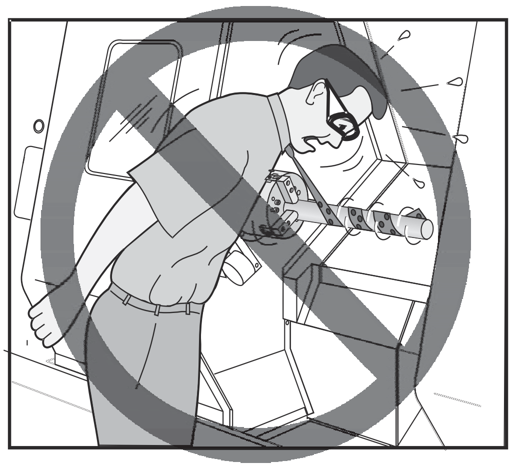
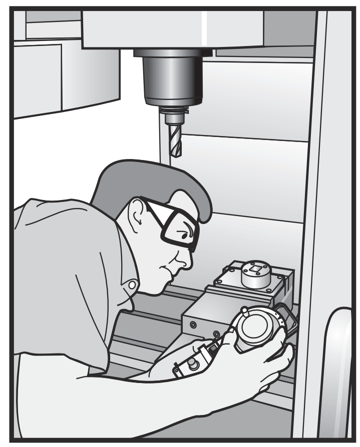
VF- Series Operators Manual 96-8000 English June 1995

HAAS SAFETY PROCEDURES

THINK SAFETY!





DON'T GET CAUGHT UP IN YOUR WORK

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

Important – This machine is to be operated only by trained personnel in accordance with the Operator's Manual, safety decals, safety procedures and instructions for safe machine operation.

Read Before Operating This Machine:

- Only authorized personnel should work on this machine. Untrained personnel present a hazard to themselves and the machine, and improper operation will void the warranty.
- Use appropriate eye and ear protection while operating the machine. ANSI approved impact safety goggles and OSHA approved ear protection are recommended to reduce the risks of sight damage and hearing loss.
- Do not operate the machine unless the doors are closed and the door interlocks are functioning properly. Rotating cutting tools can cause severe injury. When a program is running, the mill table and spindle head can move rapidly at any time in any direction.
- The Emergency Stop button is the large, circular red switch located on the Control Panel. Pressing the Emergency Stop button will instantly stop all motion of the machine, the servo motors, the tool changer, and the coolant pump. Use the Emergency Stop button only in emergencies to avoid crashing the machine.
- The electrical panel should be closed and the key and latches on the control cabinet should be secured at all times except during installation and service. At those times, only qualified electricians should have access to the panel. When the main circuit breaker is on, there is high voltage throughout the electrical panel (including the circuit boards and logic circuits) and some components operate at high temperatures. Therefore, extreme caution is required. Once the machine is installed, the control cabinet must be locked and the key available only to qualified service personnel.
- Consult your local safety codes and regulations before operating the machine. Contact you dealer anytime safety issues need to be addressed.
- DO NOT modify or alter this equipment in any way. If modifications are necessary, all such requests must be handled by Haas Automation, Inc. Any modification or alteration of any Haas Milling or Turning Center could lead to personal injury and/or mechanical damage and will void your warranty.
- It is the shop owner’s responsibility to make sure that everyone who is involved in installing and operating the machine is thoroughly acquainted with the installation, operation, and safety instructions provided with the machine BEFORE they perform any actual work. The ultimate responsibility for safety rests with the shop owner and the individuals who work with the machine.
- **This machine can cause bodily injury.**
- **Do not operate with the door open.**
- **Do not operate without proper training.**
- **Always wear safety goggles.**
- **The machine is automatically controlled and may start at any time.**
- **The electrical power must meet the specifications in this manual. Attempting to run the machine from any other source can cause severe damage and will 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.**
- **Never service the machine with the power connected.**
- **Improperly clamped parts machine at high feeds/feed may be ejected and puncture the safety door. Machining oversized or marginally clamped parts is not safe.**
- **Windows must be replaced if damaged or severely scratched - Replace damaged windows immediately.**
- **The spindle head can drop without notice. Personnel must avoid the area directly under the spindle head.**
- **Do not reset a circuit breaker until the reason for the fault is investigated. Only Haas-trained service personnel should troubleshoot and repair the equipment.**

- **Follow these guidelines while performing jobs on the machine:**

Normal operation - Keep the door closed and guards in place, while machine is operating.
Part loading and unloading – An operator opens the door or guard, completes task, closes door or guard before pressing cycle start (starting automatic motion).
Tool loading or unloading – A machinist enters the machining area to load or unload tools. Exit the area completely before automatic movement is commanded (for example, next tool, ATC/Turret FWD/REV).
Machining job set-up – Press emergency stop before adding or removing machine fixtures.
Maintenance / Machine Cleaner– Press emergency stop or power off the machine before entering enclosure.
Do not enter the machining area anytime the machine is in motion; severe injury or death may result.

Unattended Operation

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

As it is the shop owner's responsibility to set up the machines safely and use best practice machining techniques, it is also their responsibility to manage the progress of these methods. The machining process must be monitored to prevent damage if a hazardous condition occurs.

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

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

MODIFICATIONS TO THE MACHINE

DO NOT modify or alter this equipment in any way. If modifications are necessary, all such requests must be handled by Haas Automation, Inc. Any modification or alteration of any Haas machining center could lead to personal injury and/or mechanical damage and will void your warranty.

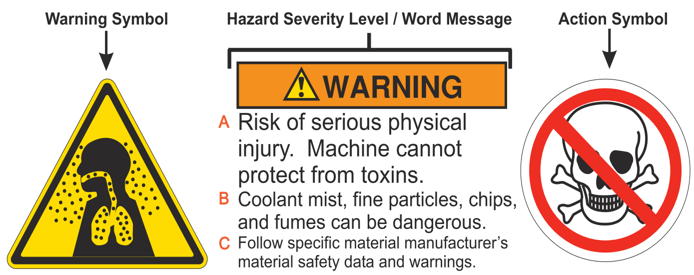
SAFETY DECALS

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

Never alter or remove any safety decal or symbol.

Each hazard is defined and explained on the general safety decal, located at the front of the machine. Particular locations of hazards are marked with warning symbols. Review and understand the four parts of each safety warning, explained below, and familiarize yourself with the symbols on the following pages.

NEVER OPERATE THIS MACHINE WITH THE DOORS OPEN



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

Word Message - Clarifies or reinforces the intent of the warning symbol.
A: Hazard.
B: Consequence if warning is ignored.
C: Action to prevent injury. Also refer to Action Symbol.

Hazard Severity Level - Color-coded to indicate risk in ignoring a hazard.
Red + "DANGER" = Hazard WILL cause death or serious injury if ignored.
Orange + "WARNING" = Hazard COULD cause death or serious injury if ignored.
Yellow + "CAUTION" = Hazard MAY cause minor to moderate injury if ignored.
Blue + "NOTICE" = Indicates an action to prevent damage to the machine.
Green + "INFORMATION" = Details about machine components.

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

MILL WARNING DECALS

DANGER

Electrocution hazard. Death by electric shock can occur. Turn off and lock out system power before servicing.

Risk of serious physical injury. Machine cannot protect from toxins. Coolant mist, fine particles, chips, and fumes can be dangerous. Follow specific material manufacturer's material safety data and warnings.

Risk of fire and explosion. Machine is not designed to resist or contain blasts or fire. Do not machine explosive or flammable materials or coolants. Refer to specific material manufacturer's material safety data and warnings.

Severe injury can occur. Moving parts can entangle, trap, and cut. Sharp tools or chips can cut skin easily. Ensure the machine is not in automatic operation before reaching inside.

Automatic Machine may start at any time. Injury or death could be caused by untrained operator. Read and understand operator's manual and safety signs before using this machine.

Risk of serious bodily injury. The enclosure may not stop every type of projectile. Double-check job set up before beginning any machining operations. Always follow safe machining practices. Do not operate with doors or windows open or guards removed.

Risk of bodily injury. Serious cuts, abrasions, and physical injury may result from slips and falls. Avoid using the machine in wet, damp, or poorly lit areas.

Risk of eye and ear injury. Flying debris into unprotected eyes can cause loss of sight. Noise levels can exceed 70 dBA. Must wear safety glasses and hearing protection when operating or in the area of machine.

Safety windows may become brittle and lose effectiveness when exposed to machine coolants and oils over time. If signs of discoloration, crazing, or cracking are found, replace immediately. Safety windows should be replaced every two years.

WARNING

Severe injury can occur. Moving parts can entangle and trap. Always secure loose clothing and long hair.

Impact hazard. Machine components can crush and cut. Do not handle any part of the machine during automatic operation. Always keep clear of moving parts.

Risk of serious bodily injury. Follow safe clamping practices. Inadequately clamped parts can be thrown with deadly force. Securely clamp workpieces and fixtures.

Moving parts can crush. The tool changer will move in and crush your hand. Never place your hand on the spindle and press ATC FWD, ATC REV, NEXT TOOL, or cause a tool change cycle.

- Do not allow untrained personnel to operate this machine.
- Do not alter or modify machine in any way.
- Do not operate this machine with worn or damaged components.
- No user serviceable parts inside. Machine must be repaired or serviced by authorized service technicians only.

NOTICE

Coolant Tank Maintenance

Gate Filter

Clean the filter screen weekly.

Remove the coolant tank cover and clean out any sediment inside the tank weekly.

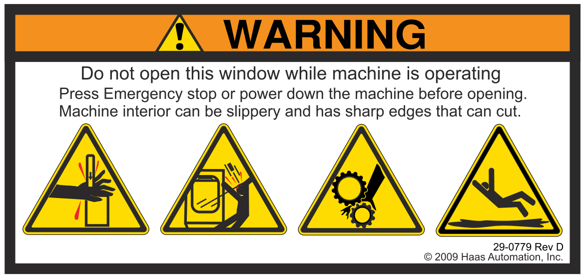
Do not use plain water, permanent corrosion damage will result. Rust inhibiting coolant is required.

Do not use toxic or flammable liquids as a coolant.

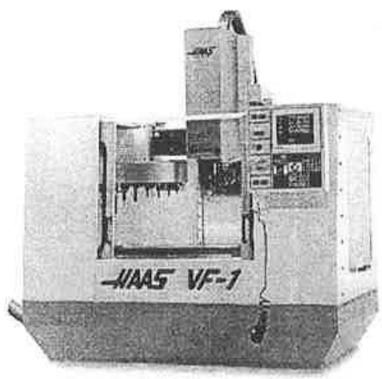
LATHE WARNING DECALS

OTHER SAFETY DECALS

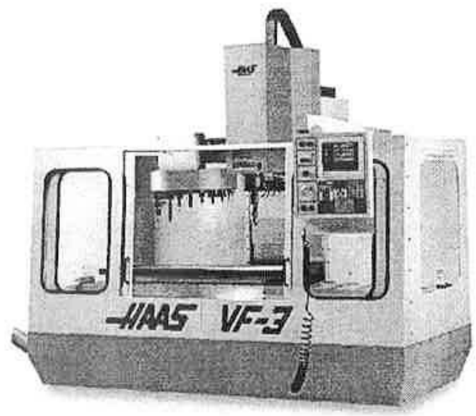
Other decals may be found on your machine, depending on the model and options installed:



VF-SERIES OPERATORS MANUAL

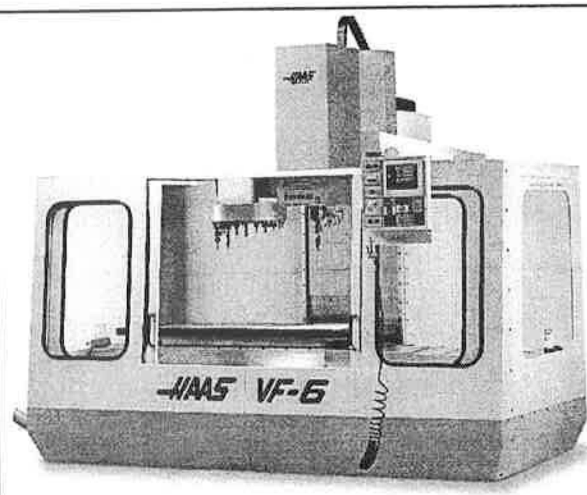


VF-1



VF-3

VF-4



New VF-6

HAAS VERTICAL MACHINING CENTERS

HAAS AUTOMATION, INC.

Haas Automation, Inc.
9601 Lurline Ave.
Chatsworth, CA. 91311
(818) 885-6050

JUNE 1995

WARRANTY REGISTRATION CERTIFICATE

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

CONTROL: (All components within the main control cabinet and pendant cabinet): Warranted against defects in material and workmanship for a period of two years from date of purchase.

NON-CONTROL: The remainder of the machine is warranted against defects in material and workmanship for a period of one year from date of purchase.

EXCLUSIONS: Items subject to wear from machining parts, such as paint, windows, 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
9601 Lurline Ave.
Chatsworth, CA 91311
Phone: (818) 885-6050 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: ______________________ Contact Name: ______________
Address: ______________________ Dealer: ______________
______________________ Date Purchased: ______________
Model No.: ______________________ Serial Number: ______________
Telephone: ( ) ______________________ FAX: ______________

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.

CONTENTS

II. PROGRAMMING

I. GENERAL INFORMATION

The operation of the VF-Series Vertical Machining Center requires that a part program be designed, written, and entered into the memory of the controller. 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 subsequently 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.

The HAAS VF-Series Vertical Machining Center control can store and save more than one part program even when the machine's power is turned off. Programming of the part program can also be done at the CNC by using the EDIT function. Editing can even be done while another program is being run.

This manual can be used as both an operator's manual and as a programmer's manual. The following two sections are to be used as a basic introduction to programming. However, a CNC programmer needs much more training and information available to him or her before attempting to program on the machine.

The remainder of this manual is divided into a *Programming* chapter (II) and an *Operation* chapter (III).

1. BASIC PROGRAMMING

An "NC" machine is a machine that is numerically controlled, i.e., the machine tool is controlled by a code system that enables it to be operated with little or no personal supervision and with a great deal of repeatability. Thus, the same task may be performed over and over with minimum human error. "CNC" is the same type of operating system, with the exception that the machine tool is monitored by a computer. The term CNC stands for **C**omputerized **N**umerical **C**ontrol.

Anyone who can operate a manual machine can learn to program an NC or CNC machine. The main difference is that instead of cranking handles to position a slide to a certain point, that dimension would be stored in the memory of the machine control **once**. The control thereafter will move the machine automatically when called out in the program, on each individual part.

Programming for parts that are complex is done by building several steps, or operations, together to form the finished product.

If a person sets out to operate and program an CNC controlled machine, that person must have a basic understanding of machining practices and a working knowledge of math. He/she needs also 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 section is to be used 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 HAAS Vertical Machining Center.

The purpose of this section is to give a basic understanding of CNC programming and its 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.

1.1 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.

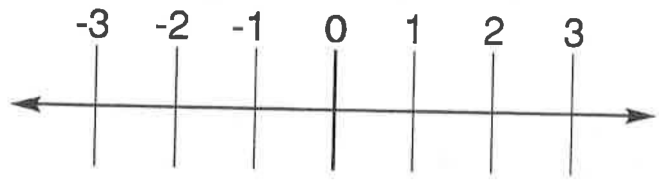


Fig. 1-1 *Horizontal number line.*

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) pictures 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.

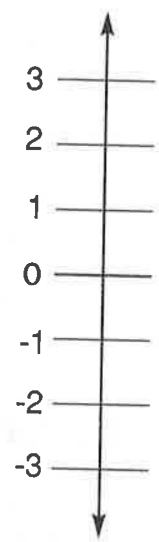


Fig. 1-2

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.

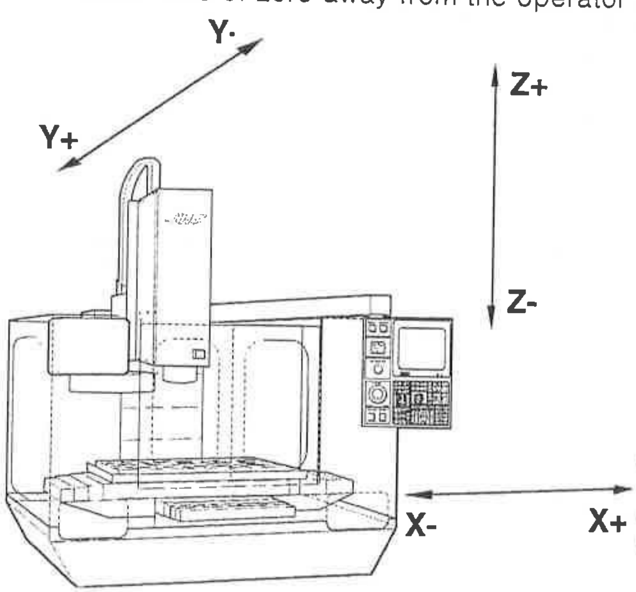


Fig. 1-3 *VF-1 showing X, Y, and Z axis lines.*

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. For the HAAS VF-1 and VF-0, we have access to 20 inches in the X-axis, 16 inches in the Y-axis, and 20 inches in the Z-axis. For the VF-2, we have access to 30 inches in the X-axis, 16 inches in the Y-axis, and 20 inches in the Z-axis. For the VF-3, we have access to 40 inches in the X-axis, 20 inches in the Y-axis, and 25 inches in the Z-axis.

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, this will be entered in the tool length offset as a negative value. The range of travel on the HAAS VF Series 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 at right 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.

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--

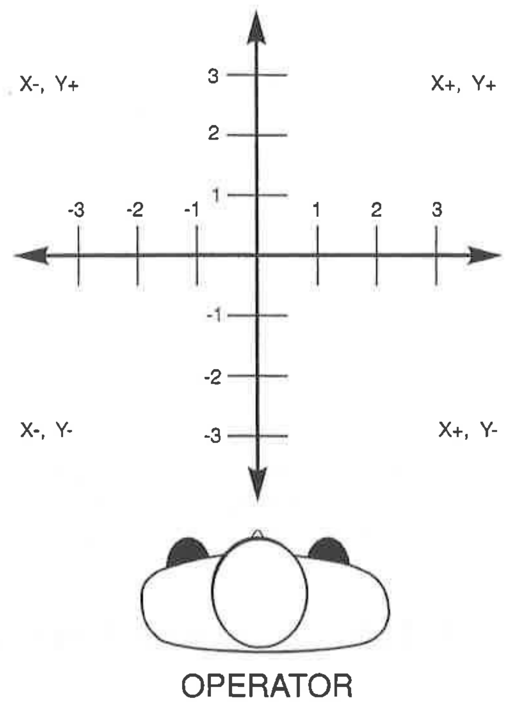


Fig. 1-4 *View of X,Y grid from above*

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, 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.

The layout of the **A** and **B** axes on the HAAS five-axis control are depicted at right and on the next page. 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

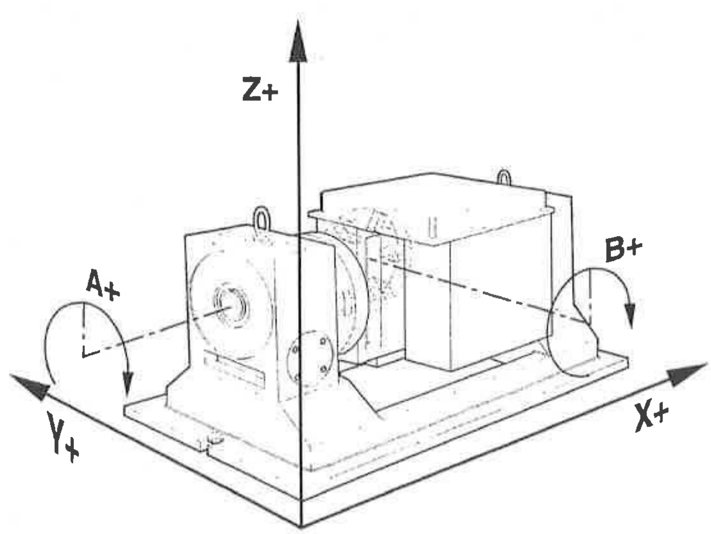


Fig. 1-5 *Positive tool movement for fifth axis machines.*

movement direction. For the right hand rule, the fingers will point opposite of the positive rotary table movement. Refer to Figures 1-5 and 1-6.

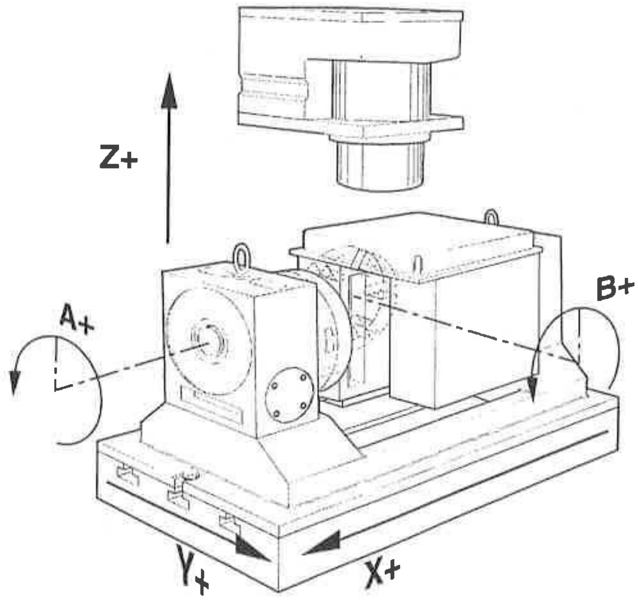


Fig. 1-6 *Positive mounting surface movement for all axes.*

NOTE: Figures 1-5 and 1-6 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.

1.2 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.

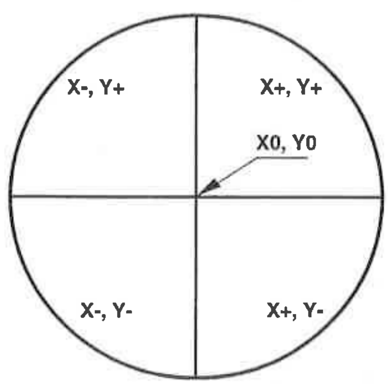


Fig. 1-7 All four quadrants will have to be accessed to machine this part.

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 it's datum lines running through the center. The setup of such a part may look like Fig. 1-7.

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.

1.3 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 or A.B.S., 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, or I.N.C. 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.

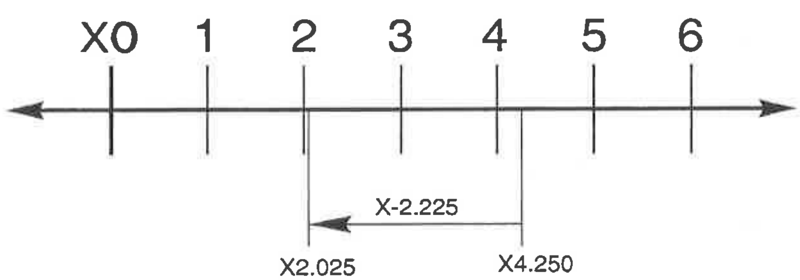


Fig. 1-8 An example of an incremental move.

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.

1.4 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 in inches per minute.

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.

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 stop program. 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.

1.5 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 VF Series are listed in the Operator/Programming Manual, and are indicated by an asterisk (*) next to the specific G codes.

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.

1.6 PROGRAM FORMAT

Program format, or program 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;

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.

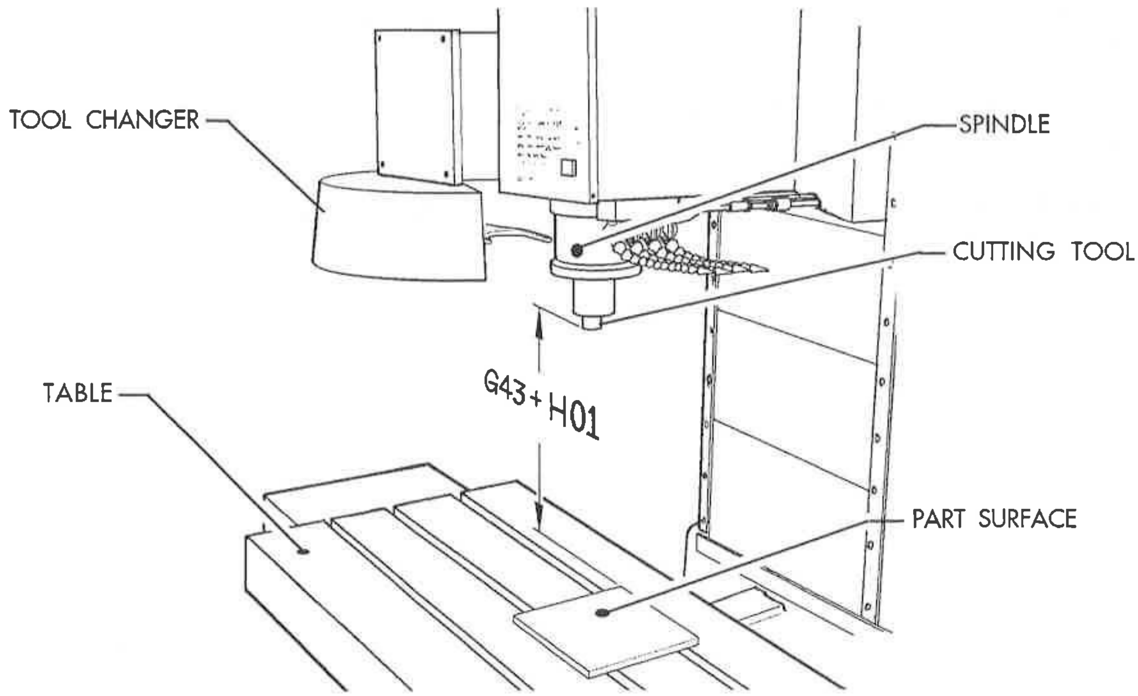


Fig. 1-9 Tool length offset and tool length compensation.

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.)

1.7 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, see Chapter II, Section 3.17.

2. PROGRAMMING EXAMPLES

2.1 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 2-1 on the facing page:

```
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
```

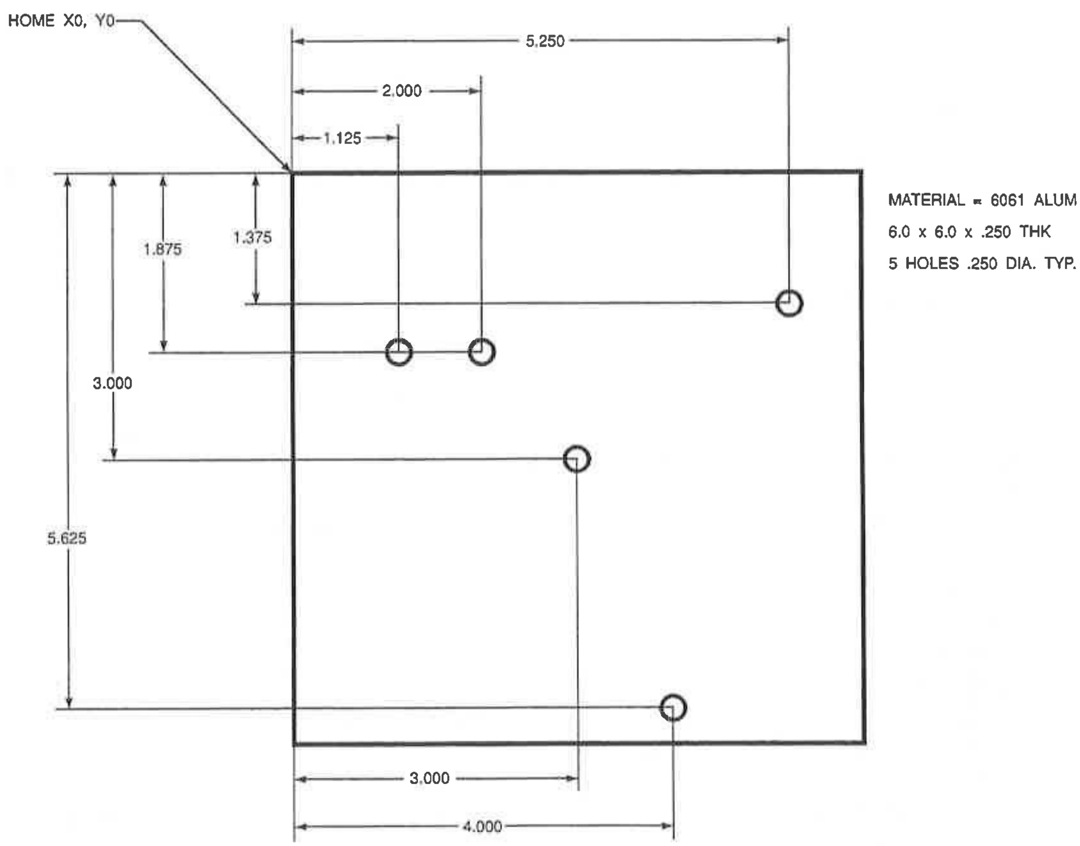


Fig. 2-1 Canned cycle programming example using aluminum block.

2.2 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
%
```

2.3 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.

```
%
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
%
```

```
%
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
%
```

2.4 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
T1 M06
G00 G90 G54 X0 Y0 S1500 M03
G43 H01 Z.1 M08
M98 P3000
G55
M98 P3000
G56
M98 P3000
G57
M98 P3000
G58
M98 P3000
G59
M98 P3000
G00 Z1.0 M09
G28 G91 Y0 Z0
M30
%
```

```
%
O3000
X0 Y0
G83 Z-1.0 F15. R.1 Q.2
G00 G80 Z.2
M99
%
```

The following illustration 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.

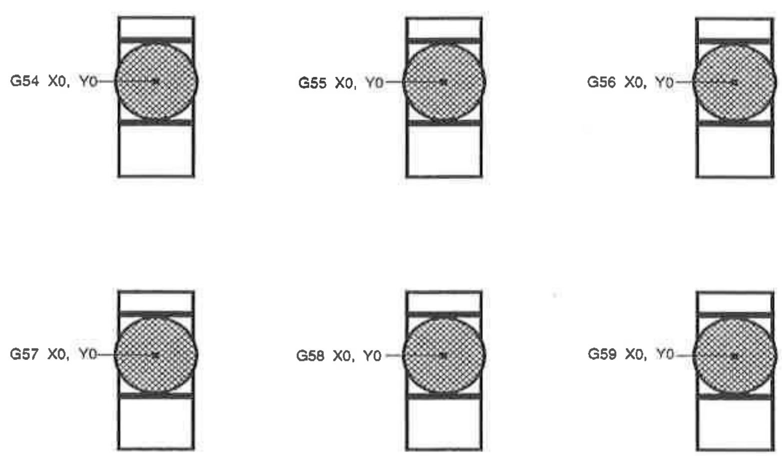


Fig. 2-2 Multiple fixture setup.

2.5 LOOPING CANNED CYCLES

Below is an example of a program using a drilling canned cycle that is incrementally looped. Compare the grid plate drawing on the next page 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
G90 Y-2.0                          (Or stay in G91 and repeat Y-1.0)
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.

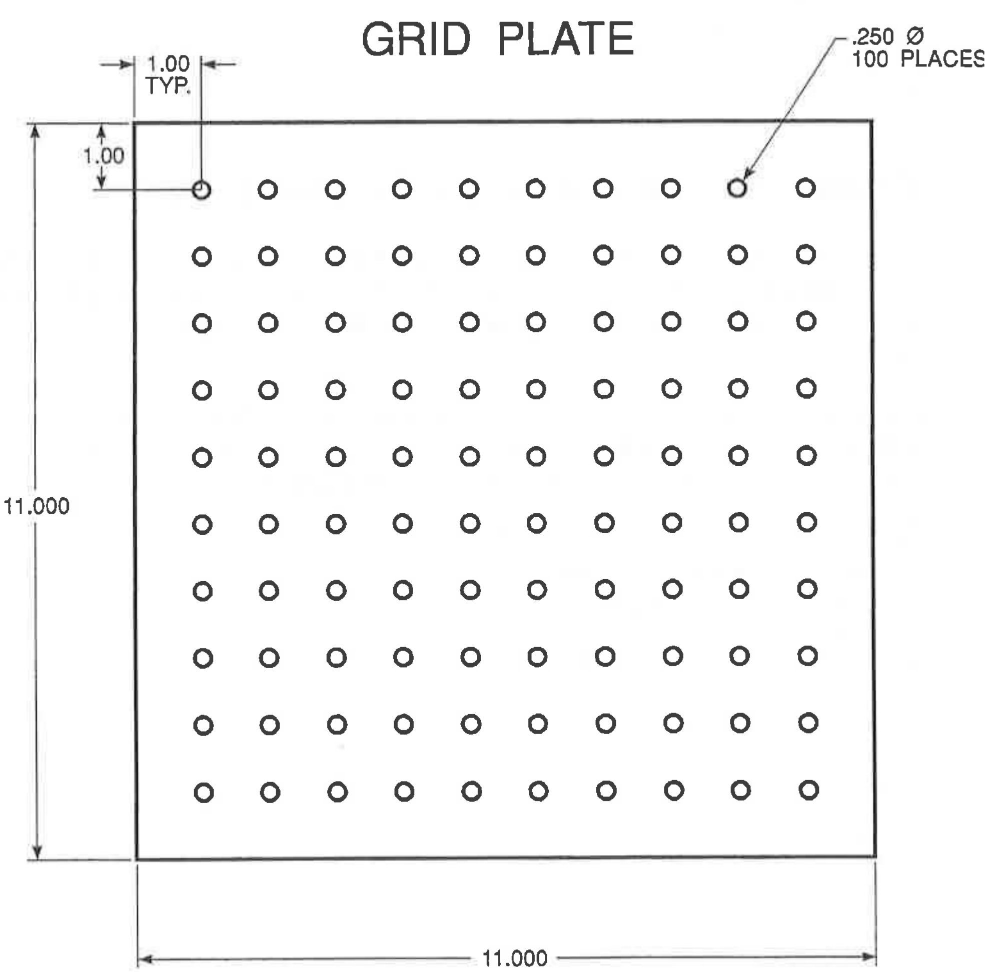


Fig. 2-3 Grid plate for multiple-fixture subprogram exercise.

2.6 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
%
```

2.7 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:

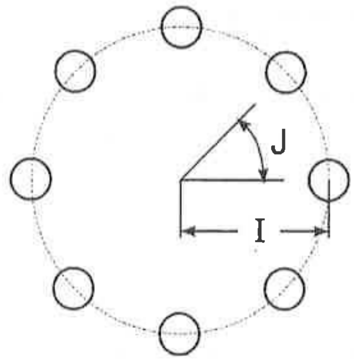
1. The tool must be placed at the center of the bolt pattern before the canned cycle execution. The center is usually X0, Y0.
2. The **J** code is the angular starting position and is always 0 to 360 degrees counterclockwise from the three o'clock position.

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:

```
%
05000
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
```

[See following page for illustrations of G70, G71, and G72.]

G70
BOLT HOLE CIRCLE

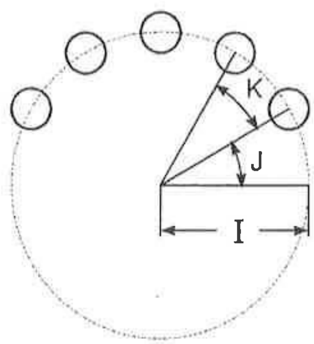


I = Radius of the bolt circle

J = Starting angle from three o'clock position

L = Number of holes (evenly spaced)

G71
BOLT HOLE ARC



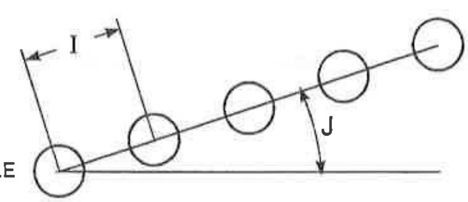
I = Radius of the bolt circle

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

2.8 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.

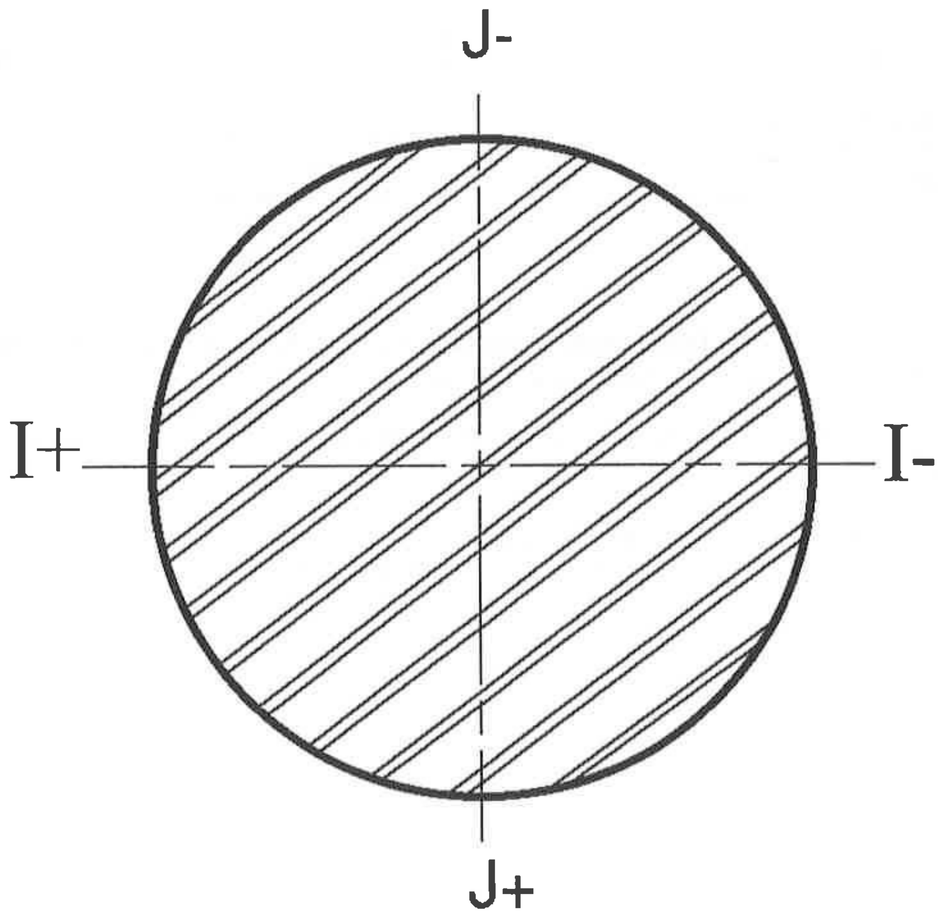
Before we get into circular interpolation and how it is used, below are a few rules about cutter compensation that have to be closely followed in order to perform successful machining operations. Always refer to these rules when programming!

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.

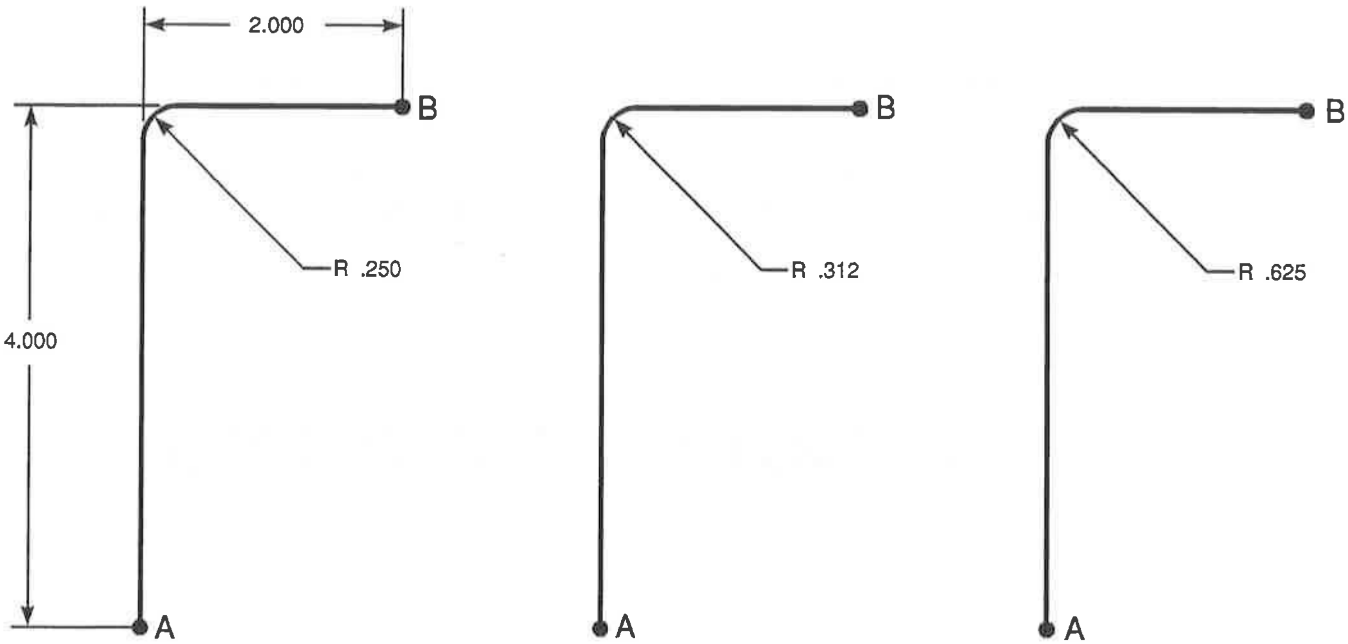
3) In most machines, during cutter compensation, a linear X,Y move that is smaller than the cutter

CIRCLE SECTIONS



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.
DO THE FOLLOWING EXERCISE FOR PRACTICE.



G01 Y3.75
G02 X.250 Y4.0 R.250
G01 X2.0

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.

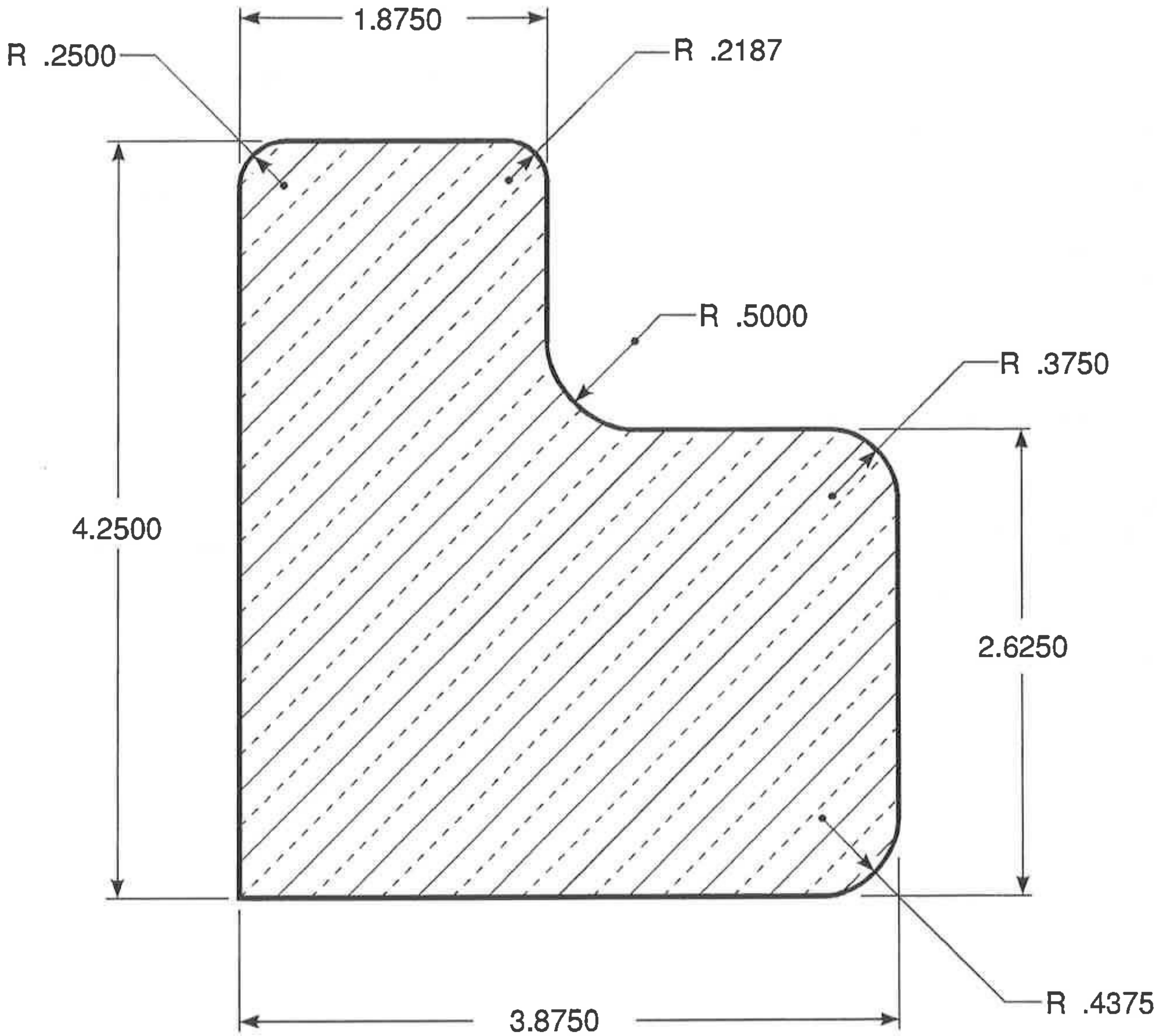


Fig. 2-4 *Part for G02, G03, and R practice.*

See following page for the correct program.
The correct program for Figure 2-4 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.

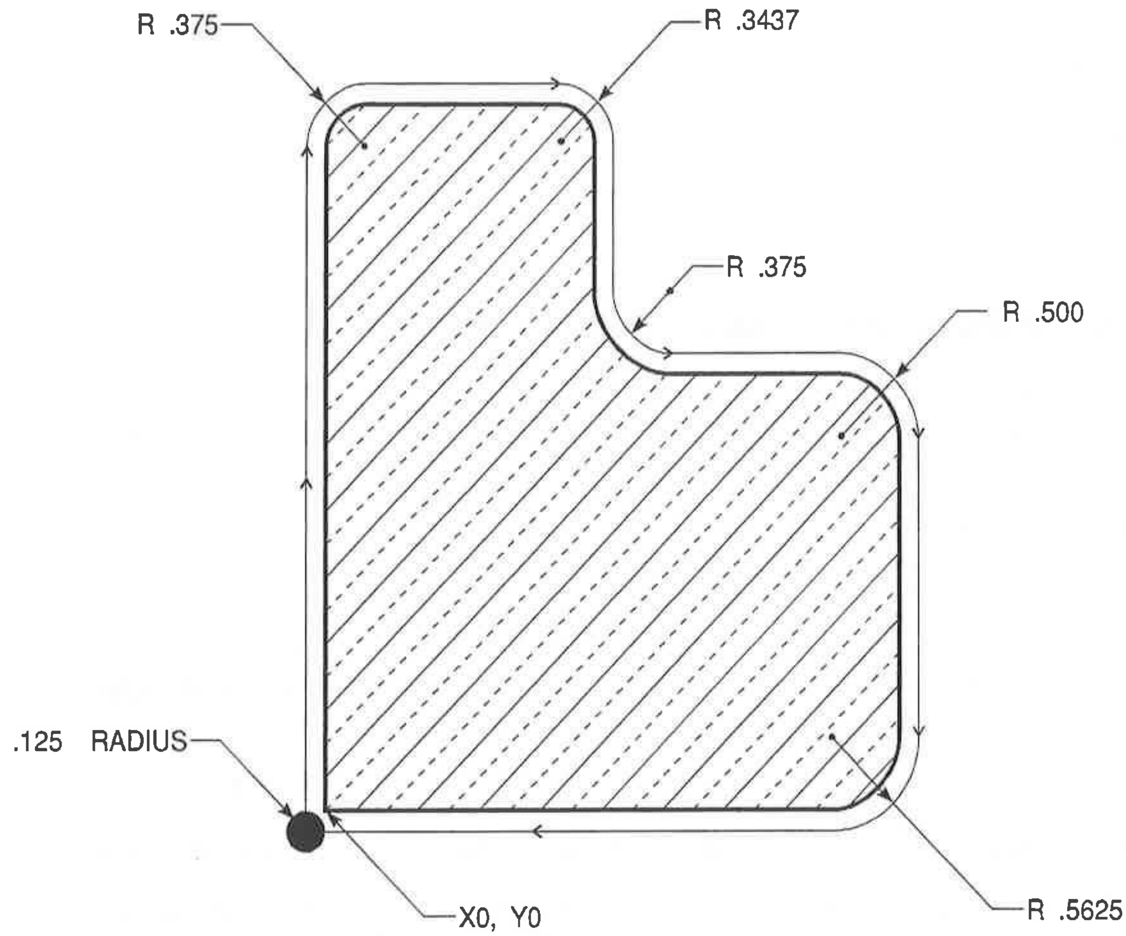


Fig. 2-5 *Programming exercise showing tool path.*

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 2-5.

```
%
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
%
```

2.9 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 Section 2.8, we can see the program below using the **R** or the **I** and **J**:

Using **R**:

```
G01  Y3.750
G02  X.250  Y4.0  R.250
G01  X2.0
```

Using **I** and **J**:

```
G01 Y3.750
G02  X.250  Y4.0  I.250  J0
G01  X2.0
```

Note the +1.250 move and compare it with the circle sections illustration on page 22.

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.

2.10 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.

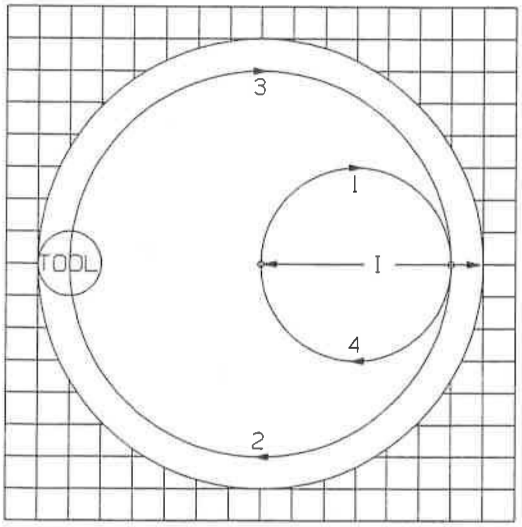
NOTE: The tool must be positioned at the center of the circle, either in a previous block or in the G12/G13 line by using **X** and **Y**.

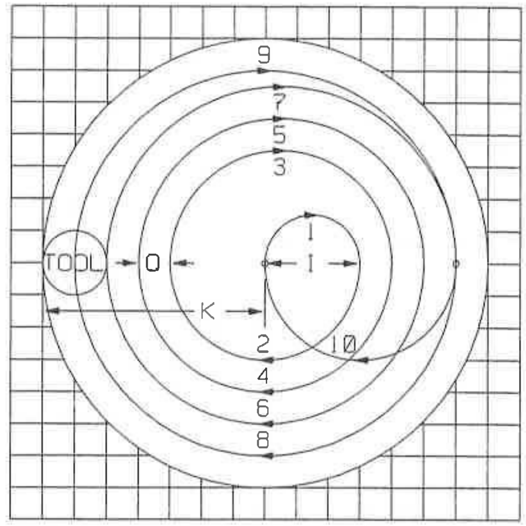
WARNING! Any software revision before 1.37, 2.8, and 3.2 will execute a rapid **Z** depth movement. Current software supports linear feed in the **Z** depth move.

The diagram on the next page 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

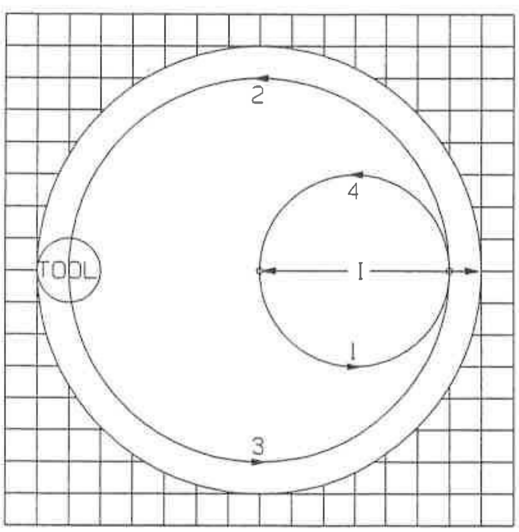


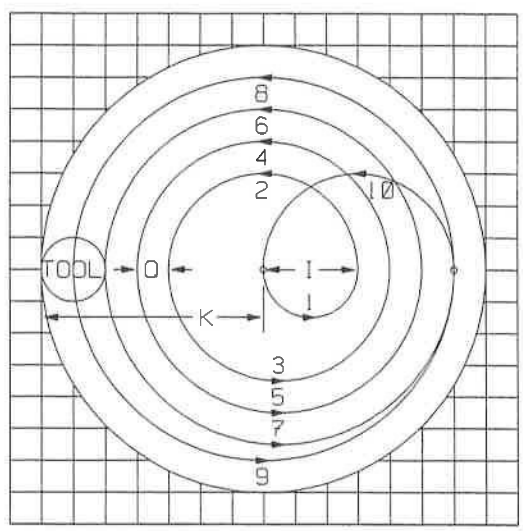


I ONLY

I, K, and Q

G13





PROGRAM LINE REQUIREMENTS:

Z = Depth of cut or increment.	
F = Feed Rate.	
I = Radius of first circle	(Finished radius if no K specified.).
K = Radius of finished circle	(If using I, K, and Q.).
Q = Radius increment or cutter overlap	(Must use with K).
D = Tool geometry offset number	(Not required).
L = Loop count for incremental Z depth stepping	(Optional).

Ex. 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
%
```

Ex. 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  K.500  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.

Ex. 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.

2.11 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
%
```

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
%
```

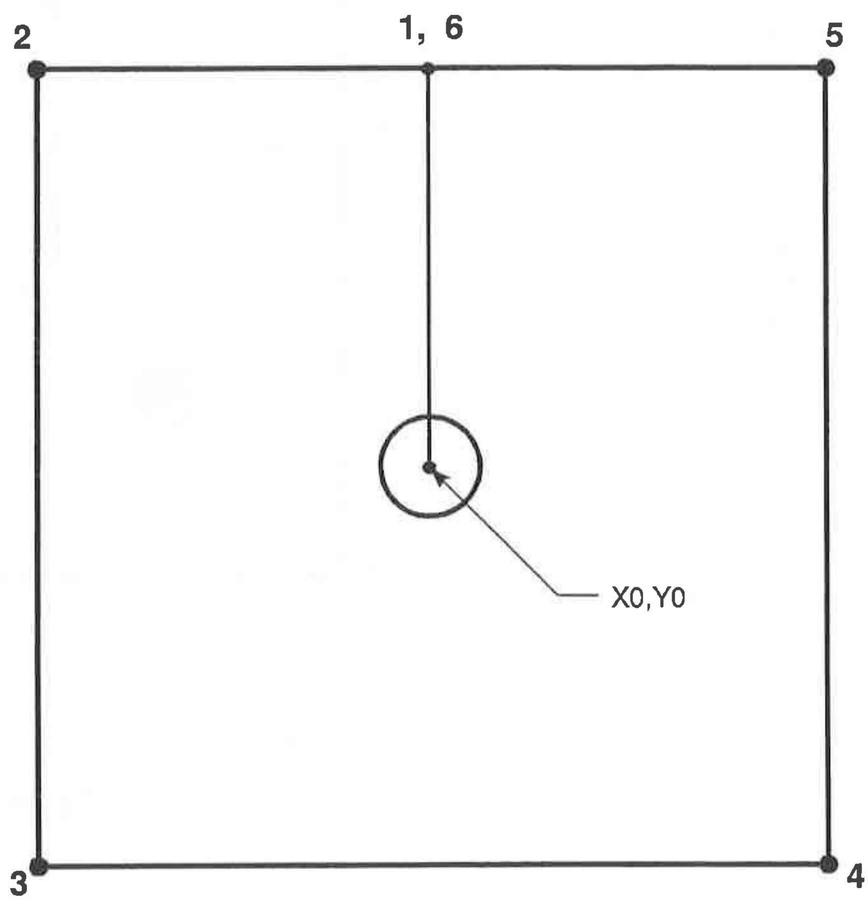


Fig. 2-6 Pocket milling exercise for G150 operation.

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.O1  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
Y4.0
X4.0
Y2.0
X2.0
Y3.0
X0
Y0
M99
%
```

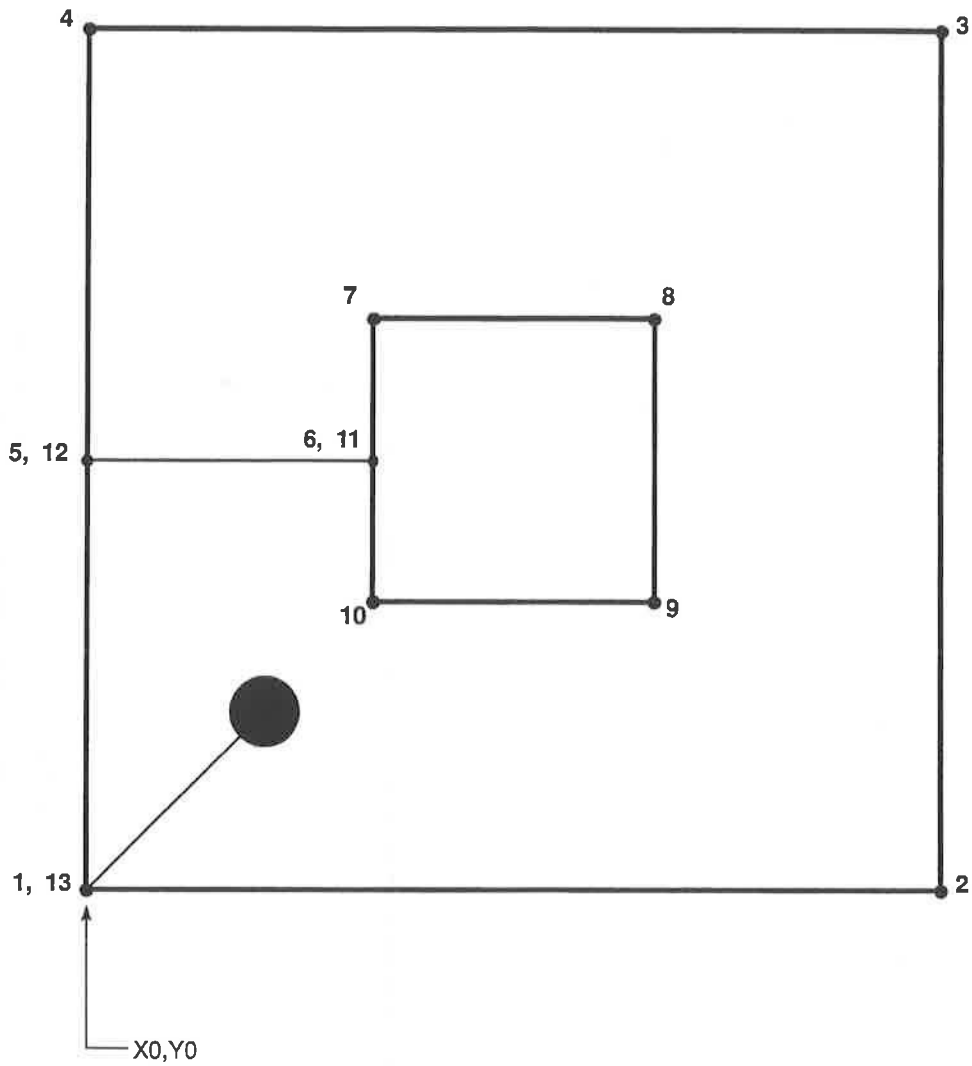


Fig. 2-7 *Square island programming exercise using G150.*

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
%
```

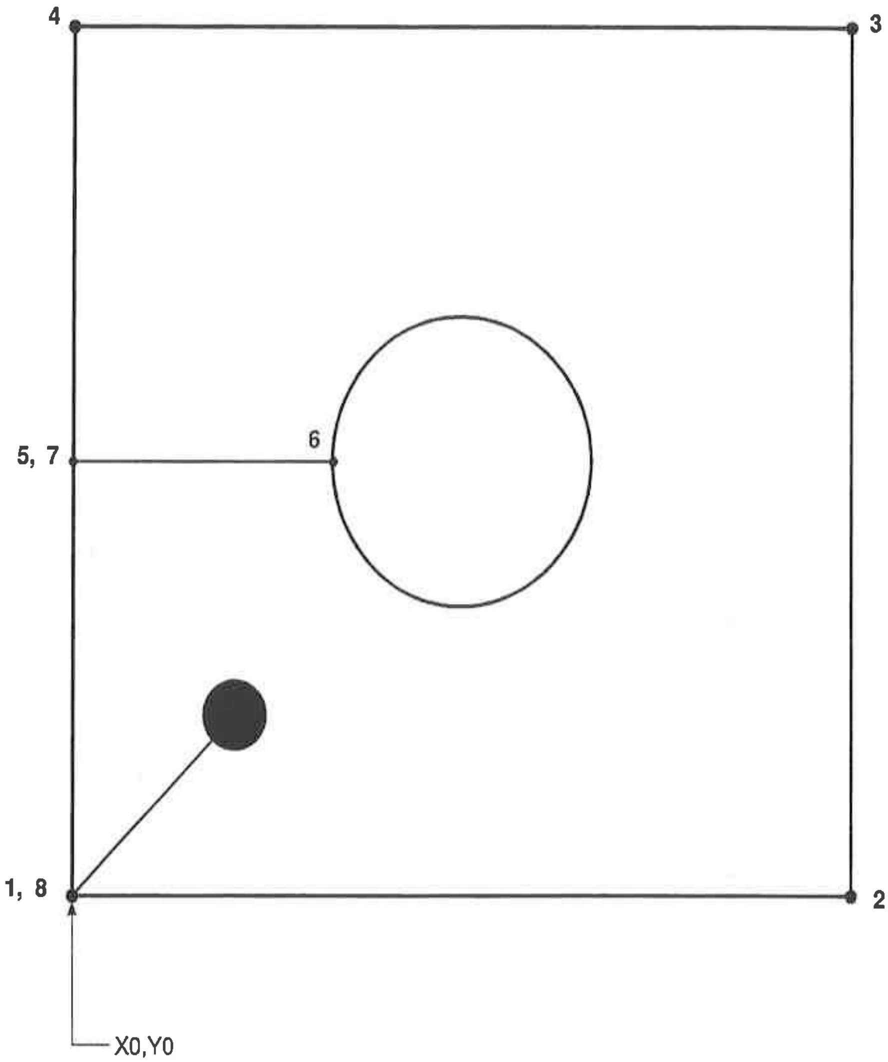


Fig. 2-8 *Round island programming exercise using G150.*

2.12 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.

Ex.

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.

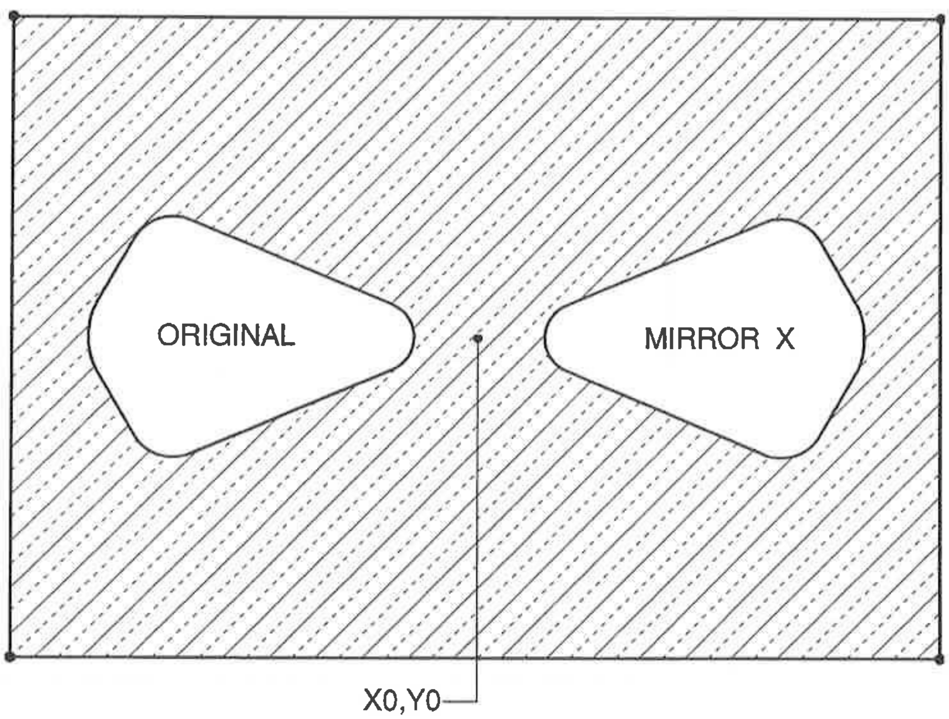


Fig. 2-9 *Mirror image and pocket milling exercise.*

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
%
```

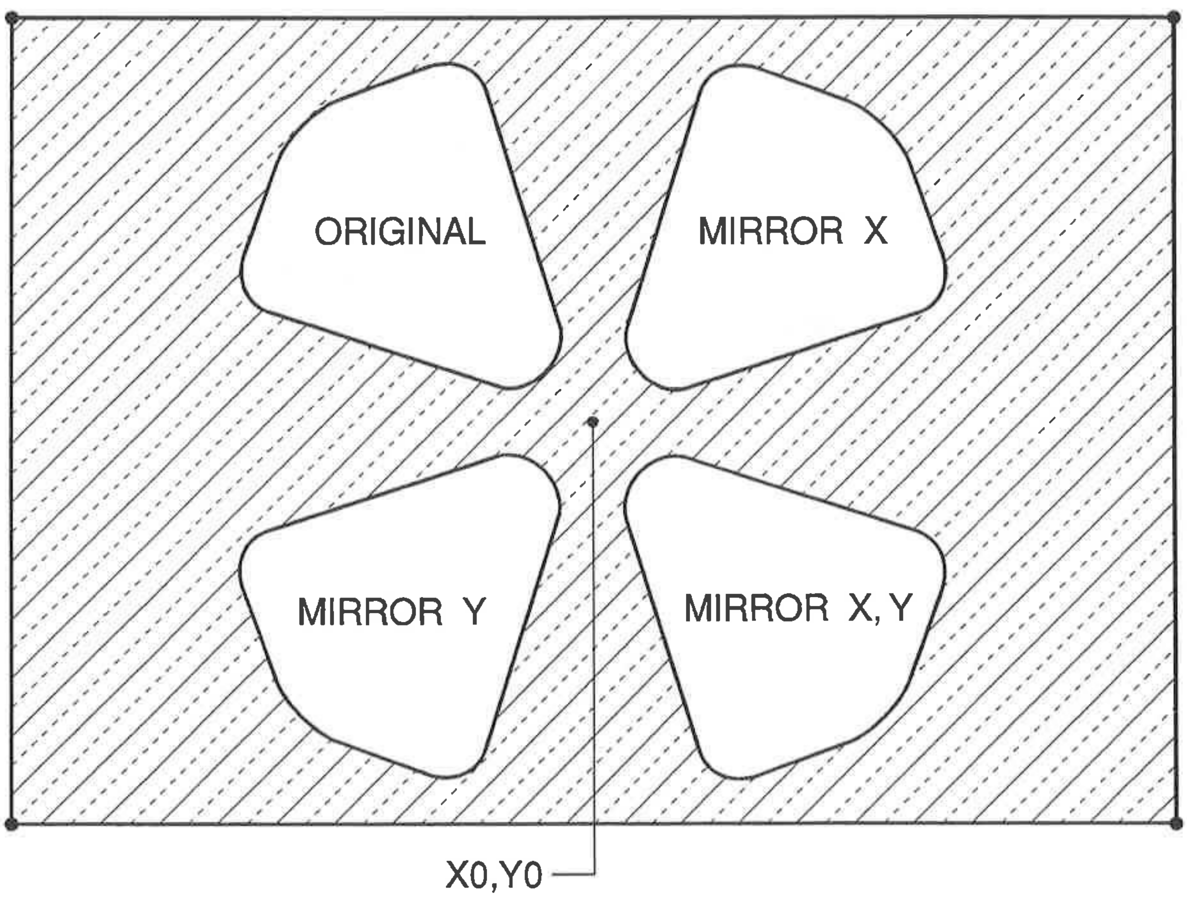


Fig. 2-10 *Mirror image (X, Y, and X-Y) and pocket milling exercise.*

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
%
```

2.13 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

Ex. 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

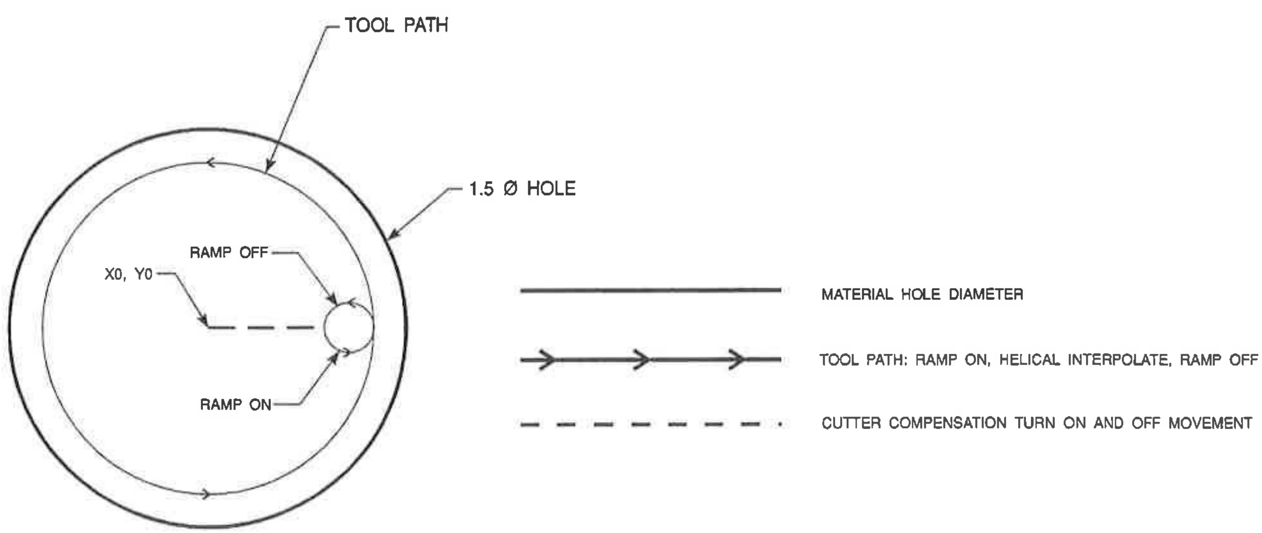


FIG. 2-11 *Thread milling exercise*

The code for Figure 2-11 (previous page) 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.200  F7.
G03  I-.375  Z-.475
G03  X.175  R.200
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.

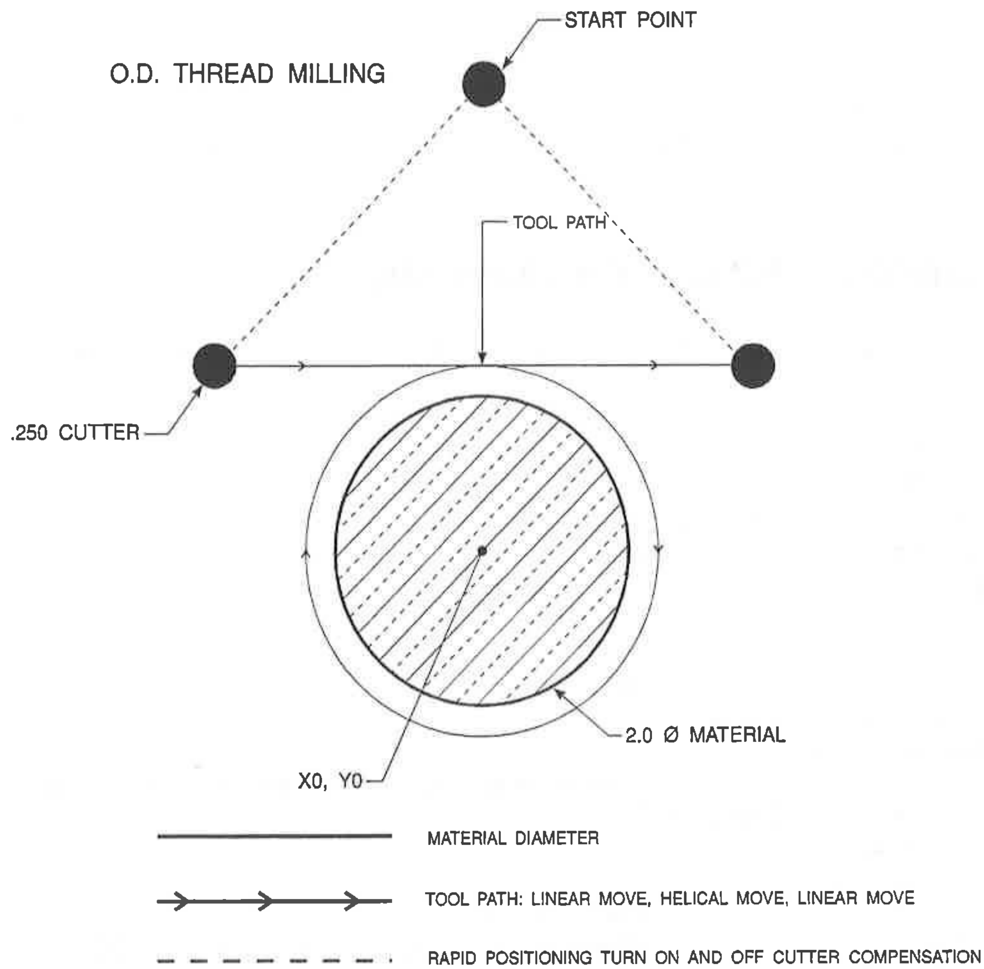


Fig. 2-12 O.D. thread milling exercise.

O.D. THREAD MILLING -

The code for the previous illustration 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.

2.14 SINGLE-POINT THREAD MILLING

Using the following data, we will write a program for single-point thread milling procedures:

DATA:

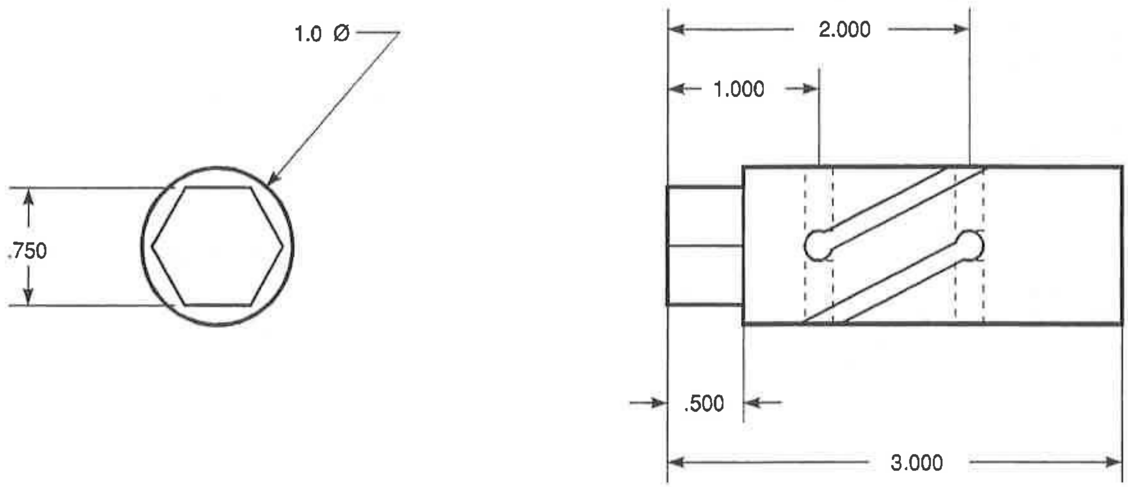
- 2.500 Ø hole
- Diameter of cutter (Subtract .750): 1.75
- Radial value (Divide by 2): .875
- Thread pitch: .0833 (12 TPI)
- Part thickness: 1.00

```
%
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
%
```

2.15 FOURTH AXIS PROGRAMMING

The following is an example of fourth axis programming. Refer to Figure 2-13 for the program code on page 40.



EIGHT (8) HOLES CHAMFERED (.2 Ø)

EIGHT (8) HOLES .1875 Ø, 90° SPACING, .4 DP.

FOUR (4) CHANNELS, .125 WIDE

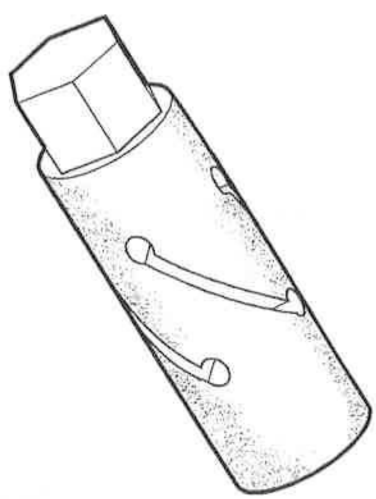


FIG. 2-13 *Fourth axis programming exercise.*

PROGRAM CODE FOR FIG. 2-19:

```
%
O1234                                    (Fourth axis program using a Haas Servo 5C)
(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                                  (Tool #1 is a .500 end mill to mill hex.)
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                                  (Tool #2 is a .375 Ø NC spot drill.)
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.
X1.0
G00  G80  Z.1  M09
T3  M06                                  (Tool #3 is a .1875 Ø stub twist drill.)
G00  G90  G54  X1.0  Y0  A0  S5000  M03
G43  H03  Z.1  M08
G83  Z-1.125  F12.  R.1  Q.25
X2.0
A90.
X1.0
G00  G80  Z.1  M09
T4  M06                                  (Tool #4 is a .125 Ø end mill.)
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                                    (Subprogram to mill hex.)
G01  Z-.125  F50.
Y.5  F35.
G00  Z.1
G91  Y-1.0  A60.
G90
M99
%
```

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
%
```

2.16 FORMULAS

TAPPING -

STANDARD thread formula:

Revolutions per minute (RPM) divided by threads per inch (TPI) = Feed rate in inches per minute

RPM/TPI = F

METRIC thread formula:

Pitch (P) multiplied by .03937 = ____ multiplied by RPM = Feed rate in inches per minute

(P x .03937) x RPM = F

SPEED AND FEEDS -

S.F.M. (Surface Feet per Minute):

.262 multiplied by the cutter diameter multiplied by the RPM = SFM

.262 x Cutter Diameter x RPM = SFM

R.P.M. (Revolutions Per Minute):

3.82 multiplied by the recommended SFM divided by the cutter diameter = RPM

(3.82 x SFM) / Cutter Diameter = 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.

(Feed/tooth x n) x RPM = 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 Diameter x d) x IPM = CIPM

II. PROGRAMMING

1. INTRODUCTION

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.

2. PROGRAM STRUCTURE

2.1 THE PARTS OF A PROGRAM

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 ½ 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                                 :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 ½ 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 **Onnnn** 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.

2.2 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 50. 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 Not used

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 150. 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 Section 3 for a detailed list of **G** codes.

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 50. 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 OFSET 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 -838.0000 and 838.0000 for inches or between -8380.000 and 8380.000 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 Section 9 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 and M98 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 **Onnnn** identification in the first block; it cannot be deleted. Altering the **O** in the first block causes the program to be renamed. An **Onnnn** 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 "**O**".

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 always a positive number in inches between 0 and 100.0.

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 -838.0000 and 838.0000 for inches or between -83800.000 and 8380.000 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 -838.0000 and 838.0000 for inches or between -8380.000 and 8380.000 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 -838.0000 and 838.0000 for inches or between -8380.000 and 8380.000 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 -838.0000 and 838.0000 for inches or between -8380.000 and 8380.000 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 **G** codes summary. A " * " indicates the default within each group, if there is one:

Code:	Group:	Function:	Description On Page:
G00	*01	Rapid Motion	51
G01	01	Linear Interpolation Motion	51
G02	01	CW Interpolation Motion	51
G03	01	CCW Interpolation Motion	52
G04	00	Dwell	53
G09	00	Exact Stop	53
G10	00	Programmable Offset Setting	53
G12	00	CW Circular Pock Milling (Yasnac)	54
G13	00	CCW Circular Pock Milling (Yasnac)	55
G17	*02	XY Plane Selection	56
G18	02	ZX Plane Selection	56
G19	02	YZ Plane Selection	56
G20	06	Inch programming selection	88
G21	06	Metric programming selection	88
G28	00	Return To Reference Point	56
G29	00	Set Return Reference Point	56
G31	00	Skip Function	56
G35	00	Automatic Tool Diameter Measurement	57
G36	00	Automatic Work Offset Measurement	57
G37	00	Automatic Tool Length Measurement	57
G40	*07	Cutter Comp Cancel	58
G41	07	Cutter Compensation Left	58
G42	07	Cutter Compensation Right	58
G43	08	Tool Length Compensation +	58
G44	08	Tool Length Compensation -	59
G49	*08	G43/G44 Cancel	59
G50	11	G51 Cancel	59
G51	11	Scaling	59
G52	12	Select work Coordinate System G52 (Yasnac)	63
G52	00	Set Local Coordinate System (Fanuc)	63
G53	00	Non-Modal Machine Coordinate Selection	63
G54	*12	Select Work Coordinate System 1	63
G55	12	Select Work Coordinate System 2	63
G56	12	Select Work Coordinate System 3	63
G57	12	Select Work Coordinate System 4	63
G58	12	Select Work Coordinate System 5	63
G59	12	Select Work Coordinate System 6	63
G60	00	Unidirectional Positioning	63
G61	13	Exact Stop Modal	64
G64	*13	G61 Cancel	64
G65	00	Macro Subroutine Call	112
G68	16	Rotation	60
G69	16	G68 Cancel	63
G70	00	Bolt Hole Circle (Yasnac)	64
G71	00	Bolt Hole Arc (Yasnac)	64
G72	00	Bolt Holes Along an Angle (Yasnac)	65
G73	09	High Speed Peck Drill Canned Cycle	68
G74	09	Reverse Tap Canned Cycle	70
G76	09	Fine Boring Canned Cycle	70
G77	09	Back Bore Canned Cycle	72
G80	*09	Canned Cycle Cancel	73

G81	09	Drill Canned Cycle	73
G82	09	Spot Drill Canned Cycle	74
G83	09	Peck Drill Canned Cycle	75
G84	09	Tapping Canned Cycle	77
G85	09	Boring Canned Cycle	78
G86	09	Bore/Stop Canned Cycle	79
G87	09	Bore/Manual Retract Canned Cycle	80
G88	09	Bore/Dwell Canned Cycle	81
G89	09	Bore Canned Cycle	82
G90	*03	Absolute	83
G91	03	Incremental	83
G92	00	Set Work Coordinates	83
G98	*10	Initial Point Return	83
G99	10	Plane Return	83
G100	00	Disable Mirror Image	84
G101	00	Enable Mirror Image	84
G102	00	Programmable Output To RS-232	84
G103	00	Block Lookahead Limit	85
G110	12	Select Work Coordinate System 7	85
G111	12	Select Work Coordinate System 8	85
G112	12	Select Work Coordinate System 9	85
G113	12	Select Work Coordinate System 10	85
G114	12	Select Work Coordinate System 11	85
G115	12	Select Work Coordinate System 12	85
G116	12	Select Work Coordinate System 13	85
G117	12	Select Work Coordinate System 14	85
G118	12	Select Work Coordinate System 15	85
G119	12	Select Work Coordinate System 16	85
G120	12	Select Work Coordinate System 17	85
G121	12	Select Work Coordinate System 18	85
G122	12	Select Work Coordinate System 19	85
G123	12	Select Work Coordinate System 20	85
G124	12	Select Work Coordinate System 21	85
G125	12	Select Work Coordinate System 22	85
G126	12	Select Work Coordinate System 23	85
G127	12	Select Work Coordinate System 24	85
G128	12	Select Work Coordinate System 25	85
G129	12	Select Work Coordinate System 26	85
G136	00	Automatic Work Offset Center Measurement	57
G150	00	General Purpose Pocket Milling	86

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

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 thru 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 operator buttons.

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 block 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

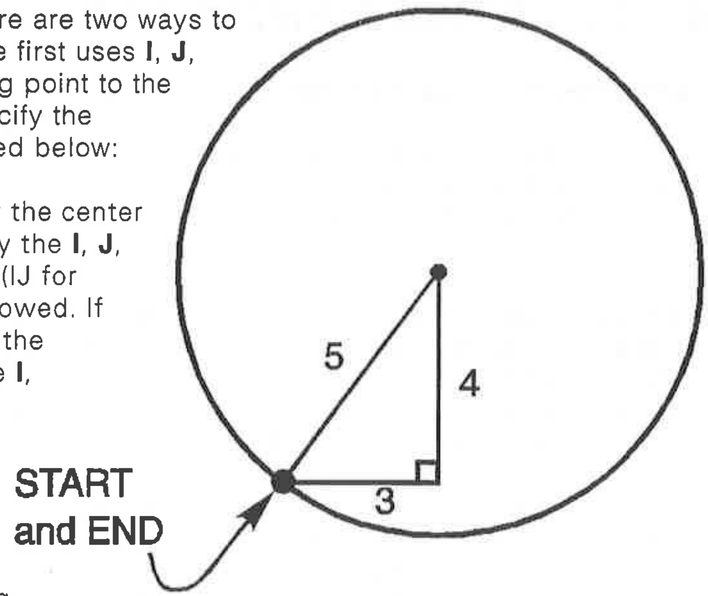
F Feed rate in inches (mm) per minute
X Optional X-axis motion command
Y Optional Y-axis motion command
Z Optional Z-axis motion command
A Optional A axis motion command

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 speeds of all axes are 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:

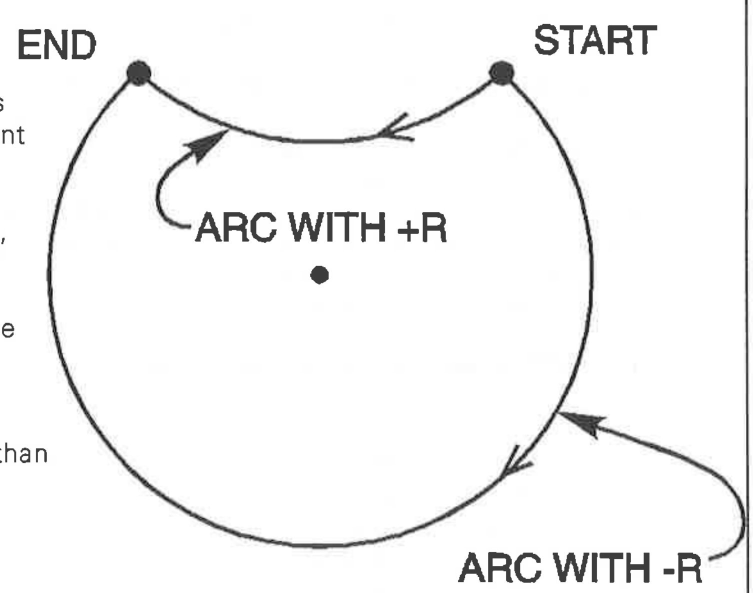


I, J, K: When **I**, **J**, or **K** are used to specify the center of the arc, **R** may not be used. Only the **I**, **J**, or **K** specific to the selected plane (IJ for G17, IK for G18, JK for G19) are allowed. If only one of the **I**, **J**, **K** is specified, the others are assumed to be zero. 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°)*b2W 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.

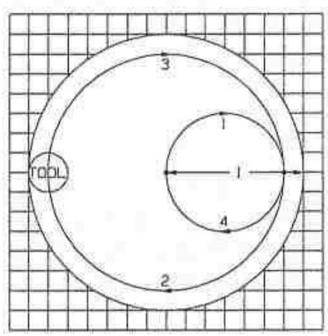
G13 Circular Pocket Milling Counterclockwise Group 00

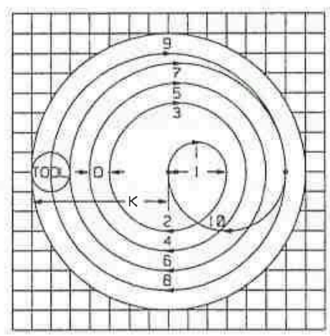
This **G** Code implies the use of G41 and is otherwise similar to G12. G13 belongs to Group zero and thus is non-modal.

```
%
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

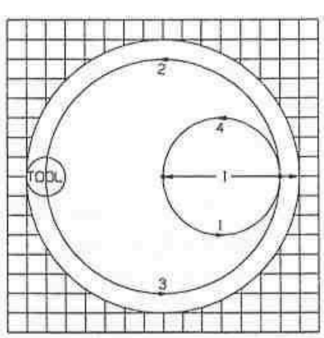


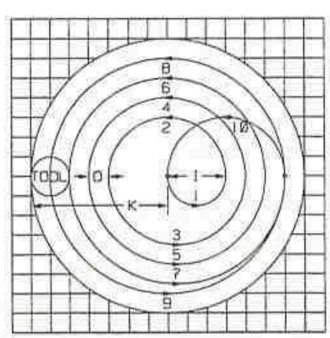


I ONLY

I, K, AND Q

G13





3.6 CIRCULAR PLANE SELECTION

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.

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.

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.

3.7 REFERENCE POINT DEFINITION AND RETURN

G28 Return To 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** code is specified on the same block, only those axes will be moved and they will be moved to the specified positions in the current coordinate system and then they will be moved to machine zero. The intermediate position, if specified, is saved for use in the 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 will also cancel tool length offsets.

G29 Set Return Reference Point Group 00

The G29 code is used to move the axes to a position via a previously-set reference point. The reference is defined with the G29. This command is normally given with the axes positioned at machine zero. The axes that are selected in this block are moved first to the intermediate reference point and then they are moved to the **X**, **Y**, **Z**, or **A** specified. The positions are interpreted in the current coordinate system.

3.8 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.

3.9 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..H50) 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.

3.10 AUTOMATIC WORK OFFSET MEASUREMENT (G36,G136)

G36	**Automatic Work Offset Measurement**	Group 00
G136	**Automatic Work Offset Center Measurement**	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 set into 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.

3.11 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.

See Section 10 for a complete description of cutter compensation operation.

3.12 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 OFSET 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.

3.13 COORDINATE ROTATION AND SCALING

This control function is optional. If you wold 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.

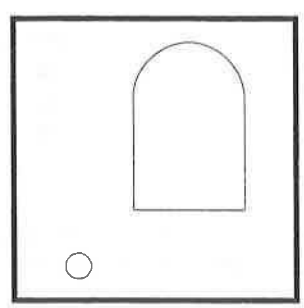
G51 is modal and modifies appropriate positional values in the blocks following the G51 command. It does not change or modify values in the block from which it is called. Axes X, Y, and Z are all scaled when the P code is used. If the P code is not used, the scaling factor currently in Setting 71 is 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.

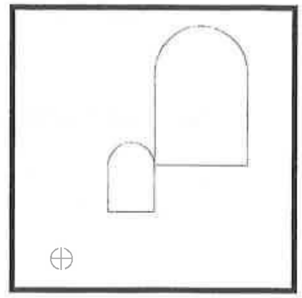
```
O0001 (GOTHIC WINDOW) ;
F20. S500 ;
G00 X1. Y1. ;
G01 X2. ;
Y2. ;
G03 X1. R0.5
G01 Y1. ;
G00 X0 Y0 ;
M99 ;
```

O = Work coordinate origin
+ = Center of scaling



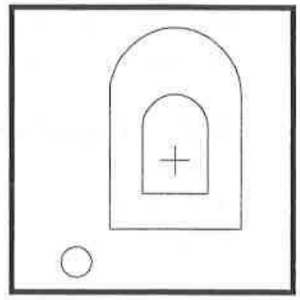
The first example illustrates how the control uses the current work coordinate location as a scaling center. Here, it is X0 Y0 Z0.

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



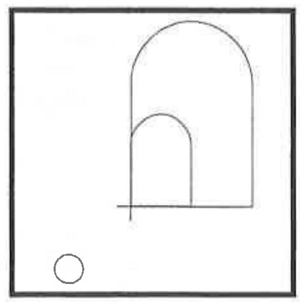
The next example specifies the center of the window as the scaling center.

```
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 I G18 I 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.

On the previous page, '**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.

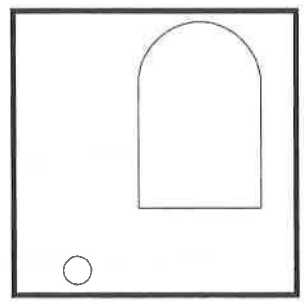
G68 is modal and modifies appropriate positional values in the blocks following the G68 command. Values in the block containing G68 are not rotated. For subsequent blocks, only the values in the plane of rotation are rotated. Thus, if G17 is the current plane of rotation, only X and Y values are affected.

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.

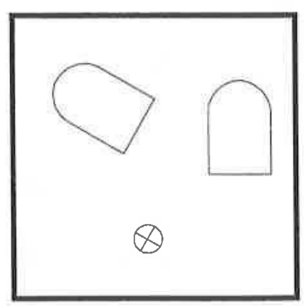
The following examples illustrate rotation using G68.



```
O0001 (GOTHIC WINDOW) ;
F20. S500 ;
G00 X1. Y1. ;
G01 X2. ;
Y2. ;
G03 X1. R0.5 ;
G01 Y1. ;
M99 ;
```

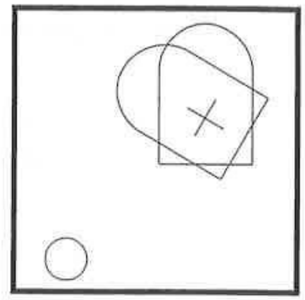
O = Work coordinate origin
+ = Center of rotation

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



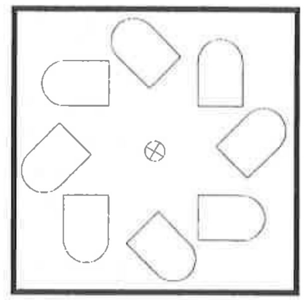
```
O0002 ;
G59 ;
G00 G90 X0 Y0 Z0 ;
M98 P1 ;
G90 G00 X0 Y0 ;
G68 R60. ;
M98 P1 ;
G69 G90 G00 X0 Y0 ;
M30 ;
```

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

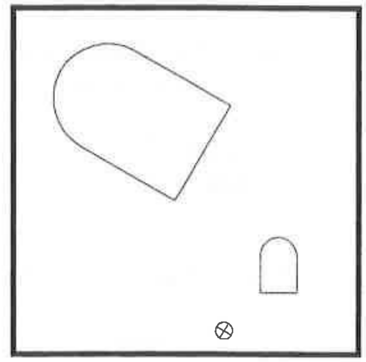
If scaling and rotation is used simultaneously, it is recommended that scaling is turned on prior to rotation, and that separate blocks be used. Use the following template when doing this.

```
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.

3.14 WORK COORDINATE SYSTEM SELECTION

Note: The G52 command works differently depending on the value of Setting 33. That setting selects either FANUC style of coordinates or YASNAC style of coordinates. They are both listed here:

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.

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.

3.15 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.

3.16 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 (see 3.12 below).

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 (+ and -)
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

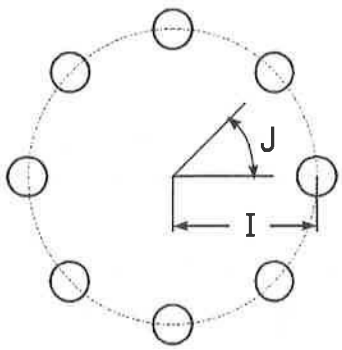
Group 00

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.

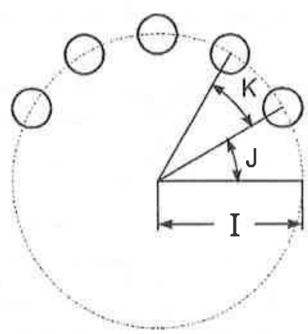
See chapter 2.7 of the general information section for examples of bolt hole pattern canned cycles.

G70
BOLT HOLE CIRCLE



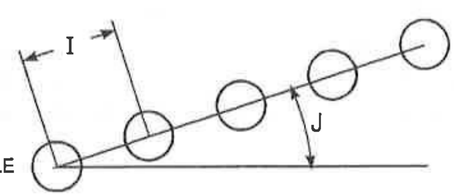
I = Radius of the bolt circle
J = Starting angle from three o'clock position
L = Number of holes (evenly spaced)

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

3.17 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:

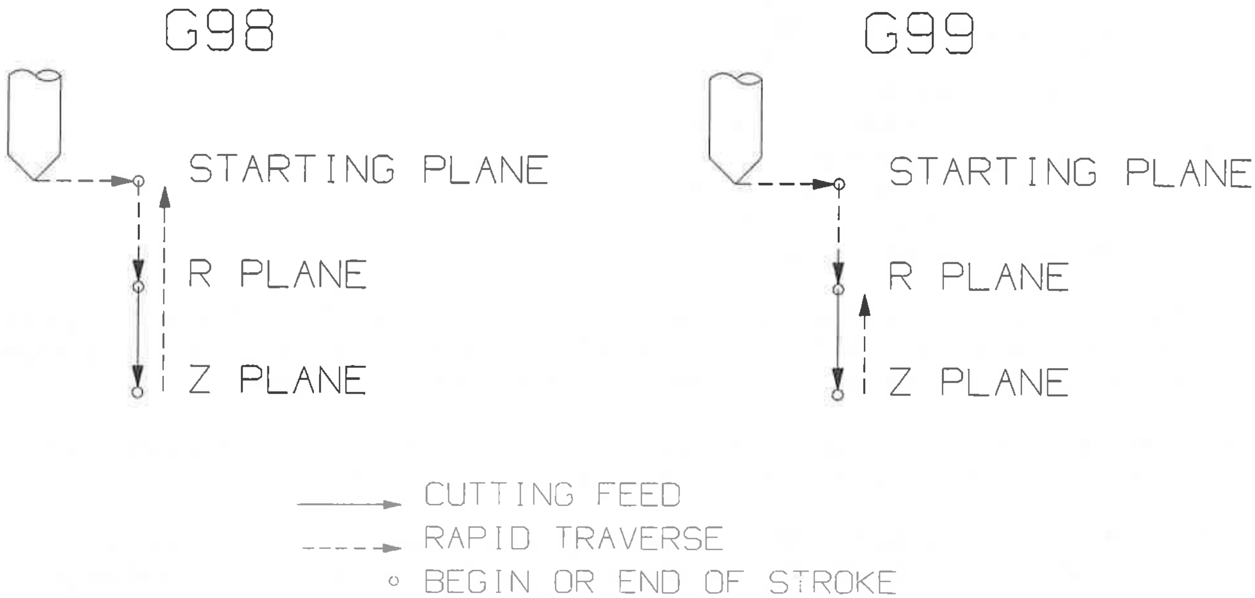
G code	Z Drilling direction	Operation at bottom of hole	Retraction Z direction	Application
G73 feed	intermittent	none	rapid	high speed peck drilling
G74	feed	spindle CW	feed	left hand tapping
G76	feed then stop	orient spindle	rapid	fine boring
G81	feed	none	rapid	spot drilling
G82	feed	dwell	rapid	counter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

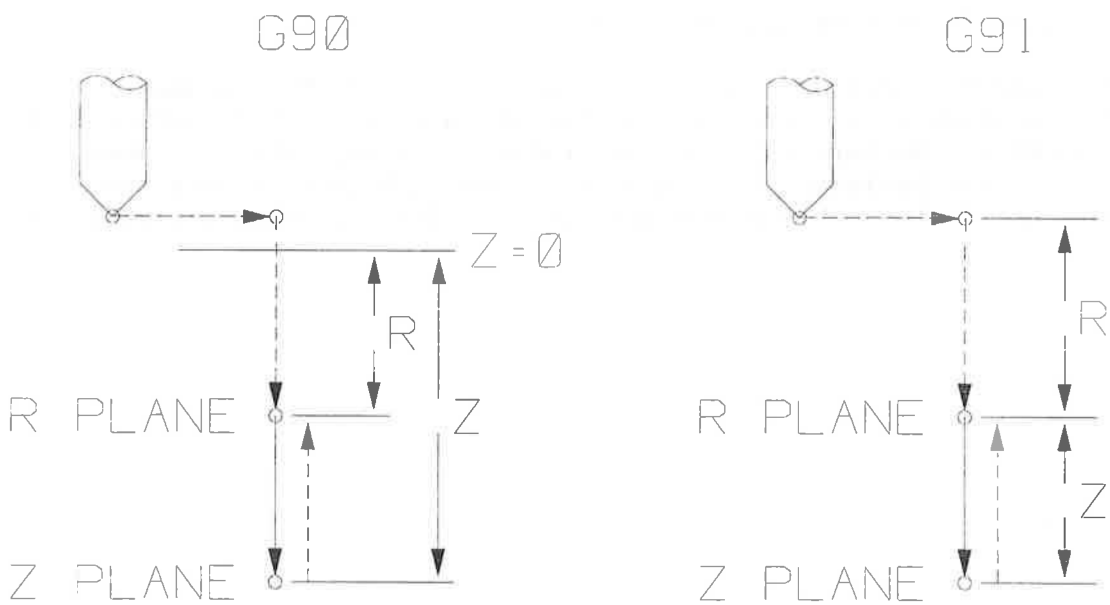
G98 and G99 are modal commands that change the way the canned cycles operate. When G98 is active, the Z-axis will be returned to the same position as at the start of the canned cycle when it completes. When G99 is active, the Z-axis will be returned to the **R** point when the canned cycle completes.

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.

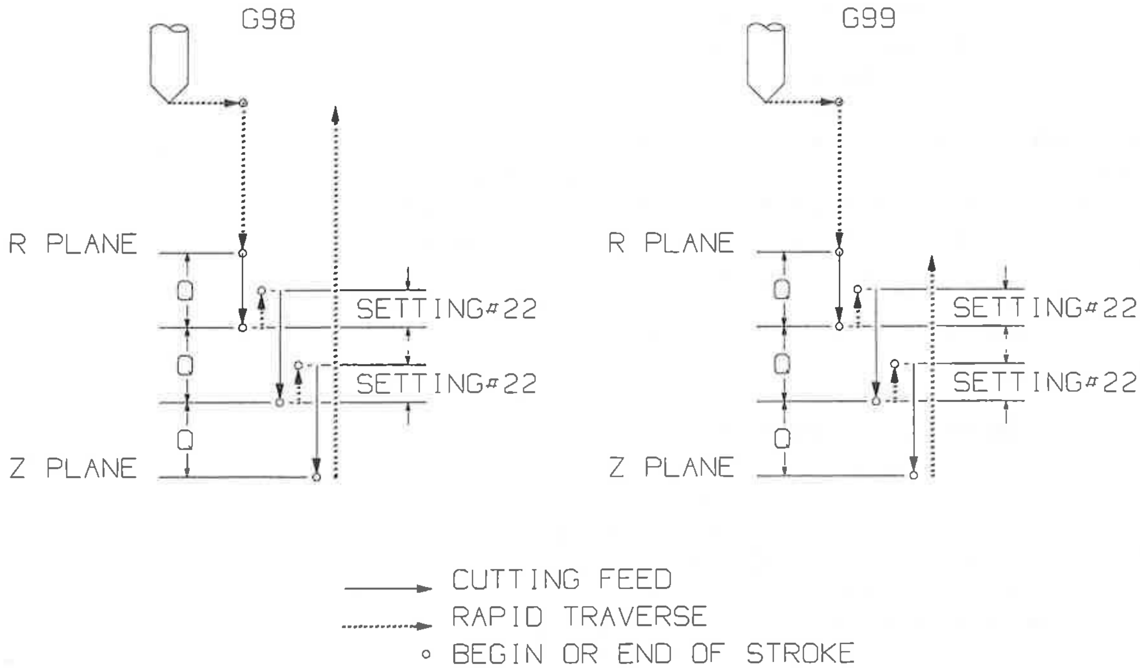
If **I**, **J**, and **K** are specified, a different operating mode is selected. The first pass will cut in by **I**, each succeeding cut will be reduced by amount **J**, and the minimum cutting depth is **K**.

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.

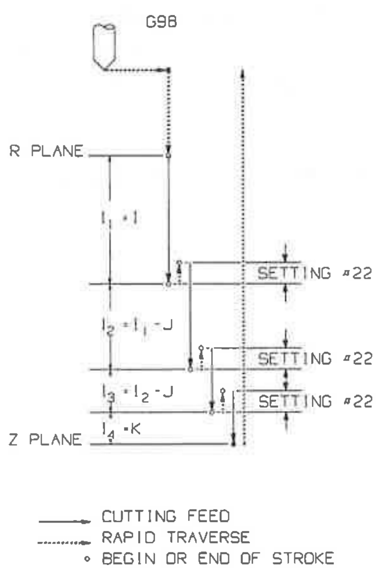
I, **J**, **K**, and **Q** are always positive numbers.

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.

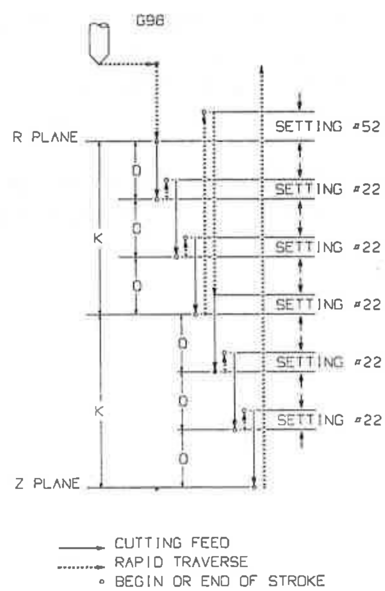
G73 PECK DRILLING CANNED CYCLE



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



G73 PECK DRILLING CANNED CYCLE
WITH K AND Q OPTIONS



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.

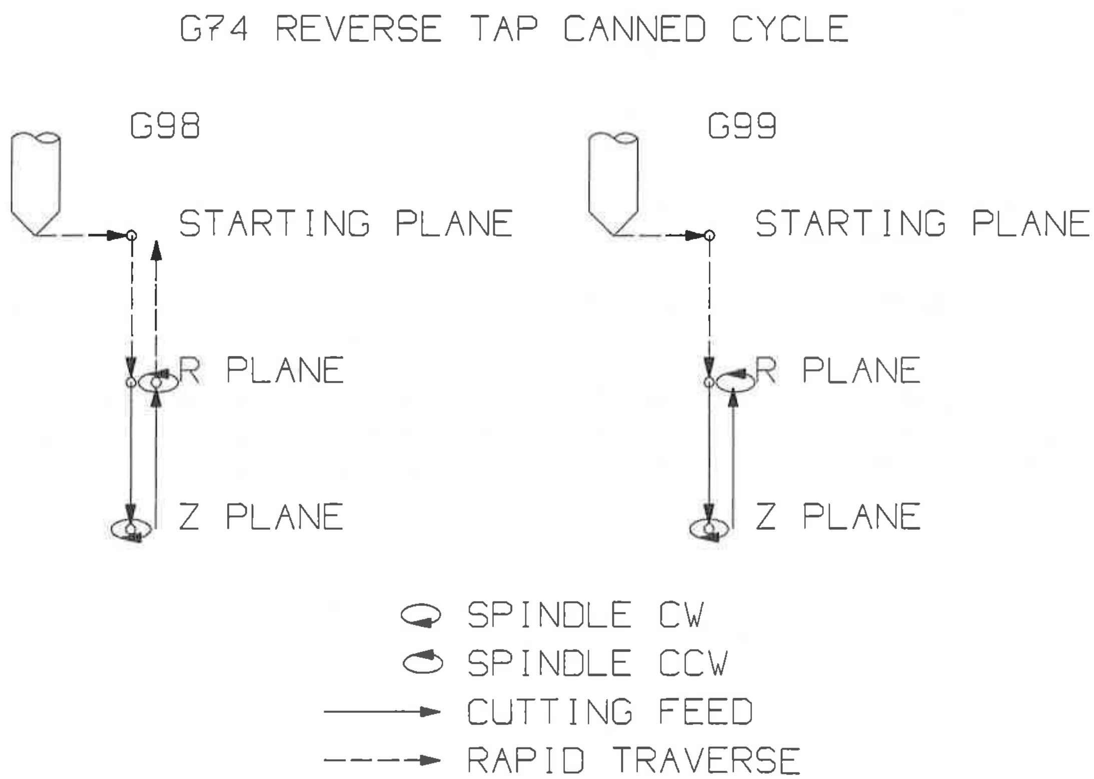
G76 Fine Boring Canned Cycle

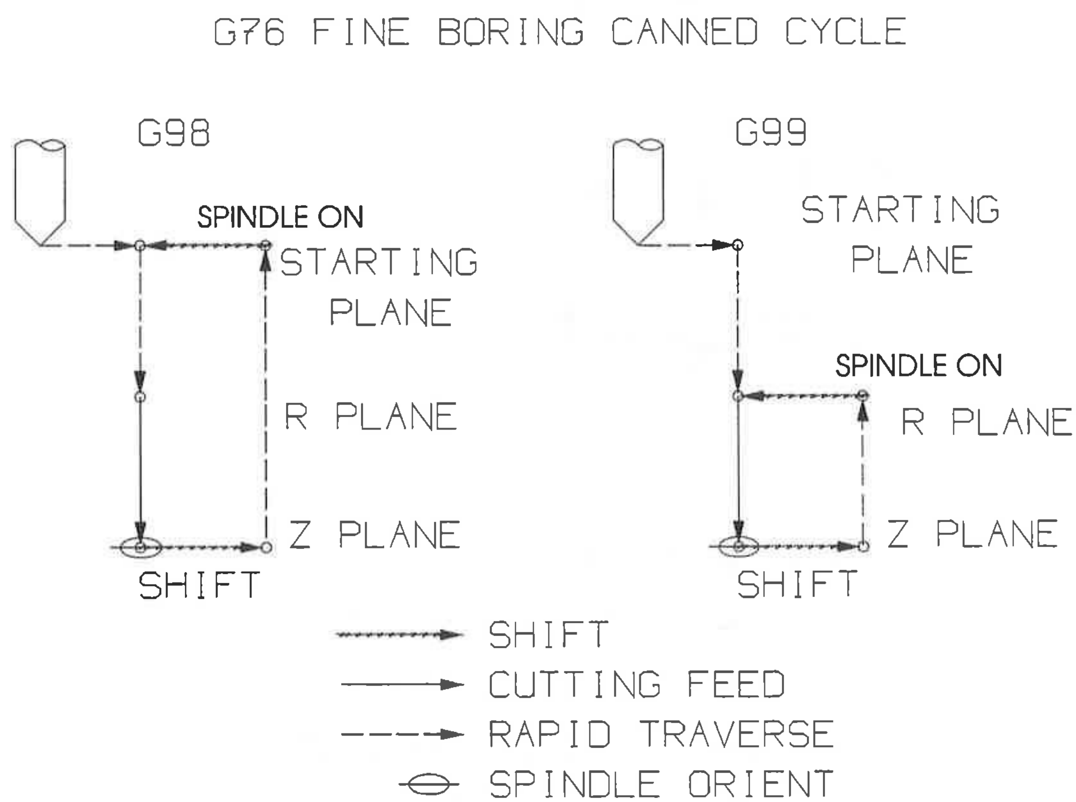
Group 09

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.


G74 REVERSE TAP CANNED CYCLE
G98
STARTING PLANE
R PLANE
Z PLANE
G99
STARTING PLANE
R PLANE
Z PLANE
SPINDLE CW
SPINDLE CCW
CUTTING FEED
RAPID TRAVERSE

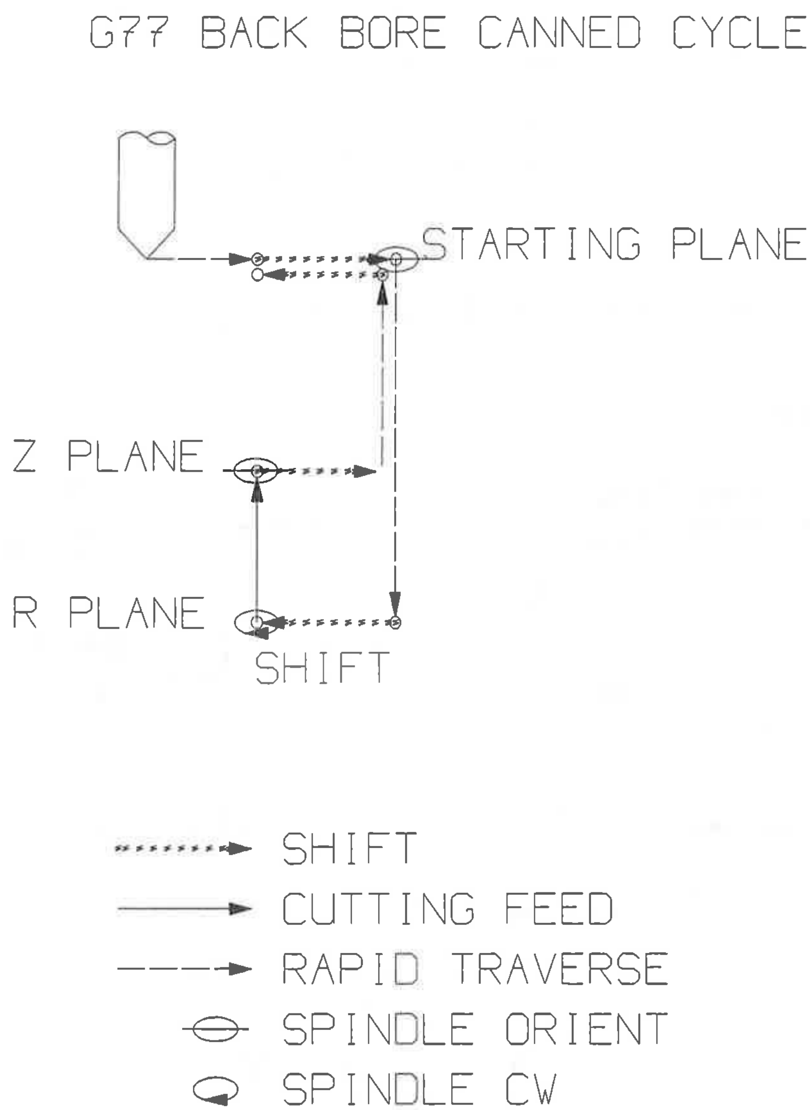

G76 FINE BORING CANNED CYCLE
G98
SPINDLE ON
STARTING
PLANE
R PLANE
Z PLANE
SHIFT
G99
STARTING
PLANE
SPINDLE ON
R PLANE
Z PLANE
SHIFT
SHIFT
CUTTING FEED
RAPID TRAVERSE
SPINDLE ORIENT

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 clead 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.



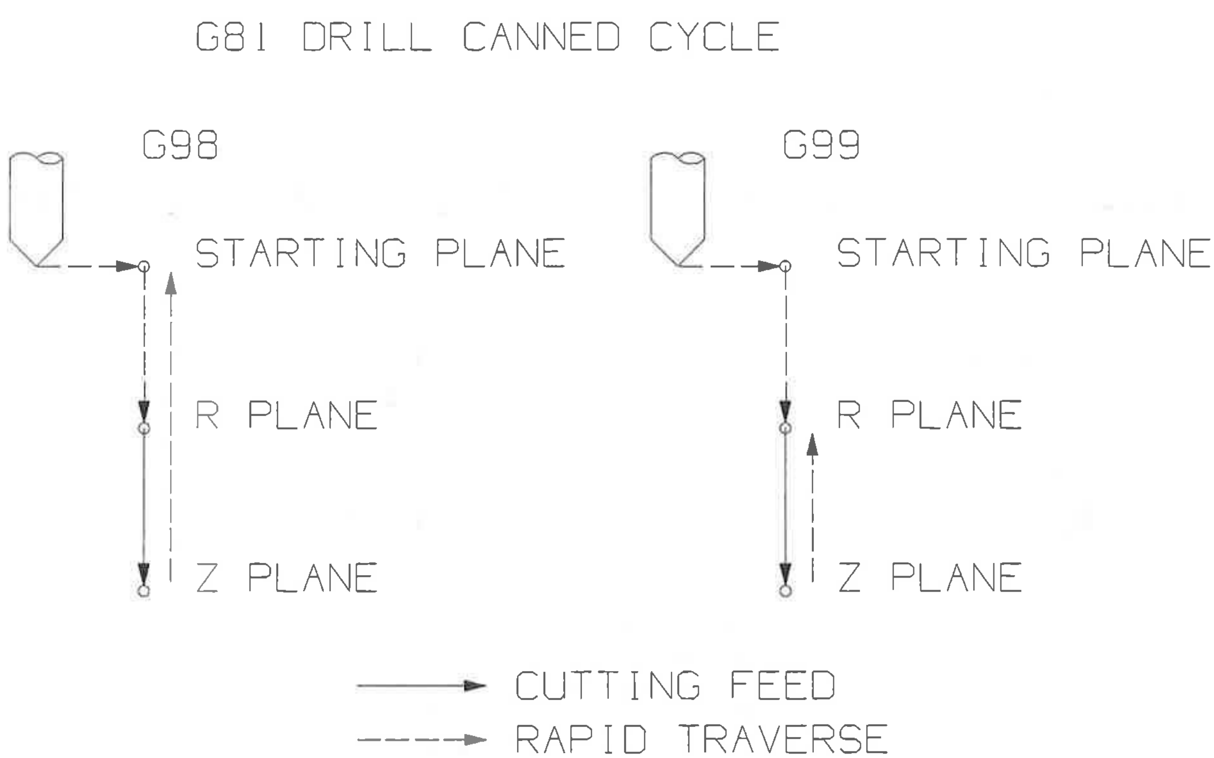
G80 Canned Cycle Cancel — Group 09

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.

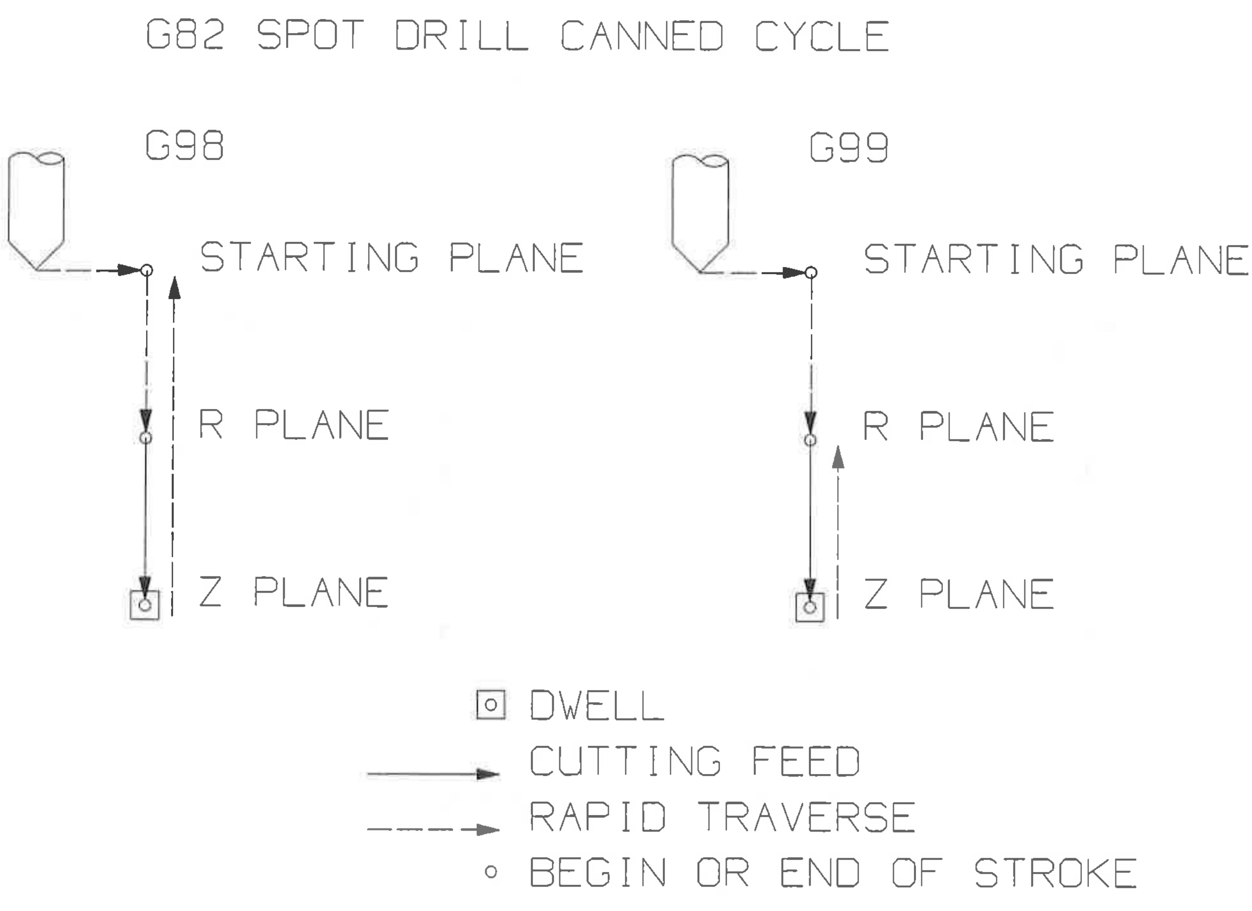


G82 Spot Drill Canned Cycle

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.



G83 Peck Drilling Canned Cycle

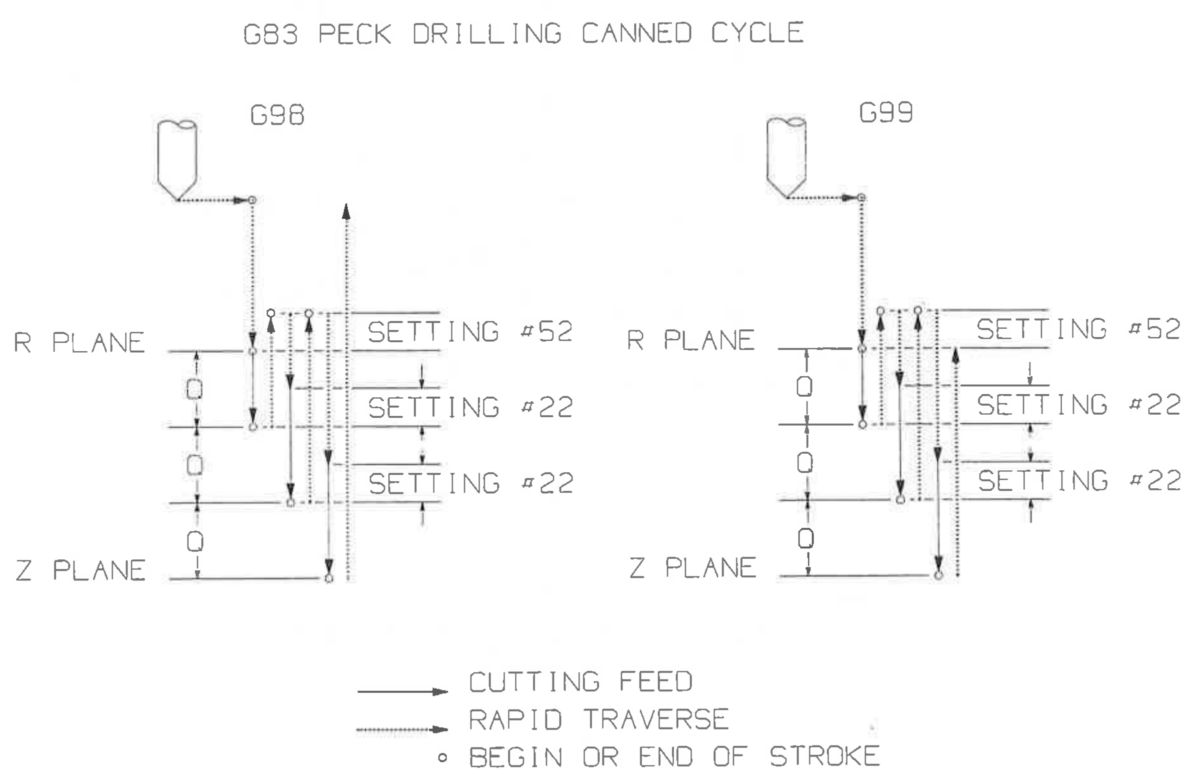
Group 09

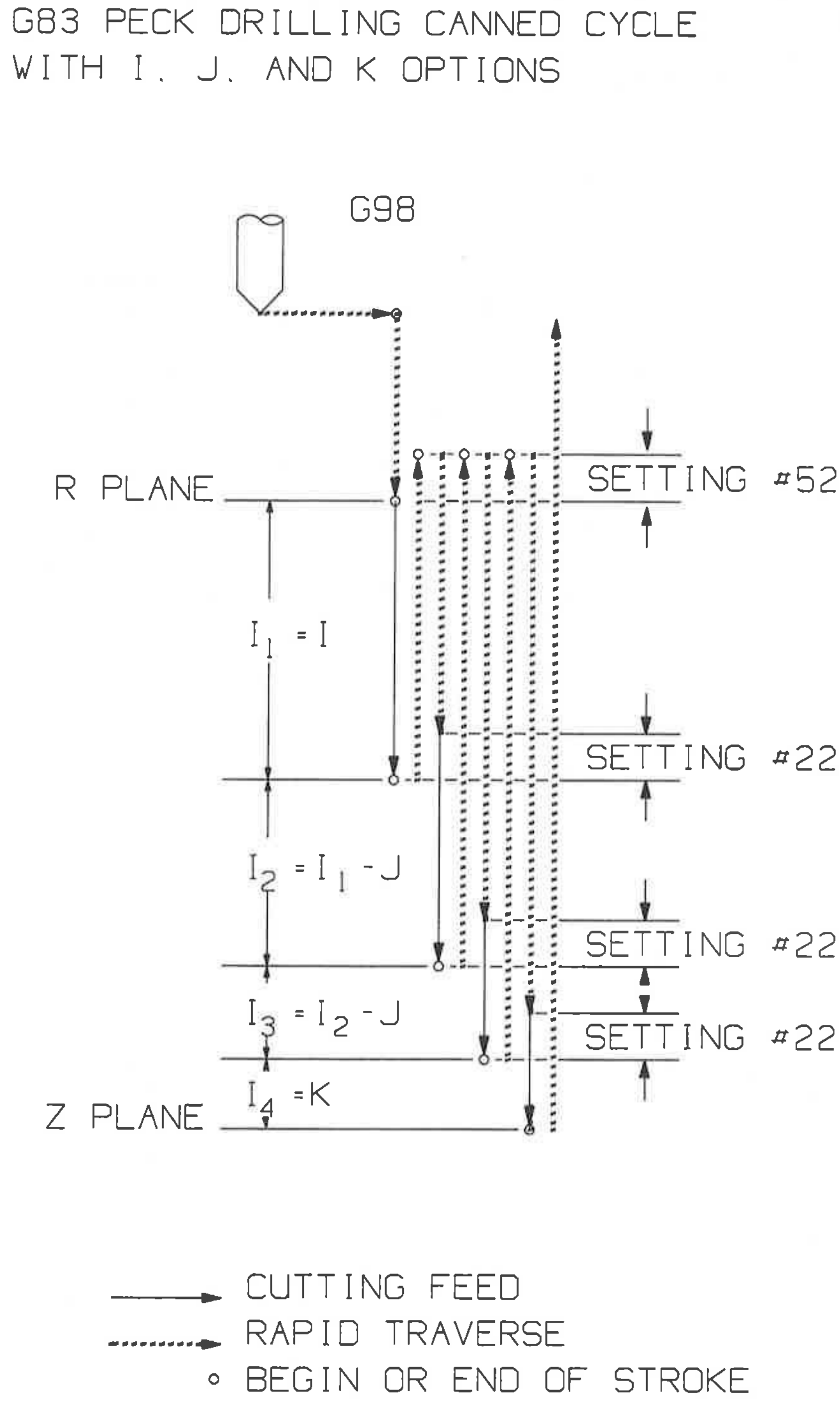
F	Feed Rate in inches (mm) per minute
I	Optional size of first cutting depth
J	Optional amount to reduce cutting depth each pass
K	Optional minimum depth of cut
L	Number of repeats
Q	The cut-in value, always incremental
R	Position of the R plane
X	Optional X-axis motion command
Y	Optional Y-axis motion 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
WITH I, J, AND K OPTIONS
G98
R PLANE
SETTING #52
I1 = I
SETTING #22
I2 = I1 - J
SETTING #22
I3 = I2 - J
SETTING #22
I4 = K
Z PLANE
CUTTING FEED
RAPID TRAVERSE
BEGIN OR END OF STROKE

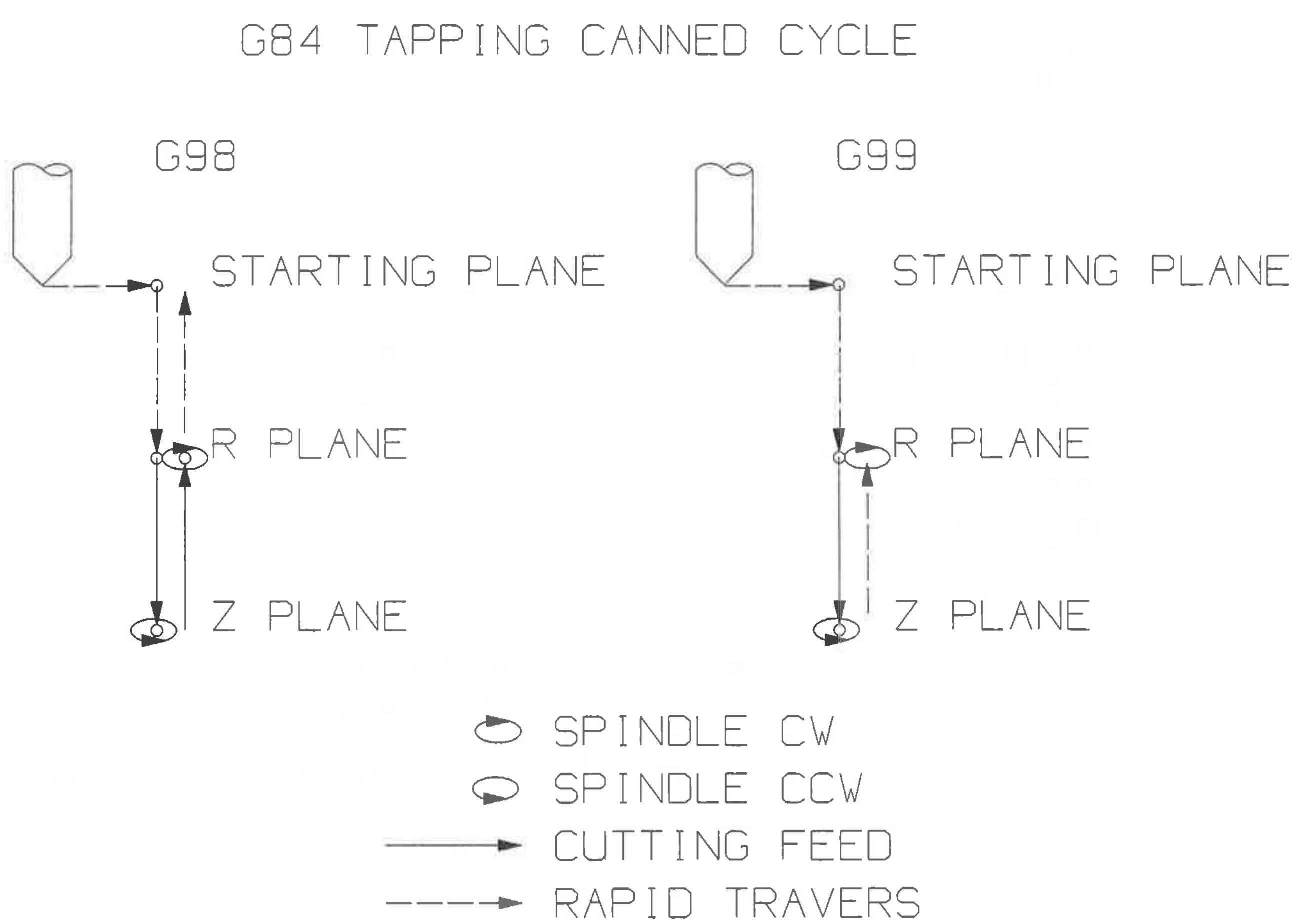
G84 Tapping Canned Cycle

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.

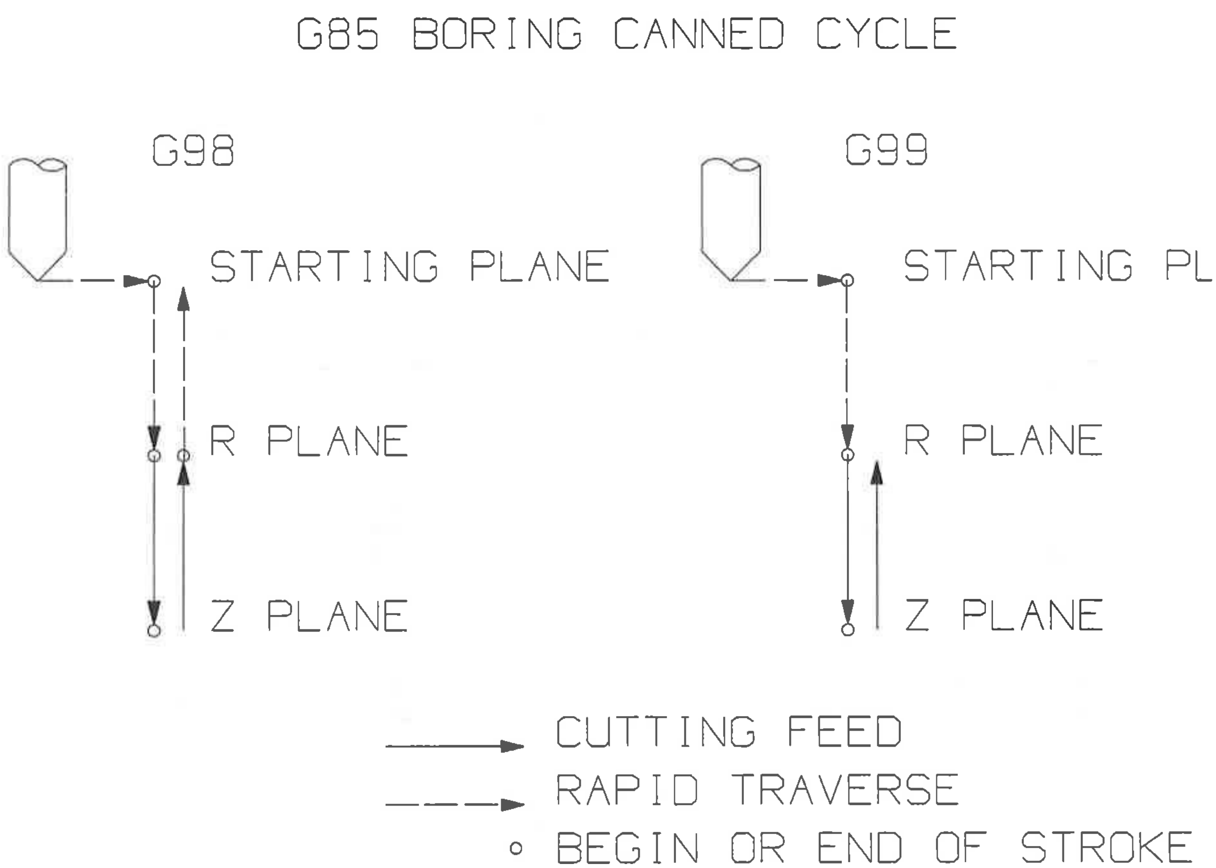
You do not need to start the spindle CW before this canned cycle. The control does this automatically.



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

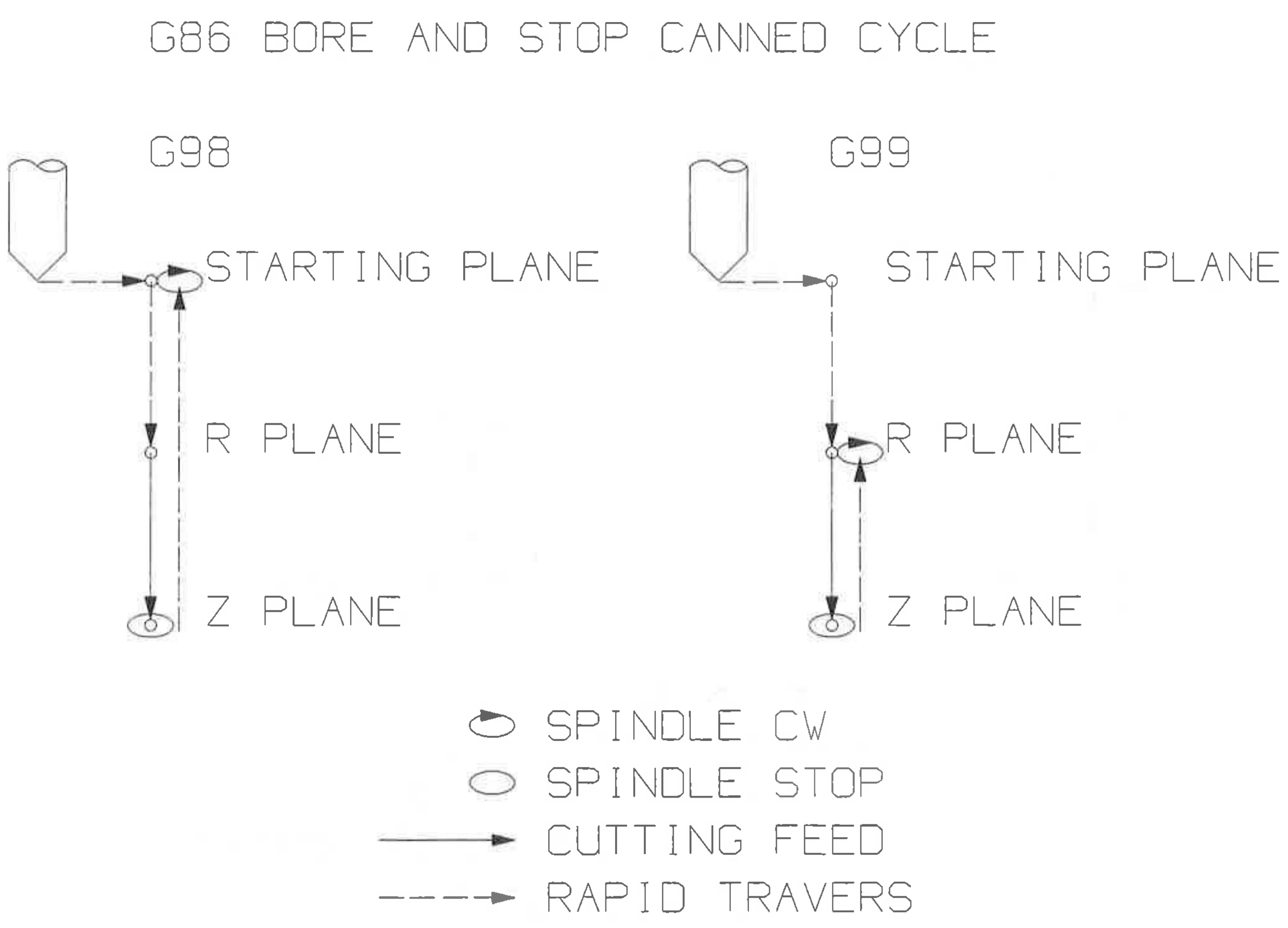
This **G** code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of **X** or **Y** will cause this canned cycle to be executed.



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.

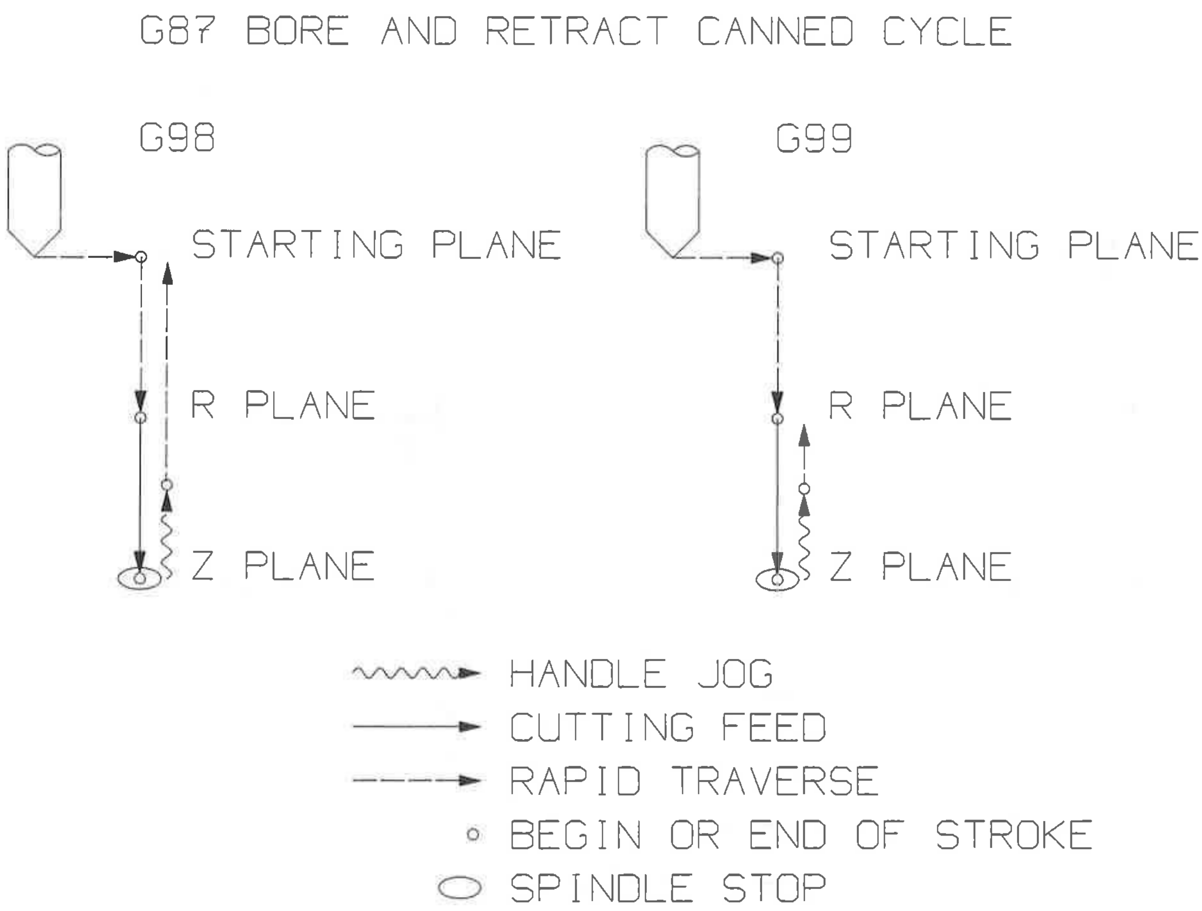


G87 Bore and Manual Retract Canned Cycle

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.

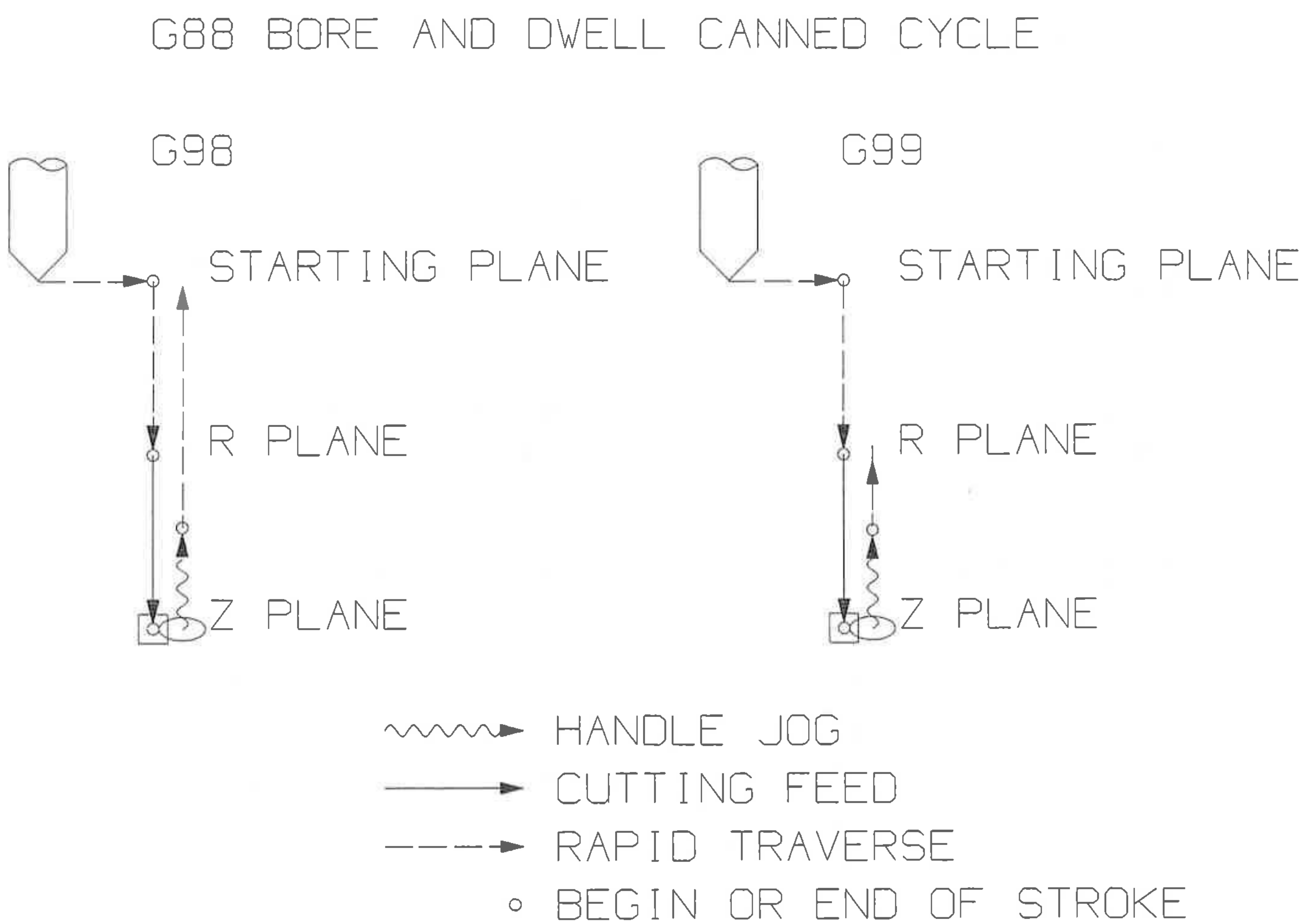


G88 Bore and Dwell Canned Cycle

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.

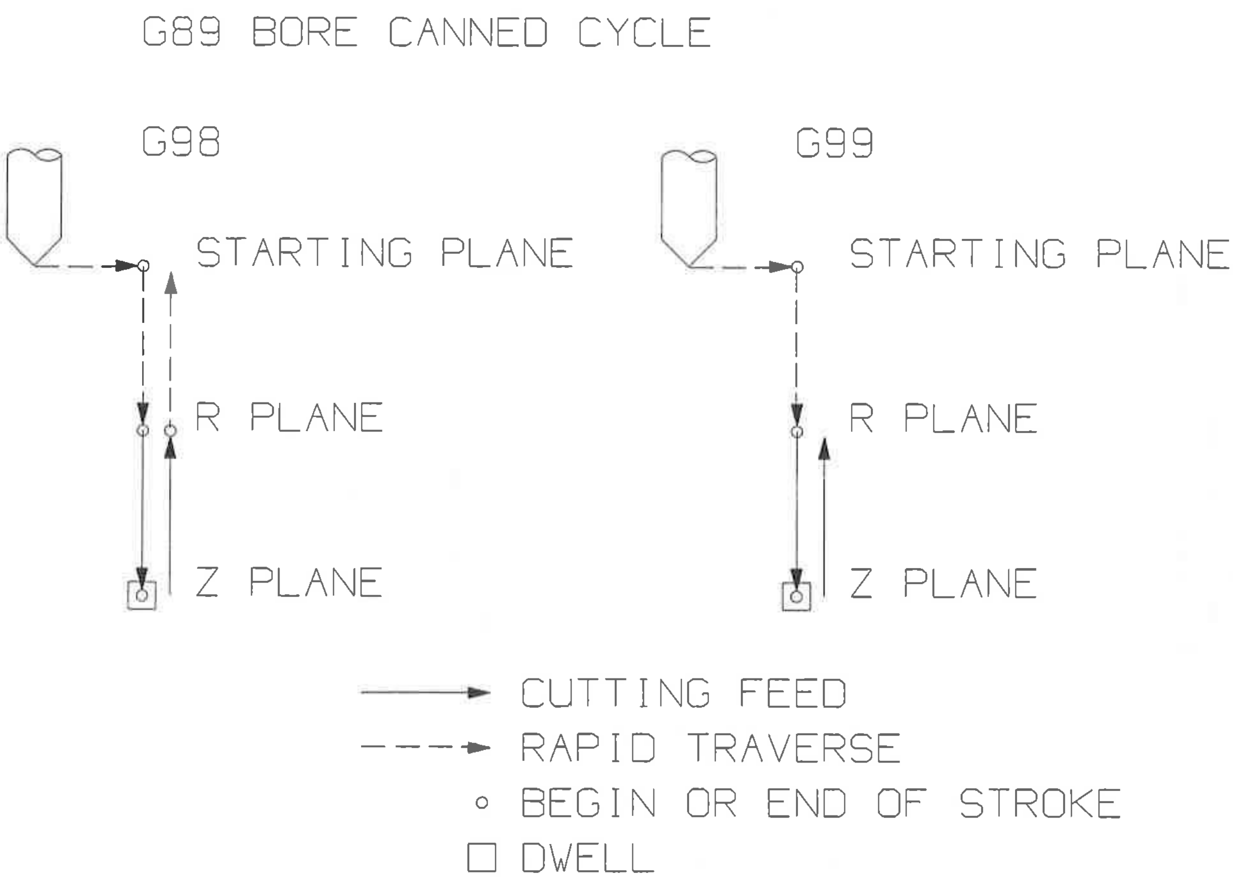


G89 Bore Canned Cycle

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.



3.18 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.

3.19 MORE WORK COORDINATE SELECTION

This command works differently depending on the value of Setting 33. That setting selects either FANUC style of coordinates 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 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.

3.20 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.

3.21 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.

3.22 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.

3.23 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.

3.23 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.

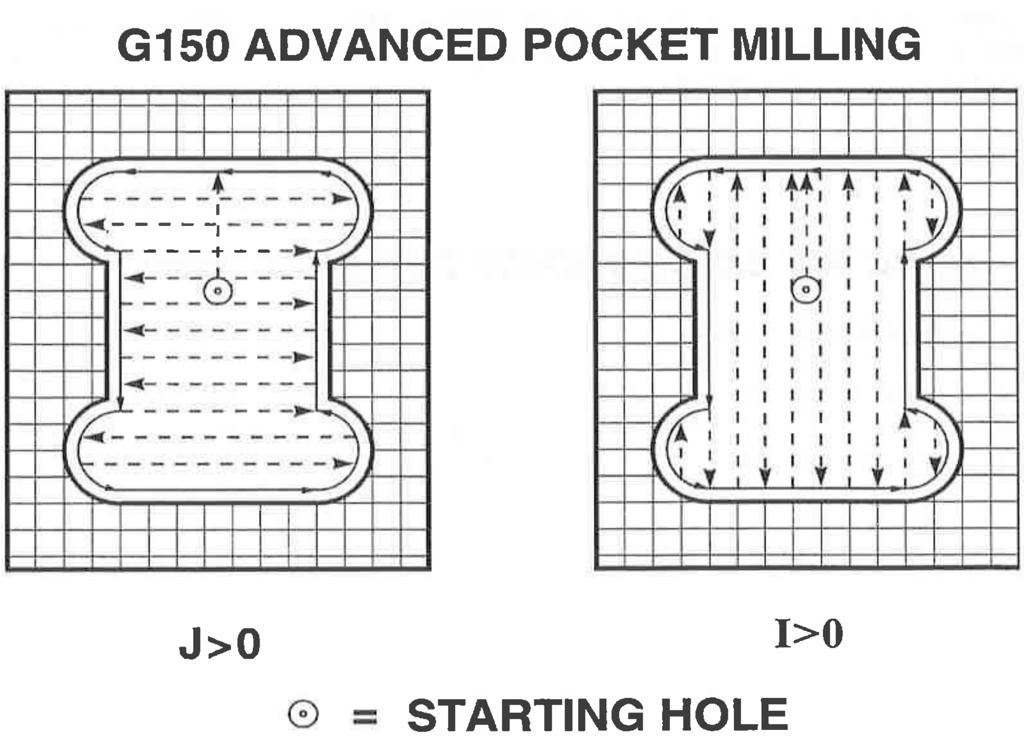
3.24 GENERAL PURPOSE POCKET MILLING FUNCTION

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
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 X-axis. If **J** is used, the pocket is cut from a series of strokes in the Y-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.

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 following 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**.

```
O0100 (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

O0200 (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)
```

4. 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.

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. 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.

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.

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.

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.

5. INCH / METRIC SELECTION (G20, G21)

Selection between inch and metric programming can only be done from the Setting page, 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 insure that the inch/metric setting is set correctly for that program.

6. 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. All motion from machine zero is in a negative direction. 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.

There are two work coordinate systems that can be used. A G92 code followed by an **X** and **Y** value and a Z0 on the first line of the program is the most common. This first line then tells the control the distance from part zero to machine home. A better way is to use the G54 through G59 or G110 through G129 work offset. They do the same thing that a G92 code does except that the G54 value is located outside the program in the offset display.

The advantages of the G54 are: 1) on power up, the machine automatically establishes its coordinate system; 2) the operator does not need to edit the program; 3) a crash cannot occur when the program is restarted at a point other than home as can in a G92 which establishes a new reference point each time it is read; 4) there are twenty six offsets available; 5) you can load and save offsets just as you load and save programs.

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. Normally the Z-axis work coordinate is zero because the offset is supplied by the tool offset number (**T** command). An exception to this would be if you wish to raise the tool above **Z** zero for program setup. By putting a Z-1.0 into the **Z** work coordinate that tool will start one inch above **Z** zero on the part. If you wish to raise all the tools, enter a Z-1.0 into the G52 work coordinate. The G52 work coordinate shifts all work coordinates by the amount entered but applies only when using the Fanuc type of coordinate system (Setting 33).

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.

Offsets can be sent and received with the RS-232 port. See Chapter III, Section 13 for a description of how to do this.

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 Section 14.1 for further details.

8. 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 AUTO POWER UP 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. 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

9. MISCELLANEOUS FUNCTIONS (M FUNCTIONS)

9.1 M CODE SUMMARY

Only one † code may be programmed per block of a program. All † codes are effective or cause an action to occur at the end of the block and only one † 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
M16	Tool Change (same as M06)
M19	Orient Spindle
M21-M28	Optional Pulsed User M Function with Fin
M27	Apply fifth axis brake, wait until M-fin signal is received through the secondary RS-232 port.
M30	Prog End and Rewind
M31	Chip Conveyor Forward
M32	Chip Conveyor Backward
M33	Chip Conveyor Stop
M34	Increment Coolant Spigot Position
M35	Decrement Coolant Spigot Position
M39	Rotate Tool Turret
M41	Low Gear Override
M42	High Gear Override
M51-M58	Optional User M turn ON
M57	Apply fifth axis brake, continue with program.
M61-M68	Optional User M turn OFF
M67	Release fifth axis brake, continue with program.
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
M97	Local Sub-Program Call
M98	Sub Program Call
M99	Sub Program Return Or Loop

9.2 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.

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.

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 is 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 is 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.

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.

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 thru 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.

M27 Release fifth axis Brake, Wait For M-fin signal.

This code activates the fifth axis brake relay, which must be connected to relay M27 on the I/O board. It activates the relay, 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.

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.

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.

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 positiveZ 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.

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.

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.

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.

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.

	HAAS	FANUC
calling program:	O0001	O0001
	...	...
	N50 M98 P2	N50 M98 P2
	N51 M99 P100	...
	...	N100 (continue here)
	N100 (continue here)	...
	...	M30
	M30	
subroutine:	O0002	O0002
	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.

10. 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).

10.1 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:

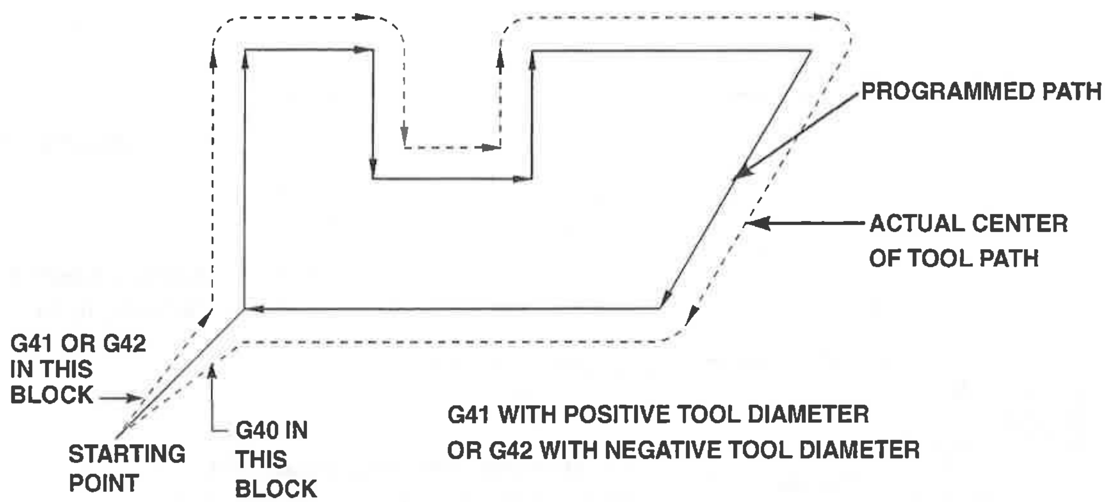
When Setting 58 is set to Yasnac, the control must be able to position the tool edge along all of the programmed cuts without overcutting the next two motions. All outside angles are joined by a circular motion.

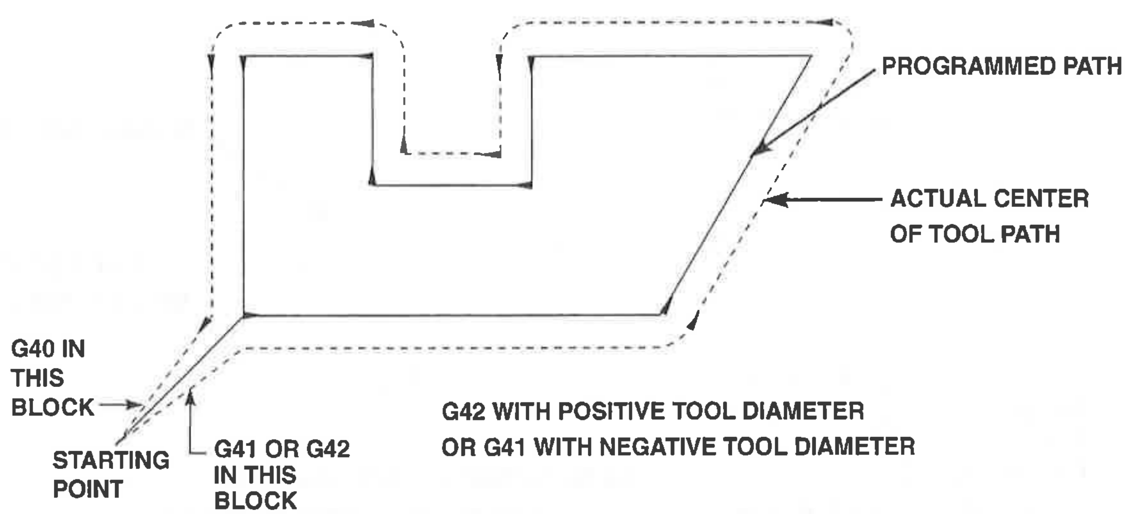
When Setting 58 is set to Fanuc, the control does not require that the tool cutting edge be placed along all programmed cuts. Overcutting, however, is still prevented and, if overcutting cannot be prevented, an alarm will still occur. Outside angles less than or equal to 270 degrees are joined by a square 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

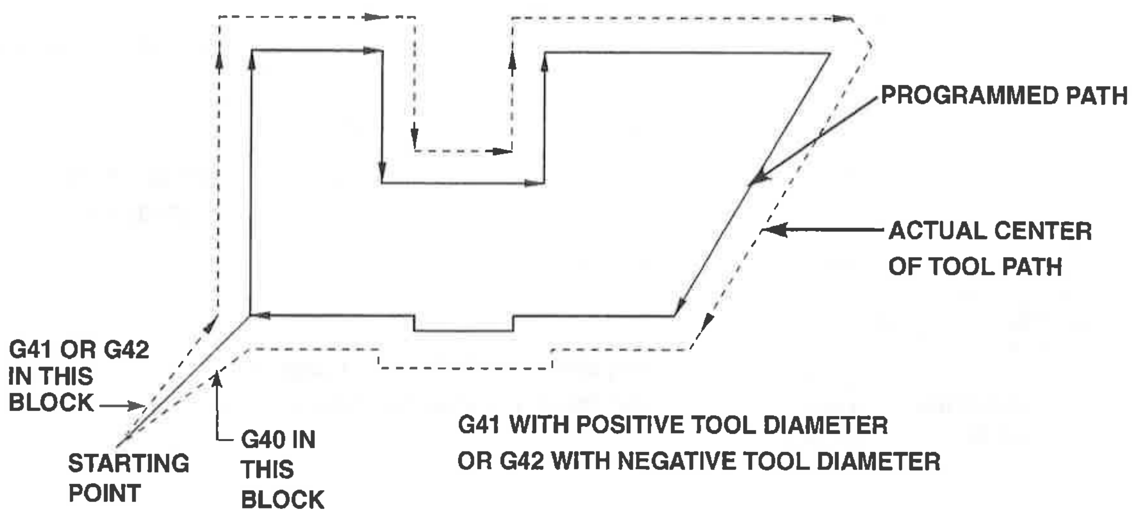
(Yasnac style)

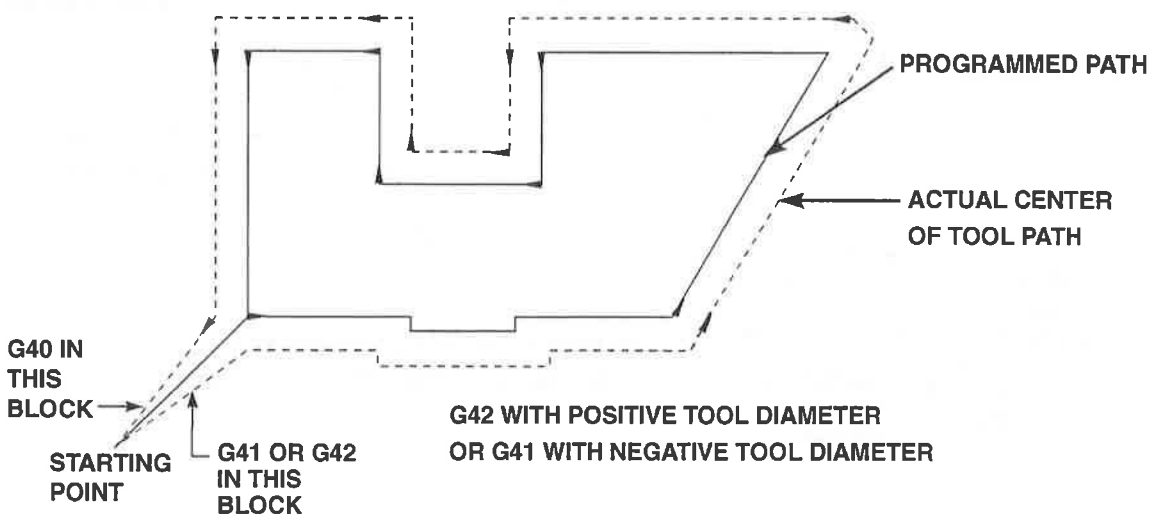




CUTTER COMPENSATION

(Fanuc style)





10.2 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 non-zero **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.

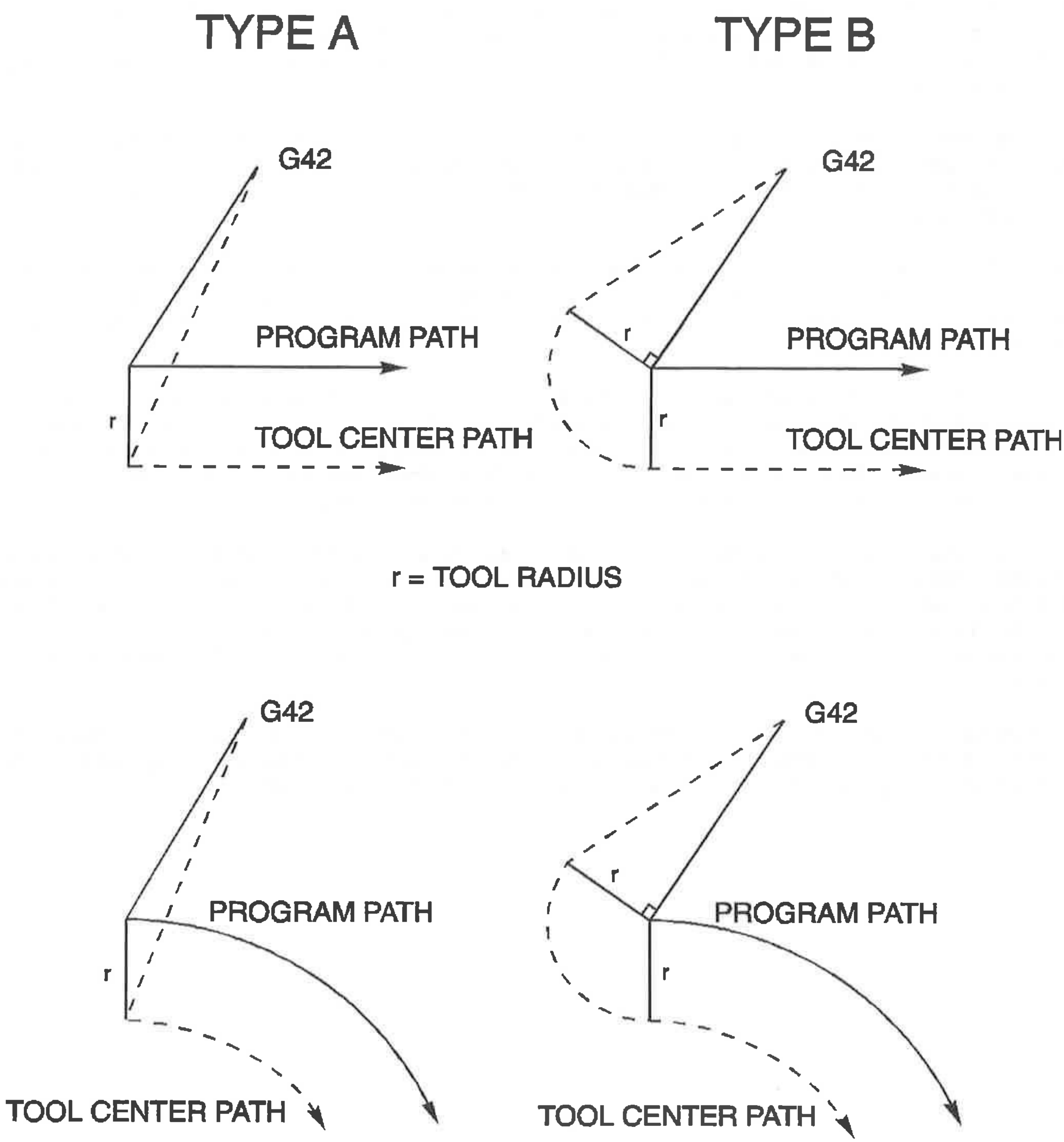
You should always turn off cutter compensation in a move which clears the tool away from the part being cut. If a program is terminated with cutter compensation still active, an alarm is generated. In addition, you cannot turn on or off cutter compensation during a circular move (G02 or G03); otherwise 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.

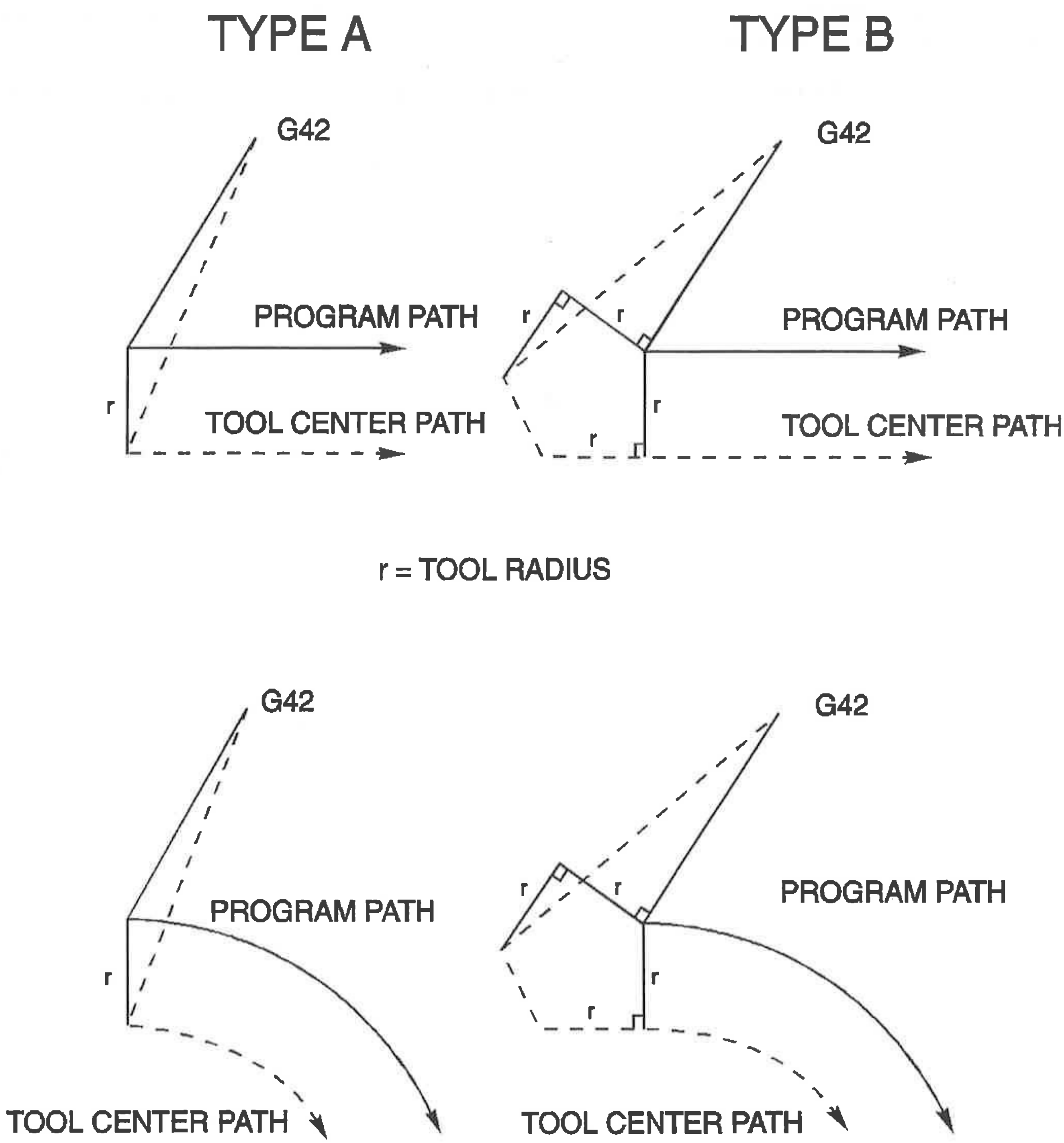
When turning on cutter compensation in a move followed by a second move at an angle of less than 90 degrees, there are two common ways of computing the first motion. They are called cutter compensation type A and B. Type A will not stay on the programmed side of the first cut but will go directly to 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.

CUTTER COMPENSATION ENTRY
(Yasnac style)



CUTTER COMPENSATION ENTRY
(Fanuc style)



10.3 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.

11. 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.

11.1 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.

11.2 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 1500000 steps/sec/sec and Parameter 113 is 375 (0.0375 seconds); the maximum velocity for exponential accel/decel would be:

1500000 x 0.0375 = 56250 steps/second
For a 2000 line encoder and 6 mm screw, this would be:
60 x 56250 / 33867 = 100 inches/minute

11.3 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 Parameters 101...104. If this is set to 2500 (corresponding to about 0.0725 inches), with Parameter 113 set to 375 (0.0375 seconds), this means that the highest feed rate which will give continuous cutter motion is:

(2500/33867)*60/0.0375 = 110 inches per minute.

11.4 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 (see example above giving 159 inches per minute). An example of the "In position limit" values follows. If Parameter 101 (for **X**) is 2000 and Parameter 5 is 33867, a rapid move of will proceed to the next block when the X-axis is within a distance of:

1000 / 33867 = 0.0295 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.

11.5 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 = 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 = (Par. 113) * (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 inches
and
Ra = 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.

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.

11.6 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.

12. 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. Normally, the control can do 500 blocks per second and 1000 is an option. No special programming or setup is required to get these speeds.

When operating at feed rates above 150 inches per minute, there is a parameter which will cause a pause at the end of each stroke. If you have a large number of very short strokes, this may be a problem. The pause is to insure that the tool is within a preset distance of the desired end point and is used to guarantee square corners. This default parameter value corresponds to 0.118 inches. If you operate faster than 100 inches per minute and do not need better accuracy or the strokes are well blended (no sharp corners), this parameter (101..104) can be changed.

Another function, called **In Position Accuracy**, can be used to insure corners are accurate by slowing the feed rate only between strokes which are not well blended. This is setting 85 and is described in the next section. The accuracy value in setting 85 is insured between strokes only when required to get that accuracy.

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.

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. 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.

12.1 IN POSITION ACCURACY

In position accuracy is a function which allows you to set the required accuracy or corner squareness where strokes meet. 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 from a program.

The In Position Accuracy function will cause a pause at the end of any block where there would otherwise have been corner rounding exceeding the specified accuracy. This pause is normally just a slight slow down of the feed rate. 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 values of parameters 101..104 may also cause a pause at the end of a stroke if you are machining faster than 150 inches per minute. In this case, these parameters can be increased to larger values and setting 85 used to slow the feed only when sharp corners are required.

The most important thing to remember with In Position Accuracy is that normal, well blended strokes should not get a slowdown or pause. Only the sharp corners need this in order to achieve the programmed accuracy. However, setting the accuracy to extremely small values (0.0005 and smaller) may cause a pause when you don't think it should. Even well blended strokes can have a slight error.

13. 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:

```
O0100 (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) ;
```

```
O0200 (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);
```

14.MACROS

14.1 INTRODUCTION

This control function is optional. If you wold 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.

14.2 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] DO1                     (Repeat T times)
G91 G81 Z#26 ;                        (Drill To Z depth)
#20=#20-1 ;                           (Decrement counter)
IF [#20 EQ 0] GOTO5 ;                 (All holes drilled)
G00 X#24 Y#25 ;                       (Move along slope)
N5 END1 ;
M99 ;                                 (Return to caller)
```

14.3 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.

■ 14.4 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	D	E	F	G	H	I	J	K
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 <u>not</u> 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.

14.5 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.

14.6 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.
The block

N1 G0 G90 X1.0 Y0 ;

can be replaced by,

N1 G#7 G#11 X#1 Y#2 ;

providing that the variables take on the values:

```
#7=0;
#11=90;
#1=1.0;
#2=0.0;
```

Here, 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
#2000-#2999	TOOL OFFSETS
#3000	PROGRAMMABLE ALARM WITH MESSAGE
#3001	MILLISECOND TIMER
#3002	HOUR TIMER
#3006	PROGRAMMABLE STOP WITH MESSAGE
#4001-#4021	LAST BLOCK GROUP CODES
#4101-#4126	LAST BLOCK ADDRESS DATA
#5001-#5004	LAST BLOCK TARGET POSITION
#5021-#5024	CURRENT MACHINE COORD POSITION
#5041-#5044	CURRENT WORK COORD POSITION
#5061-#5064	CURRENT SKIP SIGNAL POSITION
#5081-#5084	TOOL LENGTH COMPENSATION
#5221-#5224	G54 OFFSET VALUES
#5241-#5244	G55 " "
#5261-#5264	G56 " "
#5281-#5284	G57 " "
#5301-#5304	G58 " "
#5321-#5324	G59 " "
#7001-#7004	G110 ADDITIONAL OFFSET VALUES
#7021-#7024	G111 " " "
#7381-#7384	G129 " " "

14.7 SYSTEM VARIABLES IN-DEPTH

This section fully describes system variables.

• 1-Bit Discrete Inputs

For a complete description of discrete inputs, refer to the Service Manual. 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 03 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-#2050	H geometry offsets (1-50) for length.
#2200-#2250	H geometry wear (1-50) for length.
#2401-#2450	D geometry offsets (1-50) for diameter.
#2601-#2650	D geometry wear (1-50) 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);
```

In addition to displaying a message and stopping, the first 34 characters of the comment will be placed on the last line of the operator notes page. Lines 2 through 16 of operator notes are scrolled up and the first line is lost. Important information should not be placed in operator notes if programmable stops are to be used.

• 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 **J** value is #4104.

• Last Target Position

#5001-#5004 The final programmed point, target position, for the most recent motion block can be accessed through variables #5001-#5004, 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-#5024 The current position in machine coordinates can be obtained through #5021-#5024, 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-#5044 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-#5064 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-#5084 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-#5204	G52 X, Y, Z, A OFFSET VALUES
#5221-#5224	G54 " " " " " "
#5241-#5244	G55 " " " " " "
#5261-#5264	G56 " " " " " "
#5281-#5284	G57 " " " " " "
#5301-#5304	G58 " " " " " "
#5321-#5324	G59 " " " " " "
#7001-#7004	ADDITIONAL X, Y, Z, A OFFSET VALUES
#7021-#7024	" " " " " "
"	" " " " " "
#7941-#7944	ADDITIONAL X, Y, Z, A OFFSET VALUES

14.8 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.

14.9 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.

A 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 ;
G0 X[ #1 + #1 ] ;
     (Table moves to 1.0067) ;
G0 X[ ROUND[ #1 ] + ROUND[ #1 ] ] ;
     (Table moves to 1.0066) ;
G0 A[ #1 + #1 ] ;
     (Axis moves to 1.007) ;
G0 A[ ROUND[ #1 ] + ROUND[ #1 ] ] ;
     (Axis moves to 1.006) ;
D[1.67]     (Diameter 2 is made current) ;
```

B 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	Here the variable #3
#2=2.0;	0000 0010	will contain 3.0 after
#3=#1 OR #2	0000 0011	the OR operation.
#1=5.0;		Here control will
#2=3.0;		transfer to block 1
IF [#1 GT 3.0 AND #2 LT 10]		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.

C 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;
```

D 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.

E 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 proram that serializes parts:

```
O9200    (Engrave digit at current location.)
;
(D=DECIMAL DIGIT TO ENGRAVE)    ;
IF [[#7 NE #0] AND [#7 GE O] 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 brach processing based on the results of reading hardware imputs. An example might look like the following:

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

```
M99
N2  (1030=1, 1031=0)
...
M99
N3  (1030=1, 1031=1)
...
M99
```

The discrete inputs always return either 0 or 1 when read. The GOTO[expression] will branch to the appropriate 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>] GOTOn

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 ;
```

The previous 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] DO1 ;
END1 ;

<other statements>

#3001=0 (WAIT 300 MILLISECONDS) ;
WH [#3001 LT 150] DO1 ;
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,

```
DO1 ;
<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 DO1 (letter O).

14.10 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 that 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 ;
```

■ *14.11 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.

14.12 OPERATION NOTES

This section explains the additional screens and operator actions that come with macros.

• 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 COMNDS 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.

14.13 FANUC-STYLE MACRO FEATURES NOT INCLUDED IN HAAS CNC CONTROL

This section lists the FANUC macro features that have not been implemented as of this release.

M ALIASING	REPLACE G65 Pnnnn WITH Mnn PROGS 9020-9029.	
P ADDRESS IN G65	NEED TO ALLOW P ADDRESS	
G66 MODAL	CALL IN EVERY MOTION BLOCK	
G66.1 MODAL	CALL IN EVERY BLOCK	
G67 MODAL CANCEL		
M98 ALIASING, T CODE		PROG 9000, VAR #149, ENABLE BIT
M98 ALIASING, S CODE		PROG 9029, VAR #147, ENABLE BIT
M98 ALIASING, B CODE		PROG 9028, VAR #146, ENABLE BIT
SKIP/N N=1...9		
#3003 SINGLE BLOCK SUPPRESSION FLAG		
#3007 MIRROR IMAGE ON FLAG, EACH AXIS		
#3011 YEAR/MONTH/DAY		
#3012 HOUR/MINUTE/SECOND		
#3901 TOTAL NUMBER OF PARTS		
#3902 REQUIRED NUMBER OF PARTS		
#4201-#4320 CURRENT BLOCK MODAL DATA		
#5101-#5106 CURRENT SERVO DEVIATION		

ADDITIONAL OFFSETS	G54.1P## FORMAT
NAMES FOR VARIABLES	FOR DISPLAY PURPOSES
ATAN [ ]/[ ]	ARCTANGENT, FANUC VERSION
BIN [ ]	CONVERSION FROM BCD TO BIN
BCD [ ]	CONVERSION FROM BIN TO BCD
FUP [ ]	TRUNCATE FRACTION CEILING
LN [ ]	NATURAL LOGARITHM
EXP [ ]	BASE E EXPONENTIATION
ADP [ ]	RE-SCALE VAR TO WHOLE NUMBER
BPRNT [ ]	

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.

15. QUICKCODE

15.1 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 encryptive 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 suite 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 than 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 different 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.

15.2 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.

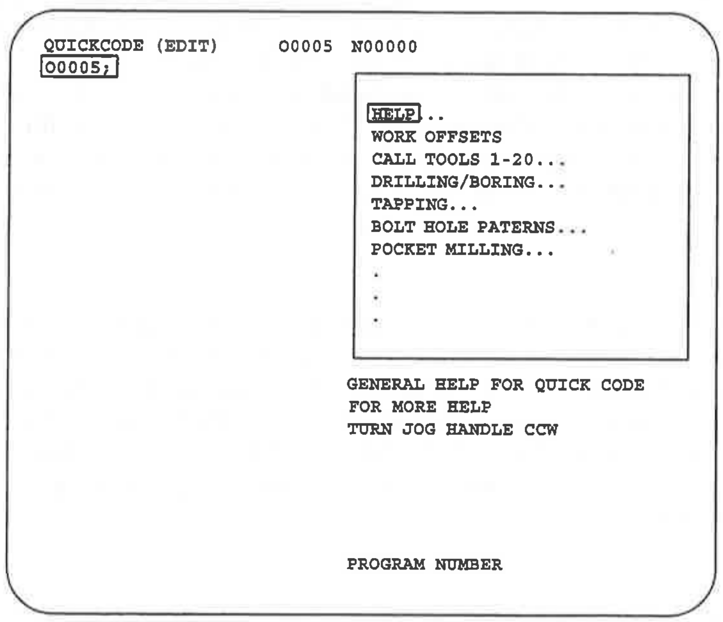


Figure 1. The QUICKCODE display

15.3 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 pressing 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.

■ 15.4 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.

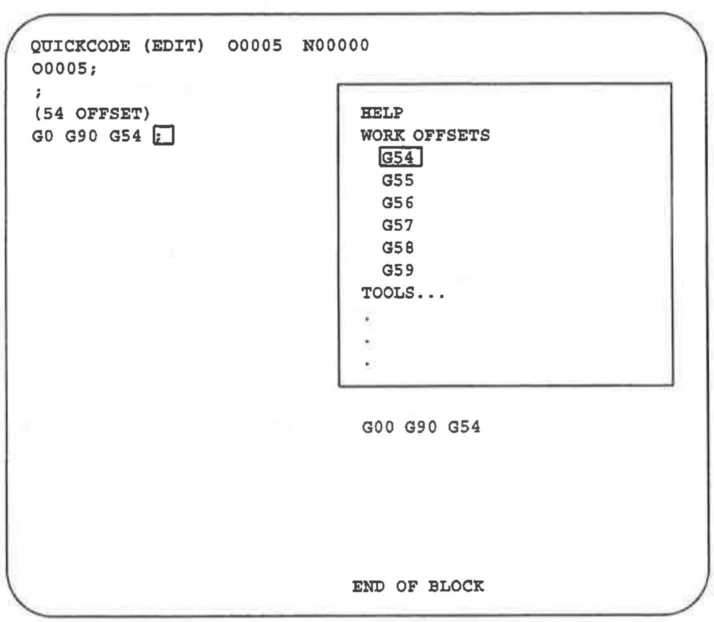


Figure 2 (Empty Program)

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.

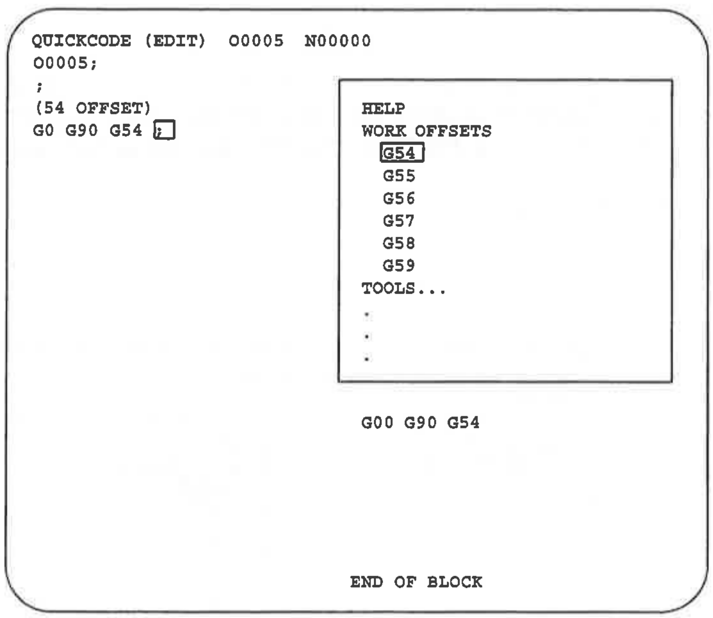


Figure 3 (Program with G54 added)

- 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.

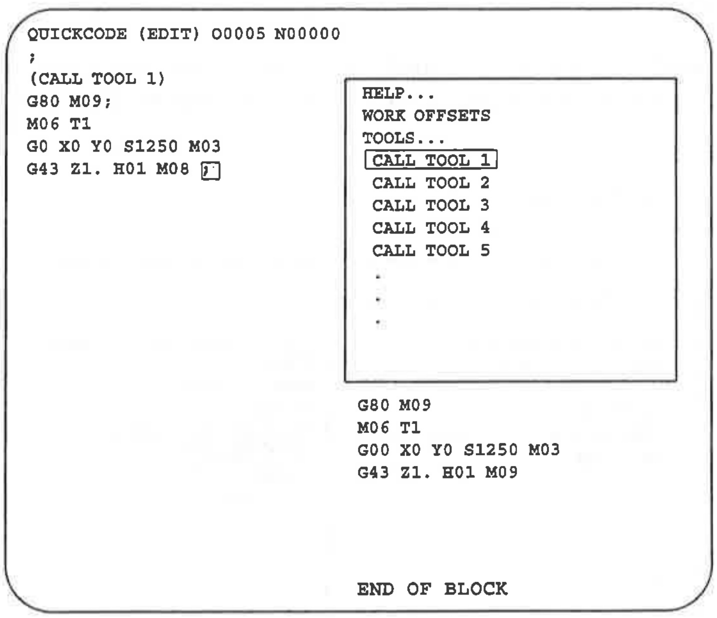


Figure 4 (Program with CALL TOOL 1 added)

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.

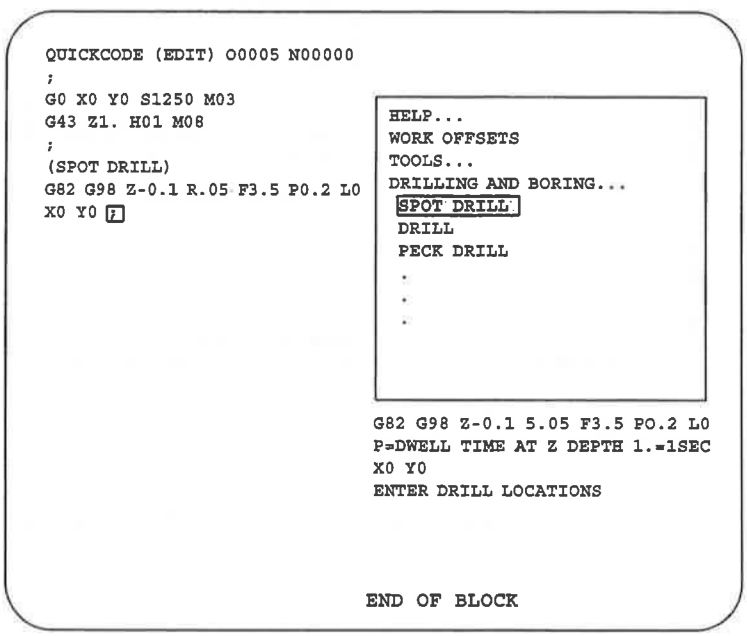


Figure 5 (Program with spot drilling invoked)

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.

L1.5 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) O0005 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

END OF BLOCK
```

Figure 6 (Program with spot drilling invoked)

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.

15.5 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 O9999 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 O9999 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.

```
%
O9999 (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 morethan 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)
G0 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.

15.6 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 you control with a proper backup scheme. Remember... this file operates the QUICKCODE feature in you HAAS machine.

The following program is a sample QUICKCODE source file. It contains many examples of how QUICKCODE can be used.

```
%
O9999 (QUICKCODE - HAAS AUTOMATION INC.)
(*HELP...)
(GENERAL HELP FOR QUICK CODE)
(FOR MORE HELP)
(TURN JOG HANDLE COUNTERCLOCKWISE)
(**MOVING THROUGH GROUP ITEMS)
(TO MOVE THROUGH GROUP ITEMS, TURN)
(THE JOG HANDLE COUNTERCLOCKWISE)
(THIS IS CALLED OPENING A GROUP)
(**MOVING THROUGH GROUPS)
(TO MOVE THROUGH QUICK CODE GROUPS)
(TURN THE JOG HANDLE CLOCKWISE)
(**QUICK CODE TERMINOLOGY)
(A GROUP IS ANY ITEM ON THE MAIN)
(QUICKCODE LIST)
(GROUP ITEMS ARE DISPLAYED WHEN)
(A GROUP IS OPENED)
(**SELECTING A GROUPS ITEM CODE)
(CODE MAY BE ASSOCIATED WITH A GROUP)
(ITEM. THE ITEMS CODE CAN BE)
(INSERTED INTO THE CURRENT PROGRAM)
(BEING EDITED BY HIGHLIGHTING THE)
(ITEM VIA THE JOG HANDLE AND)
(PRESSING THE WRITE KEY.)
(**CREATING A QUICK CODE FILE)
(YOU CAN CREATE YOUR OWN CUSTOM)
(QUICK CODE FILE BY EDITING THIS)
(FILE OR BY LOADING A NEW FILE)
(REFER TO CHAPTER ?? OF THE)
(PROGRAMMING MANUAL)
() (********************************)
(*WORK OFFSETS...)
(OFFSETS G54 THROUGH G59)
(**G54)
(G00 G90 G54)
```

```
;
(G54 OFFSET)
G00 G90 G54
(**G55)
(G00 G90 G55)
;
(G55 OFFSET)
G00 G90 G55
(**G56)
(G00 G90 G56)
;
(G56 OFFSET)
G00 G90 G56
(**G57)
(G00 G90 G57)
;
(G57 OFFSET)
G00 G90 G57
(**G58)
(G00 G90 G58)
;
(G58 OFFSET)
G00 G90 G58
(**G59)
(G00 G90 G59)
;
(G59 OFFSET)
G00 G90 G59
() (********************************)
(*TOOLS...)
(A STANDARD TOOL CHANGE SEQUENCE)
(FOR TOOLS 1 THROUGH 20)
(**CALL TOOL 1)
(G80 M09)
(M06 T1)
(G00 X0 Y0 S1250 M03)
(G43 Z1. H01 M08)
;
(CALL TOOL 1)
G80 M09
M06 T1
G00 X0 Y0 S1250 M03
G43 Z1. H01 M08
()
(**CALL TOOL 2)
(G80 M09)
(M06 T2)
(G00 X0 Y0 S1250 M03)
(G43 Z1. H02 M08)
;
(CALL TOOL 2)
G80 M09
M06 T2
G00 X0 Y0 S1250 M03
G43 Z1. H02 M08
()
(**CALL TOOL 3)
```

```
(G80 M09)
(M06 T3)
(G00 X0 Y0 S1250 M03)
(G43 Z1. H03 M08)
;
(CALL TOOL 3)
G80 M09
M06 T3
G00 X0 Y0 S1250 M03
G43 Z1. H03 M08
()
(**CALL TOOL 4)
(G80 M09)
(M06 T4)
(G00 X0 Y0 S1250 M03)
(G43 Z1. H04 M08)
;
(CALL TOOL 4)
G80 M09
M06 T4
G00 X0 Y0 S1250 M03
G43 Z1. H04 M08
()
(**CALL TOOL 5)
(G80 M09)
(M06 T5)
(G00 X0 Y0 S1250 M03)
(G43 Z1. H05 M08)

(CALL TOOL 5)
G80 M09
M06 T5
G00 X0 Y0 S1250 M03
G43 Z1. H05 M08
()
(**CALL TOOL 6)
(G80 M09)
(M06 T6)
(G00 X0 Y0 S1250 M03)
(G43 Z1. H06 M08)
;
(CALL TOOL 6)
G80 M09
M06 T6
G00 X0 Y0 S1250 M03
G43 Z1. H06 M08
()
(**CALL TOOL 7)
(G80 M09)
(M06 T7)
(G00 X0 Y0 S1250 M03)
(G43 Z1. H07 M08)
;
(CALL TOOL 7)
G80 M09
M06 T7
G00 X0 Y0 S1250 M03
```

```
G43 Z1. H07 M08
()
(**CALL TOOL 8)
(G80 M09)
(M06 T8)
(G00 X0 Y0 S1250 M03)
(G43 Z1. H08 M08)
;
(CALL TOOL 8)
G80 M09
M06 T8
G00 X0 Y0 S1250 M03
G43 Z1. H08 M08
()
(**CALL TOOL 9)
(G80 M09)
(M06 T9)
(G00 X0 Y0 S1250 M03)
(G43 Z1. H09 M08)

(CALL TOOL 9)
G80 M09
M06 T9
G00 X0 Y0 S1250 M03
G43 Z1. H09 M08
()
(**CALL TOOL 10)
(G80 M09)
(M06 T10)
(G00 X0 Y0 S1250 M03)
(G43 Z1. H10 M08)
;
(CALL TOOL 10)
G80 M09
M06 T10
G00 X0 Y0 S1250 M03
G43 Z1. H10 M08
()
(**CALL TOOL 11)
(G80 M09)
(M06 T11)
(G00 X0 Y0 S1250 M03)
(G43 Z1. H11 M08)
;
(CALL TOOL 11)
G80 M09
M06 T11
G00 X0 Y0 S1250 M03
G43 Z1. H11 M08
()
(**CALL TOOL 12)
(G80 M09)
(M06 T12)
(G00 X0 Y0 S1250 M03)
(G43 Z1. H12 M08)
;
(CALL TOOL 12)
```

```
G80 M09
M06 T12
G00 X0 Y0 S1250 M03
G43 Z1. H12 M08
()
(**CALL TOOL 13)
(G80 M09)
(M06 T13)
(G00 X0 Y0 S1250 M03)
(G43 Z1. H13 M08)

(CALL TOOL 13)
G80 M09
M06 T13
G00 X0 Y0 S1250 M03
G43 Z1. H13 M08
()
(**CALL TOOL 14)
(G80 M09)
(M06 T14)
(G00 X0 Y0 S1250 M03)
(G43 Z1. H14 M08)
;
(CALL TOOL 14)
G80 M09
M06 T14
G00 X0 Y0 S1250 M03
G43 Z1. H14 M08
()
(**CALL TOOL 15)
(G80 M09)
(M06 T15)
(G00 X0 Y0 S1250 M03)
(G43 Z1. H15 M08)
;
(CALL TOOL 15)
G80 M09
M06 T15
G00 X0 Y0 S1250 M03
G43 Z1. H15 M08
()
(**CALL TOOL 16)
(G80 M09)
(M06 T16)
(G00 X0 Y0 S1250 M03)
(G43 Z1. H16 M08)
;
(CALL TOOL 16)
G80 M09
M06 T16
G00 X0 Y0 S1250 M03
G43 Z1. H16 M08
()
(**CALL TOOL 17)
(G80 M09)
(M06 T17)
(G00 X0 Y0 S1250 M03)
```

```
(G43 Z1. H17 M08)

(CALL TOOL 17)
G80 M09
M06 T17
G00 X0 Y0 S1250 M03
G43 Z1. H17 M08
()
(**CALL TOOL 18)
(G80 M09)
(M06 T18)
(G00 X0 Y0 S1250 M03)
(G43 Z1. H18 M08)
;
(CALL TOOL 18)
G80 M09
M06 T18
G00 X0 Y0 S1250 M03
G43 Z1. H18 M08
()
(**CALL TOOL 19)
(G80 M09)
(M06 T19)
(G00 X0 Y0 S1250 M03)
(G43 Z1. H19 M08)
;
(CALL TOOL 19)
G80 M09
M06 T19
G00 X0 Y0 S1250 M03
G43 Z1. H19 M08
()
() (*******************************)
(*DRILLING / BORING...)
(ROUTINES FOR)
(SPOT DRILLING, DRILLING,)
(PECK DRILLING, BORING)
(AND MORE)
(**SPOT DRILL)
(G82 G98 Z-0.1 R0.05 F3.5 P0.2 L0)
(P=DWELL TIME AT Z DEPTH 1.=1SEC.)
(X0 Y0)
(ENTER DRILL LOCATIONS)
;
(SPOT DRILL)
G82 G98 Z-0.1 R0.05 F3.5 P0.2 L0
X0 Y0
()
(**DRILL)
(G81 G98 Z-0.5 R0.05 F3.5 L0)
(X0 Y0)
(ENTER DRILL LOCATIONS)
;
(DRILL)
G81 G98 Z-0.5 R0.05 F3.5 L0
X0 Y0
()
```

```
(**PECK DRILL)
(G73 G98 Z-1.1 R0.05 Q0.2 F3.5 L0)
(X0 Y0)
(ENTER DRILL LOCATIONS)
 ;
(PECK DRILL)
G73 G98 Z-1.1 R0.05 Q0.2 F3.5 L0
X0 Y0
()
(**PECK DRILL W/REDUCING PECK VALUE)
(G73 G98 Z-1.1 R0.05 I0.5 J0.1 K0.2 F3.5 L0)
(I=1ST PECK J=REDUCE BY K=MIN.PECK)
(X0 Y0)
(ENTER DRILL LOCATIONS)
 ;
(PECK DRILL W/REDUCING PECK VALUE)
G73 G98 Z-1.1 R0.05 I0.5 J0.1 K0.2 F3.5 L0
X0 Y0
()
(**PECK DRILL W/RETURN TO R PLANE)
(G73 G98 Z-2.2 R0.05 Q0.2 K0.6 F3.5 L0)
(Q=PECK K=CLEAR EVERY)
(X0 Y0)
(ENTER DRILL LOCATIONS)
 ;
(PECK DRILL W/RETURN TO R PLANE)
G73 G98 Z-2.2 R0.05 Q0.2 K0.6 F3.5 L0
X0 Y0
()
(**DEEP HOLE PECK DRILL)
(G83 G98 Z-2.2 R0.05 Q0.2 F3.5 L0)
(Q=PECK)
(X0 Y0)
(ENTER DRILL LOCATIONS)
 ;
(DEEP HOLE PECK DRILL)
G83 G98 Z-2.2 R0.05 Q0.2 F3.5 L0
X0 Y0
()
(**BORE IN RAPID OUT)
(G81 G98 Z-1.2 R0.05 F3.5 L0)
(X0 Y0)
(ENTER BORE LOCATIONS)
 ;
(BORE IN RAPID OUT)
G81 G98 Z-1.2 R0.05 F3.5 L0
X0 Y0
()
(**BORE IN BORE OUT)
(G85 G98 Z-1.2 R0.05 F3.5 L0)
(X0 Y0)
(ENTER BORE LOCATIONS)
 ;
(BORE IN BORE OUT)
G85 G98 Z-1.2 R0.05 F3.5 L0
X0 Y0
()
```

```
(**BORE IN SHIFT RAPID OUT)
(G76 G98 Z-1.2 R0.05 Q0.02 F3.5 L0)
(Q=SHIFT AMOUNT, SETTING NO.27=DIRECTION)
(X0 Y0)
(ENTER BORE LOCATIONS)
 :
(BORE IN SHIFT RAPID OUT)
G76 G98 Z-1.2 R0.05 Q0.02 F3.5 L0
X0 Y0
() (*******************************)
(*TAPPING...)
(READY TO USE CODE FOR MOST)
(STANDARD TAPS)
(**10-32 TAP)
(S700 M05)
(G84 G98 Z-0.8 R0.2 F21.875 L0)
(X0 Y0)
(ENTER TAP LOCATIONS)
 :
(10-32 TAP)
S700 M05
G84 G98 Z-0.8 R0.2 F21.875 L0
X0 Y0
()
(**1/4-20 TAP)
(S700 M05)
(G84 G98 Z-0.8 R0.2 F35. L0)
(X0 Y0)
(ENTER TAP LOCATIONS)
 :
(1/4-20 TAP)
S700 M05
G84 G98 Z-0.8 R0.2 F35. L0
X0 Y0
()
(**1/4-28 TAP)
(S700 M05)
(G84 G98 Z-0.8 R0.2 F25. L0)
(X0 Y0)
(ENTER TAP LOCATIONS)
 :
(1/4-28 TAP)
S700 M05
G84 G98 Z-0.8 R0.2 F25. L0
X0 Y0
()
(**5/16-18 TAP)
(S684 M05)
(G84 G98 Z-0.8 R0.2 F38. L0)
(X0 Y0)
(ENTER TAP LOCATIONS)
 :
(5/16-18 TAP)
S684 M05
G84 G98 Z-0.8 R0.2 F38. L0
X0 Y0
()
```

```
(**5/16-24 TAP)
(S696 M05)
(G84 G98 Z-0.8 R0.2 F29. L0)
(X0 Y0)
(ENTER TAP LOCATIONS)
;
(5/16-24 TAP)
S696 M05
G84 G98 Z-0.8 R0.2 F29. L0
X0 Y0
()
(**3/8-16 TAP)
(S640 M05)
(G84 G98 Z-0.8 R0.2 F40. L0)
(X0 Y0)
(ENTER TAP LOCATIONS)
;
(3/8-16 TAP)
S640 M05
G84 G98 Z-0.8 R0.2 F40. L0
X0 Y0
()
(**3/8-24 TAP)
(S600 M05)
(G84 G98 Z-0.8 R0.2 F25. L0)
(X0 Y0)
(ENTER TAP LOCATIONS)
;
(3/8-24 TAP)
S600 M05
G84 G98 Z-0.8 R0.2 F25. L0
X0 Y0
()
(**1/2-13 TAP)
(S494 M05)
(G84 G98 Z-0.8 R0.2 F38. L0)
(X0 Y0)
(ENTER TAP LOCATIONS)
;
(1/2-13 TAP)
S494 M05
G84 G98 Z-0.8 R0.2 F38. L0
X0 Y0
()
(**1/2-20 TAP)
(S500 M05)
(G84 G98 Z-0.8 R0.2 F25. L0)
(X0 Y0)
(ENTER TAP LOCATIONS)
;
(1/2-20 TAP)
S500 M05
G84 G98 Z-0.8 R0.2 F25. L0
X0 Y0
() (********************************)
(*BOLT HOLE PATTERNS...)
(PROGRAM THE MACHINE TO THE X Y)
```

```
(POSITION OF THE BOLT CIRCLE)
(CENTER OR START OF ANGLE LINE)
(SET UP A DRILL CYCLE)
(THEN SET UP A BOLT HOLE CYCLE)
(**BOLT CIRCLE)
(G70 I1.5 J90. L7)
(DRILL 7 HOLES EQUALLY SPACED)
(ON A 3.0 DIA. B.C. WITH THE)
(FIRST HOLE IN THE Y+ DIRECTION)
;
(BOLT CIRCLE)
G70 I1.5 J90. L7
()
(**BOLT HOLE ARC)
(G71 I4. J45. K22.5 L4)
(DRILL 4 HOLES 22.5 DEG. APART)
(ON A 8.0 DIA. B.C. WITH THE)
(FIRST HOLE AT 45 DEG. FROM THE)
(START POINT)
;
(BOLT HOLE ARC)
G71 I4. J45. K22.5 L4
()
(**BOLT HOLES ALONG AN ANGLE)
(G72 I0.75 J30. L6)
(DRILL 6 HOLES .750 APART)
(ON A 30DEG. LINE WITH THE)
(FIRST HOLE AT THE START POINT)
;
(BOLT HOLES ALONG AN ANGLE)
G72 I0.75 J30. L6
() (********************************)
(*CIRCULAR POCKET MILLING...)
(ROUTINES FOR ROUGHING)
(AND FINISHING, CW AND CCW, OF)
(CIRCULAR POCKETS)
(**ROUGH CIRCULAR POCKET CCW)
(G00 X0 Y0)
(ENTER CENTER OF POCKET)
(G13 I0.3 K1.5 Q0.3 F10. Z-0.25 D01)
(I=FIRST CIRCLE RADIUS K=POCKET RADIUS)
(Q=STEP OVER AMOUNT D=RADIUS OFFSET)
;
(ROUGH CIRCULAR POCKET CCW)
G00 X0 Y0
G13 I0.3 K1.5 Q0.3 F10. Z-0.25 D01
()
(**ROUGH CIRCULAR POCKET CW)
(G00 X0 Y0)
(ENTER CENTER OF POCKET)
(G12 I0.3 K1.5 Q0.3 F10. Z-0.25 D01)
(I=FIRST CIRCLE RADIUS K=POCKET RADIUS)
(Q=STEP OVER AMOUNT D=RADIUS OFFSET)
;
(ROUGH CIRCULAR POCKET CW)
G00 X0 Y0
G12 I0.3 K1.5 Q0.3 F10. Z-0.25 D01
```

```
()
(**ROUGH CIRCULAR POCKET CCW W/ Z STEP DOWN)
(G00 X0 Y0)
(ENTER CENTER OF POCKET)
(G01 Z0 F30.)
(G91)
(G13 I0.3 K1.5 Q0.3 F10. Z-0.25 D01 L3)
;
(ROUGH CIRCULAR POCKET CCW W/ Z STEP DOWN)
G00 X0 Y0
G01 Z0 F30.
G91
G13 I0.3 K1.5 Q0.3 F10. Z-0.25 D01 L3
()
(**CCW W/ Z STEP DOWN - ADDRESS CODE HELP)
(I=FIRST CIRCLE RADIUS K=POCKET RADIUS)
(Q=STEP OVER AMOUNT)
(Z=STEP DOWN EACH PASS)
(L=NUMBER OF STEP DOWN PASSES)
(D=CUTTER RADIUS OFFSET BEING USED)
()
(**ROUGH CIRCULAR POCKET CW W/ Z STEP DOWN)
(G00 X0 Y0)
(ENTER CENTER OF POCKET)
(G01 Z0 F30.)
(G91)
(G12 I0.3 K1.5 Q0.3 F10. Z-0.25 D01 L3)
;
(ROUGH CIRCULAR POCKET CW W/ Z STEP DOWN)
G00 X0 Y0
G01 Z0 F30.
G91
G12 I0.3 K1.5 Q0.3 F10. Z-0.25 D01 L3
()
(**CW W/ Z STEP DOWN - ADDRESS CODE HELP)
(I=FIRST CIRCLE RADIUS K=POCKET RADIUS)
(Q=STEP OVER AMOUNT)
(Z=STEP DOWN EACH PASS)
(L=NUMBER OF STEP DOWN PASSES)
(D=CUTTER RADIUS OFFSET BEING USED)
()
(**FINISH CIRCULAR POCKET CCW)
(G00 X0 Y0)
(ENTER CENTER OF POCKET)
(G13 I1.5 F10. Z-0.25 D01)
(I=POCKET RADIUS)
(D=CUTTER RADIUS OFFSET BEING USED)
;
(FINISH CIRCULAR POCKET CCW)
G00 X0 Y0
G13 I1.5 F10. Z-0.25 D01
()
(**FINISH CIRCULAR POCKET CW)
(G00 X0 Y0)
(ENTER CENTER OF POCKET)
(G12 I1.5 F10. Z-0.25 D01)
(I=POCKET RADIUS)
```

```
(D=CUTTER RADIUS OFFSET BEING USED)
;
(FINISH CIRCULAR POCKET CW)
G00 X0 Y0
G12 I1.5 F10. Z-0.25 D01
() (********************************)
(*USEFUL CUTS USING CRC...)
(SLOTTING, ROUND BOSS,)
(SQUARE BOSS, HEX BOSS)
(AND MORE)
(CRC IS CUTTER RADIUS COMPENSATION)
()
(**Y AXIS SLOT .75 X 2.0 1/4 EM)
(SLOT IS .75 IN X, 2.0 IN Y)
(WITH FULL RADIUS ENDS)
( )
(D## MUST MATCH CURRENT TOOL)
;
(Y AXIS SLOT)
G01 Z-0.25 F20.
G91
G01 G41 X0.25 F15. D01
(D## MUST MATCH CURRENT TOOL)
(RADIUS OFFSET = 0 FOR 1/4 EM.)
Y0.625
G03 X-0.5 R0.25
G01 Y-1.25
G03 X0.5 R0.25
G01 Y0.625
G40 X-0.25
G00 G90 Z1.
()
(**X AXIS SLOT 2.0 X .75 1/4 EM)
(SLOT IS 2.0 IN X, .75 IN Y)
(WITH FULL RADIUS ENDS)
( )
(D## MUST MATCH CURRENT TOOL)
;
(X AXIS SLOT)
G01 Z-0.25 F20.
G91
G01 G41 Y0.25 F15. D01
(D## MUST MATCH CURRENT TOOL)
(RADIUS OFFSET = 0 FOR 1/4 EM.)
X-0.625
G03 Y-0.5 R0.25
G01 X1.25
G03 Y0.5 R0.25
G01 X-0.625
G40 Y-0.25
G00 G90 Z1.
()
(**ROUND BOSS 1.0 DIA. 1. EM)
(D## MUST MATCH CURRENT TOOL)
;
(1.0 DIAMETER ROUND BOSS)
G91 G00 X-1.25
```

```
G01 G90 Z-0.5 F20.
G91 G41 X0.25 F15. D01
(D## MUST MATCH CURRENT TOOL)
(RADIUS OFFSET = 0 FOR 1.0 EM.)
G02 I1. F20.
G01 X-0.25 G40
G00 G90 Z1.
()
(**SQUARE BOSS 1.0 1.0 EM)
(D## MUST MATCH CURRENT TOOL)
;
(1.0 SQUARE BOSS)
G91 G00 X-1.25
G01 G90 Z-0.5 F20.
G91 G41 X0.25 F15. D01
(D## MUST MATCH CURRENT TOOL)
(RADIUS OFFSET = 0 FOR 1.0 EM.)
G01 Y1. F20.
X2.
Y-2.
X-2.
Y1.
G40 X-0.25
G00 G90 Z1.
()
(**HEX BOSS 1.0 FLAT 1.0 EM)
(D## MUST MATCH CURRENT TOOL)
;
(1.0 HEX BOSS)
G90 G00 Z1.
G91 X-0.977 Y1.5
G01 G90 Z-0.5 F20.
G91 G41 Y-0.5 F15. D01
(D## MUST MATCH CURRENT TOOL)
(RADIUS OFFSET = 0 FOR 1.0 EM.)
G01 X1.554 F20.
X0.578 Y-1.
X-0.578 Y-1.
X-1.154
X-0.578 Y1.
X0.578 Y1.
G00 G90 Z1.
G40 Y1.5
() (********************************)
(*POSITION A AXIS...)
(ZERO OR INDEX THE A AXIS)
(**MOVE A AXIS TO 0 DEGREES)
(G90 A0.)
;
(MOVE A AXIS TO 0 DEGREES)
G90 A0.
()
(**INCREMENT A AXIS +90 DEGREES)
(G91 A90.)
(G90)
;
(INCREMENT A AXIS +90 DEGREES)
```

```
G91 A90.
G90
()
(**DECREMENT A AXIS -90 DEGREES)
(G91 A-90.)
(G90)
;
(**DECREMENT A AXIS -90 DEGREES)
G91 A-90.
G90
() (********************************)
(*MISC. INSTRUCTIONS)
(HOME Y AND Z AXES)
(PROGRAM END, STOP, OPTIONAL STOP)
(GOTO BLOCK NUMBER)
(**HOME Z AXIS)
(G00 G91 G28 Z0 M09)
(G90 M19)
;
(HOME Z AXIS)
G00 G91 G28 Z0 M09
G90 M19
()
(**HOME Y AND Z AXIS)
(G00 Z6.)
(G00 G91 G28 Z0 Y0 M09)
(G90 M19)
;
(HOME Y AND Z AXIS)
G00 Z6.
G00 G91 G28 Z0 Y0 M09
G90 M19
()
(**END PROGRAM/RESET TO BEGINNING)
(M30)
;
(END PROGRAM/RESET TO BEGINNING)
M30
()
(**STOP PROGRAM W/O RESET)
(M00)
;
(STOP PROGRAM W/O RESET)
M00
()
(**OPTIONAL STOP W/O RESET)
(M01)
;
(OPTIONAL STOP W/O RESET)
M01
()
(**REPEAT PROGRAM)
(M99)
;
(REPEAT PROGRAM)
M99
()
```

```
(**GOTO BLOCK)
(M99 P1234)
(GOTO BLOCK N1234)
(N1234 MUST BE IN CURRENT PROGRAM)
:
(GOTO BLOCK)
M99 P1234
() (********************************)
(*MAKE SUBROUTINE...)
(GENERATES A SUBROUTINE TEMPLATE)
(FOR DOING HOLE PATTERNS. IT IS)
(RECOMMENDED THAT THESE BE PLACED)
(AFTER THE END OF PROGRAM. USE)
(KEYS TO POSTION FOR PLACEMENT.)
(**MAKE SUBROUTINE A)
(N8810 SUBROUTINE A)
(X0 Y0)
(ENTER HOLE LOCATIONS)
(M99)
:
N8810 (SUBROUTINE A)
X0 Y0
M99
()
(**MAKE SUBROUTINE B)
(N8820 SUBROUTINE B)
(X0 Y0)
(ENTER HOLE LOCATIONS)
(M99)
:
N8820 (SUBROUTINE B)
X0 Y0
M99
()
(**MAKE SUBROUTINE C)
(N8830 SUBROUTINE C)
(X0 Y0)
(ENTER HOLE LOCATIONS)
(M99)
:
N8830 (SUBROUTINE C)
X0 Y0
M99
()
(**MAKE SUBROUTINE D)
(N8840 SUBROUTINE D)
(X0 Y0)
(ENTER HOLE LOCATIONS)
(M99)
:
N8840 (SUBROUTINE D)
X0 Y0
M99
()
() (********************************)
(*DO SUBROUTINE...)
(CALLS SUBROUTINES MADE BY)
```

```
(MAKE SUBROUTINE GROUP.)
(**DO SUBROUTINE A)
(M97 P8810)
;
(DO SUBROUTINE A)
M97 P8810
()
(**DO SUBROUTINE B)
(M97 P8820)
;
(DO SUBROUTINE B)
M97 P8820
()
(**DO SUBROUTINE C)
(M97 P8830)
;
(DO SUBROUTINE C)
M97 P8830
()
```

16. 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.

16.1 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.

16.2 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 it's 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)

16.3 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

16.4 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)

III. OPERATION

1. INTRODUCTION

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.

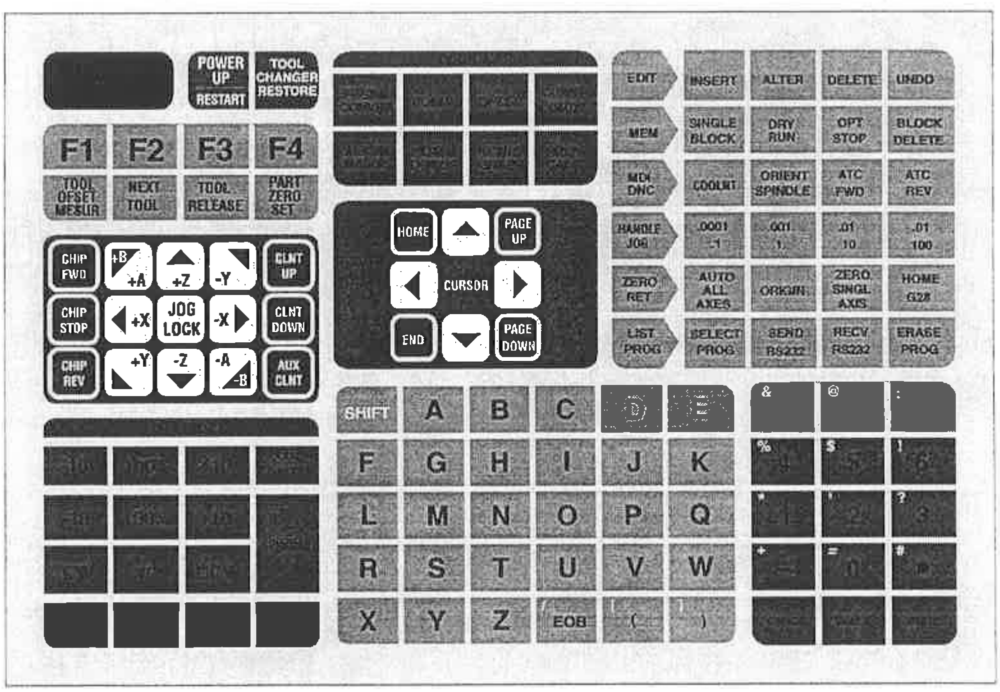


Fig. 1-1 *Control panel keypad with operating and display keys highlighted.*

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:

PROGRAM	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.

2. OPERATOR'S CONTROL PANEL

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 changes the function of these buttons with a blue character 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.

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.

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.

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.

2.1 KEYBOARD

The control panel keyboard consists of 123 keys and is divided into nine separate regions. They are:

RESET keys	Three (3) keys
FUNCTION keys	Eight (8) keys
JOG keys	Nine (9) keys
OVERRIDES	Twelve (12) 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.
AUTO POWER UP	Automatically initializes the machine at power up. After initial power up, this key can be used to reinitialize the system. The action of the key is the same as power down, but the system does not power down.
TOOL CHANGER RESTORE	Restores the tool changer to normal operation after the tool changer has encountered an interuption during a tool change. The button initiates a user prompt screen to assist the operator recover 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 OFSET 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 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 have the following actions:

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 have the following actions:

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	This key is reserved for future use.

OVERRIDES: The overrides are at the lower right 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.
RAPID	Not a key.

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	Displays the currently selected program.
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.
OFSET	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, triangle 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, it will display the entire table in the view window after F2 is pressed.

(UP ARROW) — The up arrow moves up one item, a 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) — Similar to the LEFT arrow, but moves right.

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)	Similar to the UP arrow but moves 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 blue characters on the keyboard. Pressing SHIFT and then the blue 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 port while receiving a program, it is added to the program between these two brackets.
/	The right-leaning 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.
[and]	Square brackets are used in macro expressions and functions.

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, when the block delete option is on T1 is executed.

T1/T2;
N1 G54

The HAAS does not allow multiple address codes on a line.

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

```
/ T2 M99 P1   (T2 EXECUTED WHEN BLOCK DELETE IS OFF)
T1            (T1 EXECUTED WHEN BLOCK DELETE IS ON)
N1 G54
```

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 func tion, it may not stop immediately. If the program has been inter preted 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.
MDI	Selects MDI 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.
.01, 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.
RECV RS232	Receives programs from the RS232 serial port.
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.

3. POWER ON/OFF AND SETUP

3.1 POWER ON

There is only one way to turn on this CNC. This is by 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 AUTO POWER UP 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**, or **A** 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 AUTO POWER UP 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**, and **A** axes all at rapid rate. If the Z-axis is positioned above the machine zero, the **X**, **Y**, and **A** 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 (**B**, **C**, ...) are returned to home after **X**, **Y**, **Z**, and **A**. 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.

3.2 POWER OFF

There are several ways of turning the machine off. By pushing the red POWER OFF button, power to the machine is removed instantly. Another preferable method is to use the POWER DOWN key. The POWER DOWN key will put the machine in the proper order for turn on the next time and will also sequence an orderly machine shut down. When this key is pushed, the axes will zero return, tool one will be put in the spindle, and the machine will turn off in 30 seconds.

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 on the Setting page.

A sustained overvoltage condition or sustained overheat condition will also shut this machine off automatically. If either of these conditions remains true for 4.5 minutes, the machine will start the 30 second auto-shutdown as above. 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.

3.3 SETUP PROCEDURES

The following sequence of operations is strongly recommended for setting up this machine:

1) Load a program into memory. This is done either manually entered or downloaded from a CAM package via the RS-232 interface.

2) Determine tools needed and get them ready.

3) Use a vise or fixture to hold the part and mount on 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.

4. MANUAL OPERATION

4.1 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 turned on, in 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.

4.2 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 cannot be manually jogged from the front panel. The single axis control jog button should be used for this.

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 when you are slow milling the soft jaws of a vise as an example.

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 ½ second. After ½ 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, 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.

5. AUTOMATIC OPERATION

5.1 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	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.

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.

5.2 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 (bright) 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 **Onnnn** from the keyboard and then the press SELECT PROG button.

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

5.3 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 AUTO POWER UP 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.

5.4 PROGRAM RESTART

The Program Restart function may be selected from the Setting page. It allows a program to be re-started 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.

5.5 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.

5.6 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.

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.

5.7 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.

6. OVERRIDE FUNCTIONS

6.1 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 five or 25% of maximum. If the **100%** rapid is too fast, it may be set to **50%** of maximum by Setting 10 on the Setting page.

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 "NOR-MAL" 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 "IG-NORE" 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.

6.2 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 func-tion 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.

7. AUXILIARY AXIS CONTROL

Besides the four directly controlled axes in this control, up to five additional external positioning axes may be added. These axes may be commanded directly from the program using the **B**, **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 B-axis feed at F30.0, this means that the B-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 "B 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.

In a control configured for five axes, the auxiliary axis **B** is not available. The first auxiliary axis is designated **C**.

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
B	5	Y
C	6	Z
U	1	U
V	2	V
W	3	W

Only four auxiliary axes are available with a control configured with a fifth axis.

Multiple auxiliary axes must be daisy chained through the second RS-232 port as described in the single axis control manual.

Auxiliary axes cannot be jogged from the CNC front panel. The single axis control jog button should be used for this. When any auxiliary axis is idle, the front panel JOG button for each auxiliary axis can be used to jog that axis.

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.

8. WHAT TO DO WHEN ALARMS OCCUR

8.1 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.

8.2 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.
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 changer 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	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	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. 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	Tool release piston is energized at power up. This is a possible fault in the air solenoids, relays on the IO Assembly, the draw bar assembly, or wiring.
131 Tool Not Clamped	Tool Release Piston is not Home. This is a possible fault in the air solenoids, relays on the IO Assembly, the draw bar 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.
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	Tool did not release from spindle when commanded. Check air pressure and solenoid circuit breaker CB4. Can also be caused by misadjustment

	of draw bar 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 tool. 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 Coolant	Coolant supply is below about five gallons or P7 is disconnected. To run without coolant, Setting 32 can be set to IGNORE.
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 Over Current or 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 Over Current or Drive Fault	same as 161.
163 Z-axis Over Current or Drive Fault	same as 161.
164 A-axis Over Current or 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 re leases. This will not turn the servos off but will stop the zero return operation.
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 4.5minutes 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 Spindle Coolant	Spindle coolant oil is low or low pressure condition in lines.
180 Tool Arm Rotation Fault	For Side Mount Tool Changer, the tool exchange operation did not sense the 180 degree rotation switch.
181 Tool Pot Position Fault	For Side Mount Tool Changer, the tool pot positioning mechanism is not working.
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.
196 Coolant Spigot Failure	Spigot failed to achieve commanded location after two (2) attempts.
197 Hours Unpaid Bill	
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.
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 occured in phasing initialization of brushless motor.
218 Y Axis Phasing Error	Same as above.
219 Z Axis Phasing Error	Same as above.
220 A Axis Phasing Error	Same as above.
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.
253 Program Data Error	Same as 249.
254 Program Data Error	Same as 249.
255 Program Data Error	Same as 249.
256 Program Data Error	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.
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.
308 Nest Full	Software Error; Call your dealer.
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 Code	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.
351 Buffered Block Range Error	Software error. Call your dealer.
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 VMC 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.
365	Cutter Comp Interference	G02 or G03 cut cannot be done with tool size.
366	Cutter Comp Interference	Tool doesn't fit inside of cut.
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	Inches 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 Not Enabled	An attempt was made to use a control feature not enabled by a parameter bit. Set the parameter bit to 1.
393 Invalid Motion In	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.
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.
429 "Undefined",	
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 reveiving 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.
434 "Floppy Abort"	Could not read disk.
435 "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.
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.
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.

9. 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.

■ 9.1 CREATING PROGRAMS

To create a new program, you must be in the PROGRAM DISPLAY and LIST PROG mode. Enter **O** (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 **Onnnn** 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 that when INSERT is pressed, the new data is put in after the highlighted (reverse video) 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 characters 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 **Onnnn** on the first line. If the maximum number of programs are already present, the message "DIR FULL" will de displayed and the program cannot be created. The maximum number of programs in memory is 200.

9.2 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 **Onnnn** 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 O0000 will be displayed. To change a program name, move the cursor to the existing **Onnnn**, 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 an EDIT key button.

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 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.

You can change to a different program while in the EDIT mode by using the CURSOR **up** and **down** keys, enter **Onnnn** on the input line and then press the CURSOR **up** and **down** keys or the **F4** key. **Onnnn** 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 turn this function on. 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. See Section 17 for a description of background editing.

Editing error messages:

Guarded Code .... You tried to remove the **Onnnn** 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 **Onnnn** 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.

9.3 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 OFSET 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 a program is not running, you are in EDIT mode, and you have entered **Onnnn** in the input line, pressing **F4** will change the currently–edited program to **Onnnn**. When in MEM mode and PROGRAM DISPLAY, this will select either BACKGROUND EDIT or PROGRAM REVIEW. You can enter BACKGROUND EDIT only when a program is running. 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.

9.4 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.

9.5 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 move 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 above-described manner.

10. 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.

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	0 to 65535
	82	LANGUAGE	English, German, French, Spanish
	1	AUTO POWER OFF TIMER	0 to 9999 minutes
	9	DIMENSIONING	INCH or METRIC
	77	SCALE INTEGER F	ON or OFF
	33	COORDINATE SYSTEM	FANUC or YASNAC
	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, HA5C, 7RT, 9RT, SRT, 11RT,160, 210, 310
	34	ROTARY AXIS DIAMETER	0 to 29.9999 inches
	78	5TH AXIS ENABLE	OFF, HA5C, 7RT, 9RT, SRT, 11RT
	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
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
	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, or 38400
12	PARITY SELECT	NONE, ODD, EVEN, ZERO
13	STOP BIT	1 or 2
14	SYNCHRONIZATION	XON/XOFF, RTS/CTS, or DC CODES
37	RS-232 DATA BITS	7 or 8
24	LEADER TO PUNCH	NONE, BLANK, or NULL
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 or RTS/CTS
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 PANL

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
84	TOOL OVERLOAD ACTION:	ALARM, FEEDHOLD,BEEP

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

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

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. The possible values include: 50, 110, 200, 300, 600, 1200, 2400, 4800, 7200, 9600, 19200, 38400.

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. It can be RTS/CTS or XON/XOFF. 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. 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.

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 9xxxx 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 NULL. 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
This is an On/Off setting. When it is **off**, the fourth 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.

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 either Yasnac or Fanuc. The Fanuc class of controls uses the G52 offset differently than the Yasnac class of controls. In a Fanuc control, the G52 offset will shift the work coordinates of all of the other offsets (G54 thru

G59). In a Yasnac control, the G52 offset stands on its own as another work offset. In both cases, G92 can be used to set the G52 offset from the CNC program.

34 ROTARY 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 over-shoot 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. It can be RTS/CTS or XON/XOFF. 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.

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**, 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.

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. It can be set to either Yasnac or Fanuc. The two types are similar to the method of cutter compensation available in those two 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 setting changes the way the TOOL OFSET 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.
Total Y travel = Parameter 20 / Parameter 19 (16.25 VF-0 thru VF-2)
Scale = Total Y travel / Setting 65

66 GRAPHICS X OFFSET
This setting locates the right side of the scaling window relative to the machine X zero position, (See Section 11.9). 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 Section 11.9). Its default is zero.

68 GRAPHICS Z OFFSET
Reserved for future use.

69 DPRNT LEADING SPACES
This setting 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 ;	Setting 69:	OFF	ON
G0 G90 X#1 ;	OUTPUT:	X3.0000	X 3.0000
DPRNT[X#1[44]] ;			

The default value is OFF.

70 DPRNT OPEN/CLOS DCODE
This setting controls whether the POPEN and PCLOS statements in macros send DC control codes to the serial port. When the setting is ON, these statements 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 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 **O** number of 9000 or above. When the setting is OFF, the control will not display 9000 series blocks. The default setting is ON.

75 9xxxx 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
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.

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. The languages currently available are English, German, French, and Spanish. If the language selected does not reside in the control, NOT AVAILABLE will be displayed in the message area when that language is selected.

Settings may be sent and received with the RS-232 port. See Section 13 for a description of how to do this.

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.

86 M39 LOCKOUT
Locks out the rotation of the tool changer.

87 M06 RESETS OVERRIDE
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 canceled and set to their programmed values.

11. DISPLAYS

You can select any of the following displays using the eight DISPLAY select buttons:

PRGRM	To show or edit 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 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 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. Note, if BACKGROUND EDIT is available in your machine, it is selected from the PROGRAM DISPLAY.

11.1 CRT DISPLAYS

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 things displayed on the screen are:

The present selected display in top left corner,
The present selected mode in parentheses,
The presently selected program in top right corner,
The most recent line number in top right corner,
Up to 18 lines of variable display data,

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 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.

Error response status if button pressed is not allowed. The following are in response to operator action.

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 DRY RUN 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.

There are several more responses made only 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 to the above, the CRT display can show one of eight types of data in the 18 lines of variable display. They are:

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 show 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.

11.2 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.

While you are running a program, the PROGRAM REVIEW function is available. 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 right hand display to a different part of the program. The left side display 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 **Onnnn** 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.

11.3 POSITION DISPLAYS

There are five position displays in this control:

Machine display:
This display is the machine coordinate system that is automatically set upon power up and the first ZERO RET. It cannot be changed by the operator or any work coordinate systems and will always tell you how far from machine zero you are. It can be used by a non modal G53 command.

Work display:
This display tells you how far your tool is away from your **X**, **Y** and **Z** zero of your programmed part. Upon power up, it will display the value in work offset G54 automatically. It can only be changed by G54 thru G59, G110 thru G129 or by a G92 command. The machine uses this coordinate system to run your part.

Distance to go:
This display is an incremental display that tells you the travel distance remaining to go before the axes stop. When the mode is in ZERO RET, 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.

Operator display:
This display is for the operator/setup person to use as desired. It can be Origin'ed out as needed to show position relative to where you zero it. It is not used by the control for any positioning functions. The ORIGIN button is used to set the zero position when the Operator position display is selected and the control is in JOG mode.

A fifth display shows all of the above four in small characters. All of the others are displayed 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.

11.4 HELP 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.

11.5 CALCULATOR FUNCTION

The calculator function is selected by pressing the HELP key a second time. There are three calculator pages. Trig Help, Circular Help, and Milling/Tapping Help. All of these have a simple calculator and an equation solver. The **Fn** keys also allow moving of data from other displays to/from the calculator. Trig Help, Circular, and Milling Help are selected using the PAGE UP and PAGE DOWN keys.

All of the Calculator Help functions have a simple calculator for simple add, subtract, multiply, and divide operations. The operation being performed is selected with the **left** and **right** cursor arrows. The value being operated is typed on the bottom of the screen and the WRITE button is used to perform the operation.

The calculator functions only work when the calculator value window is highlighted. The CURSOR **up** and **down** buttons are used to select the Trig, Circular, or Milling data values for entry with the WRITE button.

11.6 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.

11.7 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 push 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.

11.8 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.

11.9 GRAPHIC DISPLAY FUNCTION

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 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.

■ 11.10 CURRENT COMMAND DISPLAY

The Current Commands Display provides four types of displays. The PAGE UP and PAGE DOWN keys are used to select among:

Program Command Check,
Current Display Command,
Operation Timers,
Tool Life Timers, and
Tool Load Monitor.

The Program Command Check display will show both the programmed speed for the spindle (the last **Snnnn** command) and the actual speed command being sent to the spindle. This is shown in the Program Command Check page of the Current Commands. To get to this page, use the PAGE UP or PAGE DOWN keys. In addition to the speed, this page shows the CW, CCW, or stopped command being sent to the spindle and the current gear position. The Program Check also displays the axes' positions. The coordinates displayed are selected from the POSIT display using PAGE UP and PAGE DOWN. If the spigot is enabled, the current spigot position is displayed.

■ 11.11 RUN HOURS AND PARTS NUMBER DISPLAY

The running hour displays are selected by PAGE UP or PAGE DOWN from the CURNT COMDS display. You may use the ORIGIN key to zero any of the three displays of on-time, cycle start time, and feed time. The CURSOR **up** and **down** buttons move the highlighted title and the ORIGIN button will zero the selected item.

Listed below the run hour display 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.

11.12 LEAVING MESSAGES

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.

To enter messages, press the ALARM MESGS 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.

11.13 TOOL LIFE

A Tool Life screen is found in the CURNT COMDS display using the PAGE DOWN key. The Tool Life screen maintains the time the tool is in feed, the time the tool is selected, and the number of times the tool has been used. This can be used by the operator or setup person to assist in predicting tool life. The displays can be zeroed out by using the CURSOR and the ORIGIN keys. Put the cursor on the title line to origin all of the data in that column.

The Tool Life screen may also be used to generate an alarm when a specified number of tool uses has been reached. 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.

11.14 TOOL OFFSETS

Tool length and part zero offsets are entered in the offsets page. There are 50 **H** and **D** codes possible. You can use the same offset number for both Z-axis offset (H1) and tool diameter compensation (D1) because the offset contains separate values for each. 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 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 **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.

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 are also entered automatically when using the TOOL OFSET MESUR key during setup procedures. Note that this automatic offset measurement works with G43 only.

A parameter in the setting page 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. This is Setting 15.

PAGE DOWN in the OFSET display will go through all 50 possible tool offsets and then change to the work zero offsets. Work zero offsets may be entered and displayed from this page or using the PART ZERO SET key.

When entering offsets, pressing WRITE will cause the new value to be added to the old value. Pressing **F1** will set the value to your entry. This allows small adjustments to the offsets. Note that a negative entry and WRITE will decrease the value of the offset.

Tool diameter may be entered as either radius or diameter. Setting 40 on the setting page 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. 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. See Section 13 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:

```
O0001               (Sample coolant positioning)
G90  G54
T1   H1   M06
M08                 (Moves spigot to H1 position)
.
.
.
T2   H2   M06
M08                 (Moves spigot to H2 position)
.
.
.
H42   M08           (Position to H42 position)
                    (Note that the length and radius offsets should be the same as Tool 2)
.
.
.
M30
```

11.15 TOOL LOAD MONITOR AND DISPLAY

This machine has a standard tool load monitor and display function. With the tool load display, the operator may enter a maximum load that is expected for each tool. When this load is exceeded in a feed, an alarm will be generated. The tool load display provides for the entry of this alarm point and it also displays the largest load that tool has seen in any previous feed. The tool load display also shows the present tool number and the present tool load. The tool load display is selected with the PAGE UP or PAGE DOWN keys in CURNT COMDS display.

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 and an alarm is generated if the limit is exceeded. When the spindle is up to speed, you are in a feed motion, and the load limit is exceeded, Alarm 174 is generated. This alarm will stop the axis motors, stop the spindle motor, turn off the coolant, and disable the servos.

11.16 PARAMETER 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 SWTCH	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.

11.17 DIAGNOSTIC DATA DISPLAY

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.

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 **Service Manual** for a description of the diagnostic page inputs.

11.18 SETTINGS DISPLAY

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; See section 10 for a more detailed description.

11.19 ALARMS DISPLAY

The Alarms display can be selected at any time by pressing the ALARM MESGS 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.

■ 11.20 MESSAGE DISPLAY

The Message display can be selected at any time by pressing the ALARM MESGS 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.

12. 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 with an explanation of each is provided in the Parameters Section of the Operators Manual.

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.

■ 12.1 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.

■ 12.2 SPINDLE HEAD THERMAL COMPENSATION

This control is equipped with a compensation function to correct for thermal expansion of the spindle head. This consists of a temperature sensor mounted on the spindle head and software to compensate for this expansion in the **Y** and **Z** axes directions. There are two parameters that specify the coefficient of expansion of the spindle and this will, in turn, shift the position of the **Y** and **Z** axes to correct for the growth of the spindle head. These are Parameters 132 and 133.

The diagnostic data display also displays the temperature of the spindle head. This compensation is valid over a temperature range of 50F to 140F.

13. DATA INPUT / OUTPUT TO / FROM COMPUTER / READER / PUNCH

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 **O** for a program name, but it is always displayed as **O**.

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. Transmission can be stopped with either the XON/XOFF characters or the RTS/CTS wires.

Parameters, settings, and offsets 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 settings, offsets, and parameters 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.

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.

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, 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 Section 15 for details.

14. 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*, 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 (OFSET, PARAM, SETNG), 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:

```
%
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.

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 will 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 Oxxxx 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.

14.1 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 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 %.

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.

15. 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.

16. 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 Operation Section 7 (Auxiliary Axis Control). 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	300	30	Quick and easy to use for light loads only.
HA5C2,HA5C4	250	30	HA5C2 fits along Y and X axes; 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	80/60	100	Fits VF-3 and VF-4 with some restrictions in tool length.
HRT160	80	100	
HRT 210	70	160	
HRT 310	60	150	
HRT 450	60	150	

17.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 interface. Note, that this is the first serial port or the top connector. Do not confuse DNC with RS-232 uploading and downloading which is described in section 13.

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 RS-232 data rate available is 38400 bits per second. 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.

18. 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 **Onnnn** for the program you want to edit and pressing F4. If you do not enter the **Onnnn**, 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 new display function has been added 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.

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

19. AUTOMATIC 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.

19.1 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.

19.2 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.

19.3 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 legnth, width and size of the part.

19.4 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

19.5 M CODE SUMMARY

M34 Coolant Spigot Increment Position
M35 Coolant Spigot Decrement Position

19.6 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.

20. 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.

20.1 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.

20.2 CONVEYOR PROGRAM COMMANDS

Use M codes M31, M32 and M33 to control the conveyor from within a program or in MDI. Briefly M31 commands the conveyor forward, M32 commands the conveyor in reverse and M33 stops the conveyor. Refer to section 9.2 for a detailed description of these M codes.

20.3 CONVEYOR PARAMETERS

The parameters that control the conveyor are below. Refer to the following page for a complete description of their usage.

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.

20.4 M CODE SUMMARY

M31	Chip Conveyor Forward
M32	Chip Conveyor Backward
M33	Chip Conveyor Stop

20.5 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.

❖❖❖❖ End Of Programming and Operation Manual ❖❖❖❖

Index

A

B

C

D

E

F

G

H

I

J

K

L

M

N

O

P

R

S

T

U

V

W

X

Z

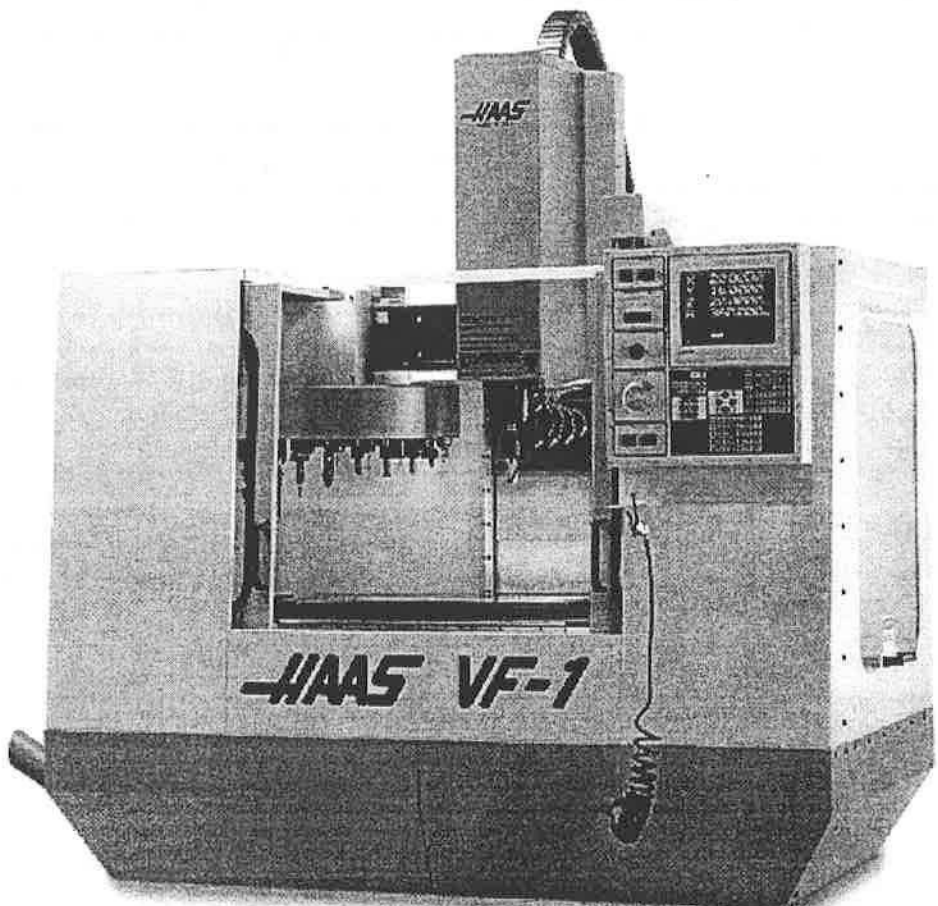
MAINTENANCE SCHEDULE AND LUBRICATION CHART FOR THE VF-SERIES

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.

INTERVAL	MAINTENANCE PERFORMED
DAILY	✓ 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	✓ Check automatic dump air line's water trap for proper operation. ✓ Check air gauge/regulator for 85 psi. ✓ Check aluminum air filters on control heat exchanger and at top of spindle motor. ✓ Clean exterior surfaces with mild cleaner. DO NOT use solvents.
MONTHLY	✓ Inspect way covers for proper operation and lubricate with light oil, if necessary. ✓ Clean the upper screen on the coolant tank. Remove the middle plate on the tank and remove any sediment inside the tank and clean the inlet filter to the rotary pump. Be careful to disconnect the coolant pump from the controller to POWER OFF the control before working on the coolant tank. ✓ 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. ✓ Place a dab of grease on the V-flange of tools.
SIX MONTHS	✓ Replace coolant and thoroughly clean the coolant tank. ✓ Check oil level in gear box by adding oil until it begins dripping from the drain tube. ✓ Lubricate counterweight chains with Mobil Vactra #2 over full legnth of chain. ✓ Inspect chain for any abnormal wear or cracks.
ANNUALLY	✓ 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. Slowly refill with oil until it overflows at bottom of the heastock. ✓ Check way lube covers; oil if necessary. ✓ Check oil filter and clean out residue at bottom of filter.

SYSTEM:	WAY LUBE AND PNEUMATICS	TRANSMISSION	COOLANT TANK
LOCATION	Under the control panel at the rear of the machine	Above the spindle head	Front 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	30 gallons
LUBRICANT	Mobil Vactra #2	Mobil DTE 25	Water soluble, synthetic

HAAS AUTOMATION, INC.



INSTALLATION MANUAL FOR THE VF-0, VF-1, VF-2

TABLE OF CONTENTS

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 AUTO POWER UP 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 instructions and in 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.

Never reach through the holes in the vertical column. The large counterweight inside can cause severe injury as it moves up and down.

KEY SPECIFICATIONS	VF-0	VF-1	VF-2
Installation Height (in.)	104	104	104
Installation Width (in.)	84	84	98
Installation Depth (in.)	102	102	102
Shipping Weight (lb.)	7,000	7,000	7,500
Machine Height (in.)	106	106	106
Machine Width (in.)	76	76	90
Machine Depth (in.)	86	86	86
Machine Weight (lb.)	6,500	6,500	7,000

SERVICE REQUIREMENTS

ELECTRICITY

The VMC runs on a three-phase, 195-260 V, 50- or 60-Hz supply. Power supplied to the machine should not fluctuate more than ± 5%.

The power required is 25 amps.

If the service run from the electrical panel is less than 120', use No. 10 wire; if the run is longer, use No. 8 wire.

A separate earth, or cold-water-pipe, ground is required (a conduit-type ground does not provide sufficient noise immu nity).

Do not connect to 480 V under any circumstances!

WARNING!

Maximum voltage leg-to-leg or leg-to-ground
should not exceed 260 volts!

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.** 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

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.

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. See Appendix B (Service Notes) for instructions on how to clean this element.

The air must be supplied through a minimum3/8" hose and fittings must be at least ¼" NPT.

NOTE: The nipple between the air filter/regulator and the Bijur oil lubricator (See illustration at left center of page 6) 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 ¾" hex wrenches (one open end or box and one ratchet).
✓ A forklift that is capable of lifting more than 7,500 pounds. The forks must be at least 5' long by 6" wide.
✓ Claw hammer.

CAUTION THE VMC CRATE CAN ONLY BE MOVED BY A FORK LIFT.

MATERIALS REQUIRED

✓ Wire and air hose or piping as specified in the Service Requirements section.
✓ 20/30 gallons of water-soluble synthetic, or cutting oil coolant.
✓ A small amount of grease.
✓ Way lube for the lubricator (Vactra #2).

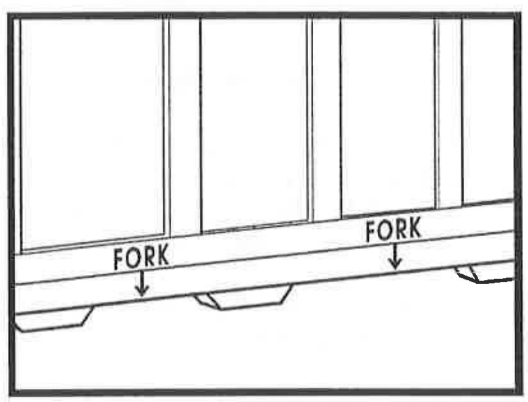
MOVING THE CRATE

CAUTION! The fork positions are marked on the sides of 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.



1. Pry off the clips around the top of the crate with a claw hammer and remove the top panel.

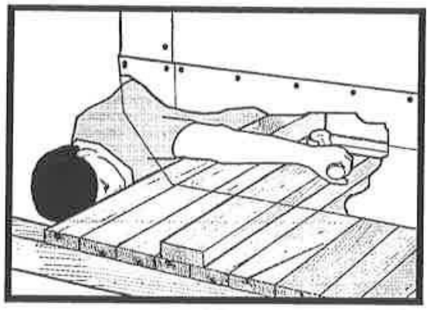
2. Pry off all but one clip at each corner of the crate.

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. Carefully remove the staples that fasten the plastic cover to the pallet. 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 system) and the red tool box on the left side of the machine.



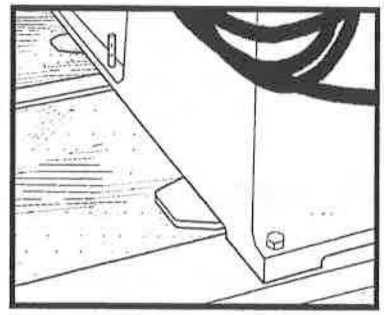
Remove the cleats that held them in place.

7. Slide the drip pan out from under the machine, or, pull the drain bucket out of the rear of the machine, whichever is applicable.

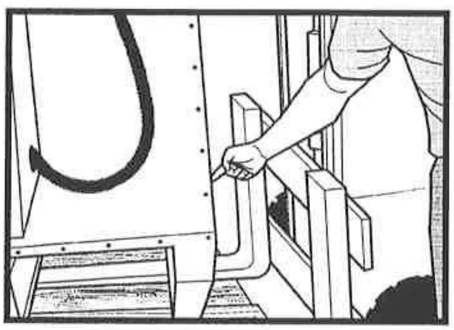
8. Remove the touch-up paint that is taped to the gutter of the table.

9. Using ¾" wrenches, remove the nuts (under the pallet) from the retaining bolts at the four corners of the machine. (A ratchet on top and a wrench on the bottom is the easiest way to do this.) Leave the bolts in the holes after the nuts are removed.

Setting in Place



CAUTION! Much of the weight of the VMC is concentrated in the column at the back. Therefore, it is imperative that the forks of a forklift be positioned as far to the back of the machine as possible with the fork at the back of the machine located next to the pads. Also, when moving the machine, do not lift it any farther than necessary off the floor and move as slowly and cautiously as possible. Dropping the machine, even from a height of a few inches, not only has the potential to cause injury, it also results in expensive repairs and voids the warranty.



WARNING! The only acceptable way to move a VMC 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 27" apart (measured inside to inside of the forks). 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 a foot clearance between the forklift and the side of the machine. Attempting to move the machine any other way may void the warranty.

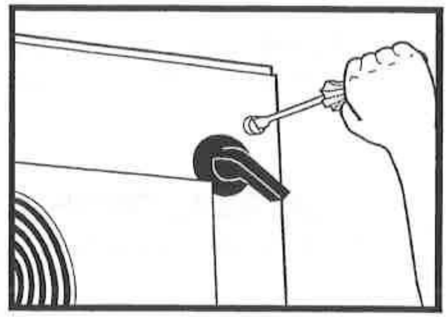
1. Lift the machine until the bolts clear the pallet. Pull the bolts out of the holes in the machine.

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 ¾-10 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.

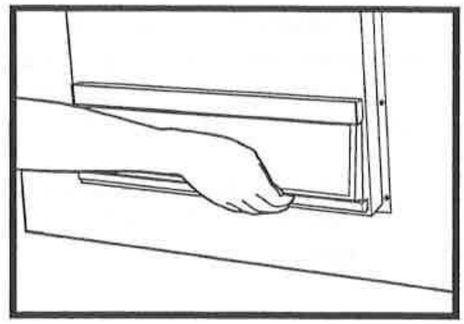
INITIAL SETUP



WARNING! At this point, there should be NO electrical connection to the machine. The electrical panel should be closed and the three screws 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, loosen the three screws on the panel door and open the door.



3. The manuals and air filter screen are located at the bottom left of the panel.

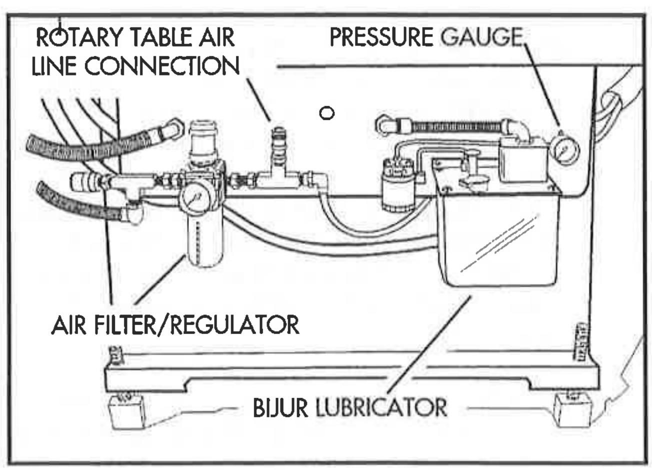
4. Install the air filter toward the bottom on the outside of the panel door (push it up into the top mounting bracket and then push in the bottom edge).

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.

1. The air filter is empty, and the air lubricator and the lubricator reservoir tank are full when the machine leaves the factory. 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. Prime the Bijur lubricator to make sure it is working.

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.

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. When you pull up on the handle, you can see the pressure increase on the pressure gauge.

ELECTRICAL CONNECTIONS

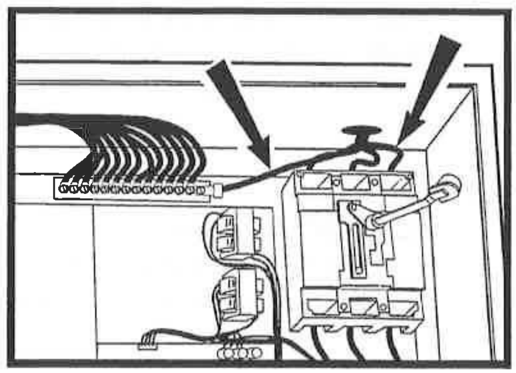
NOTE: The machine must have air pressure at the air gauge or an interlock will prevent it from powering up.

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 screws 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.

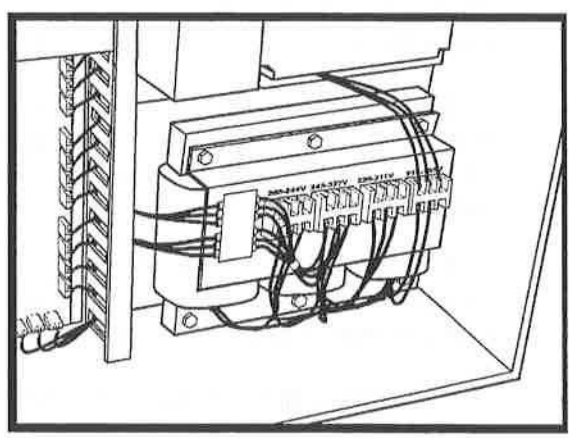
1. Hook up the three power lines to the terminals on top of the main switch at upper right of electrical panel and the separate ground line to the ground bus to the left of the terminals. It is not necessary to be concerned with phase rotation (which wire is connected to L1, L2, and L3).

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.

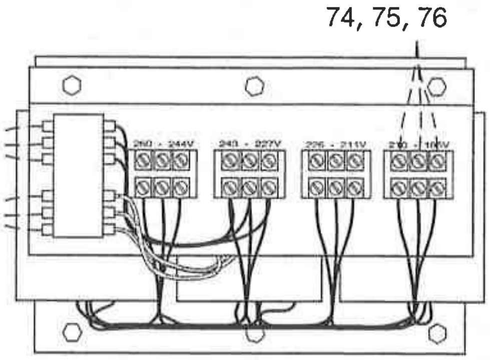
NOTE: wide voltage fluctuations are common in many industrial areas; you need to know the minimum and maximum voltage which will be supplied to the machine while it is in operation. U.S. National Electrical Code specifies that machines should operate with a variation of +5% to -5% around an average supply 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.

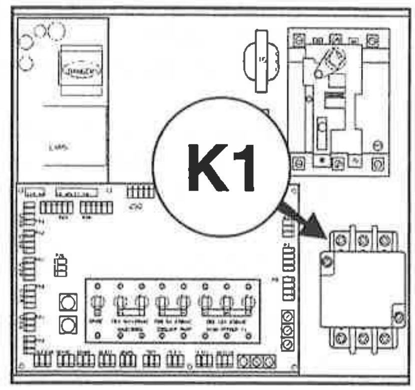
3. Check the connections on the transformer at the bottom-right corner of the rear cabinet. The three black wires labeled **74, 75,** and **76** must be moved to the terminal block triple which corresponds to the average voltage measured in **step 2** above. There are four positions for the input power to this transformer. The input voltage range for each terminal block is as follows:

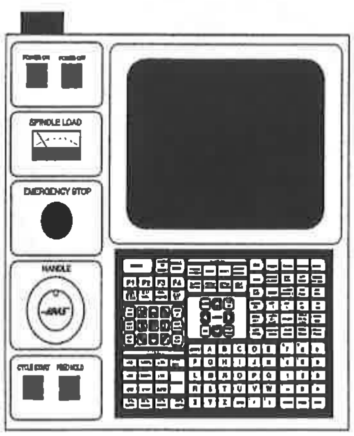
195 to 210	right side
211 to 226	right center
227 to 243	left center
244 to 260	left side



4. Set the main switch to ON (rotate the shaft that engages the handle on the panel door clockwise until it snaps intothe 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.

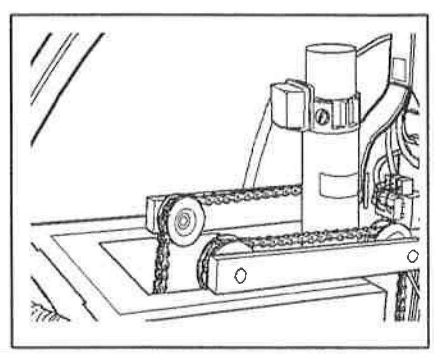
7. Turn off the power (rotate the shaft that engages the handle on the panel door counterclockwise until it snaps into the OFF position). Also, set the main switch handle on the panel door to OFF. (Both the handle and the switch must be set to OFF before the door can be closed). Close the door, screw the screws into place, and turn the power back on.

START UP

CAUTION! Do NOT press AUTO POWER UP on the control panel while the wooden head support is under the spindle.

1. Check at the top of the column to make sure that the counterweight chains are properly set on the sprockets.

2. 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.

3. 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 you start the machine for normal operation BUT NOT FOR THIS START-UP ROUTINE.)

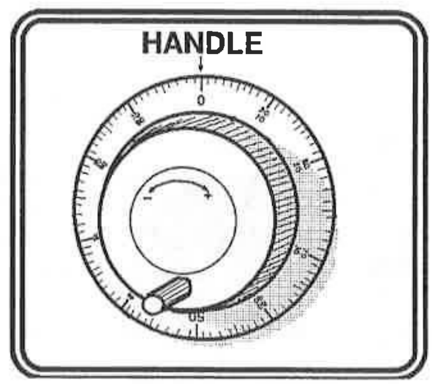
4.Press and release the RESET button twice, or until you have no alarms, to turn the servos on. (The message "ALARM" appears at the lower right of the screen if one or more alarms are in effect.)

NOTE: Some alarms, such as low way lube or low air pressure, cannot be cleared with the RESET button. If you cannot clear the alarms, press and release the ALARM MESGS button for more information on the alarms. If you are unable toc l e a r the alarms, copy down the alarm numbers and call the factory.

5. 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 "JOGGING¼" message shows the X- or Y-axis instead of Z, press and release the +Z button.

CAUTION! Do not touch the -Z button while the wooden head support is located under the spindle.

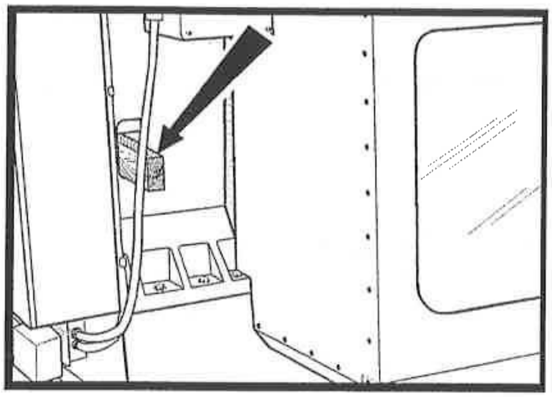


6. Turn the HANDLE clockwise (in the positive direction) to raise the head off the wooden head support.

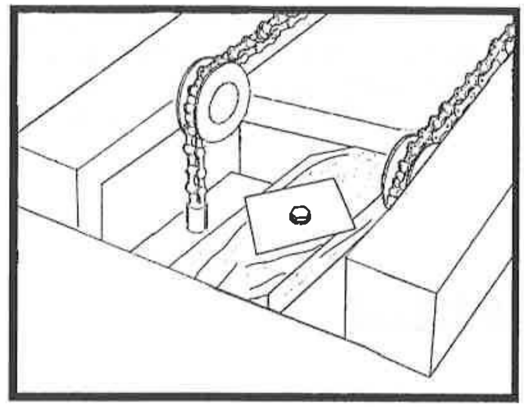
CAUTION! Do not turn the handle counterclockwise while the wooden head support is located under the spindle.

7. Remove the head support and the cardboard.

8. Turn the HANDLE counterclockwise to move the head down a couple of inches (until the counterweight lifts off the counterweight support — the two-by-six that extends through the column).



9. Remove the counterweight support and the counterweight stabilizer (the wooden piece at the top-front of the counterweight) — lift it out of the top of the column. If it is hard to get the stabilizer out, inspect the counterweight for misalignment and the chains to make sure that they are on the sprockets and operating correctly. If there is a problem, lower the head (with the HANDLE), put the counterweight support back through the column, raise the head (with the HANDLE) to lower the counterweight onto the support, and set the chains on the sprockets, etc. Then proceed as before to remove the counterweight support.

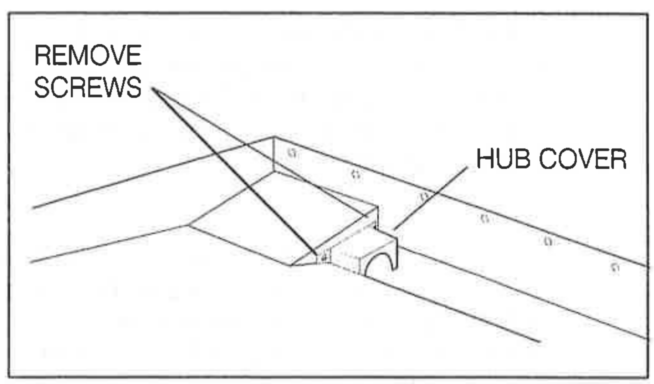


10. Once you are certain that the Z-axis is working correctly (that it operates smoothly and there are no strange noises, etc.), make sure that all alarms are clear — check for the "ALARM" message at the lower right of the screen. (See item 4 in this section for information on alarms.) Next, close the 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.

11. The machine is now ready for leveling.

CHIP CONVEYOR INSTALLATION



1. Unpack the conveyor and drain tube and remove the motor hub cover.

2. Install gasket and slide the discharge tube up and onto studs. Attach the eight nuts with locking washers and tighten uniformly.

3. Slide the conveyor into the discharge tube and then slip opposite end onto motor hub. Fasten to motor hub with the 5/16-18 x 2½" bolt.

4. Replace motor hub cover.

ELECTRICAL CONNECTION

The following instructions are for connecting the power supply to machines equipped with the optional chip conveyor. Please read instructions entirely before connecting.

1. Connect the power supply to the machine as shown in the Installation Manual

2. Check the operation to 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 1-0 or 0-1 so that it establishes a new forward direction.

MAINTENANCE

During normal operation, most chips are discharged from machine at the discharge tube. Very small chips may flow through the drain and collect in the chip trap tray located in the coolant tank. To prevent drain blockage, clean trap regularly. Should drain become clogged and coolant collects in the machine's pan, stop the machine, loosen the chips blocking the drain and allow the coolant to drain. Empty the chip trap from the coolant tank, then resume operation.

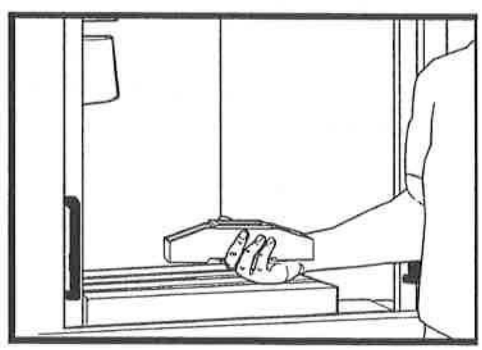
LEVELING

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 level** 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.

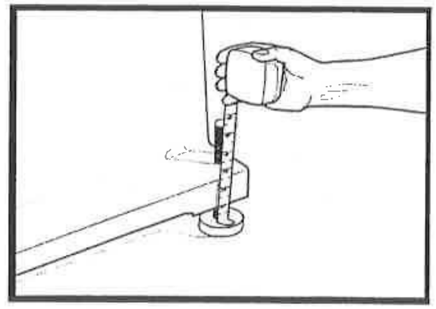
Use a precision bubble level with each division equal to **0.0005** inch per **10** inches, or **.05**mm per meter, or **10** seconds per division. Before starting, check the accuracy of your level. Set it on the table on the X-axis and mentally record the reading. Then turn it **180°** and the reading should be the same. If it is not, the level is out of calibration andshould be adjusted before you continue.



1. Press and release SETNG GRAPH. Then page down to the last page (press and release PAGE DOWN several times). Cursor to Setting **51**, DOOR HOLD OVERRIDE (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. This bypasses the door interlock so you can operate the machine with the doors open.

NOTE: This setting automatically resets to off when the machine is powered up again. DO NOT disconnect the door interlock switches (this makes you liable for any damage or injury).

ROUGH LEVELING --



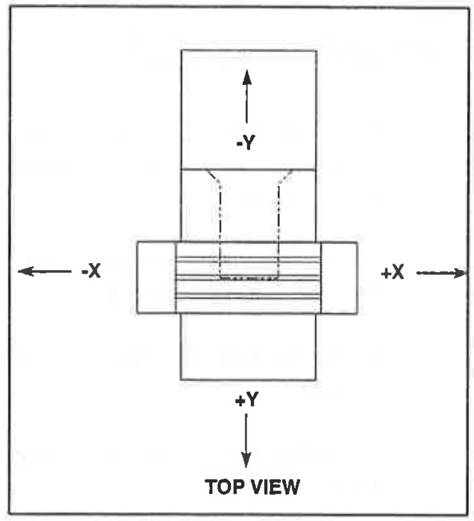
2. Screw the four leveling screws at the corners through the base (use the 3/8" allen wrench in the tool box) until the base is 2½" to 3" above the floor. That translates into a minimum of one inch of the leveling screw extending out of the bottom of the base of the machine, or one inch between the pads 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).

NOTE: The two center leveling screws should not be touching anything.

3. Use Handle Jog set for **0.01** on the X- and Y- axes for the leveling procedure. (See the Start Up 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 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.

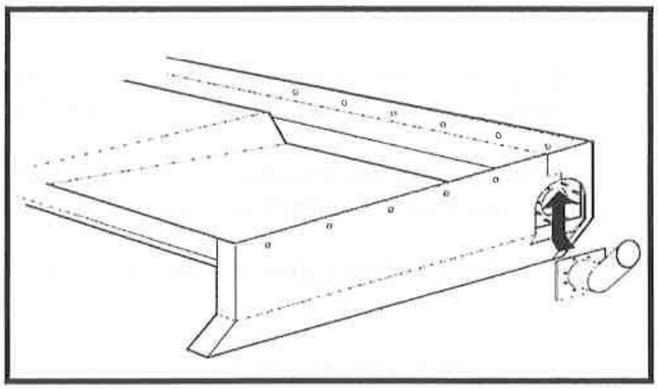
IMPORTANT!!!

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 as was 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.

COOLANT SYSTEM

1. Connect the coolant tank power line to the connection on the right side of the electrical panel. A plaque above the connections specifies where the connection is made.



2. Connect the loose hose on the left side of the column to the hose fitting on the pump on top of the coolant tank.

3. Fill the tank with the appropriate amount of coolant 30 gallons

4. Position the coolant tank against the left side of the machine.

5. Place the drip pan under the machine or place the bucket under the drain pipe at the rear of the machine.

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 ¾" wrenches (one open end or box ,and one ratchet).
✓ A forklift that is capable of lifting more than ,7,500 pounds. The forks must be at least 5' 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 (7' x 7').
✓ 6-mil plastic cover (18' x 6" x 20').
✓ Coolant box (cardboard).
✓ Wire ties.
✓ Foam pads.
✓ Fiber banding.
✓ ½-13 x 10" bolts (4).
✓ ½" washers (4).
✓ ½-13 Nylock nuts (4).
✓ 16d nails as required.
✓ Wooden supports as specified in Appendix D.

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

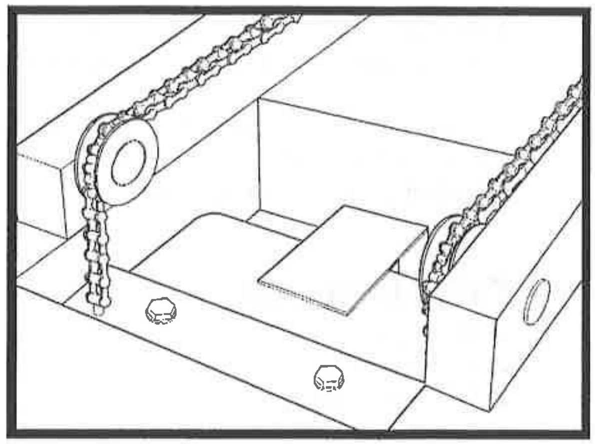
1.Remove any tools from the tool carousel and spindle.

2. Spray all metal parts around the way covers, table, spindle (including up into the spindle), with rust preventive oil. (See the cautionary notes in the Cleaning section of the Maintenance Briefs.) After spraying, clean the interior of the enclosure, including the windows, as necessary.

3. In Jog mode, use the HANDLE to center the table under the spindle, and move the head practically all the way down. This is approximately X = 10, Y = 8, and Z = 13.

4. Place the counterweight support (a 2' long 2" x 4") through the access holes at either side of the column to support counterweight. (The support must extend out of the holes on either side of the column but not more that 2" on the left side of the column.)

5. Slip the counterweight stabilizer into the top of the column on the front side of the counterweight (with the lip pointed toward the back of the machine). This keeps the counterweight from rocking back and forth during shipment.

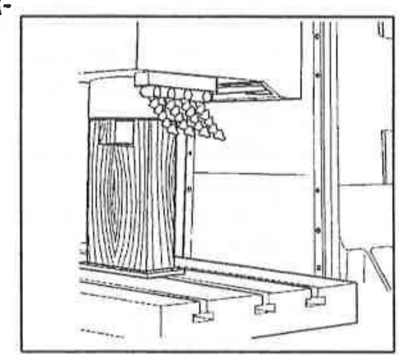


6. Using the HANDLE, raise the head (Z-axis) far enough so the counterweight rests on the counterweight support and the head support can be inserted between the spindle and the table (about 10" of clearance between the table and the head).
NOTE: You can hear a slight thump as the counterweight comes to rest on the support.

7. Set the head support on a piece of cardboard (to protect the table against scratching) in the center of the table. Position the support so the spindle housing is resting on it and NOT the spindle shaft.

NEVER PUT ANY SUPPORT ON THE SPINDLE!! THIS WILL DESTROY THE SPINDLE BEARING!!!

8. Use the HANDLE to bring the head down until it rests on the wood. Then press EMERGENCY STOP and the machine is locked up for shipping.



9. Power down (press the POWER OFF button at upper left of control).

10. Tape the touch-up paint to the gutter of the table in the front of the machine.

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 screws 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.

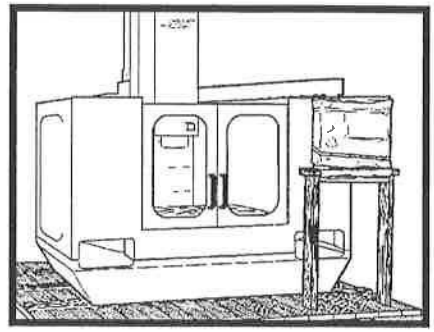
6. Remove the air-filter screen mounted toward the bottom on the outside of the panel door (lift it up and then pull the bottom edge out). Clean it (with compressed air) if necessary.

7. Place the air-filter screen, along with the manuals and other documentation, inside the panel at the bottom left.

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 tighten the screws.

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. For new machines, make sure that the warranty certificate is positioned against the screen on the control panel.

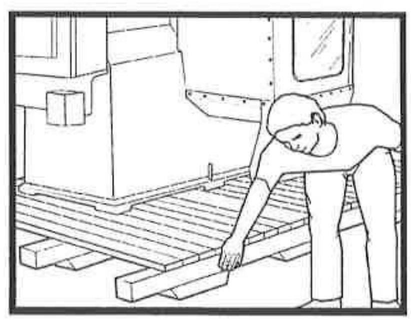
4. Wrap some padded packing material around the pendant (control box) and secure with some heavy-duty tape. Also, place a couple inches thickness of foam or other packing material on the enclosure behind the control panel.

5. Wrap some padded packing material around the pendant support's crossmember (See Appendix E) and place the support in position as shown.

6. Nail the pendant support to the pallet to secure. Once the support is secured, slip packing material behind the pendant wherever it comes into contact with the machine. This not only protects the machine but also makes the support more sturdy and the control panel more secure.

MOVE ONTO PALLET

NOTE: If you did not store your shipping materials, pallets and crates are available from Haas Automation. Contact the factory for information.



WARNING! The only acceptable way to move a VMC 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 27" apart (measured inside to inside of the forks). The forks must be positioned all the way to the back of the machine and they must extend at least 3" past the far side of the machine base. Also, there must be about a foot clearance between the forklift and the side of the machine. Attempting to move the machine any other way may void the warranty.

CAUTION! Much of the weight of the VMC is concentrated in the column at the back. Therefore, it is imperative that the forks of a forklift be positioned as far to the back of the machine as possible with the fork at the back of the machine located next to the pads. Also, when moving the machine, do not lift it any farther than necessary off the floor and move as slowly and cautiously as possible. Dropping the machine, even from a height of a few inches, not only has the potential to cause injury, it also results in expensive repairs and voids the warranty.

1. Lift the machine a few inches off the floor.

2. Remove the leveling screws at all four corners.

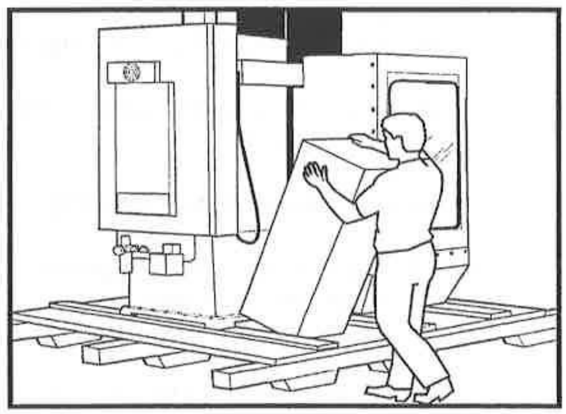
3. Insert 1/2-13 x 10" retaining bolts with washers in the leveling screw holes at all four corners of the machine. (These are secured on the pallet with 1/2-13 Nylock nuts). Putting the bolts in the machine at this time provides a guide for lining up with the holes in the pallet.

4. Cover the pallet with a 6-mil, 7' square, contiguous plastic sheet.

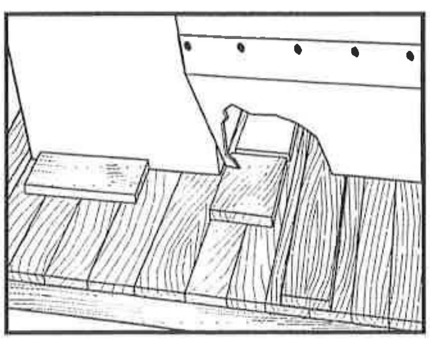
NOTE: This and the other plastic that is added later provide a shield against moisture while the machine is in transit. HOWEVER, the plastic wrapping is not 100 percent waterproof and the crate must be covered with waterproof tarpaulins any time there is a chance it can be exposed to rain, splashing from wet roads, or other conditions where it could get wet.

5. Move the machine over the pallet and align the bolts with the holes in the pallet. **NOTE:** You can see three supports at each end of the pallet. The center support is offset toward one side. The machine must be placed on the pallet with the rear of the machine on this side of the pallet.
(**NOTE:** The plastic sheet was left off pallet for some of the illustrations so the holes and other details would be visible.)

6. Block the two retaining bolts at the front of the machine with short lengths (about 2') of two-by-four so they cannot slip back out of the machine. Then, as the machine is lowered onto the pallet, make sure that the retaining bolts slip into the holes in the pallet. (Move the pallet as necessary. Also, the two bolts at the back can be driven into the holes.)



7. 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.



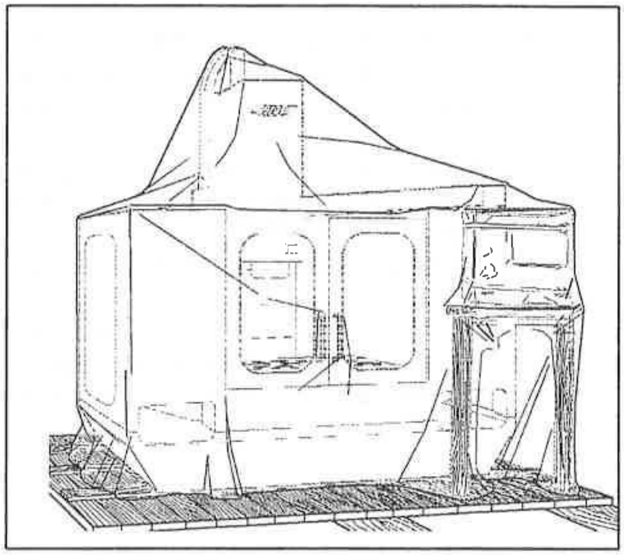
8. Put the leveling screws, pads, allen wrench, and screwdriver in the tool box and position it between the coolant box and the support strip toward the front of the machine. Then nail a 12", two-by-four cleat to the pallet with 16d nails to hold the tool box in place.

9. 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

1. Cover the machine with a sheet of 20' W, 6-mil plastic cut to a length of 18'-6". 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 (See the illustration page 4). **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 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
✓ ¾" Wrench (Open-end or Box-end)
✓ A forklift capable of lifting more than 7,000 pounds. The forks must be at least 5' long and 6" wide.

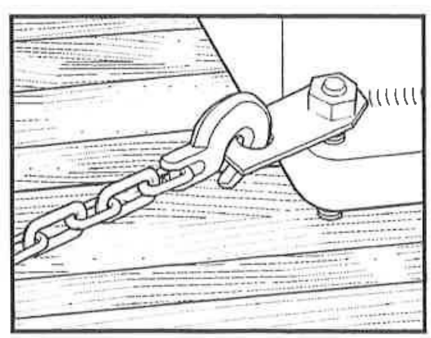
MATERIALS REQUIRED

✓ Four metal tie-downs.
✓ Four ¾-11 nuts (in addition to the four nuts that come with the machine).
✓ Two 4 x 4's at least 3' long.

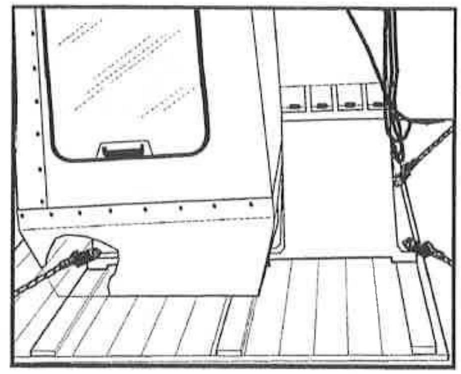
PROCEDURE

1. Prepare the machine as described in Appendix C up to the point of putting it on a pallet. Make sure the spindle is on tool number one. Secure the counterweight and support the spindle -- even for the shortest moves. (An unsecured counterweight can cause severe damage to the ball screw in the Z-axis.) 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 a protrude at least ½" through the bottom of the machine. Tighten the ¾-11 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 ¾-11 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 ¾-11 nuts.

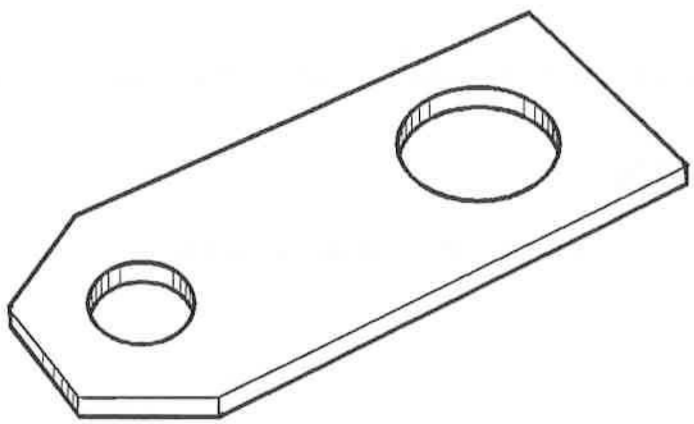


3. Place two 4 x 4's that are least 3' long on the truck. Lower the VMC onto the truck and align the two pieces of lumber under the leveling screws at the corners of the machine. Remove the forklift and have the rigger secure the machine using the large holes in the four tie-downs as the attachment points for the riggers' chains. When the chains are tightened, the leveling screws dig into the4 x 4's to help hold the machine in place.



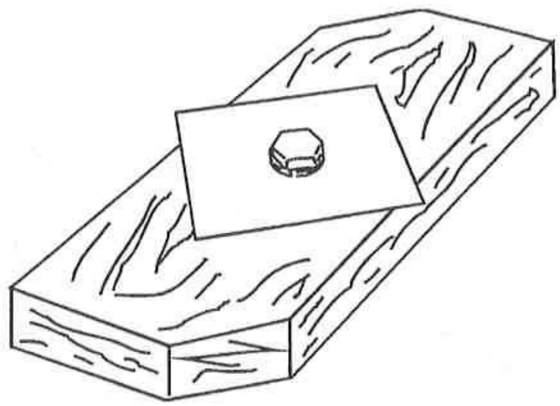
4. Using the four corner tie-downs, the machine can be secured without dragging chains across its top. It is a fast, easy, and secure transportation method that assures against damage. Be sure to explain to the rigger that this is how you want the machine to be moved BEFORE he does something else.

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.

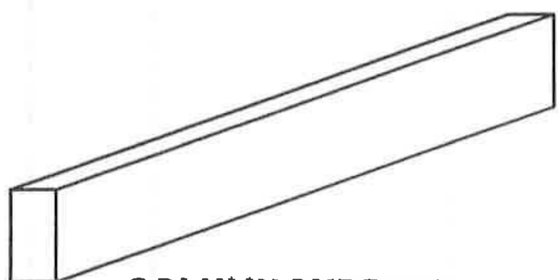


Steel tie-down.

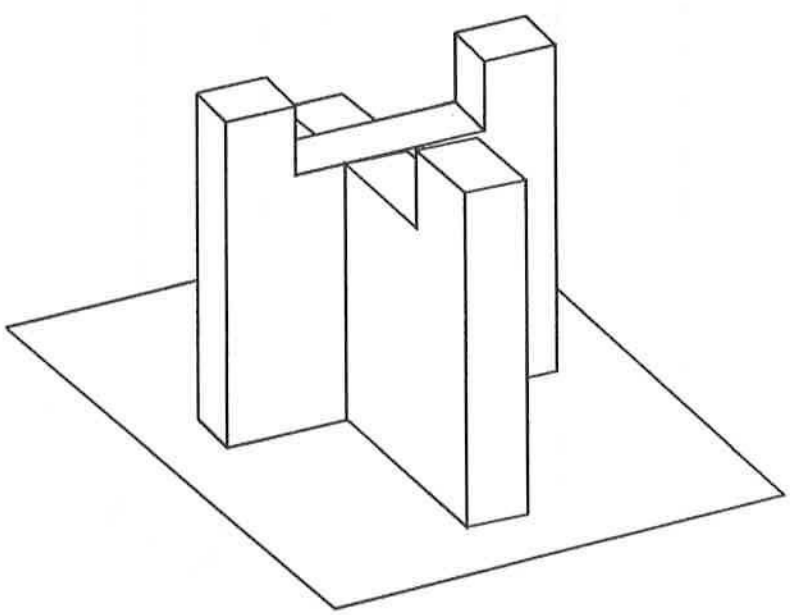
SHIPPING SUPPORTS



COUNTERWEIGHT STABILIZER

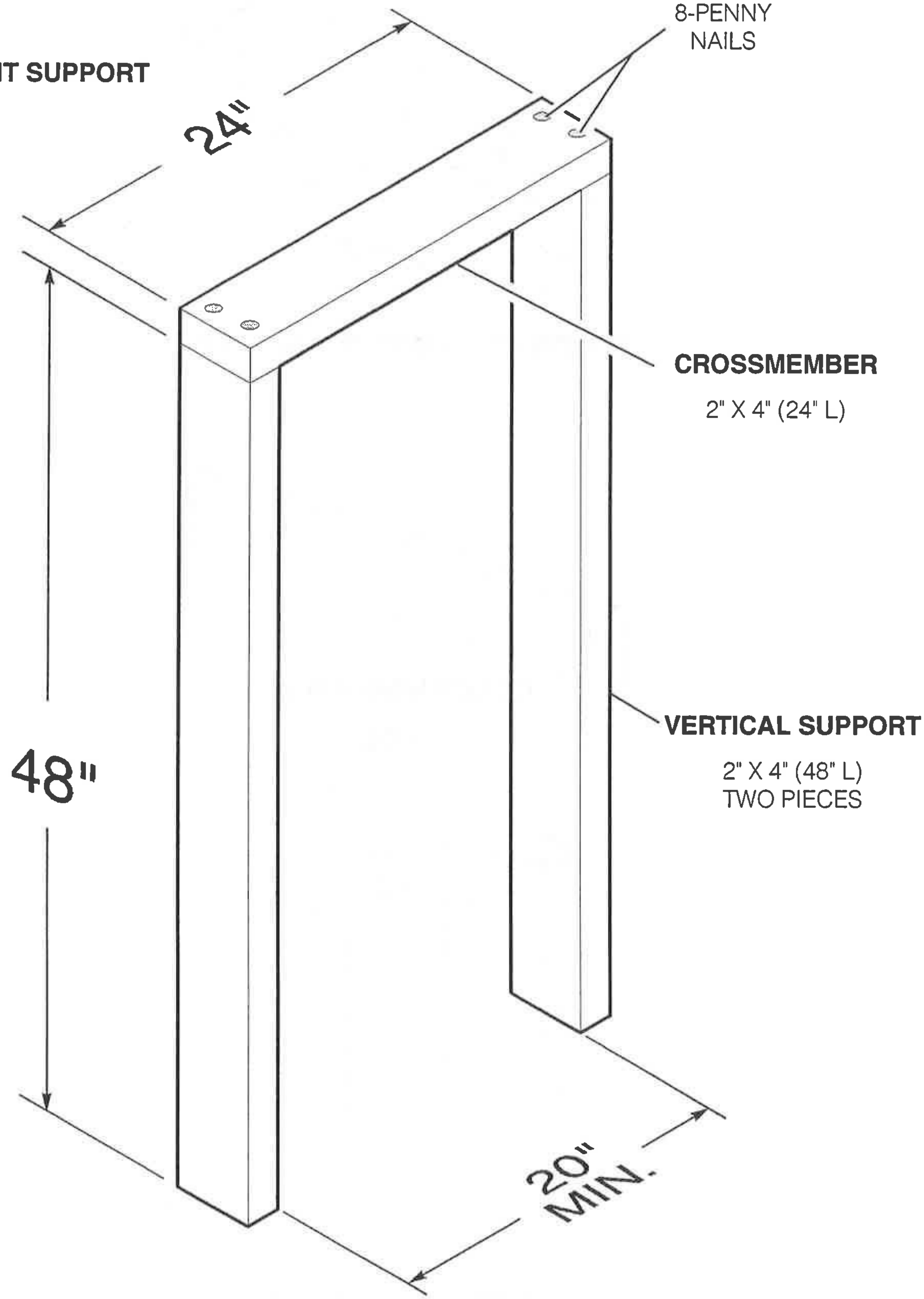


COLUMN SUPPORT
2X6

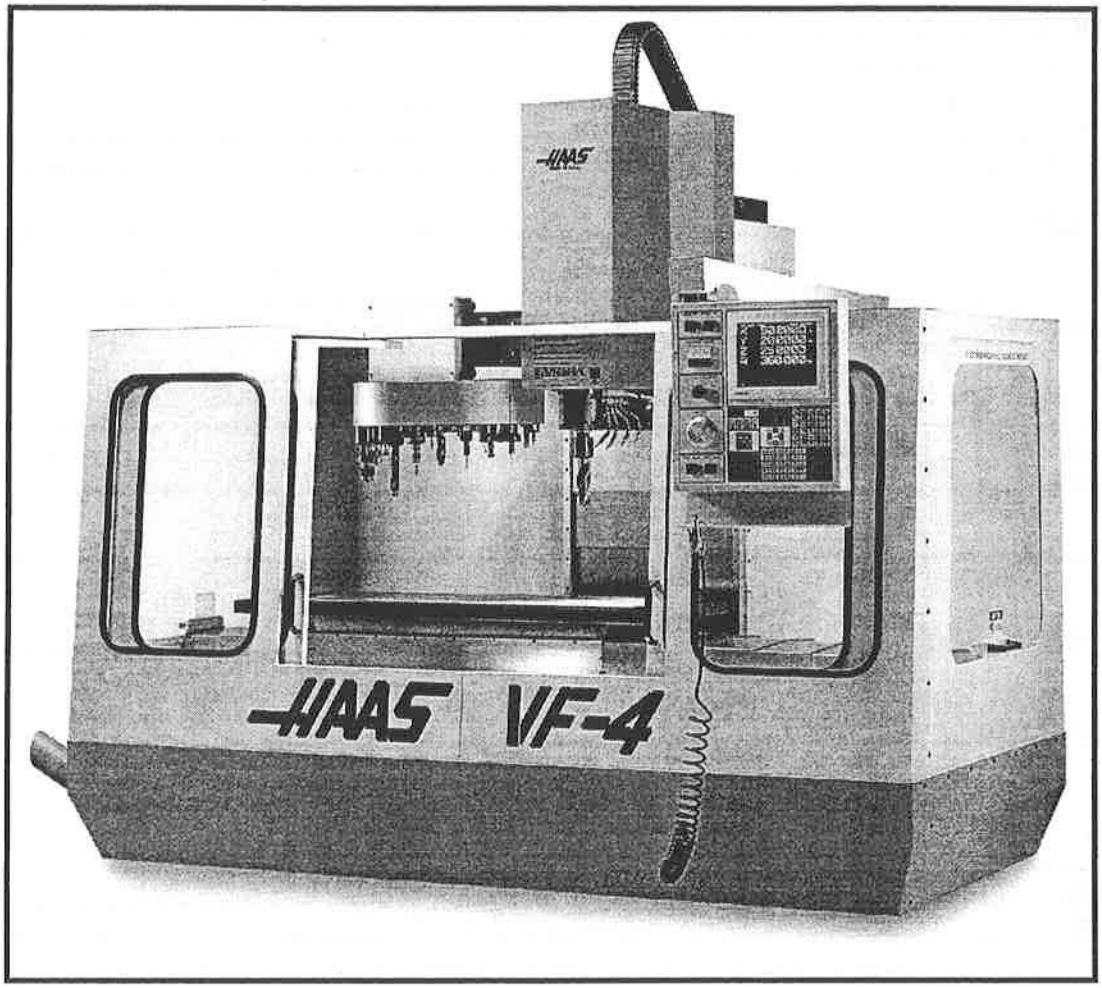


SPINDLE SUPPORT

PENDANT SUPPORT



HAAS AUTOMATION, INC.



INSTALLATION MANUAL FOR THE VF-3, VF-4, VF-6

TABLE OF CONTENTS

INSTALLING THE VF-3/VF-4

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 AUTO POWER UP 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 instructions and in 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.

Never reach through the holes in the vertical column. The large counterweight inside can cause severe injury as it moves up and down.

KEY SPECIFICATIONS	VF-3	VF-4	VF-6
Installation Height (in.)	117	117	126
Installation Width (in.)	102	102	156
Installation Depth (in.)	144	144	111
Shipping Weight (lb.)	13,540	14,000	20,000

SERVICE REQUIREMENTS

ELECTRICITY

The VMC runs on a three-phase, 195 - 260 V, 50- or 60 Hz supply. Power supplied to the machine should not fluctuate more than ± 5%. The power required is 25 amps.

If the service run from the electrical panel is less than 120', use No. 10 wire. If the run is longer, use No. 8 wire.

A separate earth, or cold-water-pipe, ground is required (a conduit-type ground does not provide sufficient noise immunity).

Do not connect to 480 V under any circumstances!

Maximum voltage leg-to-leg or leg-to-ground
should not exceed 260 volts!

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.** 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 possiblewith this machine.**

AIR

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 H.P. compressor, with a minimum 20-gallon tank, that turns on when the pressure drops to 100 PSI.

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. See Appendix B (Service Notes) for instructions on how to clean this element.

The air must be supplied through a minimum 3/8" hose and fittings must be at least ¼" NPT.

NOTE: The nipple between the air filter/regulator and the Bijur oil lubricator (see illustration at bottom right of page 6) 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 level, such as one calibrated to show 0.0005 inch per 10 inches.
✓ Test indicator (0.0005).
✓ ¼" allen wrench.
✓ 5/16" allen wrench.
✓ 3/8" allen wrench.

✓ 9/16" hex wrench.
✓ Two ¾" hex wrenches (one open end or box and one ratchet).
✓ A forklift that is capable of lifting more than 20,000 pounds. The forks must be at least 8' long by 6" wide.
✓ Claw hammer.

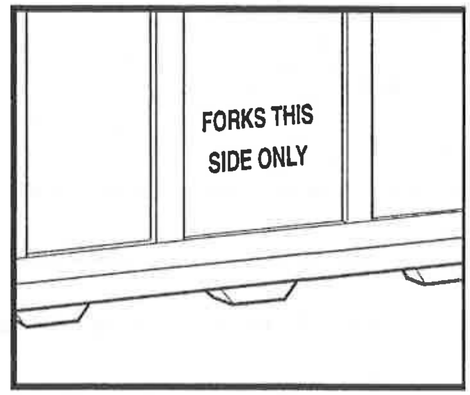
MATERIALS REQUIRED

✓ 20/30 gallons of water-soluble synthetic, or cutting oil coolant.
✓ A small amount of grease.
✓ Way lube for the lubricator (Vactra #2).

UNPACKING THE VMC

MOVING THE CRATE

CAUTION! The fork positions are marked on the back of the crate. (Also, note that there are three skids at front and back of the pallet. The heavy part of the machine [the back] is noted on the back panel.) 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.



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.

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. 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.

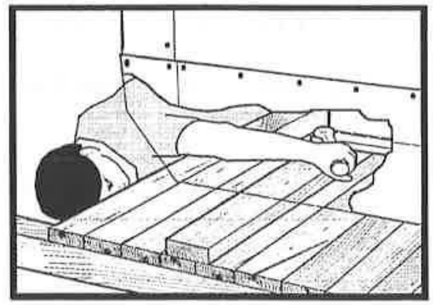
4. Carefully remove the staples that fasten the plastic cover to the pallet. Pull the plastic cover off the machine, fold it, and set it aside to store with the crate. CAUTION! Do not put undue pressure on the top of the machine as you remove the plastic.

5. Lift out the large box (containing the coolant system) and the red tool box on the left side of the machine. Remove the cleats that held them in place.

6. Pull the drain bucket out of the rear of the machine.

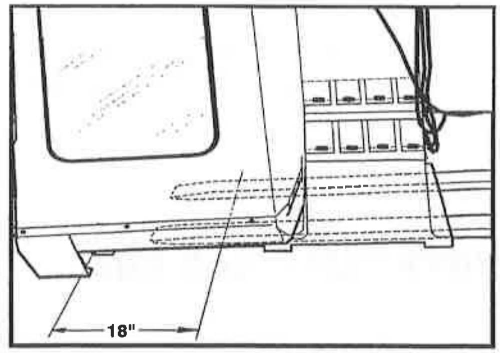
7. Remove the touch-up paint that is taped to the gutter of the table.

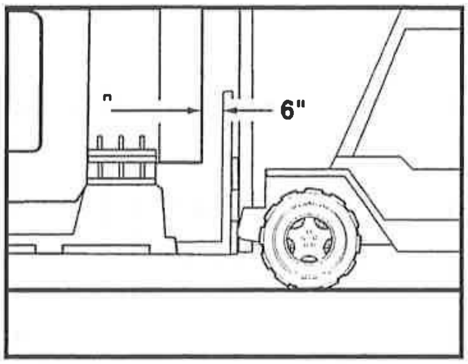
8. Using ¾" wrenches, remove the nuts (under the pallet) from the retaining bolts at the four corners of the machine. (A ratchet on top and a wrench on the bottom is the easiest way to do this.) Leave the bolts in the holes after the nuts are removed.



SETTING IN PLACE

CAUTION! Much of the weight of the VMC is concentrated in the column at the back. Therefore, it is imperative that the forks of a forklift be positioned as shown in the picture above with the fork at the back of the machine located next to the pads. Front of the forks must reach to within a **maximum** of 18" from the front of the base. Also, when moving the machine, do not lift it any farther than necessary off the floor and move as slowly and cautiously as possible. Dropping the machine, even from a height of a few inches, not only has the potential to cause injury, it also results in expensive repairs and voids the warranty.





WARNING! The only acceptable way to move a VMC is to pick it up from the back with a forklift. The forklift must be capable of lifting more than 20,000 pounds. The forks must be at least 8' long by 6" wide and they must be set 31" apart (measured outside to outside of the forks). Also, there must be about 6" clearance between the forklift and the side of the machine. Attempting to move the machine any other way may void the warranty.

1. Lift the machine until the bolts clear the pallet. Pull the bolts out of the holes in the machine.

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 ¾-10 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 the six pads out of the tool box, grease the dimple in each pad, and locate them under the six leveling screws, then lower the machine.

4. Remove all banding and packing material around the control panel and the doors.

5. Remove the two bolts fastening the arm to the rear cabinet and slowly swing the support arm around to the front of the machine, taking care to not pinch the cables inside the arm and supporting the weight of the control panel box.

CAUTION!! Do not pinch or unnecessarily bend the cables inside the arm! Irreparable damage and/or intermittent opera tion of the control panel may result.

6. Push the support arm toward the rear of the machine and lock into place. Insert the two bolts (previously removed from the angle bracket) into the two holes at the support arm's joint. Do not tighten at this time.

7. Level the control panel box (bottom of the box approximately horizontal to the floor) and tighten the two bolts on the support arm.

8. Connect the matching electrical leads from the support arm to the work light and door switch at the top right of the machine enclosure. The leads are marked and cannot be interchanged.

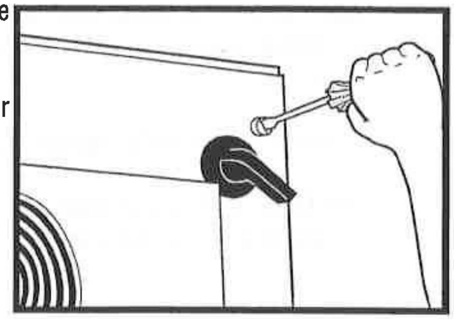
INITIAL SETUP

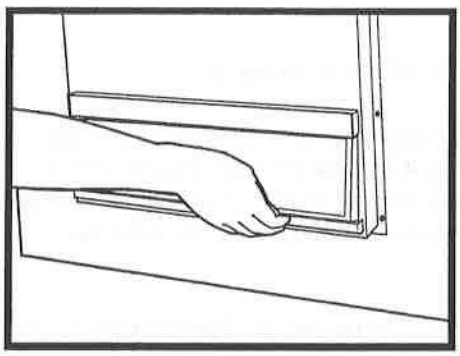
WARNING! At this point, there should be NO electrical connection to the machine. The electrical panel should be closed and the three screws on the door should be secured at all times except during installation and service. At those times, only qualified electricians should have access to the panel. When 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, loosen the three screws on the panel door and open the door.

3. The manuals and air filter screen are located at the bottom left of the panel.





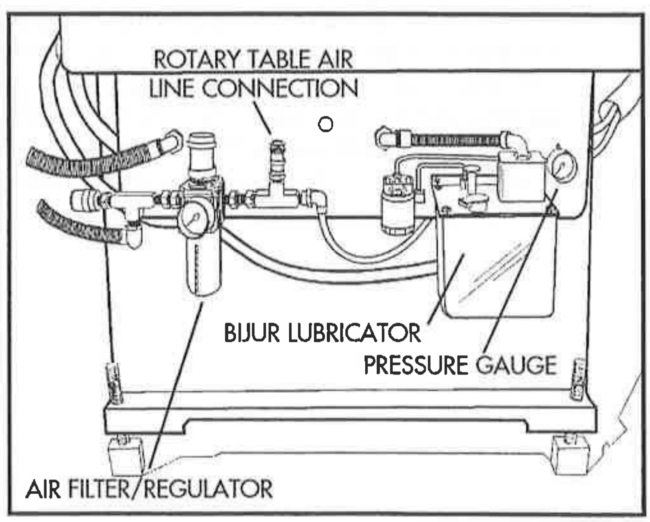
4. Install the air filter toward the bottom on the outside of the panel door (push it up into the top mounting bracket and then push in the bottom edge).

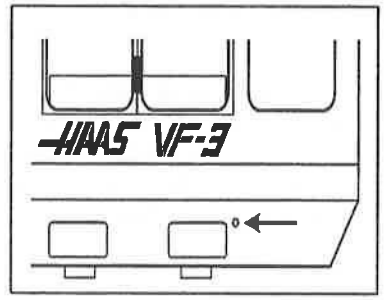
5. Take sufficient time to check all the components and connectors associated with the circuit boards. With the power off, push on them gently to make sure that they are seated in their sockets. Look for any cables that have become disconnected, look for any signs of damage, and loose parts in the bottom 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.

1. The air filter is empty, and the air lubricator and the lubricator reservoir tank are full when the machine leaves the factory. However, they should be checked and serviced if required before compressed air is supplied to the machine (See details in the Maintenance Briefs section of this manual).





2. Connect the air blow gun to the air supply fitting on the right apron of the machine.

3. 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.

4. Start the compressor, set it between 100 and 150 PSI. Set the regulator on the machine to 85 to 90 PSI. Prime the Bijur lubricator to make sure it is working.

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.

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. When you pull up on the handle, you can see the pressure increase on the pressure gauge.

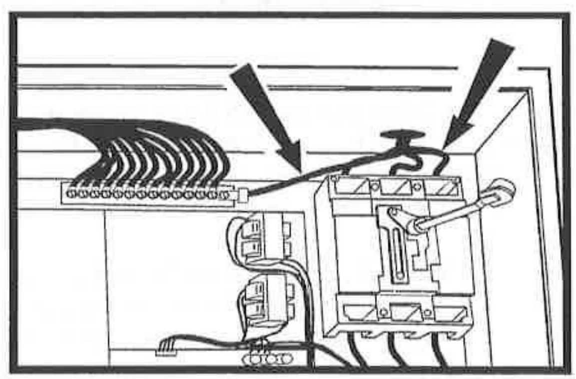
ELECTRICAL CONNECTIONS

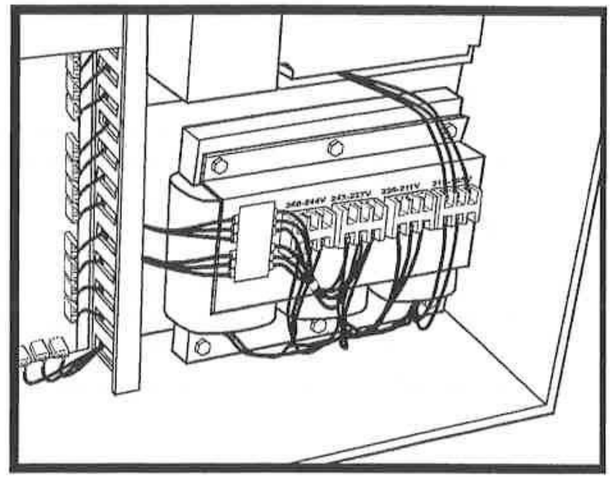
NOTE: The machine must have air pressure at the air gauge or an interlock will prevent it from powering up.

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 screws 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 L30. 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.



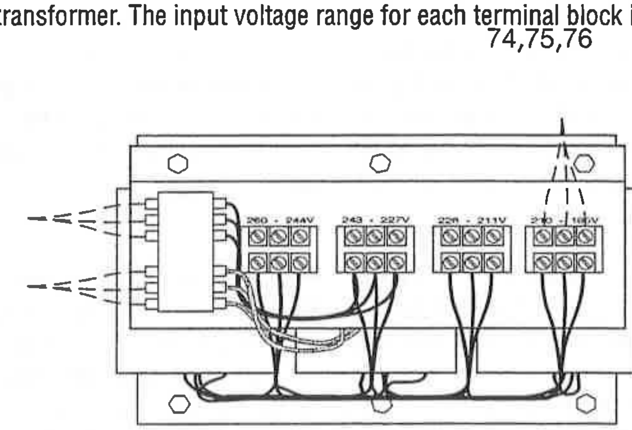


2. After the line voltage is connected to the machine, make sure that main circuit breaker (at top right of cabinet) is OFF (rotate the shaft that connects to the breaker counterclockwise until it snaps off). Turn ON the power at the source. Using an accurate digital voltmeter and appropriate safety 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. 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.

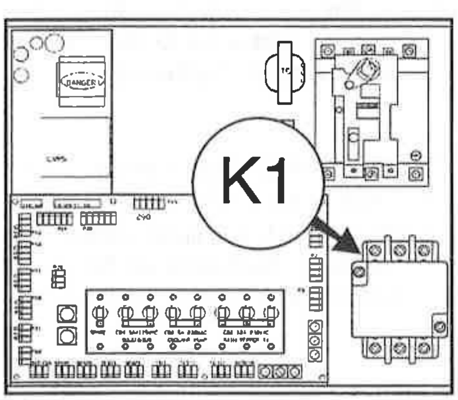
CAUTION! Make sure that the main breaker is set to OFF and the power is off at your supply panel BEFORE you change the transformer connections. Make sure that all three black wires are moved to the correct terminal block and that they are tight.

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 to this transformer. The input voltage range for each terminal block is as follows:

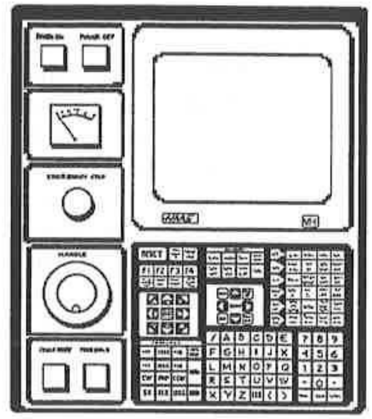
195 to 210	right side
211 to 226	right center
227 to 243	left center
244 to 260	left side



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.

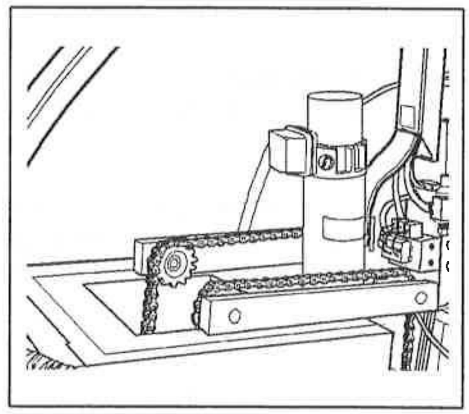


6. Check the DC voltage displayed in the second page of Diagnostic data on the CRT. It is labeled DC BUS. This voltage must be between 150 and 175 volts. If the voltage is outside these limits, turn off the power and re-check the incoming power and the transformer wiring (repeat steps 2 and 3). If the 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, screw the screws into place, and turn the power back on.

STARTUP

CAUTION! Do NOT press AUTO POWER UP on the control panel while the shipping support bracket is under the spindle.



1. Check at the top of the column to make sure that the counterweight chains are properly set on the sprockets.

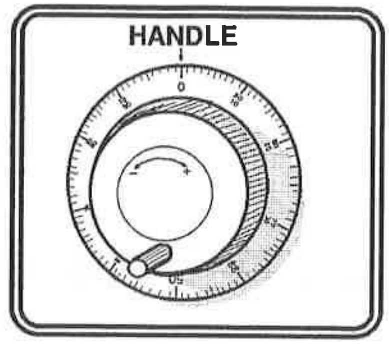
2. With the main switch on the electrical panel set to ON, press and release POWER ON at the upper left of the control panel. You will hear a click in the back of the machine and the fans will energize. (If you don't hear these sounds, the machine is not getting power and, with all necessary safety precautions, you should check the connections to the electrical panel.) After a few seconds, the display will appear on the screen.

3. 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 you start the machine for normal operation (BUT NOT FOR THIS START-UP ROUTINE.)

4. 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: Some alarms, such as low way lube or low air pressure, cannot be cleared with the RESET button. If you cannot clear the alarms, press and release the ALARM MESGS button for more information on the alarms. If you are unable to clear the alarms, copy down the alarm numbers and call the factory.

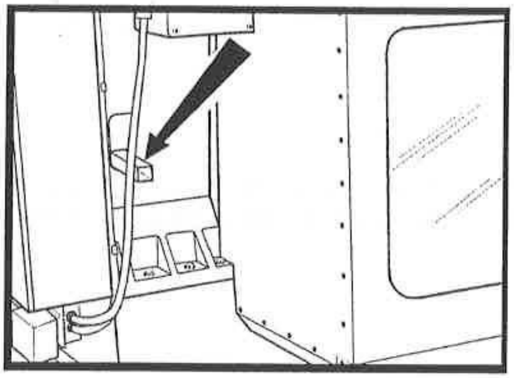
5. Press and release the HANDLE JOG button and check the screen for the "JOGGING Z AXIS HANDLE .001" message. Verify that the head will travel SLOWLY (not more than 0.001 inch per impulse — the ".001" part of the Z-axis message). If the message does not read .001, press and release the .001 button next to the HANDLE JOG button. (NOTE: The upper numbers on the buttons next to HANDLE JOG are inch settings and the lower numbers are for use when the machine is set for metric operation). If the "JOGGING . . ." message shows the X- or Y-axis instead of Z, press and release the +Z button. CAUTION! Do not touch the -Z button while the support bracket is located under the spindle.

6. Using the 5/16" allen wrench, remove the two bolts from both ends of the support bracket.



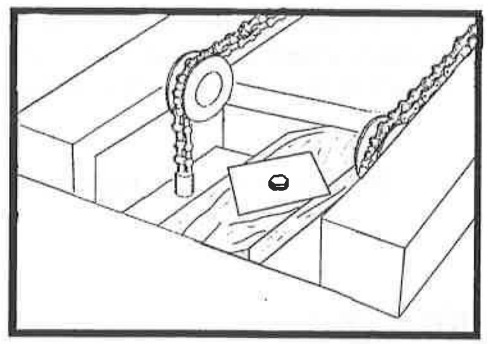
7. Turn the HANDLE clockwise (in the positive direction) to raise the head off the wooden head support. **CAUTION!** Do not turn the handle counterclockwise while the wooden head support is located under the spindle.

8. Remove the support bracket, the cardboard, and the T-nut from the mill table.



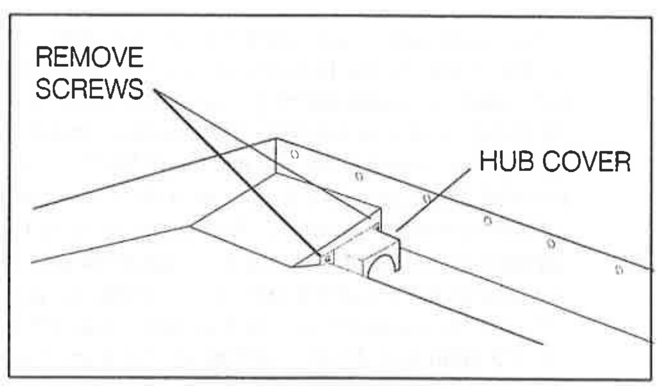
9. Turn the HANDLE counterclockwise to move the head down a couple of inches (until the counterweight lifts off the counterweight support that extends through the column).

10. Remove the counterweight support and the counterweight stabilizer (the wooden piece at the top-front of the counterweight) — . If it is hard to get the stabilizer out, inspect the counterweight for misalignment and the chains to make sure that they are on the sprockets and operating correctly. If there is a problem, lower the head (with the HANDLE), put the counterweight support back through the column, raise the head (with the HANDLE) to lower the counterweight onto the support, and set the chains on the sprockets, etc. Then proceed as before to remove the counterweight support.



11. 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. (See item 4 in this section for information on alarms.) 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.

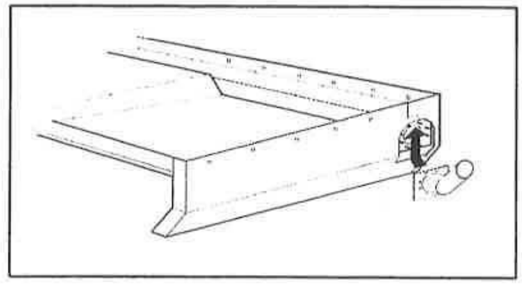
CHIP CONVEYOR INSTALLATION



1. Unpack the conveyor and drain tube and remove the motor hub cover.

3. Slide the conveyor into the discharge tube and then slip opposite end onto motor hub. Fasten to motor hub with the 5/16-18 x 2½" bolt.

4. Replace motor hub cover.



ELECTRICAL CONNECTION

The following instructions are for connecting the power supply to machines equipped with the optional chip conveyor. Please read instructions entirely before connecting.

1. Connect the power supply to the machine as shown in the Installation Manual

2. Check the the conveyor during operation to ensure the direction of auger 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 1-0 or 0-1 to establishe a new forward direction.

MAINTENANCE

During normal operation, most chips are discharged from machine at the discharge tube. Very small chips may flow through the drain and collect in the chip trap tray located in the coolant tank. To prevent drain blockage, clean trap regularly. Should drain become clogged and coolant collects in the machine's pan, stop the machine, loosen the chips blocking the drain and allow the coolant to drain. Empty the chip trap from the coolant tank, then resume operation.

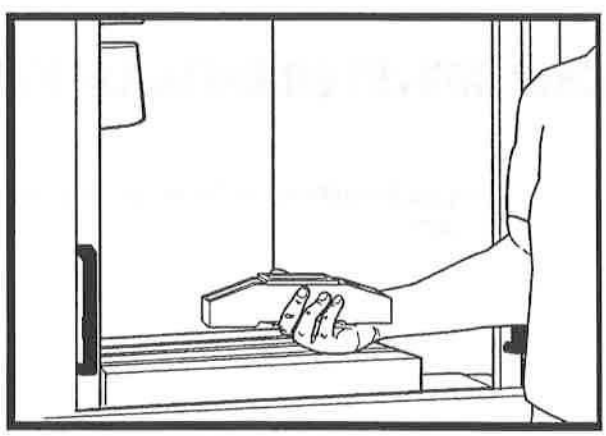
12. The machine is now ready for leveling.

LEVELING

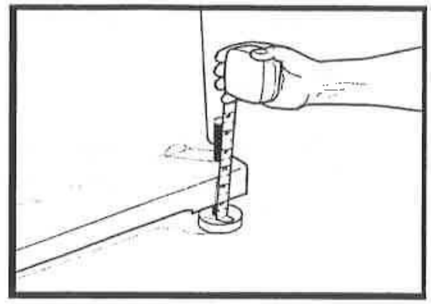
Leveling of the machine is required to obtain the correct right angle geometry of the VMC's X, Y, and Z axes. Incorrect level will result in out-of-round circle milling and incorrect linear interpolation.

Leveling is done in two steps: **rough level** 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 .05mm 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 mentally 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

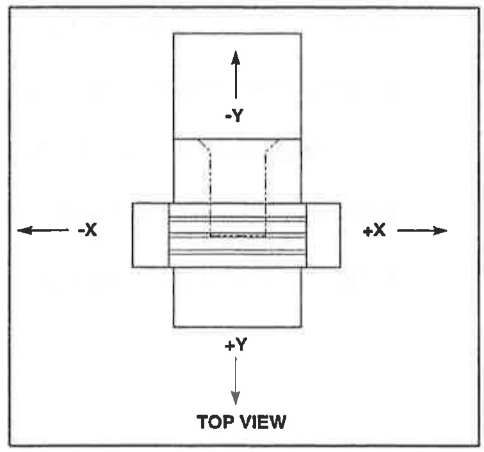


1. Press and release SETNG GRAPH. Then page down to the last page (press and release PAGE DOWN several times). Cursor to Setting 51, DOOR HOLD OVERRIDE (with the cursor **down** key). Press and release the cursor **right** key and then press and release the WRITE key at the extreme lower right of the control panel to turn this setting on. This bypasses the door interlock so you can operate the machine with the doors open. NOTE: This setting automatically resets to off when the machine is powered up again. DO NOT disconnect the door interlock switches (this makes you liable for any damage

2. 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). NOTE: The two center leveling screws should not be touching anything.

3. Use Handle Jog set for 0.01 on the X- and Y- axes for the leveling procedure. (See the Start Up 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 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 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 as was 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.

COOLANT SYSTEM

1. Connect the coolant tank power line to the connection on the right side of the electrical panel. A plaque above the connections specifies where the connection is made.

2. Connect the loose hose on the left side of the column to the hose fitting on the pump on top of the coolant tank.

3. Fill the tank with the appropriate amount of coolant (20 or 30 gallons).

4. Position the coolant tank against the left side of the machine.

5. Place the bucket under the drain pipe at the rear of the machine.

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 ¾" wrenches (one open end or box and one ratchet).
✓ A forklift that is capable of lifting more than 7,500 pounds. The forks must be at least 5' 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 (7' x 7').
✓ 6-mil plastic cover (18' x 6" x 20').
✓ Coolant box (cardboard).
✓ Wire ties.
✓ Foam pads.
✓ Fiber banding.
✓ ½-13 x 10" bolts (4).
✓ ½" washers (4).
✓ ½-13 Nylock nuts (4).
✓ 16d nails as required.
✓ Wooden supports as specified in Appendix D.

CLEANING

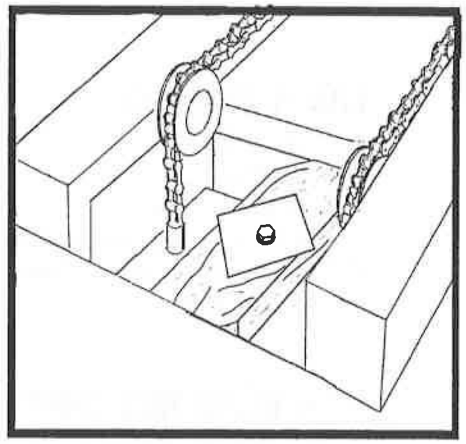
Clean the machine as necessary before preparing it for shipping. (See the cleaning notes in the Maintenance Briefs section.)

LOCK UP THE MACHINE FOR SHIPPING

1. Remove any tools from the tool carousel and spindle.

2. Spray all metal parts around the way covers, table, spindle (including up into the spindle), with rust preventive oil. (See the cautionary notes in the Cleaning section of the Maintenance Briefs.) After spraying, clean the interior of the enclosure, including the windows, as necessary.

3. In Jog mode, use the HANDLE to center the table under the spindle, and move the head practically all the way down. This is approximately X = 10, Y = 8, and Z = 13.

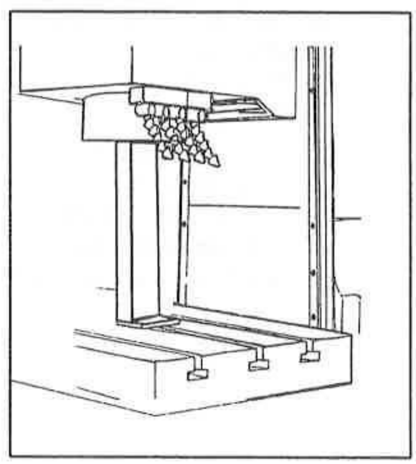


4. Place the counterweight support through the access holes at either side of the column to support counterweight.

5. Using the HANDLE, raise the head (Z-axis) far enough so the counterweight rests on the counterweight support and the head support can be inserted between the spindle and the table (about 10" of clearance between the table and the head). Tighten the locking bolts on both sides of support bar.

6. Bolt the counterweight stabilizer into the top of the column on top of the counterweight itself. This keeps the counterweight from rocking back and forth during shipment.

NOTE: You can hear a slight thump as the counterweight comes to rest on the support.



7. Set the head support on a piece of cardboard (to protect the table against scratching) in the center of the table. Position the support so the spindle housing is resting on it and NOT the spindle shaft.

NEVER PUT ANY SUPPORT ON THE SPINDLE!! THIS WILL DESTROY THE SPINDLE BEARING!!!

8. Use the HANDLE to bring the head down until it rests on the wood. Then press EMERGENCY STOP and the machine is locked up for shipping.

9. Power down (press the POWER OFF button at upper left of control).

10. Tape the touch-up paint to the gutter of the table in the front of the machine.

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 screws 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.

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 tighten the screws.

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.

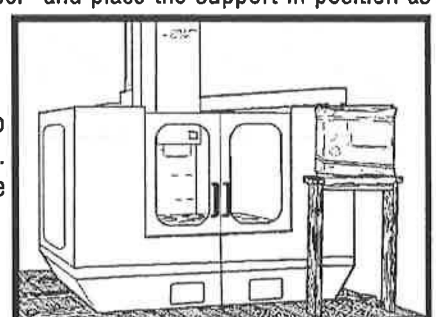
1. Make sure that the machine is locked up and remove any loose material from the table area.

2. Place a pad between the doors, pull them closed, and secure them with two wire ties through the handles.

3. For new machines, make sure that the warranty certificate is positioned against the screen on the control panel.

4. 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.

5. Wrap some padded packing material around the pendant support's crossmember and place the support in position as shown.



6. Nail the pendant support to the pallet to secure. Once the support is secured, slip packing material behind the pendant wherever it comes into contact with the machine. This not only protects the machine but also makes the support more sturdy and the control panel more secure.

MOVE ONTO PALLET

NOTE: If you did not store your shipping materials, pallets and crates are available from Haas Automation. Contact the factory for information.

WARNING! The only acceptable way to move a VMC 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 27" apart (measured inside to inside of the forks). The forks must be positioned all the way to the back of the machine and they must extend at least 3" past the far side of the machine base. Also, there must be about a foot clearance between the forklift and the side of the machine. Attempting to move the machine any other way may void the warranty.

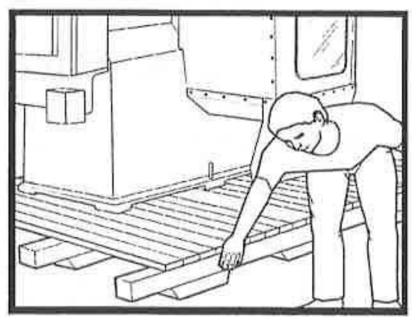
CAUTION! Much of the weight of the VMC is concentrated in the column at the back. Therefore, it is imperative that the forks of a forklift be positioned as far to the back of the machine as possible with the fork at the back of the machine located next to the pads. Also, when moving the machine, do not lift it any farther than necessary off the floor and move as slowly and cautiously as possible. Dropping the machine, even from a height of a few inches, not only has the potential to cause injury, it also results in expensive repairs and voids the warranty.

1. Lift the machine a few inches off the floor.

2. Remove the leveling screws at all four corners.

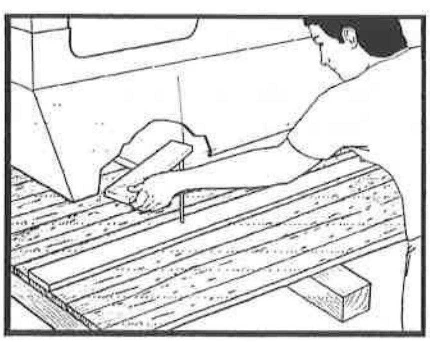
3. Insert 1/2-13 x 10" retaining bolts with washers in the leveling screw holes at all four corners of the machine. (These are secured on the pallet with 1/2-13 Nylock nuts). Putting the bolts in the machine at this time provides a guide for lining up with the holes in the pallet.

4. Cover the pallet with a 6-mil, 7' square, 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.

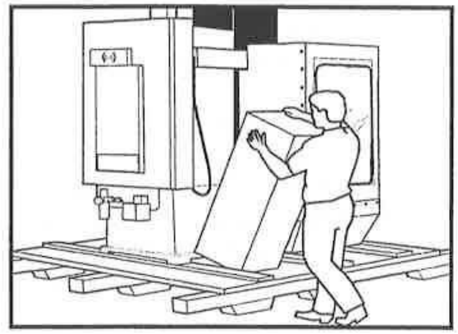


5. Move the machine over the pallet and align the bolts with the holes in the pallet. **NOTE:** You can see three supports at each end of the pallet. The center support is offset toward one side. The machine must be placed on the pallet with the rear of the machine on this side of the pallet.

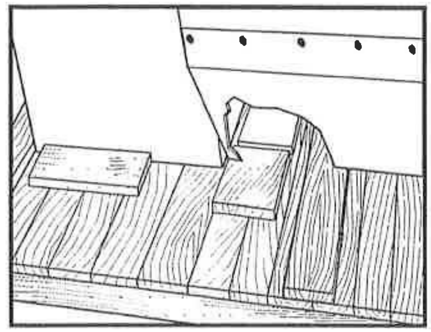
(**NOTE:** The plastic sheet was left off pallet for some of the illustrations so the holes and other details would be visible.)



6. Block the two retaining bolts at the front of the machine with short lengths (about 2') of two-by-four so they cannot slip back out of the machine. Then, as the machine is lowered onto the pallet, make sure that the retaining bolts slip into the holes in the pallet. (Move the pallet as necessary. Also, the two bolts at the back can be driven into the holes.)



7. 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.



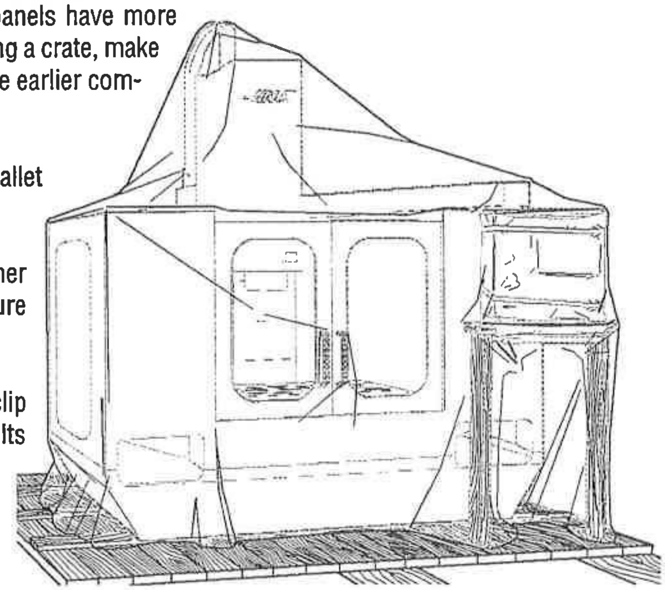
8. Put the leveling screws, pads, allen wrench, and screwdriver in the tool box and position it between the coolant box and the support strip toward the front of the machine. Then nail a 12", two-by-four cleat to the pallet with 16d nails to hold the tool box in place.

9. 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

1.Cover the machine with a sheet of 20' W, 6-mil plastic cut to a length of 18'-6". 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 (See the illustration page 4). **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.

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.

APPENDIX B: SHIPPING WITHOUT A PALLET

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.

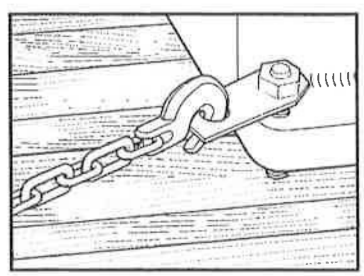
TOOLS REQUIRED

✓ 9/16" Hex Wrench
✓ ¾" Wrench (Open-end or Box-end)
✓ A forklift capable of lifting more than 7,000 pounds. The forks must be at least 5' long and 6" wide.

MATERIALS REQUIRED

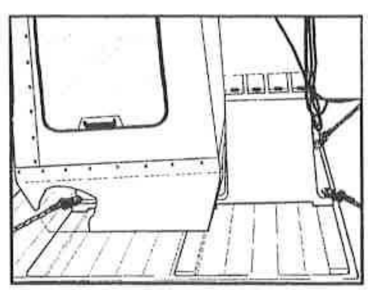
✓ Four metal tie-downs.
✓ Four ¾-11 nuts (in addition to the four nuts that come with the machine).
✓ Two 4 x 4's at least 3' long.

PROCEDURE



1. Prepare the machine as described in Appendix C up to the point of putting it on a pallet. Make sure the spindle is on tool number one. Secure the counterweight and support the spindle -- even for the shortest moves. (An unsecured counterweight can cause severe damage to the ball screw in the Z-axis.) 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 a protrude at least ½" through the bottom of the machine. Tighten the ¾-11 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 ¾-11 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 ¾-11 nuts.



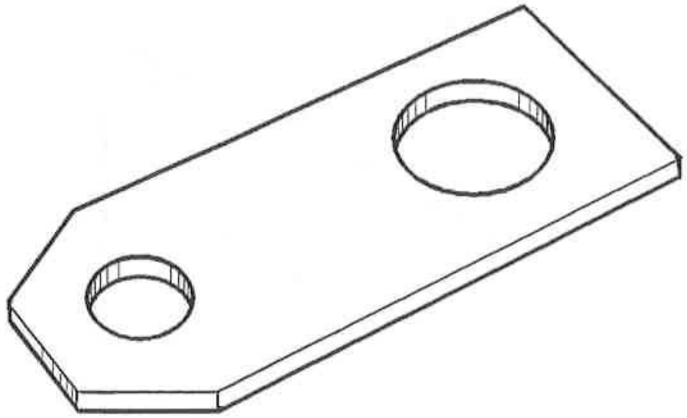
3. Place two 4 x 4's that are least 3' long on the truck. Lower the VMC onto the truck and align the two pieces of lumber under the leveling screws at the corners of the machine.

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.

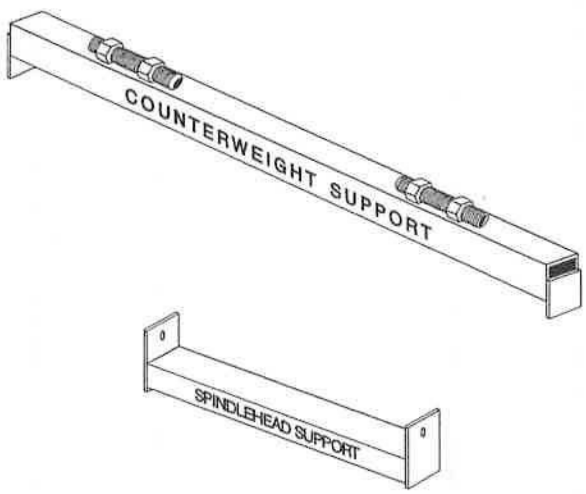
4. Using the four corner tie-downs, the machine can be secured without dragging chains across its top. It is a fast, easy, and secure transportation method that assures against damage. Be sure to explain to the rigger that this is how you want the machine to be moved BEFORE he does something else.

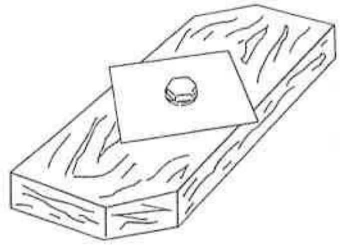
5. If you wish to tarp the machine, you should be aware of a few potential trouble spots. If a tarp is simply placed over the machine and tightened down, it can do considerable damage, such as bending the conduit carrier for the headstock, wearing paint off the machine, distorting the enclosure, and 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.

APPENDIX C: SHIPPING SUPPORTS

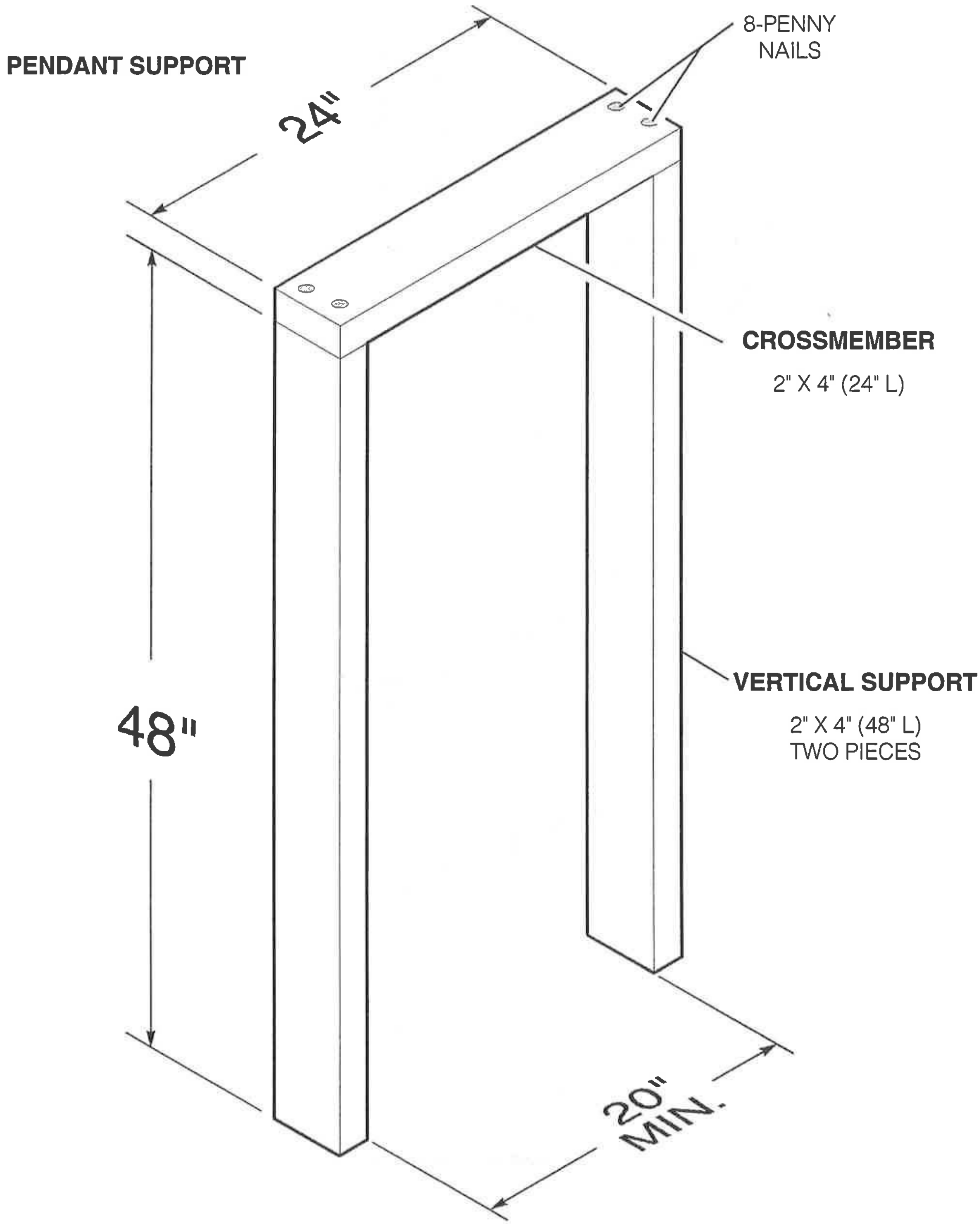


Steel tie-down.

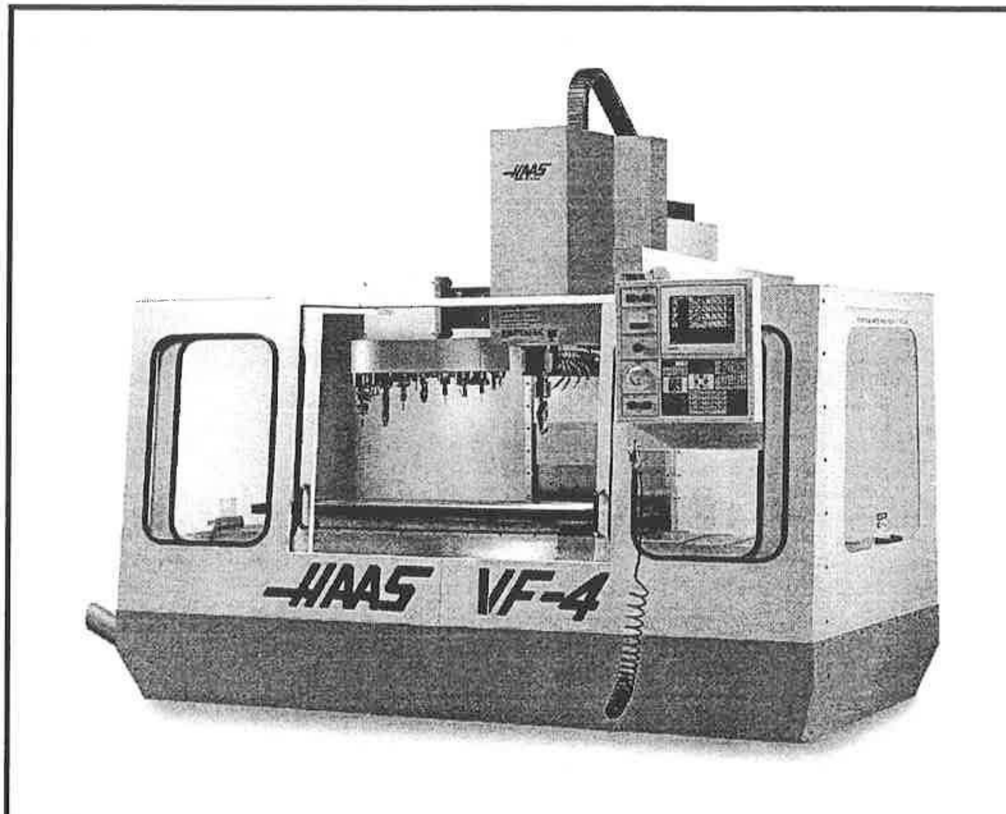




COUNTERWEIGHT STABILIZER


PENDANT SUPPORT
24"
8-PENNY NAILS
CROSSMEMBER
2" X 4" (24" L)
48"
VERTICAL SUPPORT
2" X 4" (48" L)
TWO PIECES
20" MIN.

VMC CABLE LIFT INSTRUCTIONS



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.

PLEASE READ THIS ADDENDUM IN ITS ENTIRETY BEFORE ATTEMPTING TO LIFT VMC.

LIFT KITS for the HAAS VERTICAL MACHINING CENTERS

VF-0/1/2: (LIFTING SUPPLIES IN TOOLKIT 30-7900E)	
(3) 5/8-11 x 1¼" EYE BOLTS	(3) 5/8" FLAT WASHERS
VF-3/4: (LIFTING SUPPLIES IN TOOLKIT 30-9900E)	
(2) 90° LIFT BRACKETS	(6) ½" FLAT WASHERS
(2) ½-13 x 1½" SHCS	(2) ¾-10 x 1¼" EYE BOLT
(2) ¾" FLAT WASHERS	

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.

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.

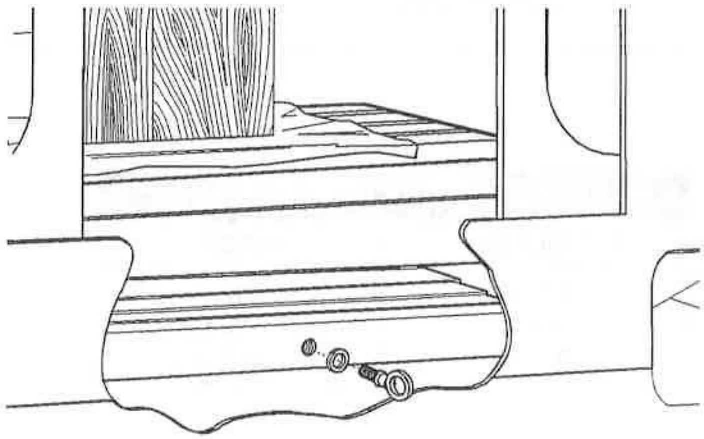
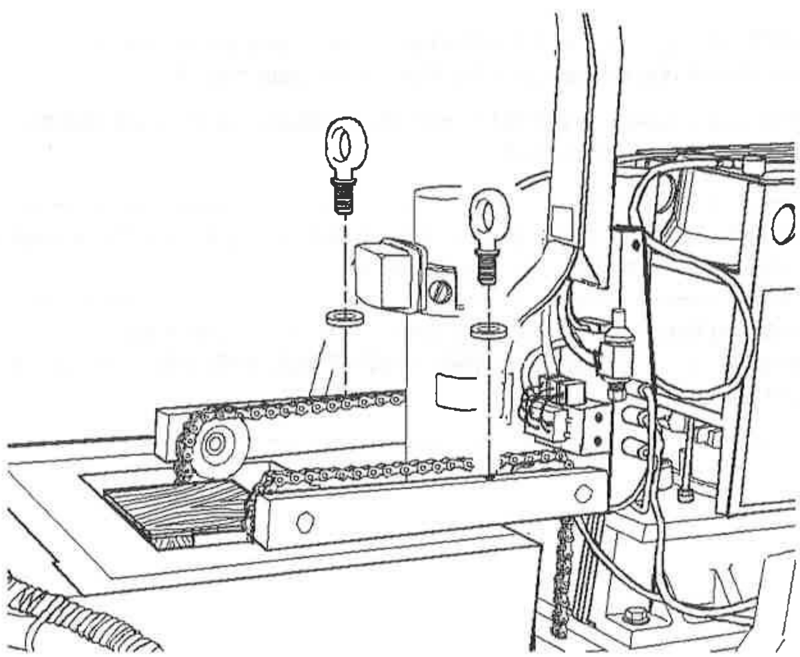


Fig. 1-1 *Eyebolt mounting location for VF-0, VF-1, and VF-2.*

2. Place a washer on each of the 5/8" x 1¼" eyebolts and mount to the counterweight support bars as shown below. Tighten securely.



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

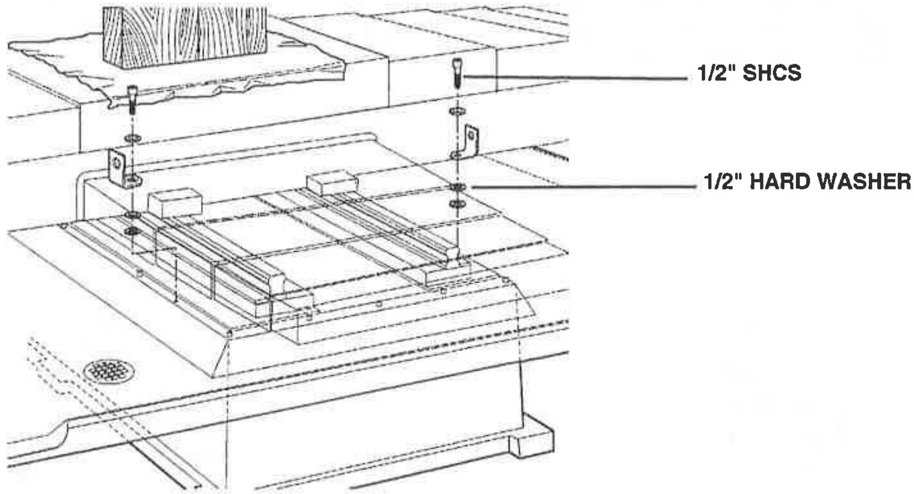


Fig. 1-3 *Lifting bracket mounting location for VF-3 and VF-4.*

4. Install the two (2) ¾-10 x 1¼" eyebolts to the column casting as shown below. Tighten securely.

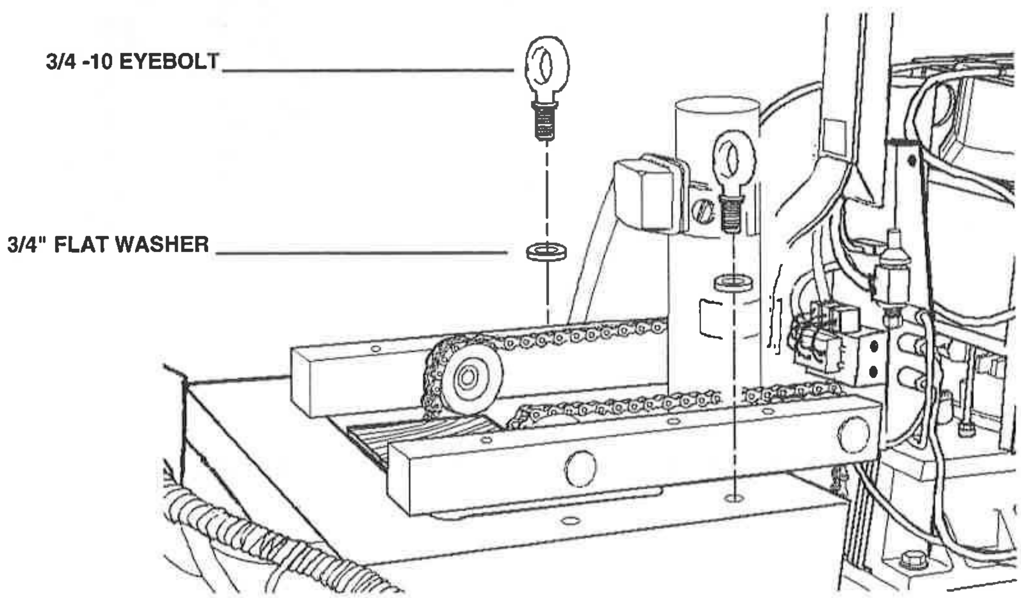


Fig. 1-4 *VF-3/VF-4 eyebolt locations.*

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.

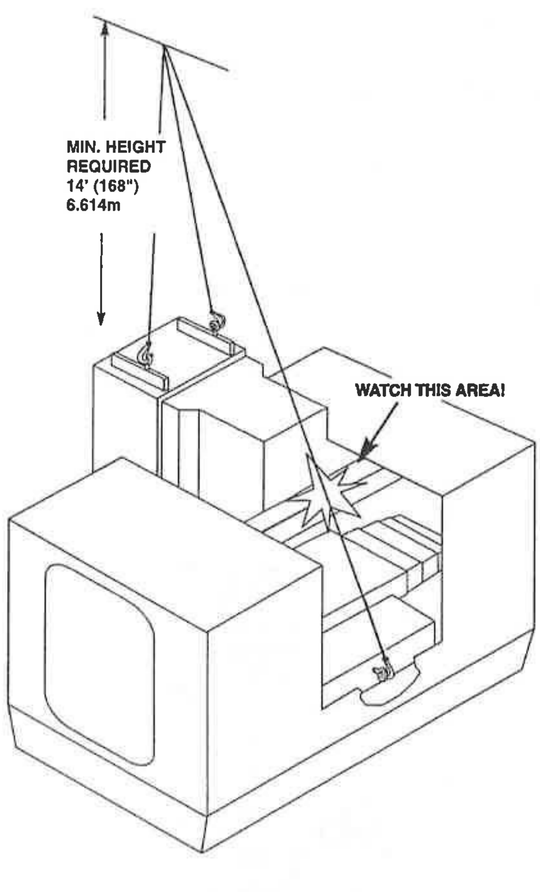


Fig. 1-4 *VF-0/1/2 lifting positions.*

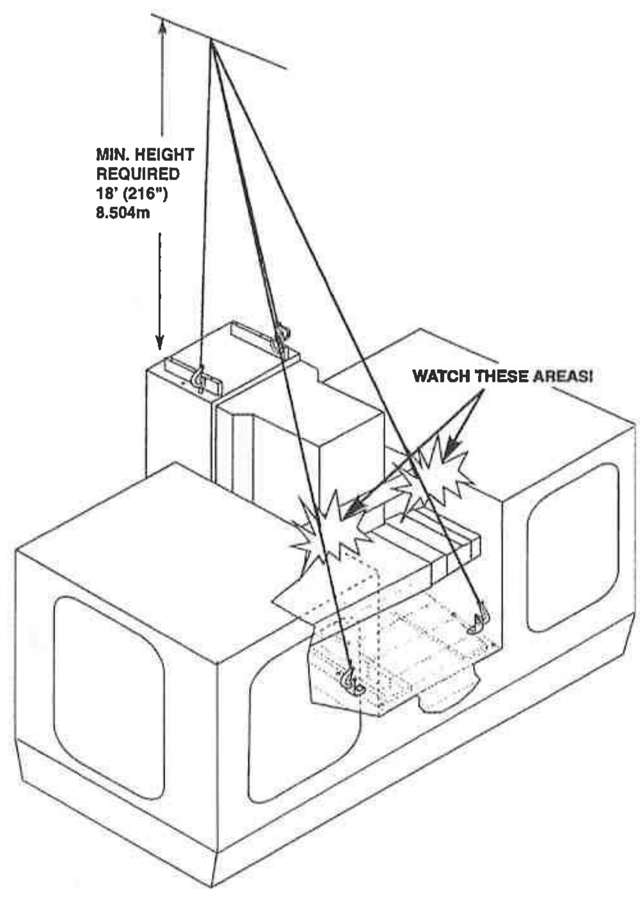


Fig. 1-5 *VF-3/4 lifting positions.*

CONTENTS

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.

FU5 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 CB4.1.2 Tool Change Sequence

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) Shuttle moves in to grab tool,
5) Tool unclamps,
6) Z axis moves up,
7) Tool Changer rotates,
8) Z axis moves down,
9) Tool clamps,
10) Shuttle moves out.

1.1 Tool Change 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.

1.4 Shuttle In/Out Switches

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.

1.5 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.

1.6 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 diagnos tic display will show this status of this input switch as "TOOL #1". A “1” indicates that tool #1 is in position.

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 14. When the relay is activated, 115V AC is applied to the solenoid. This applies air pressure to release the tool. Relay 14 is on relay card #2, relay #6. Circuit breaker CB4 will interrupt power to this solenoid.

2.3 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.

2.4 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. For the VF-1 thru 6, speeds at and below 1250 RPM automatically select low gear. Speeds at and above 1251 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 Sense Switch

A normally-closed switch 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 is wired to the spindle drive and commands it into the Coast Stop condition. This is done to make sure that the spindle motor is not powered when the pin is locking the spindle.

3.5 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. SERVO MOTOR ENCODERS

Attached to each DC servo motor, there is an incremental encoder that is either 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.

Encoders' 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.

4.1 Servo Characteristics

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 con stant 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 x 0.075 = 90000 steps/second

For a 2000 line encoder and 6 mm screw, this would be:

60 x 90000 / 33867 = 159 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).

4.2 Servo Drive Assembly

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.

4.3 160 Volt DC Power Supply

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.

4.4 Servo Cooling Fan

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.

4.5 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 causes 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 (860)
P7	115V AC to fan
P8	160V DC supply to tool changer
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 T1, T2, T3 and T4. They are ½ amp, 240V AC, AGC type. FU3 protects the regenerative load circuit from a short circuit.

4.5 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 microproces sor 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:

P1 160V DC from SDIST PCB
P2 low voltage AC power from SDIST PCB
P3 PWM and H drive control signals from Motor Interface and overcurrent sense back
P4 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,

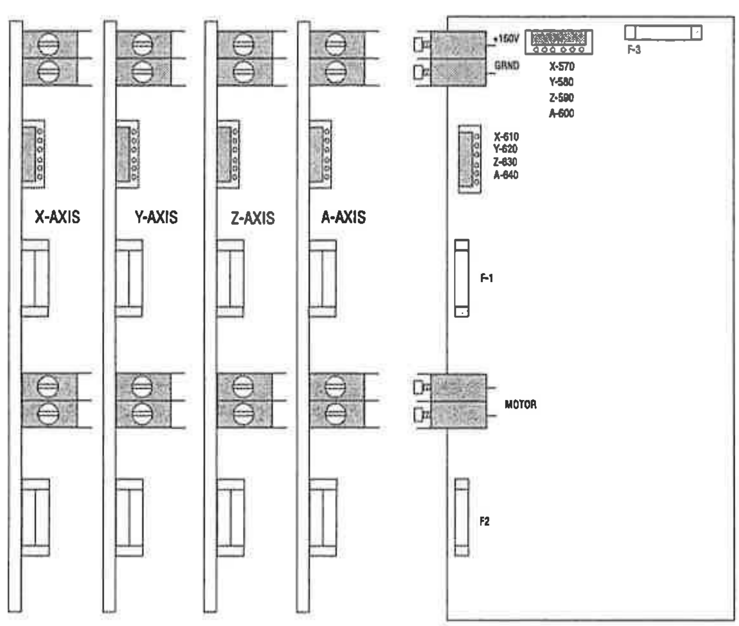


Fig. 4-1 Servo Drive Assembly

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 connectors on the IOPCB are:

P1	16-pin relay drivers from MOTIF 1 to 8 (510)
P2	16-pin relay drivers from MOTIF 9 to 16 (520)
P3	16-pin relay drivers from MOTIF 17 to 24 (M21-M28) (530)
P4	34-pin inputs to MOTIF (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)
P11	+12V DC from power supply (860A)
P12	115V AC to spindle head solenoids (880)
P13	Tool changer status inputs (820)
P14	Low coolant input (900)
P15	Spindle head status inputs (890)
P16	Emergency stop input (770)
P17	Low Lube input (960)
P18	Low 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 inputs 21-24 (790)
P25	Spare inputs 31-32 (200)
P26	Spare terminals for M21 to M24
P27	M28 output
P28	115V AC from CB4 (910)
P29	A-axis brake solenoid output (390)
P30	Tool changer shuttle motor output (810A)
P31	FU5 connection for tool changer (840)
P32	160V DC for tool changer (80)
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 to oiler (870)

P38	Door open (1050)
P39	Tool changer turret motor output (810)
P41	Operator lamp switch connection (800A)
P43	Ground fault sense signal input (1060) Axis Brake
P44	5TH axis brake
P45	HTC Shuttle
P46	Chip Conveyor
P47	Skip input signal (1070)
P48	Kill
P49	k210
P50	Spigot Motor (200)
p51	16 PIN Relay drivers 25-32 (540)
P52	A1 B1 Input
P53	Spigot Sense
P54	Servo Brake

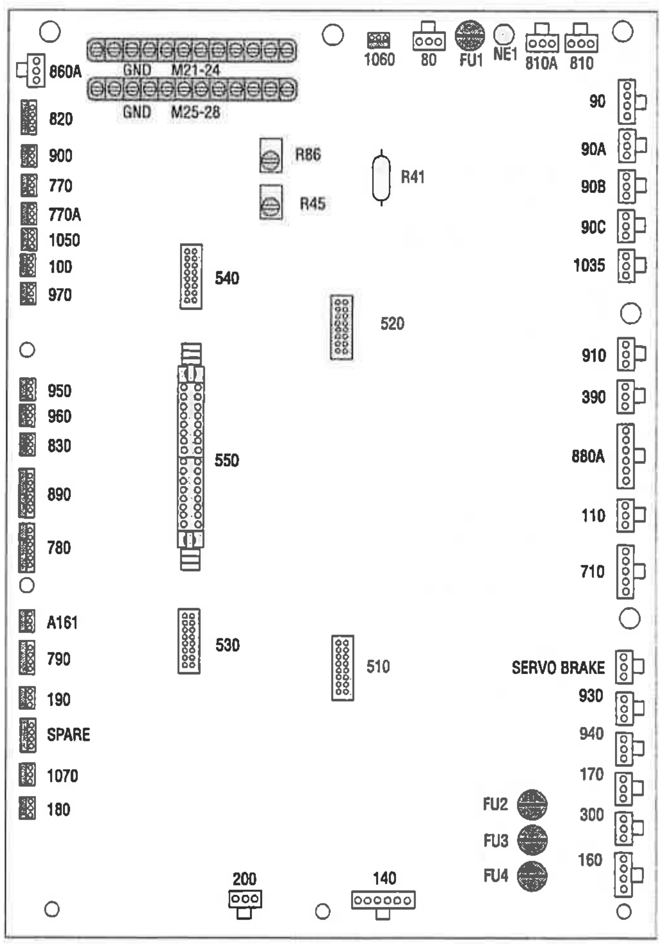


Fig. 5-1 Input/Output Board

6. TWO-SPEED GEAR TRANSMISSION (VF-1, VF-2, VF-3 ,VF-4, VF-6)

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 of low gear. Circuit breaker CB4 will interrupt power to these solenoids.

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, go 8,
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 but 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 four types of spindle drive that might be used in your control. They are all equivalent in performance but are adjusted differently. These drives are: Fuji, Mitsubishi Z200, Mitsubishi Z300, and Yaskawa.

The Yaskawa drive is adjusted similarly to the Mitsubishi drive.

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.The volume of this beeper can be adjusted from pot R2 on the Keyboard Interface PCB inside the control panel. Controls built after January 1991 have this volume adjustment permanently set to high.

If the beeper is not audible when buttons are pressed, the problem could be in either the keypad 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: 68EC030, VIDEO, and MOTIF. 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 4MB of CMOS RAM and betwen 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.**
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.**
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.**
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.**
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.**
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.**
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.**
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.**
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 (2) two-position DIP switches on the processor PCB labled S1 and S2. 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 S1-2 is used to change the default data rate for power-up communications. If the switch is OFF, the rate is 9600; if S1-2 is ON, the rate is 38400.

Switch S2-1 is used to enable FLASH. If it is disabled it will not possible to write to FLASH.
Switch S2-2 enables the processors CACHE memory.

The processor connectors are:

J1	Address buss
P2	Data buss
P4	Serial port #1 (for upload/download/DNC) (850)
P5	Serial port #2 (for auxiliary 5th axis) (850A)
P3	Power connector
P6	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 and Keyboard PCB (VIDEO2)

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	P4	Keyboard (700)
P2	Address buss	P5	EGA extended video connector (option)
P3	Video connector (760)	P6	Data buss

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)
P23	A-to-D input spare
P24	Home switch inputs X, Y, Z (990)
P25	Spare inputs
P26	A-to-D input spare
P27	A-to-D inputs spare
P28	A-to-D inputs spareP29 A-to-D inputs spare

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 5 H.P. continuous rating, a 7.5 H.P. five-minute rating, and a 9 H.P. one-minute rating. The spindle drive is protected by CB1 at 30 amps. 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 near the center of the control cabinet. It contains the servo and spindle drive regen load resistors.

10.1 Spindle Drive Regen Resistor

A 20- or 30-ohm, 600-watt resistor 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 50-ohm, 100-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-open 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. Main incoming power is brought to this assembly and any fuses or circuit breakers that might trip in operation are located on this 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 30 amps 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 investi gating the cause of the trip. These 30 amps could correspond to as much as 15 horsepower.

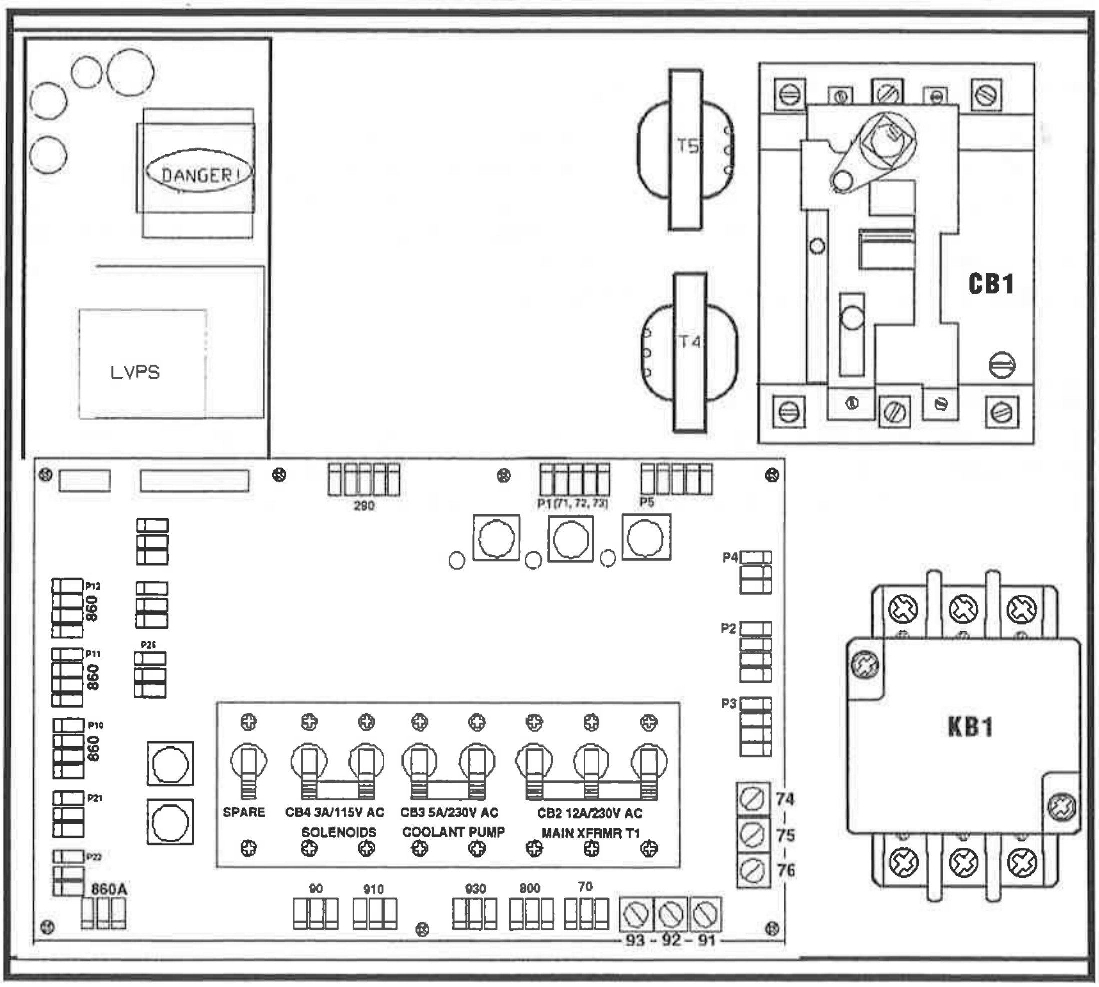


Fig. 27-1 Power Supply Assembly.

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 (See Fig. 27-1). 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
P8	115V AC from IOPCB for low voltage supply and solenoids (910)
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 (800)
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
P23	Spare circuit breaker CB5
P24	Spare fuse FU7
P25	Spare fuse FU8
P26	+12V DC option connector
P27	+5/+12/Gnd form low volt supply to logic boards (860)
P28	Option connector for alternate supply
P29	Option connector for alternate supply

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 T1. 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

Five more circuit breakers are on the Power supply assembly.

CB2 controls the power to the servo transformers and, if tripped, will turn off the servo motors and air solenoids. CB2 also controls the 115V AC from the T1 secondary and could be blown by a severe servo over load.

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 T2 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 230/208V to three-phase 115V. The 115V is used primarily 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 a polyphase bank transformer. It is rated at 3750 VA and its primary is protected at 12 amp.

This transformer is replaced by one that is much larger and can adjust the primary voltage applied to the spindle drive. The new transformer has four voltage connections that allow for a range of inputs from 195V to 260V.

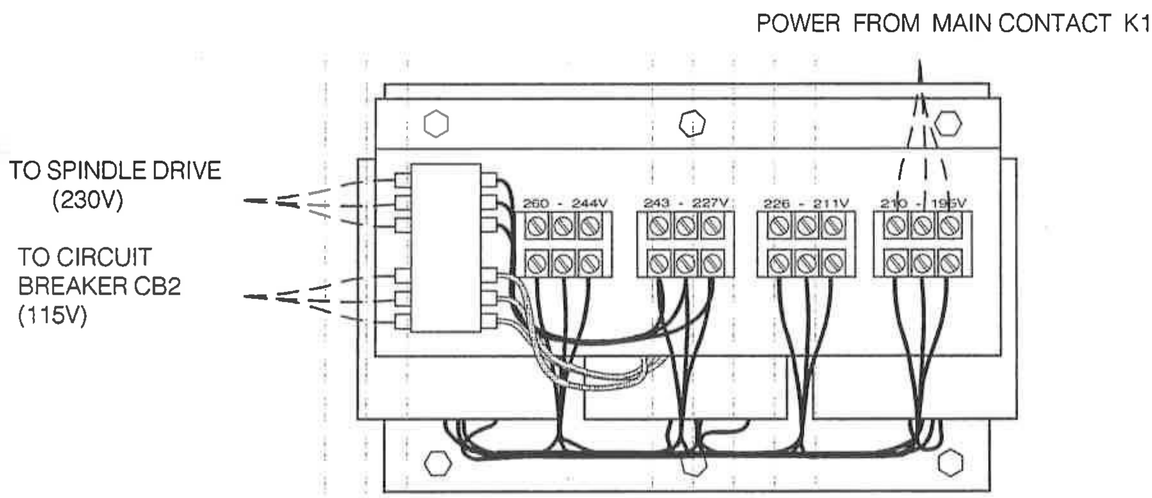


FIG. 12-1 Polyphase bank transformer assembly

12.1 Primary Connection To T1

Input power to T1 is supplied from the power assembly through CB2; a 12 amp, three leg magnetic circuit breaker. Three-phase 230 to T1 is connected to the first three terminals of TB10. With the newer transformer, CB2 protects the secondary of transformer T1 and is rated at 25 amps.

The newer transformer acts as an auto-transformer and thus has an output connection on its primary side. This connection supplies 230V AC power to the spindle drive and coolant pump.

12.2 Voltage Selection Taps

With the older main transformer, terminal block TB11 is used to select the 208 or 230V taps of transformer T1. To select between 208 and 230, three terminals must be moved. The two positions for each of these terminals are marked 208 and 230.

With the newer transformer, 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. It is available on the last three terminals of TB10.

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
TOOL CH	ABC	5	250V	" "	
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	"	

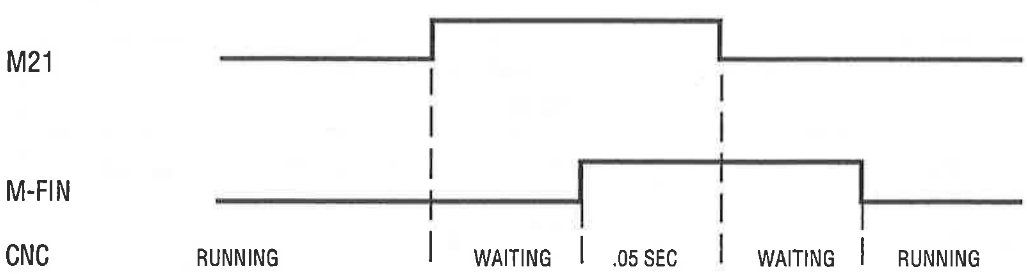
14. SPARE USER M CODE INTERFACE

The M code interface uses outputs M21-24 and one discrete input circuit. M codes M21 through M24 will activate relays labled 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 PUMP

The lubrication pump is powered whenever the spindle is on or any axes are in motion. It operates from 115V AC On a cyclic basis, it will pump oil to the screws and guides. It cycles at least once every 30 minutes.

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 plastic enclosure.

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.

16.3 X, Y, and Z Limit Switches

The machine zero position is defined by a limit switch for each of the X, Y, and Z axes. After the search for machine zero has been completed, these switches are used to limit travel in the positive direction. In addition, travel in the negative direction is limited by stored stroke limits. It is not normally possible to command the servo axes past the machine zero as servo travel look-ahead will decelerate and stop each motor prior to exceeding the stroke limits. All limit switches are wired through connector P5 on the side of the control cabinet. P5 also contains the wiring to the lubrication pump and an alternate connection to the DOOR OPEN switches.

Prior to performing an AUTO POWER UP 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.

17. COUNTERBALANCE (VF-6)

The counterweight system supports most of the weight of the spindle head giving the motor/ballscrew one direction of preload. The pneumatic cylinders are connected to a surge tank which allows the lifting piston to move without large pressure variations. The pneumatic system is closed with a one direction regulated air supply feed, which fills the system to a minimum 54psi.

The regulated air is supplied through a check valve and only provides air to fill and maintain a minimum system pressure. With the head in full up position the pistons are at largest volume- the air system fills to its lowest constant pressure, allowing the spindle head to be suspended in place.

When the spindle head is lowered, the pistons compress the air, the surge tank limits this compression to an acceptable rise of approximately 11 psi. NOTE: The compressed air is not released from the system, it is conserved. The air spring action results in an increased counterweight force of approximately 200lbs. at the bottom of the spindle head travel. The top of the air cylinders are open and fitted with filters.

18. LEADSCREW BRAKE (VF-6)

The Z-axis leadscrew is fitted with an electric power off brake. When electric power to the brake is cut off the brake prevents the screw from rotating(POWER OFF, E-STOP, ALARMS CONDITIONS, INTERUPPTED AIR SUPPLY). The brakes electrical supply is interuppted with a pressure swith connected to the counterbalance closed air system. When the counterbalance pressure falls below 45 psi, the switch opens disconnecting the brake from its power suppy. The brake is sufficient to hold the spindle head in position and overcurrent the axis motor.

19. 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 light or not as described in Section 24.1.

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:

DISCRETE INPUTS

#	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 SP*	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	A161	Safety Interlock status
7	SP LOW	Spindle In Low	23		spare
8	EM STP	Emergency Stop	24		spare
9	DOOR S	Door Open Switch	25	UNCLA*	Remote tool unclamp
10	M-FIN*	Not M Func Finish	26	LO PH A	Low voltage in phase 1
11	OVERV*	Not Over Voltage	27	LO PH B	spare
12	LO AIR	Low Air Pressure	28	LO PH C	spare
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	conveyor	

DISCRETE OUTPUTS

#	Name	Description	#	Name	Description
1	SRV PO	Servo Power On	17	M21	Spare M Functions
2	SP FOR	Spindle Forward	18	M22	
3	SP REV	Spindle Reverse	19	K111	Spindle & T.C. Enable
4	SP RST	Spindle Reset	20	K210	E-Stop Enable
5	4TH BK	4th Axis Brk Rel	21	UNCLPR	Unclamp pre-charge
6	COOLNT	Coolant Pump	22	M26	
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	spare	
10	TC OUT	Tool Changer Out	26	spare	
11	TC CW	Tool Changer CW	27	spare	
12	TC CCW	Tool Changer CCW	28	spare	
13	SP HIG	Spindle High Gear	29	spare	
14	SP LOW	Spindle Low Gear	30	spare	
15	T UNCL	Tool Unclamped	31	spare	
16	SP LOK	Spindle Lock Cmd	32	spare	

The 32 inputs are numbered the same as the 32 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 ZIRQ	X-axis Z channel interrupt
Y Z CH	Y-Axis Z Channel	Y ZIRQ	Y-axis Z channel interrupt
Z Z CH	Z-axis Z Channel	Z ZIRQ	Z-axis Z channel interrupt
A Z CH	A-axis Z Channel	A ZIRQ	A-axis Z channel interrupt
X HOME	X-axis Home/Lim Switch	1K IRQ	1 kHz Interrupt
Y HOME	Y-axis Home	Z IRQ	Z channel interrupt
Z HOME	Z-axis Home	SPZIRQ	Spindle encoder Z interrupt
A HOME	A-axis Home	SELF T	Self-Test Input
X OVRH	X Motor OverTemp	X CABL	Broken cable to X encoder
Y OVRH	Y Motor OverTemp	Y CABL	Broken cable to Y encoder
Z OVRH	Z Motor OverTemp	Z CABL	Broken cable to Z encoder
A OVRH	A Motor OverTemp	A CABL	Broken cable to A encoder
OVC X	X Drive Overcurrent	spare	
OVC Y	Y Drive Overcurrent	spare	
OVC Z	Z Drive Overcurrent	spare	
OVC A	A Drive Overcurrent	AD EOC	A-to-D End of Conversion

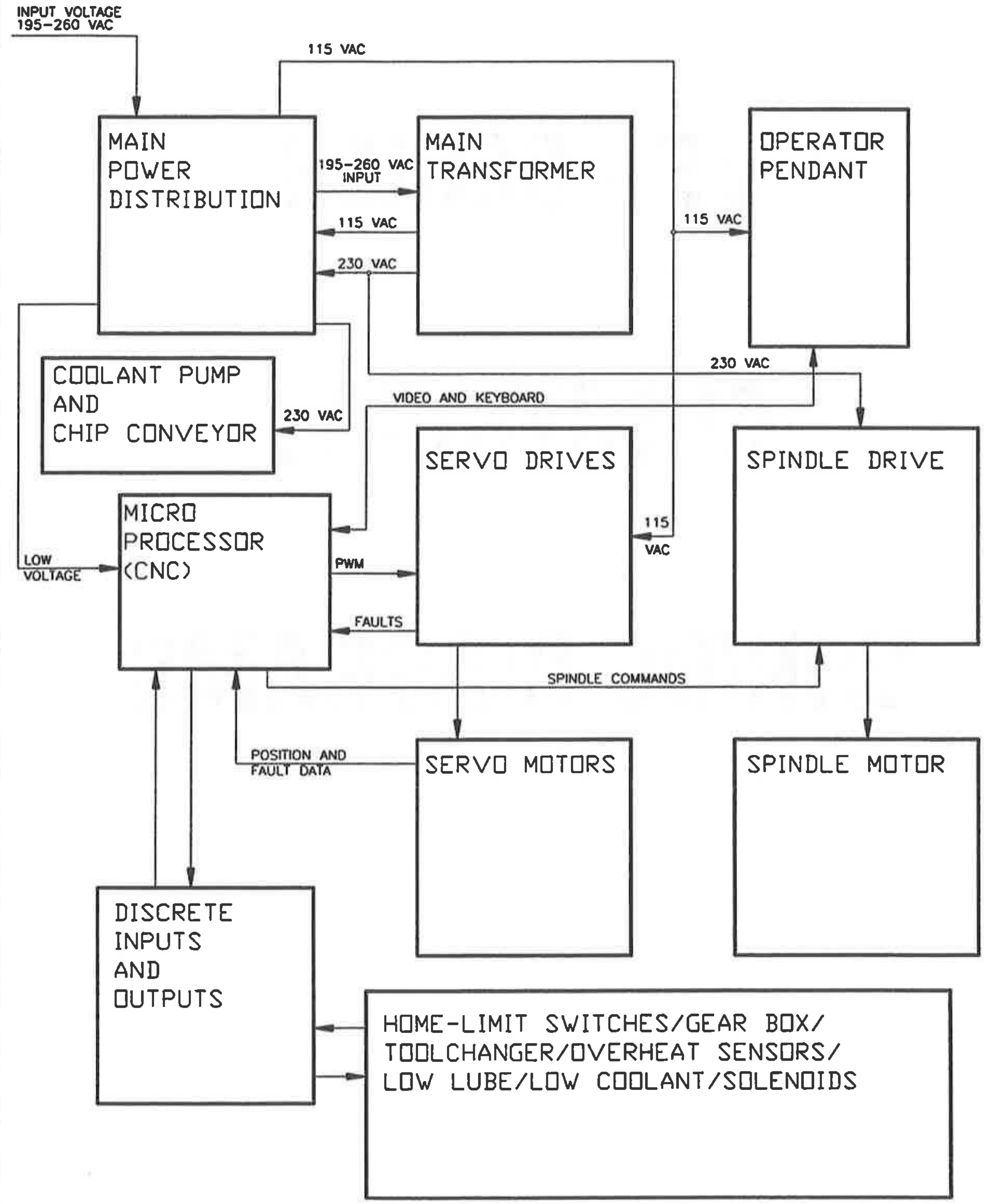
ANALOG DATA

Name	Description
DC BUSS	DC Servo Buss Voltage
SP TEMP	Spindle temperature F
SP LOAD	Spindle load in %
AUX TMP	Not used
SP SPEED	Spindle RPM CW or CCW

VF - SERIES

ELECTRICAL

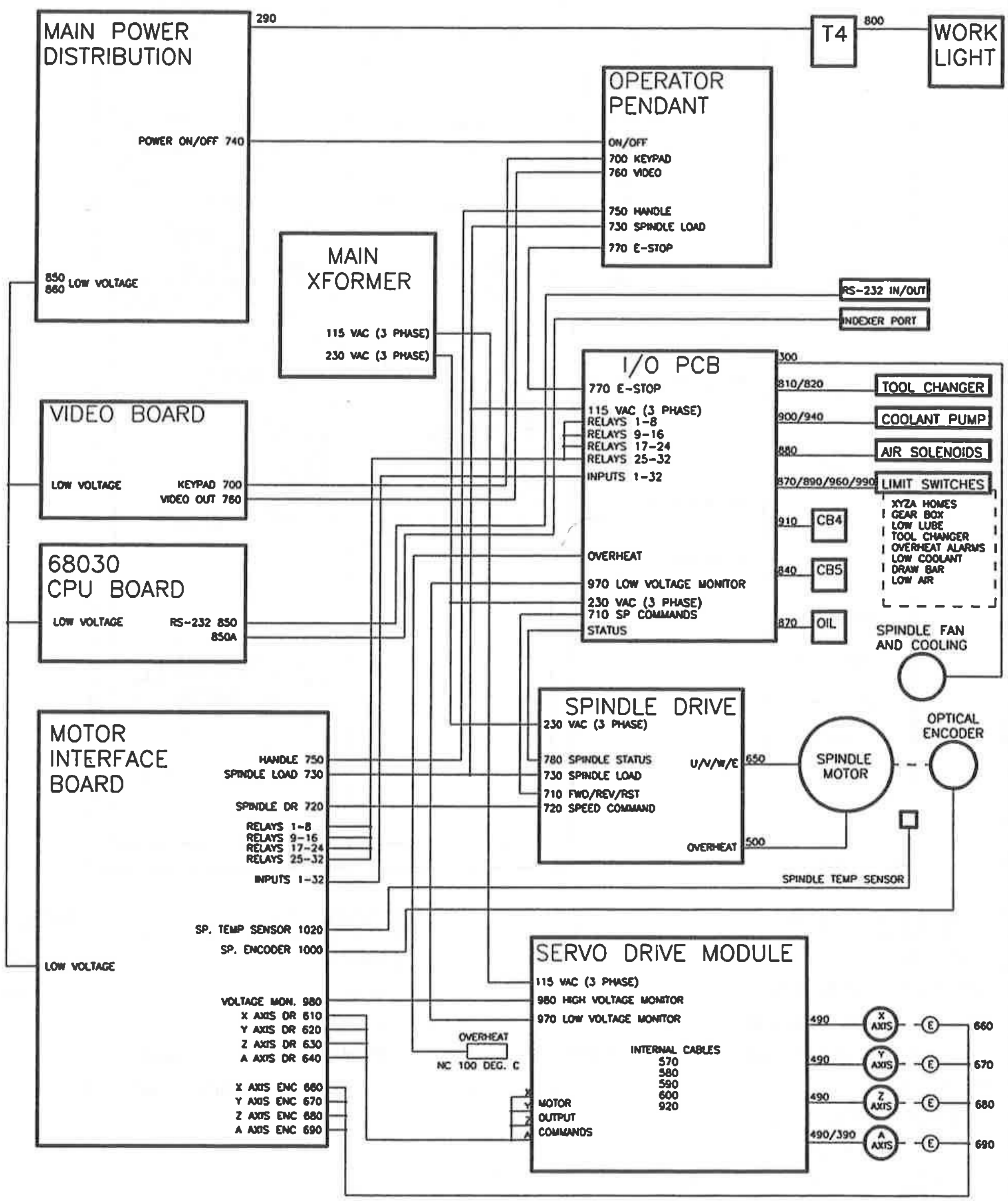
WIRING DIAGRAMS



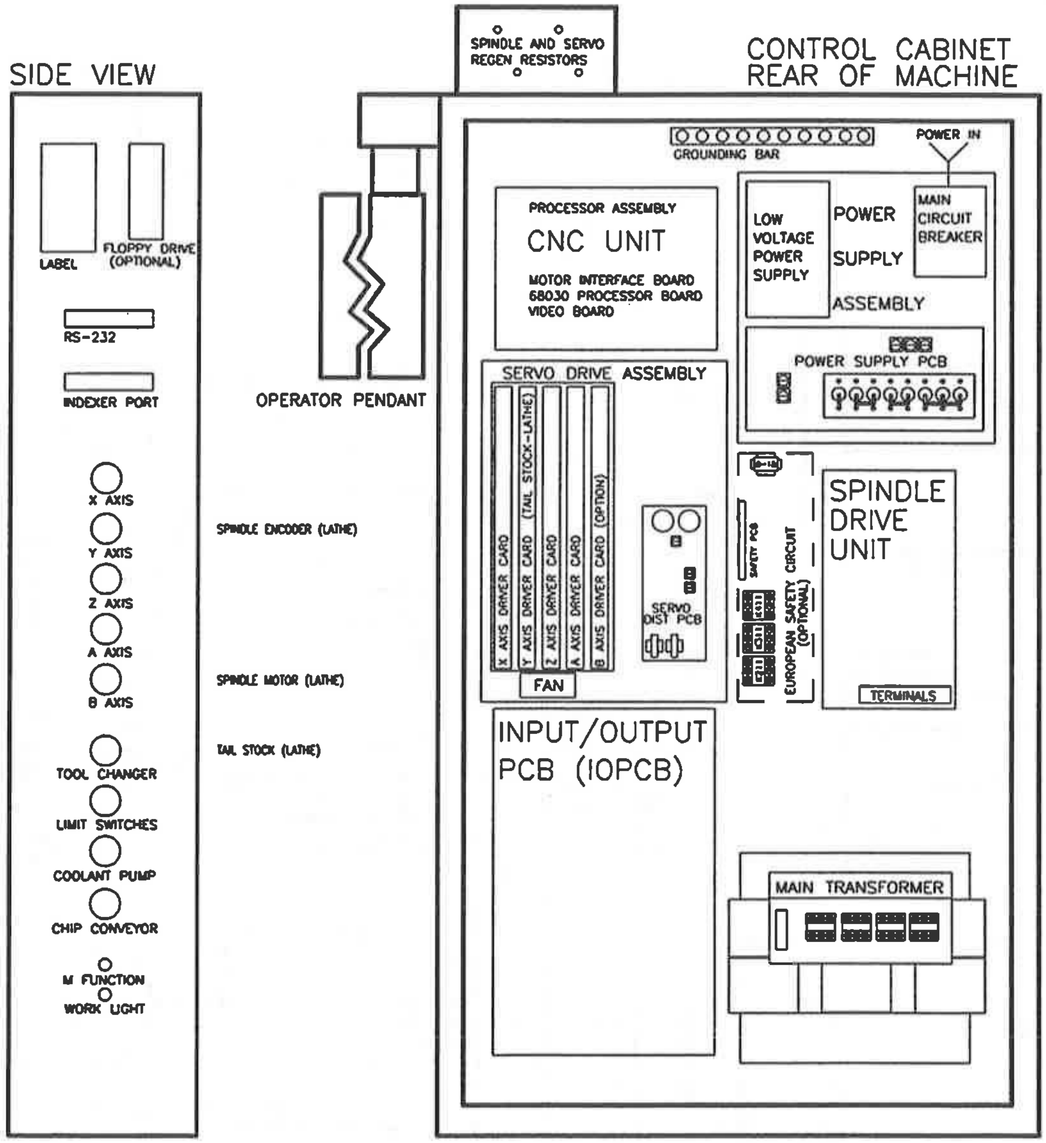
NOTE FOR HIGH VOLTAGE IN, SEE PAGE 16.

SYSTEM BLOCK DIAGRAM - LOW VOLTAGE

6/95 HAAS AUTOMATION VF/HS/HL SERIES PAGE B



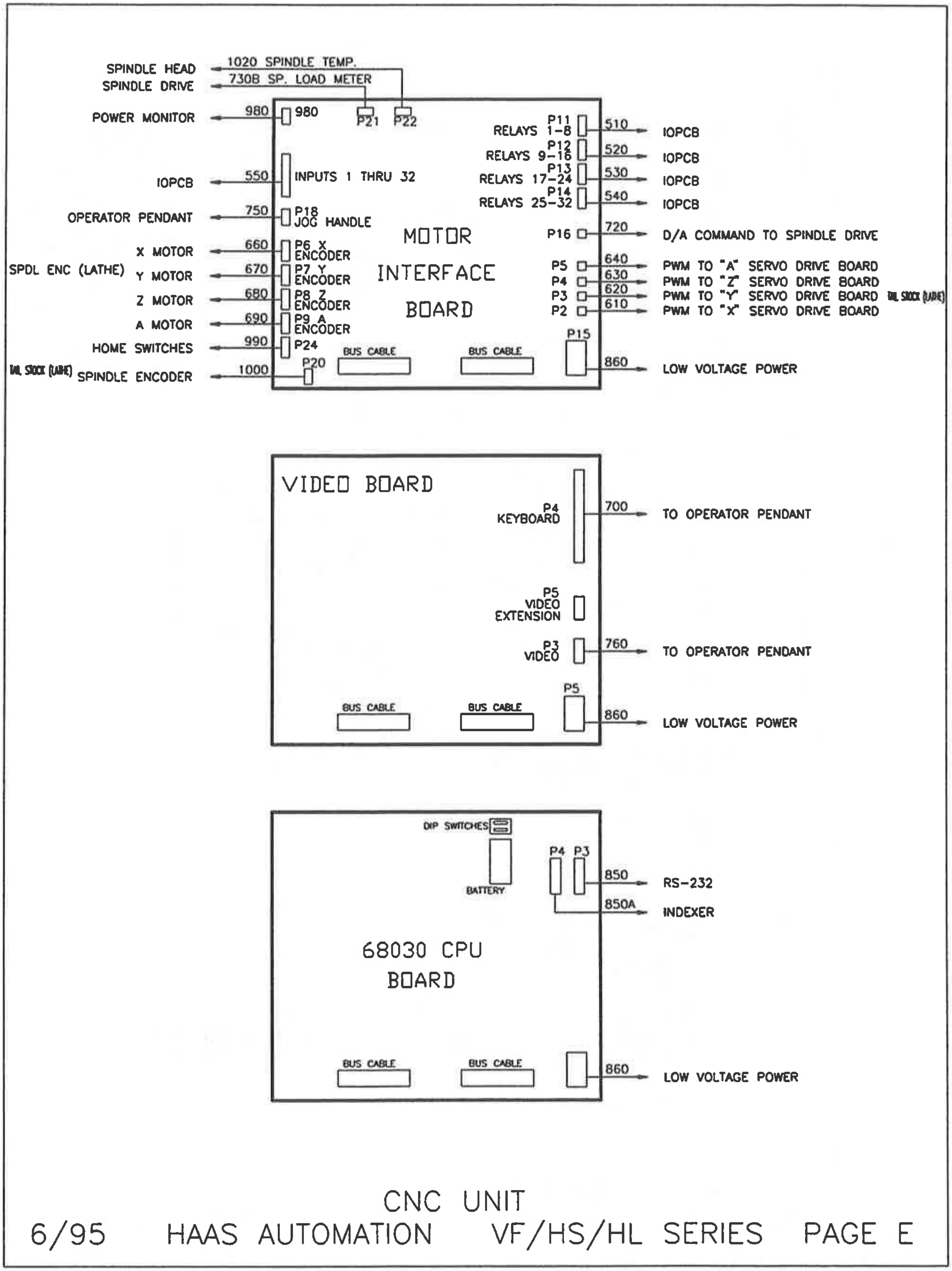
CABLE INTERCONNECT DIAGRAM
6/95 HAAS AUTOMATION VF/HS/HL SERIES PAGE C

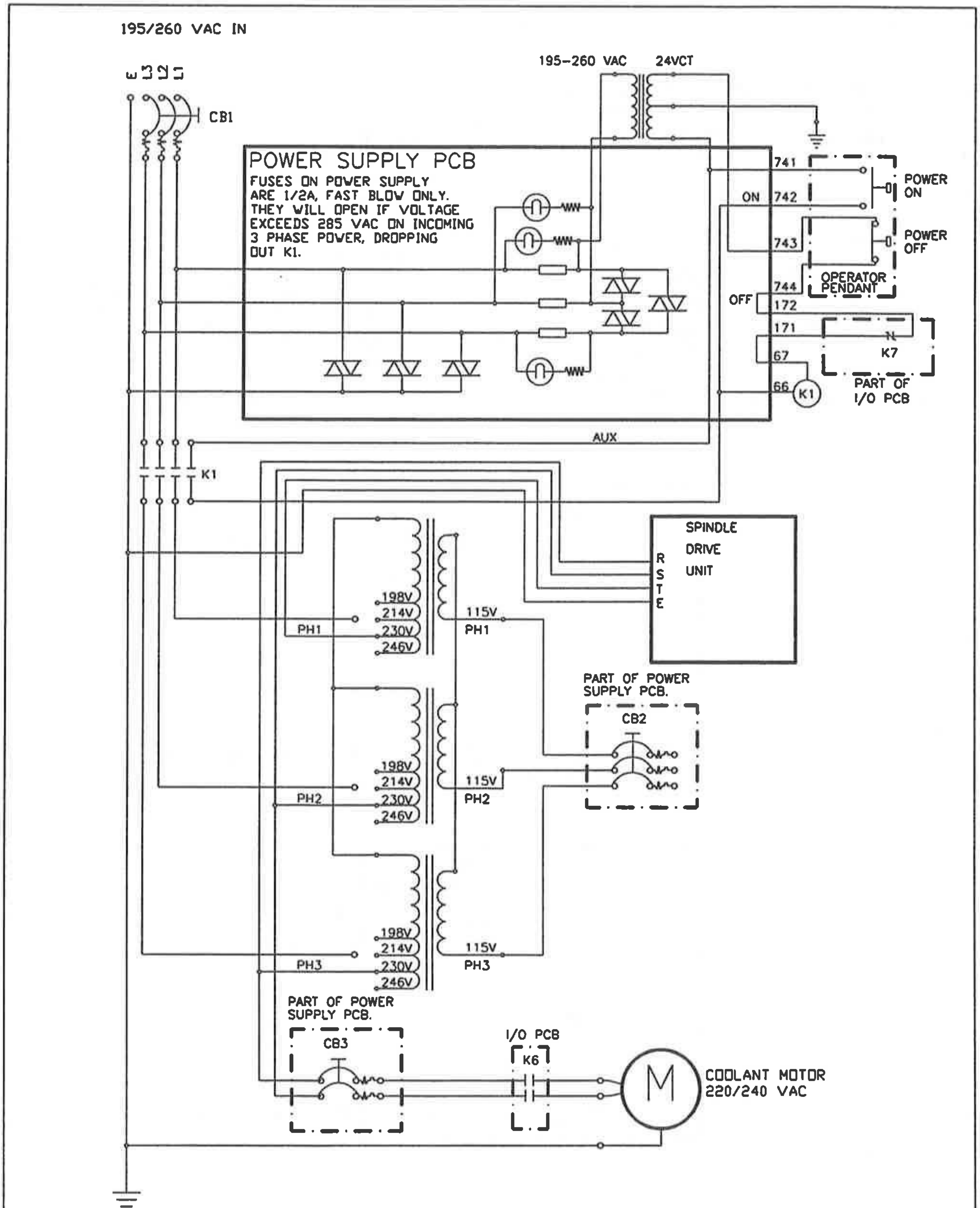


ITEM DESCRIPTION	PAGE #	ITEM DESCRIPTION	PAGE #
CNC UNIT	0E	TOOL CHANGER	10
SPINDLE DRIVE UNIT	02	SOLENOIDS	04
SERVO DRIVE ASSEMBLY	03	SERVO DRIVE CARDS	03
INPUT/OUTPUT BOARD	5,6,7,8,AND 9	TRANSFORMER HIGH VOLTAGE	16
MAIN TRANSFORMER	01	CHIP CONVEYOR /SPIGOT MOTOR	17
OPERATOR PENDANT	12		
CONNECTORS P1 THRU P4	11		
CONNECTORS P5, P6, AND P7	13		

CONTROL LAYOUT DIAGRAM

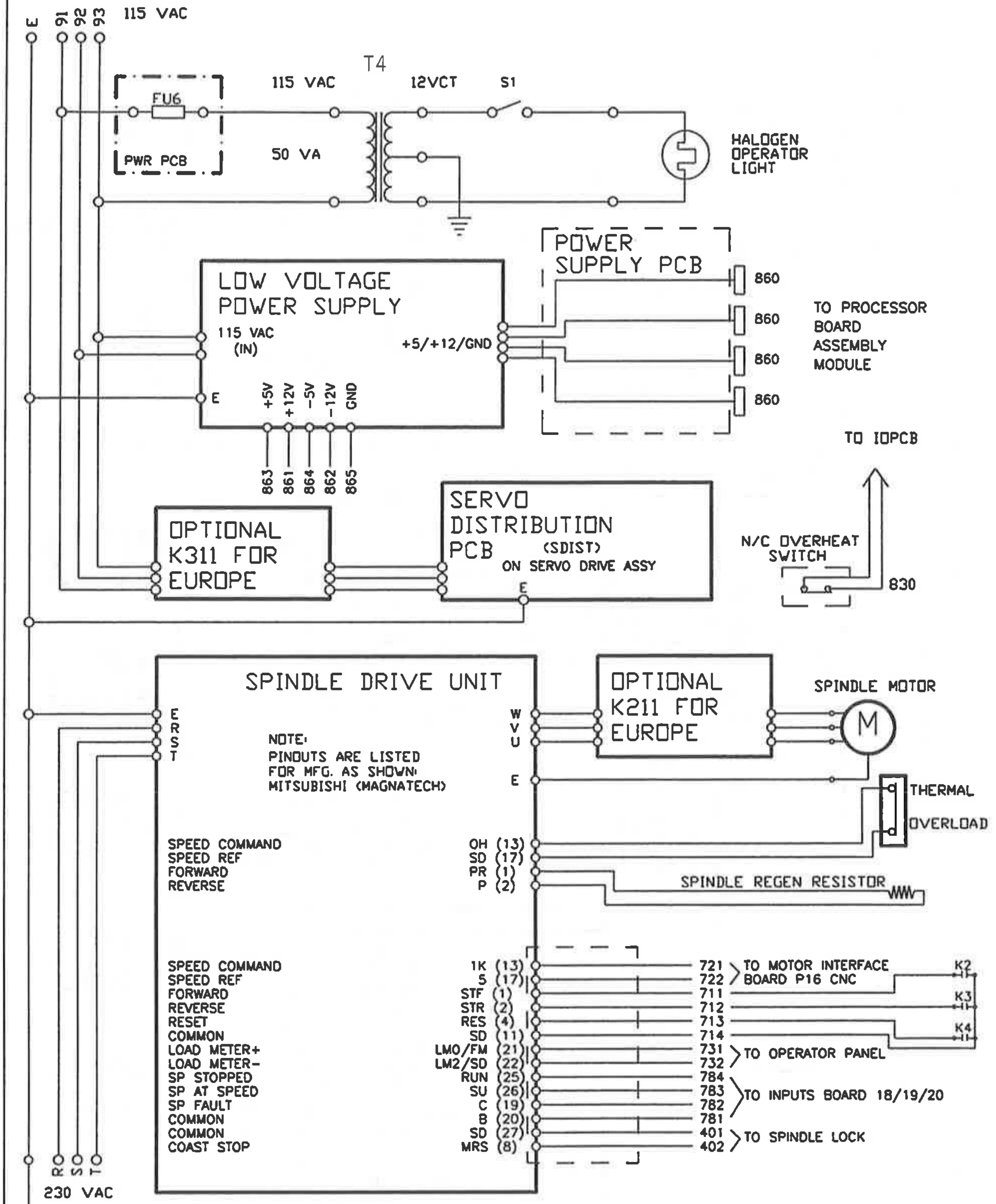
6/95 HAAS AUTOMATION VF/HS/HL SERIES PAGE D


SPINDLE HEAD
SPINDLE DRIVE
1020 SPINDLE TEMP.
730B SP. LOAD METER
POWER MONITOR
980
P21
P22
IOPCB
550
INPUTS 1 THRU 32
OPERATOR PENDANT
750
P18 JOG HANDLE
X MOTOR
SPDL ENC (LATHE)
Y MOTOR
Z MOTOR
A MOTOR
HOME SWITCHES
TAIL STOCK (LATHE)
SPINDLE ENCODER
660
670
680
690
990
1000
P6 X ENCODER
P7 Y ENCODER
P8 Z ENCODER
P9 A ENCODER
P24
P20
BUS CABLE
MOTOR INTERFACE BOARD
RELAYS 1-8 P11
RELAYS 9-16 P12
RELAYS 17-24 P13
RELAYS 25-32 P14
510
520
530
540
IOPCB
P16
720
D/A COMMAND TO SPINDLE DRIVE
P5
P4
P3
P2
640
630
620
610
PWM TO "A" SERVO DRIVE BOARD
PWM TO "Z" SERVO DRIVE BOARD
PWM TO "Y" SERVO DRIVE BOARD
PWM TO "X" SERVO DRIVE BOARD
TAIL STOCK (LATHE)
P15
860
LOW VOLTAGE POWER
VIDEO BOARD
P4 KEYBOARD
700
TO OPERATOR PENDANT
P5 VIDEO EXTENSION
P3 VIDEO
760
TO OPERATOR PENDANT
P5
860
LOW VOLTAGE POWER
DIP SWITCHES
BATTERY
P4
P3
850
RS-232
850A
INDEXER
68030 CPU BOARD
BUS CABLE
860
LOW VOLTAGE POWER
CNC UNIT
6/95 HAAS AUTOMATION VF/HS/HL SERIES PAGE E



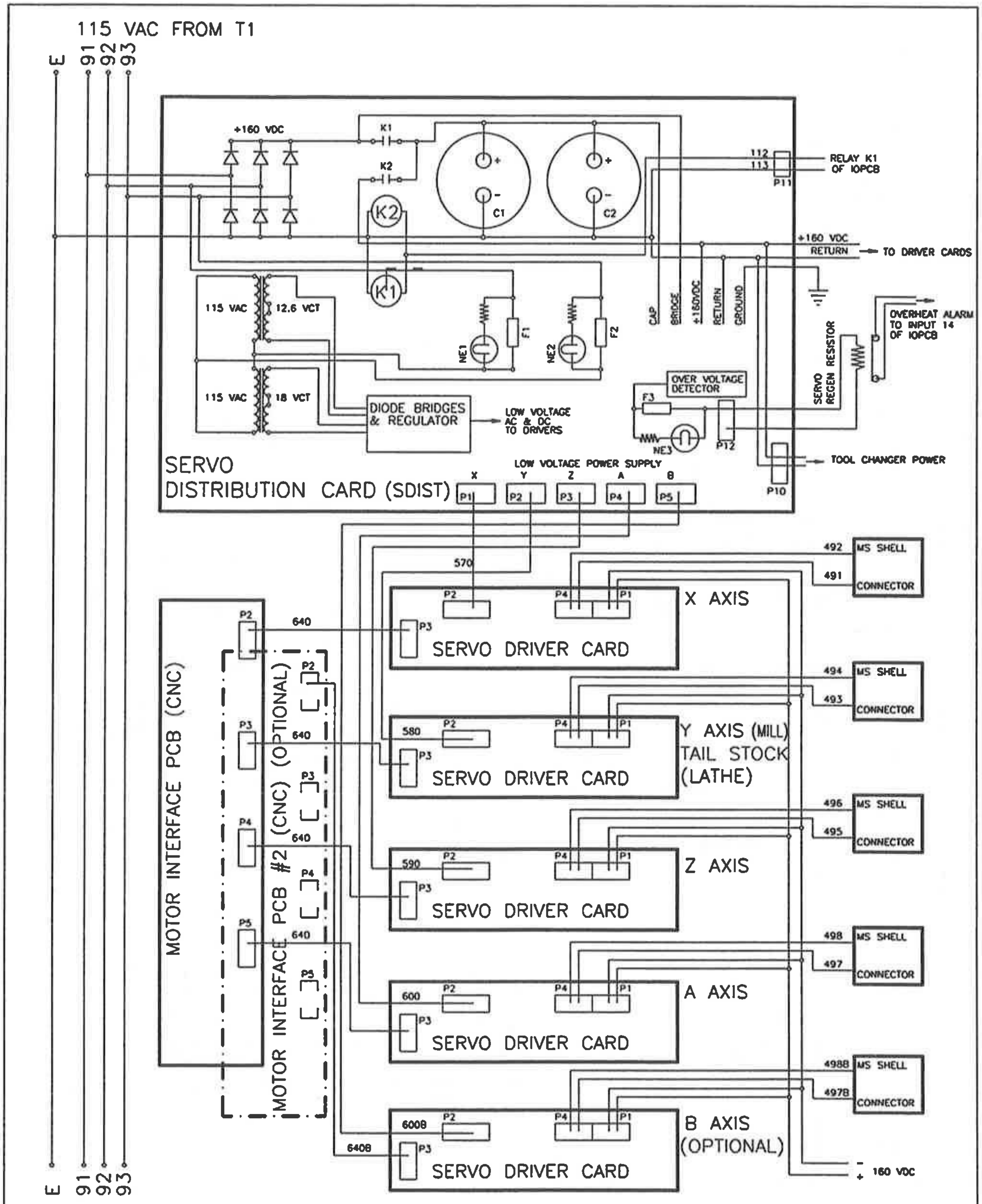
MAIN TRANSFORMER – LOW VOLTAGE

6/95 HAAS AUTOMATION VF/HS/HL SERIES PAGE 1



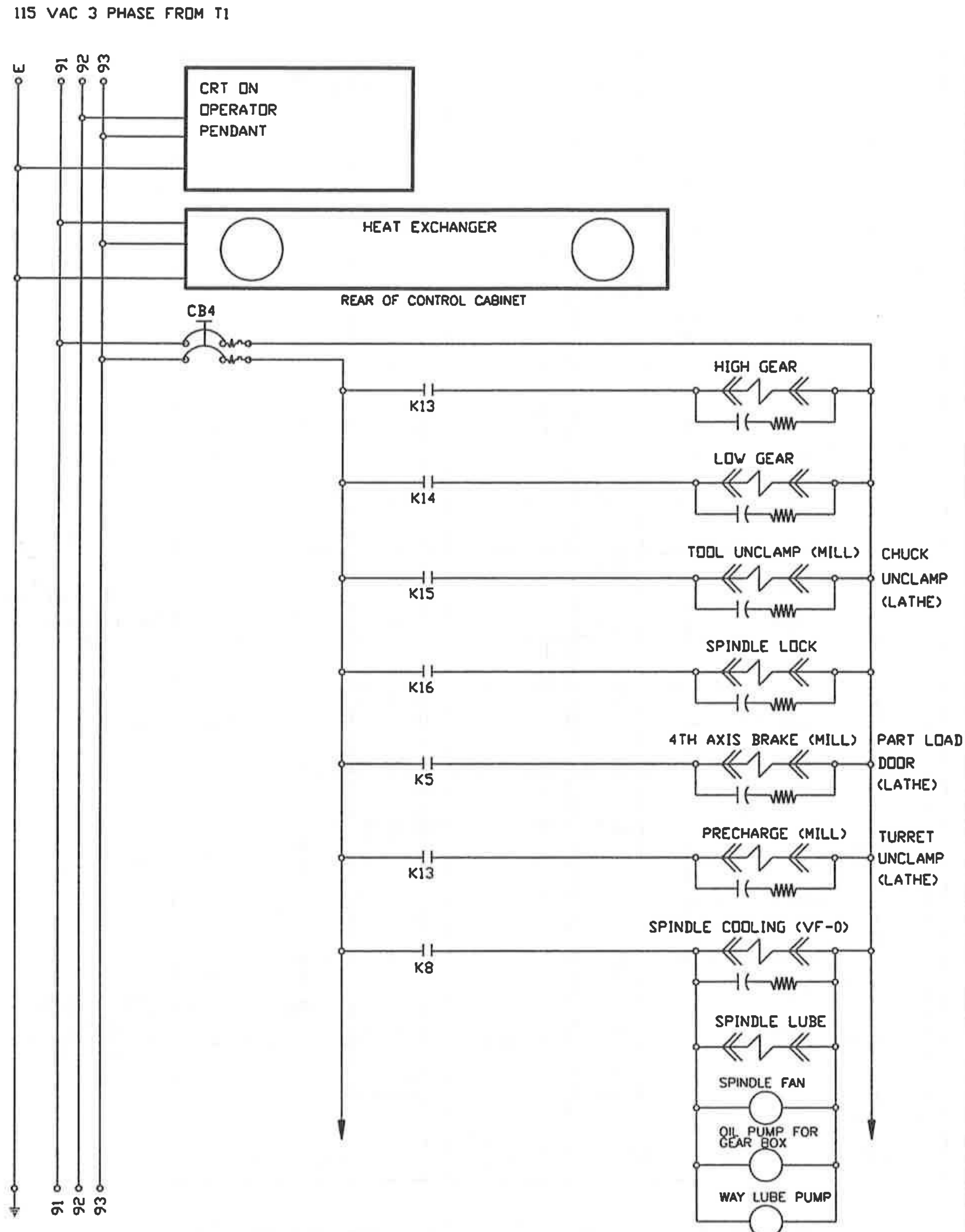
SPINDLE DRIVE UNIT

6/95 HAAS AUTOMATION VF/HS/HL SERIES PAGE 2



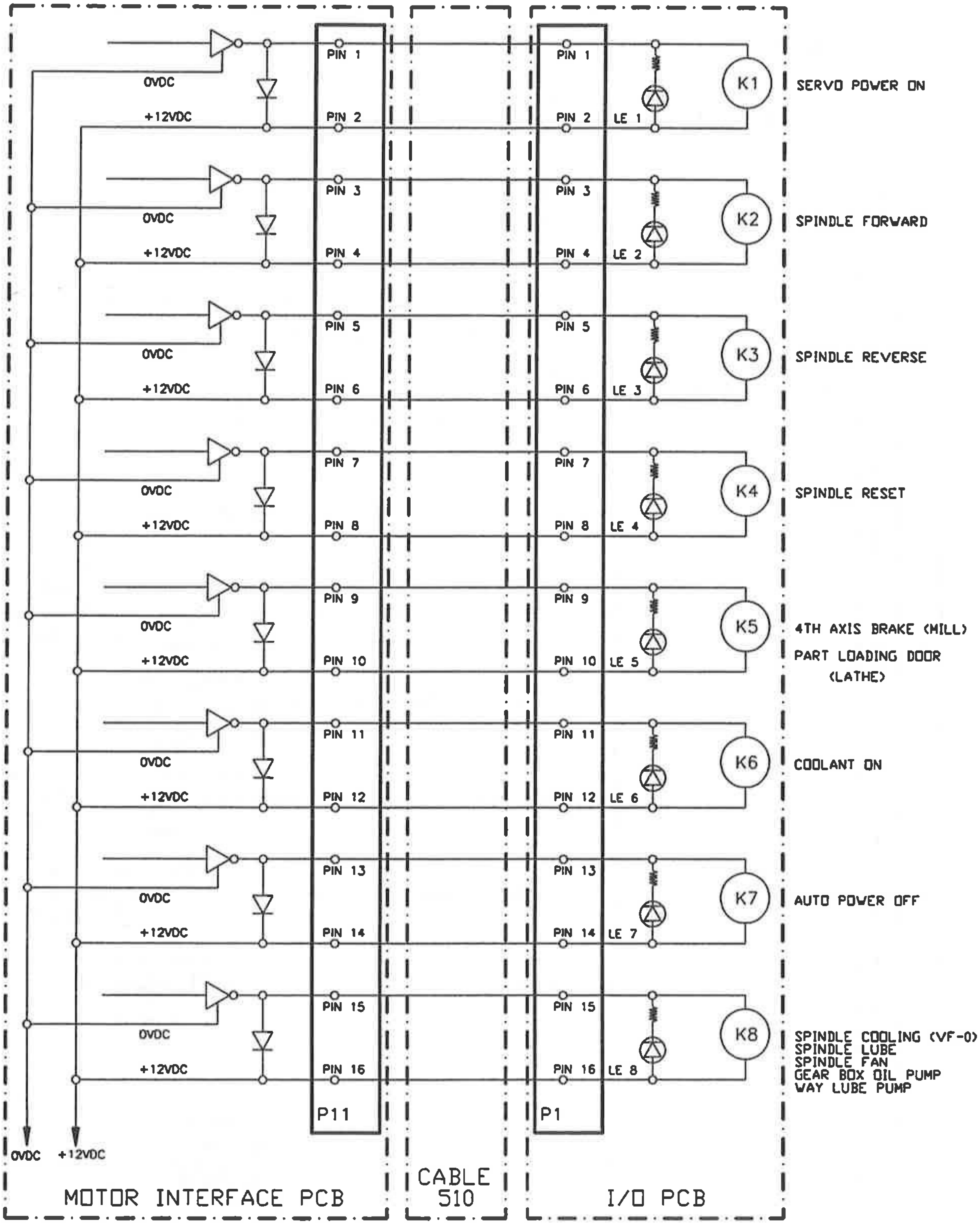
SERVO SYSTEM

6/95 HAAS AUTOMATION VF/HS/HL SERIES PAGE 3



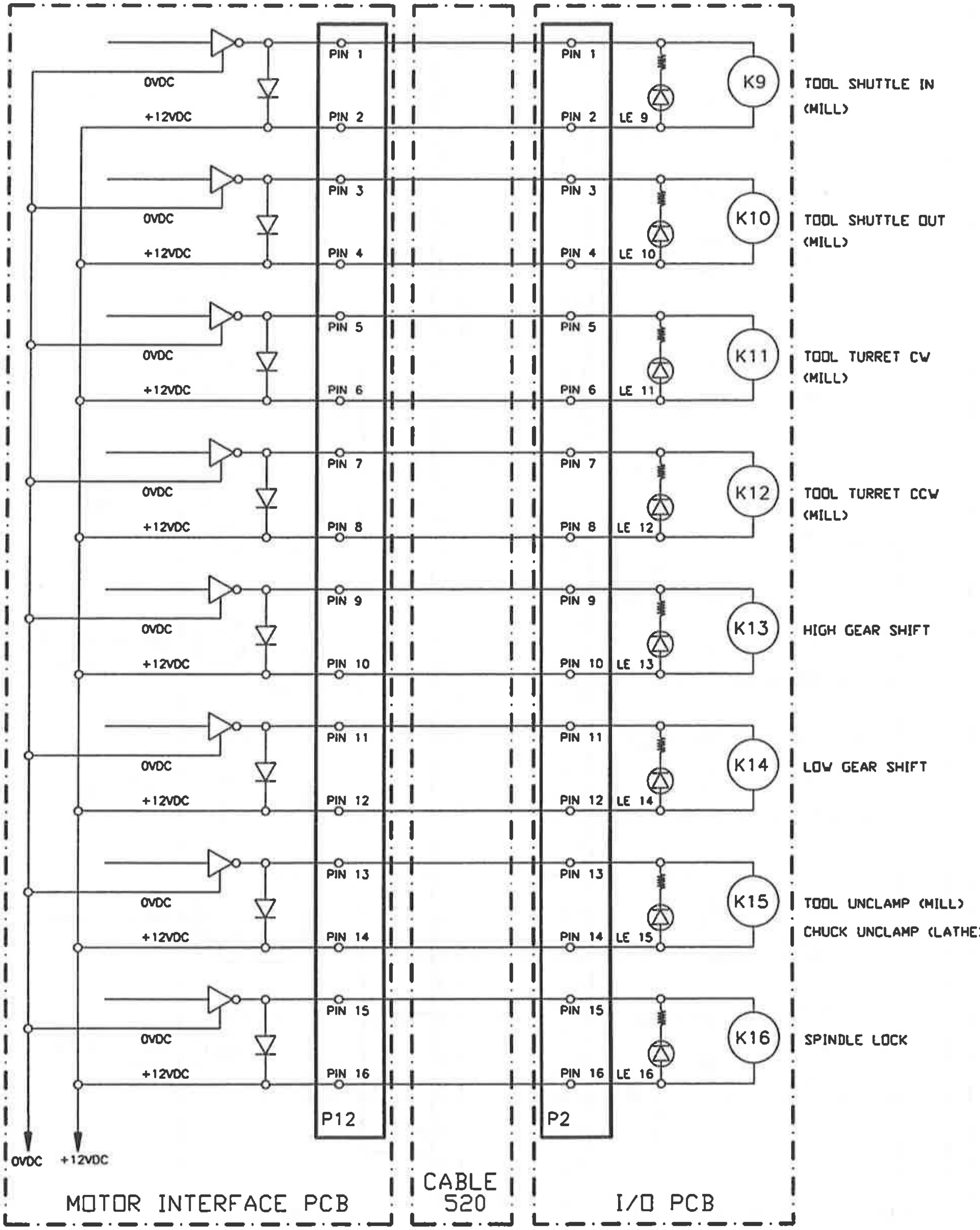
115 VAC CIRCUITS

6/95 HAAS AUTOMATION VF/HS/HL SERIES PAGE 4



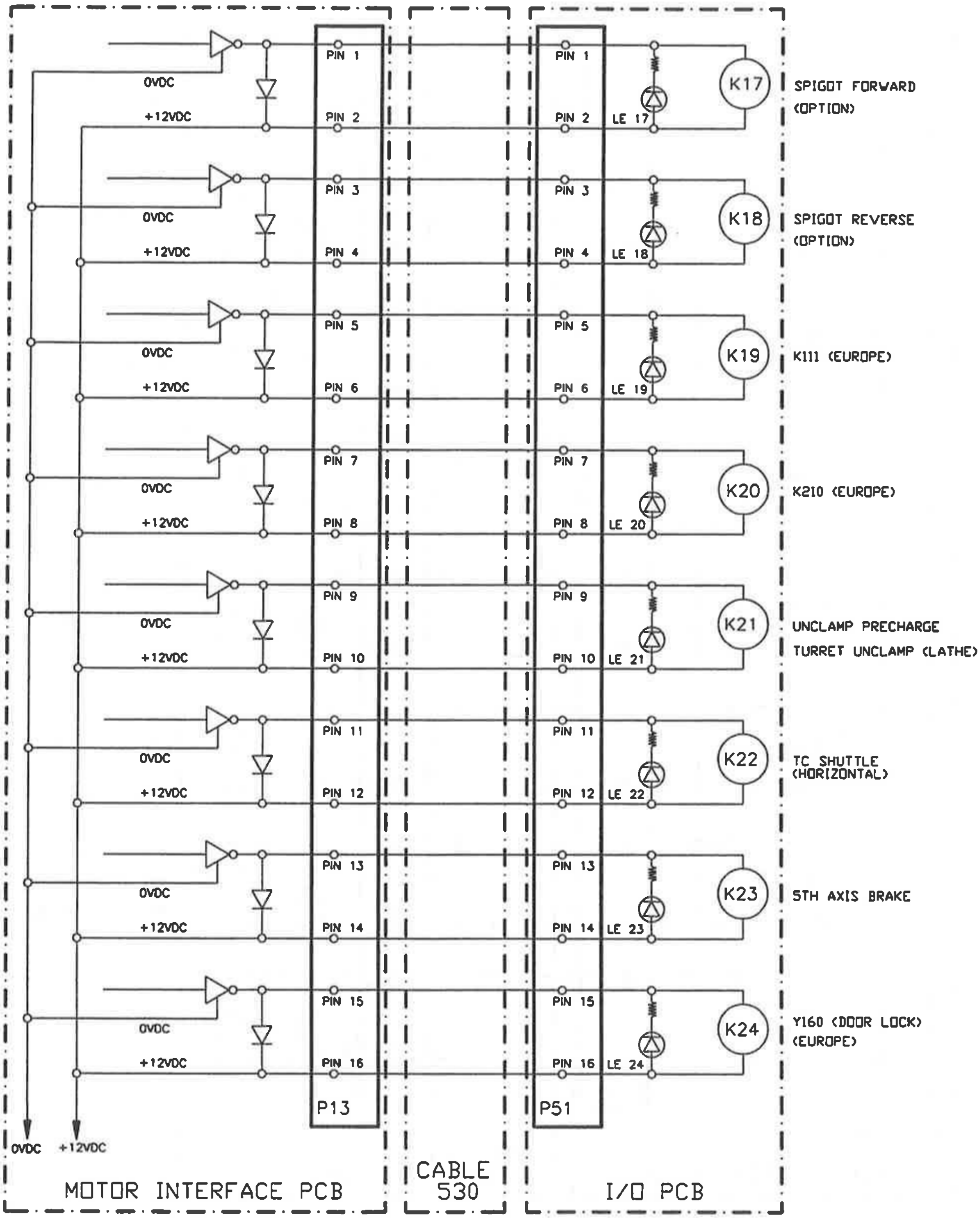
RELAY COIL DRIVERS, K1 THROUGH K8

6/95 HAAS AUTOMATION VF/HS/HL SERIES PAGE 5



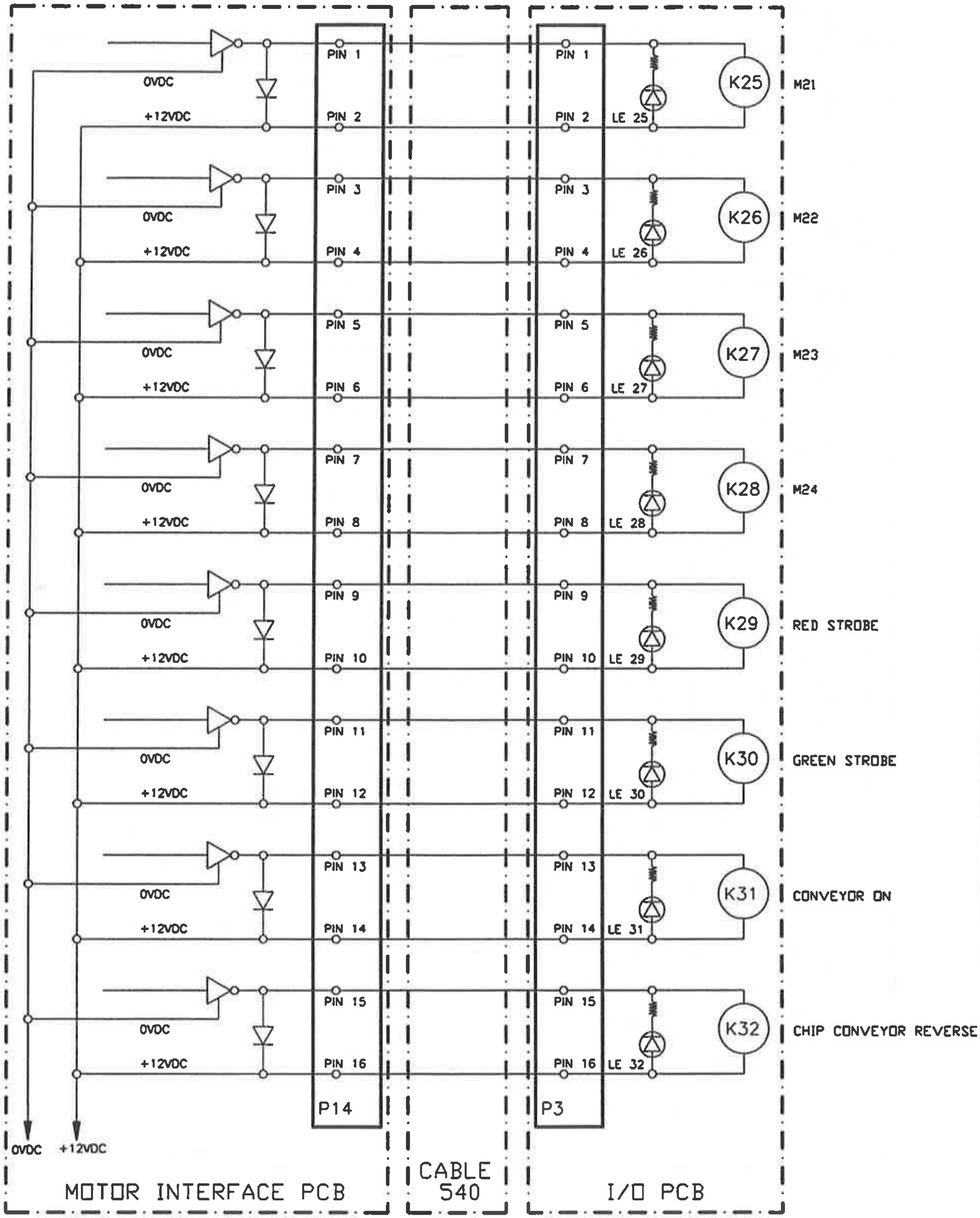
RELAY COIL DRIVERS, K9 THROUGH K16

6/95 HAAS AUTOMATION VF/HS/HL SERIES PAGE 6

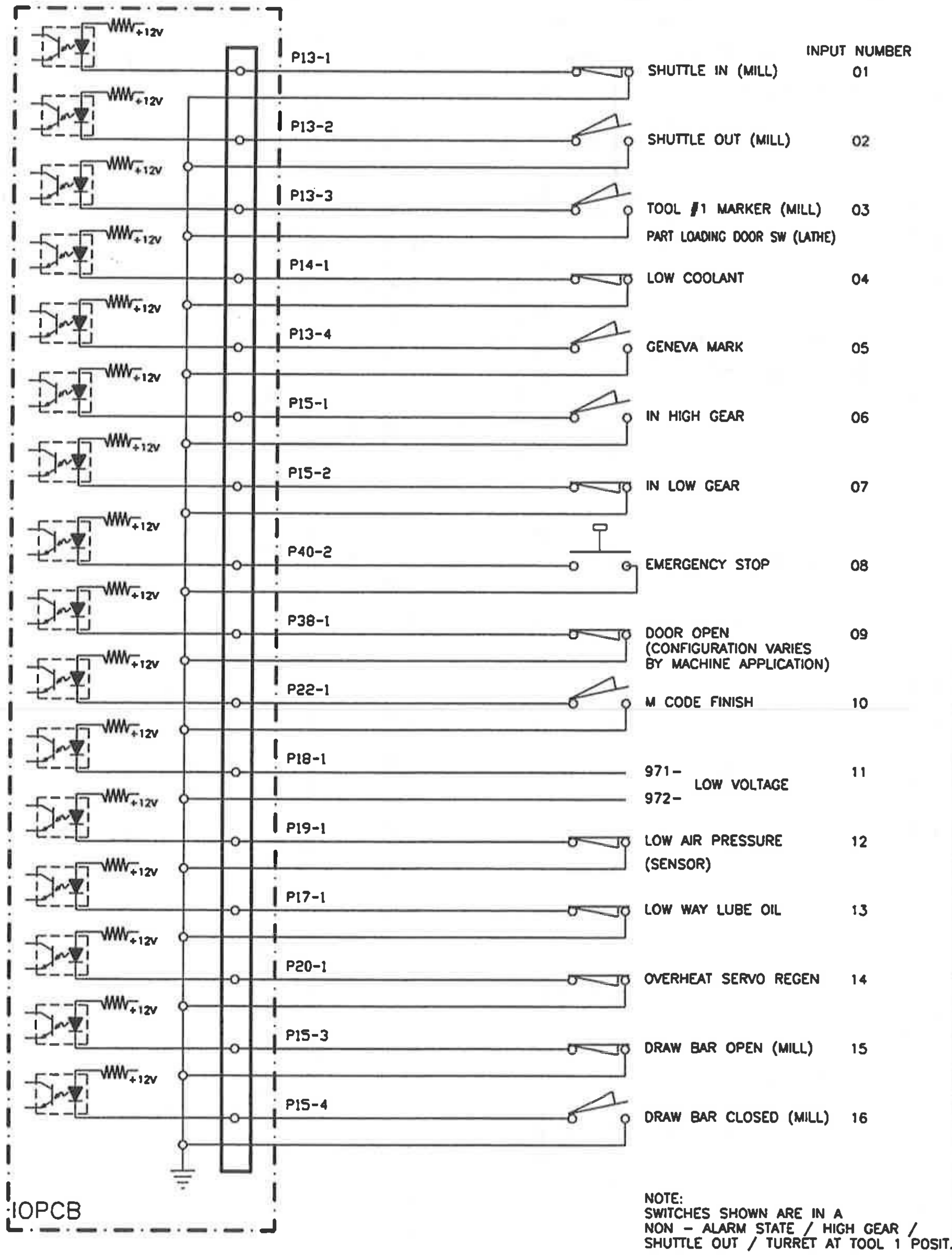


RELAY COIL DRIVERS, K17 THROUGH K24

6/95 HAAS AUTOMATION VF/HS/HL SERIES PAGE 7

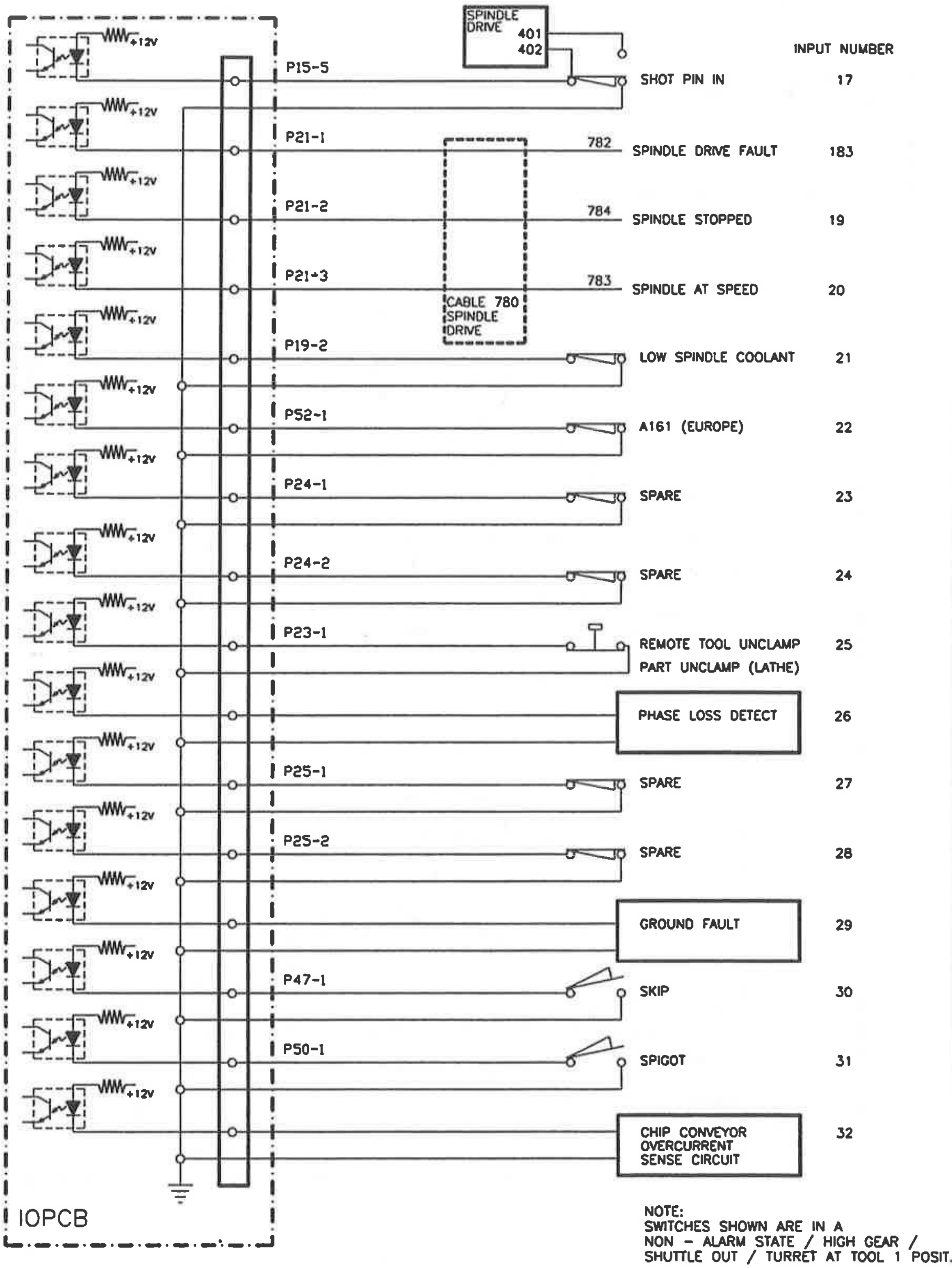


RELAY COIL DRIVERS, K25 THROUGH K32
6/95 HAAS AUTOMATION VF/HS/HL SERIES PAGE 8



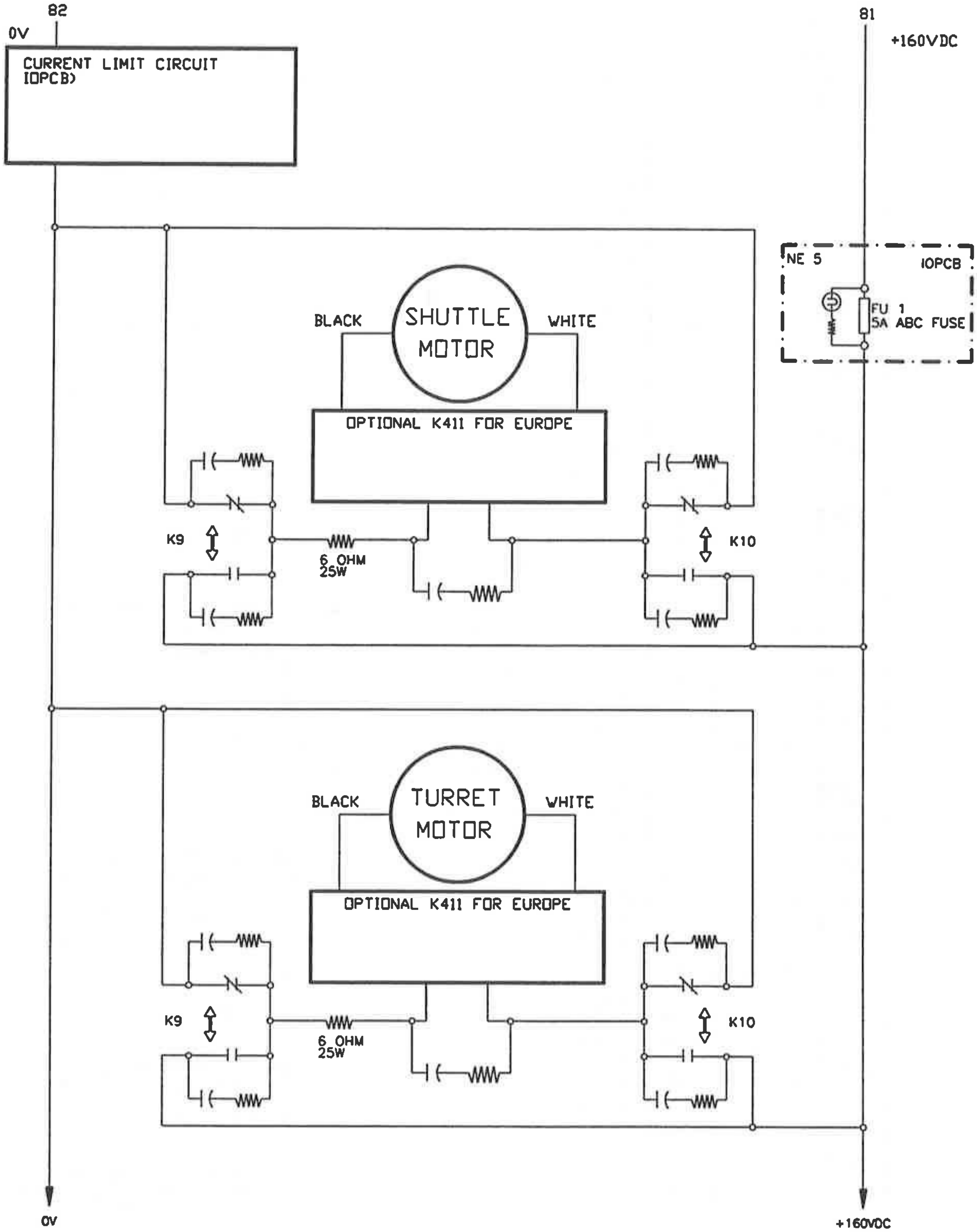
DISCRETE INPUTS 1 THROUGH 16

6/95 HAAS AUTOMATION VF/HS/HL SERIES PAGE 9



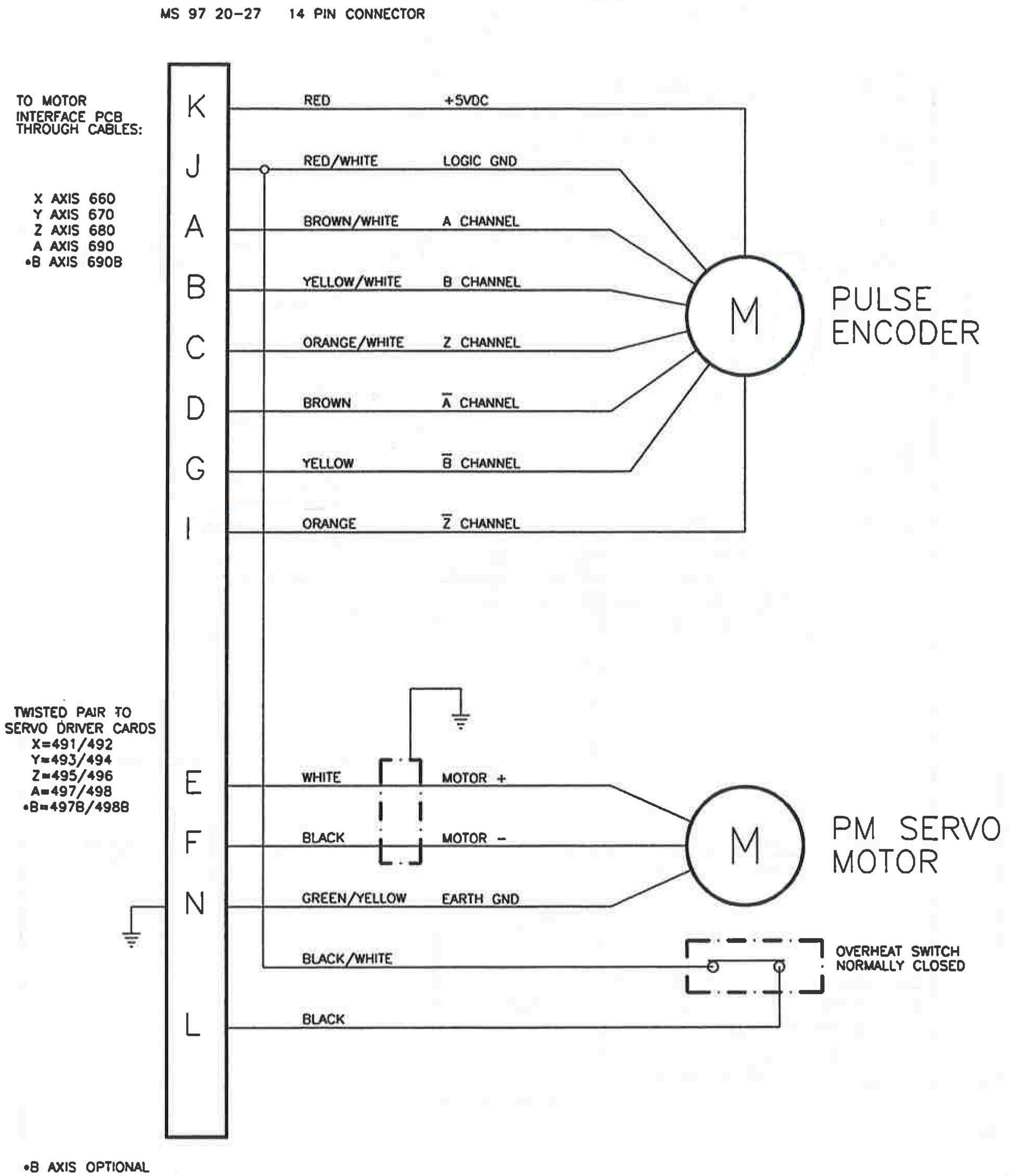
DISCRETE INPUTS 17 THROUGH 32

6/95 HAAS AUTOMATION VF/HS/HL SERIES PAGE 10



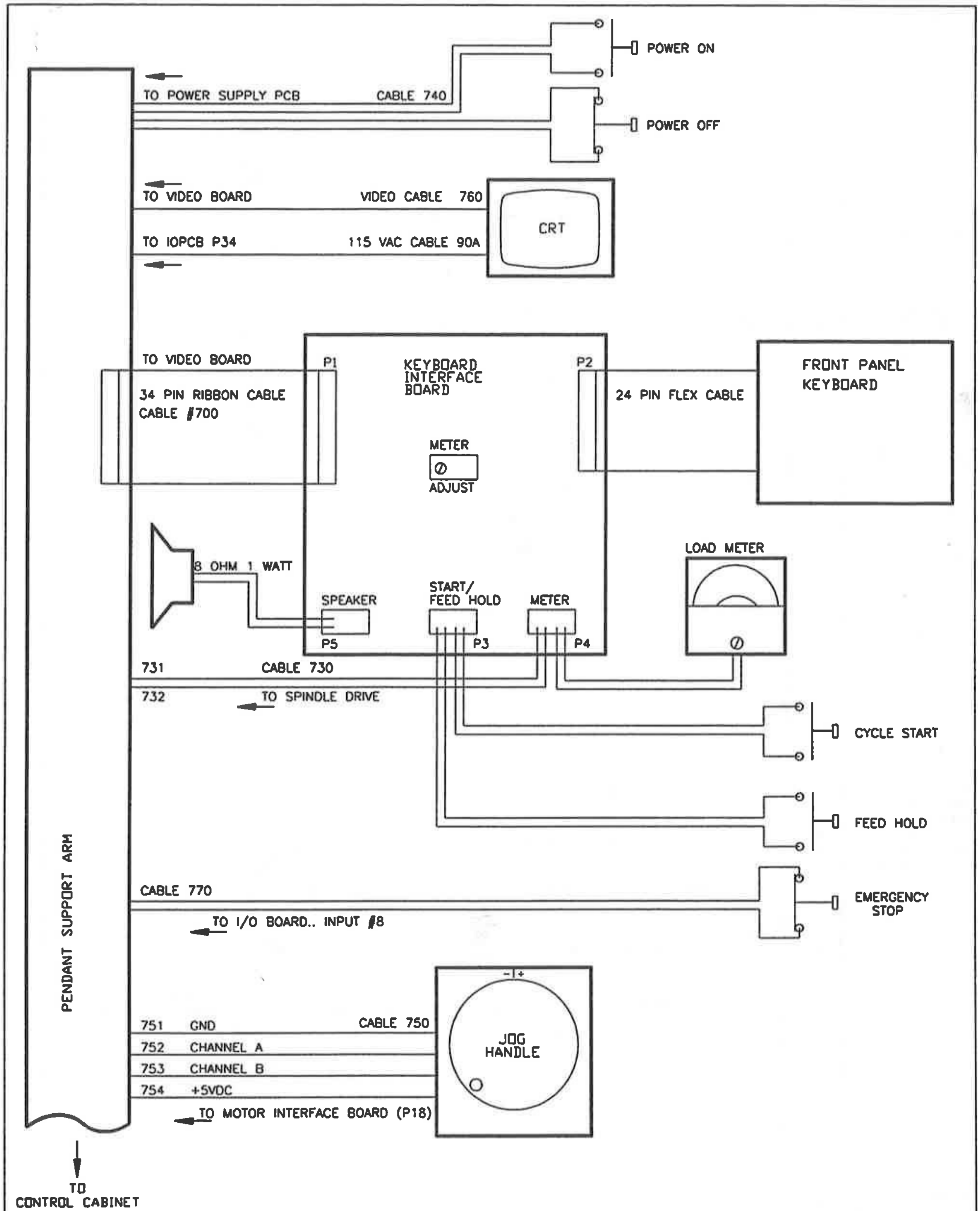
TOOL CHANGE MOTORS

6/95 HAAS AUTOMATION VF/HS/HL SERIES PAGE 11



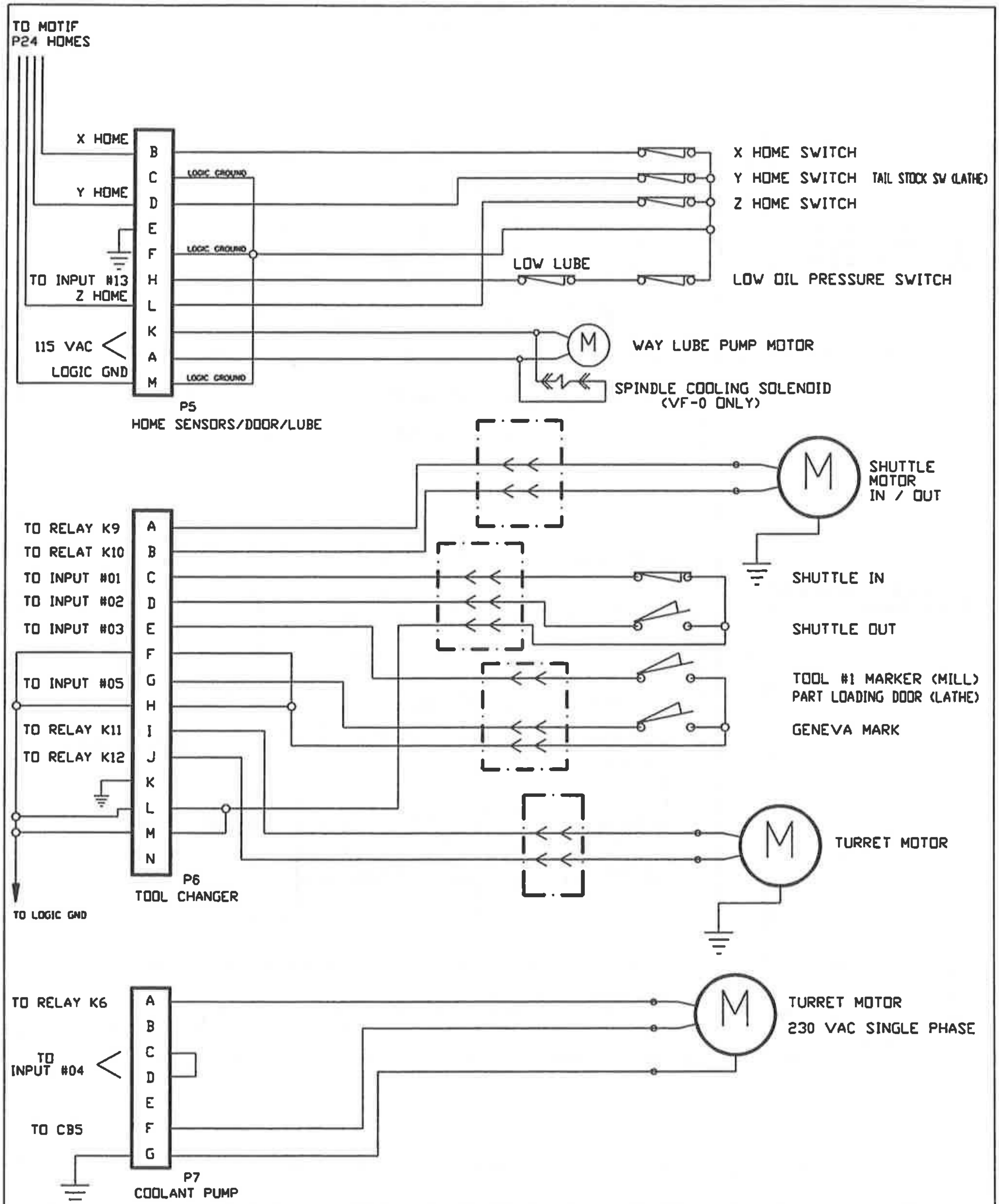
X/Y/Z AXIS MOTOR & ENCODER

6/95 HAAS AUTOMATION VF/HS/HL SERIES PAGE 12



OPERATOR PENDANT

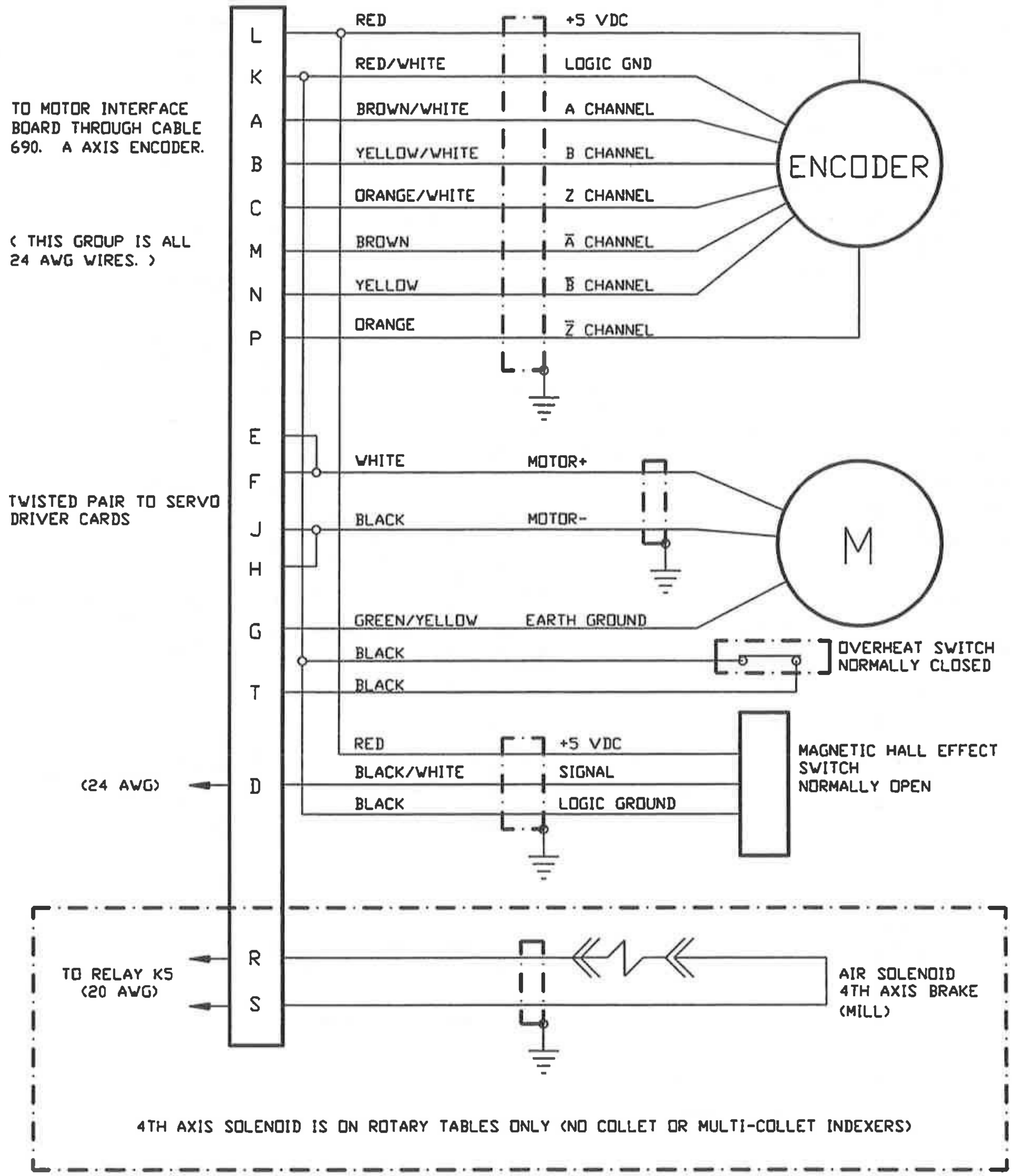
6/95 HAAS AUTOMATION VF/HS/HL SERIES PAGE 13



NOTE: CONNECTORS ARE LOCATED ON SIDE OF CONTROL CABINET.

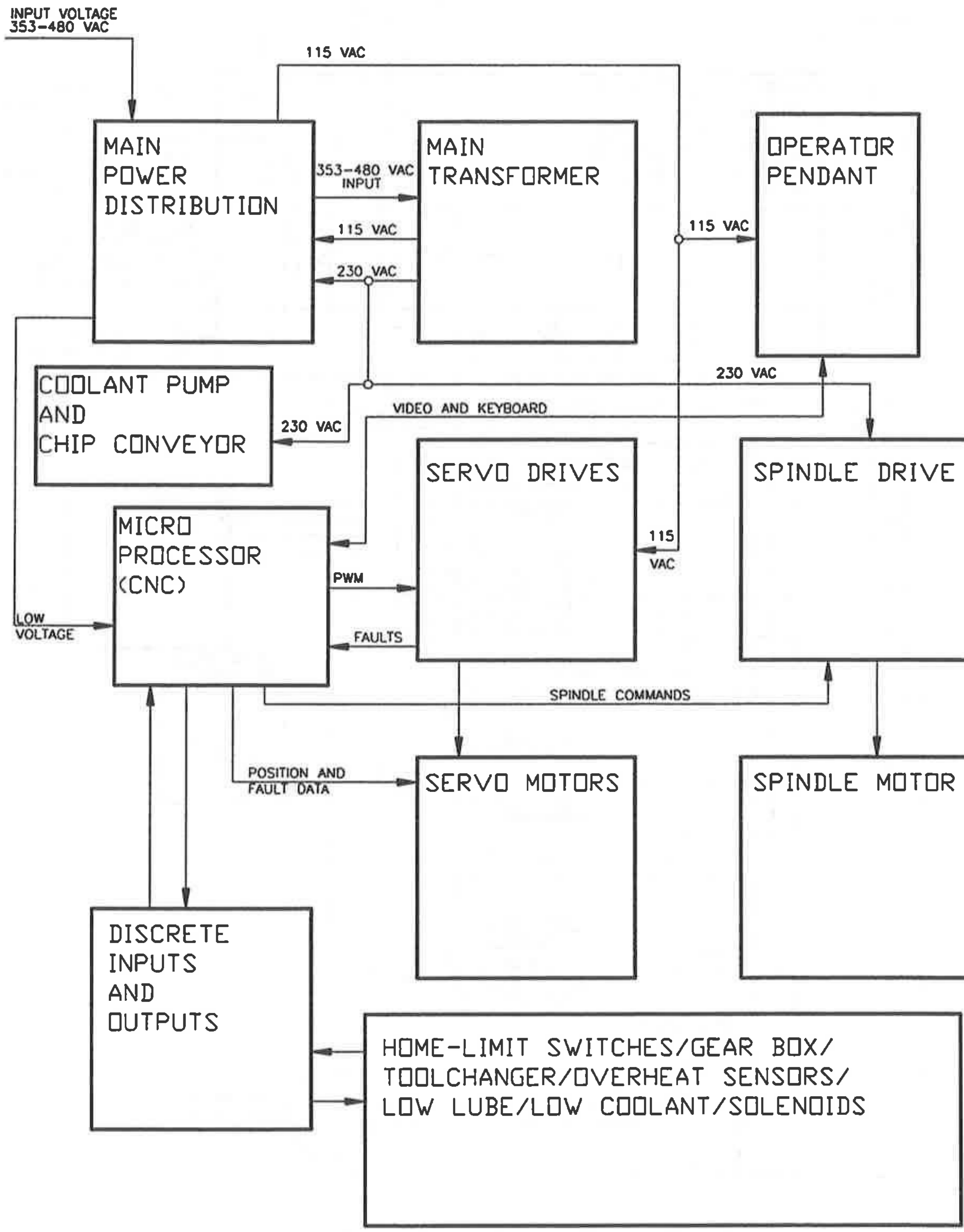
CABINET CONNECTORS

6/95 HAAS AUTOMATION VF/HS/HL SERIES PAGE 14



A AXIS

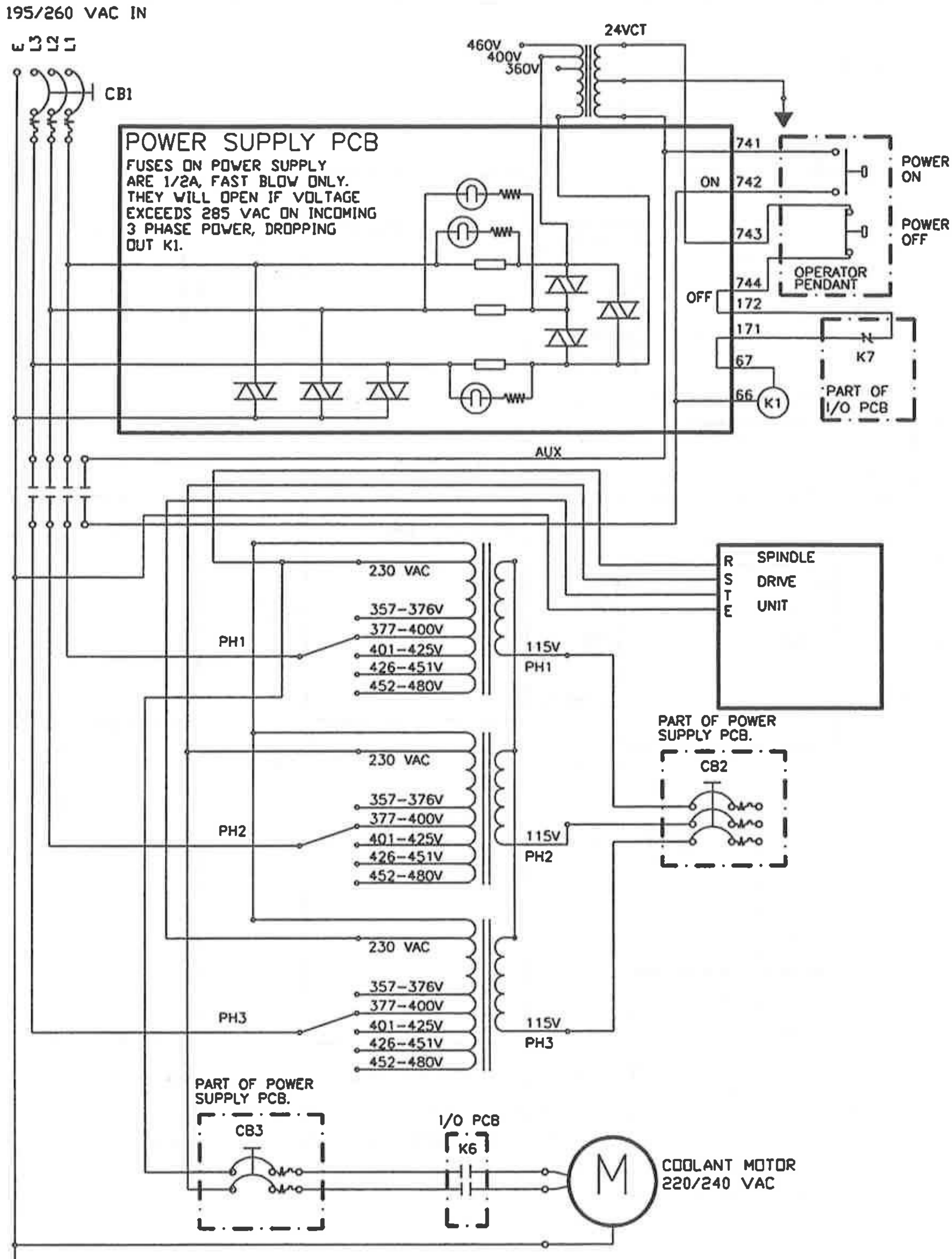
6/95 HAAS AUTOMATION VF/HS/HL SERIES PAGE 15



NOTE FOR LOW VOLTAGE IN, SEE PAGE B.

SYSTEM BLOCK DIAGRAM – HIGH VOLTAGE6

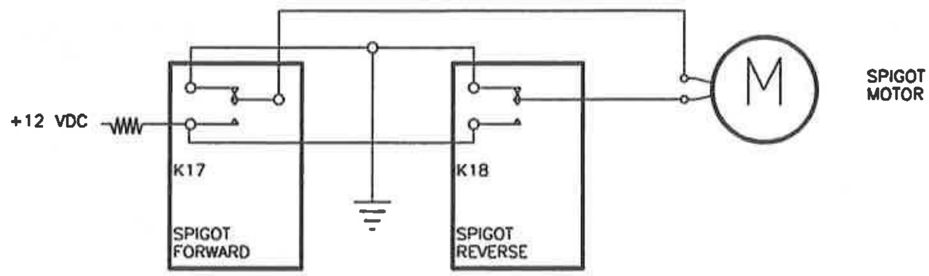
6/95 HAAS AUTOMATION VF/HS/HL SERIES PAGE 16

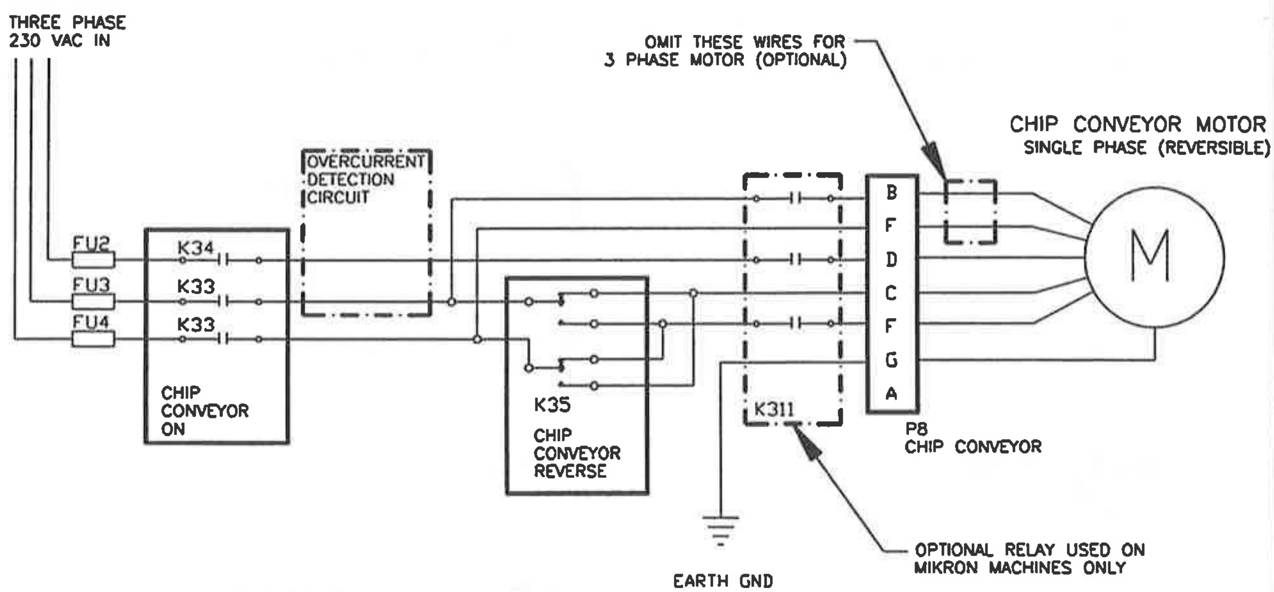


NOTE: FOR LOW VOLTAGE IN, SEE PAGE 1.

MAIN TRANSFORMER – HIGH VOLTAGE

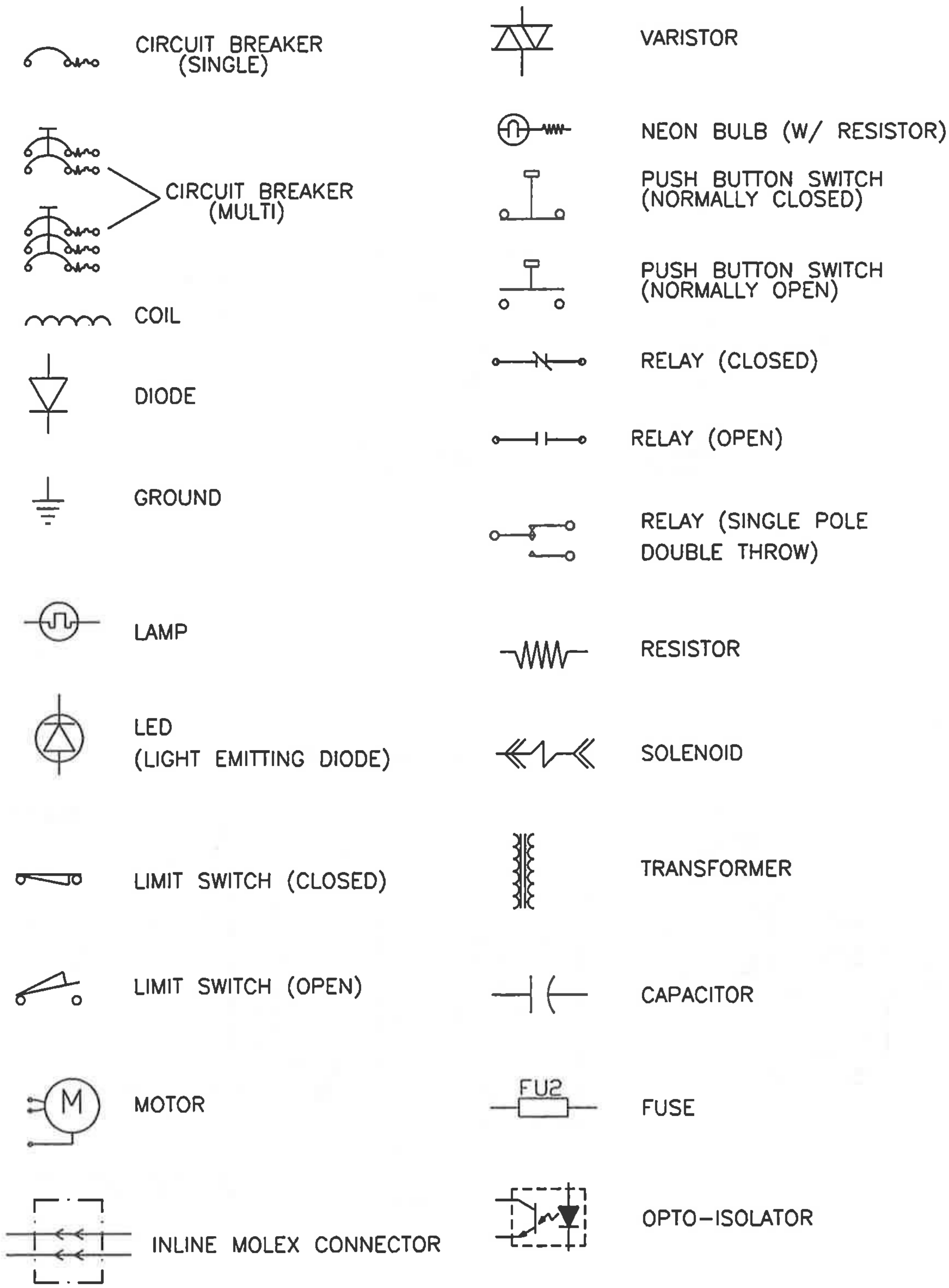
6/95 HAAS AUTOMATION VF/HS/HL SERIES PAGE 17





SPIGOT AND CHIP CONVEYOR MOTORS

6/95 HAAS AUTOMATION VF/HS/HL SERIES PAGE 18



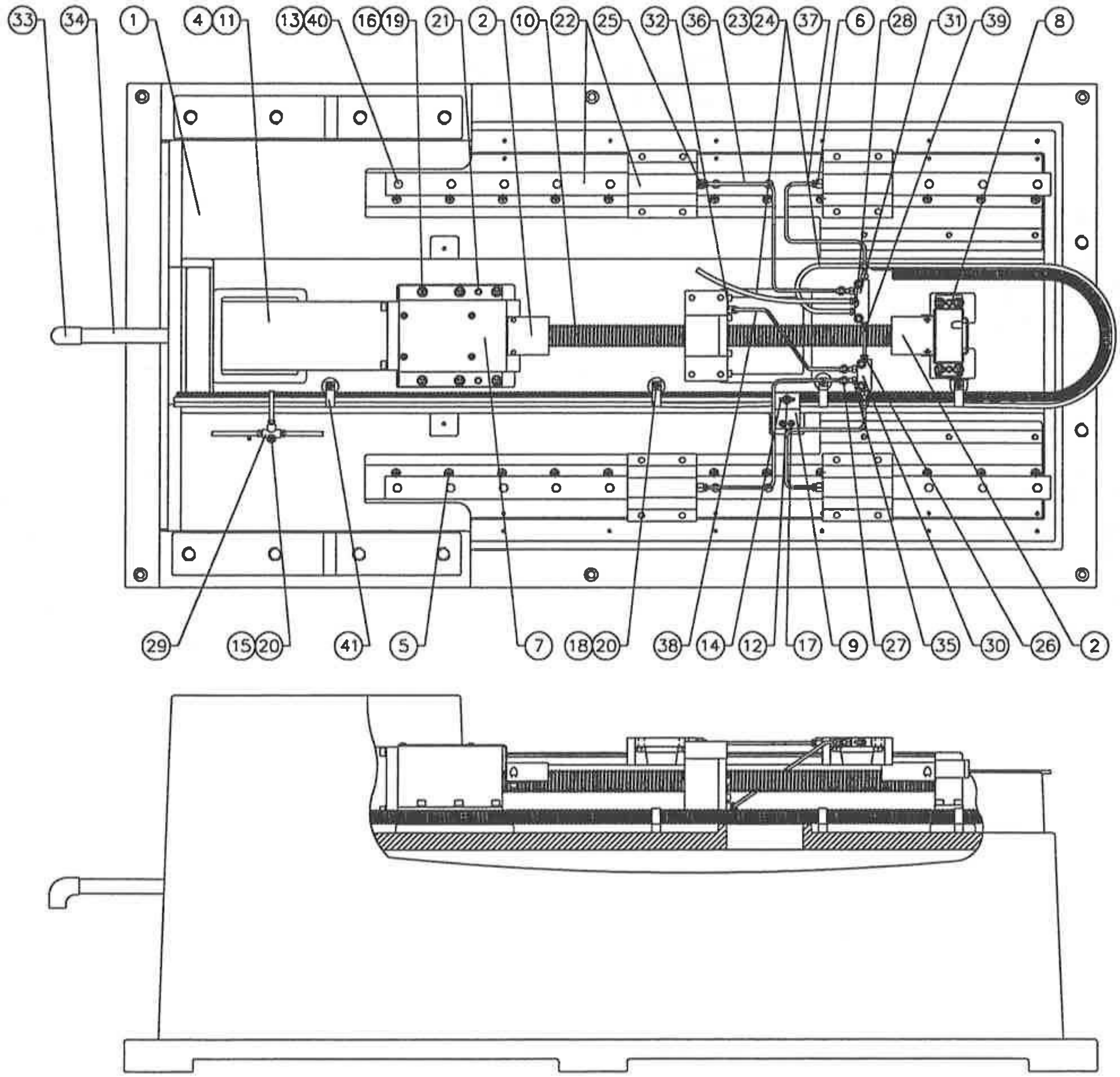
ELECTRICAL SYMBOLS

6/95 HAAS AUTOMATION VF/HS/HL SERIES PAGE 19

ASSEMBLY DRAWINGS

Contents

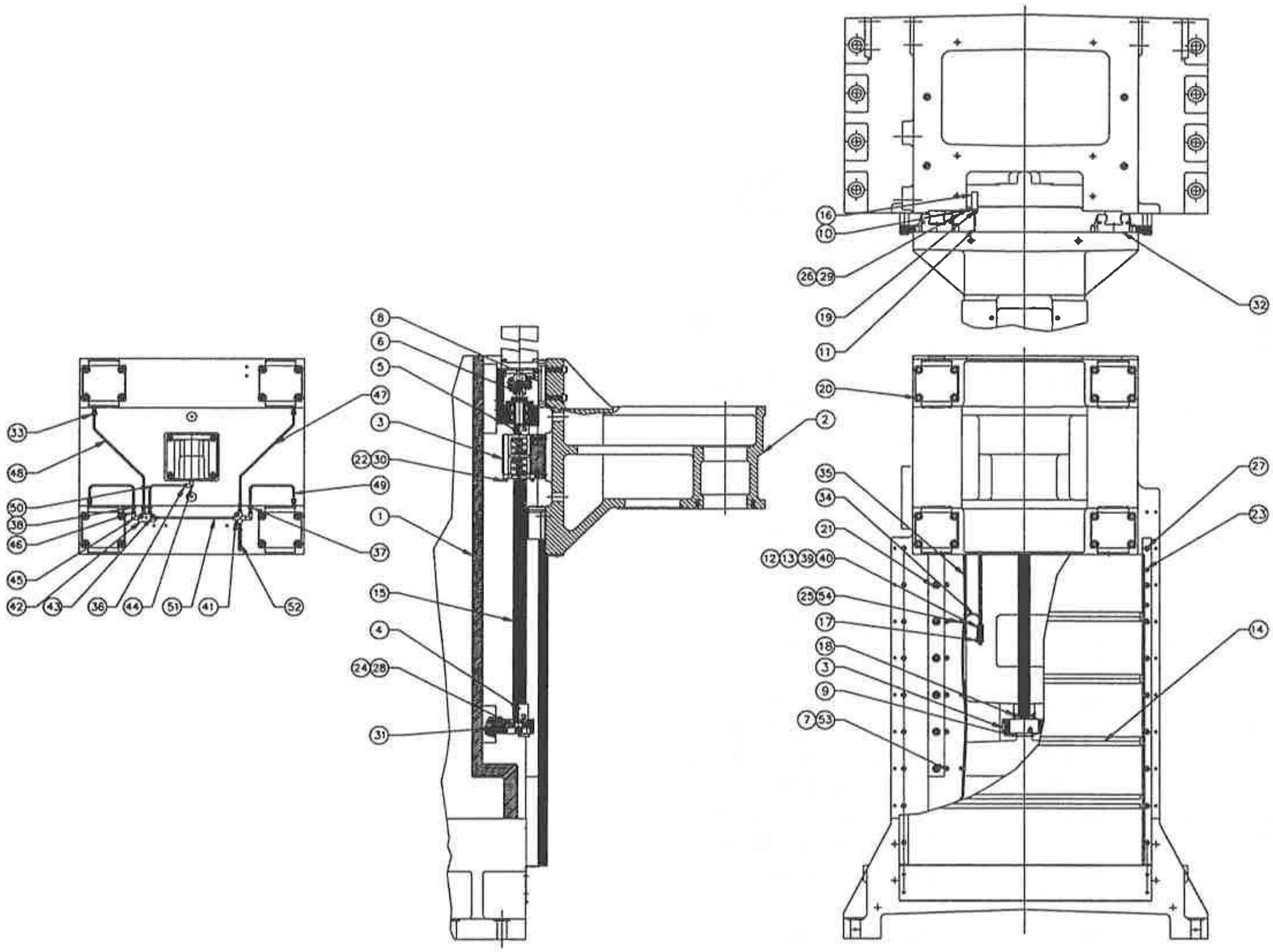
VF-1 BASE


33 34 1 4 11 13 40 16 19 21 2 10 22 25 32 36 23 24 37 6 28 31 39 8
29 15 20 41 5 7 18 20 38 14 12 17 9 27 35 30 26 2

VF-1 BASE

IT	QTY	DWG_NUM	TITLE
1	1	20-9002	BASE, MACHINED
2	1	20-9007	NUT HOUSING
3	1	20-9095	BUMPER, Y-AXIS, BRG. END
4	1	20-9096	BUMPER, Y-AXIS, MOTOR END
5	1	22-2629	KEY, 0.1875/0.1870 SQUARE
6	24	22-7458	CAM SCREW, LINEAR GUIDE
7	1	25-7042	COVER PLATE, LEAD SCR.
8	1	25-7080	BUMPER BRACKET, BRG. HSG.
9	1	25-7267	Y AXIS MOUNT BRACKET
10	1	30-1210	LEAD SCREW ASSEMBLY (Y,Z)
11	1	32-1600	Y AXIS MOTOR ASSEMBLY
12	1	32-2031	TELEMECH. SWITCH 62 IN Y-AXIS
13	4	40-1632	SHCS, 1/4-20 x 1/2"
14	2	40-16413	MSHCS, M3 x 5
15	24	40-1660	SHCS, 1/2-13 x 1 1/2"
16	4	40-1667	SHCS, 5/16-18 x 1 1/4"
17	5	40-1697	SHCS, 1/4-20 x 3/4"
18	5	40-1705	FHCS, 10-32 x 1"
19	14	40-1715	SHCS, 5/16-18 x 1 1/2"
20	2	40-1850	SHCS, 10-32 x 3/8"
21	3	40-2026	SHCS, 10-32 x 3/8"
22	14	45-1600	WASHER, SPLIT LOCK 5/16" MED.
23	5	45-1620	WASHER, SPLIT LOCK #10 MED.
24	4	48-0045	PIN, PULL 3/8" x 1 1/2"
25	1	50-9011	Y,Z AXIS LINEAR GUIDE
26	4	58-1550	1/8 NPT CONN. (BIJUR B3488)
27	1	58-2000	NYLON TUBING, 1/4" CL, 5.83 FT
28	1	58-2010	NYLON TUBING, 5/32" CL, 7.71 FT
29	12	58-2100	SLEEVE, LUBE ASSY
30	9	58-2110	SLEEVE NUTS, LUBE ASSY
31	5	58-2111	COMPRESSION NUT B-1095
32	6	58-2130	SLEEVE, COMP. NYLON TUBING
33	1	58-2763	3-WAY JUNCTION
34	1	58-3000	MANIFOLD, 4 WAY
35	1	58-3005	FITTING MANIFOLD, 5 WAY
36	1	58-3030	M6-1 TO 5/16-24 ELBOW
37	1	58-3054	90 DEG. 1/2 NPT ELBOW
38	1	58-3505	NIPPLE, 1/2-14 NPT x 10"
39	5	58-4000	FLOWMETER, FJB-000
40	2	58-9105	COPPER TUBING - YA
41	2	58-9106	COPPER TUBING - YC
42	1	58-9107	COPPER TUBING - YE
43	1	58-9108	COPPER TUBING - YF
44	24	59-6650	PLUG, GUDE RAIL, THK C-12
45	3	63-1031	CABLE CLAMP, 1/4"
46	3	63-1032	CABLE CLAMP, KEYSTONE #8110

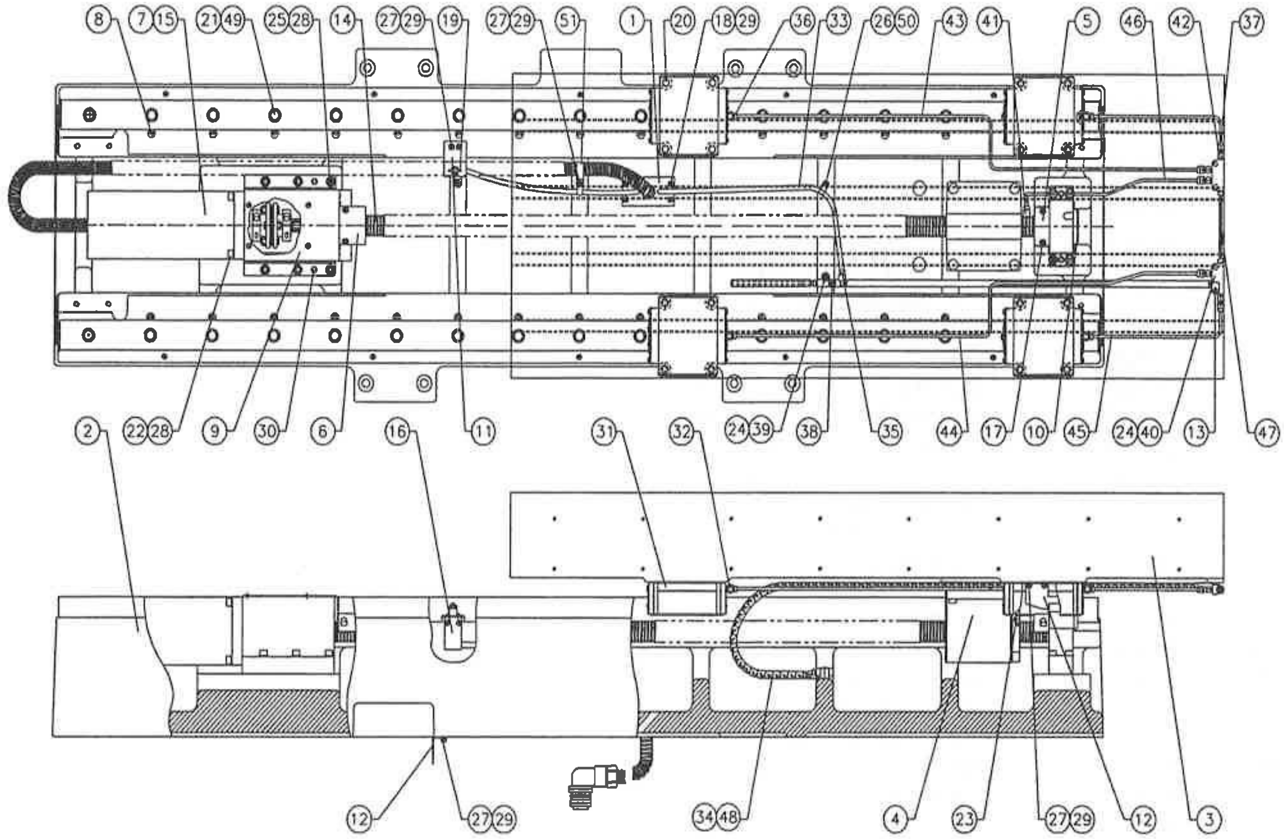
VF-1 COLUMN


16
10
26 29
19
11
32
8
6
5
3
22 30
1
15
4
24 28
31
2
20
35
34
21
12 13 39 40
25 54
17
18
3
9
7 53
27
23
14
33
47
48
50
38
46
49
37
45
42
43
36
44
51
41
52

VF-1 COLUMN

IT	QTY	DWG_NUM	TITLE
1	1	20-9001A	COLUMN, MACHINED-FROM CASTING 14-9001
2	1	20-9005A	SPINDLE HEAD, MACHINED
3	1	20-9007	NUT HOUSING, VF-3
4	1	20-9057	BUMPER, BL. SCR - X - BRG. END
5	1	20-9058	BUMPER, BL. SCR. -X- MTR. END
6	1	22-2629	KEY, 0.1875/0.1870 SQUARE
7	24	22-7458	CAM SCREW, LINEAR GUIDE
8	1	25-7042	COVER PLATE, LEAD SCREW
9	1	25-7080	BUMPER BRACKET BRG. HSG.
10	1	25-7267	Y AXIS MOUNT BRACKET
11	1	25-7459	TRIP BRACKET, TABLE
12	1	25-7485	BRACKET, OL LINE CARRIER (L)
13	1	25-7486	BRACKET, OIL LINE CARRIER (R)
14	1	25-9040	WAY COVER, Z AXIS VF-3
15	1	30-1210	LEAD SCREW ASSM.VF-3 (Y,Z)
16	1	32-2041	TELEMECH 90 IN Z-AXIS VF-3
17	1	40-16204	SHCS, 10-32 X 1 5/8"
18	4	40-1632	SHCS, 1/4-20 x 1/2"
19	2	40-16413	MSHCS, M3 x 5
20	16	40-1655	MSHCS, M12 x 65
21	24	40-1660	SHCS, 1/2-13 x 1 1/2"
22	5	40-1697	SHCS, 1/4-20 x 3/4"
23	4	40-1705	FHCS, 10-32 x 1"
24	14	40-1715	SHCS, 5/16-18 x 1 1/2"
25	8	40-1750	BHCS, 10-32 x 3/8"
26	2	40-1850	SHCS, 10-32 x 3/8"
27	18	40-2021	FHCS, 1/4-20 x 3"
28	18	45-1600	WASHER, SPLIT LOCK 5/16" MED.
29	2	45-1620	WASHER, SPLIT #10 MED
30	4	45-1800	WASHER, SPLIT LOCK 1/4" MED.
31	4	48-0045	PIN, PULL 3/8 x 1 1/2"
32	1	50-9011	Y,Z AXIS LINEAR GUIDE VF-3
33	4	58-1550	1/8 NPT CONN.(BIJUR B3488)
34	6.42FT	58-2000	NYLON TUBING, 1/4 INCH CL
35	8.5 FT	58-2010	NYLON TUBING, 5/32 INCH C
36	14	58-2100	SLEEVE, LUBE ASSM.
37	11	58-2110	SLEEVE NUTS, LUBE ASSM.
38	5	58-2111	COMPRESSION NUT B-1095
39	2	58-2130	SLEEVE, COMP.NYLON TUBING
40	1	58-2760	FITTING MANIFOLD, 2 WAY
41	1	58-3000	MANIFOLD, 4-WAY
42	1	58-3012	MANIFOLD, 6-WAY, B-3109
43	2	58-3015	CLOSURE PLUG, B-3784
44	1	58-3030	M6-1 TO 5/16-24 ELBOW
45	1	58-3045	P/N LE90585 ELBOW
46	5	58-4000	FLOWMETER, FJB-000
47	1	58-9109	COPPER TUBING - ZA VF-3
48	1	58-9110	COPPER TUBING - ZB VF-3
49	2	58-9111	COPPER TUBING - ZC VF-3
50	1	58-9112	COPPER TUBING - ZE VF-3
51	1	58-9113	COPPER TUBING - ZF VF-3
52	1.46FT	59-6150	PLASTIC CARRIER 0130.06
53	24	59-6650	PLUG, GUIDE RAIL, THK C-12
54	8	63-1032	CABLE CLAMP, KEYSTONE #8110

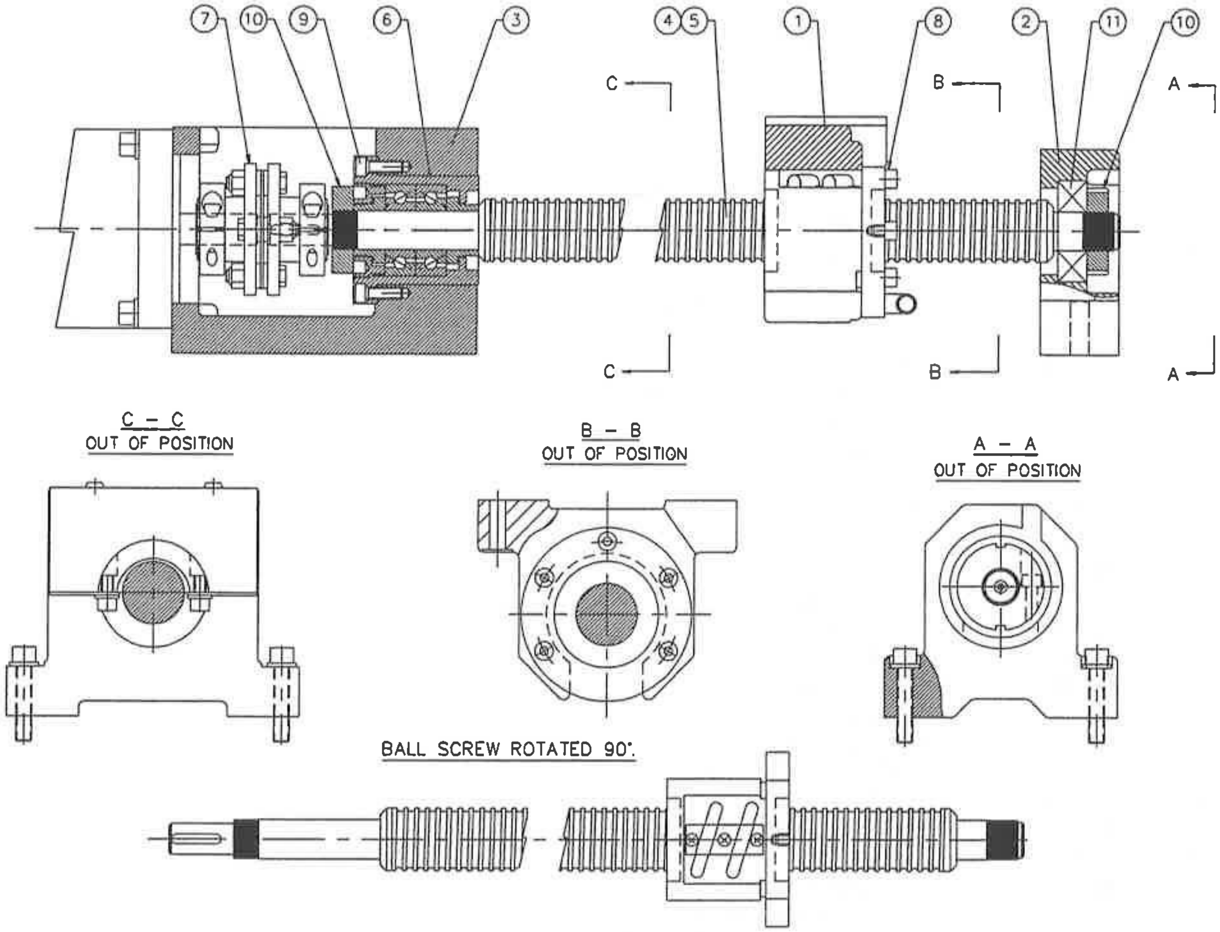
VF-1 SADDLE


8
7 15
21 49
25 28
14
27 29
19
27 29
51
1
20
18 29
36
33
26 50
43
41
5
46
42
37
2
22 28
9
30
6
16
11
31
32
24 39
38
35
44
17
10
45
24 40
13
47
12
27 29
34 48
4
23
27 29
12
3

VF-1 SADDLE

IT	QTY	DWG_NUM	TITLE
1	2	20-7456	COND. STRAIN RELIEF, SADDLE
2	1	20-9003	SADDLE, MACHINED
3	1	20-9004	TABLE, MACHINED
4	1	20-9007	NUT HOUSING
5	1	20-9057	BUMPER, BL. SCR. -X - BRG. END
6	1	20-9058	BUMPER, BL. SCR. -X - MTR. END
7	1	22-2629	KEY, 0.1875/0.1870 SQUARE
8	30	22-7458	CAM SCREW, LINEAR GUIDE
9	1	25-7042	COVER PLATE, LEAD SCREW
10	1	25-7080	BUMPER BRACKET, BRG. HSG.
11	1	25-7267	Y AXIS MOUNT BRACKET
12	2	25-7459	TRIP BRACKET
13	2	25-7485	BRACKET, OIL LINE CARRIER
14	1	30-1200	LEAD SCREW ASSY, VF-3 (X)
15	1	32-1401	X AXIS MOTOR ASSY (VF-3)
16	1	32-2050	TELEMECH. 120 IN. X-AXIS 2-3
17	4	40-1632	SHCS, 1/4-20 x 1/2"
18	4	40-1640	SHCS, 10-32 x 1/2"
19	2	40-16413	MSHCS, M3 x 5
20	16	40-1656	MSHCS, M12 x 130
21	34	40-1660	SHCS, 1/2-13 x 1 1/2"
22	4	40-1667	SHCS, 5/16-18 x 1 1/4"
23	5	40-1697	SHCS, 1/4-20 x 3/4"
24	5	40-1705	FHCS, 10-32 x 1"
25	14	40-1715	SHCS, 5/16-18 x 1 1/2"
26	1	40-1750	BHCS, 10-32 x 3/8"
27	8	40-1850	SHCS, 10-32 x 3/8"
28	18	45-1600	5/16" SPLIT LOCK WASHER
29	12	45-1620	WASHER, SPLIT LOCK #10 MED.
30	4	48-0045	PIN, PULL 3/8 x 1 1/2"
31	1	50-9010	X-AXIS LINEAR GUIDE, VF-3
32	4	58-1550	1/8 NPT CONN. (BIJUR B3488)
33	1	58-2000	NYLON TUBING, 1/4" CL., 1.54FT
34	1	58-2010	NYLON TUBING, 5/32", 3.15FT
35	14	58-2100	SLEEVE, LUBE ASSY
36	11	58-2110	SLEEVE NUTS, LUBE ASSY
37	5	58-2111	COMPRESSION NUT B-1095
38	2	58-2130	SLEEVE, COMP. NYLON TUBING
39	1	58-2760	FITTING MANIFOLD, 2 WAY
40	2	58-3000	MANIFOLD, 4 WAY
41	1	58-3030	M6-1 TO 5/16-24 ELBOW
42	5	58-4000	FLOWMETER, FJB-000
43	1	58-9100	COPPER TUBING - XA, VF-3
44	1	58-9101	COPPER TUBING - XB, VF-3
45	2	58-9102	COPPER TUBING - XD, VF-3
46	1	58-9103	COPPER TUBING - XE, VF-3
47	1	58-9104	COPPER TUBING - XF, VF-3
48	1	59-6150	PLASTIC CARRIER 0130.06, 1.46FT
49	34	59-6650	PLUG, GUIDE RAIL, THK C-12
50	1	63-1031	CABLE CLAMP 1/4"
51	2	63-1032	CABLE CLAMP, KEYSTONE #8110

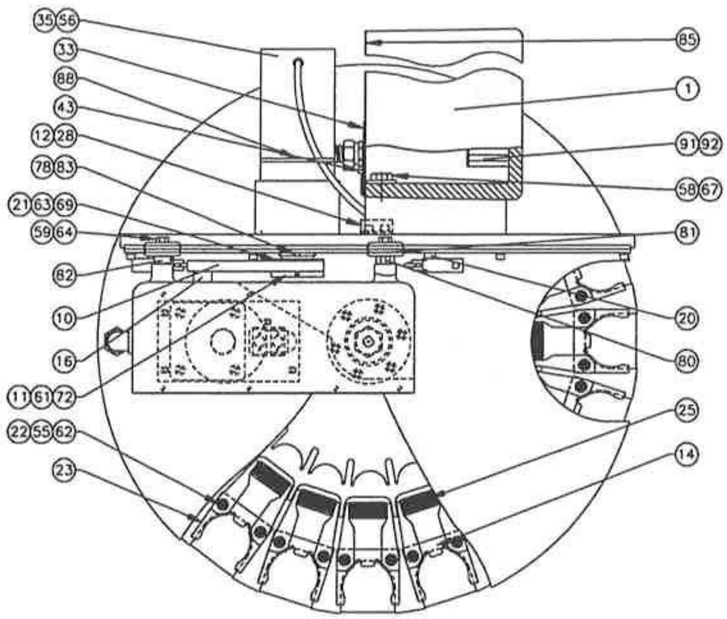
VF-1 LEADSCREW

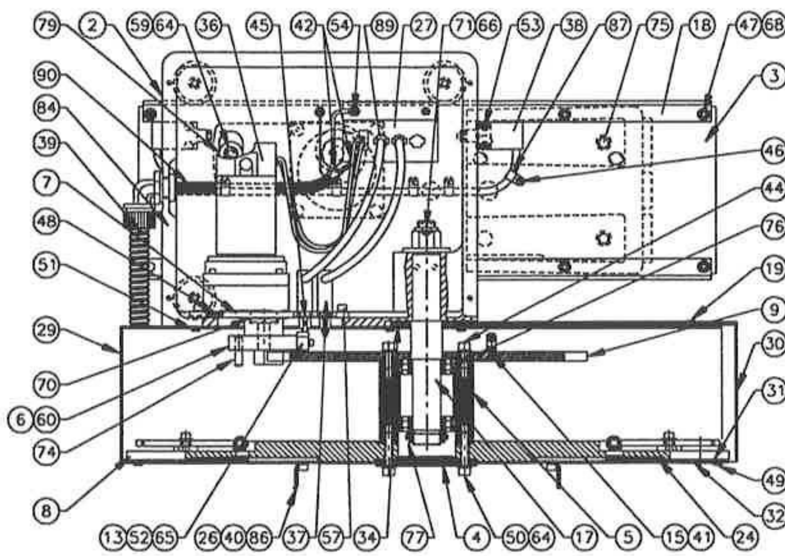


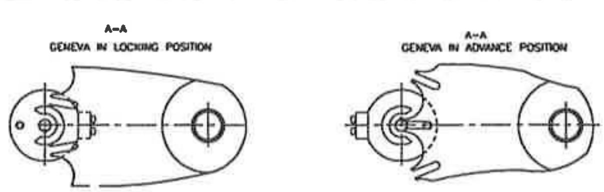
VF-1 LEADSCREW

IT	QTY	DWG_NUM	TITLE
1	1	20-7008F	FOR REFERENCE ONLY (SEE 30-1000B)
2	1	20-7009	BEARING HOUSING
3	1	20-7010	MOTOR MOUNTING
4	1	22-2629	KEY, 0.1875/0.1870 SQUARE
5	1	24-7146	LEAD SCREW, MODIFIED
6	1	24-7478	BALL SCREW SPT. BRG. ASSY
7	1	30-1220P	COUPLING ASSEMBLY, SERVO DRIVE
8	5	40-1610	SHCS, 1/4-20 x 1"
9	6	40-1697	SHCS, 1/4-20 x 3/4"
10	2	51-2012	BEARING LOCKNUT
11	1	51-2025	BEARING, FAFNIR RADIAL #304PP

VF-1 TOOL CHANGER VF-1 BASE

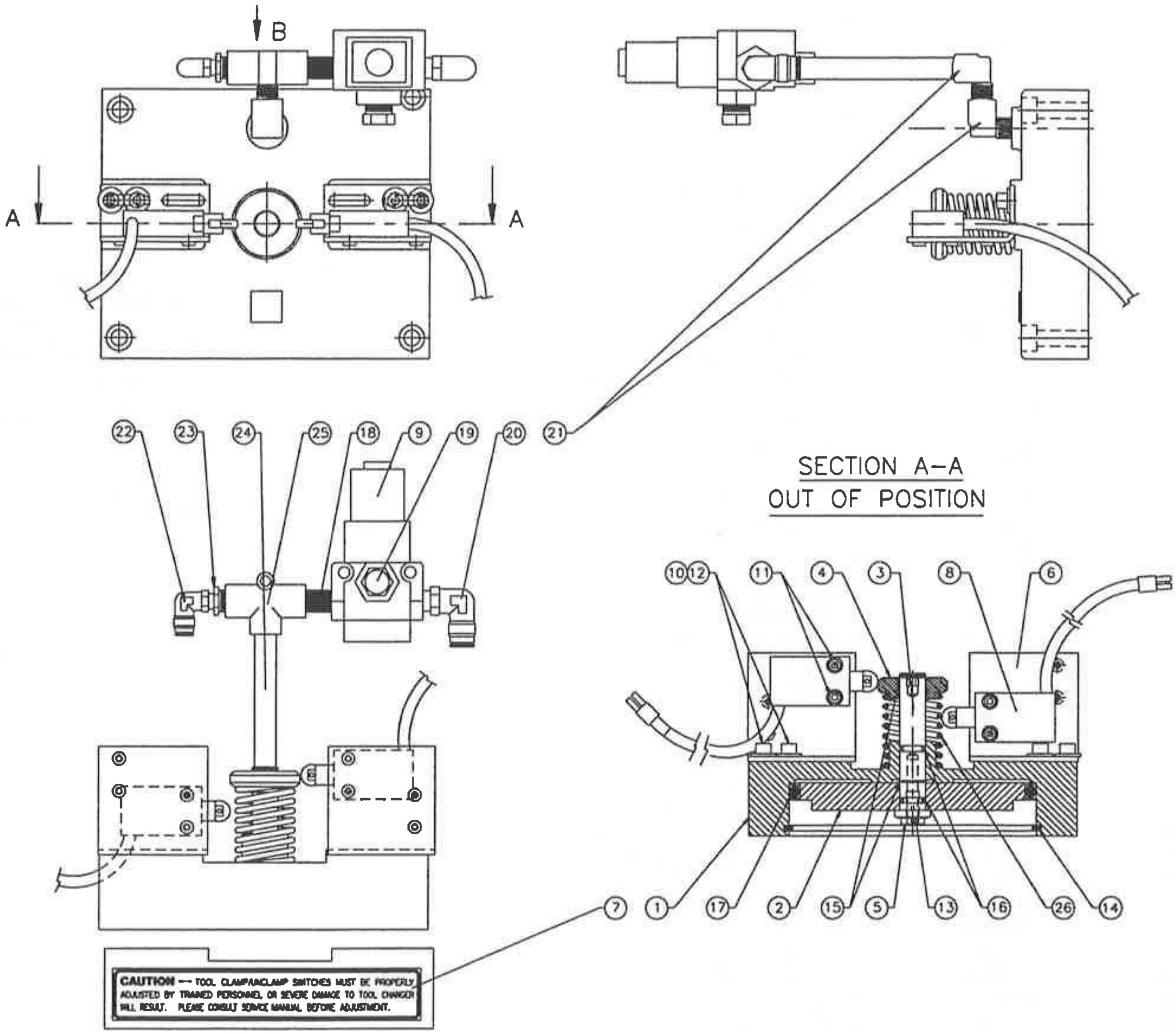






ITEM	QTY	PT#	DESCRIPTION
1	1	20-7029B	TOOL HOLDING ARM, MACH.
2	1	20-7030E	TOOL CARRIAGE
3	1	20-7033F	HOLDING PLATE
4	1	20-7036	CAP, TOOL CHANGER
5	1	20-7038	BEARING HOUSING, TOOL CH.
6	1	20-7039T	GENEVA DRIVE
7	20	20-7067F	KEY, EXTRACTOR
8	1	20-7236	MOTOR MOUNTING PLATE
9	1	20-7255A	TOOL #1 STANDOFF
10	1	20-7352A	20 POCKET CAROUSEL
11	1	20-7353	20 POCKET GENEVA STAR
12	1	20-7475	ARM, SLIP CLUTCH
13	1	20-7476	HUB, SLIP CLUTCH
14	1	22-2065	LOCATING PIN
15	1	20-7026A	CAM, GENEVA DRIVER
16	1	22-7034	SPACER, CAM FOLLOWER
17	1	22-7035G	VERTICAL AXLE
18	2	22-7106	'V' TRACK, TOOL CHANGER
19	3	22-7163	RIDER - TRAP DOOR
20	2	22-7263	SWITCH MOUNTING BLOCK
21	1	22-7477	PRESSURE PLATE
22	40	22-9256	BUSHING, EXTRACTOR
23	40	22-9574A	CT-EXTRACTOR
24	20	24-2010	COMPRESSION SPRING
25	20	24-9257	EXTRACTOR SPRING
26	20	25-7143	NUMBER BRACKET, TOOL CH.
27	1	25-7162	CONNECTOR BRACKET
28	1	25-7168	DOOR OPENER BRACKET
29	1	25-7237C	TOOL CHANGER COVER
30	1	25-7238C	TRAP DOOR, TOOL CHANGER
31	20	25-7249	SLIDING PANEL
32	20	25-7250B	SLIDING PANEL COVER
33	1	25-9253	CONDUIT MOUNTING PLATE
34	2	26-7239	SPACER RING
35	1	32-1800	SHUTTLE MOTOR ASSM.
36	1	32-1900A	TURRET MOTOR ASSM.
37	2	32-2000	TELEMECHANIQUE 8 INCHES
38	2	32-2010	TELEMECHANIQUE 24 INCHES CABLE ASSM.
39	1	32-7011	TOOL CHANGER CONDUIT ASSM.
40	1	32-7610A	CONDUIT ASSM., TOOL CARRIAGE
41	1	40-16091	BHCS, 10-32 x 1"
42	40	40-16095	SHCS, 10-32 x 1/4"
43	3	40-1632	SHCS, 1/4-20 x 1/2"
44	6	40-16385	SHCS, 5/16-18 x 3/4"
45	4	40-16413	MSHCS, M3 x 5
46	6	40-1669	BHCS, 8-32 x 3/8"
47	10	40-1697	SHCS, 1/4-20 x 3/4"
48	4	40-1702	FHCS, 10-32 x 5/8"
49	40	40-1704	FHCS, 10-32 x 1/4"
50	6	40-1715	SHCS, 5/16-18 x 1 1/2"
51	6	40-1750	BHCS, 10-32 x 3/8"
52	2	40-1800	SHCS, 8-32 x 3/4"
53	4	40-1803	SHCS, 8-32 x 1 1/4"
54	12	40-1850	SHCS, 10-32 x 3/8"
55	40	40-1860	SHCS, 1/4-20 x 7/8"
56	4	40-1970	FHCS, 1/4-28 x 1"
57	4	40-2000	SHCS, 1/4-20 x 5/8"
58	4	43-1602	HHB, 1/2-13 x 3"
59	5	43-7000	HHB, 5/16-18 x 1 3/4"
60	1	44-1622	SSS, K CUP PT 1/4-20 x 1/4"
61	1	44-1710	SSS, CUP PT 1/4-20 x 3/8"
62	40	45-0045	WASHER, BLK HRD 1/4 x 1/8"THK
63	1	45-0050	WASHER, 5702-313-120
64	17	45-1600	WASHER, SPLIT LOCK 5/16 MED.
65	2	45-1603	WASHER, SPLIT LOCK #8 MED.
66	1	45-1725	WASHER, FLAT CUT 3/4"
67	4	45-1740	WASHER, BLACK HARD 1/2"
68	10	45-1800	WASHER, SPLIT LOCK 1/4" MED.
69	2	45-2020	WASHER, NYLON
70	1	45-2021	WASHER, DELRIN 2"
71	1	46-1705	LOCK NUT, ELASTIC, 3/4-10
72	1	48-0005	PIN, DOWEL 3/16 x 3/8"
73	2	48-0020	PIN, DOWEL 1/4 x 1"
74	1	48-1661	PIN, DOWEL 5/16 x 1 1/4"
75	2	48-1750	DOWEL PIN, 1/2 x1 1/2"
76	2	51-0010	BEARING, DEEP GRV #6206-2NSL
77	1	51-0012	BRNG LOCKNUT, BH-06
78	1	51-6000	BRNG LOCKNUT, NT-05
79	1	54-0010	CAM FOLLOWER, TOOL CHANGER
80	2	54-0020	BUSHING, GUIDE WHEEL
81	4	54-0030	GUIDE WHEEL
82	2	54-0040	STANDARD BUSHING, GUIDE WHEEL
83	2	55-0010	SPRING WASHER, B2500-080
84	1	57-7378	GASKET, TOOL CARRIAGE
85	1	57-7379	GASKET, TOOL HOLDING ARM
86	1	59-8000	NUMBER SET, 1-20
87	6	63-1031	CABLE CLAMP, 1/4"
88	1	70-0050	PLT4S-M CABLE TIES, BLACK
89	1	75-15721	LARGE MOLEX HOUSING MALE
90	1	78-1996	SPLIT FLEX TUBING 1/2" I.D.
91	1	79-1000	WIRE CHANNEL, 1" x 2", 1.75 FT.
92	1	79-1001	COVER, 1" WIRE CHANNEL, 1.75 FT.

VF-1 TOOL RELEASE PISTON

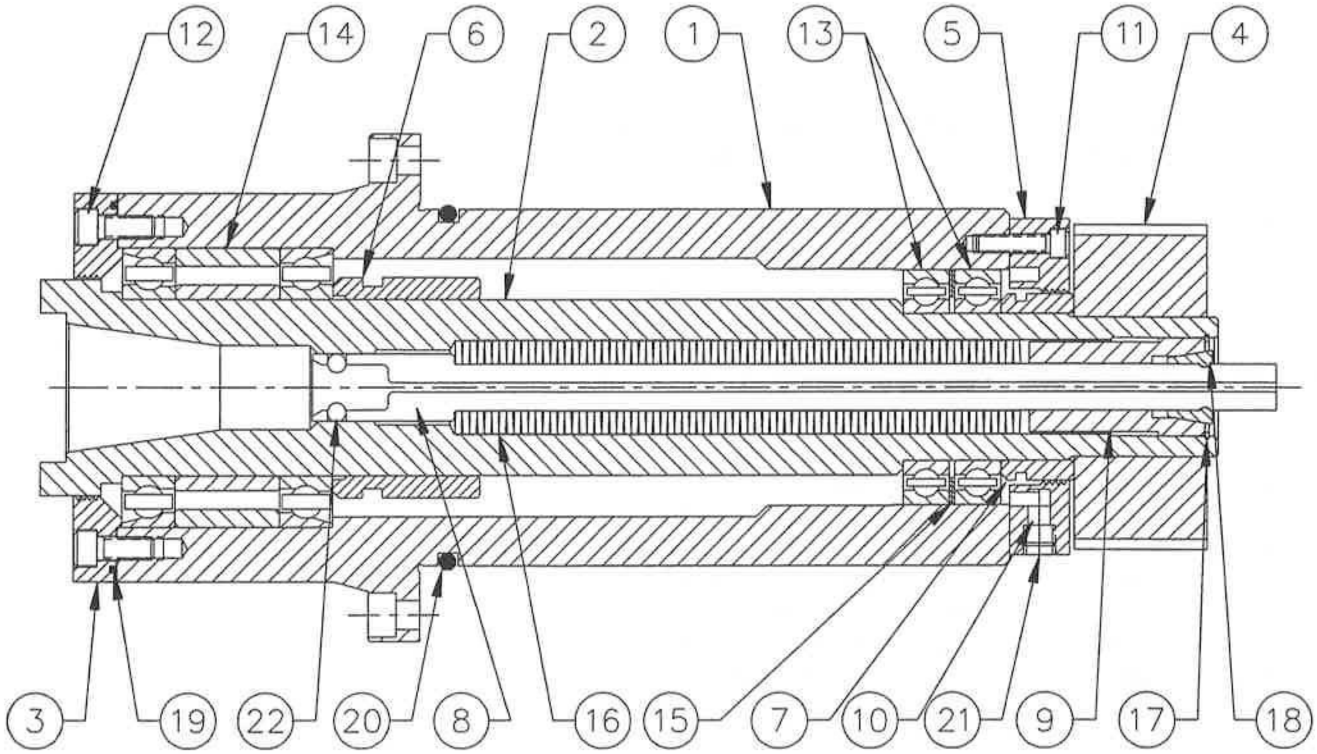


VIEW B
OUT OF POSITION

VF-1 TOOL RELEASE PISTON

IT	QTY	DWG_NUM	TITLE
1	1	20-7007	CYLINDER HOUSING, TOOL RELEASE MECHANISM
2	1	20-7043	PISTON, TOOL RELEASE
3	1	20-7044C	SHAFT, TOOL RELEASE
4	1	22-7045	SPRING RETAINER, TOOL RELEASE
5	1	22-7246	TOOL RELEASE BOLT
6	2	25-7050B	SWITCH MOUNTING BRACKET
7	1	29-7397	LABEL, TOOL RELEASE PISTON
8	2	32-2010	TELEMECH. 24 IN CABLE ASSY
9	1	32-5620	TRIP SOLENOID VALVE ASSY
10	4	40-1632	SHCS, 1/4-20 x 1/2"
11	4	40-1800	SHCS, 8-32 x 3/4"
12	4	45-0040	1/4" HARD WASHER
13	4	45-2000	1/4 SHIM WASHER, 0.010 THK.
14	1	56-0030	SNAP RING, TRU ARC #N-5000-600
15	2	56-0040	SNAP RING, TRU ARC #N-5100
16	2	57-0040	O'RING, #2-111, SHAFT
17	1	57-0090	O'RING, #2-433, TOOL RELEASE
18	1	58-2165	FITTING CLOSE NIPPLE 1/4
19	1	58-2265	AIR MUFFLER, 3/8 FLAT
20	2	58-3050	ELBOW, 1/4 NYLON TUBING
21	2	58-3618	1/4 STREET ELBOW, 90 DEG.
22	1	58-3670	1/4 NPT M - 1/8 REDUCER
23	1	58-3685	MALE 1/4 NPT - 3/8 TUBE-SWVL
24	1	58-3727A	1/4 NPT x 4" NIPPLE
25	1	59-2230A	SHUTTLE VALVE, NVR 1220NO2
26	1	59-2760	COMP. SPRING/LARGE WIRE

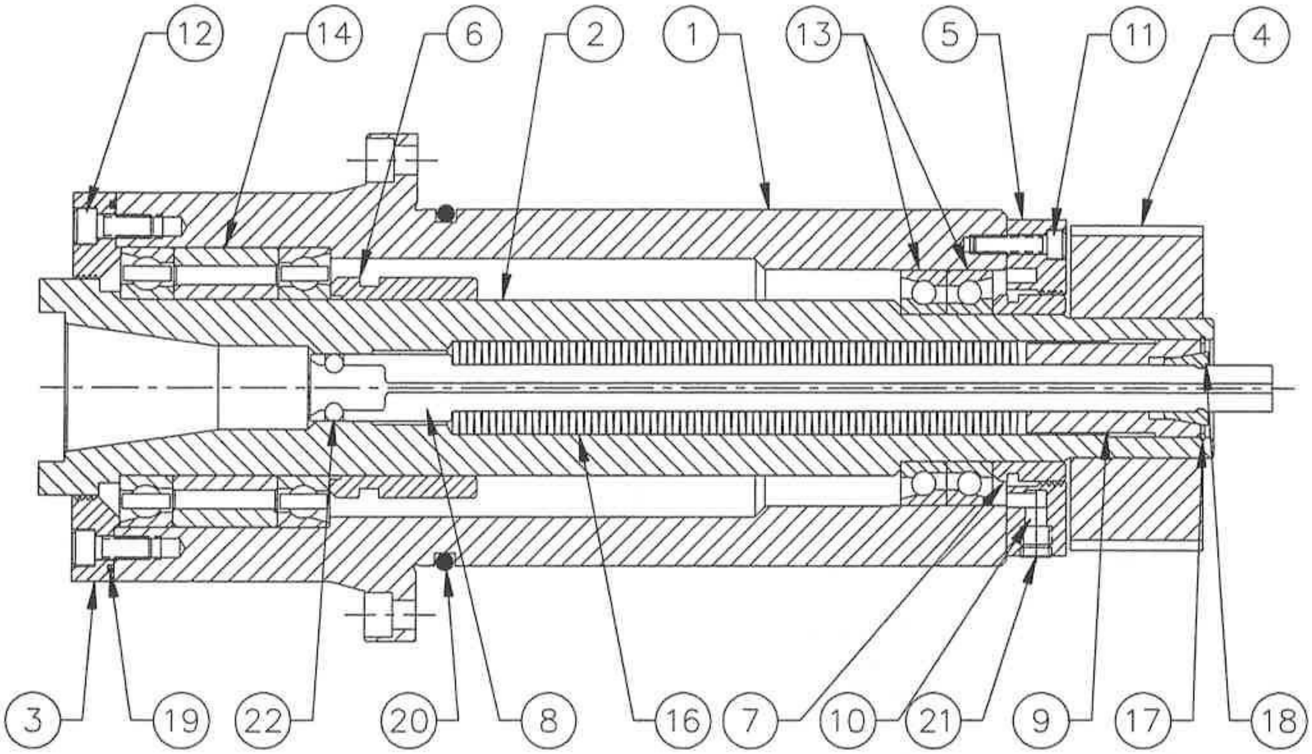
7.5K SPINDLE



7.5K SPINDLE

IT	QTY	DWG_NUM	TITLE
1	1	20-7016B	SPINDLE HOUSING
2	1	20-7018L	SPINDLE SHAFT
3	1	20-7022B	SPINDLE LOCK
4	1	20-7373	1 7/8 DIA PULLEY
5	1	20-7442B	OIL INJECT. COVER
6	1	20-7530	LOCK, 60mm BEARING
7	1	20-7531	LOCK, 50mm BEARING
8	1	22-7024J	DRAW BAR
9	1	22-7535	SPRING RETAINER
10	1	24-4200	BRONZE FILTER ELEMENT
11	4	40-1610	SHCS, 1/4-20 x 1"
12	6	40-16385	SHCS, 5/16-18 x 3/4"
13	2	51-0021	BEARING, 6010
14	1	51-1012A	112 ANGULAR CONTACT DUPLEX
15	1	55-0020	WAVE WASHER 3118
16	77	55-0030	SPRING WASHER, B1250-089
17	1	56-0075	SNAP RING, 5000-131
18	2	56-2985	VALVE KEEPER, LOCK CLIP
19	1	57-2984	O'RING, 2-158 VITON
20	1	57-2990	O-RING, 2-348 BUNA
21	1	58-1627	1/8-27 PIPE PLUG
22	4	59-2058	1/4" STEEL BALLS

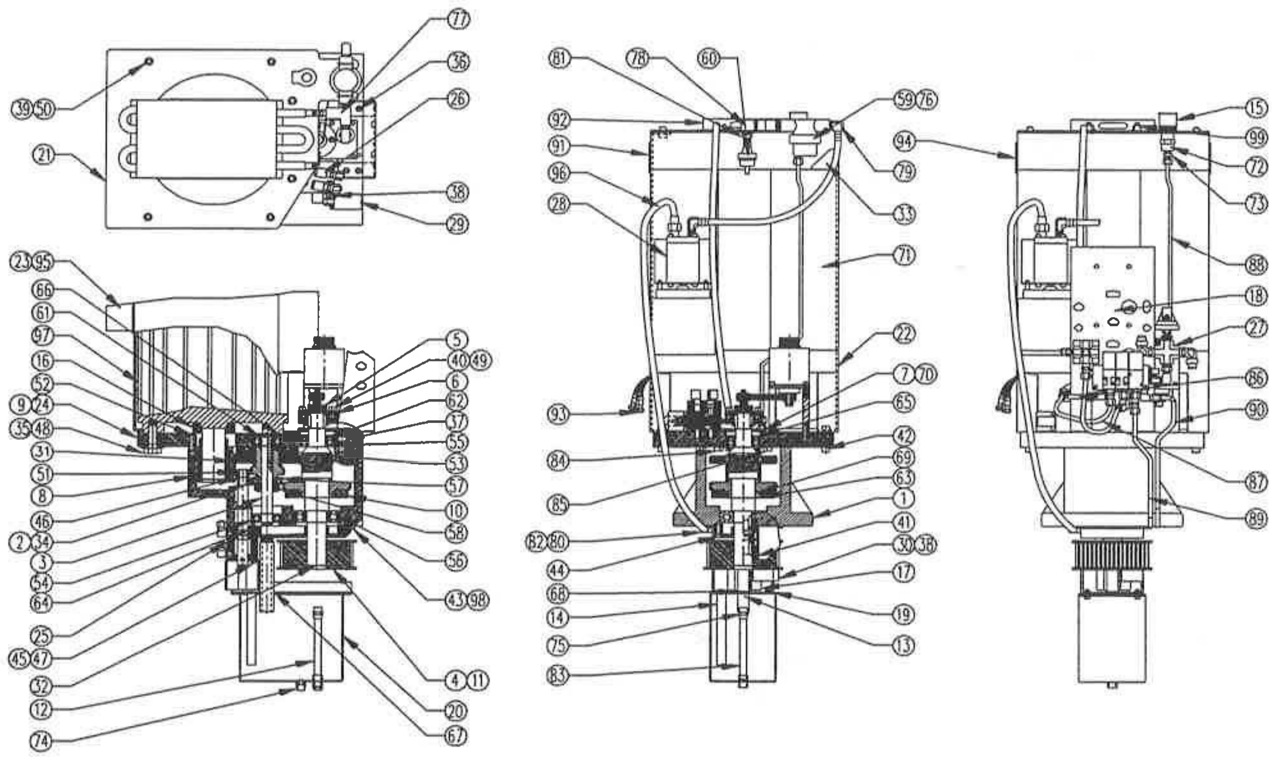
10K SPINDLE



10K SPINDLE

IT	QTY	DWG_NUM	TITLE
1	1	20-7016B	SPINDLE HOUSING
2	1	20-7018L	SPINDLE SHAFT
3	1	20-7022B	SPINDLE LOCK
4	1	20-7373	1 7/8 DIA PULLEY
5	1	20-7442B	OIL INJECT. COVER
6	1	20-7530	LOCK, 60mm BEARING
7	1	20-7532	LOCK, 50 mm ANGULAR CONT DUPLEX
8	1	22-7024J	DRAW BAR
9	1	22-7535	SPRING RETAINER
10	1	24-4200	BRONZE FILTER ELEMENT
11	4	40-1610	SHCS, 1/4-20 x 1"
12	6	40-16385	SHCS, 5/16-18 x 3/4"
13	1	51-1002	ANGULAR CONTACT DUPLEX, 50 mm
14	1	51-1012A	112 ANGULAR CONTACT DUPLEX
15	77	55-0030	SPRING WASHER, B1250-089
16	1	56-0075	SNAP RING, 5000-131
17	2	56-2985	VALVE KEEPER,LOCK CLIP
18	1	57-2984	O'RING, 2-158 VITON
19	1	57-2990	O-RING, 2-348 BUNA
20	1	58-1627	1/8-27 PIPE PLUG
21	4	59-2058	1/4" STEEL BALLS

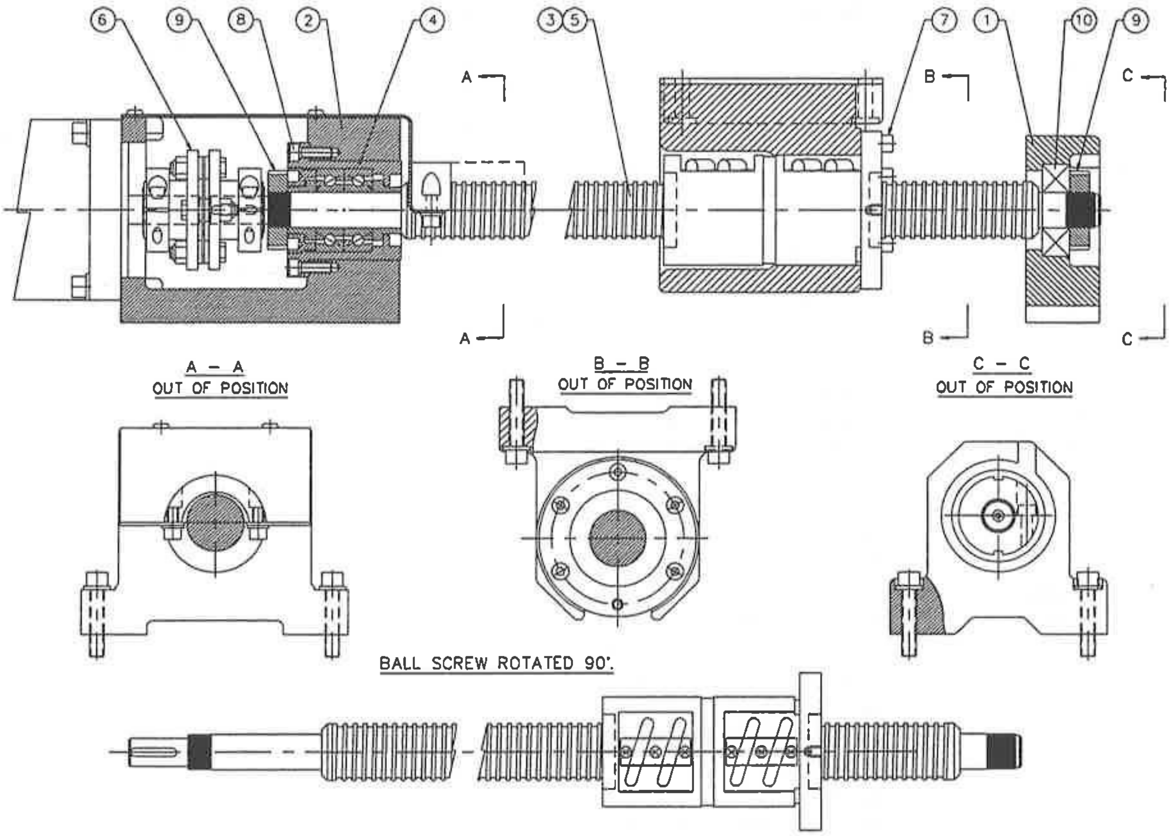
GEARBOX



IT	QTY	DWG_NUM	TITLE
1	1	20-7011B	TRANSF.HOUSING, MACHINING (14-7011 CAST)
2	1	20-7062	BEARING FORK, GEAR CLUSTER
3	1	20-7064	SHAFT, GEAR CLUSTER
4	1	20-7374	1 1/8" SPROCKET
5	1	20-7427	CLAMPING RING, SHOT PIN
6	1	20-7428	POSITIONING RING
7	1	20-7435	OILER PLATE
8	1	20-9125	SPUR GEAR, MOTOR SHAFT
9	1	20-9126	TOP PLATE, GEAR BOX
10	2	22-7081	COUNTERWEIGHT, GEAR
11	2	22-7376	SPROCKET FLANGE
12	1	20-7415	OVERFLOW DRAIN, DRY SUMP
13	1	22-7445A	DRAIN TUBE DRY SUMP
14	1	22-7446	PICK UP TUBE DRY SUMP
15	1	22-7487	OIL FILL CAP
16	1	22-9127	OIL SLINGER, VF-3
17	1	25-7264	SWITCH MOUNTING BRACKET
18	1	25-7336	SOLENOID MTG BRACKET
19	1	25-7433	SUMP BRACKET
20	1	25-7434	SUMP TANK
21	1	25-9129A	FAN BRACKET, DRY SUMP TRANSM.
22	1	25-9130A	MOTOR SHROUD
23	1	25-9175	MOTOR SHROUD COVER
24	1	29-9128	LABEL, TRANSMISSION VF-3

25	1	30-3130B	PISTON ASSY, 7500 RPM	88	1	58-9114A	TRANS FILL TUBE, VF-3
26	1	30-3140A	SHOT PIN ASSY	89	1	58-9117	HIGH GEAR TUBE
27	1	30-3150C	AIR MANIFOLD ASSY	90	1	58-9118	LOW GEAR TUBE
28	1	30-3260	OIL GEAR PUMP ASSY	91	4	58-1482	NYLON FINISH PLUG, 13/16
29	2	32-2010	TELEMECH. 24 IN CABLE ASSY.	92	1	59-2910	OIL COOLER
30	1	32-2011	TELEMECH. 40 IN CABLE ASSY	93	3	59-4005	CABLE CLAMP, 1/4 - 25/32
31	1	35-7065	TRANSFER GEAR CLUSTER	94	2	59-7222	GROMMET, 1 1/2"
32	1	35-7170	DRIVE SHAFT	95	1	59-9135A	SOUNDCOAT, MTR/GEAR SHRD
33	1	36-3035A	SPINDLE MOTOR FAN ASSY.	96	1	59-9179	SOUNDCOAT, FRNT SHRD CVR
34	2	40-1602	FHCS, 1/4-28 x 5/8"	97	1	20-9136	SPINDLE MOTOR, LINCOLN 10 HP
35	3	40-1603	HHB, 1/2-13 x 1 1/2"	98	1	63-1029	WIRE CLAMP, 3/8
36	16	40-1632	SHCS, 1/4-20 x 1/2"	99	3	76-2420	CRIMP RING, 12-10 STUD
37	12	40-16385	SHCS, 5/16-18 x 3/4"				
38	4	40-16413	MSHCS, M3 x 5				
39	4	40-1669	BHCS, 8-32 x 3/8"				
40	4	40-1697	SHCS, 1/4-20 x 3/4"				
41	4	40-1700	SHCS, 10-32 x 2"				
42	4	40-1715	SHCS, 5/16-18 x 1 1/2"				
43	5	40-1850	SHCS, 10-32 x 3/8"				
44	5	40-1950	SHCS, 10-32 x 3/4"				
45	3	41-1500	PPHS, 8-32 x 3/8"				
46	1	45-1682	WASHER, SPLIT LOCK 7/16 MED.				
47	8	45-1700	WASHER, INTERNAL LOCK #8				
48	3	45-1740	WASHER, BLACK HARD 1/2"				
49	4	45-1800	WASHER, SPLIT LOCK 1/4" MED.				
50	8	46-1617	NUT, HEX 8-32				
51	1	46-1654	NUT, HEX 7/16-20				
52	2	48-0020	PIN, DOWEL 1/4 x 1"				
53	1	48-0050	PIN, DOWEL 1/8 x 7/16"				
54	2	51-2031	RADIAL BEARING, OPEN 6303				
55	1	51-2032	RADIAL BEARING, OPEN 6205				
56	1	51-2033	RADIAL BEARING, OPEN 6306				
57	1	51-2034	RADIAL BEARING, OPEN 9105				
58	1	51-2041	BRG. LOCK NUT, BH-05				
59	1	53-1005	OIL FILTER				
60	1	53-3002	PRESSURE SWITCH PS-126				
61	2	55-0035	SPRING WASHER, #BS-204				
62	2	55-0036	SPRING WASHER, #BS-205				
63	2	56-0055	SNAP-RING, #5100-212				
64	3	56-0060	SNAP-RING, #5100-66				
65	1	56-0070	SNAP RING, #5000-187				
66	1	56-2087	SNAP RING, #5000-206				
67	1	57-0020	O'RING				
68	1	57-0040	O'RING, #2-111, SHAFT				
69	2	57-0095	O-RING, 2-327				
70	1	58-1627	1/8-27 PIPE PLUG				
71	3	58-2022	1/2 OD NATURAL TUBING				
72	3	58-2065	COUPLING, 1/4 NPT				
73	1	58-2070	1/4 NPT MALE TO 3/8 COMP.				
74	1	58-2745	MAGNETIC OIL PLUG				
75	1	58-2746	1/8 x 1/8 FEMALE COUPLER				
76	1	58-2747	MAGNET, OIL FILTER				
77	1	58-3001	1/4 SREET TEE				
78	1	58-3057	90 DEG. 1/8 UNION ELBOW				
79	2	58-3616	3/8" 90 DEGREE ELBOW 1/4 NPT				
80	1	58-3618	1/4 STREET ELBOW				
81	1	58-3670	1/4 NPT M - 1/8 F REDUCER				
82	1	58-3680	1/4 NPT M - 3/8 STRT TUBE				
83	1	58-3735	1/8 NPT x 4" PIPE NIPPLE				
84	1	58-7357	TOP PLATE TUBE - A				
85	1	58-7358	TOP PLATE TUBE - B				
86	1	58-7359B	SHOT PIN TUBE - A				
87	1	58-7360B	SHOT PIN TUBE - B (SERVICE)				

VF-3 LEAD SCREW

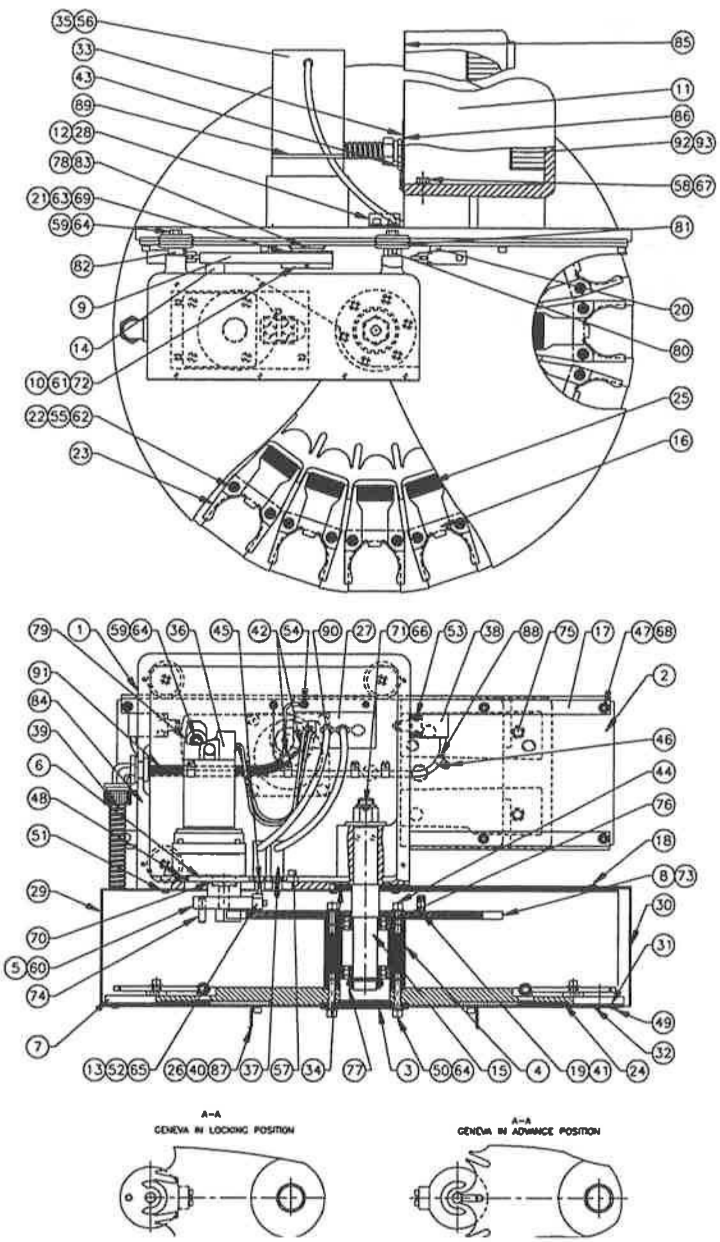


VF-3 LEAD SCREW

3

IT	QTY	DWG_NUM	TITLE
1	1	20-7009	BEARING HOUSING
2	1	20-7010	MOTOR MOUNTING
3	1	22-2629	KEY, 0.1875/0.1870 SQUARE
4	1	24-7478	BALL SCREW SPT. BRG. ASSY
5	1	24-9012	X-AXIS BALL SCREW, VF-3
6	1	30-1220P	COUPLING ASSEMBLY, SERVO DRIVE
7	5	40-1610	SHCS, 1/4-20 x 1"
8	6	40-1697	SHCS, 1/4-20 x 3/4"
9	2	51-2012	BEARING LOCKNUT
10	1	51-2025	BEARING, FAFNIR RADIAL #304PP

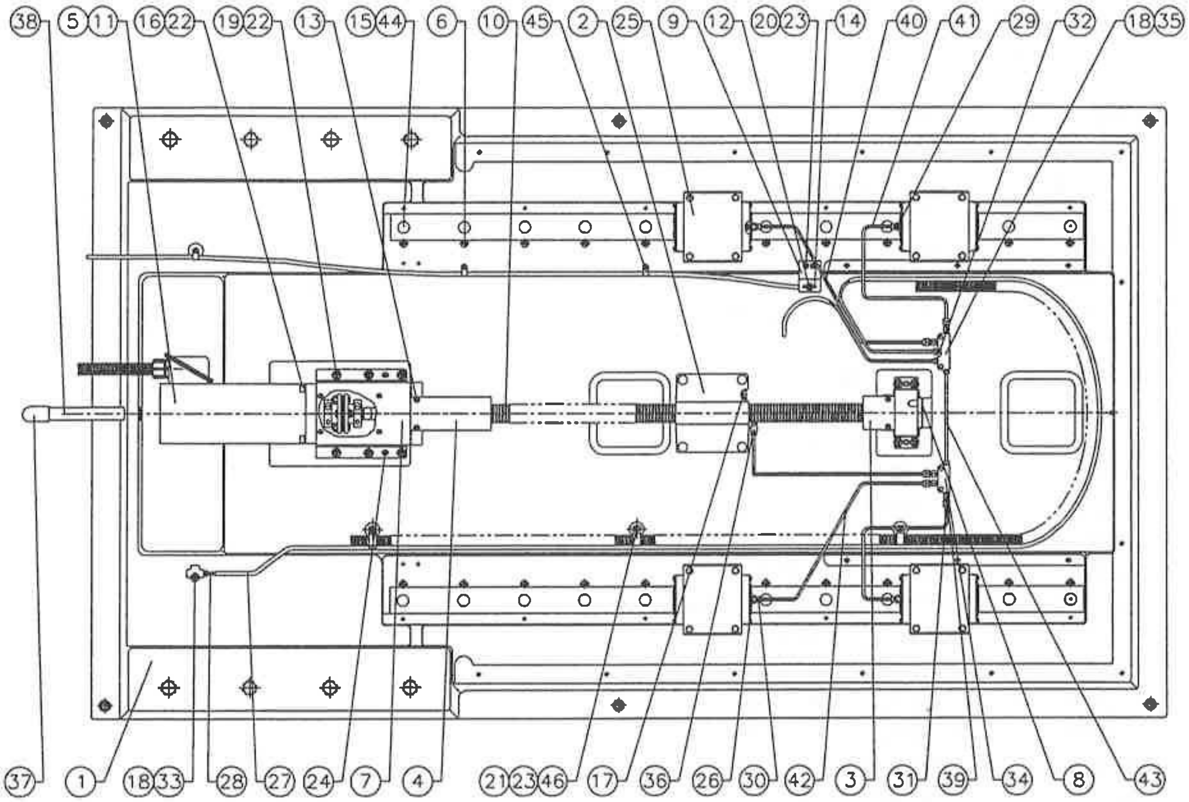
VF-3 TOOL CHANGER



VF-3 TOOL CHANGER

IT	QTY	DWG_NUM	TITLE
1	1	20-7016B	SPINDLE HOUSING
2	1	20-7018L	SPINDLE SHAFT
3	1	20-7022B	SPINDLE LOCK
4	1	20-7373	1 7/8 DIA PULLEY
5	1	20-7442B	OIL INJECT. COVER
6	1	20-7530	LOCK, 60mm BEARING
7	1	20-7531	LOCK, 50mm BEARING
8	1	22-7024J	DRAW BAR
9	1	22-7535	SPRING RETAINER
10	1	24-4200	BRONZE FILTER ELEMENT
11	4	40-1610	SHCS, 1/4-20 x 1"
12	6	40-16385	SHCS, 5/16-18 x 3/4"
13	2	51-0021	BEARING, 6010
14	1	51-1012A	112 ANGULAR CONTACT DUPLEX
15	1	55-0020	WAVE WASHER 3118
16	77	55-0030	SPRING WASHER, B1250-089
17	1	56-0075	SNAP RING, 5000-131
18	2	56-2985	VALVE KEEPER, LOCK CLIP
19	1	57-2984	O'RING, 2-158 VITON
20	1	57-2990	O-RING, 2-348 BUNA
21	1	58-1627	1/8-27 PIPE PLUG
22	4	59-2058	1/4" STEEL BALLS

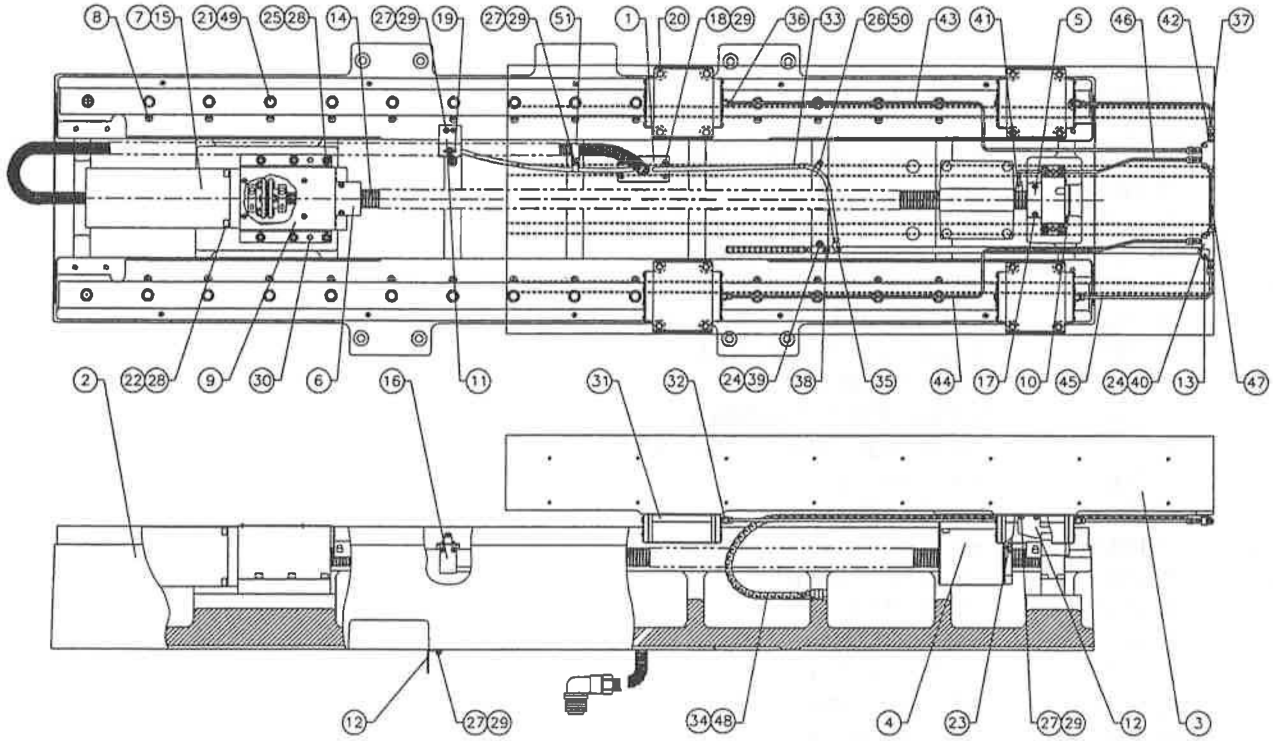
VF-3 BASE


38
5
11
16
22
19
22
13
15
44
6
10
45
2
25
9
12
20
23
14
40
41
29
32
18
35
37
1
18
33
28
27
24
7
4
21
23
46
17
36
26
30
42
3
31
39
34
8
43

VF-3 BASE

IT	QTY	DWG_NUM	TITLE
1	1	20-9002	BASE, MACHINED
2	1	20-9007	NUT HOUSING
3	1	20-9095	BUMPER, Y-AXIS, BRG. END
4	1	20-9096	BUMPER, Y-AXIS, MOTOR END
5	1	22-2629	KEY, 0.1875/0.1870 SQUARE
6	24	22-7458	CAM SCREW, LINEAR GUIDE
7	1	25-7042	COVER PLATE, LEAD SCR.
8	1	25-7080	BUMPER BRACKET, BRG. HSG.
9	1	25-7267	Y AXIS MOUNT BRACKET
10	1	30-1210	LEAD SCREW ASSEMBLY (Y,Z)
11	1	32-1600	Y AXIS MOTOR ASSEMBLY
12	1	32-2031	TELEMECH. SWITCH 62 IN Y-AXIS
13	4	40-1632	SHCS, 1/4-20 x 1/2"
14	2	40-16413	MSHCS, M3 x 5
15	24	40-1660	SHCS, 1/2-13 x 1 1/2"
16	4	40-1667	SHCS, 5/16-18 x 1 1/4"
17	5	40-1697	SHCS, 1/4-20 x 3/4"
18	5	40-1705	FHCS, 10-32 x 1"
19	14	40-1715	SHCS, 5/16-18 x 1 1/2"
20	2	40-1850	SHCS, 10-32 x 3/8"
21	3	40-2026	SHCS, 10-32 x 3/8"
22	14	45-1600	WASHER, SPLIT LOCK 5/16" MED.
23	5	45-1620	WASHER, SPLIT LOCK #10 MED.
24	4	48-0045	PIN, PULL 3/8" x 1 1/2"
25	1	50-9011	Y,Z AXIS LINEAR GUIDE
26	4	58-1550	1/8 NPT CONN. (BIJUR B3488)
27	1	58-2000	NYLON TUBING, 1/4" CL, 5.83 FT
28	1	58-2010	NYLON TUBING, 5/32" CL, 7.71 FT
29	12	58-2100	SLEEVE, LUBE ASSY
30	9	58-2110	SLEEVE NUTS, LUBE ASSY
31	5	58-2111	COMPRESSION NUT B-1095
32	6	58-2130	SLEEVE, COMP. NYLON TUBING
33	1	58-2763	3-WAY JUNCTION
34	1	58-3000	MANIFOLD, 4 WAY
35	1	58-3005	FITTING MANIFOLD, 5 WAY
36	1	58-3030	M6-1 TO 5/16-24 ELBOW
37	1	58-3054	90 DEG. 1/2 NPT ELBOW
38	1	58-3505	NIPPLE, 1/2-14 NPT x 10"
39	5	58-4000	FLOWMETER, FJB-000
40	2	58-9105	COPPER TUBING - YA
41	2	58-9106	COPPER TUBING - YC
42	1	58-9107	COPPER TUBING - YE
43	1	58-9108	COPPER TUBING - YF
44	24	59-6650	PLUG, GUDE RAIL, THK C-12
45	3	63-1031	CABLE CLAMP, 1/4"
46	3	63-1032	CABLE CLAMP, KEYSTONE #8110

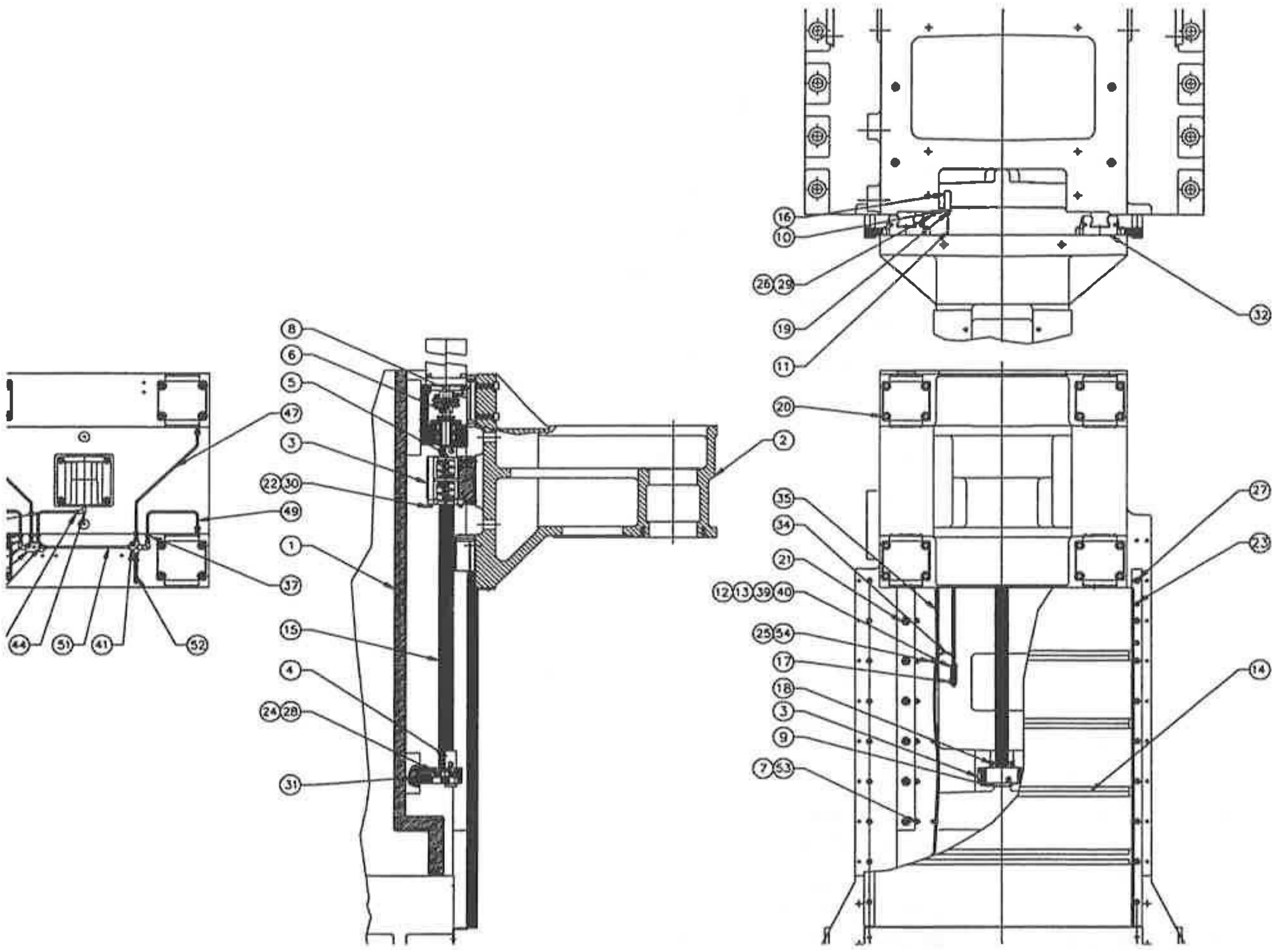
VF-3 BASE



VF-3 BASE

IT	QTY	DWG_NUM	TITLE
1	2	20-7456	COND. STRAIN RELIEF, SADDLE
2	1	20-9003	SADDLE, MACHINED
3	1	20-9004	TABLE, MACHINED
4	1	20-9007	NUT HOUSING
5	1	20-9057	BUMPER, BL. SCR. -X - BRG. END
6	1	20-9058	BUMPER, BL. SCR. -X - MTR. END
7	1	22-2629	KEY, 0.1875/0.1870 SQUARE
8	30	22-7458	CAM SCREW, LINEAR GUIDE
9	1	25-7042	COVER PLATE, LEAD SCREW
10	1	25-7080	BUMPER BRACKET, BRG. HSG.
11	1	25-7267	Y AXIS MOUNT BRACKET
12	2	25-7459	TRIP BRACKET
13	2	25-7485	BRACKET, OIL LINE CARRIER
14	1	30-1200	LEAD SCREW ASSY, VF-3 (X)
15	1	32-1401	X AXIS MOTOR ASSY (VF-3)
16	1	32-2050	TELEMECH. 120 IN. X-AXIS 2-3
17	4	40-1632	SHCS, 1/4-20 x 1/2"
18	4	40-1640	SHCS, 10-32 x 1/2"
19	2	40-16413	MSHCS, M3 x 5
20	16	40-1656	MSHCS, M12 x 130
21	34	40-1660	SHCS, 1/2-13 x 1 1/2"
22	4	40-1667	SHCS, 5/16-18 x 1 1/4"
23	5	40-1697	SHCS, 1/4-20 x 3/4"
24	5	40-1705	FHCS, 10-32 x 1"
25	14	40-1715	SHCS, 5/16-18 x 1 1/2"
26	1	40-1750	BHCS, 10-32 x 3/8"
27	8	40-1850	SHCS, 10-32 x 3/8"
28	18	45-1600	5/16" SPLIT LOCK WASHER
29	12	45-1620	WASHER, SPLIT LOCK #10 MED.
30	4	48-0045	PIN, PULL 3/8 x 1 1/2"
31	1	50-9010	X-AXIS LINEAR GUIDE, VF-3
32	4	58-1550	1/8 NPT CONN. (BIJUR B3488)
33	1	58-2000	NYLON TUBING, 1/4" CL., 1.54FT
34	1	58-2010	NYLON TUBING, 5/32", 3.15FT
35	14	58-2100	SLEEVE, LUBE ASSY
36	11	58-2110	SLEEVE NUTS, LUBE ASSY
37	5	58-2111	COMPRESSION NUT B-1095
38	2	58-2130	SLEEVE, COMP. NYLON TUBING
39	1	58-2760	FITTING MANIFOLD, 2 WAY
40	2	58-3000	MANIFOLD, 4 WAY
41	1	58-3030	M6-1 TO 5/16-24 ELBOW
42	5	58-4000	FLOWMETER, FJB-000
43	1	58-9100	COPPER TUBING - XA, VF-3
44	1	58-9101	COPPER TUBING - XB, VF-3
45	2	58-9102	COPPER TUBING - XD, VF-3
46	1	58-9103	COPPER TUBING - XE, VF-3
47	1	58-9104	COPPER TUBING - XF, VF-3
48	1	59-6150	PLASTIC CARRIER 0130.06, 1.46FT
49	34	59-6650	PLUG, GUIDE RAIL, THK C-12
50	1	63-1031	CABLE CLAMP 1/4"
51	2	63-1032	CABLE CLAMP, KEYSTONE #8110

VF-3 COLUMN



IT	QTY	DWG_NUM	TITLE
1	1	20-9001A	COLUMN, MACHINED-FROM CASTING 14-9001
2	1	20-9005A	SPINDLE HEAD, MACHINED
3	1	20-9007	NUT HOUSING, VF-3
4	1	20-9057	BUMPER, BL. SCR - X - BRG. END
5	1	20-9058	BUMPER, BL. SCR. -X- MTR. END
6	1	22-2629	KEY, 0.1875/0.1870 SQUARE
7	24	22-7458	CAM SCREW, LINEAR GUIDE
8	1	25-7042	COVER PLATE, LEAD SCREW
9	1	25-7080	BUMPER BRACKET BRG. HSG.
10	1	25-7267	Y AXIS MOUNT BRACKET
11	1	25-7459	TRIP BRACKET, TABLE
12	1	25-7485	BRACKET, OL LINE CARRIER (L)
13	1	25-7486	BRACKET, OIL LINE CARRIER (R)
14	1	25-9040	WAY COVER, Z AXIS VF-3

VF-3 COLUMN

15	1	30-1210	LEAD SCREW ASSM. VF-3 (Y,Z)
16	1	32-2041	TELEMECH 90 IN Z-AXIS VF-3
17	1	40-16204	SHCS, 10-32 X 1 5/8"
18	4	40-1632	SHCS, 1/4-20 x 1/2"
19	2	40-16413	MSHCS, M3 x 5
20	16	40-1655	MSHCS, M12 x 65
21	24	40-1660	SHCS, 1/2-13 x 1 1/2"
22	5	40-1697	SHCS, 1/4-20 x 3/4"
23	4	40-1705	FHCS, 10-32 x 1"
24	14	40-1715	SHCS, 5/16-18 x 1 1/2"
25	8	40-1750	BHCS, 10-32 x 3/8"
26	2	40-1850	SHCS, 10-32 x 3/8"
27	18	40-2021	FHCS, 1/4-20 x 3"
28	18	45-1600	WASHER, SPLIT LOCK 5/16" MED.
29	2	45-1620	WASHER, SPLIT #10 MED
30	4	45-1800	WASHER, SPLIT LOCK 1/4" MED.
31	4	48-0045	PIN, PULL 3/8 x 1 1/2"
32	1	50-9011	Y,Z AXIS LINEAR GUIDE VF-3
33	4	58-1550	1/8 NPT CONN.(BIJUR B3488)
34	6.42FT	58-2000	NYLON TUBING, 1/4 INCH CL
35	8.5 FT	58-2010	NYLON TUBING, 5/32 INCH C
36	14	58-2100	SLEEVE, LUBE ASSM.
37	11	58-2110	SLEEVE NUTS, LUBE ASSM.
38	5	58-2111	COMPRESSION NUT B-1095
39	2	58-2130	SLEEVE, COMP. NYLON TUBING
40	1	58-2760	FITTING MANIFOLD, 2 WAY
41	1	58-3000	MANIFOLD, 4-WAY
42	1	58-3012	MANIFOLD, 6-WAY, B-3109
43	2	58-3015	CLOSURE PLUG, B-3784
44	1	58-3030	M6-1 TO 5/16-24 ELBOW
45	1	58-3045	P/N LE90585 ELBOW
46	5	58-4000	FLOWMETER, FJB-000
47	1	58-9109	COPPER TUBING - ZA VF-3
48	1	58-9110	COPPER TUBING - ZB VF-3
49	2	58-9111	COPPER TUBING - ZC VF-3
50	1	58-9112	COPPER TUBING - ZE VF-3
51	1	58-9113	COPPER TUBING - ZF VF-3
52	1.46FT	59-6150	PLASTIC CARRIER 0130.06
53	24	59-6650	PLUG, GUIDE RAIL, THK C-12
54	8	63-1032	CABLE CLAMP, KEYSTONE #8110

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.

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. If you need to change parameters, contact your dealer. An explanation of each 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.

Parameter 1 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:

REV ENCODER	Used to reverse the direction of encoder data.
REV POWER	Used to reverse direction of power to motor.
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 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/3	Scale ratio is interpreted as divided by 3.

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/INCH)

The number of steps of the encoder per inch of travel. Encoder steps supply four times their line count per revolution. Thus a 2000 line encoder and a 6mm pitch screw give:

			2000 x 4 x 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 x 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 x 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/INCH 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.
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/INCH) 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/INCH)
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	Enables display of **Z** channel test plot.
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.
SPIN COOLANT	Enables spindle low oil pressure detection.
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.
ENA CONVERSE	Enables conversational programming.
SIDE MNT. TC	Enables side mount T.C. logic.
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 — LEAD COMPENS SHIFT
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.

Parameter 59 — MAX FEED RATE (UNIT)
Maximum feed rate in inches per minute.

Parameter 60 — TURRET START DELAY
Maximum delay allowed in start of tool turret. Units are milliseconds. After this time, an alarm is generated.

Parameter 103 IN POSITION LIMIT Z
Same definition as Parameter 101.

Parameter 104 IN POSITION LIMIT A
Same definition as Parameter 101.

Parameter 105 HOLDING LIMIT X
Fuse level in % of max power to motor. Applies only when motor is stopped.

Parameter 106 HOLDING LIMIT Y
Same definition as Parameter 105.

Parameter 107 HOLDING LIMIT Z
Same definition as Parameter 105.

Parameter 108 HOLDING LIMIT A
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 x 0.075 = 90000 steps/second**
For a 2000 line encoder and 6 mm screw, this would be
60 x 90000 / 33867 = 159 inches min

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 BEMF BIAS
Back EMF bias for **X** motor. This is arbitrary units.

Parameter 122 Y BEMF BIAS
See Parameter 121 for description.

Parameter 123 Z BEMF BIAS
See Parameter 121 for description.

Parameter 124 A BEMF BIAS
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	Same as Parameter 145.
Parameter 147	Z	Same as Parameter 145.
Parameter 148	A	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 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:

HORZ T.C.	Enables tool changer for horizontal machining centers (VH models only).
RST STOPS T.C.	Tool changer can be stopped with RESET button.
M21-28 @ 540	When enabled (1), M21-M28 is installed at cable 540.
ENA CONVEYOR	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.
RESERVED	
RESERVED	
RESERVED	
SPIGOT KEY INV	When (1) the commands to the conveyor motor are reversed so that forward becomes reverse. If the conveyor bit is wired incorrectly, this bit can be set so that the conveyor runs in the proper direction.
REV CONVEYOR	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.
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 pallete rotate/part ready; inputs enabled if set to 1.
RMT TOOLS RLS	If set to 1 allows use of remote tool release button.

Parameter 210	RESERVED
Parameter 211	RESERVED
Parameter 212	RESERVED
Parameter 213	RESERVED

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 consectutive overcurrents in which the overcurrents is considered another retry. If this amount of time passes be tween overcurrents then the retry count is set to (0). Default is 1500, 30 minutes.

Parameter 221 Reserved
Parameter 222 Reserved
Parameter 223 Reserved
Parameter 224 Reserved
Parameter 225 Reserved
Parameter 226 CIRC MAX FEED CNTRL
This parameter is used to limit the feed rate for circles with a small radius. This parameter limits following error to 1/4 the dimension of the radius of the circular arc. The larger the value of this parameter, the more the control will limit feed during circular interpolation.