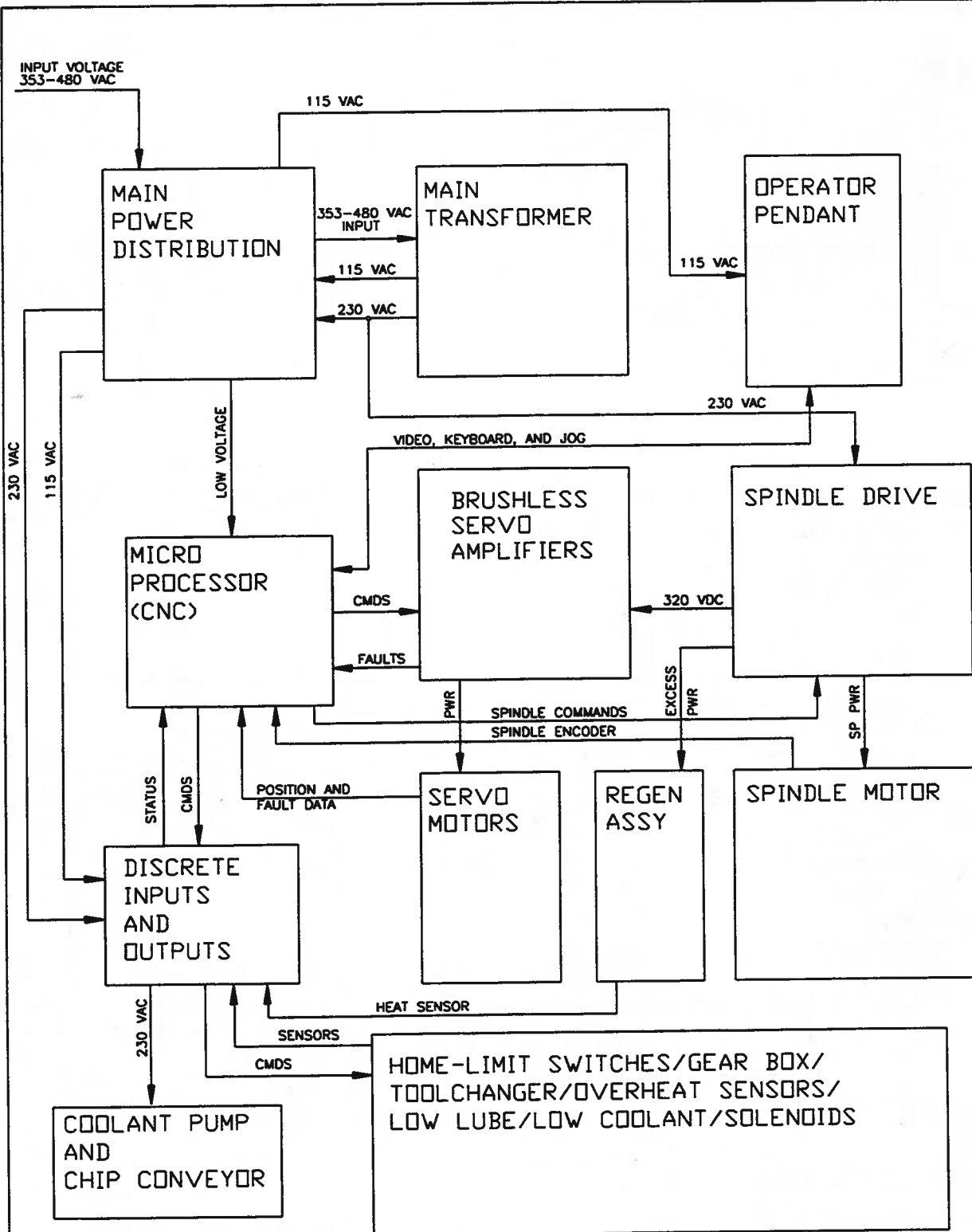
ITEM DESCRIPTION	PAGE #	ITEM DESCRIPTION	PAGE #
CNC LAYOUT	A	RELAY COIL DRIVERS, IOPCB	8-11
SYSTEM BLOCK DIAGRAM	B,C	SPINDLE DRIVE UNIT	12
CABLE INTERCONNECT DIAGRAM	D	AXIS MOTOR & ENCODER	13,14
SERVO SYSTEM	1	CABINET CONNECTORS	15
MAIN TRANSFORMER	2,3	TOOL CHANGE MOTORS	16
CNC UNIT	4	CHIP CONVEYOR/SPIGOT MOTOR	17
115VAC CIRCUITS	5	OPERATOR PENDANT	18
INPUTS IOPCB	6,7	ELECTRICAL SYMBOLS	19

CONTROL LAYOUT DIAGRAM



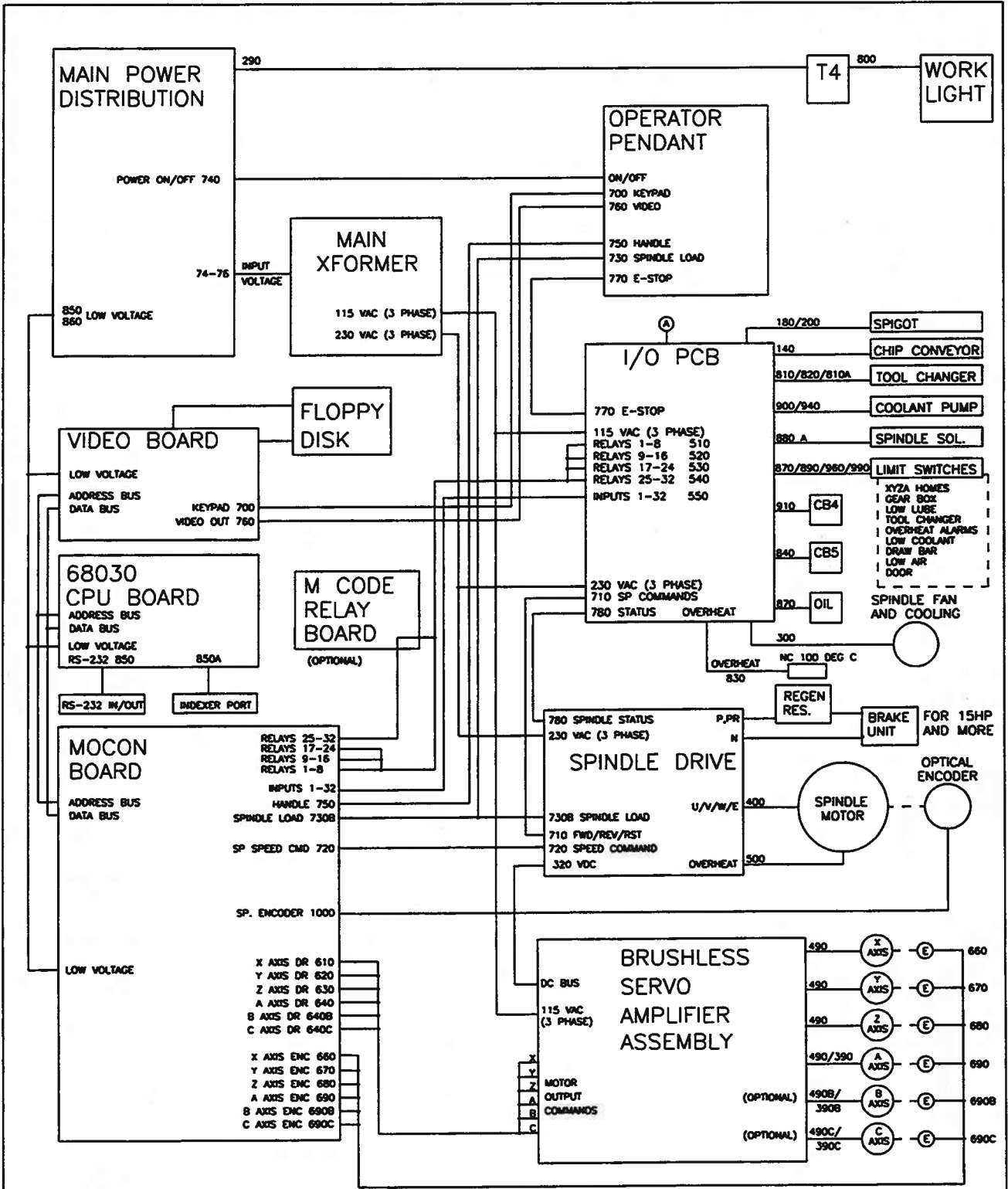
NOTE FOR HIGH VOLTAGE IN, SEE PAGE C.

SYSTEM BLOCK DIAGRAM - LOW VOLTAGE



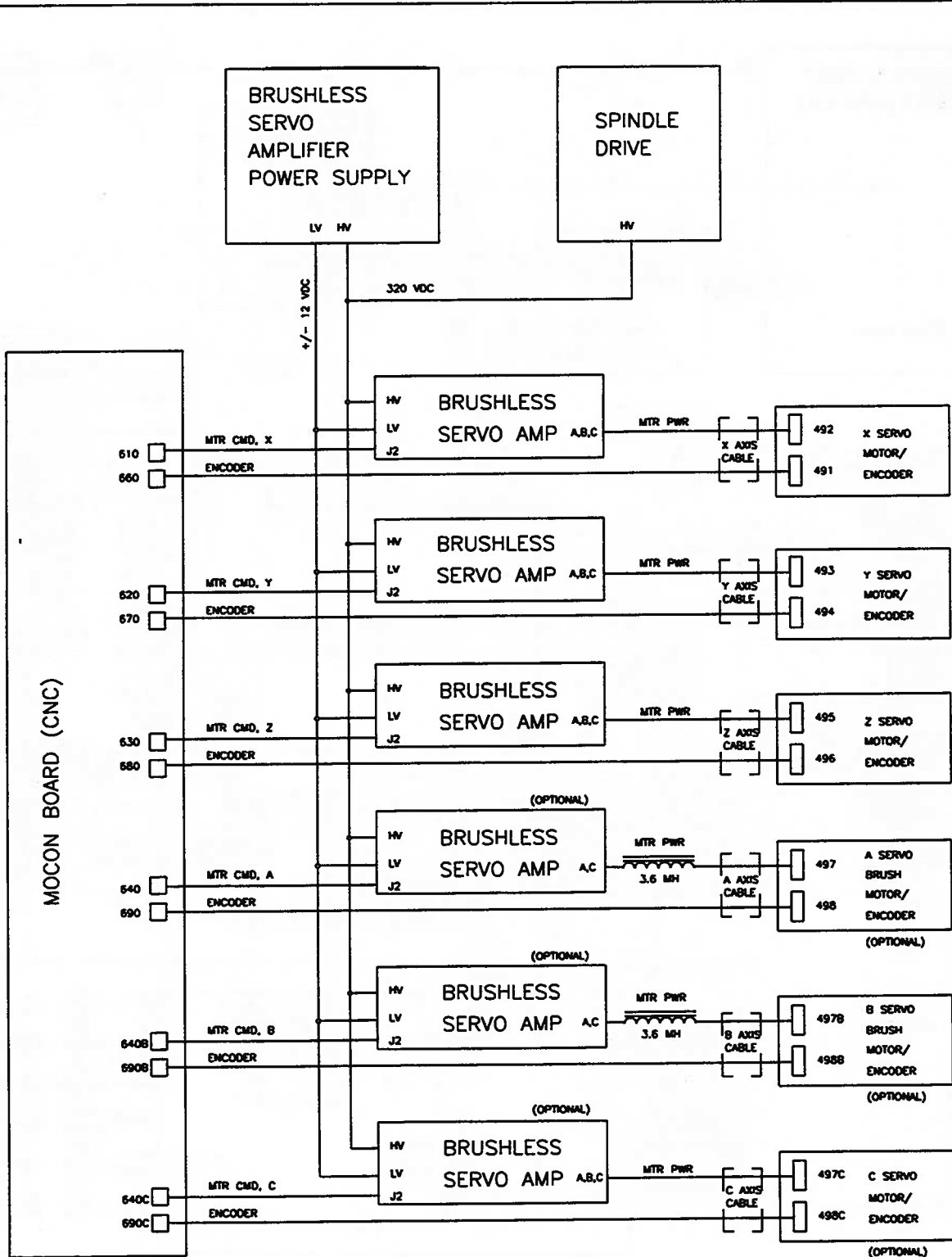
NOTE FOR LOW VOLTAGE IN, SEE PAGE B.

SYSTEM BLOCK DIAGRAM - HIGH VOLTAGE



Ⓐ 190, 1070, 90, 90A, 90B, 90C, 1050, 280

4/96 CABLE INTERCONNECT DIAGRAM HAAS AUTOMATION VF SERIES PAGE D

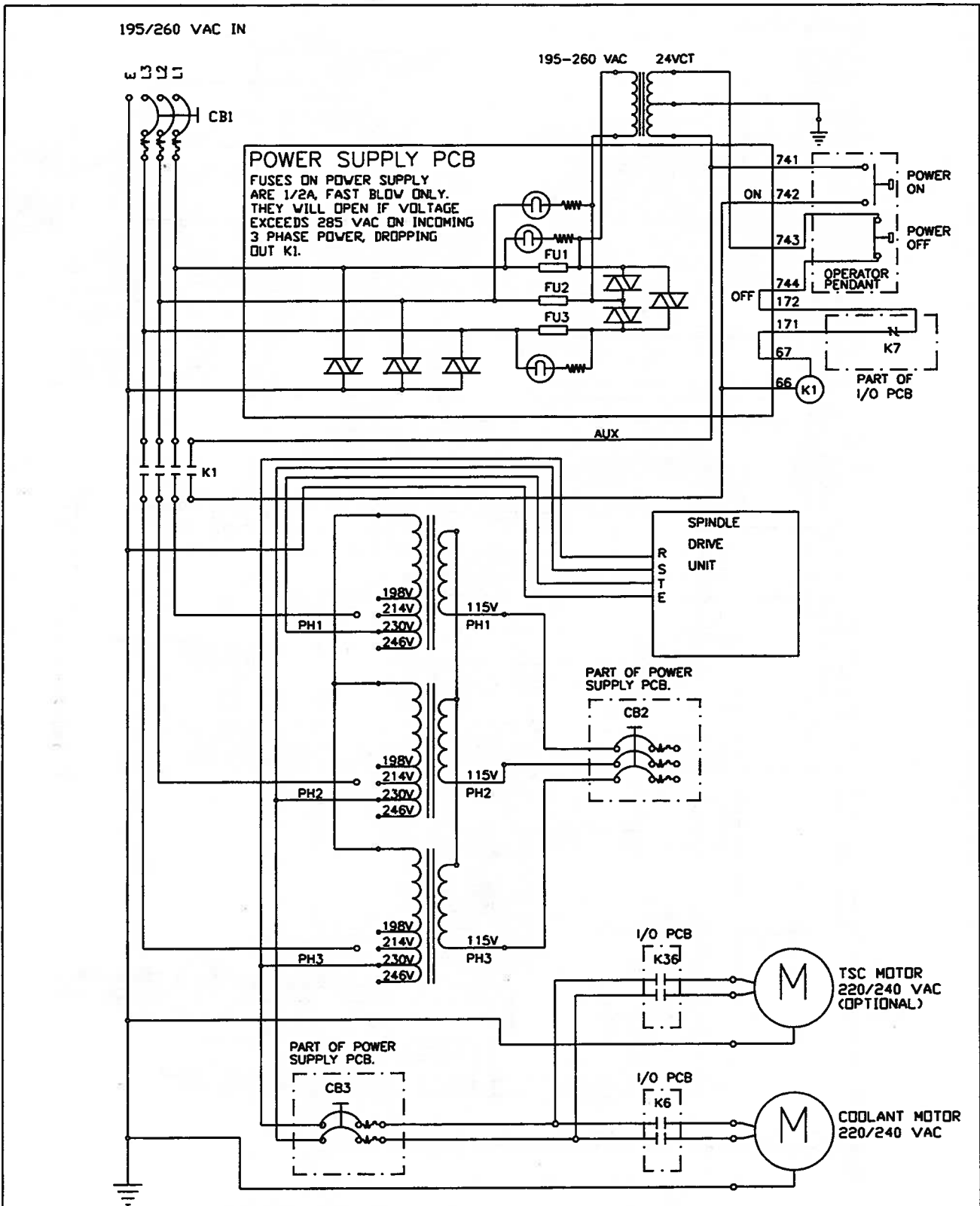


SERVO SYSTEM

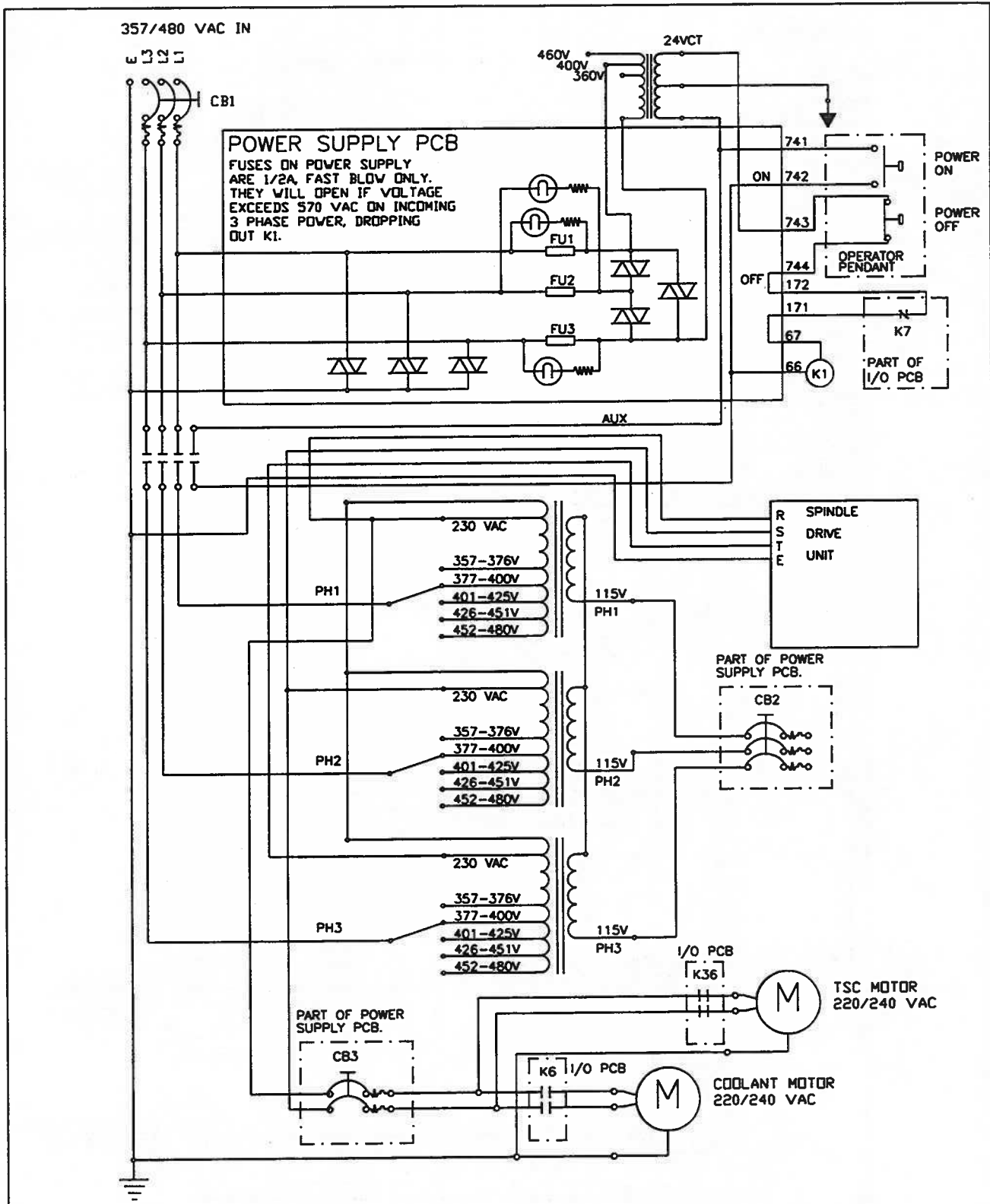
4/96

HAAS AUTOMATION

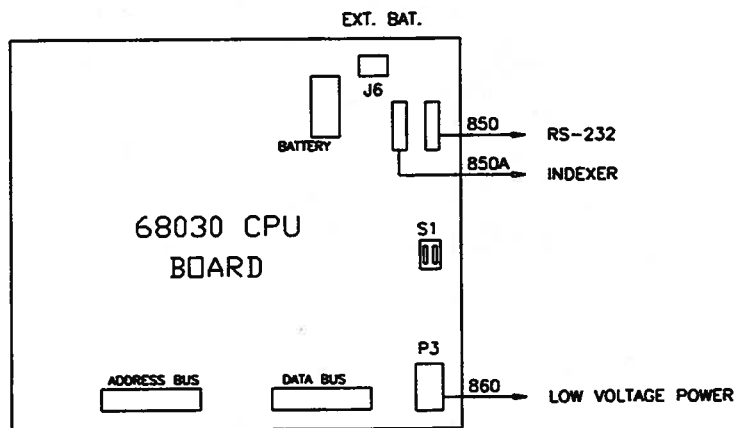
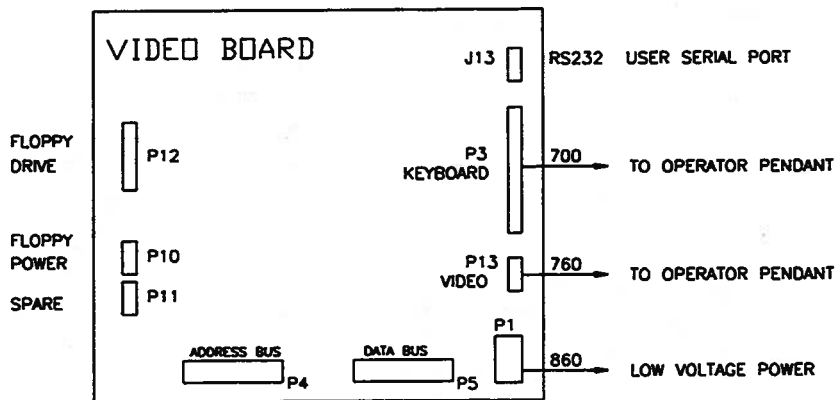
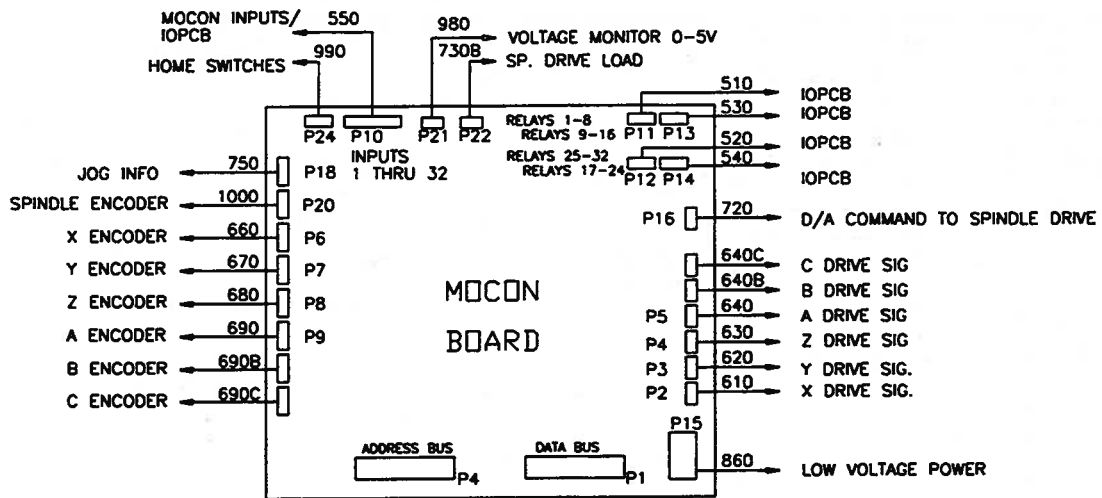
VF SERIES PAGE 1



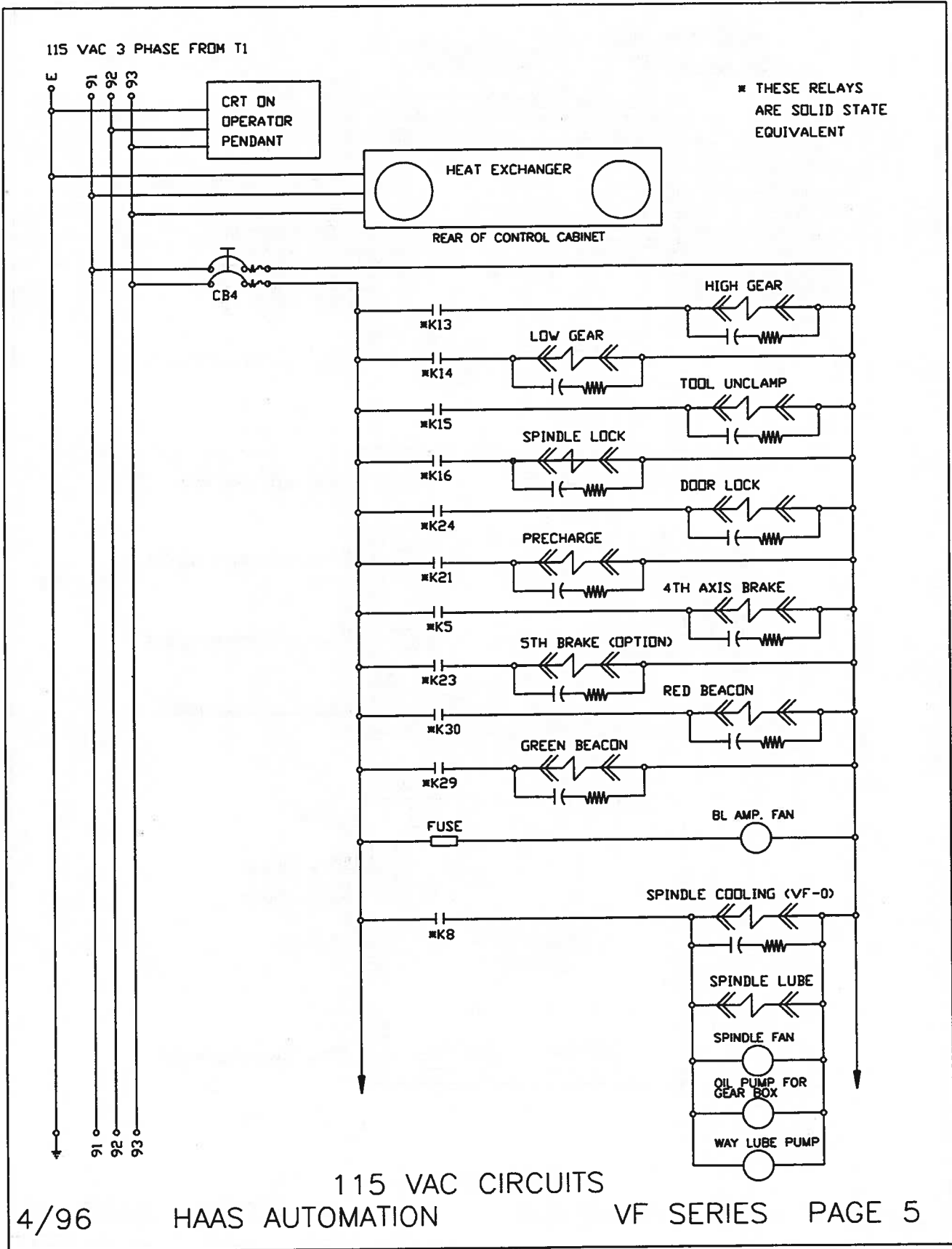
4/96 MAIN TRANSFORMER - LOW VOLTAGE HAAS AUTOMATION VF SERIES PAGE 2



4/96 MAIN TRANSFORMER - HIGH VOLTAGE HAAS AUTOMATION VF SERIES PAGE 3



CNC UNIT

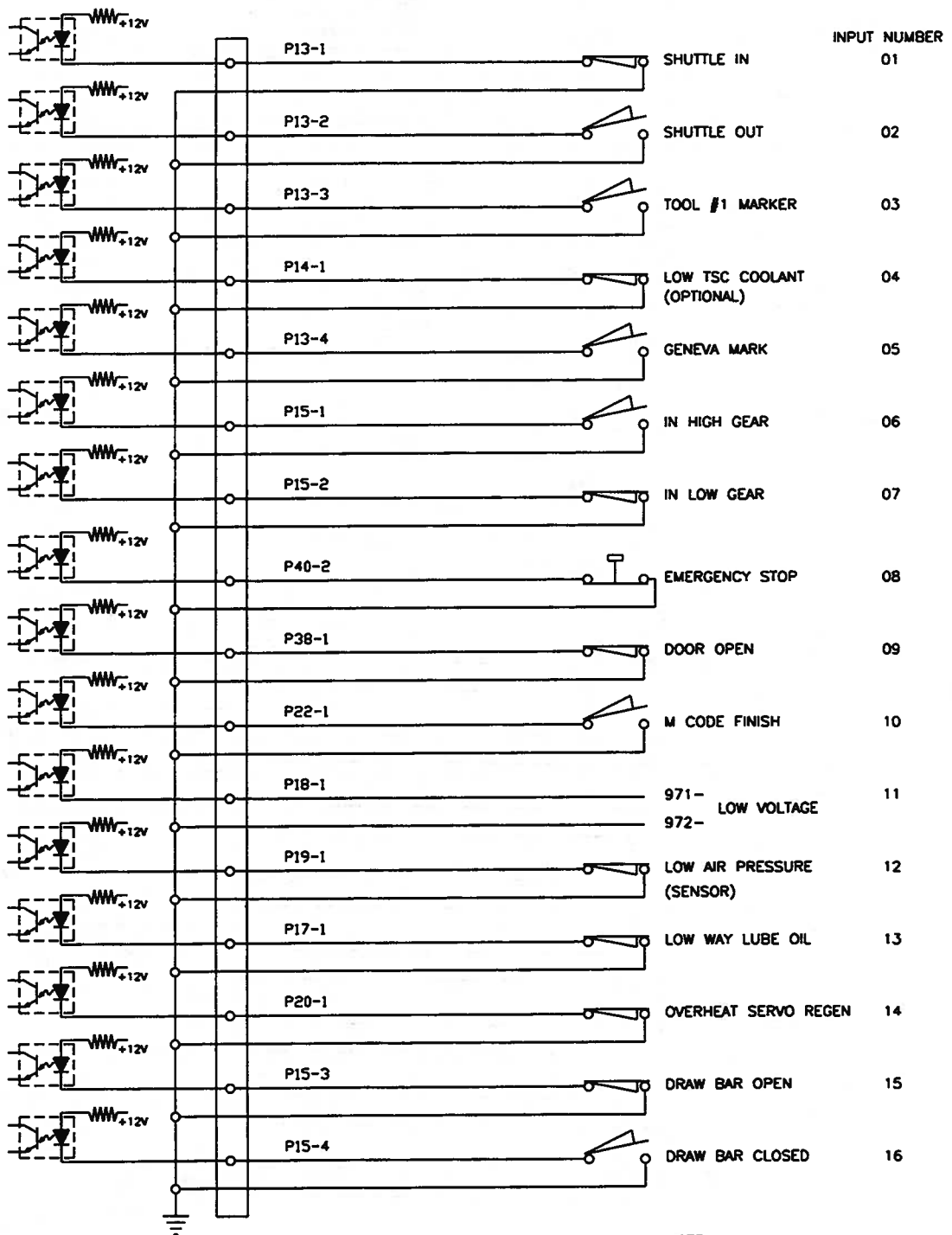


115 VAC CIRCUITS

4/96

HAAS AUTOMATION

VF SERIES PAGE 5



IOPCB CABLE 550

NOTE:
SWITCHES SHOWN ARE IN A
NON - ALARM STATE / HIGH GEAR /
SHUTTLE OUT / TURRET AT TOOL 1 POSIT.

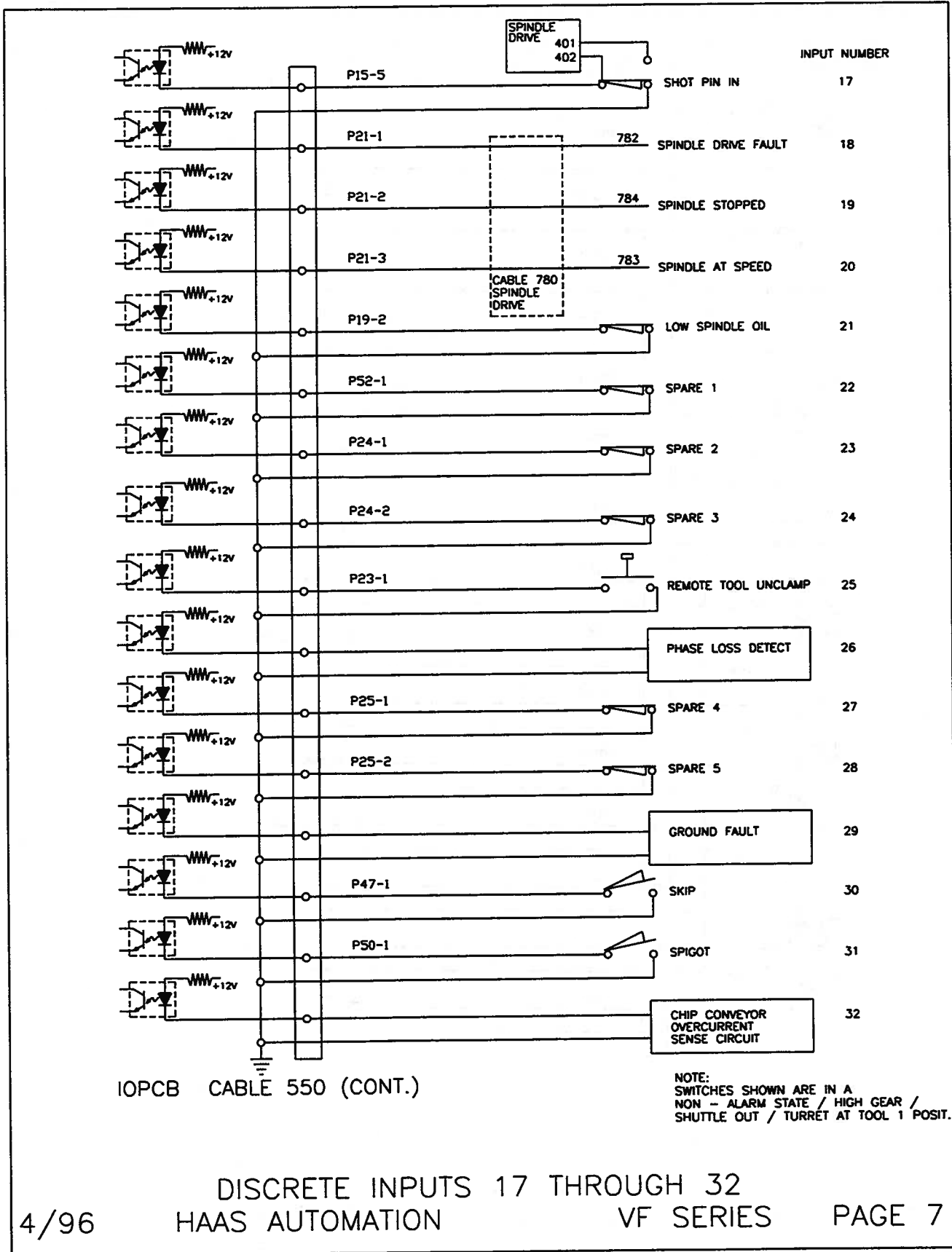
DISCRETE INPUTS 1 THROUGH 16

4/96

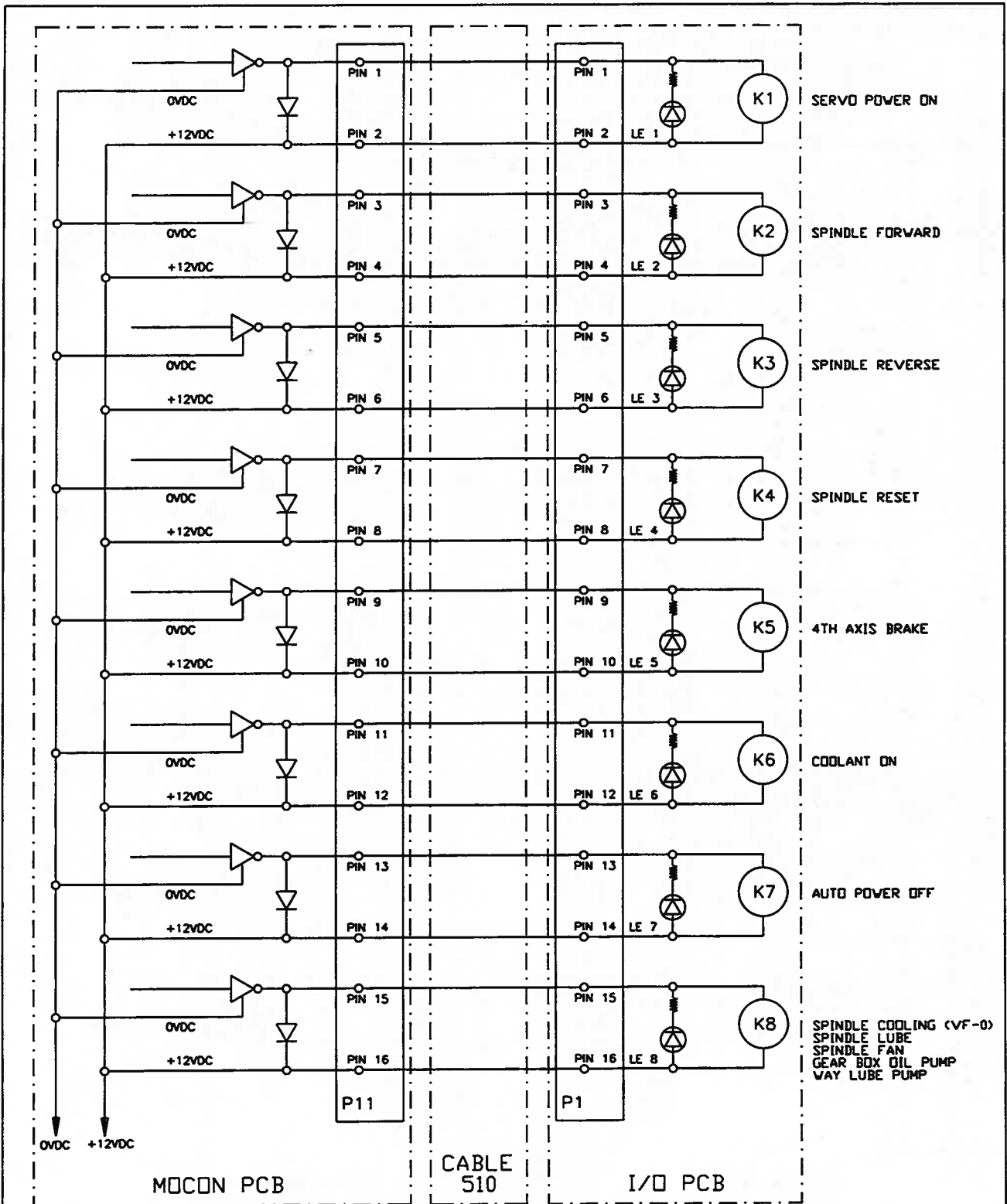
HAAS AUTOMATION

VF SERIES

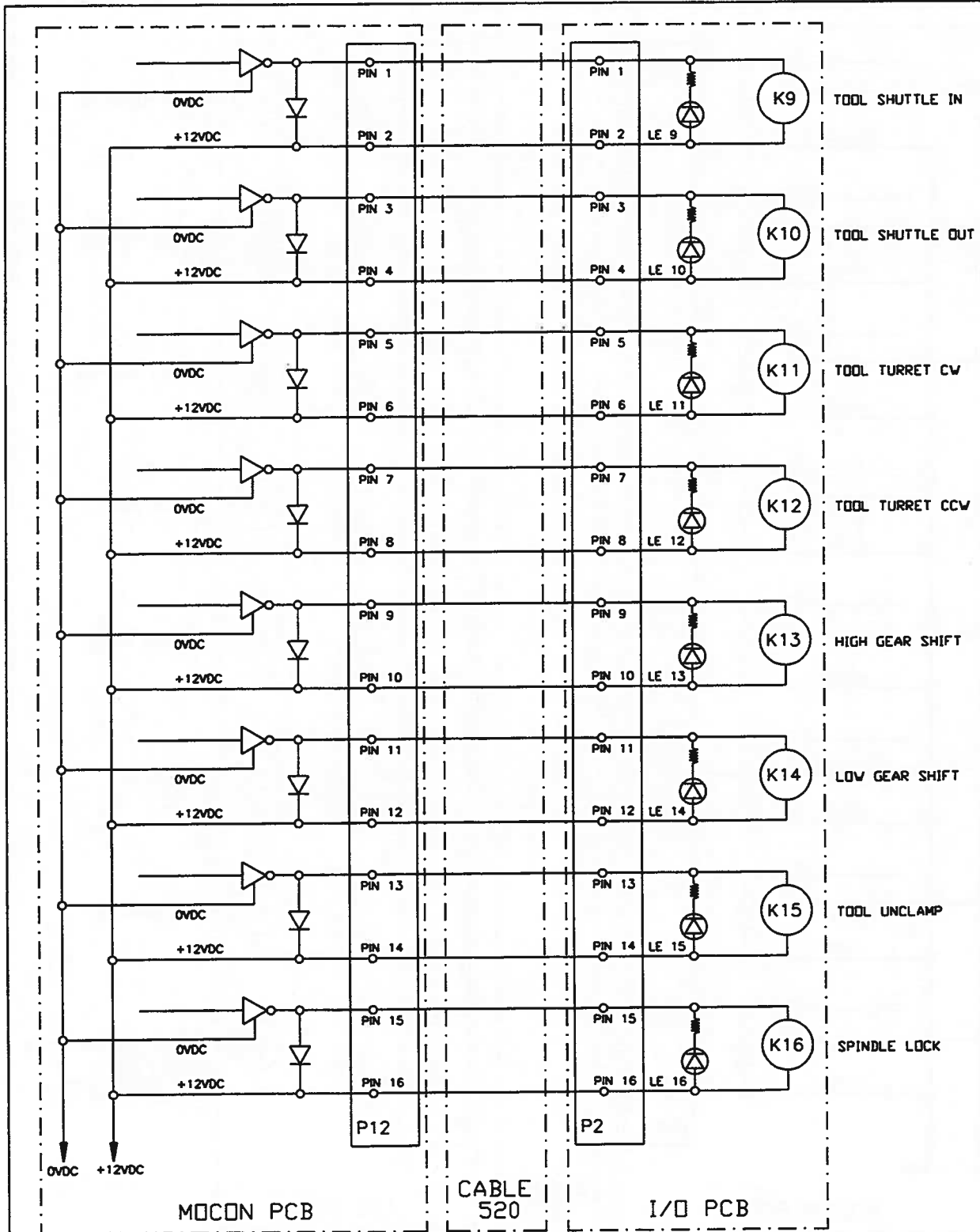
PAGE 6



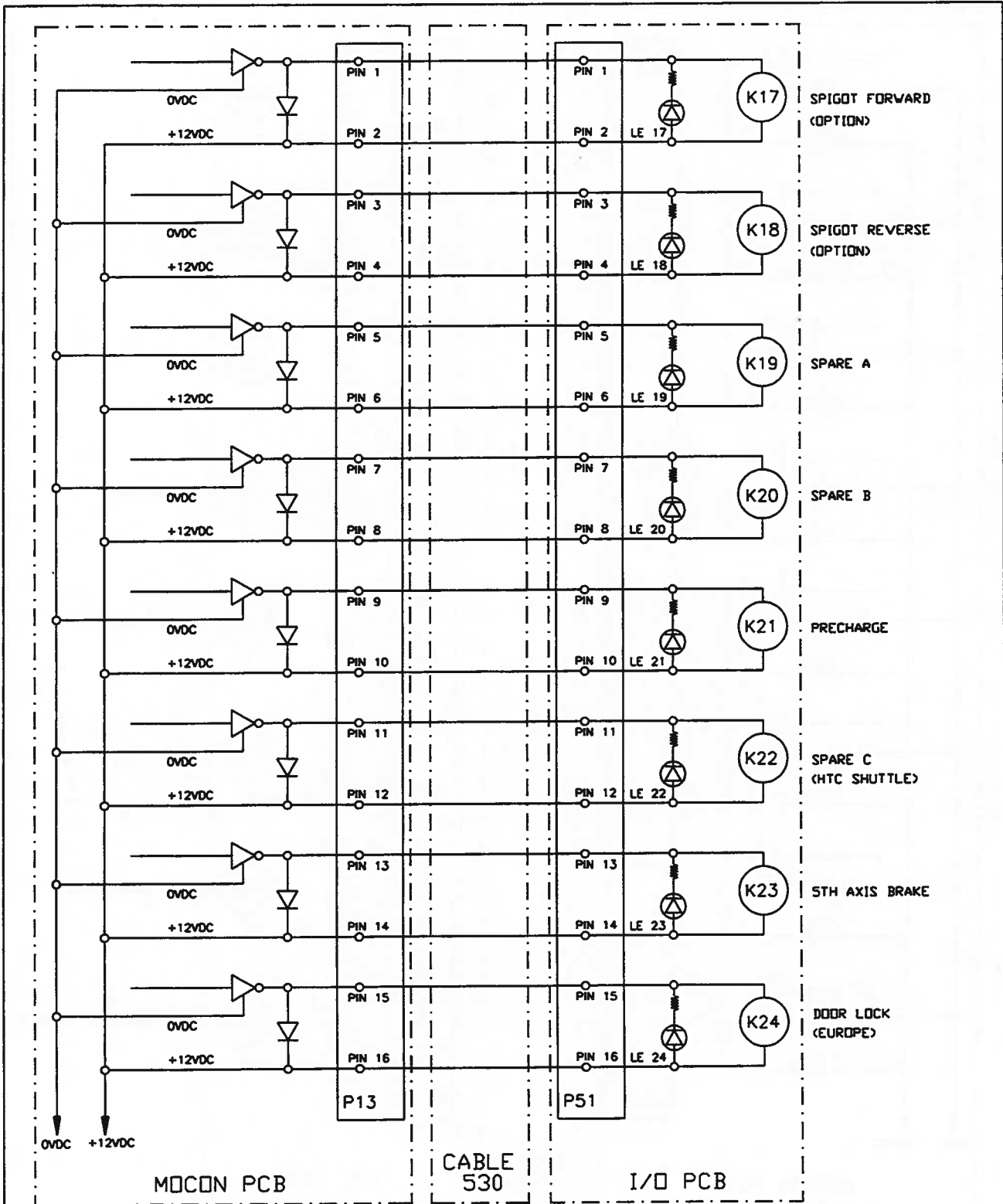
DISCRETE INPUTS 17 THROUGH 32



RELAY COIL DRIVERS, K1 THROUGH K8
HAAS AUTOMATION VF SERIES



RELAY COIL DRIVERS, K9 THROUGH K16
HAAS AUTOMATION VF SERIES

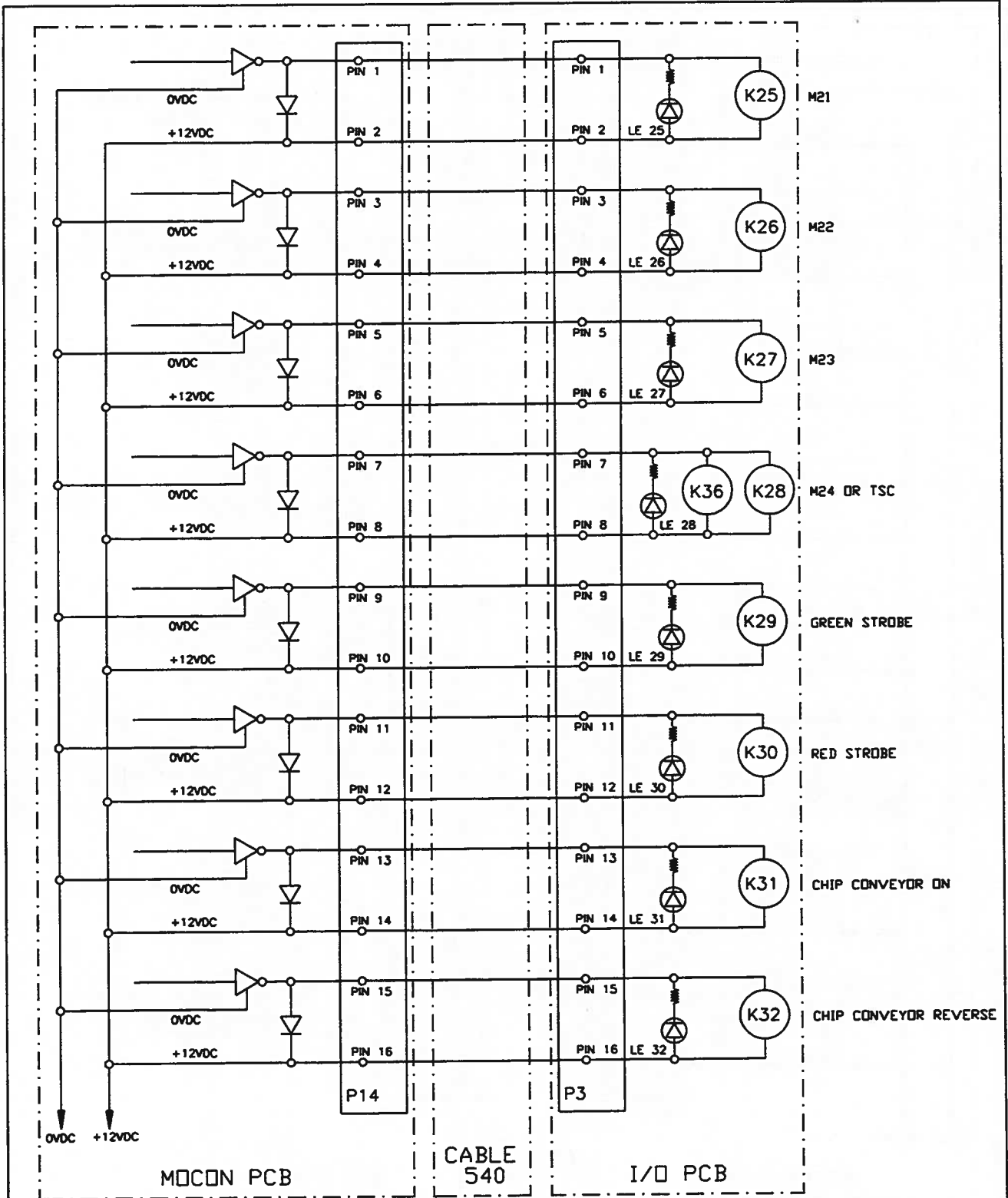


RELAY COIL DRIVERS, K17 THROUGH K24

4/96

HAAS AUTOMATION

VF SERIES PAGE 10

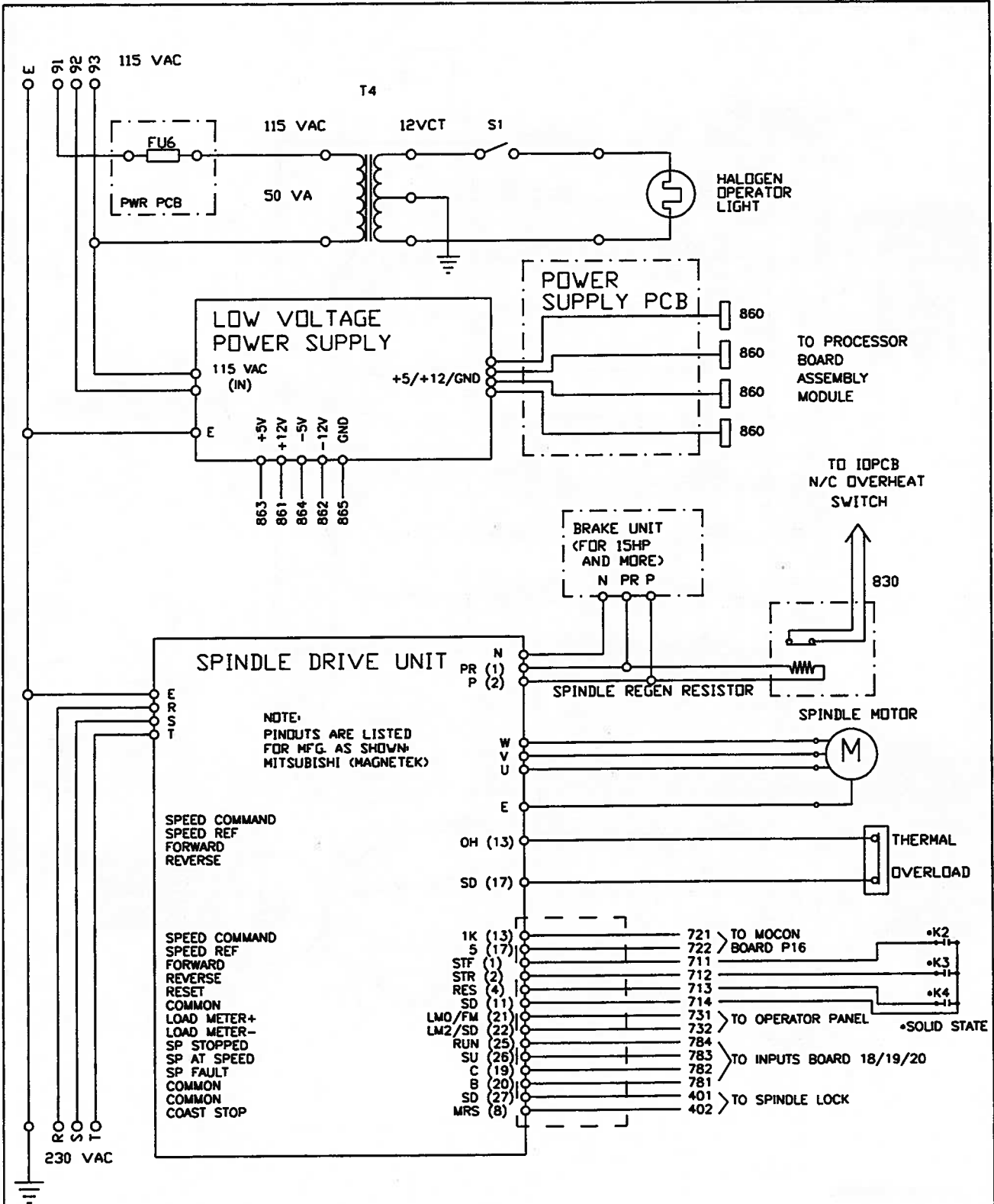


RELAY COIL DRIVERS, K25 THROUGH K32

4/96

HAAS AUTOMATION

VF SERIES PAGE 11



SPINDLE DRIVE UNIT

4/96

HAAS AUTOMATION

VF SERIES

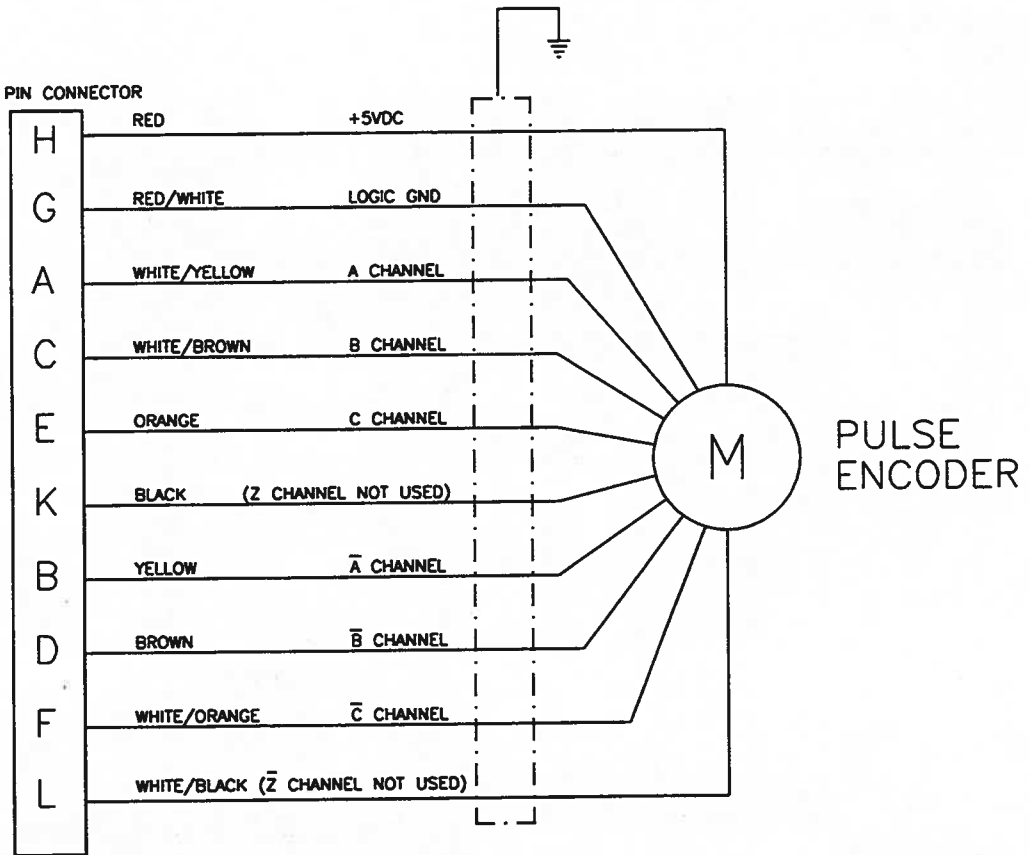
PAGE 12

17 PIN CONNECTOR

TO MOCON PCB
THROUGH CABLES:

32-1425A TO
32-1430A

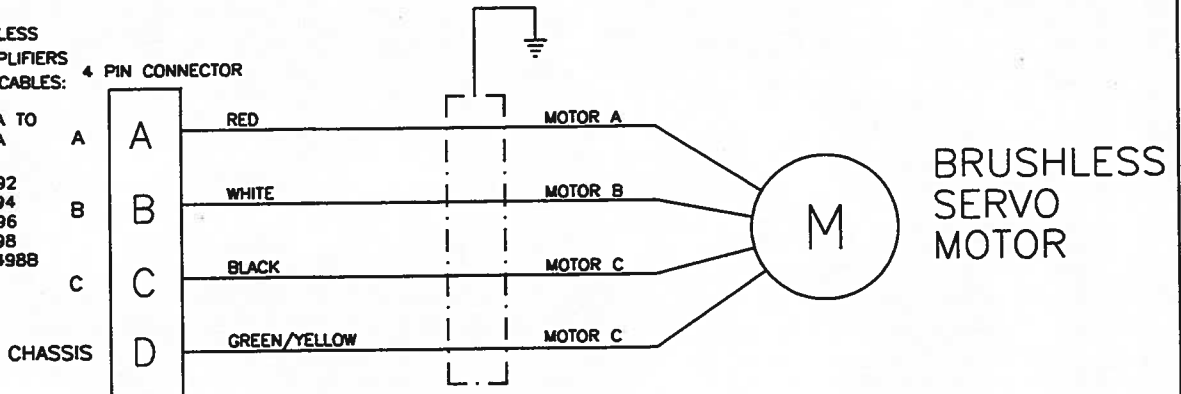
- X AXIS 660
- Y AXIS 670
- Z AXIS 680
- A AXIS 690
- B AXIS 690B



TO BRUSHLESS
SERVO AMPLIFIERS
THROUGH CABLES:

32-1425A TO
32-1430A

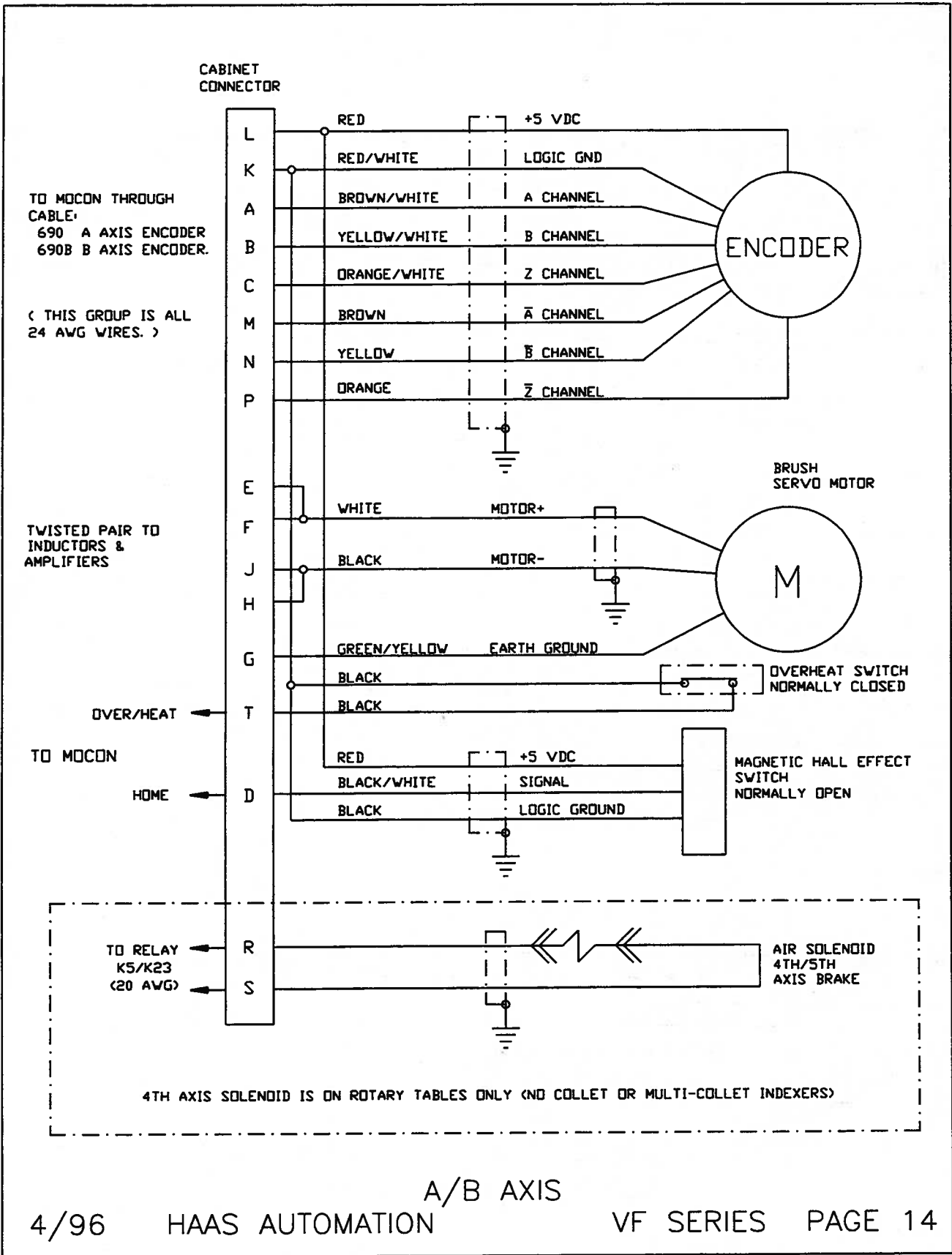
- X=491/492
- Y=493/494
- Z=495/496
- A=497/498
- B=497B/498B

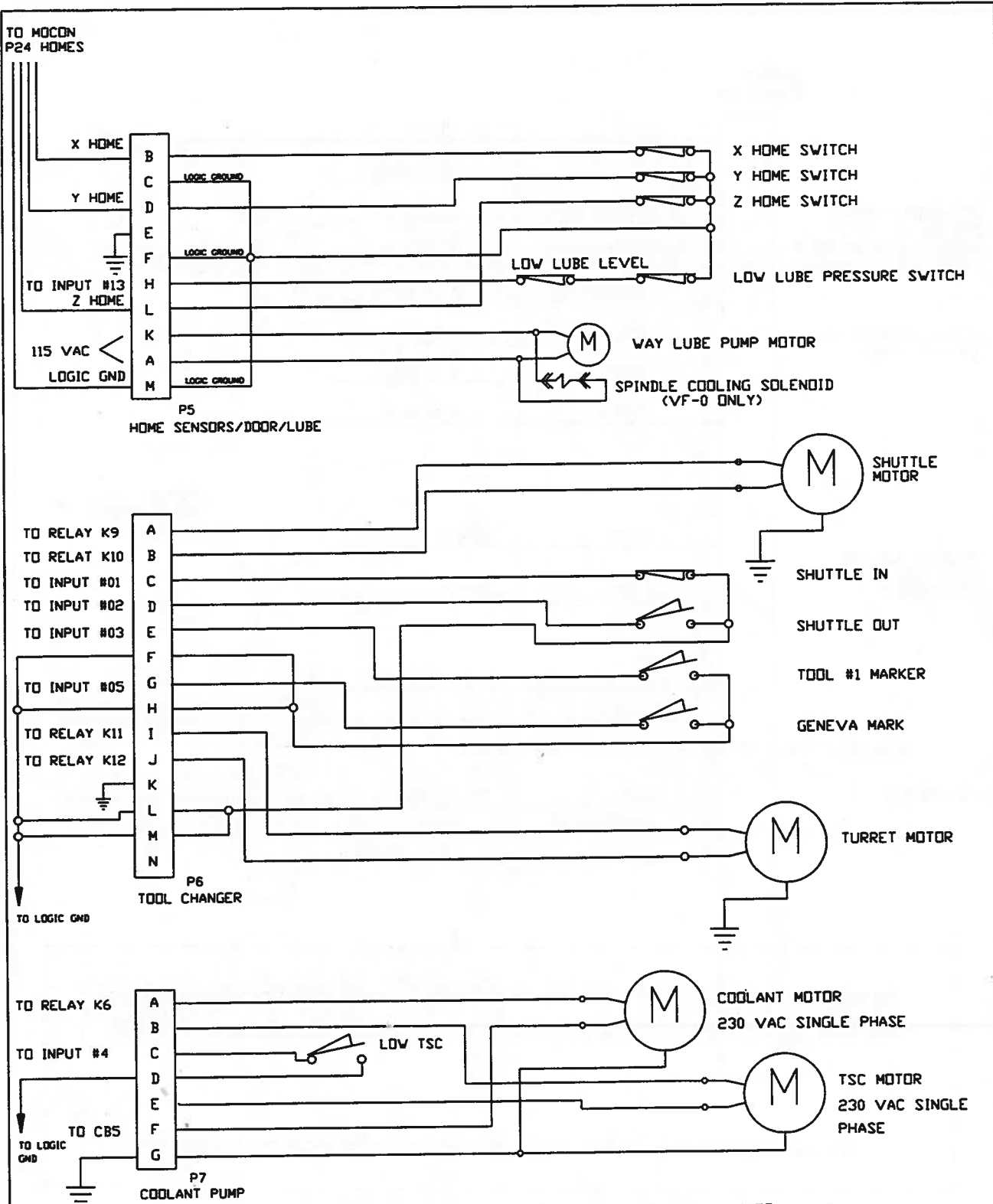


•B AXIS OPTIONAL

X/Y/Z AXIS MOTOR & ENCODER
HAAS AUTOMATION VF SERIES PAGE 13

4/96





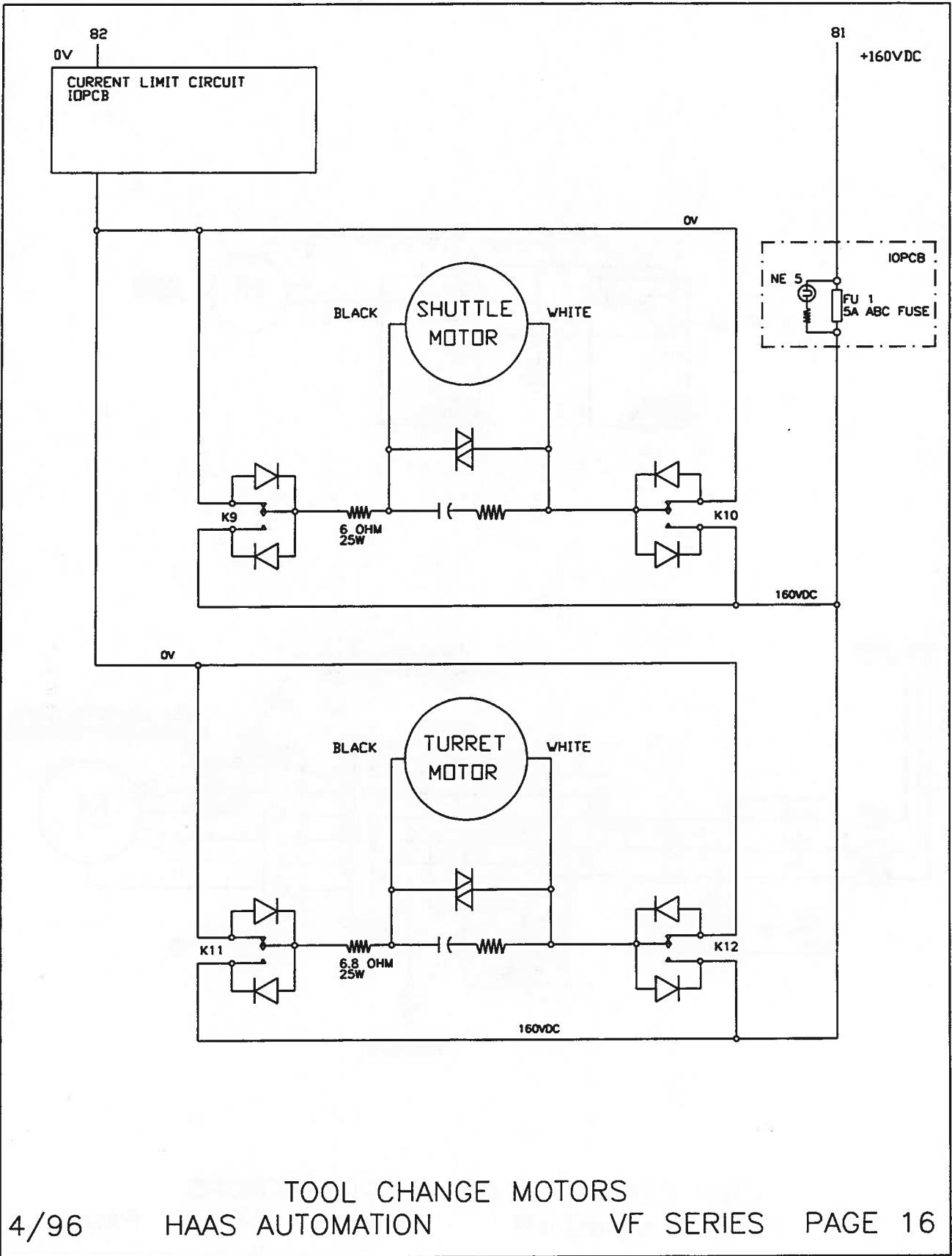
NOTE: CONNECTORS ARE LOCATED ON SIDE OF CONTROL CABINET.

CABINET CONNECTORS

4/96

HAAS AUTOMATION

VF SERIES PAGE 15



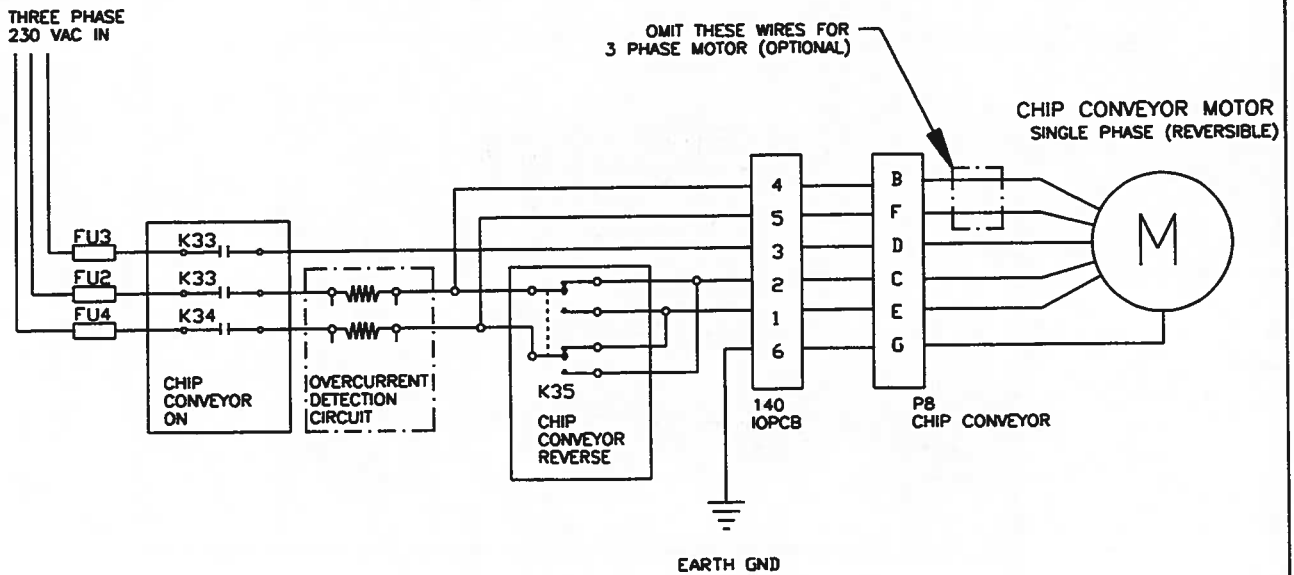
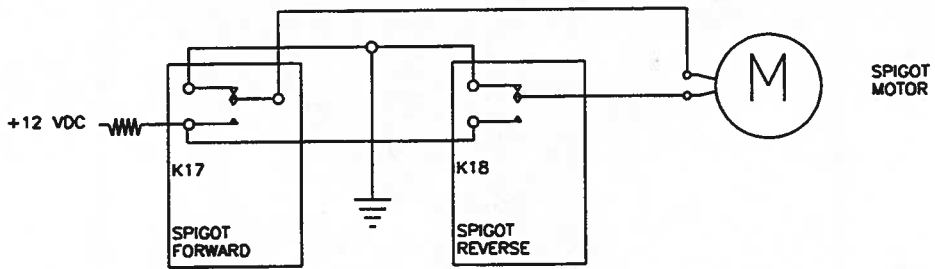
TOOL CHANGE MOTORS

4/96

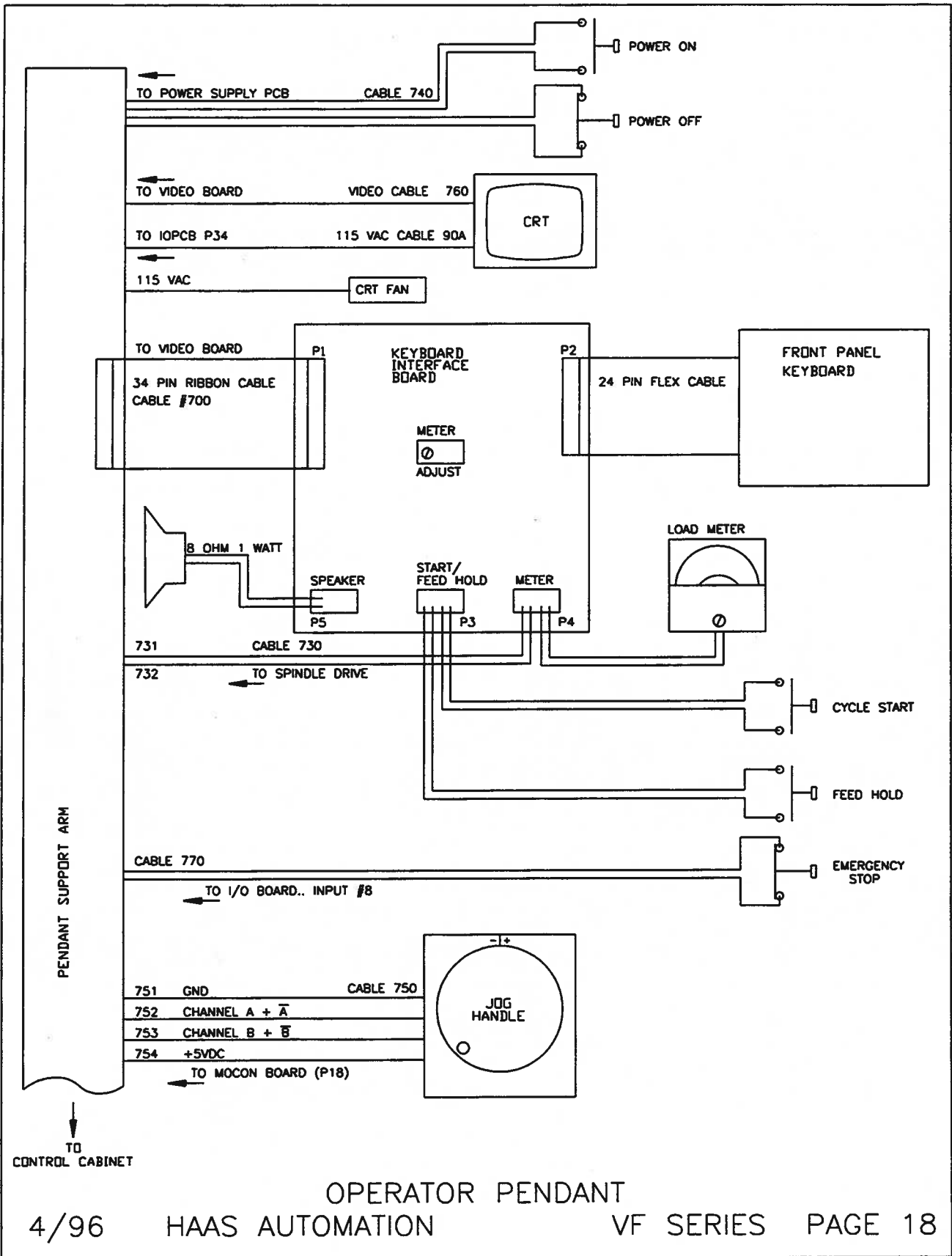
HAAS AUTOMATION

VF SERIES

PAGE 16



4/96 CHIP CONVEYOR AND SPIGOT MOTORS HAAS AUTOMATION VF SERIES PAGE 17

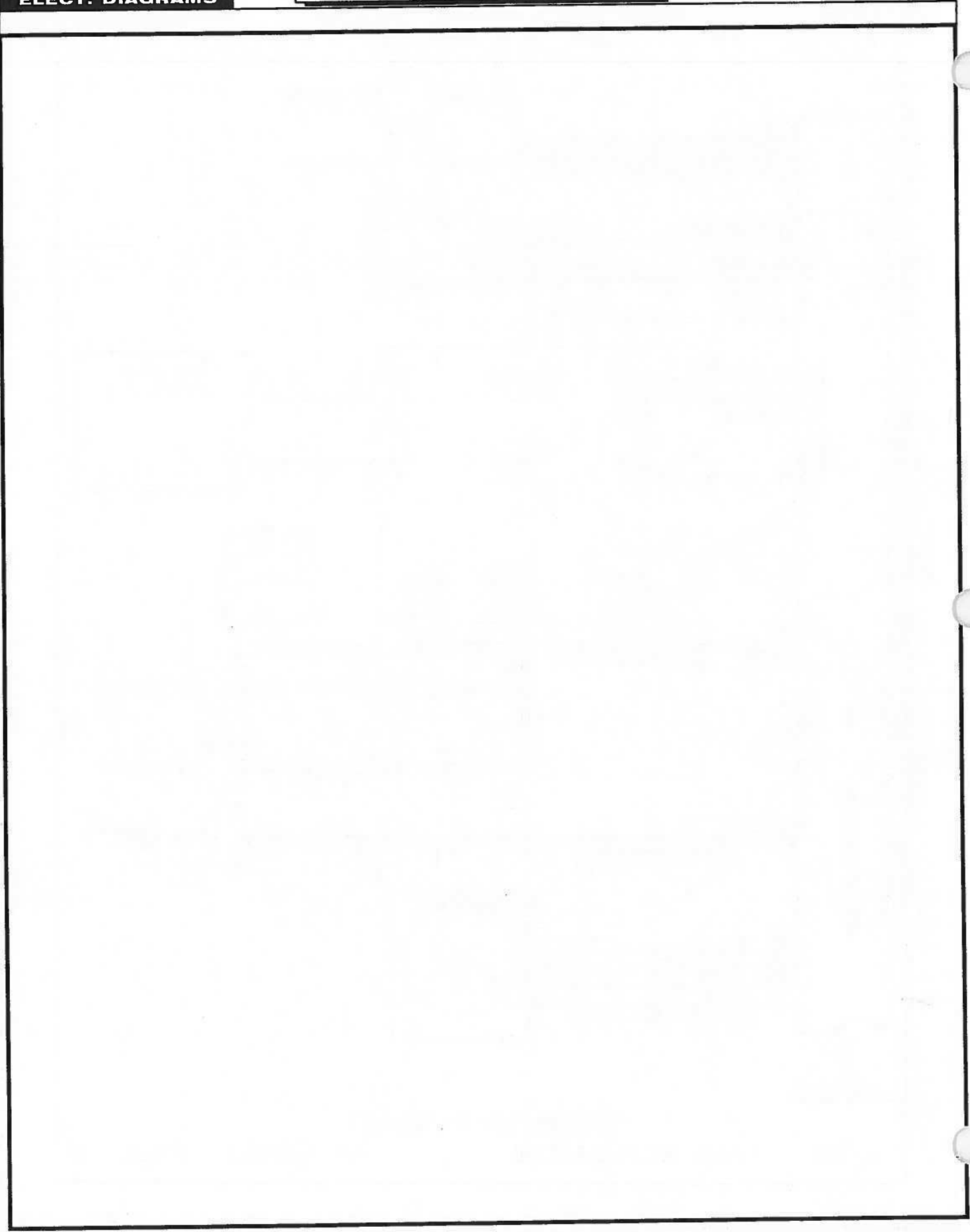


VF-SERIES



OPERATORS MANUAL

ELECT. DIAGRAMS



Index

() 16
(DOWN ARROW) 16, 52
(LEFT ARROW) 16
(MOCON) 292
(RIGHT ARROW) 16
(UP ARROW) 16
/ 16
1-Bit Discrete Inputs 144
160 Volt DC Power Supply 280

A

abnormal stops 23
Absolute and Incremental Positioning 4
Acceleration
 Exponential 232
 In Circular Moves 233
 In Feed Motions 232
Address Constant Substitution 148
Air Solenoid
 Gear Box 288
 Spindle Orientation 278
 Tool Clamp/Unclamp 277
Alarm List 177
ALARM / MESSAGES 163
Alarm condition 23
ALARM MESSAGES 15, 165, 169, 170
Alpha Keys 16
Alphabetic Addressing 141
Alphabetical Address Codes 36
ALTER 17, 24, 25, 27
Alternate Alphabetic Addressing 141
ANALOG DATA 305
Assignment Statements 152
ATC FWD 18, 20, 162
ATC REV 18, 20, 162
AUTO ALL AXES 18, 19, 87, 177, 179
Automatic Acceleration/Deceleration 232
 Acceleration in Rapid Moves 233
 Constant Acceleration 232
Automatic Tool Measurement (G35, G37) 49
Automatic Work Offset Measurement (G36, G136) 50
Autoreversing Tapping Heads 237
AUX CLNT 14
Auxiliary Axis Control 225

B

Background Edit **12, 26, 166**
Basic Programming
 Coordinate System **2**
Battery, Memory Retention **291**
BLOCK DELETE **17**
Block Lookahead Limit (G103) **76**
Block Operations **27**

C

CANCEL **19**
Canned Cycles **9, 60**
 Modifying **94**
 Special **95**
CCW **15**
Chip Conveyor **239**
CHIP FWD **14**
CHIP REV **14**
CHIP STOP **14**
Circular Interpolation Help **174**
Circular Plane Selection **47**
Circular Pocket Milling **46, 47, 101**
CLNT DOWN **14**
CLNT UP **14**
Communication With External Devices - DPRNT[] **156**
Complete Crating for the VMC **264**
CONTROL PANEL **214, 289**
Control Statements **152**
Control Status **172**
COOLANT SPGT **167**
Coolant Spigot
 Positioning **167**
COOLNT **18**
Coordinate System **2**
Creating Programs **24**
CRT **12**
CRT Displays **171**
CURNT COMDS **11, 15, 22, 163, 165, 166, 188**
Cursor Keys **15, 22, 25, 165, 169, 170**
Cutter Compensation **51**
 Entry and Exit **121**
 Feed Adjustments **121**
 General Description **118**
CW **14**
CYCLE START **12**

D

Data Input / Output **28**
 RS-232 **29, 33**
DC CODES **30, 216**
Deceleration
 In Circular Moves **233**

DEFINING A GROUP IN THE GROUP LIST (*) 135
DEFINING AN ITEM BELONGING TO A GROUP (**) 136
DELETE 17, 25, 27, 170
DELETING PROGRAMS 26
Diagnostic Data 303
 Analog Data 305
 Inputs2 305
Diagnostic data display 171
Direct Numerical Control (DNC) 31
DISCRETE INPUTS 304
DISCRETE OUTPUTS 304
DISPLAY 11
Display Title Area 171
Displays 163
 CRT 171
 Current Command 168
 Help Function 173
 Parameter 170
 Position 166
 Tool Offsets 166
Displays Keys 15
Distance To Go 166
DNC 11, 31, 33
Door Open Sense Switch 300
DRIPMODE 32
DRY RUN 17, 163, 171, 216
Dry Run Operation 162

E

EDIT 10, 12, 17, 20, 21, 22, 26, 27, 165, 175
EDIT WINDOW 125, 126
Editing 158, 214
Editing Programs 25
 Changing to a different program 25
ELECTRICAL WIRING DIAGRAMS 325
EMERGENCY STOP 11
Emergency Stop 23
Emergency Stop Switch 289
Encoder
 JOG Handle 289
 Servo Motor 279
END 16, 25
Engraving (G47) 51
EOB 16
ERASE PROG 19, 20

F

F1 12, 13, 27
F2 12, 27
F3 12, 27
F4 12, 26, 27, 166, 172
Fanuc Compatibility 233
Feed Adjustments In Cutter Compensations 121

FEED HOLD **12**
FEED RATE **14, 21, 175**
Feed/Rapid/Spindle Overrides **175**
Fifth Axis **231**
Fifth Axis Control **226**
Floating Tap Holders **236**
Floppy - Direct Numerical Control (DNC) **33**
Formulas **117**
Fourth Axis Programming **115**
FUNCTION KEYS **13**
Functions **149**
Fuses **298**

G

G Codes **41, 58**
G81 Drilling Canned Cycle **89**
G82, G83, G84 Canned Cycles **90**
Gear Box Sense Switches **288**
Gear Change Sequence **288**
General Purpose Pocket Milling
 Square Island **106**
 Square Pocket **105**
General Purpose Pocket Milling Function **77**
GENERAL WARNINGS **247**
Global Variables **143**
Graphic Display Function
 Control Status **172**
 Display Title Area **171**
 Key Help Area **171**
 Locator Window **172**
 Position Window **172**
 Tool Path Window **172**
 Z Axis Window **172**
GRAPHICS **214**
Graphics Simulation Mode **171**
GROUP **125**
GROUP CODE **135**
GROUP HELP **135**
GROUP WINDOW **125, 126**

H

HANDLE JOG **10, 18, 21**
HANDLE/JOE **21**
Helical Motion **45**
Help / Calculator Displays **165**
HELP WINDOW **125, 126**
HELP/CALC **11, 15, 24, 163, 165, 236**
High Speed Machining **234**
HOME **15, 19, 25**
HOME G28 **18, 19**
HYDRAULIC COUNTERBALANCE **303**

I

I And J
 Programming Using **44, 101**
Input/Output Assembly **285**
INPUTS2 **305**
INSERT **12, 17, 20, 25, 27**
INSTALLING THE VMC **247**
Integer Argument Passing **141**
Interpolation Commands **43**
ITEM **125**
ITEM CODE **137**
ITEM HELP **136**

J

Jog Handle **11, 21, 25**
Jog Keys **13**
JOG LOCK **14, 21**

K

Key Help Area **171**
Keyboard **13**
Keyboard Beeper **290**
Keypad **10, 11**

L

Lamp On/Off Switch **300**
Last Block (MODAL) Group Codes **146**
Lead Screw Compensation **211**
LEVELING **257**
LIST PROG **10, 12, 21, 25, 26, 27, 29, 30, 33, 85, 183, 190**
Load meter **11**
Locator Window **172**
Low Lubrication and Low Pressure Sense Switches **300**
Lubrication
 Spindle Orientation **278**
 Tool Changer **276**
LUBRICATION CHART **243**

M

M Code Detailed Description **80**
M Code Interface **299**
M Function Relays **299**
M Functions, Turning On And Off **299**
M-FIN Discrete Input **299**
Machine Defaults **7**
Machine Display **166**
MACHINE OPERATION **161**
Machine Start Up **254**
Macro Constants **142**
Macro Arguments **141**
Macro Statements **149**

Assignment Statements 152
Control Statements 152
Expressions 151
Operators 150
Macro Subroutine Call (G65) 139
Macro Variables 142
MACRO VARIABLES DISPLAY 169
Macros 138
 Aliasing 140
 Fanuc-Style Macro Features Not Included In Haas CNC 159
 Runtime Execution 157
Main Circuit Breaker CB1 295
Main Contactor K1 295
MAINTENANCE SCHEDULE 243
Manual Operation
 Handle/Jog 20
MDI 10, 11, 12, 20, 21, 25, 26, 27, 31, 33, 138, 162, 165, 171, 219
MEM 10, 11, 12, 17, 20, 21, 22, 25, 26, 162, 165, 166, 171
Microprocessor Assembly 290
Milling/Tapping Help 174
Mirror function 161
Mirror Image
 Programmable 108
Miscellaneous G Codes 45, 58
MODE KEYS 17
Modifying Canned Cycles 94
Motor Controller (MOCON) Brushless 292

N

NEXT TOOL 13, 162
NUMERIC KEYS 19

O

Offsets 147
OFSET 11, 15, 27, 162, 163, 165, 166
Onnnn address code 36
OPERATION
 Displays 175
 Setting Up a Fourth Axis 224
 Travel Limits 175
Operation Notes 158
Operation timers Display 169
Operator Display 166
Operator Reset 23
OPERATORS CONTROL PANEL 10
Operator's Lamp
 Transformer 297
OPT STOP 17
ORIENT SPINDLE 18
ORIGIN 18, 19, 169
OVERRIDES 214
Overrides Keys 14

P

- PAGE DOWN **16, 22, 25, 166, 169, 172**
- PAGE UP **16, 25, 161, 165**
- PARAM / DGNOS **15, 163, 165, 170**
- Parameter Display **170**
- Parameters **195**
 - Lead Screw Compensation **211**
- Parameters / Diagnostics Displays **165**
- Part Program Storage and Edit **24**
- PART ZERO SET **13, 161, 162, 167**
- PCB's
 - Motor Interface (MOTIF) **292**
 - Power (POWER) **296**
 - Servo Distribution (SDIST) **281**
 - Servo Drive (DRIVER) **281**
 - Video and Keyboard (VIDEO2) **291**
- POSIT **11, 15, 163**
- Position Display **165**
- Position Mark, Geneva Wheel **302**
- Position Window **172**
- Power Off **20**
- Power On/Off Switches **289**
- Power Supply Assembly **295**
 - Low Voltage Power Supply **295**
- Power Transformer Assembly (T1) **297**
 - Primary Connection To T1 **297**
 - Secondary Connection To T1 **297**
- POWER UP / RESTART **22, 87**
- Power-off **23**
- PREPARATION FOR SHIPPING **260**
- Preparatory Functions (G Codes) **41**
 - Canned Cycle Auxiliary Functions **74**
 - General Purpose Pocket Milling Function **77**
 - Reference Point Definition and Return **49**
 - Tool Length Compensation **51**
 - Work Coordinate System Selection **58**
- PRGRM / CONVRs **11, 12, 15, 24, 26, 31, 34, 163, 166**
- Primary Connection To T1 **297**
- Program Restart **22**
- Program Selection **22**
- Program Displays **165**
- Program Format **7**
- Program Review **12, 166**
- Program Structure **35**
- Programmable Output to RS-232 (G102) **75**
- Programmable Messages **145**
- Programmable Mirror Image (G100, G101) **75, 108**
- Programmable Offset Setting **45**
- Programmable Stop **146**
- PROGRAMMING
 - Cutter Compensation Description **118**

Inch / Metric Selection (G20, G21) 48
Spindle Speed Functions 88
Tool Functions (Tn) 86
Programming Examples 89
Circular Interpolation and Cutter Compensation 97
Formulas 117
Fourth Axis Programming 115
Modifying Canned Cycles 94
Programmable Mirror Image 108
Single-Point Thread Milling 114
Special Canned Cycles 95
Subprograms and Canned Cycles 91

Q

QUICKCODE 124
QUICKCODE TERMINOLOGY 125

R

Rapid Position Commands 43
RECV RS-232 18, 29, 33
Reference Point Definition and Return 49
Remote Tool Unclamp Switch 277
RESET 12, 13, 21, 23, 31, 34, 82, 87, 156, 169, 177, 180, 188, 217, 225
RESET Keys 13
Resistor 294
Resistor Assembly
 Servo Drive Regen Resistor 294
 Spindle Drive Regen Resistor 294
Rigid Tapping 236
Rotation (G68) 55
ROTATION WITH CUTTER COMPENSATION 58
ROTATION WITH SCALING 57
RS-232 28, 29, 33
RS-232 PORTS 213
Runtime Execution 157

S

Scaling (G51) 54
Secondary Circuit Breakers 296
Secondary Connection To T1 297
SELECT PROG 18, 22, 24
SEND RS-232 18, 29, 30, 33
Sense Switch
 Low Lubrication and Low Pressure 300
 Overheat 294
 Tool Clamp/Unclamp 277
Servo Amplifiers (Brushless) 283
Servo Characteristics (Brush) 279
Servo Characteristics (Brushless) 283
Servo Cooling Fan 280
Servo Drive Assembly 280
Servo Drive Regen Resistor 294

Servo Encoders (Brushless) 283
SETNG / GRAPH 15, 163, 165, 166, 171
Setting / Graphic Display Function 171
Setting Up a Fifth Axis 231
Setting Up a Fourth Axis 224
Settings 212
Settings / Graphics Displays 165
SHIFT 11, 16
SHIPPING THE MACHINE WITHOUT A PALLET 265
Shuttle In/Out Motor 276
SINGLE BLOCK 12, 17, 23, 175, 221
Single-Point Thread Milling 114
SPACE 19
Spare User M Code Interface 299
 M-FIN Discrete Input 299
 Turning M Functions On And Off 299
SPECIAL KEYS 127
Special Canned Cycles 95
SPECIAL FUNCTION KEYS 27
Spigot, Coolant 167
Spigot Parameters 238
SPINDLE 14, 175
Spindle Drive Assembly 294
Spindle Drive Regen Resistor 294
Spindle Load Meter 289
Spindle Operation 278
Spindle Orientation 278
 Sequence 278
Spindle Speed Functions 88
STOP 14
Subroutines 85
SWITCHES 300
System Variables In-Depth 144
System Overrides 146

T

Tapping
 Autoreversing Tapping Heads 237
 Thread Milling 237
Tapping With The VF Series CNC Mill 236
Thread Milling 111, 237
 Single-Point 114
THROUGH THE SPINDLE COOLANT 240
Timers 145
TOOL CHANGER 13
Tool Changer 275
Tool Clamp/Unclamp 277
 Air Solenoids 277
Tool Functions (Tn) 86
Tool Life display 169
Tool Load Monitor and Display 169
Tool Offsets 145
TOOL OFFSET MESUR 13, 37, 51, 162, 167, 220

Tool Path Window **172**
TOOL RELEASE **13, 83, 87, 162**
Transformer
 Operator's Lamp **297**
 Power-Up Low Voltage Control **296**
Transmission
 Air Solenoids **288**
 Lubrication **288**
Travel Limits **175**
Trigonometry Help Function **173**
Troubleshooting
 Diagnostic Data **303**
TSC Parameters **241**
Turret Rotation Motor **276**
Two-Speed Gear Transmission **288**
 Sense Switches **288**

U

UNDO key **17, 27**
UNPACKING THE VMC **249**

V

VMC TRAVELS **2**
Voltage
 Selection Taps **297**

W

Way and Screw Lubrication Pump
 Low Lubrication and Low Pressure Sense Switches **300**
Work Coordinate System **161**
Work Coordinate System Selection **58, 74, 76**
Work Display **166**
Working Lights **24**
WRITE **19, 24, 158, 167, 212**

X

X, Y, and Z Limit Switches **300**
XON/XOFF **29, 30**
X,Y Plane Obstacle Avoidance In A Canned Cycle **95**

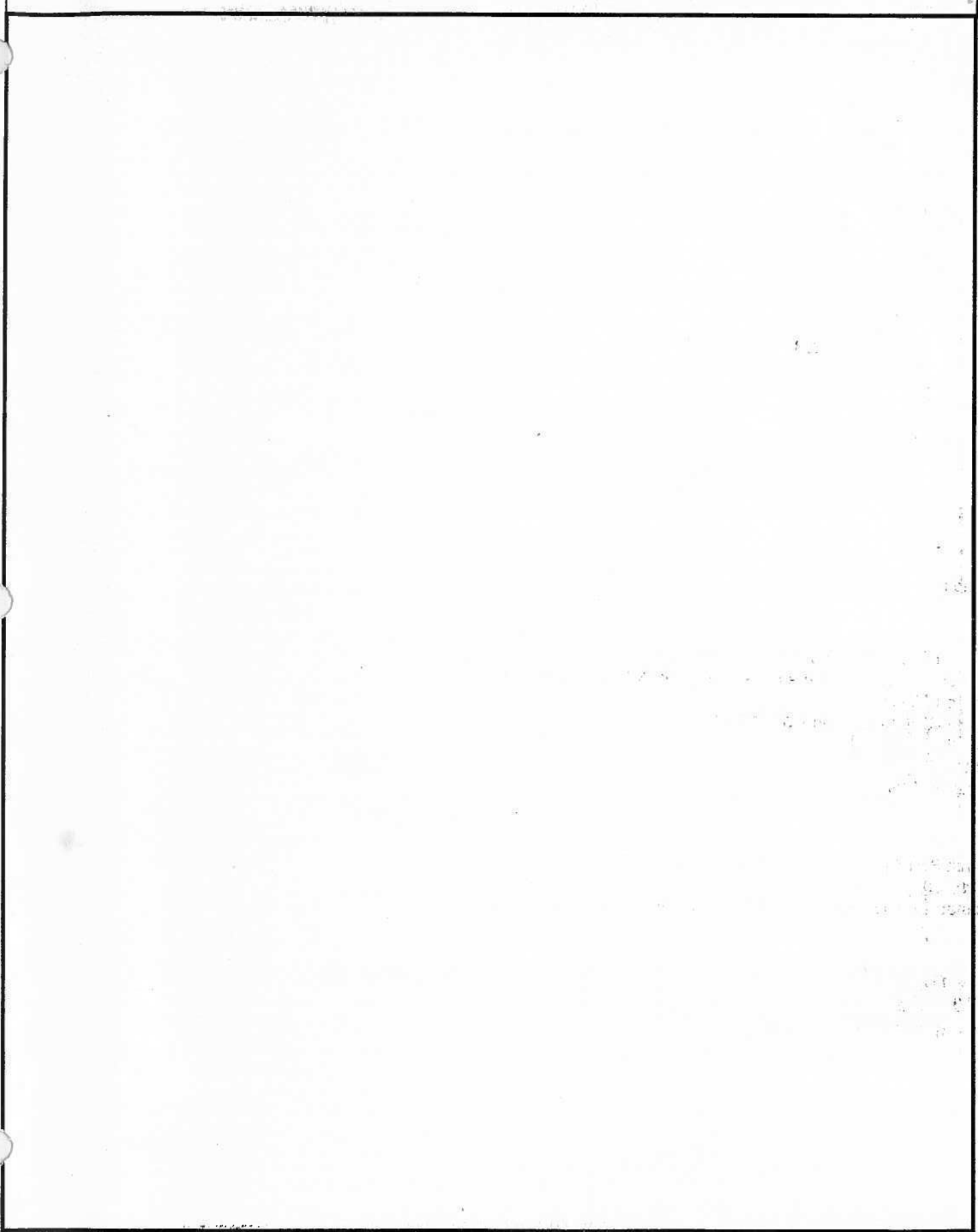
Z

Z Axis Window **172**
ZERO RET **10, 19, 21, 87, 166, 177, 187**
ZERO SINGL AXIS **18, 19**

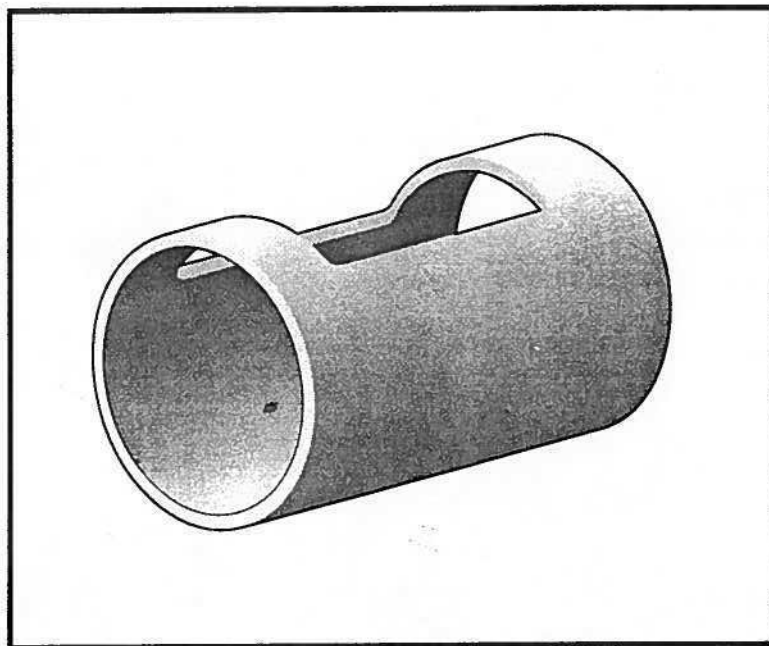


VF-SERIES OPERATORS MANUAL

INDEX



**CYLINDRICAL MAPPING (G107)
AND
INVERSE TIME MODE (G93 /94)**



CYLINDRICAL MAPPING

OVERVIEW:

This new feature translates all programmed motion occurring in a specified linear axis into the equivalent motion along the surface of a cylinder (attached to a rotary axis). It is a Group 0 G-Code, but its default operation is subject to Setting 56 (M30 RESTORE DEFAULT G).

FEATURES:

- * Any linear axis can be cylindrically mapped to any rotary axis (Only one at a time).
- * Existing linear-axis G-Code programs can be cylindrically mapped without modification; all that is required is the prior execution of a single G107 command, which is either placed at the beginning of the G-Code program itself or, if Setting 56 is set to OFF, can even be specified in a previous G-Code program, provided a RESET has not been issued in the interim.
- * The radius (or diameter) of the cylindrical surface can be redefined in a G-Code program, allowing cylindrical mapping to occur along surfaces of different diameters without the operator having to change the program.
- * The radius (or diameter) of the cylindrical surface can either be synchronized with, or be independent of, the rotary axis diameter(s) specified in the settings page.
- * G107 is used to either activate or deactivate cylindrical mapping.
- * G107 can also be used to set the default diameter of a cylindrical surface, independently of any cylindrical mapping that may be in effect.

PROGRAMMING AND OPERATION ADDENDUM

G107 DESCRIPTION

1) Three optional parameters (Address codes) can accompany G107:

X, Y, or Z - Either an X, Y, or Z address parameter can be specified. Its presence denotes that cylindrical mapping is to be initiated, and specifies the linear axis that will be mapped to the specified rotary axis (A or B). Any associated address value for this parameter will be ignored.

If one of these linear axes are specified, a rotary axis must also be specified, or an alarm will be generated.

A or B - Either an A or a B address parameter can be specified, and must be specified if any of the other parameters are specified. It serves to identify which rotary axis is being used to house the cylindrical surface; any associated address value for this parameter will be ignored.

Q - Diameter of the cylindrical surface. If this parameter is specified, a rotary axis must also be specified, or an alarm will be generated at run time. If neither this parameter or the R parameter are specified, then the diameter assumed for the cylindrical surface will be whatever was last specified for this rotary axis in a previously issued G107 since the machine was powered up.

If no G107 was issued since the machine was powered up, or if the last value specified was zero, then the diameter will be whatever is currently defined in the CNC Settings for this rotary axis ("4th Axis Diameter", etc.)

The value specified for this parameter will become the new G107 default for the specified rotary axis until the machine is turned off.

If this parameter was not specified in a previous G107 since the machine was powered up, or if its last specified value was zero, or if its last specified value was numerically equal to that defined for the rotary axis in the CNC Settings page, then any change made to the rotary axis diameter in the CNC Settings page will also change the default value for this parameter. In such a situation, the default G107 diameter is said to be "in sync" with the CNC Settings.

If the last non-zero G107 diameter value specified since the machine was powered up was not numerically equal to the Settings page value however, then the G107 diameter is said to be "out of sync" with the Settings page, and the default value thus established will be retained regardless of any changes made to the Settings page, until or unless they become equal in value.

This means that the rotary axis diameter as specified in the CNC Settings page is always used as the default cylindrical surface diameter, unless a previously issued G107 has established a different value. This allows the default cylindrical surface diameter to either be synchronized with, or operate independently from, the rotary axis diameter defined in the CNC Settings.

To force the default cylindrical surface diameter to be "in sync" with the Settings page Rotary Axis Diameter, which may be desirable if the current default cylindrical surface diameter status is unknown, the following G107 command can be used:

G107 A0 Q0

This command will reset the A-axis default cylindrical surface diameter to zero, which forces the Settings page value to be used as the default for that axis.

R - This parameter specifies the radius of the rotary axis. If Q has already been specified, the R will be ignored. Except as noted above, the conditions and/or restrictions associated with the use of this parameter are exactly the same as the Q parameter.

2) The above parameters can be entered in any order.

PROGRAMMING AND OPERATION ADDENDUM

- 3) Pressing the RESET key will always turn off any Cylindrical Mapping that is currently in effect, regardless of the status of Setting 56. Cylindrical Mapping will also be turned off automatically whenever the G-Code program ends, but only if Setting 56 is set to ON. If Setting 56 is set to OFF, any Cylindrical Mapping that is in effect when the program ends will remain in effect.
- 4) A G107 code issued without any parameters will turn off any cylindrical mapping currently in effect. This should normally be done before a program ends. If Setting 56 is set to OFF however, it may be desirable not to turn Cylindrical Mapping off, so that the mapping established can be used by other programs.
- 5) Re-issuing a G107 with either the same or a different linear axis specification will turn off any cylindrical mapping that is currently in effect, before the new mapping is initiated.
- 6) Issuing a G107 code with only a rotary axis specification and either a Q or an R will only change the default diameter for the specified rotary axis; it will not turn on or off any cylindrical mapping.

Note Due to the fact that a rotary axis's maximum acceleration (as parameterized in steps/seconds/second in the parameters page) is different from a linear axis's maximum acceleration, there is a maximum achievable Feed Rate that can be programmed for a Circular Interpolated Move, when one of the Circular axes has been Cylindrically Mapped. Refer to the next section for a description of FEED RATE ALTERATION.

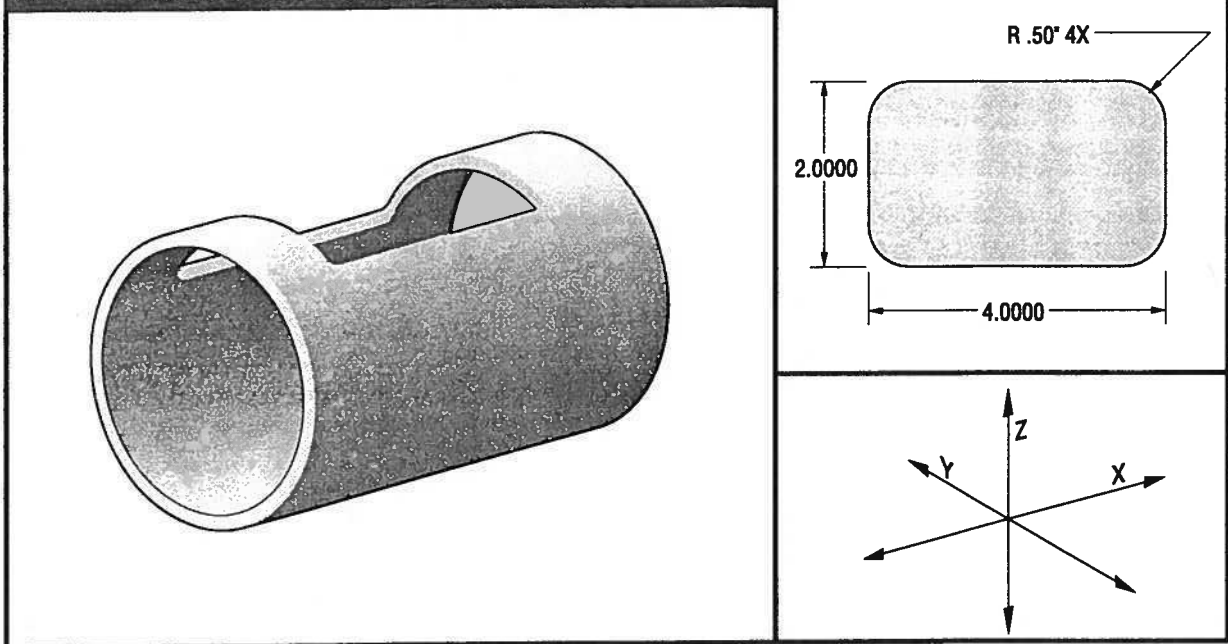
FEED RATE ALTERATION TO ENSURE CONCURRENT AXIS MOTION

The actual FeedRate achieved in any interpolated motion in the CNC Mill is limited by the maximum speed achievable in each axes of motion. The FeedRate may be lowered from its programmed value if that value would cause the Mill to command an axis to move *faster* than its parameterized maximum speed.

This means that the actual FeedRate achieved may be less than or equal to the programmed FeedRate, ensuring that all axes of interpolated motion move concurrently. When this condition occurs, the warning 'LIM' will be displayed on the screen immediately to the left of the 'FEED' display.

Note also that the **MAX FEED RATE EXCEEDED** alarm will still be issued if the programmed feed rate exceeds parameter 59 MAX FEEDRATE, (currently 300 ipm), but ONLY if the mill is not in Inverse Time mode.

G107 Cylindrical Mapping



SAMPLE PART PROGRAM

```

%
00079 (G107 TEST)
G00 G40 G49 G80 G90
G28 G91 A0
G90
G00 G54 X1.75 Y0 S5000 M03
G107 A0 Y0 R2 (IF NO R OR Q VALUE, MACHINE WILL USE VALUE IN SETTING 34)
G43 H01 Z0.25
G01 Z-0.25 F25.
G41 D01 X2. Y0.5
G03 X1.5 Y1. R0.5
G01 X-1.5
G03 X-2. Y0.5 R0.5
G01 Y-0.5
G03 X-1.5 Y-1. R0.5
G01 X1.5
G03 X2. Y-0.5 R0.5
G01 Y0.
G40 X1.75
G00 Z0.25
M09
M05
G91 G28 Z0.
G28 Y0.
G90
G107
M30
%
```

INVERSE TIME

OVERVIEW:

This new feature specifies that all 'F' code values are to be interpreted as '**strokes per minute**' (except Auxiliary Axes). This is equivalent to saying that the 'F' code value, when DIVIDED INTO 60, is the number of seconds that the motion should take to complete.

G93 activates Inverse Time Mode and a G94 deactivates it. These are implemented as Group 5 G-Codes.

FEATURES:

* Any interpolated motion that involves only the auxiliary axes is NOT affected by Inverse Time Feed Mode - the 'F' code specified will still be interpreted as Feed per Minute.

* When active, the Feed Rate specification is MANDATORY for all interpolated motion blocks. i.e.: Each non-rapid motion block MUST have its own Feed Rate specification. If it doesn't, a "**NO FEED RATE**" alarm is generated.

* Mixing auxiliary axes with regular axes in a G1/2/3 motion in G93 mode will generate the alarm

"AUX AXIS IN G93 BLOCK".

* All Group 9 motion commands, as well as any G12, G13, G70, G71, G72, & G150 commands will generate a syntax alarm when in G93 mode.

* Pressing Reset will reset the machine to G94 (Feed per Minute) mode.

* The diameter settings (4th & 5th axis) are meaningless in inverse time.

* Alarm 309 **EXCEEDED MAX FEEDRATE** will not be generated by G93 because the machine will automatically be limited by the slowest axis.

ALARMS:

Alarm 348

MOTION NOT ALLOWED IN G93 MODE

This alarm is generated if the Mill is in Inverse Time Feed Mode, and either a G12, G13, G70, G71, G72, G150, or any Group 9 motion command is issued.

Alarm 397

AUX AXIS IN G93 BLOCK

This new alarm is specific to G93/G94 and is generated if a G-Code block specifies any form of interpolated motion that involves BOTH one or more of the regular axes (X,Y,Z,A,B,etc...) AND one or more of the auxiliary axes (C,U,V,W).



ADVANCED EDITOR

This addendum describes the features of the new HAAS Advanced Editor. Information provided is in addition to what is currently in the Operators Manual.

The HAAS Advanced Editor gives the user the ability to view and edit two CNC programs at a time. This makes it easier to modify existing programs and to create new ones. The editor has an 80 column display, and includes pull-down menus that allow the user to access the features of the editor. Additionally, a context-sensitive help function is available to provide information on all of the editor's features.

The following terms are used throughout this addendum to describe the advanced editor:

CURRENT PROGRAM	The program that is expected to be run from MEM or MDI mode.
ACTIVE PROGRAM	The program that is altered by user input.
INACTIVE PROGRAM	The program opposite the active program in the editor.
CONTEXT-SENSITIVE HELP	A help function that provides information based on what the user is currently doing.
PULL-DOWN MENUS	Menus accessed, or "pulled down", via the "menu bar" at the top of the screen. Only one menu can be accessed at any one time. When a menu is pulled down, items appear that can be scrolled through and selected.
HOT KEY	A key that, when pressed, will immediately execute an editor function.

The advanced editor is entered by pressing the EDIT key. The old 40 column editor can be accessed by pressing the PRGRM/CONVRS key. Another press of PRGRM/CONVRS will place the user into the QuickCode display. The QuickCode application can also be accessed from within the F1:HELP pull-down menu. A third press of the PRGRM/CONVRS key will access the advanced editor. The user can alternate between the advanced editor, the 40 column editor, and QuickCode with successive presses of the PRGRM/CONVRS key.

All of the functions of the 40 column editor are still available in the advanced editor. However, the F1 key, which used to mark the beginning of selected text, now provides access to the pull-down menus and context-sensitive help. Pressing the F1 key activates a pull-down menu, which appears descending from the menu bar. If the pull-down menu system has not been accessed during the current editor session, the Help menu is pulled down. Otherwise the most recently accessed menu is pulled down.

Whenever the pull-down menu system is active, the current menu is pulled down and one item is highlighted. The user can then use the up and down arrow keys to scroll through the items of that menu, or use the left and right arrow keys to open other menus.

When a menu item is highlighted, the user can see a brief description of it by pressing the F1 key. This activates context-sensitive help, which describes what that menu item does. The context-sensitive help explains any prompts that may appear, what keys are available for action, and what the "hot key" (if one exists) is for that menu item.

The UNDO key is used to deactivate the pull-down menu system. Pressing RESET will also deactivate the pull-down menu, but UNDO is preferred. If UNDO is pressed after invoking an executing function from a pull-down menu, it will abort that function. A context-sensitive help session can also be ended by pressing the UNDO key. This will return the user to the active program.

The EDIT key can be used to "switch" between two programs that have been selected to edit.

Pressing the F4 key will open a copy of the current program in the inactive window. The user can quickly edit two different locations in the same program by pressing F4, moving to the second location, and then using the EDIT key to move back and forth between the two locations. If the user enters 'Onnnn' and then presses F4, program Onnnn is opened in the inactive window.

Figure 1 illustrates the layout of the advanced editor.

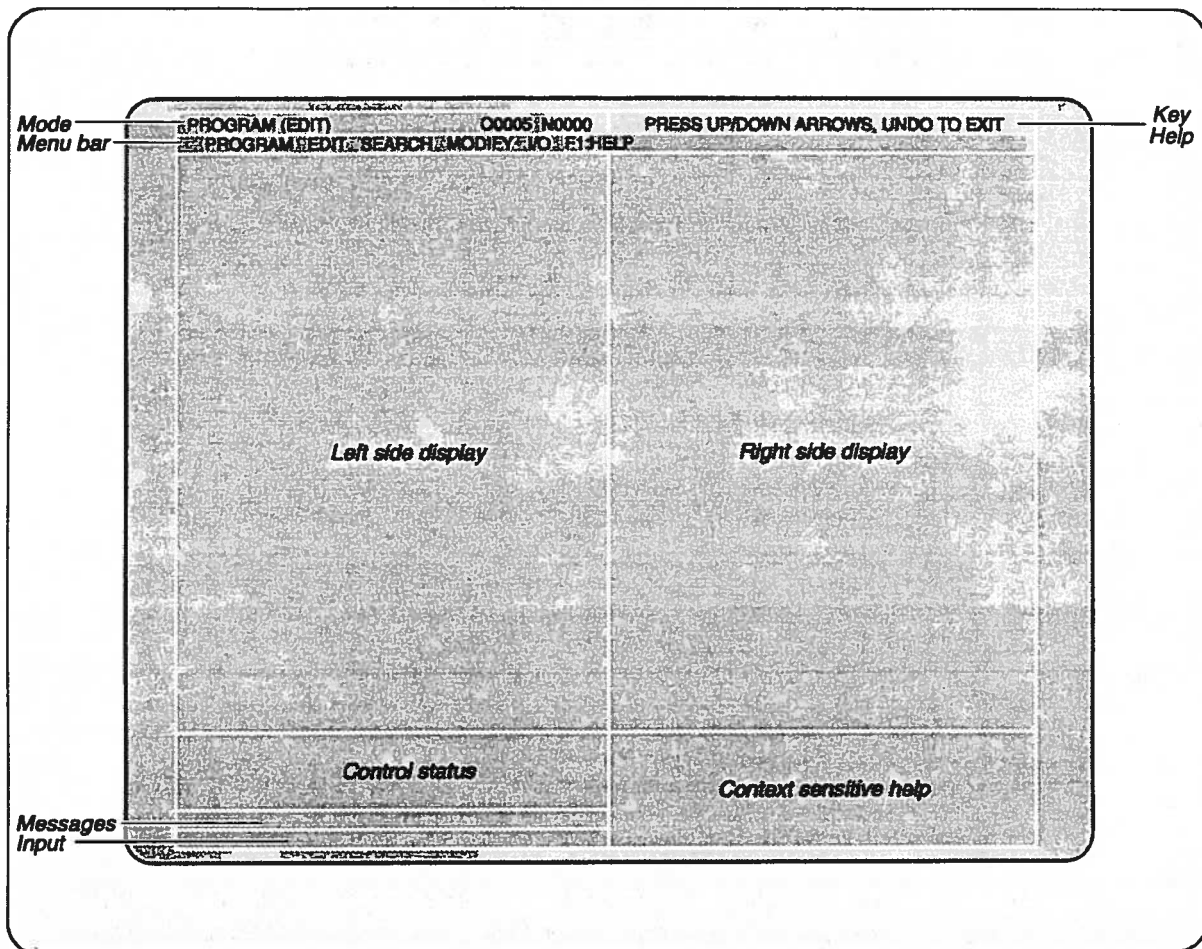


Figure 1. The advanced editor screen layout.

The advanced editor screen is divided into the following areas:

- **Mode and Control Status** - contain the Current Display page, Operating mode and Control status.
- **Messages** - where control status messages are displayed. This area is used to display prompts for user input in the advanced editor.
- **Input** - where the user's input is displayed.
- **Menu Bar** - contains the pull-down menu banner.
- **Key Help** - contains short immediate help messages. These are meant to show the user the most important keys that can be used in the current context or operation.
- **Left/Right Side Display** - shows the active and inactive programs. When first entering the editor, the current program will be displayed on the left side and the right side will be blank. These areas can also display program lists and help pages when the appropriate menu item is selected from the pull-down menu.
- **Context-Sensitive Help** - where context-sensitive help is displayed when a menu is active, or when the user is entering data in response to selecting a menu item.

FEATURES

This section briefly describes each feature of the advanced editor, each item found in the pull-down menus, and any prompts that might appear.

CONTEXT SENSITIVE HELP (The F1 key)

While in the advanced editor, press the F1 key to get into the menus, use the arrow keys to highlight the menu item, then press F1 again. The help text for that item will appear in the lower right corner of the screen. The jog handle, up and down arrow keys, and the PAGE UP and PAGE DOWN keys can then be used to view the help text. Press CANCEL or UNDO to go back to the menu, or RESET to exit the menus entirely.

THE PROGRAM MENU

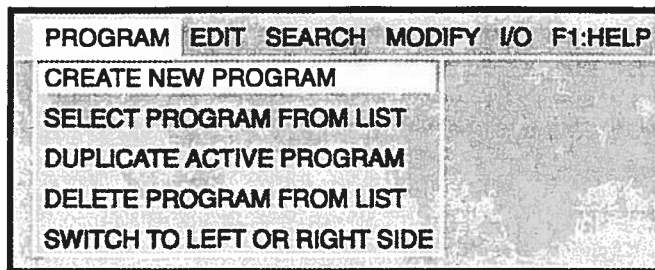


Figure 2. The PROGRAM menu items.

CREATE NEW PROGRAM

This menu item will create a new program, providing there is room in the program directory and enough memory is available. Enter a program name (Onnnn) in the range of 0 through 9999 that is not already in the program directory.

SELECT PROGRAM FROM LIST

The HAAS control maintains a directory of programs that the user can select. Select this menu item to edit a program that exists in the directory.

When this menu item is selected, a list of programs is presented for viewing. Scroll through the list by using the cursor keys or the jog handle. Pressing the WRITE key will select the program that is highlighted and will replace the selection list with the selected program. The selected program is now active, and the previously active program will appear on the inactive edit screen.

Program size and memory usage appear at the bottom of the display. SELECT PROG is the hot key for this item.

DUPLICATE ACTIVE PROGRAM

This menu item will create a new program, copy the contents of the current program into it, rename it as specified, and make it the active program.

ENTER NEW PROGRAM NUMBER: If no program code is present on the input line when this menu item is invoked, this prompt will appear. Enter a valid program number (Onnnn), then press the WRITE key. Only numeric inputs will be accepted.

DELETE PROGRAM FROM LIST

This menu item will delete a program from the program directory. A list of all programs is presented, with 'ALL' at the end.

To delete a single program, cursor to the program number and press the WRITE key. A prompt will ask for a confirmation of the deletion operation. Enter 'Y' to delete the highlighted program. If any other key is pressed, the program will not be deleted. After a program is deleted, the list of programs will again be presented.

To delete all programs, cursor to 'ALL' and press WRITE. Confirm deletion of all programs by pressing 'Y'. When all programs are deleted, program 00000 is created, and it is made the active program.

Press UNDO to exit this menu item and return to the active program. Program size and memory usage are displayed at the bottom of the screen.

ERASE PROG is the hot key for this item.

SWITCH TO LEFT OR RIGHT SIDE

This will make the active program inactive and the inactive program active. If there is no inactive program, then nothing happens. There are only two possible displays: one on the left and one on the right. The inactive display is used when a second program is selected or created.

The EDIT key is the hot key for this item.

THE EDIT MENU

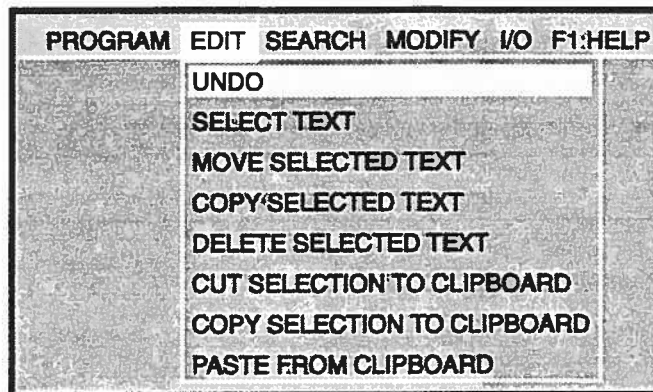


Figure 3. The EDIT menu items.

UNDO

The last insert, delete, or alter operation will be undone. Pressing UNDO again will restore the previous editing operation, up to the last 9 editing operations. If a block has been selected, choosing this item will simply exit block select mode without undoing anything. UNDO is the hot key for this item.

SELECT TEXT

This item will set the start point of the text selection. To set the end point, scroll up or down to the desired place, and press the F2 or WRITE key. The selected text will then be highlighted. To deselect the text, press UNDO. This function works the same as in the old editor, except the menu option is used to start selecting text, instead of the F1 key, and either the WRITE or F2 key can be used to end the selection. The following prompt will appear when this item is selected:

SCROLL TO END POINT, PRESS WRITE OR F2: Press the WRITE or F2 key to complete the text selection.

MOVE SELECTED TEXT

All selected text will be moved to the line following the current cursor position.

COPY SELECTED TEXT

All selected text will be copied to the line following the current cursor position.

DELETE SELECTED TEXT

This item deletes any selected text. If no text is selected, the currently highlighted item is deleted. This function works the same as in the old editor, except that if the cursor is in the middle of a comment, it will delete the entire comment instead of just the highlighted character. The UNDO key will restore any deleted comment. The DELETE key deletes individual characters from comments and is the hot key for this menu item.

CUT SELECTION TO CLIPBOARD

All selected text will be moved from the current program to a new program called the clipboard. Any previous contents of the clipboard are deleted. The program number for the clipboard is specified by Parameter 226 and can be altered, if necessary.

COPY SELECTION TO CLIPBOARD

All selected text will be copied from the current program to a new program called the clipboard. Any previous contents of the clipboard are deleted. The program number for the clipboard is specified by Parameter 226 and can be altered, if necessary.

PASTE FROM CLIPBOARD

The contents of the clipboard are copied into the current program at the line following the current cursor position.

THE SEARCH MENU

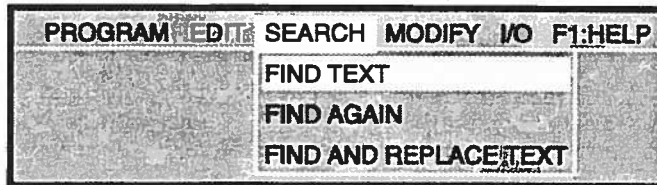


Figure 4. The SEARCH menu items.

FIND TEXT

This menu item will search for one or more G-Code items in the current program. The search can be performed in either the forward or backward direction from the current cursor location. If the item is found, the cursor will be positioned on it. The following prompts will appear when this item is selected:

ENTER TEXT/ITEM TO SEARCH FOR: Type in one or more G-code items, a single address code character, or a comment to be searched for. Press the WRITE key to enter this input. If a full G-Code item is specified, only items that exactly match will be found. If a single address code character is specified, the next matching address code will be found, regardless of the numeric value associated with it. And finally, if a comment is specified, the next comments that contain the text specified will be found.

FORWARD OR BACKWARD (F/B) ? Type in either 'F' or 'B'; all other input will be ignored. Entering 'F' will commence the search for the specified G-code item in the forward direction. Entering 'B' will commence the search in the backward direction; i.e. it will find the previous occurrence of the specified G-Code.

FIND AGAIN

This menu item will search the current program for the last text that was searched for. It will begin to search at the current cursor location, in the direction that was specified in the previous search. This function will search both selected and unselected blocks.

FIND AND REPLACE TEXT

This menu item will search the current program for one or more occurrences of a specified G-Code item and optionally replace each (or all) with another G-Code item. The search can be performed in either the forward or backward direction from the current cursor location. As each G-Code item is found, the cursor will be positioned on it, and a prompt will ask whether to replace the item, continue the search, both, or neither. This function affects both selected and unselected blocks. The following prompts will appear when this item is selected:

ENTER TEXT/ITEM TO SEARCH FOR: Enter either one or more G-code items, a single address code character, or a comment. Press the WRITE key to enter the input. If one or more G-code items are specified, only G-code items that exactly match will be found. If a single address code character is specified, all matching address codes will be found, regardless of their associated numeric value. And finally, if a comment is specified, all comments that contain the specified text will be found.

ENTER REPLACEMENT TEXT/ITEM: Enter one or more G-code items that will replace each occurrence of the G-Code item(s) found. If nothing is entered at this prompt, all occurrences of the found items will be deleted, upon verification.

FORWARD OR BACKWARD (F/B) ? Enter either an 'F' or a 'B'; all other inputs will be ignored. Entering 'F' will commence the search for the specified G-code item in the forward direction. Entering 'B' will commence the search in the backward direction.

REPLACE (YES/NO/ALL/CANCEL) ? As each G-Code item is found, the cursor will be positioned on it, and this prompt will appear. Typing in a 'Y' (for 'Yes') will replace the item, and continue the search in the specified direction. Typing an 'N' (for 'No') will not replace the item, but will continue the search in the specified direction. Typing an 'A' (for 'All') will replace all occurrences of the item with the replacement text, and end the search process. Typing a 'C' (for 'Cancel') will abort the search process.

THE MODIFY MENU

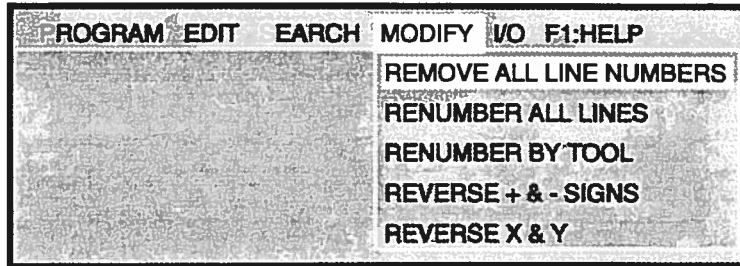


Figure 5. The MODIFY menu items.

REMOVE ALL LINE NUMBERS

This menu item will automatically remove all unreferenced N-Codes from the edited program. If a block is selected, only the G-Code blocks contained within it will be affected.

RENUMBER ALL LINES

This menu item will either renumber all G-Code blocks in the program or, if a block is selected, renumber only those G-Code blocks contained in that block. Below are the prompts that may be encountered while in this item, with a brief explanation of each:

ENTER STARTING N-CODE NUMBER: Type in the starting N-Code number, then press the WRITE key to enter the number. The maximum value accepted is 99999. Any non-numeric input will be ignored.

ENTER N-CODE INCREMENT: Type in the desired numeric difference between consecutive N-Codes, then press the WRITE key to enter the number.

RESOLVE OUTSIDE REFERENCES (Y/N) ? This prompt will appear only if a selected block was defined prior to execution of this menu item. Entering a 'Y' (for Yes) will cause G-Code items outside of the selected block that refer to N-Codes inside the block (such as a GOTO) to be changed to reference the new N-Codes correctly. If an 'N' (for No) is entered, G-Code references that exist outside the selected block will not be changed.

RENUMBER BY TOOL

Searches selected text, or the entire program, for T codes and rennumbers program blocks grouped by T code. The following prompts will appear when this item is selected:

ENTER STARTING N-CODE NUMBER: This prompt will appear when a T code is found. Blocks of code will be renumbered, starting with the code entered here, until the next T code is found.

ENTER N-CODE INCREMENT: Each block that is renumbered is incremented by the amount entered here.

REVERSE + & - SIGNS

This menu item will reverse the signs of all numeric values associated with one or more address codes in the program. If a certain block is selected, only the address codes in the selected block will be affected. The following prompt will appear when this item is selected:

ENTER ADDRESS CODE(S) TO CHANGE: Type in the valid address code characters whose associated values in the program are to have their signs reversed. They can be entered in any order, but duplicate entries will be ignored. Entries that don't make sense in the context of reversing signs, such as the 'G' character, will also be ignored.

REVERSE X & Y

When this menu item is selected, all X address codes in the program will be changed to Y address codes, and Y's to X. If a block is selected, only the address codes contained within it will be affected.

THE I/O MENU

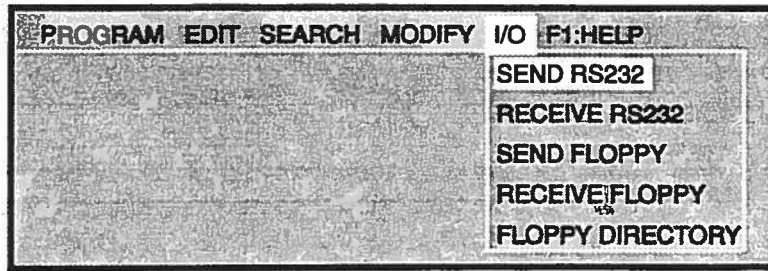


Figure 6. The I/O menu items.

SEND RS232

This menu will send program(s) that are selected from the program directory to the RS-232 port. When this menu item is selected, a list of all the programs in memory is presented, with 'ALL' at the end.

To select a program, cursor to the program number and press the INSERT key. A highlighted character will appear before the program to indicate it has been selected. Pressing INSERT again will deselect the program, and the highlighted character will disappear. The DELETE key can be used to deselect all selected programs.

To send the selected program(s), press the WRITE key. When the cursor is on 'ALL', pressing WRITE will send all programs to RS-232, regardless of the number of programs selected.

RECEIVE RS232

This menu item will receive program(s) from the RS-232 serial port. The program(s) will then be stored in the CNC memory with the corresponding Onnnn program number(s).

SEND FLOPPY

This menu item will send program(s) to the floppy. When this menu item is selected, a list of all the programs in memory is presented, with 'ALL' at the end. Refer to the "SEND RS232" section for a description of how to select and send program(s).

ENTER FLOPPY FILENAME: Type in the desired floppy filename (in standard PC format) for the selected program(s), then press the WRITE key.

RECEIVE FLOPPY

This menu item will receive programs from the floppy. The program(s) will then be stored in the CNC memory with the corresponding Onnnn program number(s).

ENTER FLOPPY FILENAME: Type in the filename (in standard PC format) of the floppy file being received, then press the WRITE key.

FLOPPY DIRECTORY

This menu item will generate a Disk Directory Listing that will be saved in program Onnnn, where nnnn is defined in Parameter 227. The default value is 8999.

THE F1:HELP MENU

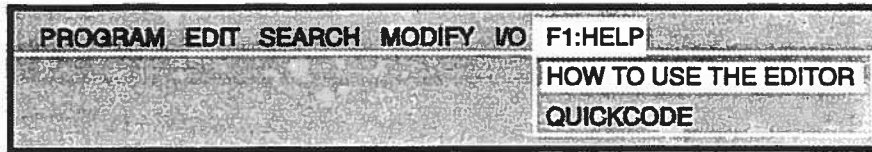


Figure 7. The F1:HELP menu items.

HOW TO USE THE EDITOR

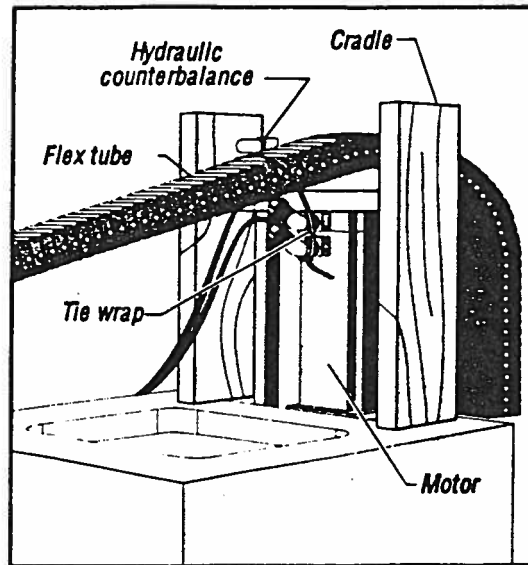
Selecting this menu item will place an on-line help manual on the inactive screen. This help manual gives a brief description of the editor and its features. The arrow keys, PAGE UP and PAGE DOWN keys, and the jog handle can be used to maneuver through this help manual. Pressing the UNDO key returns the user to the active program.

QUICKCODE

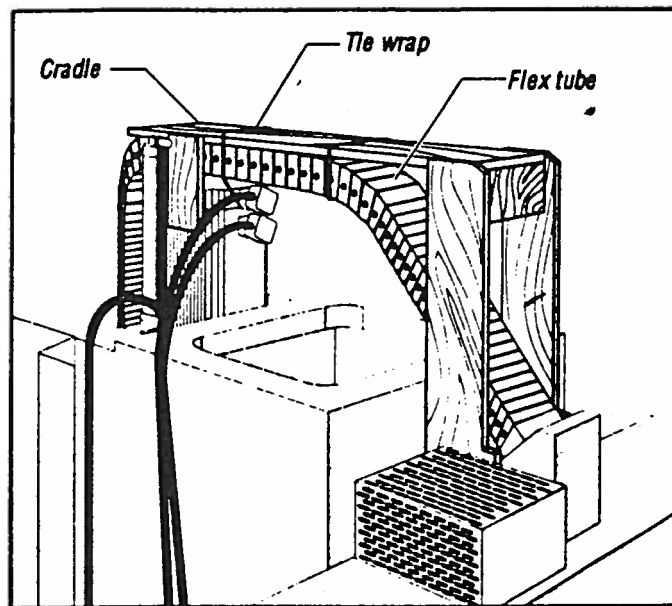
Selecting this menu item will place QuickCode on the inactive side of the editor. All QuickCode functions are now available to the user. Refer to the Operator's Manual for a full description of QuickCode. Pressing the EDIT key will exit QuickCode.

WARNING !!

FLEX TUBE CRADLE REMOVAL



Flex Tube Cradle for VF-0/1/2



Flex Tube Cradle for VF-3/4

VF -1/2 PROCEDURE

1. **BEFORE POWERING UP MACHINE**, remove tie wrap.
2. **POWER UP** the machine following installation instructions.
3. With Z-axis at top of travel, **REMOVE FLEX TUBE CRADLE**.

VF - 3/4 PROCEDURE

1. **BEFORE POWERING UP MACHINE**, remove tie wrap.
2. **LIFT AND REMOVE FLEX TUBE CRADLE**.