



INSTALLATION FOR THE VF-SERIES MILL





1. INSTALLING THE VMC

GENERAL WARNINGS

It is the shop operator's responsibility to make sure that everyone who is involved in installing and operating the Vertical Machining Center (VMC) is thoroughly acquainted with the installation, operating, and safety instructions provided with the machine BEFORE they perform any actual work.

NOTE: The ultimate responsibility for safety rests with the shop operator and the individuals who work with the machine.

The electrical power must meet the specifications in this manual. Attempting to run the VMC from any other source can cause severe damage and void the warranty.

DO NOT press POWER UP/RESTART on the control panel until after the installation is complete.

DO NOT attempt to operate the machine before all of the installation instructions have been completed.

Observe all of the warnings and cautions in the "Installation" section of the operation manual.

Move the crated VMC with care — position forklift forks only as specified by the marks on the crate and the instructions in this manual.

NEVER service the VMC with the power connected.

NEVER leave the control boxes open when the power is on. A deadly high voltage may be present in **ALL** areas of the control boxes.

IMPORTANT!!! Once the machine is installed, the control cabinet must be locked and the key given to the shop manager.

IF YOU ARE INSTALLING AN APC WITH YOUR MILL PLEASE REFER TO THE APC SECTION OF THIS MANUAL FOR COMPLETE INSTALLATION INSTRUCTIONS.



OPERATING DIMENSIONS

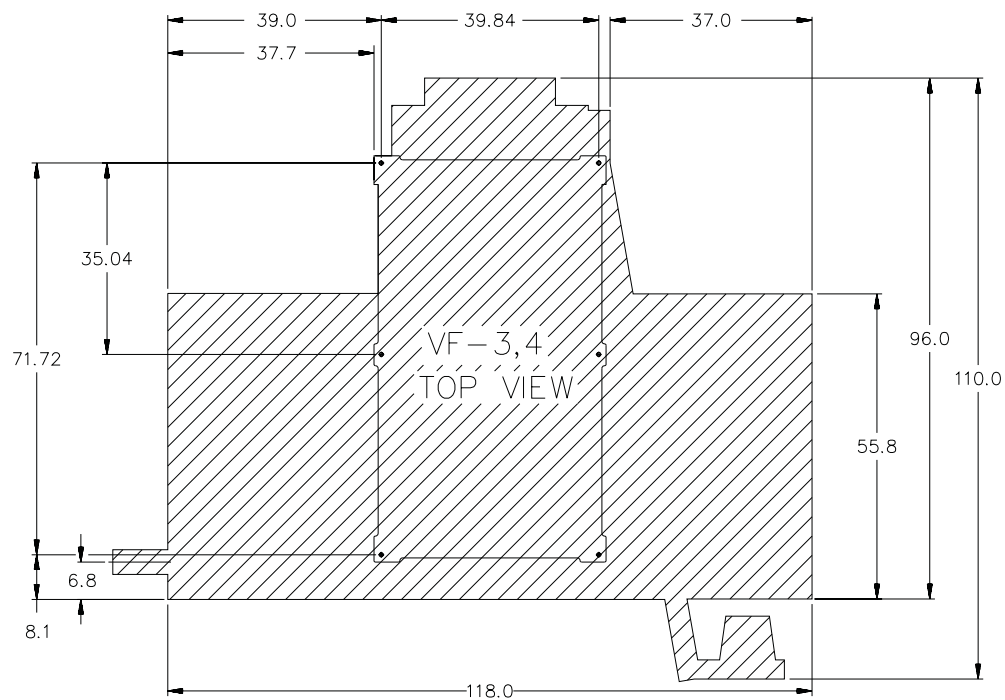
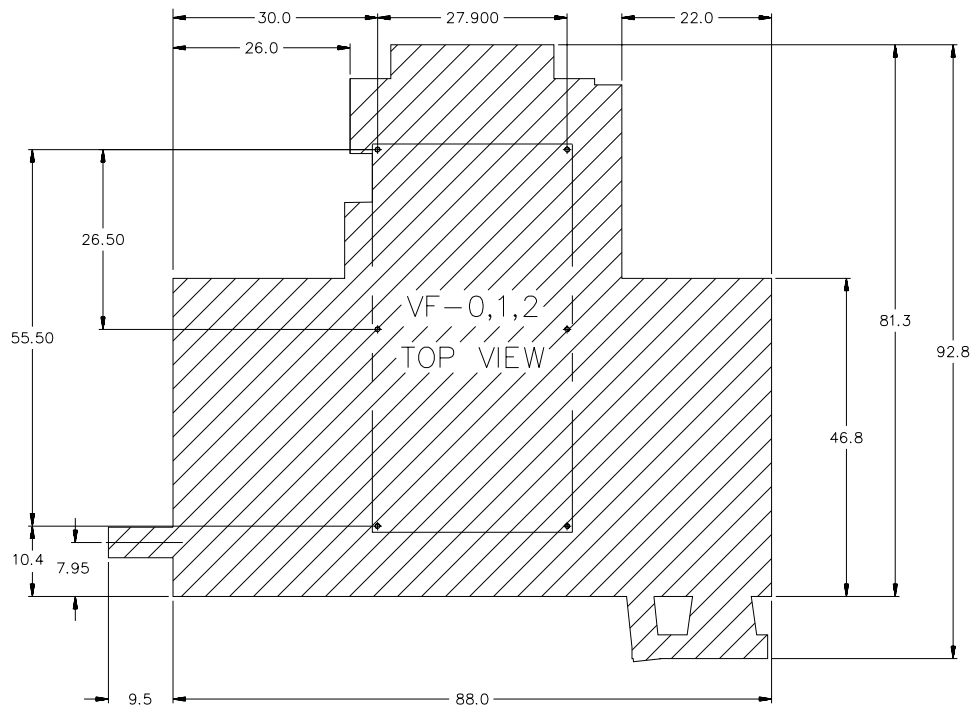
OPERATING	VF-O/E	VF-OE/EXT	VF-1	VF-2	VF-3	VF-4	VF-5/40	VF-5/50
Height (in.)	107	107	104.5	104.5	117.5	117.5	118.2	126.5
Width (in.)	109	109	138	109	138	138	148	148
Depth (in.)	137	137	157	137	157	157	163	163
Weight (lb.)	7,000	7,900	7,100	8,000	12,500	13,300	14,500	14,600
	VF-6	VF-7	VF-8	VF-9	VF-10	VF-11	50Taper	
Height (in.)	124	124	124	124	124	124	138	
Width(in.)	178	190	178	190	255	257	---	
Depth (in.)	160	160	176	176	160	178	---	
Weight (lb.)	21,000	23,000	24,000	25,000	28,000	29,400	---	

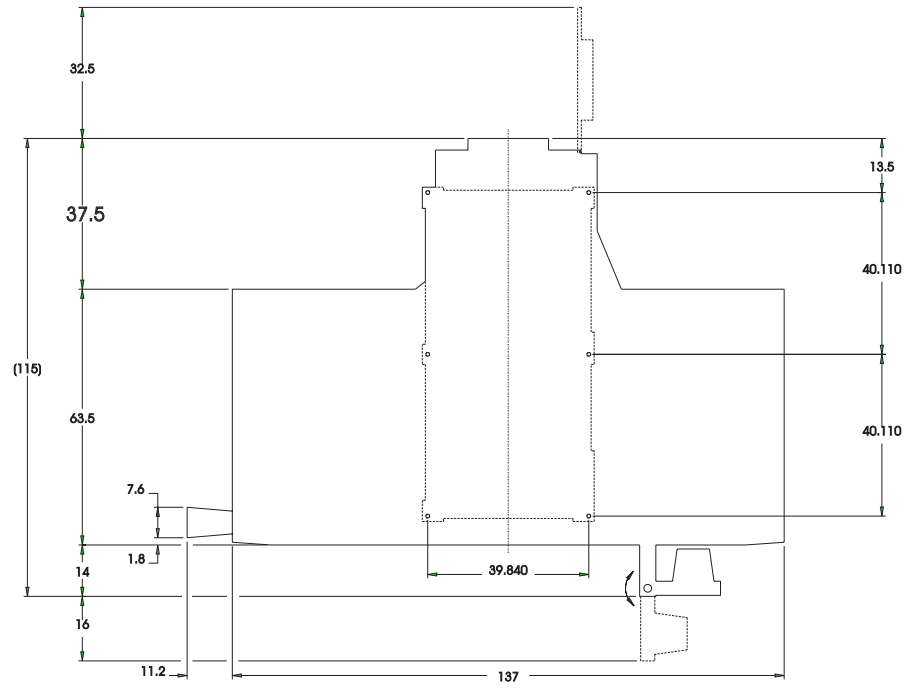
NOTE: The *operating dimensions* are the maximum dimensions of the machine during operation. These are the dimensions of the machine with the control panel door completely open, the spindle head at it's highest point, the control at it's most forward position, and with the discharge tube installed.

Out of Crate	VF-O/1/2	VF-3/4	VF-5/40	VF-5/50	VF-6
Height (in.)	91	102	101	105	107
Width (in.)	88	125	137	137	156
Depth (in.)	93	96	98.5	98.5	96
Out of Crate	VF-7	VF-8	VF-9	VF-10	VF-11
Height (in.)	107	107	107	107	107
Width (in.)	183	156	183	257	257
Depth (in.)	96	117	117	96	117

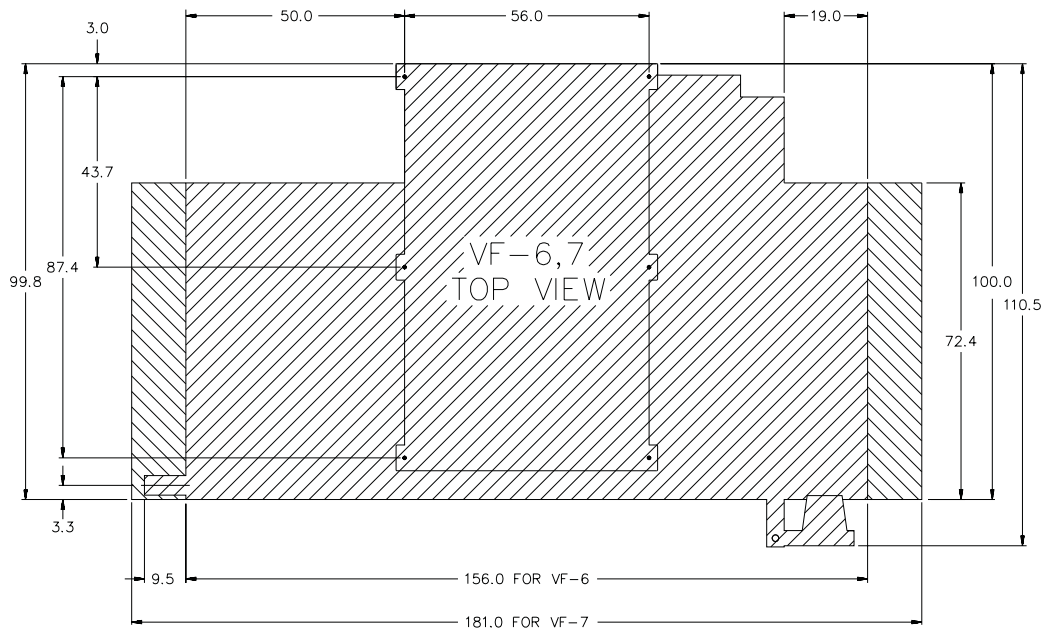
**MACHINE FOOTPRINTS**

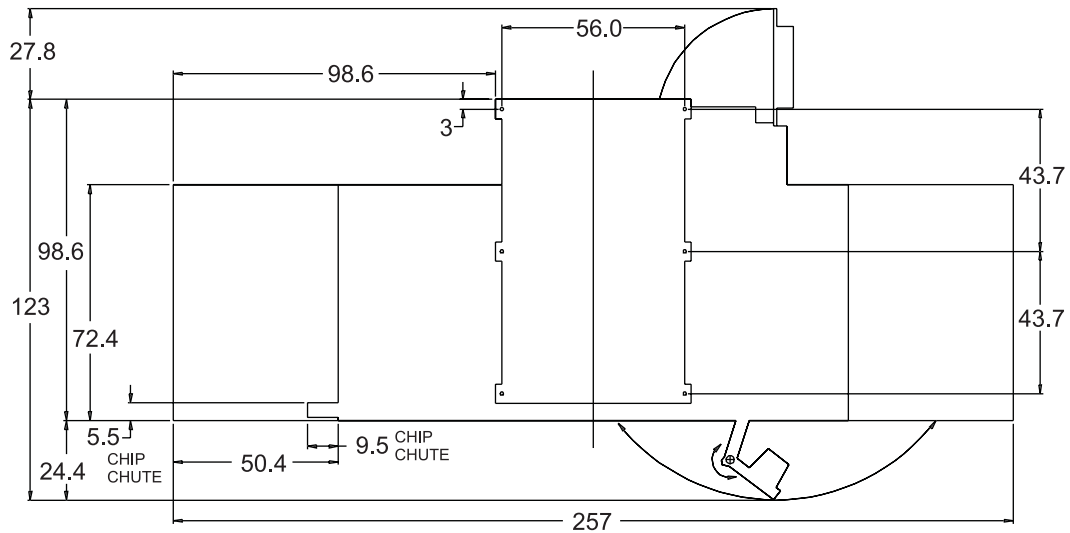
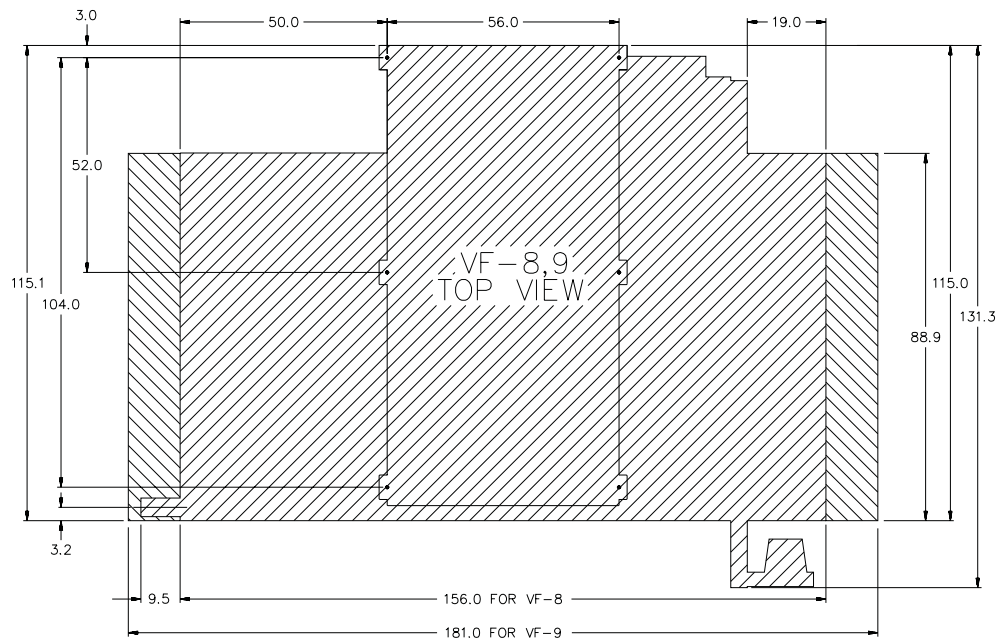
Dimensions are in inches





VF-5 FOOTPRINT



**VF-10 FOOT PRINT**



SERVICE REQUIREMENTS

GENERAL REQUIREMENTS

Operating Temperature Range 5 to 40°C
 Storage Temperature Range -20 to 70°C
 Ambient Humidity: less than 90% relative humidity, non-condensing
 Altitude: 0-7000 ft.

ELECTRICITY REQUIREMENTS

IMPORTANT! REFER TO LOCAL CODE REQUIREMENTS BEFORE WIRING MACHINES.

ALL MACHINES REQUIRE:

AC input power is three phase Delta or Wye power, except that the power source must be grounded (e.g. leg or center leg for delta, neutral for Wye)

Frequency range is 47-66 Hz

Line voltage that does not fluctuate more than +/-5%

Voltage imbalance or no more than 2%

Harmonic distortion is not to exceed 10% of the total RMS voltage

20-15 HP System	Voltage Requirements (195-260V)	High Voltage Requirements (354-488V)	20-15 HP Products
Power Supply	50 AMP	25 AMP	SL-20
Haas Circuit Breaker	40 AMP	20 AMP	Standard VF
If service run from ele. panel is less than 100' use:	8 GA. WIRE	12 GA. WIRE	Standard HS 1/2 10K (VF-E / EX)
If service run from ele. panel is more than 100' use:	6 GA. WIRE	10 GA. WIRE	
40-30 HP System	Voltage Requirements (195-260V)	High Voltage Requirements (354-488V)	40-30 HP Products
Power Supply	100 AMP	50 AMP	SL-30 / 40
Haas Circuit Breaker	80 AMP	40 AMP	5000 RPM (SL)
If service run from ele. panel is less than 100' use:	4 GA. WIRE	8 GA. WIRE	HT10K (VF) HT 10K (HS) VR-11
If service run from ele. panel is more than 100' use:	2 GA. WIRE	6 GA. WIRE	50 Taper (VF) Lathe BB

**WARNING!**

- A separate earth ground wire of the same conductor size as the input power is required to be connected to the chassis of the machine. This ground wire is required for operator safety and for proper operation. This ground may be supplied from the main plant ground, a local cold-water-pipe, or ground rod adjacent to the machine.
- Do not connect to 480V, unless machine includes high voltage option.
- Maximum voltage leg-to-leg or leg-to-ground should not exceed 260 volts or 540 volts for high voltage machines!
- The current requirements shown reflect the circuit breaker size internal to the machine. This breaker has an extremely slow trip time. It may be necessary to size the external service breaker up by 20-25% for proper operation.

AIR REQUIREMENTS

The VMC requires a minimum of 100 PSI at 4 scfm 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: Add 2 scfm to the above minimum air requirements if the operator will be using the air nozzle during pneumatic operations

Machine Type	Main Air Regulator	Input Airline Hose Size
VF-1 through VF-11 (40 taper)	85 psi	3/8"
VF-5 through VF-11 (50 taper)	85 psi	1/2"

The recommended method of attaching the air hose is to the barb fitting at the back of the machine with a hose clamp. If a quick coupler is desired, use a 1/4" for 40 taper machines, or a 3/8" for 50 taper machines and machines with the side mount tool changer option.

WARNING

AN INADEQUATE AIR SUPPLY
WILL CAUSE TOOL CHANGER FAULTS

FOLLOW THESE GUIDELINES:

MINIMUM AIR SUPPLY PRESSURE TO MACHINE IS 100 PSI.
OBSERVE GAGE DURING TOOL CHANGE - 10 PSI MAX. DROP.
USING THE AIR GUN DURING TOOL CHANGES MAY CAUSE
FAULTS IF THE AIR SUPPLY TO THE MACHINE IS MARGINAL.
ALLOW 2 HP OF AIR COMPRESSOR PER MACHINE,
(I.E., 5 MACHINES REQUIRE A 10 HP AIR COMPRESSOR).
USE MINIMUM 3/8 ID HOSE FOR 40 TAPER MACHINES,
MINIMUM 1/2 ID HOSE FOR 50 TAPER & HS MACHINES.
AVOID QUICK DISCONNECTS IN SUPPLY LINES - THEY ARE
RESTRICTIVE.

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 oil and/or water to pass into the machine.

NOTE: The nipple between the air filter/regulator and the Bijur oil lubricator (See illustration in "Air Connection" section) reservoir tank below the control box on the back of the machine is for the optional rotary table. DO NOT use this as a connection for an auxiliary air line. Auxiliary connections should be made on the left side of the air filter/regulator.



WARNING!

WHEN THE MACHINE IS OPERATING AND THE PRESSURE GAUGE (ON THE MACHINE REGULATOR) DROPS BY MORE THAN 10 PSI DURING TOOL CHANGES, INSUFFICIENT AIR IS BEING SUPPLIED TO THE MACHINE.

INSTALLATION TOOLS REQUIRED

Precision bubble level (0.0005 inch per 10")
 1 1/8" hex wrench or ratchet
 1 1/2" wrench
 Forklift with the following specifications:

Test indicator (0.0005)
 3/4" wrench
 Claw hammer

	VF-O/E	VF-OE/EXT	VF-1	VF-2	VF-3	VF-4	VF-5/40
Machine Weight *	7,000	7,900	7,100	8,000	12,500	13,000	14,500
Fork Length	8'	8'	8'	8'	8'	8'	8'

	VF-5/50	VF-6	VF-7	VF-8	VF-9	VF-10	VF-11
Machine Weight *	14,600	2,100	23,000	24,000	25,000	28,000	29,400
Fork Length	8'	8'	8'	8'	8'	8'	8'

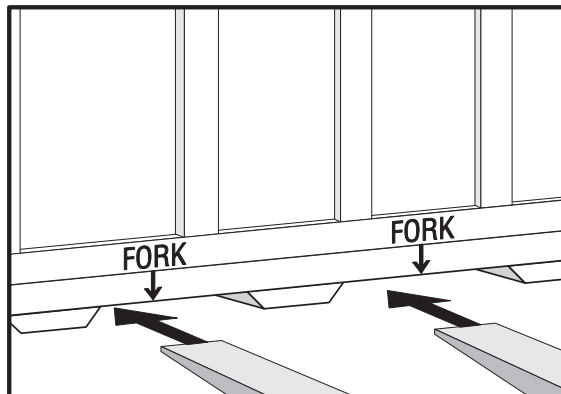
* The forklift must be capable of lifting **at least** this weight.

MATERIALS REQUIRED

Wire and air hose or piping as specified in the Service Requirements section
 A small amount of grease
 Way lube for the lubricator (Vactra #2)

**MOVING THE CRATE****CAUTION!**

THE VMC CRATE CAN ONLY BE MOVED WITH A FORKLIFT.



CAUTION! The fork positions are marked on the crate. (Also, note that there are three skids at each side of the pallet. The heavy part of the machine [the back] is positioned over the two skids that are closest together.) If the fork positions are ignored, there is a good chance that the retaining bolts will be sheared off by the forks and also that the machine will tip over when it is picked up.



UNPACKING THE VMC

UNCRATING

NOTE: Unless you are certain that you will not be shipping the machine, the crate and packing materials should be stored for reuse. Be careful not to damage the crate and the other packing materials.

CAUTION! Use extreme caution not to damage the pressurized hydraulic counterbalance when removing the machine from its container.

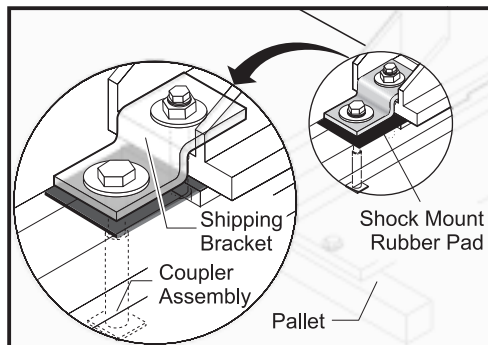
1. Pry off the clips around the top of the crate with a claw hammer and remove the top panel.
2. Pry off all but one clip at each corner of the crate.
3. Remove plastic cover.

CAUTION! Do not put undue pressure on the top of the machine as you remove the plastic.

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. Remove the coolant tank and the cleats that held them in place.



6. Unbolt shipping bracket.
7. Remove the nuts, on the leveling screws, holding the shipping bracket to the base casting. Remove the shipping brackets.
8. Lift the machine off the pallet.

SETTING IN PLACE

Keep in mind when moving the VMC that much of its weight is concentrated in the column at the back. Therefore, when lifting a **VF-0/1/2** from the side, it is important that the forks of the forklift be positioned as close to the back of the machine as possible without being on the pads.

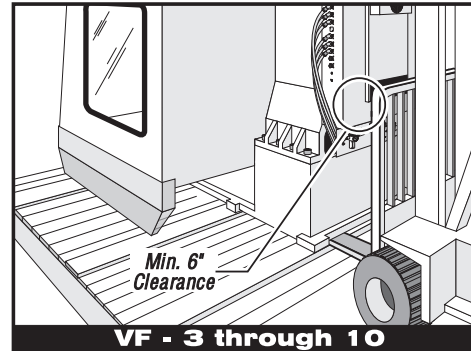
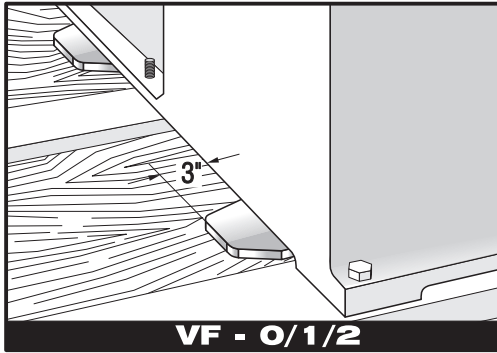
CAUTION! Do not lift the machine any farther than necessary off the floor when moving it, and move as slowly and cautiously as possible. Dropping the machine, even from a height of a few inches, can cause injury, result in expensive repairs, and void the warranty.

For the VF-0/1/2: The only acceptable way to move a **VF-0/1/2** is to pick it up from the **SIDE** with a forklift. Follow the machine weight and fork length specifications described earlier. The forks must be set as far apart as possible without being on the pads. The forks must be positioned all the way to the back of the VMC and they must extend at least 3" past the far side of the machine base. Also, there must be about approximately 6" clearance between the forklift and the side of the machine.

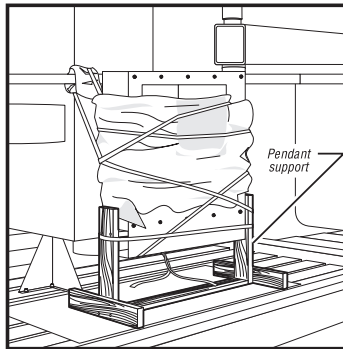
For the VF-3 through 10: The **VF-3 through 10** must be lifted from the **BACK** of the machine with a forklift. **IMPORTANT!** Follow the machine weight and fork length specifications described earlier. The forks must be set as far apart as possible without being on the pads. Also, there must be about approximately 6" clearance between the forklift and the back of the machine.

Attempting to move the machine any other way may void the warranty.

CAUTION! When lifting the machine with a forklift, be careful not to damage the sheet metal aprons with the forks.



1. Lift the machine clear of the pallet.
2. Move the machine to where it will be located. Grease the dimple in each leveling pad, and locate them under the leveling screws at the four corners. Then lower the machine.



3. Remove all banding and packing material around the control panel and the doors.
4. On the **VF-6/8**, remove the pendant support.
5. Remove the control arm shipping brace. On the **VF-3/4**, swing the control arm into position and bolt it to the support on the top front of the machine enclosure. On the **VF-6/8**, swing the control arm to the proper position.

**ANCHORING INSTRUCTIONS**

Anchors must be installed prior to setting the machine in place.

IMPORTANT! READ THE INSTRUCTIONS THOROUGHLY BEFORE ATTEMPTING TO ANCHOR THE MACHINE.

ANCHORING KIT: (P/N# 30-0088)	QTY
5/8-11 x 6 hex head bolt	6
5/8-11 x 2 1/8 coupling nut	6
5/8-11 x 8 stud	6
5/8 washer	6
5/8-11 hex nut	6
3/4-10 x 6 hex head bolt (FULLY THREADED)	6
3/4 10 hex nut	6
anchor strap	6
floor plate	6
epoxy (chock fast gray, PR-610 FR)	11 lbs

Note: There is only enough epoxy to fill the 1.5" Dia. x 8.5" deep holes.

REQUIRED TOOLS:

Carpenter's square
Hand drill and mixer blade

The anchoring kit contains enough supplies to build six complete anchors. Small vertical mills only require four anchors. Refer to the anchor pattern for the specific machine requirements.

Before drilling the anchor cores, make sure there is enough clearance around the machine to install the coolant tank. Refer to the appropriate page for machine footprint and coolant tank dimensions.

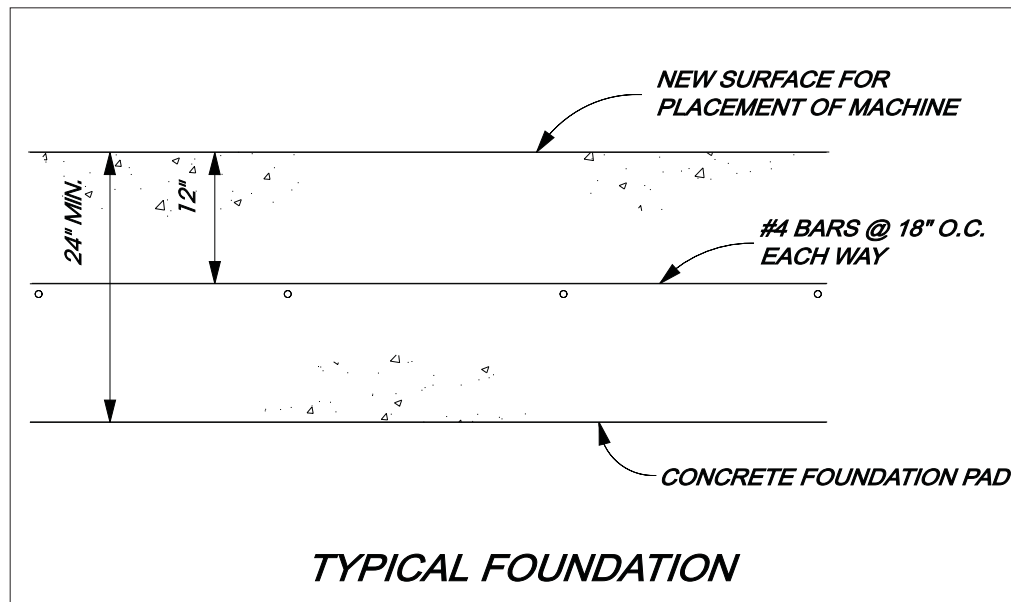
Note: To avoid breakage in the concrete ensure that bored holes are at least 12 inches away from any concrete joints.



FOUNDATION REQUIREMENTS:

A minimum 2-foot concrete foundation is required. The foundation should extend at least 1 ft. from the bored holes.

Machines with outriggers must have the foundation extend at least 1 ft. from the outrigger leveling pads.



FOUNDATION NOTES:

1. Concrete shall be 3500 psi at 28 day strength. Concrete aggregate shall be a 1" mix.
2. Steel reinforcing shall be 40 K.S.I.
3. The excavation shall be cut neat against the undisturbed soil. Any loose material in the excavation shall be removed so that the concrete bears on the undisturbed natural soil. This will help to prevent uneven settling.

ANCHORING OVERVIEW

Setting the anchors is a three step process. First, the anchors are partially assembled hand-tight and placed into cored holes with epoxy. This assembly keeps the anchors perpendicular to the floor while the epoxy is curing. Once the epoxy has cured, the studs and floor plates are removed so that the machine can be moved into position.

Once the machine is in position, the studs are threaded back into the coupling nuts with the long threaded end of the stud facing upwards. The anchors are then reassembled loosely and the machine leveled.

Once the machine is leveled, the anchor bolts are torqued in increments to a specified rating.

**CORING INSTRUCTIONS**

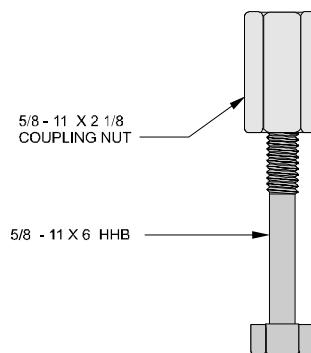
Refer to the appropriate hole pattern and machine footprint before drilling.

Cored holes must be 1.5 inch (3.81cm) diameter by 8.5 inches (21.59cm).

Note: Thoroughly clean and dry the cored holes. The holes must be completely free of oil and water and if blasting with air, ensure that the air is free of oil.

STEP 1 ANCHOR PRE-ASSEMBLY

Screw the 5/8-11 HHB into the coupling nut halfway.



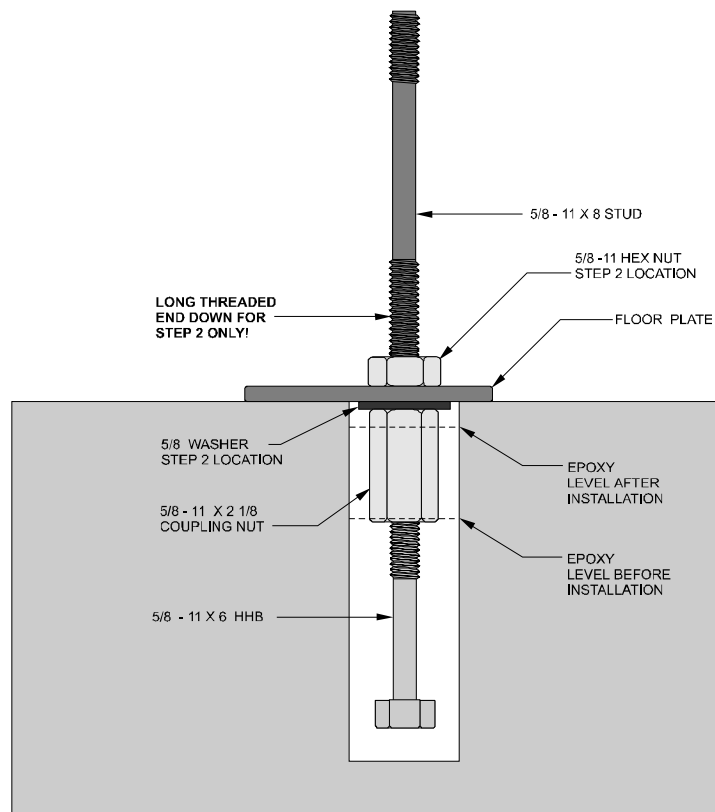
Pre-assembly diagram .

Screw the 5/8-11 nut onto the long threaded side of the stud halfway.

Place the floor plate and the 5/8 washer onto the stud. Screw the stud into the coupling nut until it bottoms out. Then tighten the hex nut until the floor plate is held tight between the two nuts (see Figure).



Step 2 Setting the Anchors



Step 2 diagram

Mix the epoxy according to the manufactures instructions. Working time for the epoxy is approximately 30 - 35 once it is mixed together.

Fill the hole until the epoxy is approximately 2 inches from the top of the hole (Refer to Figure).

Important! Do not overfill the hole because the epoxy will rise once the assembly is placed into the hole. Make sure that the epoxy does not seep onto the top of the coupling nut or you may not be able to remove the stud.

Place the anchor assembly into the hole (stud end sticking up), then lift the assembly out of the hole like a dipstick and check to see how much of the coupling nut is covered in epoxy. At least **half** of the nut must be covered in epoxy. At this point work as quickly as possible since the epoxy will begin to harden.

Use a carpenter's square to make sure the studs are perpendicular to the floor.

The epoxy will take 18 - 24 hours to cure.

**FINAL ANCHORING**

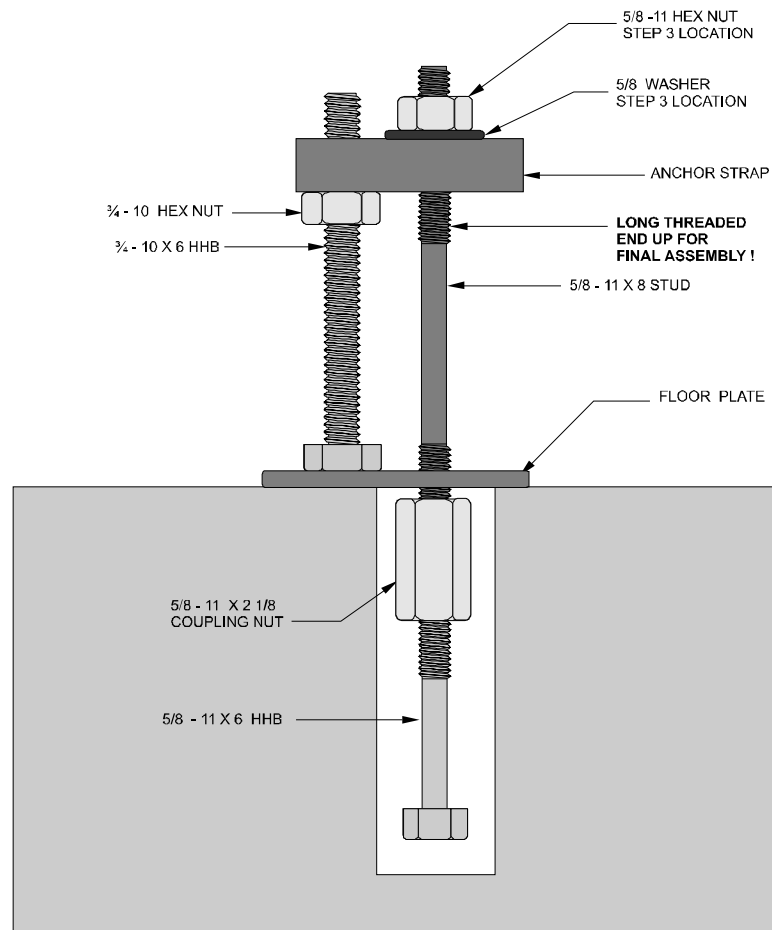
CAUTION! The Initial Setup and Leveling Procedures must be completed before final anchoring.

Once the epoxy has cured remove the studs, the floor plates, and the 5/8 nut and washers, from each hole.

Move the machine into position.

Screw the short threaded end of the studs into the coupling nuts, making sure the base casting does not interfere with the studs.

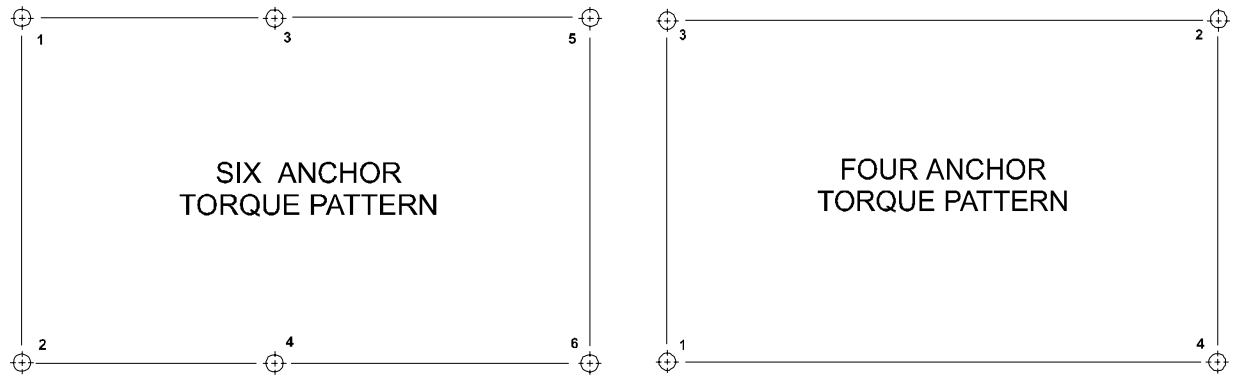
Assemble the anchors as shown in the Figure.



Final Anchor assembly



The anchor bolts must be torqued in three increments. Once the machine is level, torque the anchors to 30 FT-LB. in the pattern shown below, then recheck the level. At this point it is possible to adjust the leveling screws without loosening the anchors, if needed.



Machine torque patterns

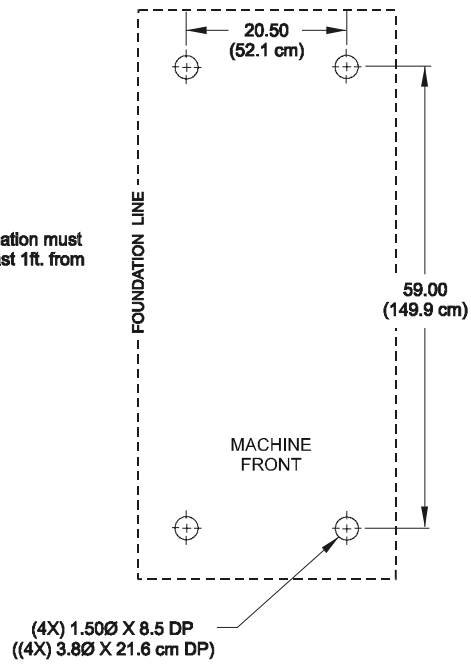
Continue to torque the anchors up to 50 FT-LB. in the same pattern, then recheck the level and make any necessary adjustments.

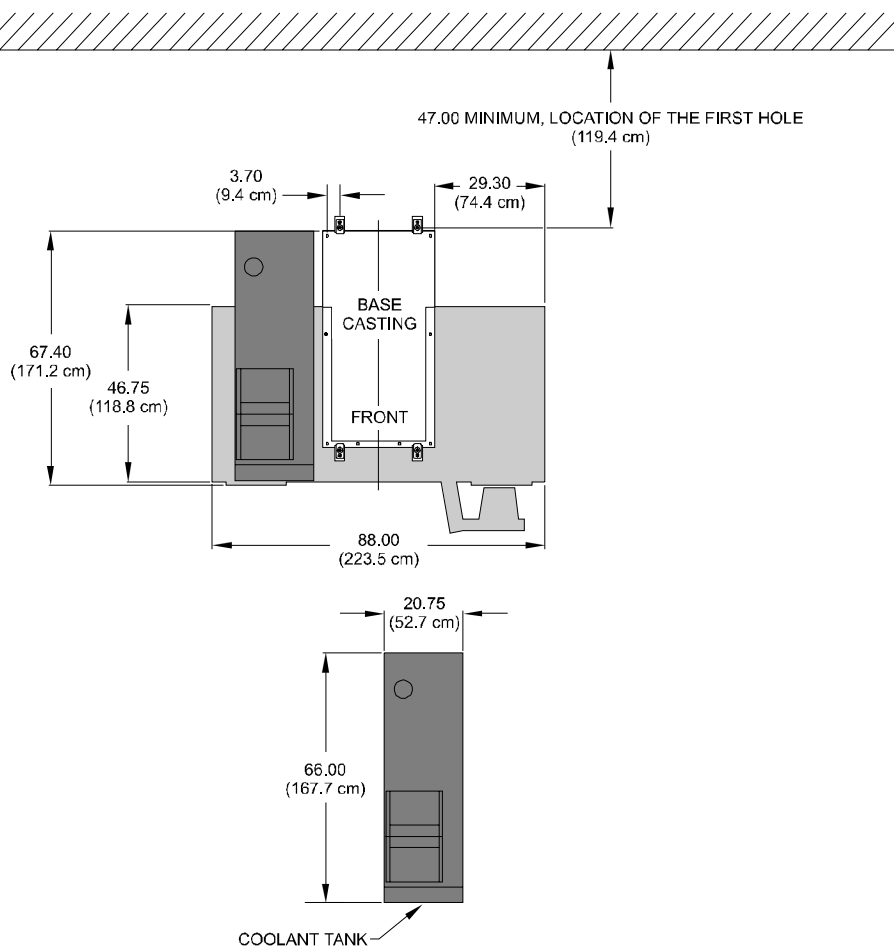
Finally torque the anchors to 80 FT-LB and recheck the level. At this point you will only be able to make very minor adjustment without loosening the anchors.

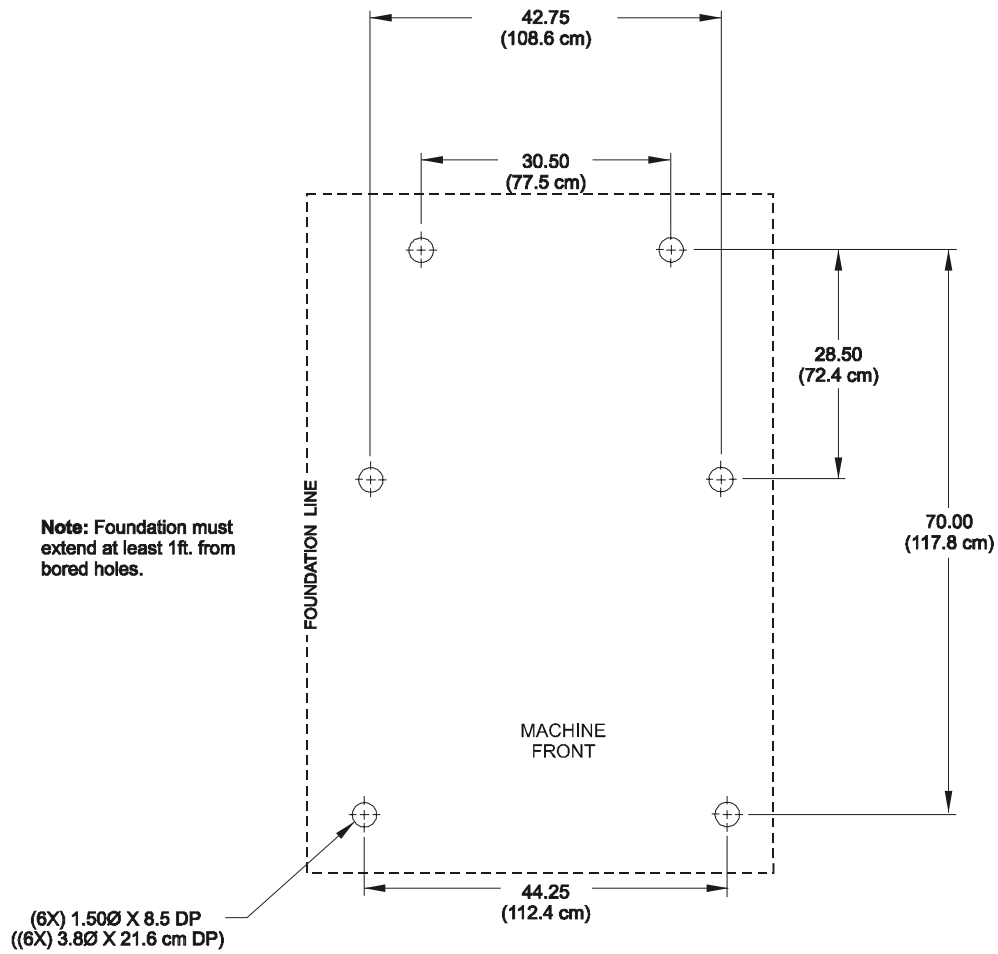
Do not overtighten the bolts or you could crack the concrete!

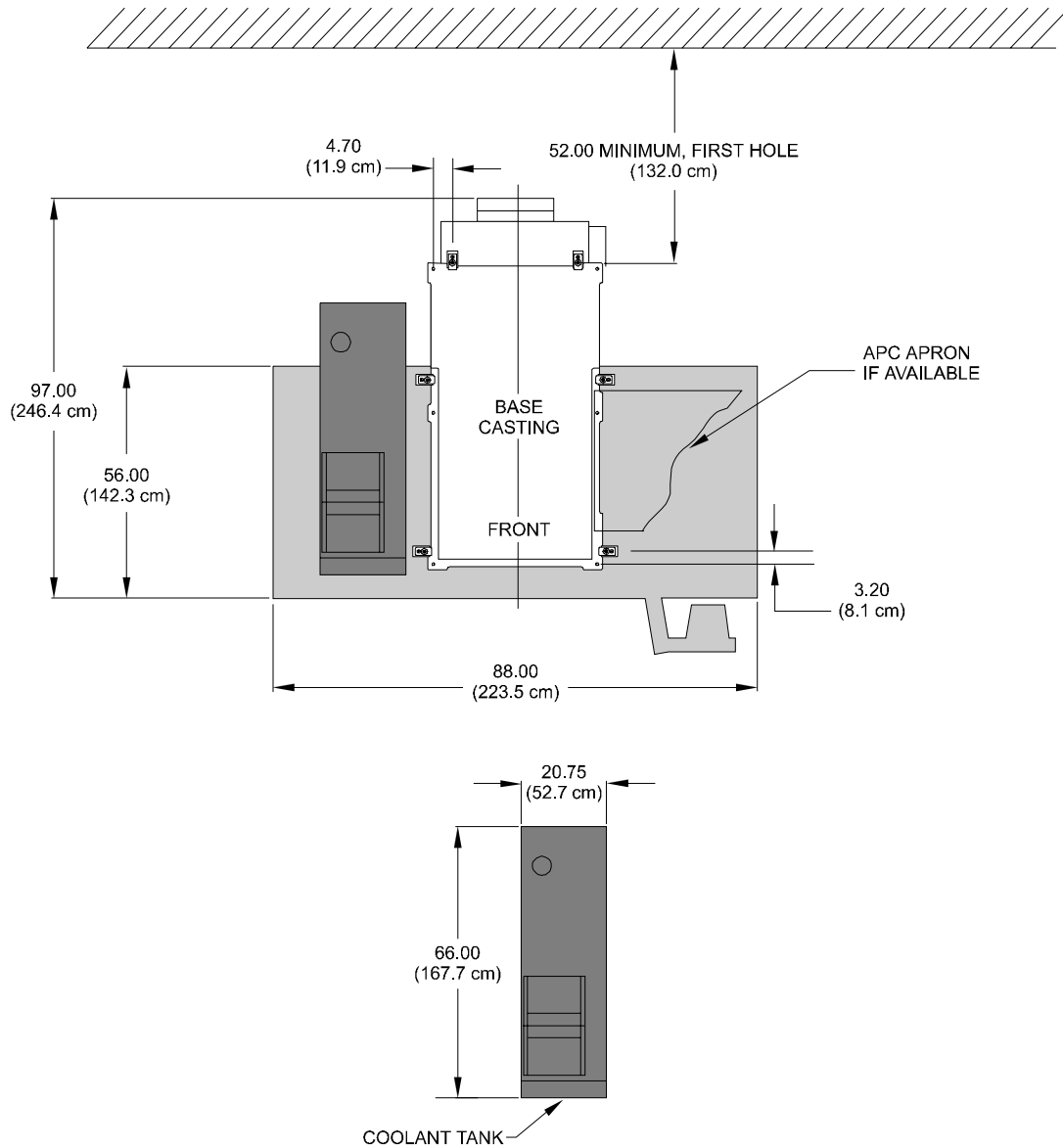
**ANCHOR PATTERN: VF-0/E/1/2**

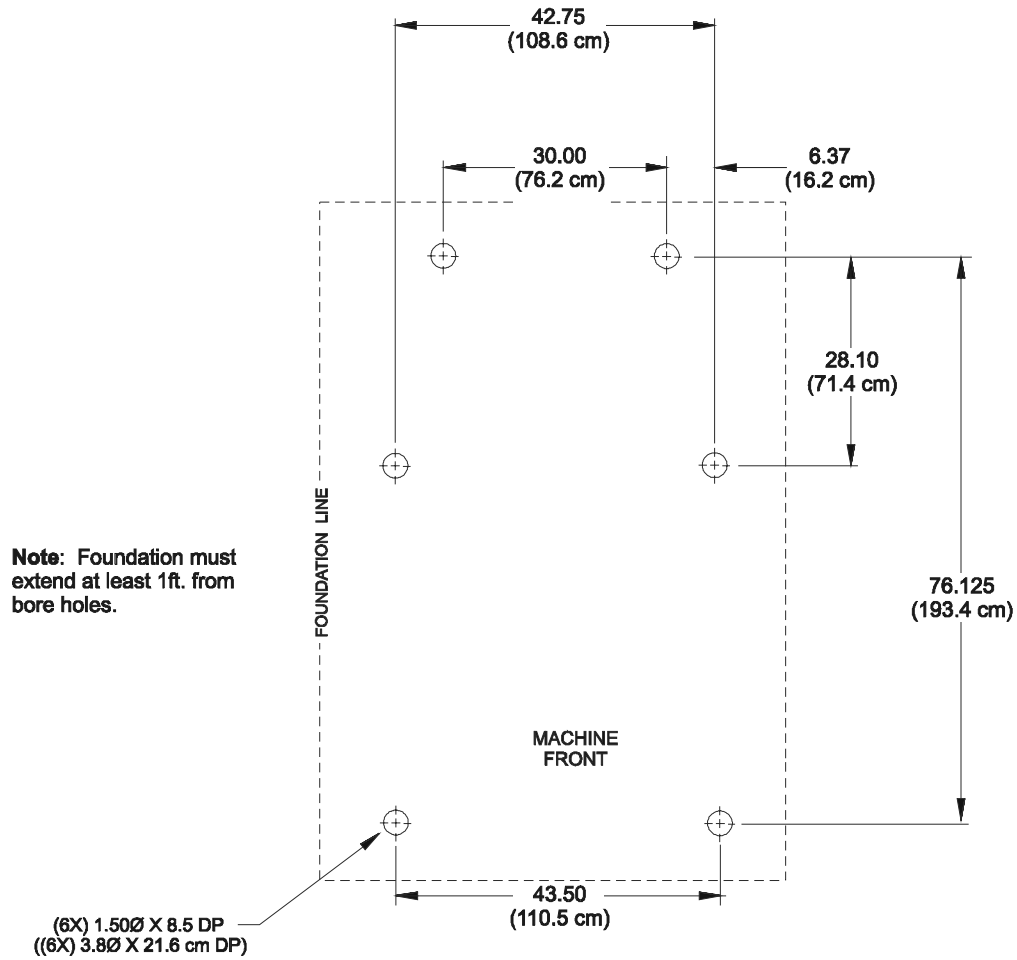
Note: Foundation must extend at least 1ft. from bored holes.

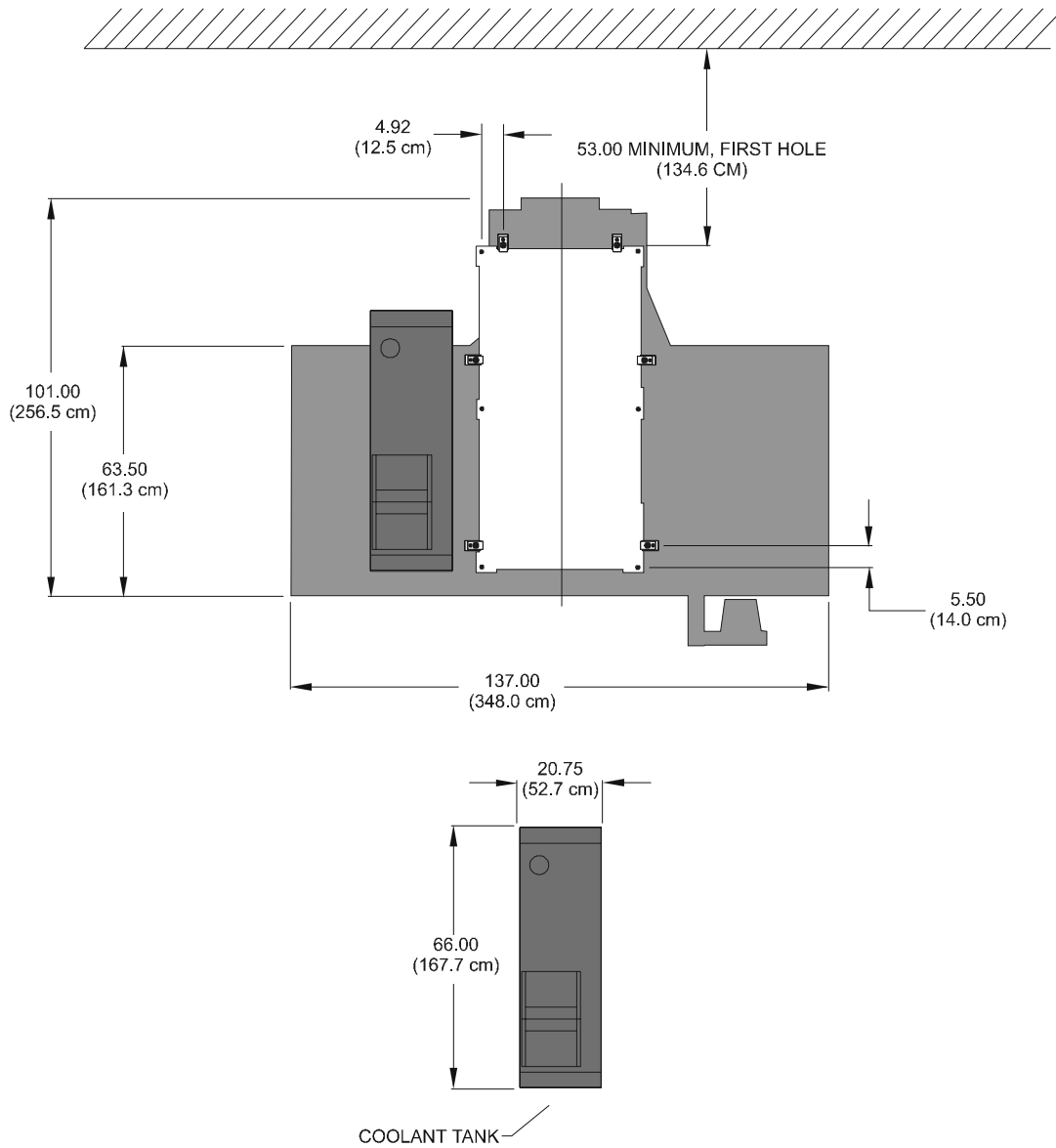


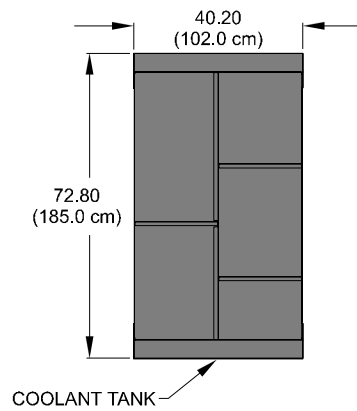
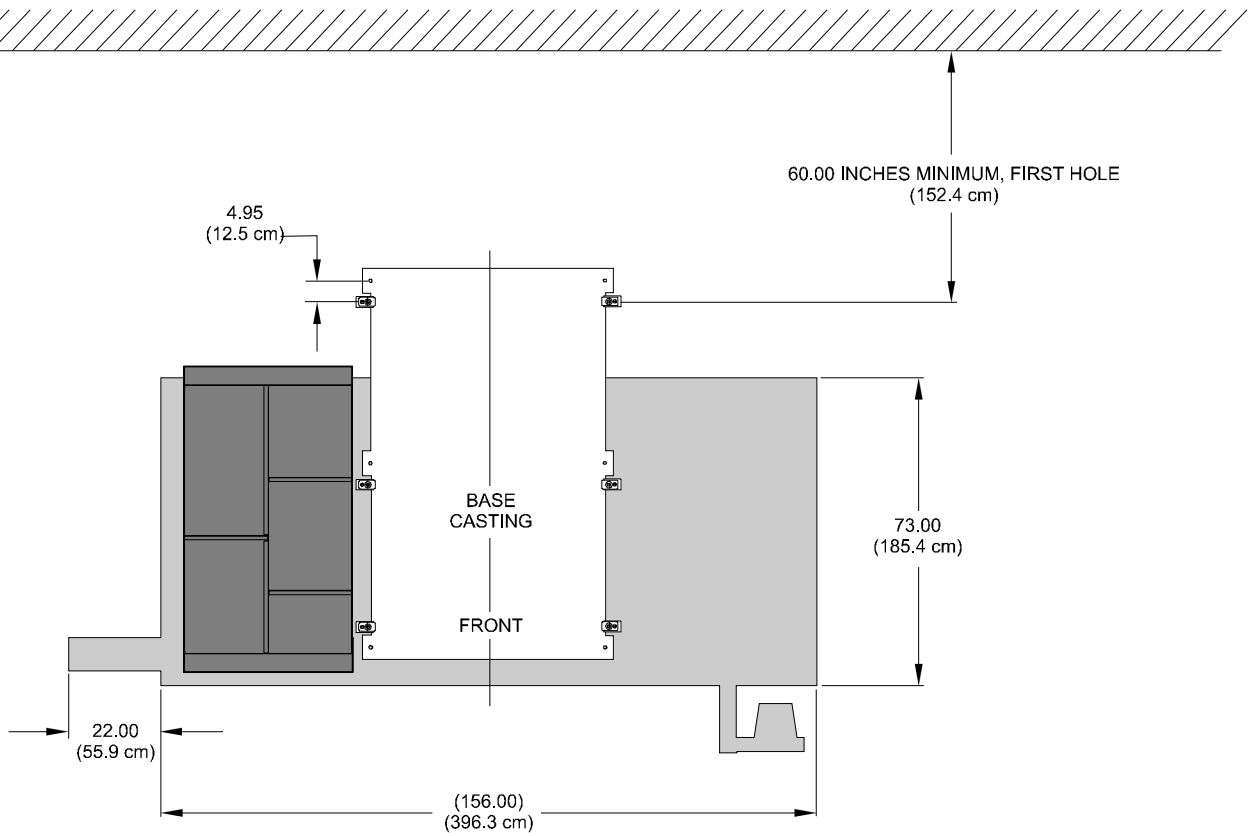
**MACHINE FOOTPRINT: VF-0/E/1/2**

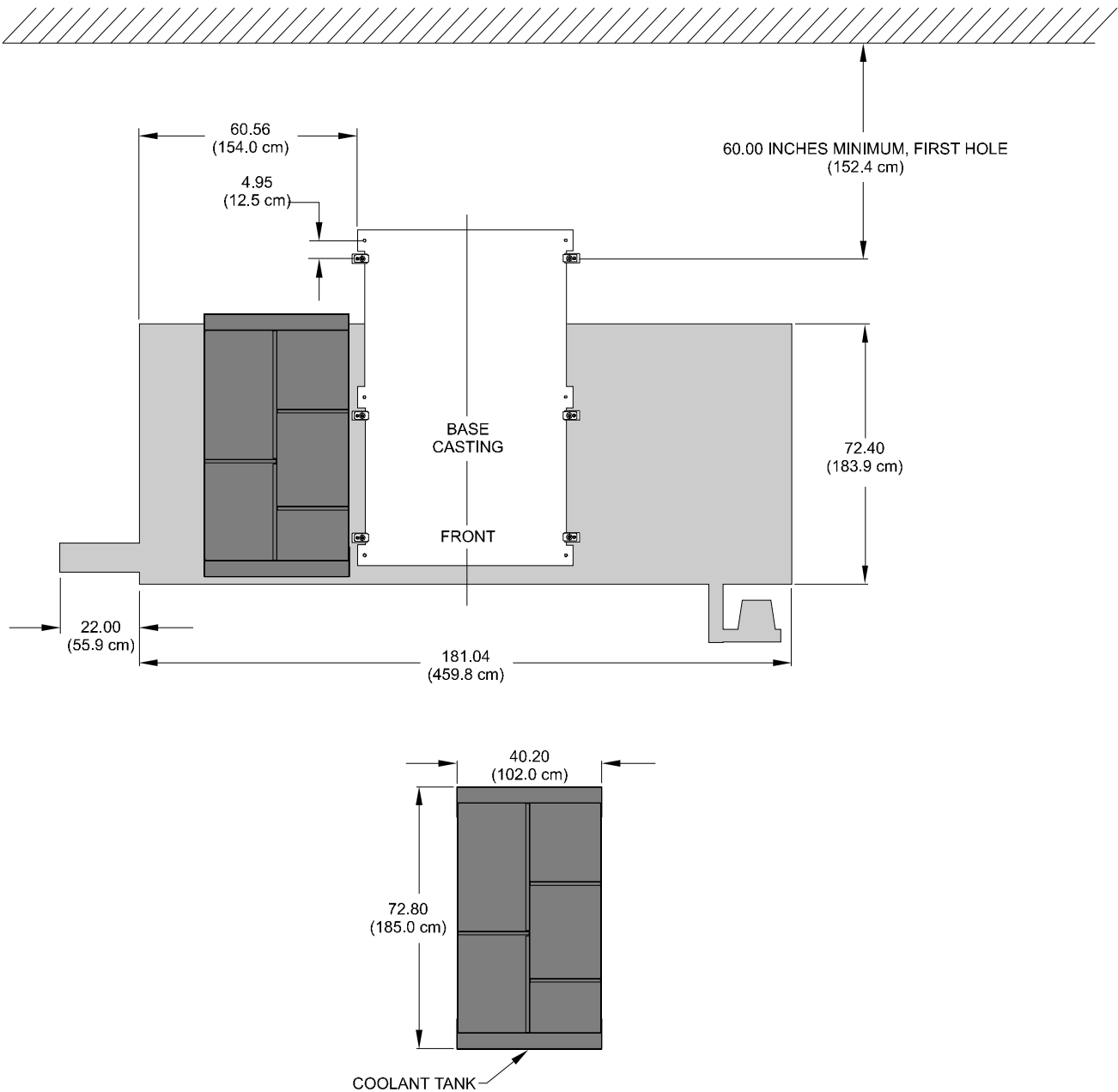
**ANCHOR PATTERN: VF-3/4**

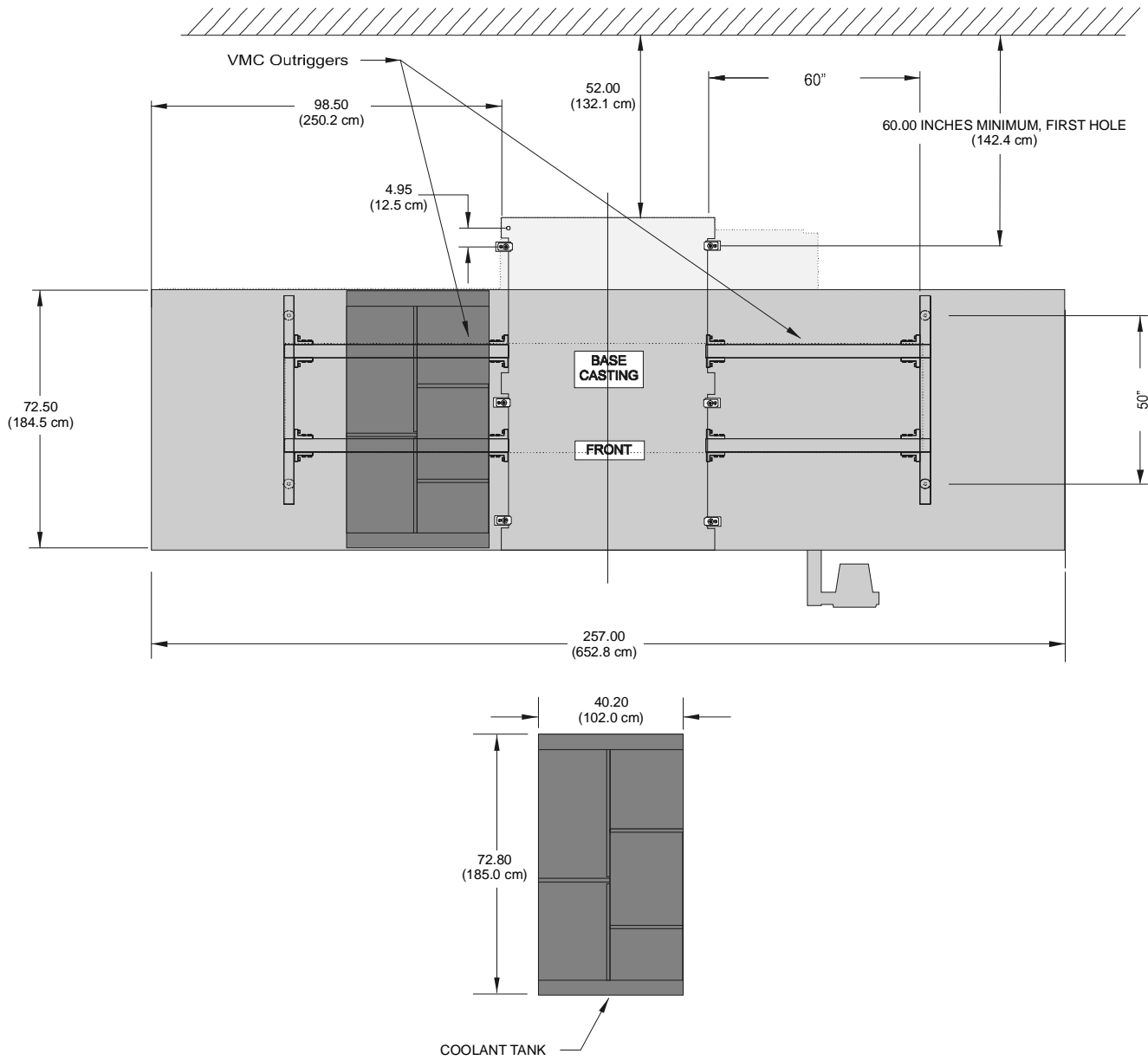
**MACHINE FOOTPRINT: VF-3/4**

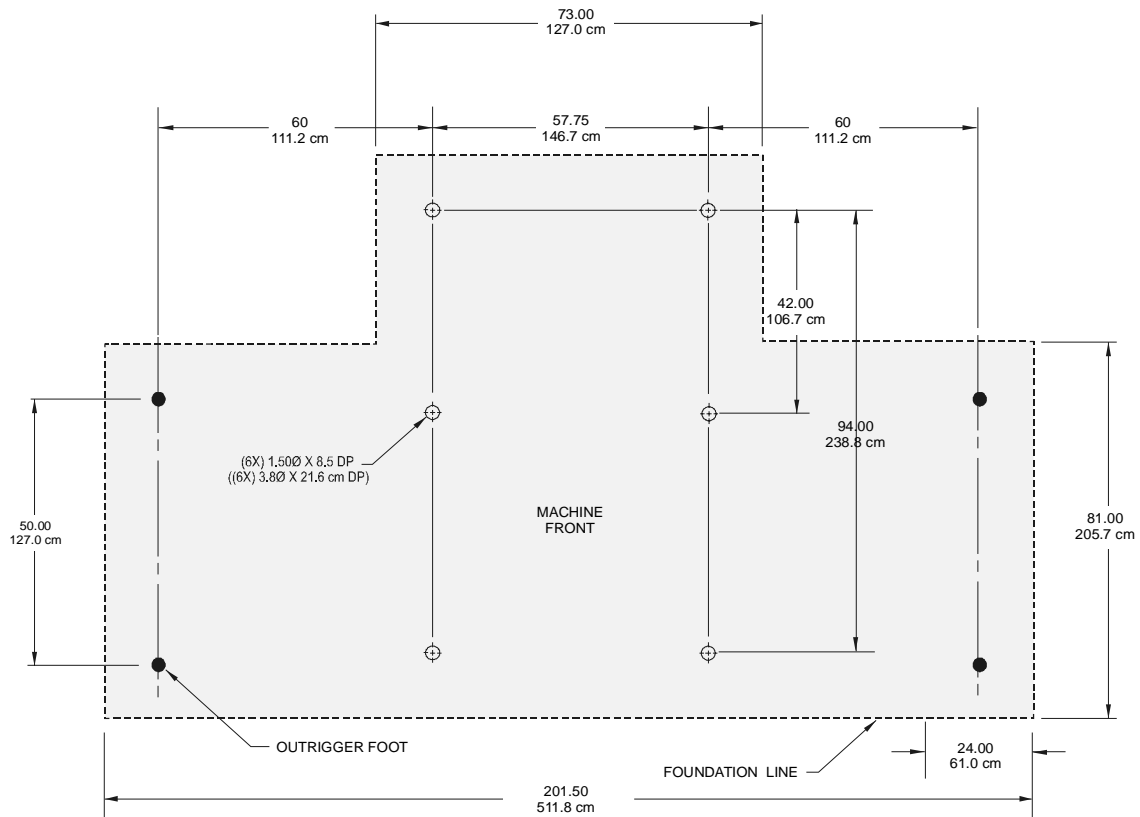
**ANCHOR PATTERN: VF-5**

**MACHINE FOOTPRINT: VF-5**

**MACHINE FOOTPRINT: VF-6**

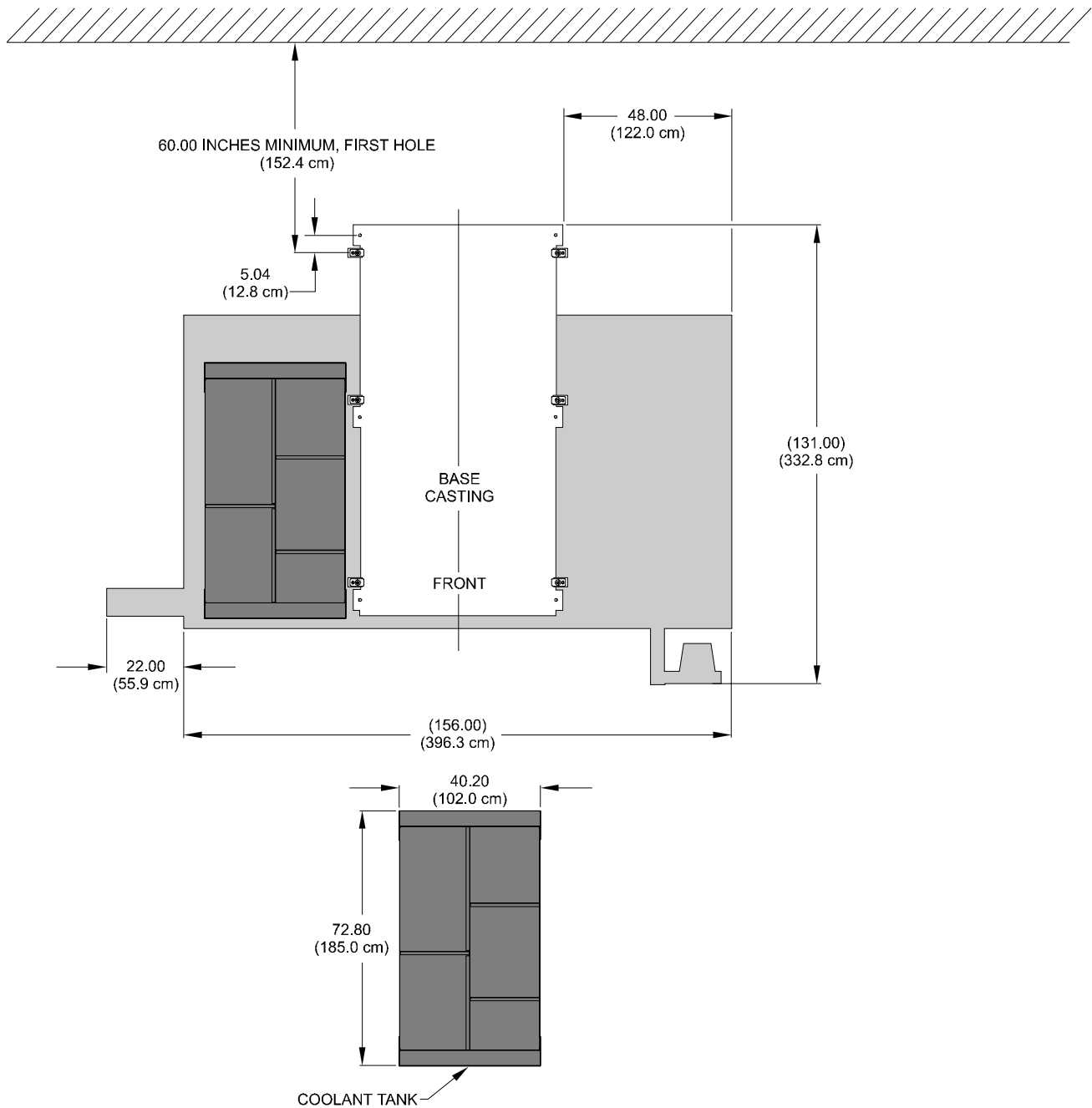
**MACHINE FOOTPRINT: VF-7**

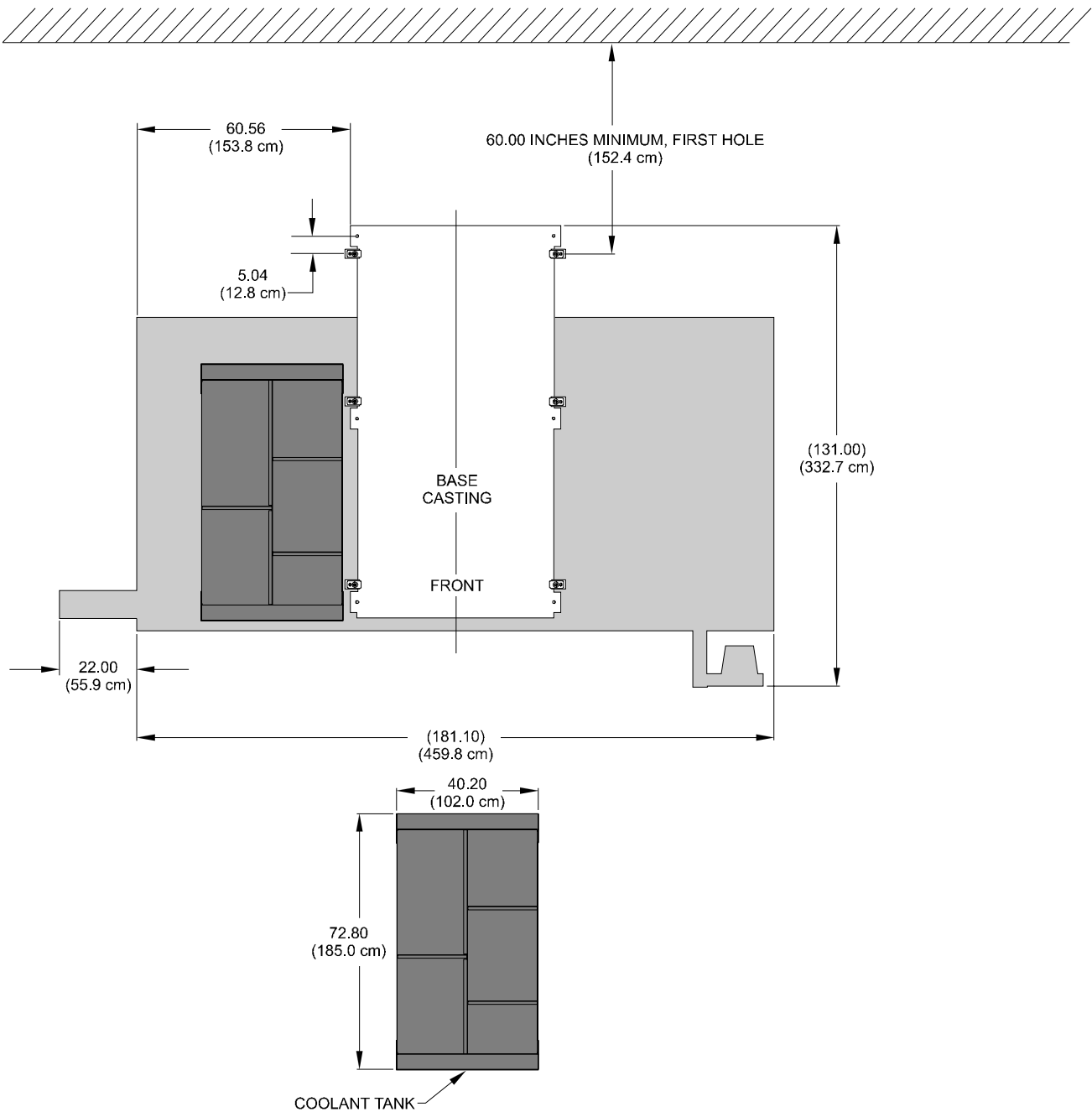
**MACHINE FOOTPRINT: VF-10**

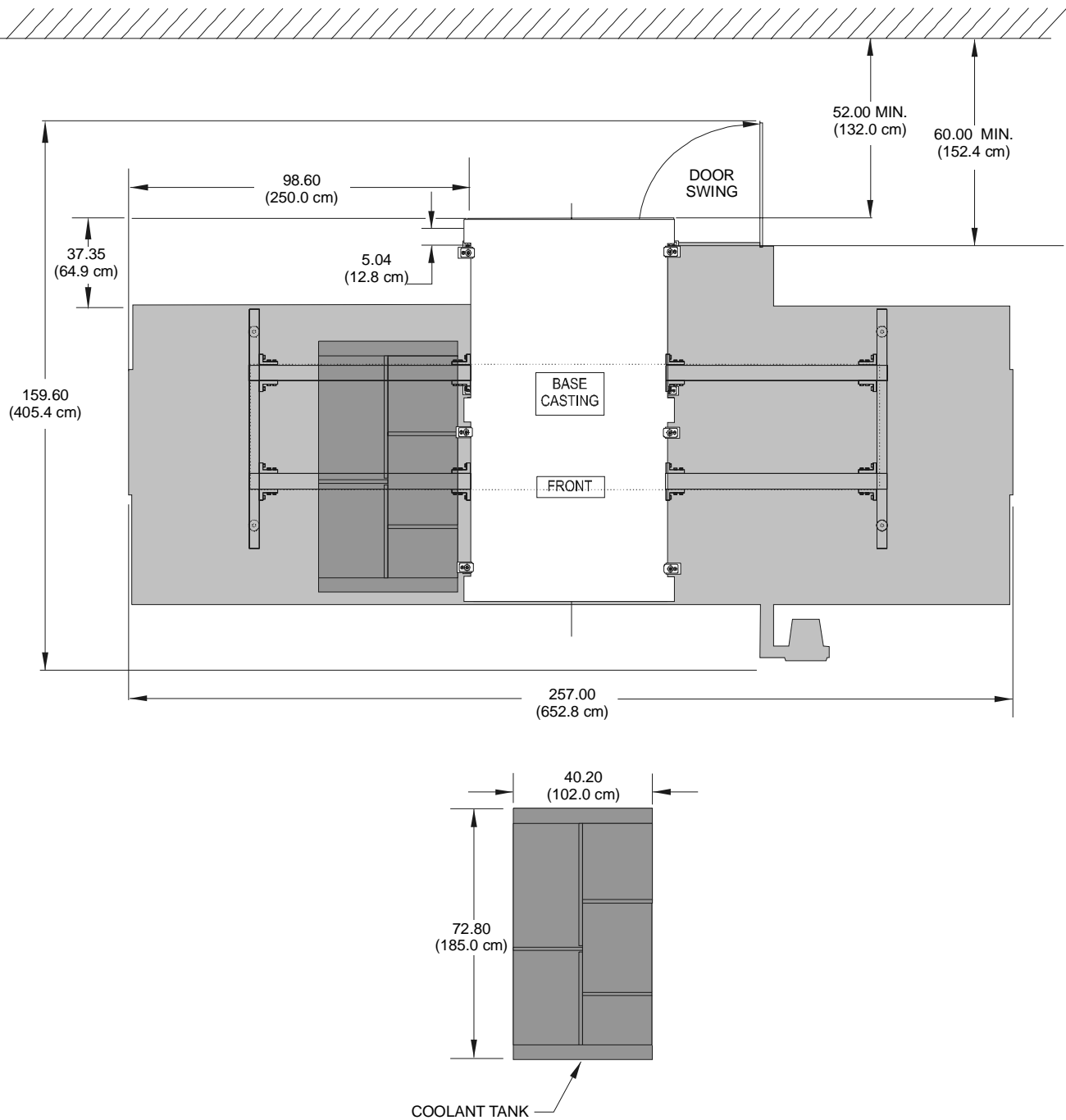
**ANCHOR PATTERN FOR VF-8/9/11**

Note: Foundation must extend at least 1ft. from bored holes.

Machines with outriggers must have the foundation extend at least 1 ft. from the outrigger leveling pads.

**MACHINE FOOTPRINT: VF-8**

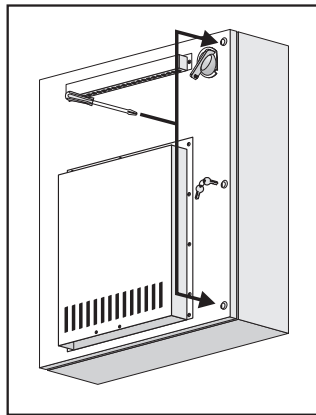
**MACHINE FOOTPRINT: VF-9**

**MACHINE FOOTPRINT: VF-11**

**INITIAL SETUP****WARNING!**

At this point, there should be NO electrical connection to the machine. The electrical panel should be closed and the three latches on the door should be secured at all times except during installation and service. At those times, only qualified electricians should have access to the panel. When the main switch is on, there is high voltage throughout the electrical panel (including the circuit boards and logic circuits) and some components operate at high temperatures. Therefore, you must exercise extreme caution when you are working in the panel.

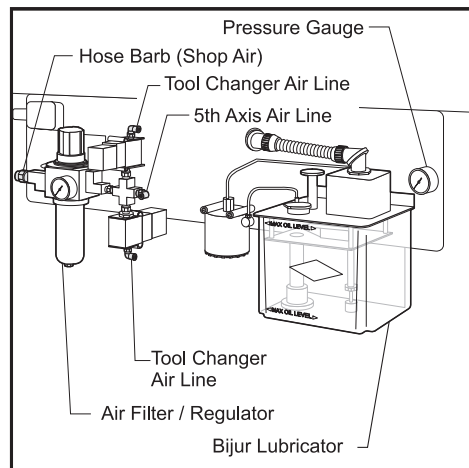
1. Set the main switch at the upper right of the electrical panel on the back of the machine to OFF.



2. Using a screwdriver, unlock the two latches on the panel door, unlock the cabinate with the key, and open the door.
3. The manuals are located at the bottom left of the panel.
4. 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 was mishandled, 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.

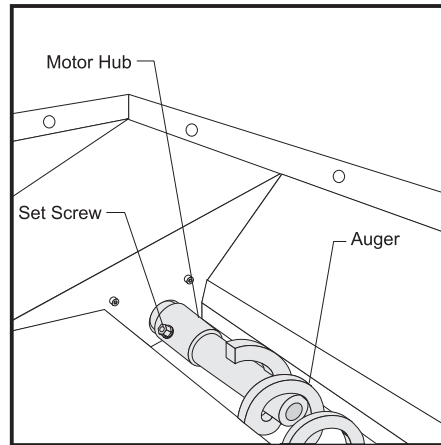


1. When the machine leaves the factory, the air filter is empty, and air lubricator and the lubricator reservoir tank are full. However, they should be checked and serviced if required before compressed air is supplied to the machine.
2. With the pressure off in the air line, connect the air supply to the hose barb next to the air filter/regulator (below the electrical panel). If the fitting supplied is not compatible, simply replace it.
3. Start the compressor, set it between 100 and 150 PSI. Set the regulator on the machine to 85 to 90 PSI.
4. Prime the Bijur lubricator to make sure it is working. To prime the lubrication system, pull up on the handle on top of the reservoir tank. **CAUTION! NEVER** push down on the primer handle! It gradually returns to the down position by itself, and the corresponding pressure increase can be seen on the pressure gauge.

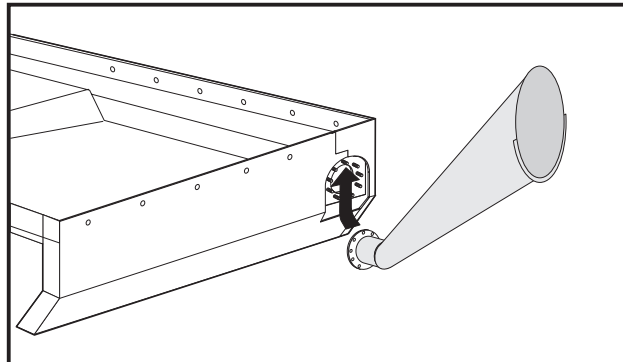
NOTE: Depending on the position of the cam that drives it, the lubrication system may not activate until a few minutes after the machine is started. However, if there is a problem with the system, an alarm will stop the machine.

**OPTIONAL CHIP AUGER INSTALLATION**

1. Unpack the auger and discharge tube.



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



3. Install gasket and slide the discharge tube up and onto studs. Attach the eight nuts with locking washers and tighten uniformly.
4. After machine start-up, check the operation of the auger to ensure the direction of rotation will move the chips toward the discharge tube. If the auger is turning so that the chips are not being moved toward the discharge tube, change the bit switch in PARAM 209 from 1 to 0 or 0 to 1 to establish a new forward direction.

MAINTENANCE

During normal operation, most chips are discharged from the machine at the discharge tube. However, very small chips may flow through the drain and collect in the coolant tank strainer and pan coolant drain (under the pan). To prevent drain blockage, clean this trap regularly. Should the drain become clogged and cause coolant to collect in the machine's pan, stop the machine, loosen the chips blocking the drain, and allow the coolant to drain. Empty the coolant tank strainer, then resume operation.

Note: Auger and discharge tube are subject to wear. Abrasive swarf, hard steel chips and continuous use will accelerate this wear



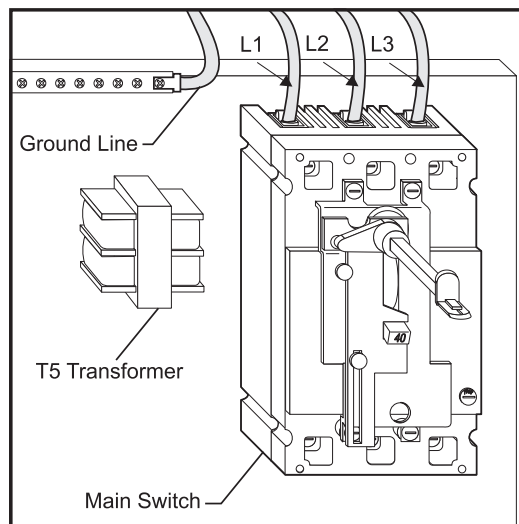
ELECTRICAL CONNECTIONS

NOTE: The machine must have air pressure at the air gauge, or a "Low Air Pressure" alarm will be present on power up.

CAUTION! Working with the electrical services required for the VMC can be extremely hazardous. The electrical power must be off and steps must be taken to ensure that it will not be turned on while you are working with it. In most cases this means turning off a circuit breaker in a panel and then locking the panel door. However, if your connection is different or you are not sure how to do this, check with the appropriate personnel in your organization or otherwise obtain the necessary help **BEFORE** you continue.

WARNING!

THE ELECTRICAL PANEL SHOULD BE CLOSED AND THE THREE LATCHES ON THE DOOR SHOULD BE SECURED AT ALL TIMES EXCEPT DURING INSTALLATION AND SERVICE. AT THOSE TIMES, ONLY QUALIFIED ELECTRICIANS SHOULD HAVE ACCESS TO THE PANEL. WHEN THE MAIN CIRCUIT BREAKER IS ON, THERE IS HIGH VOLTAGE THROUGHOUT THE ELECTRICAL PANEL (INCLUDING THE CIRCUIT BOARDS AND LOGIC CIRCUITS) AND SOME COMPONENTS OPERATE AT HIGH TEMPERATURES. THEREFORE, EXTREME CAUTION IS REQUIRED.



1. Hook up the three power lines to the terminals on top of the main switch at upper right of electrical panel and the separate ground line to the ground bus to the left of the terminals.

WARNING!

THROUGH THE SPINDLE COOLANT (TSC) PUMP IS A THREE PHASE PUMP AND MUST BE PHASED CORRECTLY! IMPROPER PHASING WILL CAUSE DAMAGE TO THE TSC PUMP AND VOID THE WARRANTY. REFER TO THE TSC START UP SECTION.

THE 50 TAPER SIDE MOUNT TOOL CHANGER HAS A THREE PHASE MOTOR WHICH MUST BE PHASED CORRECTLY. CHECK DIRECTION OF CAROUSEL ROTATION TO VERIFY PROPER PHASING.



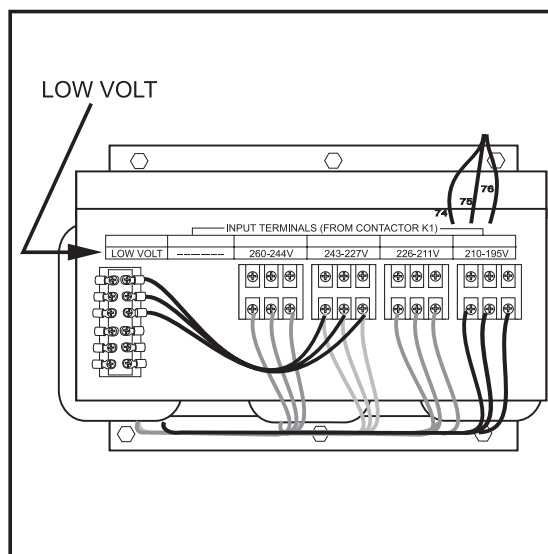
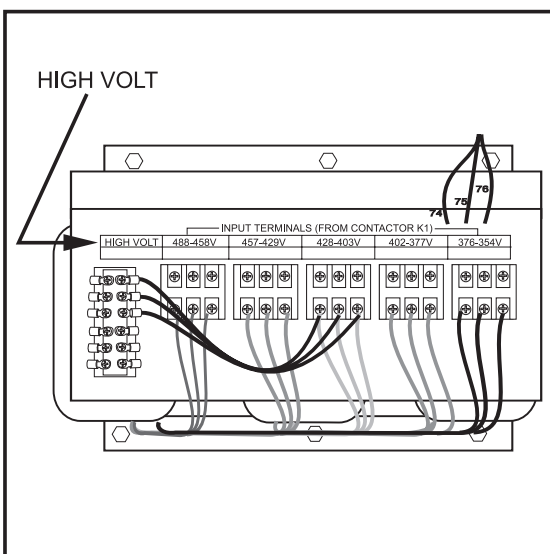
NOTE: Make sure that the service wires actually go into the terminal-block clamps. (It is easy to miss the clamp and tighten the screw. The connection looks fine but the machine runs intermittently or has other problems, such as servo overloads.) To check, simply pull on the wires after the screws are tightened.

2. After the line voltage is connected to the machine, make sure that main circuit breaker (at top-right of rear cabinet) is OFF (rotate the shaft that connects to the breaker counterclockwise until it snaps OFF). Turn ON the power at the source. Using an accurate digital voltmeter and appropriate safety procedures, measure the voltage between all three pair phases at the main circuit breaker and write down the readings. The voltage must be between 195 and 260 volts (360 and 480 volts for high voltage option).

NOTE: Wide voltage fluctuations are common in many industrial areas; you need to know the minimum and maximum voltage which will be supplied to the machine while it is in operation. U.S. National Electrical Code specifies that machines should operate with a variation of +5% to -5% around an average supply voltage. If problems with the line voltage occur, or low line voltage is suspected, an external transformer may be required. If you suspect voltage problems, the voltage should be checked every hour or two during a typical day to make sure that it does not fluctuate more than +5% or -5% from an average.

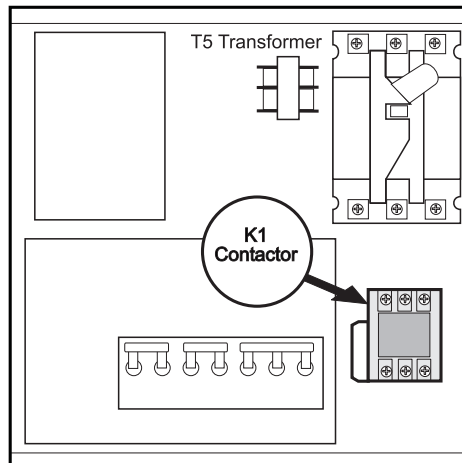
CAUTION! Make sure that the main breaker is set to OFF and the power is off at your supply panel BEFORE you change the transformer connections. Make sure that all three black wires are moved to the correct terminal block and that they are tight.

3. Check the connections on the transformer at the bottom-right corner of the rear cabinet. The three black wires labeled **74**, **75**, and **76** must be moved to the terminal block triple which corresponds to the average voltage measured in **step 2** above. There are four positions for the input power for the 260 volt transformer and five positions for the 480 volt transformer. The input voltage range for each terminal is as follows:





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.



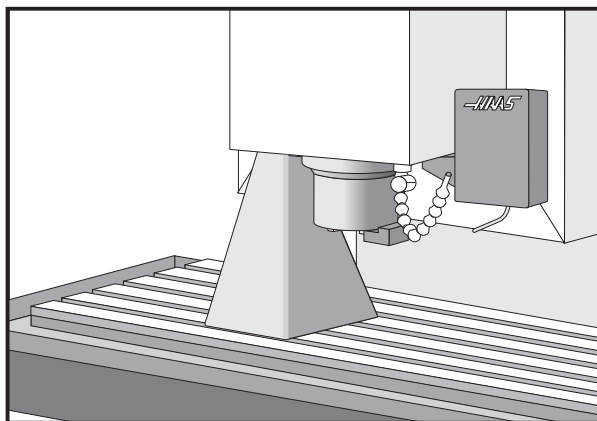
5. After the power is on, measure the voltage across the upper terminals on the contactor K1 (located below the main circuit breaker). It should be the same as the measurements where the input power connects to the main breaker. If there are any problems, check the wiring.
6. Apply power to the control by pressing the Power-On switch on the front panel. Check the high voltage buss on the Vector Drive (pin 2 with respect to pin 3 on the terminal bus at the bottom of the drive). It must be between 310 and 360 volts. If the voltage is outside these limits, turn off the power and recheck steps 2 and 3. If the voltage is still outside these limits, call the factory. Next, check the DC voltage displayed in the second page of the Diagnostic data on the CRT. It is labeled DC BUS. Verify that the displayed voltage matches the voltage measured at pins 2 and 3 of the Vector Drive +/- 7 VDC.
7. Turn off the power (rotate the shaft that engages the handle on the panel door counterclockwise until it snaps into the OFF position). Also, set the main switch handle on the panel door to OFF. (Both the handle and the switch must be set to OFF before the door can be closed). Close the door, lock the latches, and turn the power back on.
8. Remove the key from the control cabinet and give it to the shop manager.

Transformer (t5) supplies 24 VAC to K1 contactor coil. this transformer has a multi-tap wiring at the primary side 200/240V (360/400/480V version for high volt machines).

Note: Choose the correct tap according to the incoming power to the CNC machine.

**MACHINE START UP**

1. On the **VF-0/1/2/3/4**: before POWERING UP machine, remove the tie wrap that holds the flex tube in place.



2. Loosen the four SHCS (three screws are in the table, and one is in the spindle head) holding the shipping bracket under the spindle head, and remove the bracket.
3. With the main switch on the electrical panel set to ON, press and release POWER ON at the upper left of the control panel. You will hear a click in the back of the machine and the fans will energize. (If you don't hear these sounds, the machine is not getting power and, with all necessary safety precautions, you should check the connections to the electrical panel.) After a few seconds, the display will appear on the screen.

CAUTION! DO NOT press POWER UP/RESTART on the control panel while the shipping bracket is under the spindle. Also, do not touch the X, Y, or Z buttons or the jog handle while the shipping bracket is located under the spindle.

4. Press and release SETNG GRAPH. Then page down to the last page (press and release PAGE DOWN several times). Cursor to Setting 53, JOG W/O ZERO RETURN (with the cursor **down** key). Press and release the cursor **right** key and then press and release the WRITE key at the extreme lower right of the control panel to turn this setting on. Turning on JOG W/O ZERO RETURN bypasses the zero return interlock.

NOTE: This setting, like many others, resets to OFF when the machine is powered up. This prevents the machine from operating until a zero return has been executed — the machine control cannot determine position until it has been set by a zero return routine. For this reason, it is important that you execute a zero return immediately each time the machine is started for normal operation **BUT NOT FOR THIS START-UP ROUTINE.**

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

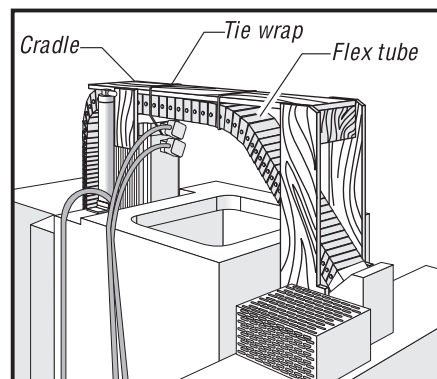
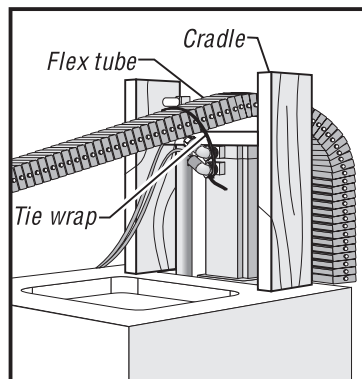


NOTE: If any alarms are present and cannot be cleared with the RESET button, press and release the ALARM MESSAGES button for more information on the alarms. If you are unable to clear the alarms, write down the alarm numbers and call the factory.

6. Press and release the HANDLE JOG button and check the screen for the "JOGGING Z AXIS HANDLE .001" message. Verify that the head will travel SLOWLY (not more than 0.001 inch per impulse — the ".001" part of the Z-axis message). If the message does not read .001, press and release the .001 button next to the HANDLE JOG button.

NOTE: The upper numbers on the buttons next to HANDLE JOG are for the jog handle use, and the lower numbers the jog speed in inches per minute when using the JOG buttons on the keypad. If the "JOGGING __" message shows the X- or Y-axis instead of Z, press and release the +Z button.

7. For the VF-0/1/2/3/4, jog the Z-axis to the top of its travel, and remove the flex tube cradle (as shown).



8. Once you are certain that the Z-axis is working correctly (that it operates smoothly and there are no strange noises, etc.), make sure that all alarms are clear — check for the "ALARM" message at the lower right of the screen. Next, close the doors and press and release the ZERO RETURN button followed by the AUTO ALL AXES button. The Z-axis moves up slowly. Then, after it has reached its home position, the X- and Y-axes move to their home positions.

CAUTION! If you hear any strange noises, hit the EMERGENCY STOP button immediately and call the factory.

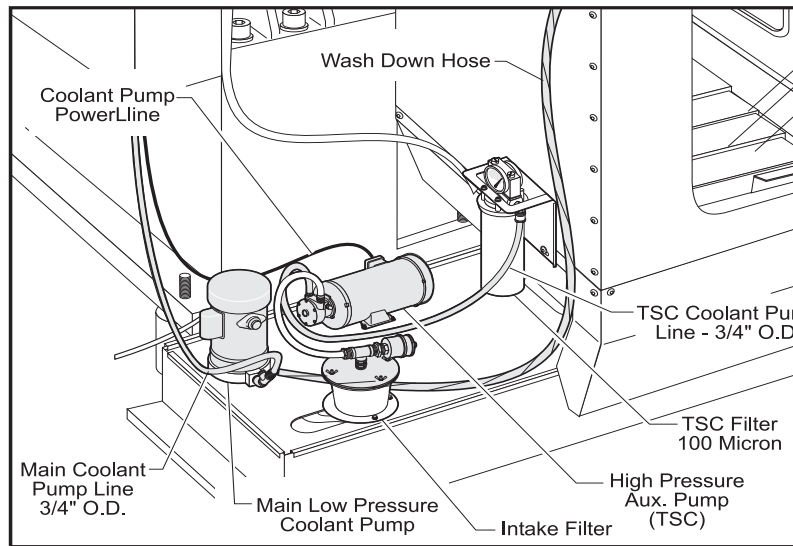
IMPORTANT!! To verify correct hydraulic counterbalance pressure, jog the head to the top of its travel and check pressure with those printed below and on the tank.

	VF-0/1/2/6/7/8/10	VF-6/7/10 w/50T Spindle	VF-3/4	VF-50 w/40T	VF-50 w/50T
Machine at top of travel	750 psi	1150 psi	1150 psi	950psi	1100psi

**COOLANT TANK INSTALLATION**

1. Position the coolant tank under the left side of the machine as shown.
2. Connect the main coolant line (3/4" O.D.) to the main pump.
3. Connect the main pump power line to the outlet on the right side of the electrical panel.
4. If machine includes Through the Spindle Coolant option, attach the 3/4" O.D. coolant line to the TSC pump.

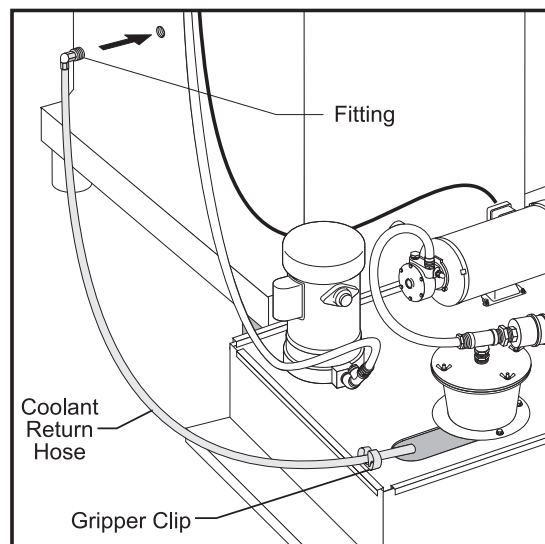
NOTE: The TSC power line is hard wired to the main pump power line.



5. Fill coolant tank with coolant (approximately 40 gallons). VF-6 through 10 (approx. 80 gallons)

NOTE: Before operating the coolant system, ensure the drain is positioned half way over tank strainer.

6. Route the coolant return hose as shown.

**THROUGH THE SPINDLE COOLANT SYSTEM STARTUP****WARNING!**

THROUGH THE SPINDLE COOLANT (TSC) PUMP IS A THREE PHASE PUMP AND MUST BE PHASED CORRECTLY! IMPROPER PHASING WILL CAUSE DAMAGE TO THE TSC PUMP AND VOID THE WARRANTY.

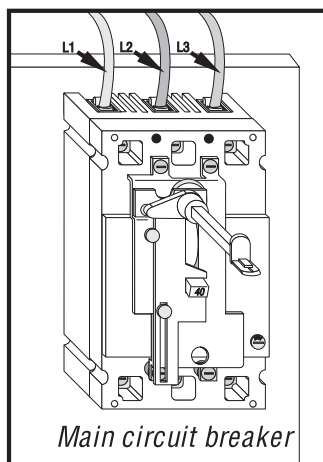
1. Fill the coolant tank with coolant and connect all hoses and power cords. Zero Return the machine.
2. Switch to MDI. Turn on TSC by pressing the AUX CLNT button on the control panel. Quickly go to the back of the machine and check if the TSC pump motor is turning. Check if the pump is pulling coolant up the clear intake hose. If it is, the machine is properly phased. If it is not pumping coolant, immediately stop the pump by pressing the RESET button. Check that the intake hose connection is tight. The pump cannot prime itself if the intake hose is loose. If the motor is not turning, check that the power cords are connected and the circuit breaker inside the control box is on.

The TSC pump will not pump if it is rotating backwards. The motor rotation should be counterclockwise when viewed from the fan end. If the motor is rotating in the correct direction, but the pump is not pumping, call your Haas Service Technician.

Caution: Running The TSC pump dry for more than 60 seconds can cause serious damage to the pump.



3. If the machine is improperly phased perform the following procedure:
 - a. Make sure there is no power at the input side (top) of the main circuit breaker.
MEASURE THE VOLTAGE!
 - b. Exchange any two wires at the input side (top) of the main circuit breaker as shown.



- c. Close the control box.
 - d. Return to Step 1 and test for proper phasing.



OPTIONAL AUXILIARY FILTER FOR TSCHP SYSTEM

APPLICATION

This Auxiliary Filter is used to protect the high pressure TSC pump from particle damage. It is recommended for customers doing medium to high production machining of cast aluminum, cast iron or titanium. It may also be useful for customers who perform high speed milling operations and produce small powder-like chips.

The filter unit consists of a #2 filter bag, filter housing, pipe fittings and hoses. The large size of the filter bag will extend the time interval between bag changes. To reduce wear and prolong the filter system, a small plastic hose is connected between the filter housing and the primary coolant pump to prime the system with coolant. The filter is automatically topped off whenever P-Cool is used. The amount of flow through the priming hose has an unnoticeable effect on normal coolant operation.

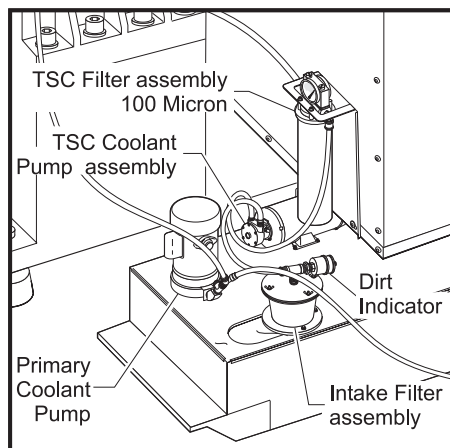


Figure 1.0 Stock Filtration Setup

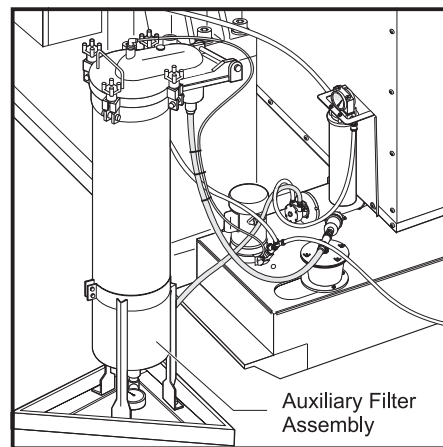


Figure 2.0 Filtration Retrofit

INSTALLATION

1. Place the filter assembly approximately six inches behind the coolant tank, for tanks that pull out from the side or for Horizontal machines. For large vertical machines, place the filter one foot behind the enclosure with the inlet turned to the left so the hinged lid does not strike the enclosure when it is opened for filter changes. For tanks that pull out from the rear of the machine, place the Auxiliary Filter beside the coolant tank next to the enclosure.
2. Secure the filter unit to the floor using 1/4" concrete anchors. (not provided)
3. Using a pipe wrench, turn the tee on top of the intake filter clockwise 1/8 turn so the hose is between the wing nuts. Remove the wing nuts from the filter and pull off the cover. Remove the filter element and re-install the cover with the hose pointing out toward the edge of the coolant tank.



4. For large vertical machines, remove the original hose from the intake filter and replace with the longer hose included in the kit. Secure it to the hose barb using the supplied hose clamp. The hose may be trimmed shorter if desired for appearance. Attach the hose from the intake filter to the inlet of the auxiliary filter.
5. Disconnect the 1/2" check valve from the 45° elbow at the base of the normal coolant pump. Install the 1/2" street-tee from the kit between the 45° elbow and the check valve. Install the pipe reducer from the kit into the side outlet of the street-tee. Install the 1/4" push-in elbow into the pipe reducer. (See figure 3.0).

Note: The cap of the check valve should point upward for the check valve to function properly.

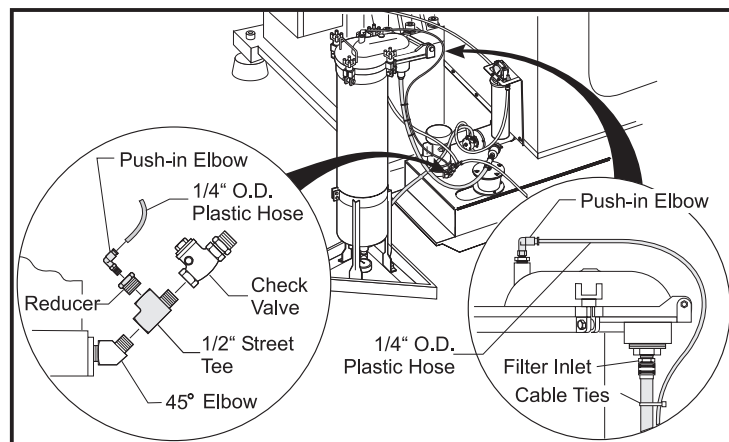


Figure 3.0 Auxiliary Filter Priming System

6. Insert the 1/4" OD plastic hose into the push-in elbow on the pump. Route the hose along the intake filter hose and around the hinge of the auxiliary filter. Trim the plastic hose to length and insert it into the push-in elbow at the top of the filter. Secure the plastic hose to the inlet hose with the supplied cable ties. (See figure 3.0).
7. Attach the hose from the bottom of the auxiliary filter to the inlet of the TSC gear pump.
8. Check that the filter lid is securely closed. Using a wrench handle or metal bar, tighten the two rear bar nuts first and then the front pair. Torque the bar nuts according to the manufacturer's recommendations. (approximately 30-50 ft-lbs)



9. Run the primary coolant system for four minutes to prime the bag filter housing before using the TSC system.

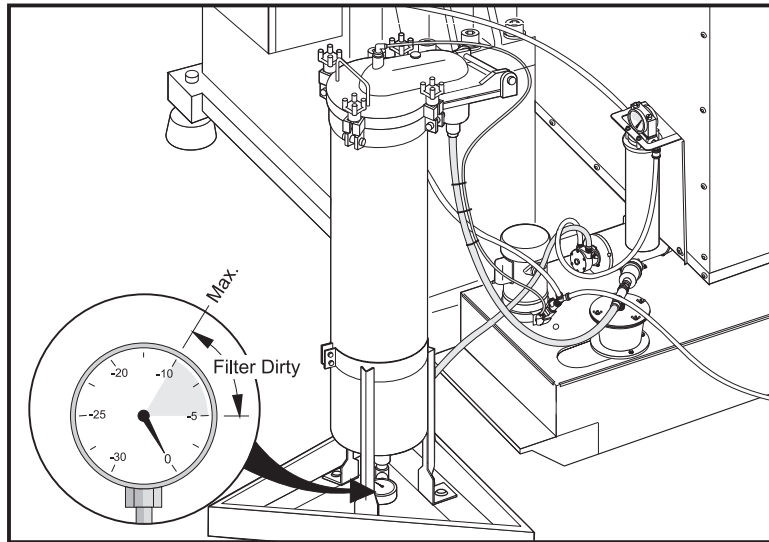


Figure 4.0 Filter Gauge Indicator

REPLACEMENT FILTER BAGS

Change the filter bag when the filter gauge indicator displays a vacuum level of -5 inHg or more. Do not allow the suction to exceed -10 inHg or pump damage may occur (refer to figure 4.0). HAAS recommends using 25 micron rated filter bags (one is provided with the unit). Replacement bags can be purchased from local filter suppliers or from HAAS (Part No. 93-9130). (Finer micron ratings can be used if desired)

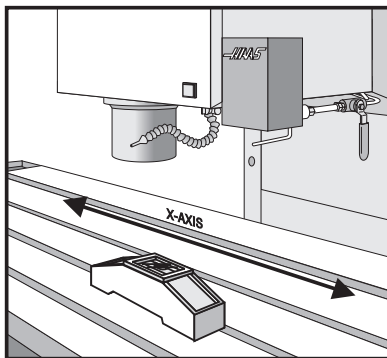


LEVELING THE VMC

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

Leveling is done in two steps: **rough leveling** to ensure the machine is level for coolant and oil drainage, and **fine leveling** for axes' geometry. Finally, the spindle sweep is checked. Leveling is done without removing any covers.

NOTE: Many factors can affect a machine's ability to remain level — the rigidity of the floor, the stability of the support under the floor, trains or trucks passing nearby, seismic activity, and so on. Therefore, until your experience shows how often re-leveling is required, you should check the machine's level frequently after it is installed.

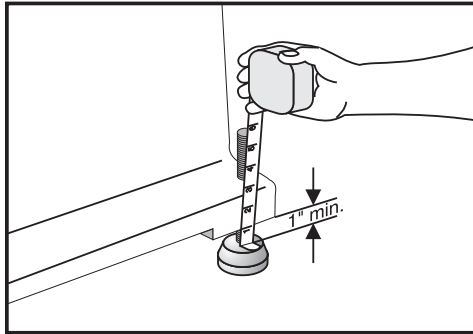


Use a precision bubble level with each division equal to **0.0005** inch per **10** inches, or **.05** mm per meter, or **10** seconds per division. Before starting, check the accuracy of your level. Set it on the table on the X-axis and record the reading. Then turn it **180°** and the reading should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

ROUGH LEVELING

NOTE: Be sure to have removed the plastic bolt covers from the retaining bolts before continuing.

NOTE: For the VF-0/1/2, it may be necessary to pull the coolant tank toward the back of the machine to access the leveling screws.



1. Screw the four leveling screws at the corners through the base 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).
2. Install the two center leveling screws, ensuring that they do not touch the floor.
3. Use HANDLE JOG set for **0.01** on the X- and Y- axes for the leveling procedure. (See the "Introduction" section for details on selecting jog rate and axis.) This provides a good rate of travel as you manually move the table.
4. Using the jog handle, center the table under the spindle. You do not need to move the table while rough-leveling the machine.
5. Place the level parallel to the X-axis (side-to-side) on the table and observe the bubble. If the bubble is centered, the table is level on this axis. If the bubble is off to the left of the level, it means that the left side of the table is high. And, conversely, if the bubble is off to the right, it means that the right side of the table is high.

NOTE: Each time you read the level, make sure that the bubble has steadied before you take the reading.

6. Turn the screws on the low side of the machine clockwise (screw them in) a little at a time and check the level until the bubble is centered.

NOTE: In most cases it is better to raise a side or corner than it is to lower it — when you lower a machine there is a greater risk of running out of adjustment.

7. Repeat the previous steps with the level on the Y-axis (front-to-back).
8. Continue this process until the machine is level on both axes.

NOTE: If the level is off on both axes, it indicates that one corner of the machine is high or low.



9. As the process continues, the leveling screws are turned in smaller increments — 1/4 turn, 1/8 turn, and smaller. Also, as the machine is leveled, make sure that the tension continues to be equal on the screws at all four corners.

NOTE: The following procedure for fine leveling the machine must be performed exactly as noted to ensure machine will meet all quality standards for machining operations. Failure to follow these guidelines will prevent the machine from being truly leveled and result in poor machining finishes.

FINE LEVELING

10. With the table centered, place the bubble level in the center of the table parallel to the X-axis. Using the jog handle, move the Y-axis, stopping at the front, middle, and back of the travels. The objective is to adjust the level to make the Y-axis guides parallel. The bubble level must indicate the same reading at each position (front, middle, back). Note the movement of the bubble and if the table is at front or back of travel. If the bubble moves, for example, to the right and the table is at the front of the travel, lower the right front corner adjustment screw slightly. Repeat the procedure until you get the bubble steady from front to back. This is the only leveling adjustment that can be done.

The following procedure is simply a check of machine level. If it does not meet specifications, then you must repeat this operation. Do not adjust the middle screws at this point.

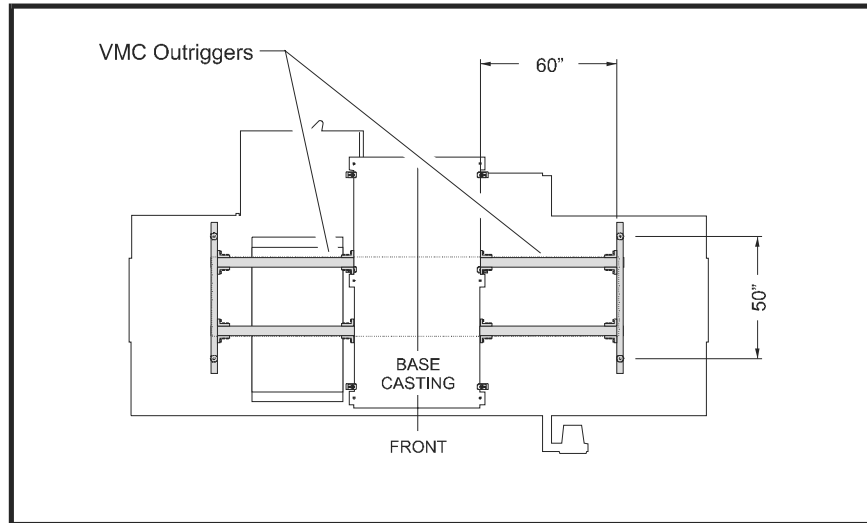
Refer to the Machine Inspection Report that accompanies your machine. Check your results with those of the report under the Table Travel Flatness verification. By duplicating these results exactly, you will obtain the same alignment specifications that were achieved at the factory.

11. Place a **0.0005** test indicator in the spindle and sweep a **10" diameter** circle on the table (See the Machine Inspection Report in the manual for the results of this test at the factory). Grease the dimple in each of the two remaining pads, locate them under the middle leveling screws, and use these screws to compensate for any error. If there is no error, tighten the screws evenly until they contact the pads.



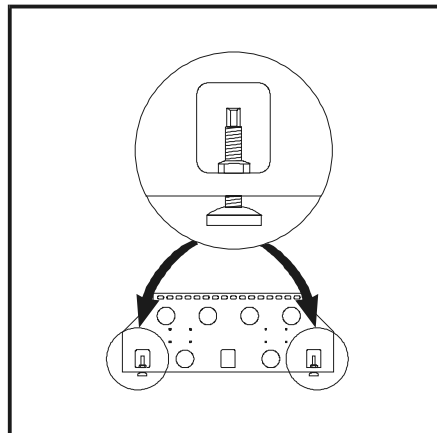
OUTRIGGER LEVELING PROCEDURES

NOTE: The standard VMC leveling procedures must be completed before starting this section.



VMC Outrigger Locations

1. Locate the leveling pads underneath the outrigger leveling screws. Grease the dimples and keep the screws at least 1/4" above the pads.
2. Place a level in the center of the table and parallel to the X-axis.
3. Jog the Y-axis back (toward the column) max travel.
4. Starting from either left or right of the machine, jog the table to the max X-travel and over the rear outrigger leveling screw. **DO NOT MOVE THE LEVEL.**



Outrigger Leveling Screw Locations



5. Lower the leveling screw onto the rear pad. Raise the table to zero the level.
6. Jog the Y-axis forward to max travel and lower the leveling screw onto the front pad. Raise the table to zero the level.
7. Jog the X-axis to max travel over the other outrigger. Jog the Y-axis to the rear (toward column) max travel.
8. Repeat steps 5 and 6.
9. Check level through full X and Y range of travel.

CAUTION! Over tightening the outrigger leveling screws will result in poor machine performance.

SPINDLE BREAK-IN

Caution! Spindle break-in program must be run before spindle can be run above 1000 RPM. Failure to run this program can result in spindle over heating and failure.

Before running the spindle, a spindle break-in **must** be performed. A program has been supplied with the machine, which will slowly run the spindle up to speed (approx. 2 hrs.). This will purge out any oil which may have settled at the nose of the spindle, due to long idle time. The program is # O02021 Spindle Break-In and will be used for all spindle types and RPM's. For machines equipped with 7,500 RPM or 10,000 RPM spindle run the above programs with spindle speed override at 100%. For machines equipped with 15,000 RPM spindle, run spindle speeds overrides at 150%. For machines equipped with a 50 taper spindle, run spindle speed override at 50%.

If this program was not supplied with the machine, contact your dealer.

The spindle should be checked periodically for spindle temperature rise. If the temperature rises above 150 degrees, start the program from the beginning (refer to service manual for overheating troubleshooting).



PREPARATION FOR SHIPPING THE VMC

NOTE: See the illustrations in the "Installation" and "shipping the Machine Without a Pallet" section for additional information.

IMPORTANT!

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
9/16" hex wrench
1 1/8" wrench

Claw hammer
3/4" wrench

Forklift with the following specifications:

	VF-O/E	VF-OE/EXT	VF-1	VF-2	VF-3	VF-4	VF-5/40
Machine Weight *	7,000	7,900	7,100	8,000	12,500	13,000	14,500
Fork Length	8'	8'	8'	8'	8'	8'	8'

	VF-5/50	VF-6	VF-7	VF-8	VF-9	VF-10	VF-11
Machine Weight *	14,600	2,100	23,000	24,000	25,000	28,000	29,400
Fork Length	8'	8'	8'	8'	8'	8'	8'

* The forklift must be capable of lifting **at least** this weight.

Cleaning

Clean the machine as necessary before preparing it for shipping.



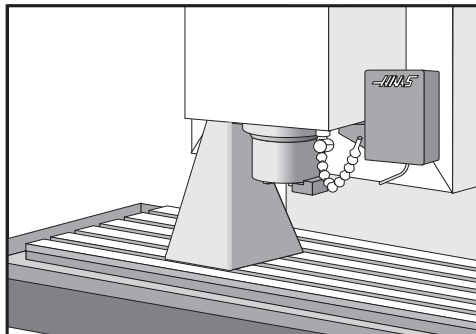
MATERIALS REQUIRED

One crate (including pallet and 40 corner clips)
3/8" washers (18)
6-mil plastic cover
Wire ties
Fiber banding
1/2" washers (4)
16d nails (as required)

3/8" x 3" lag bolts (18)
6-mil plastic sheet
Coolant box (cardboard)
Foam pads
1/2-13 x 10" bolts (4)
1/2-13 Nylock nuts (4)

LOCK UP THE MACHINE FOR SHIPPING

1. Remove any tools from the tool carousel and spindle.
2. Clean the interior of the enclosure, including the windows, as necessary. After cleaning, spray all metal parts around the way covers, table, spindle (including up into the spindle), with rust preventive oil. (See the cautionary notes on cleaning the machine in the "Maintenance" section.)
3. Use the jog handle to center the table under the spindle.
NEVER PUT ANY SUPPORT ON THE SPINDLE!! THIS WILL DESTROY THE SPINDLE BEARING!!!

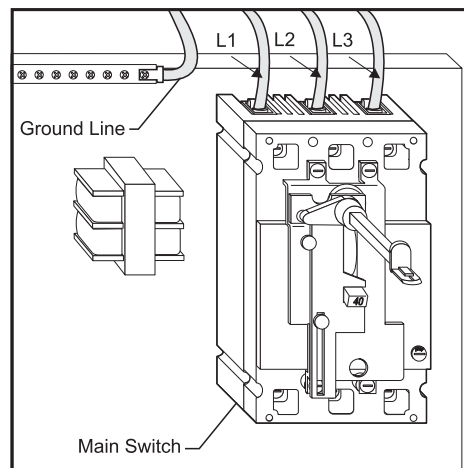


4. Set the shipping bracket on the table, then use the JOG HANDLE to bring the head down until it rests on the bracket. Attach the shipping bracket to the table and head with the four SHCS.
5. Press EMERGENCY STOP and the machine is locked up for shipping.
6. Power down (press the POWER OFF button at upper left of control).
7. Remove the discharge tube. Remove the chip conveyor from the motor hub. Pack them both for shipping.
8. Place all loose items in the drain bucket. These items include the control cabinet key, washdown kit, brush, leveling feet, and the leveling screws.

**DISCONNECT SERVICES****WARNING!**

WORKING WITH THE ELECTRICAL AND AIR SERVICES REQUIRED FOR VMC CAN BE EXTREMELY HAZARDOUS. MAKE SURE THAT PRESSURE HAS BEEN REMOVED FROM THE AIR LINE BEFORE YOU DISCONNECT IT FROM THE MACHINE. SIMILARLY, THE ELECTRICAL POWER MUST BE OFF AND STEPS MUST BE TAKEN TO ENSURE THAT IT WILL NOT BE TURNED BACK ON WHILE YOU ARE WORKING WITH IT. IN MOST CASES THIS MEANS TURNING OFF A CIRCUIT BREAKER IN A PANEL AND THEN LOCKING THE PANEL DOOR. HOWEVER, IF YOUR CONNECTION IS DIFFERENT OR YOU ARE NOT SURE HOW TO DO THIS, CHECK WITH THE APPROPRIATE PERSONNEL IN YOUR ORGANIZATION OR OTHERWISE OBTAIN THE NECESSARY HELP BEFORE YOU CONTINUE.

1. Disconnect the air supply line at the pressure regulator (below the electrical panel on the back of the machine).
2. Turn off the electrical power to the machine at its source (see the caution at the beginning of this section).
3. Set the main switch on the machine (upper right of the electrical panel on the back of the machine) to OFF.
4. Release the three latches along the edge of the panel door and open the door.



5. Using a screwdriver to release the terminals, disconnect the three electrical leads to the terminals behind the main switch and the adjacent ground-bus connection. Pull the wires out of the top of the panel box and secure them safely.
6. Remove the air-filter screen mounted toward the bottom on the outside of the panel door (lift it up and then pull the bottom edge out). Clean it with compressed air if necessary.
7. Place the air-filter screen, along with the manuals and other documentation, inside the panel at the bottom left.
8. Set the main switch to OFF (rotate the shaft that engages the handle on the panel door counter-clockwise as far as it will go and then let it snap into the OFF position). Also, set the main switch handle on the panel door to OFF. (Both the handle and the switch must be set to OFF before the door can be closed). Close the door and lock the latches.

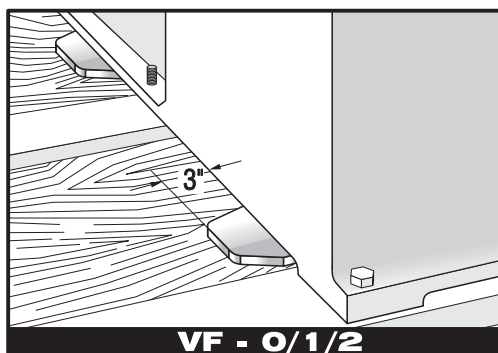
**SECURE THE FRONT**

CAUTION! No heavy or abrasive ties, chains, wraps, or bindings should be placed in contact with painted or plastic parts of the machine. Damage caused by ignoring this caution is NOT covered by the warranty.

1. Make sure that the machine is locked up and remove any loose material from the table area.
2. Place a pad between the doors, pull them closed, and secure them with two wire ties through the handles.
3. Wrap some padded packing material around the pendant (control box) and secure with some heavy-duty tape. Also, place a couple inches thickness of foam or other packing material on the enclosure behind the control panel.
4. For the **VF-3/4**, remove the support bolt from the control arm, and swing the arm back and into shipping position. Put the shipping brace into place and secure with two bolts.

MOVE ONTO THE PALLET

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

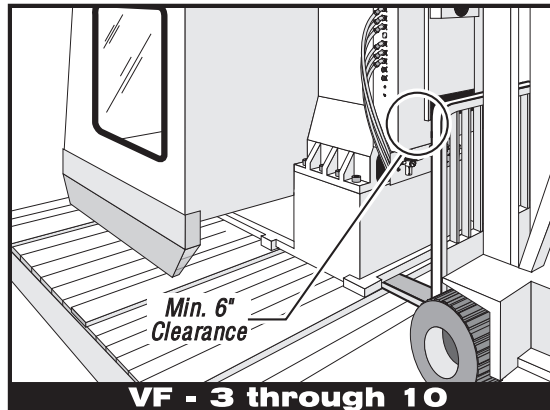


IMPORTANT! Keep in mind when moving the VMC that much of its weight is concentrated in the column at the back. Therefore, when lifting a **VF-0/1/2** from the side, it is important that the forks of the forklift be positioned as close to the back of the machine as possible without being on the pads.

CAUTION! Do not lift the machine any farther than necessary off the floor when moving it, and move as slowly and cautiously as possible. Dropping the machine, even from a height of a few inches, can cause injury, result in expensive repairs, and void the warranty.

**WARNING!**

FOR THE VF-0/1/2: THE ONLY ACCEPTABLE WAY TO MOVE A VF-0/1/2 IS TO PICK IT UP FROM THE SIDE WITH A FORKLIFT. IMPORTANT! FOLLOW THE MACHINE WEIGHT AND FORK LENGTH SPECIFICATIONS DESCRIBED EARLIER. THE FORKS MUST BE SET AS FAR APART AS POSSIBLE WITHOUT BEING ON THE PADS. THE FORKS MUST BE POSITIONED ALL THE WAY TO THE BACK OF THE VMC AND THEY MUST EXTEND AT LEAST 3" PAST THE FAR SIDE OF THE MACHINE BASE. ALSO, THERE MUST BE ABOUT APPROXIMATELY 6" CLEARANCE BETWEEN THE FORKLIFT AND THE SIDE OF THE MACHINE.



For the VF-3 through 10: The VF-3 through 10 must be lifted from the **BACK** of the machine with a forklift. **IMPORTANT!** Follow the machine weight and fork length specifications described earlier. The forks must be set as far apart as possible without being on the pads. Also, there must be about approximately 6" clearance between the forklift and the back of the machine.

Attempting to move the machine any other way may void the warranty.

CAUTION! Use caution when lifting the machine with a forklift not to damage the sheet metal aprons with the forks.

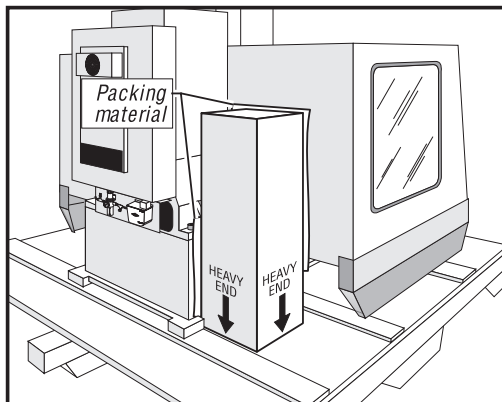
1. Lift the machine a few inches off the floor.
2. Cover the pallet with a 6-mil thick contiguous plastic sheet.

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

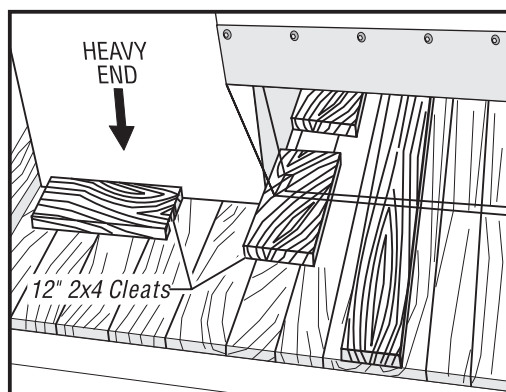
3. Move the machine over the pallet and align the leveling screws with the holes in the pallet. Note that the pallet center support is offset to one side. When the machine is placed on the pallet, the rear of the machine must be on this side of the pallet.



4. Remove the leveling screws at the four corners of the base. Insert the 3/4-10 mounting bolts through the leveling screw holes. Tighten the bolts to the coupler mounted in the pallet.



5. Drain and clean the coolant system. Pack it in the box it came in and slip the box, heavy end down, into the space on the left side of the machine between the panel, at the back of the machine, and the enclosure.

**WARNING!**

THE BOX MUST BE POSITIONED WITH THE HEAVY END DOWN. THEN SLIP PACKING MATERIAL BETWEEN THE BOX AND THE MACHINE AT ALL POINTS WHERE THERE IS CONTACT. AND FINALLY, NAIL A 12", TWO-BY-FOUR CLEAT TO THE PALLET WITH 16D NAILS TO HOLD THE BOX IN PLACE.

6. Put the leveling screws, pads, allen wrench, and screwdriver in the tote kit 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 tote kit in place.
7. Clean the drip pan as necessary and slide it under the machine. If the machine is furnished with the drain bucket, place it at the rear of the machine, under the air/lube panel, and tie into place.

**CRATING THE VMC**

1. Cover the machine with a sheet of 6-mil thick plastic. Then carefully roll the edges of the bottom plastic and the cover together and staple them to the pallet. This forms a moisture resistant cocoon around the machine. **HOWEVER**, as noted earlier, the plastic wrapping is not 100% waterproof and the crate must be covered with waterproof tarpaulins any time there is a chance it can be exposed to rain, splashing from wet roads, or other conditions where it could get wet.
2. Set the four sides of the crate around the machine. Note that the front and back of the pallet are shorter than the sides. Therefore, the side panels have more reinforcing strips than the front and back. Also, if you are reusing a crate, make sure that the fork location marks are in the correct location (see earlier comments).
3. Position one of the sides (longer panels) on the edge of the pallet and secure it with five 3/8" lag bolts.
4. Position an adjoining side and secure it in place with a corner clip (use a claw hammer to snap the clip into place). Then secure it with four lag bolts.
5. Repeat the above two steps for the remaining two sides--clip the corner and secure the side. (Remember to use four lag bolts for the short side and five lag bolts for the long side.)
6. Complete clipping the four corners. There should be a total of five clips at each corner.
7. Place the top panel on top of the crate and clip each side. Use five clips per side.
8. If the fork locations are not painted on the sides, add these marks (refer to earlier figure).

CAUTION! The fork positions marked on the sides of the crate are the key to safely moving the crated VMC. If the fork positions are ignored, there is a good chance that the retaining bolts will be sheared off by the forks and also that the machine will tip over when it is picked up.

This completes packing the VMC for shipping. If you have questions, contact your dealer or the Haas Automation factory.



SHIPPING THE MACHINE WITHOUT A PALLET

To ship a VMC a short distance, it may not be necessary to mount it on a pallet and crate if you employ professional machinery movers. Nevertheless, you must make specific preparations for moving it and certain precautions are necessary.

NOTE: We strongly recommend using an air-ride suspension truck to minimize jarring the machine. It is well worth the few extra dollars involved. And make sure to ask specifically for air-ride suspension -- not all riggers have trucks of this type.

IMPORTANT!

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)

Forklift with the following specifications:

	VF-O/E	VF-OE/EXT	VF-1	VF-2	VF-3	VF-4	VF-5/40
Machine Weight *	7,000	7,900	7,100	8,000	12,500	13,000	14,500
Fork Length	8'	8'	8'	8'	8'	8'	8'

	VF-5/50	VF-6	VF-7	VF-8	VF-9	VF-10	VF-11
Machine Weight *	14,600	2,100	23,000	24,000	25,000	28,000	29,400
Fork Length	8'	8'	8'	8'	8'	8'	8'

* The forklift must be capable of lifting **at least** this weight.

MATERIALS REQUIRED

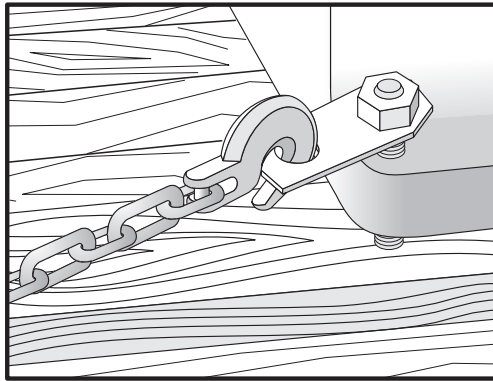
Four metal tie-downs

Four ¾-11 nuts (in addition to the four nuts that come with the machine)

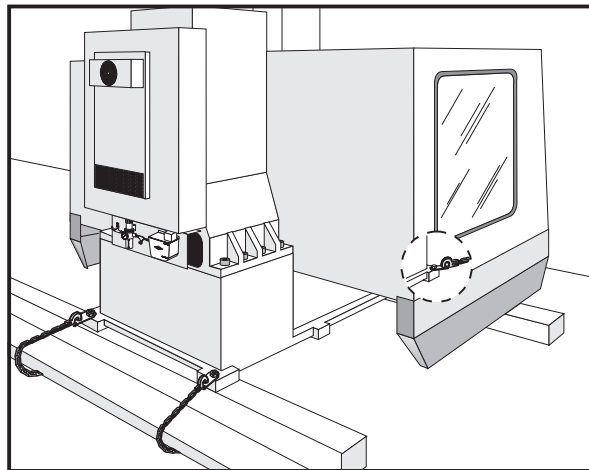
Two 4 x 4's at least 5' long

**PROCEDURE**

1. Prepare the machine as described in the "Preparation for Shipping the VMC" section up to the point of putting it on a pallet. Make sure the spindle is on tool number one. Replace and secure the shipping bracket -- even for the shortest moves. Tie-wrap the doors together and stow the air filter from the control panel door (on the back of the machine) inside the panel.



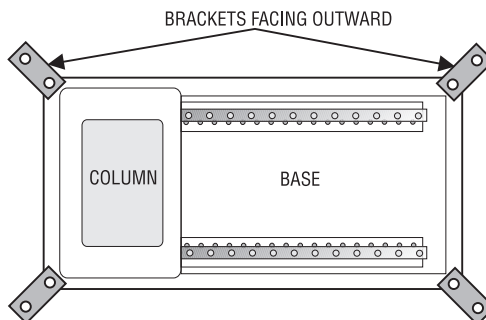
2. To transport the machine without a pallet, you need four metal tie-downs. (Additional tie-downs can be purchased from Haas if you have lost yours.) After raising the machine with a forklift, turn the four corner levelling adjustment screws clockwise, with the 9/16" hex wrench, until they protrude at least 1/2" through the bottom of the machine. Tighten the 3/4-11 nuts until the levelling screws are secured in the casting. Place a metal tie-down on each of the four levelling screws (the small hole in the tie-down slips over the top of the screw and rests on the top of the 3/4-11 nut). Position the tie-downs so they are at 45° angles to the front, back, and sides of the machine and secure them on the screws with additional 3/4-11 nuts.



3. Place two 4 x 4's that are least 5' long on the truck. Lower the VMC onto the truck and align the two pieces of lumber under the leveling screws at the corners of the machine. Remove the forklift and have the rigger secure the machine using the large holes in the four tie-downs as the attachment points for the riggers' chains. When the chains are tightened, the leveling screws dig into the 4 x 4's to help hold the machine in place.



CAUTION! Do not cross the chains.



4. Using the four corner tie-downs, the machine can be secured without dragging chains across its top. It is a fast, easy, and secure transportation method that protects against damage. If there is limited access, wrap perimeter of base above feet with chain tension as necessary. Be sure to explain to the rigger that this is how you want the machine to be moved **BEFORE** he does something else.
5. If you wish to tarp the machine, you should be aware of a few potential trouble spots. If a tarp is simply placed over the machine and tightened down, it can do considerable damage, such as bending the conduit carrier for the headstock, wearing paint off the machine, distorting the enclosure, and damaging the clear plastic parts. If you must use a tarp, make sure that it is supported off the machine by a wooden frame or other secure structural means.
6. If the fork locations are not painted on the sides, refer to the earlier figures and add them.
CAUTION! These fork positions are very important, and if they are ignored, there is a good chance that the retaining bolts will be sheared off by the forks and that the machine will tip over when it is lifted.

This completes packing the VMC for shipping. If you have questions, contact your dealer or the Haas Automation factory.

**CABLE LIFT INSTRUCTIONS**

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



***It is mandatory that preparation
and lifting of the VMC be performed
by PROFESSIONAL RIGGERS and
that this section be used as a
reference guide only.***



LIFT KITS for the HAAS VERTICAL MACHINING CENTERS

VF-O/1/2: (LIFT KIT 30-7900E)	
(1) 5/8-11 x 1 1/16" EYE BOLT	(1) WASHER, SAE, 5/8"
(2) 3/4-10 X 1 3/16 EYE BOLTS	(2) WASHER, FLAT CUT 3/4"
(1) INSTRUCTIONS	
VF- 3/4: (LIFT KIT 30-9900E)	
(2) LIFTING BRACKETS	(2) 3/4 FLAT CUT WASHERS
(2) 1/2-13 x 1 1/2" SHCS	(2) 3/4-10x 1 3/16" EYE BOLT
(6) 1/2" BLACK HARD, WASHER	(1) INSTRUCTIONS
VF-5: (TOOL KIT 30-0321)	
(2) LIFTING BRACKETS	(2) 3/4" FLAT CUT WASHERS
(2) 1/2-13 X 1 1/2" SHCS	(2) 3/4-10 X 1 3/16" EYEBOLT
(6) 1/2" BLACK HARD, WASHER	(2) 5/8 x 1 x 1 1/16" EYEBOLT
(1) INSTRUCTIONS	(2) SPREADER BARS*
VF-6 THRU 10: (TOOL KIT 30-7910E)	
(2) 1 1/8-7 x 2 3/4 EYEBOLT	
(1) INSTRUCTIONS	
(2) SPREADER BARS*	

NOTE: PRIOR TO JAN. 1996 , 1- 8 EYE BOLT, HEX NUT (P/N 46-1696) AND HOLE PLUG (P/N 22-9918) ARE REQUIRED FOR THOSE MACHINES WITH EYEBOLT THROUGH HOLES.

* Spreader bars and spacer blocks are not provided with the lift kits

EYEBOLT WORKLOAD RATING

EYEBOLT SIZE	RATING
1/2-13	2,800 lbs
5/8-11	5,100 lbs
3/4-10	7,000 lbs
1-8	9,000 lbs
1 1/8-7	12,000 lbs

LIFTING PRECAUTIONS

IMPORTANT: The chains attached to the top of the column are the main lifting points and should remain vertical to the floor throughout the lift.

The front chains attached to the base are to help stabilize the machine and should not be used to carry the main load.

When lifting the machine, slowly pull up on the chains until taut, then lift the machine off the floor, ensuring the machine remains balanced.

Before installing the lift kit or lifting the machine, ensure the machine is prepared for shipping as noted in the PREPARATION FOR SHIPPING section in the Installation Manual.



SECTION 1: 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.

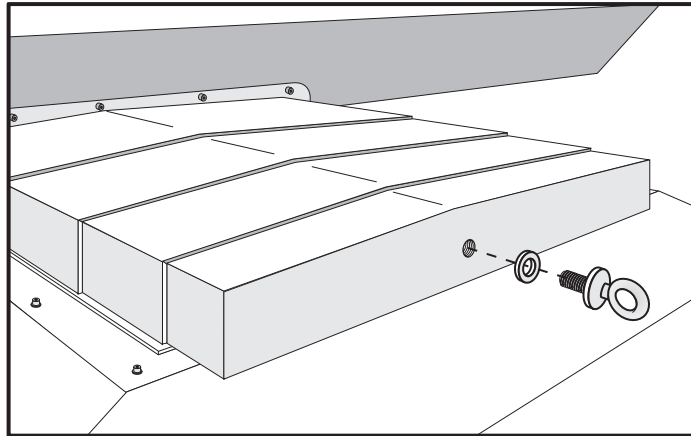


Fig. 1-1 Eyebolt mounting location for VF-0, VF-1, and VF-2.

2. Place a washer on each of the 3/4-10 x 1³/₁₆" eyebolts and mount to the top of the column as shown below. Tighten securely.

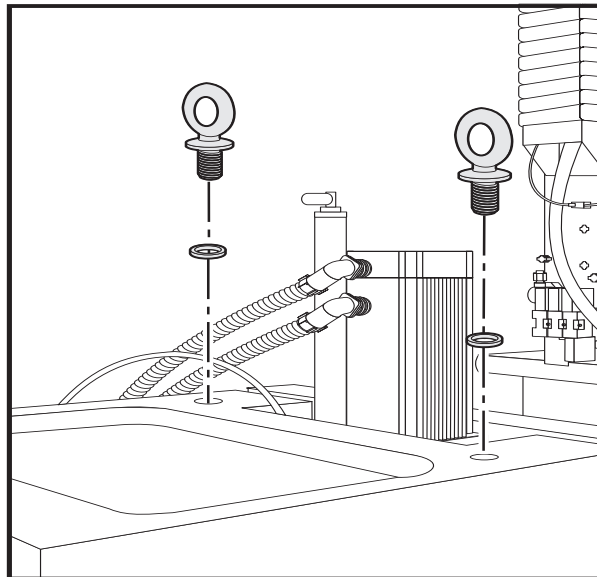


Fig. 1-2 VF-0/1/2 eyebolt locations.

**INSTALLING THE LIFT KIT****VF- 3 / 4**

3. Install the lifting brackets with (2) 1/2"-13 SHCS and (2) 1/2" hard washers, [(2) washers under bracket and (1) above bracket], to the base as shown in figure 1-3. Tighten securely.

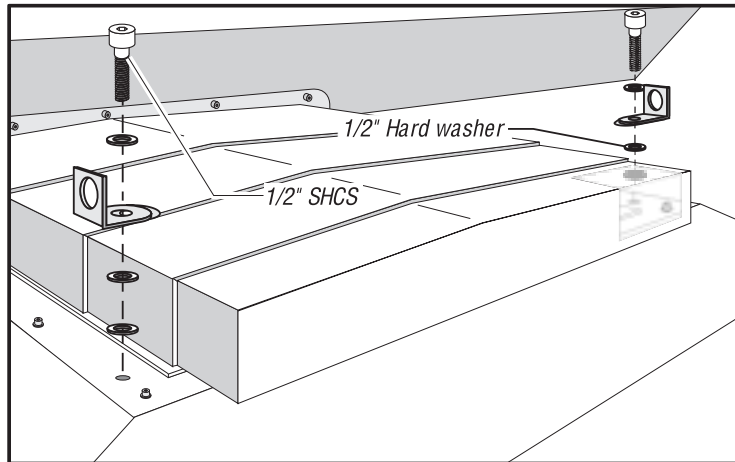


Fig. 1-3 Lifting bracket mounting location for VF-3 and VF-4.

4. Install (2) 3/4-10 x 1 1/4" eyebolts and (2) 3/4" hard washers to the column casting as shown in figure 1-4. Tighten securely.

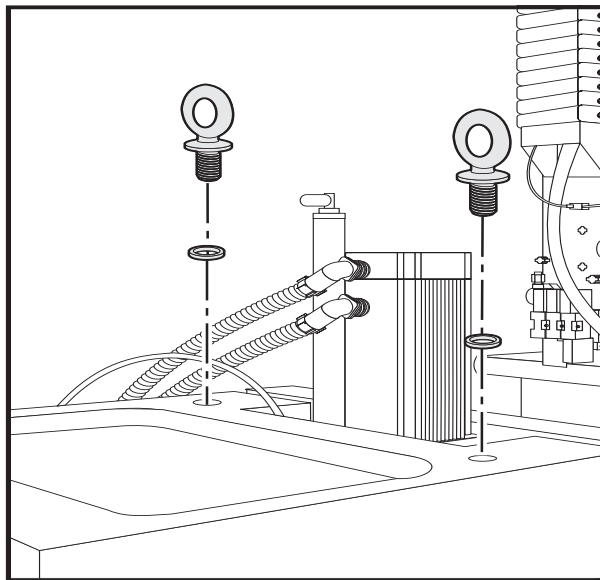


Fig. 1-4 VF-3/VF-4 eyebolt locations.



SINGLE POINT LIFTING

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 sheetmetal enclosure.

A **minimum** chain height requirement to lift the machine without damaging the enclosure and to keep the rear chains perpendicular to the floor. The chain length is measured from the top of the column casting.

IMPORTANT: The chains attached to the top of the column are the main lifting points and should remain vertical to the floor throughout the entire lift.

The front chains attached to the base are to help stabilize the machine and should not carry the main load.

When lifting the machine, slowly pull up on the chains until taut, then lift the machine 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 section of the manual.

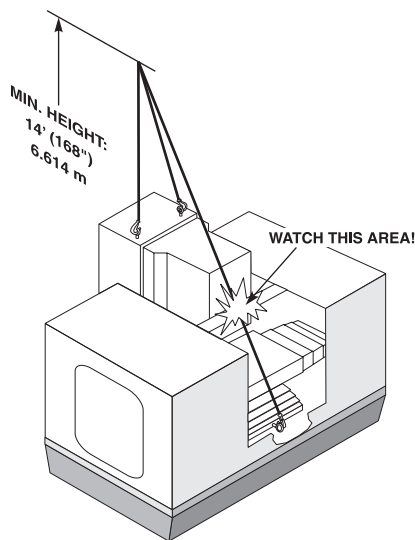


Fig. 1-5 VF-0/1/2 lifting positions.

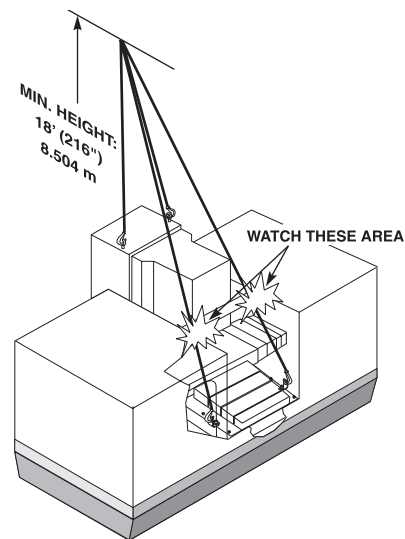


Fig. 1-6 VF-3/4 lifting positions.



SECTION 2: INSTALLING THE LIFT KIT

VF-5 40 / 50 TAPER

1. Install the lifting brackets with (2) 1/2" -13 SHCS and (2) 1/2" hard washers, [(2) washers under bracket and (1) above bracket], to the base as shown in Figure 2-1. Tighten Securely.

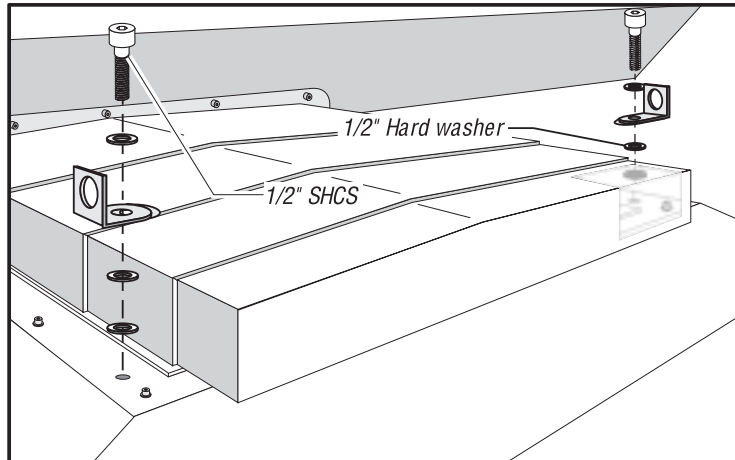


Fig. 2-1 Lifting bracket mounting location for VF-5.

2. Install (2) 5/8 x 1 x 11/16" eyebolts and (2) 3/4" hard washers to the column as shown in Figure 2-2. Tighten securely.

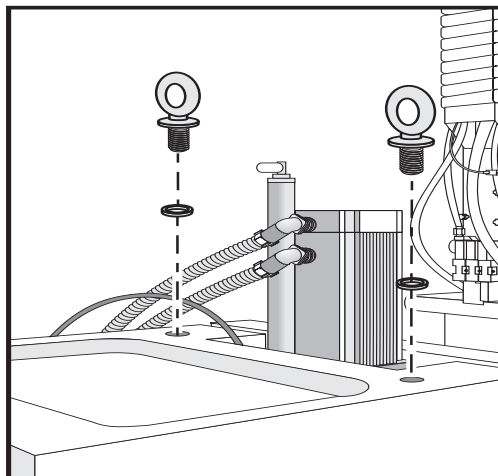


Fig. 2-2 VF-5 eyebolt locations on top of column.



BEAM STYLE LIFTING

VF-5 40 / 50 TAPER

3. Attach the lifting chains to the rigging beam as shown. Use the spreader bars to protect against any possible contact points.

IMPORTANT! The spreader bar above the column is used to keep the chains parallel to the column and evenly distribute the load on the eyebolts.

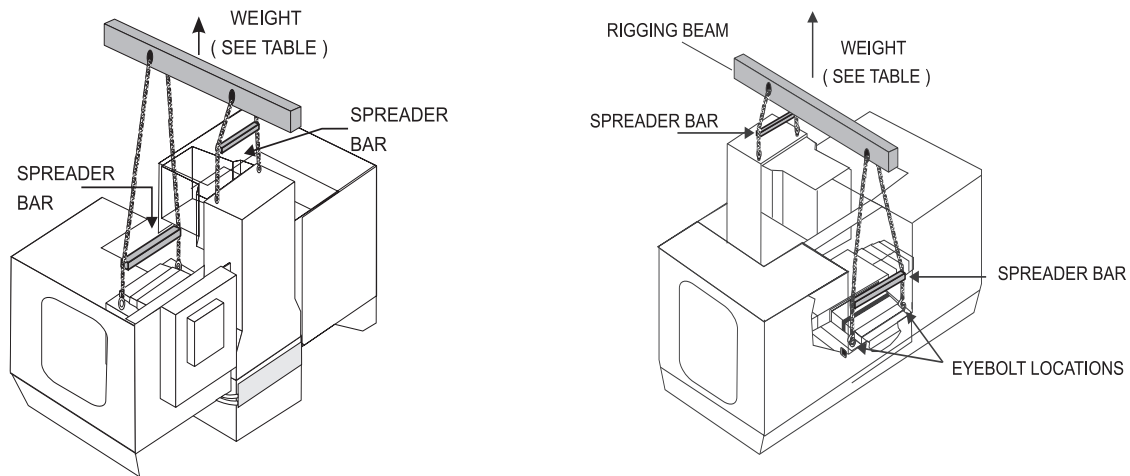


Fig. 2-3 VF-5 lifting positions.



SECTION 3: INSTALLING THE LIFT KIT

VF-6 thru 10

* On machines with eyebolt through holes, remove the eyebolt plugs on the base casting.

1. Install the two 1 $\frac{1}{8}$ - 7 eyebolts to the base casting as shown below and tighten. Eyebolts should be facing towards the front of the machine when tightened.

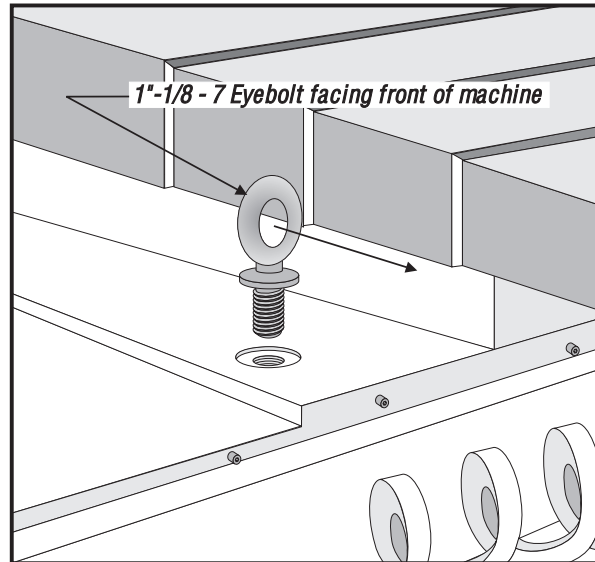


Fig. 3-1 Eyebolt and position access hole for VF-6 thru 10.

2. These specifications are an approximation of the weight distribution and chain length needed to lift the VF-6 thru 10.

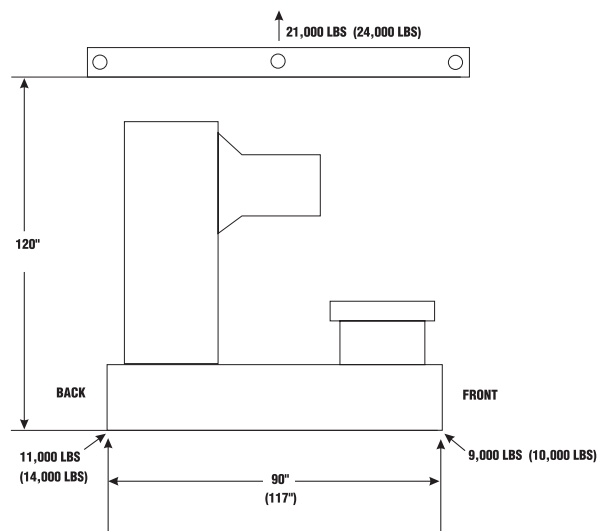


Fig. 3-2 VF- weight specifications.



3. Attach the lifting chains and hooks to the lifting points, with the rigging beam, spreader bar, and spacer block in place. Position the spreader bar so the lifting chains do not come into contact with the waycovers.

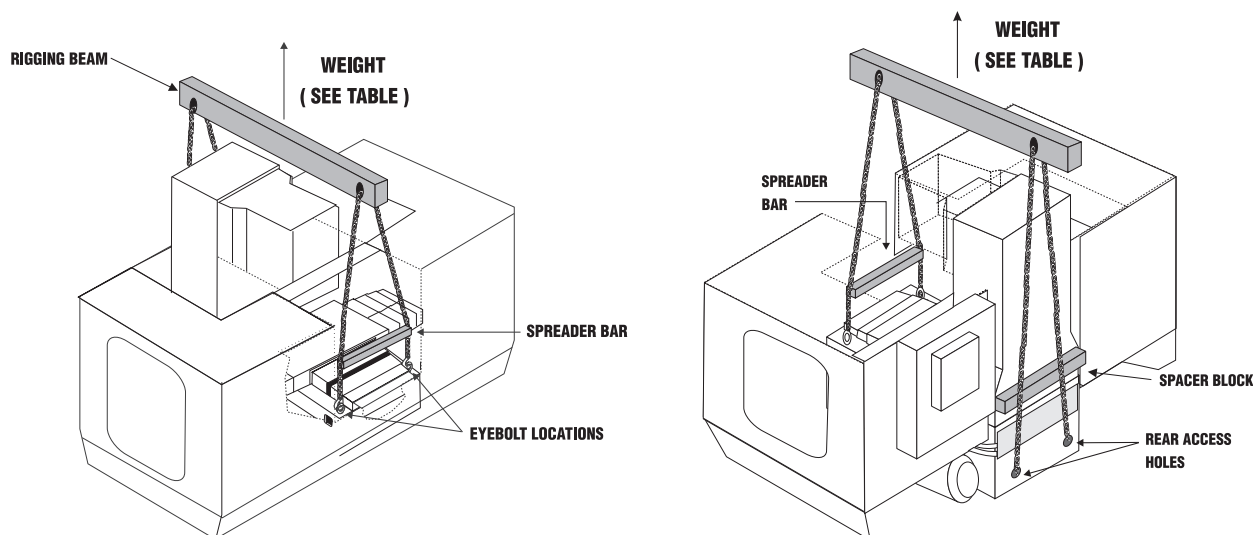


Fig. 3-3 VF-6/8 lifting positions.

4. Use the spacer block to keep the chains from damaging the rear conduit lines.

* On machines with eyebolt through holes, reinstall the hole plugs.

5. The following weight specifications are an approximation of weight distribution and chain length required to lift the machines.

Machine	Weight (pounds)	Chain Length (column)
VF-O/E	7,000	-
VF-OE/EXT	7,900	-
VF-1	7,100	-
VF-2	8,000	-
VF-3	12,500	-
VF-4	13,300	-
VF-5/40	14,500	-
VF-5/50	14,600	-
VF-6	21,000	120"
VF-7	23,000	120"
VF-8	24,000	120"
VF-9	25,000	120"
VF-10	28,000	120"
VF-11	29,400	120"



2. MAINTENANCE SCHEDULE

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	<ul style="list-style-type: none">✓ Check coolant level each eight hour shift (especially during heavy TSC usage).✓ Check way lube lubrication tank level.✓ Clean chips from way covers and bottom pan.✓ Clean chips from tool changer.✓ Wipe spindle taper with a clean cloth rag and apply light oil.
WEEKLY	<ul style="list-style-type: none">✓ Check Through the Spindle Coolant (TSC) filters. Clean or replace element if needed.✓ Check for proper operation of auto drain on filter regulator.✓ On machines with the TSC option, clean the chip basket on the coolant tank. Remove the tank cover and remove any sediment inside the tank. Be careful to disconnect the coolant pump from the controller and POWER OFF the control before working on the coolant tank. Do this MONTHLY for machines without the TSC option.✓ Check air gauge/regulator for 85 psi.✓ For machines with the TSC option, place a dab of grease on the V-flange of tools. Do this MONTHLY for machines without the TSC option.✓ Clean exterior surfaces with mild cleaner. DO NOT use solvents.✓ Check the hydraulic counterbalance pressure according to the machine's specifications.
MONTHLY	<ul style="list-style-type: none">✓ Check oil level in gear box. For 40 taper spindles: Remove inspection cover beneath spindle head. Add oil slowly from top until oil begins dripping from overflow tube at bottom of sump tank. For 50 taper spindles: Check oil level in sightglass. Add from side of gearbox if necessary.✓ Inspect way covers for proper operation and lubricate with light oil, if necessary.✓ Place a dab of grease on the outside edge of the guide rails of the tool changer and run through all tools.
SIX MONTHS	<ul style="list-style-type: none">✓ Replace coolant and thoroughly clean the coolant tank.✓ Check all hoses and lubrication lines for cracking.
ANNUALLY	<ul style="list-style-type: none">✓ Replace the gearbox oil. Drain the oil from the bottom of the gearbox. Remove inspection cover beneath spindle head. Add oil slowly from top until oil begins dripping from overflow tube at bottom of sump tank. For 50 taper spindles, add oil from the side of the transmission.✓ Check oil filter and clean out residue at bottom of filter.✓ Replace air filter on control box every (2) years.



2.1 TSC MAINTENANCE

- Check dirt indicator on 100 micron filter with TSC system running and no tool in the spindle. Change element when the indicator reaches the red zone.
- On newer machines, clean pump intake filter when indicator is in red zone. Reset indicator with button. All intake filters can be cleaned with a wire brush.
- After changing or cleaning filter elements, run TSC system with no tool in spindle for at least one minute to prime system.

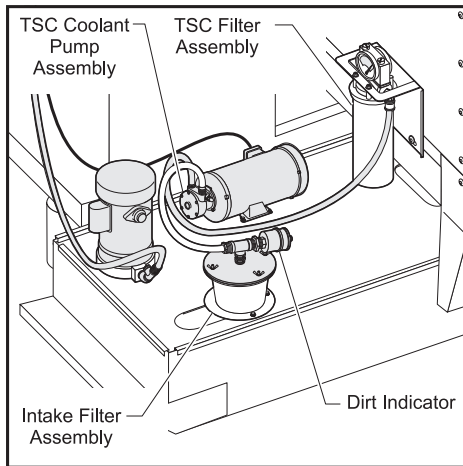


Figure 2-1. TSC coolant pump assembly.

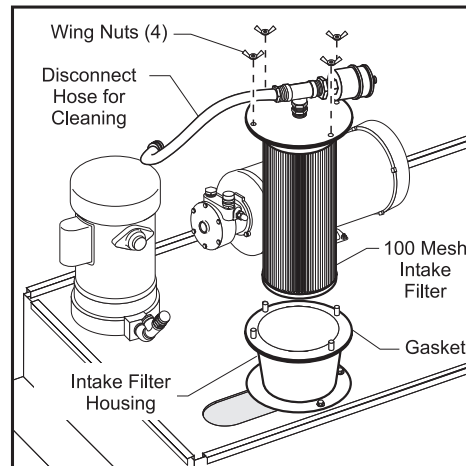
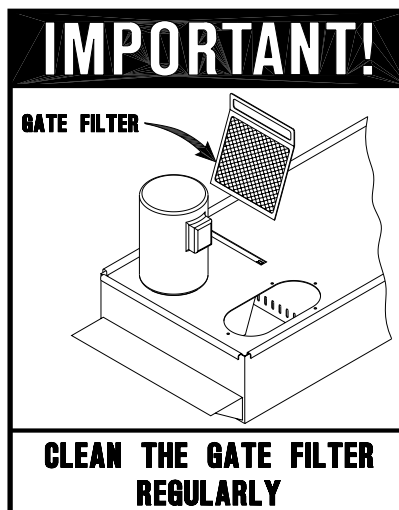


Figure 2-2. Cleaning the intake filter.



To clean the filter:

Turn off the coolant pump.
Remove the filter.
Clean and reinstall filter.

**2.2 LUBRICATION CHART**

SYSTEM	WAY LUBE AND PNEUMATICS	TRANSMISSION	COOLANT TANK
LOCATION	Under the control panel at the rear of the machine	Above the spindle head	Side of machine
DESCRIPTION	Piston pump with 30 minute cycle time. Pump is only on when spindle is turning or when axis is moving.		
LUBRICATES	Linear guides, ball nuts and spindle	Transmission only	
QUANTITY	2-2.5 Qts. depending on pump style	40 taper 2-Qt 50 taper 36 oz	40 Gallons 80 Gallons VF 6-10
LUBRICANT	Mobil Vactra #2	Mobile DTE 25	Water soluble, synthetic

WARNING!

The TSC pump is a precision gear pump and will wear out faster and lose pressure if abrasive particles are present in the coolant.

When machining castings, sand from the casting process and the abrasive properties of cast aluminum and cast iron will shorten pump life unless a special filter is used in addition to the 100 mesh suction filter. Contact Haas Automation for recommendations.

Machining of ceramics and the like voids all warranty claims for wear and is done entirely at the customer's risk. Increased maintenance schedules are absolutely required with abrasive swarf. The coolant must be changed more often, and the tank thoroughly cleaned of sediment on the bottom. A larger coolant tank is recommended.

Shortened pump life, reduction of pressure and increased maintenance are normal and to be expected in abrasive environments and is not covered by warranty.



This section contains the following:

- Introduction/ basic machine overview
- Control panel overview
- Set up and operation

3. OPERATION

3.1 BASIC INTRODUCTION

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

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

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

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

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

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

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



3.2 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.

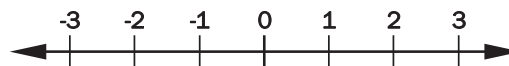
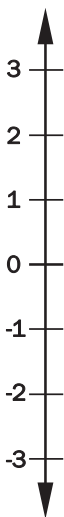


Fig. 3-1 Horizontal number line.

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

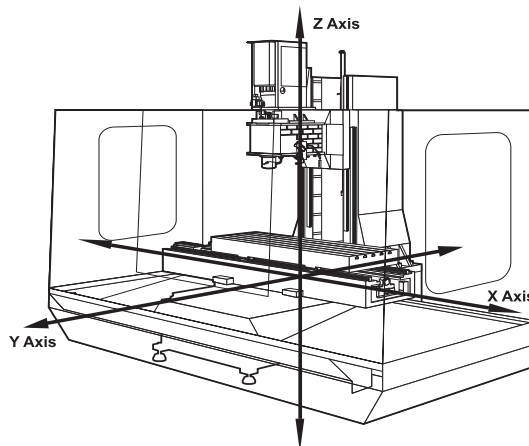


Fig. 3-2

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. 3-3 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.

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



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

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

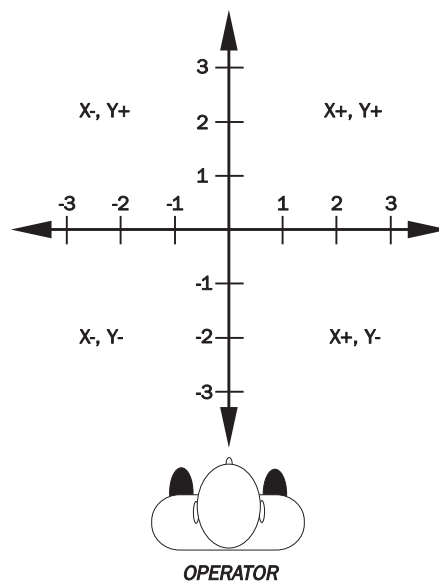


Fig. 3-4 View of X,Y grid from above.

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

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

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



3.3 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.

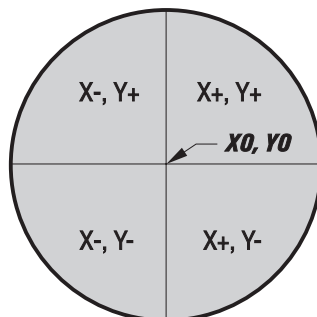


Fig. 3-5 All four quadrants will have to be accessed to machine this part.

Sometimes it is useful in the machining of a part to utilize more than one of the X,Y quadrants. A good example of this is a round part that has it's datum lines running through the center. The setup of such a part may look like Fig. 3-5.

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



3.4 ABSOLUTE AND INCREMENTAL POSITIONING

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

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

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

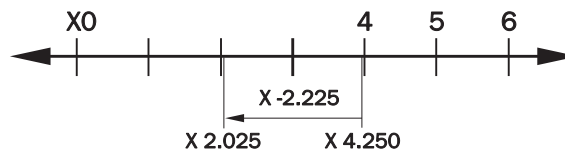


Fig. 3-6 An example of an incremental move.

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



3.5 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 has an alphabetical address. The universal rule here is that except for G codes and macro calls, Codes with the same alphabetical address cannot be used more than once on the same line.
- 2) **G** codes come in groups. Each G code group has a specific group number. G codes from the same group cannot be used more than once on the same line.
- 3) There are modal **G** codes groups which, once established, remain effective until replaced with another code from the same group.
- 4) There are non-modal **G** codes groups 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.
- G01 Linear interpolation motion; Used for actual machining and metal removal. Governed by programmed feed rate.
- G02 Circular interpolation - Clockwise
- G03 Circular interpolation - Counterclockwise
- G28 Machine home (Rapid traverse)
- G40 Cutter compensation cancel
- G41 Cutter compensation to **left** of path
- G42 Cutter compensation to **right** of path
- G43 Read tool length compensation
- G54 Work coordinate #1 (Part zero)
- G80 Canned cycle cancel
- G81 Drill canned cycle
- G82 Spot drill canned cycle
- G83 Peck drill canned cycle
- G84 Tapping canned cycle

- G90 Absolute programming
- G91 Incremental programming
- G98 Initial point return
- G99 Reference plane return

M CODES:

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

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



3.6 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 when it was never specified in the actual program, it is because the machine automatically reads G54 upon start-up. That is a **default**.

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

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

G00	Rapid traverse
G17	X,Y 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	Command absolute programming
G98	Canned cycle initial point return

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



3.7 PROGRAM FORMAT

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

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

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

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

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

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

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

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. They show that the machine will rapid position, that the machine is in the absolute coordinate mode, and most important, the work zero coordinates. 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. However 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.

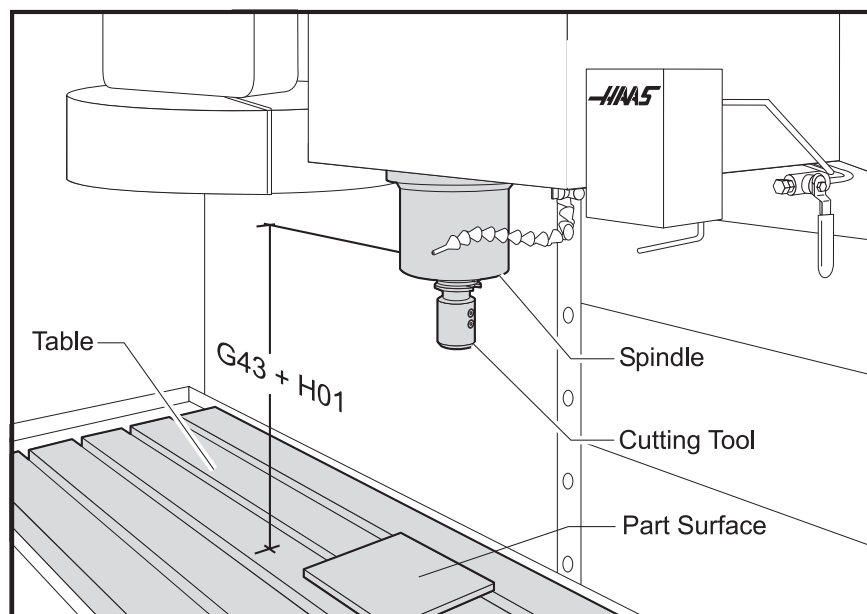


Fig. 3-7 Tool length offset and tool length compensation.

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



3.8 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** and/or **Y** move. G98 and G99 are modal commands which change the way the canned cycles operate. When G98 (the system default) is active, the Z-axis will be returned to the starting position at the completion of the canned cycle. When G99 is active, the Z-axis will be returned to the reference plane when the canned cycle is completed.

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

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



3.9 SETUP PROCEDURES

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

- 1) Load a program into memory. This is either manually entered or downloaded from a CAM package via the RS-232 interface.
- 2) Determine the tools needed and get them ready.

CAUTION!

If machine is equipped with the optional 50 taper spindle, follow these guidelines:

- 25 lb. maximum per tool, and 300 lb. maximum total tool weight.
- Extremely heavy tool weights should be distributed evenly.

If machine is equipped with the 20 or 32 pocket tool changer, follow these guidelines:

- 12 lb. maximum per tool (200 lb. maximum total tool weight for 32 pocket tool changer).
- Extremely heavy tool weights should be distributed evenly.
- Ensure there is adequate clearance between tools in the tool changer before running an automatic operation. This distance is 3.6" for 20 pocket, and is 3.4" for 32 pocket.

Note: If machine is equipped with the optional **extended Z-clearance table**, follow these guidelines:

- Avoid extreme loads on center of table or far end of table. Weight of part should be distributed evenly over the table or one pad.
- Flatness of part should be within .002"

- 3) Use a vise or fixture to hold the part and mount it on the mill.
- 4) Locate the **X** and **Y** zero points of your program on the part. Usually these points will coincide with the print reference point where dimensioning begins and needs to be clearly indicated by the programmer. Use an edge finder or indicator to locate this point with the handle function. After locating the programmed zero point, push the display offset key and page down until the work zero offset page appears. Cursor down to G54 **X**, the work zero offset. 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 OFFSET 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 OFFSET 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 position 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 OFFSET 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 keys other than JOG, TOOL OFFSET MESUR, and NEXT TOOL. Also note that this automatic offset measurement works with G43 only.



3.10 OPERATORS CONTROL PANEL

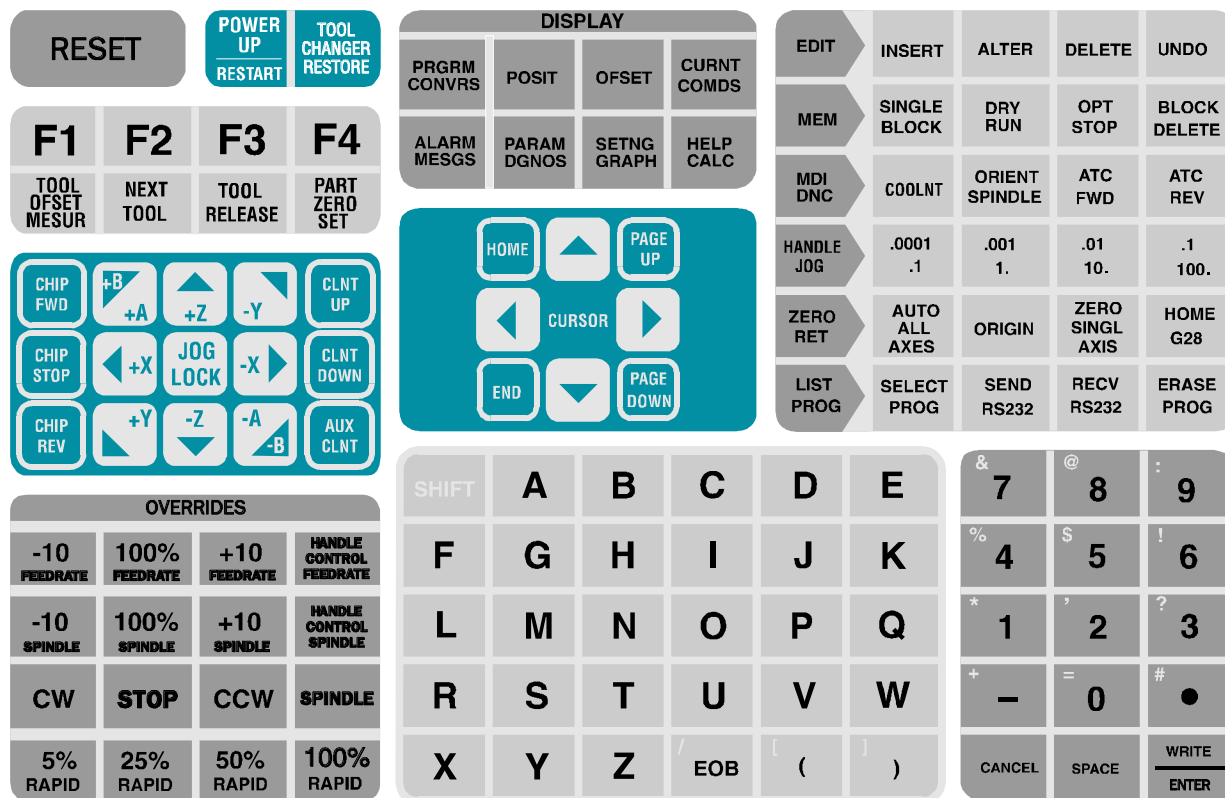


Figure 3-8 Control panel keypad with operating and display keys highlighted.

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

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



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

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

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

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

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

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

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

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

The **load meter** measures the power to the spindle motor. At 100%, the spindle motor can be operated continuously. The 150% level can be sustained for no more than ten (10) minutes, and at 200% level no more than three (3) minutes. After the specified time, the spindle may begin to slow and even stall. A 200% load should be reduced to 150% by reducing spindle speed or decreasing the feed rate. Spindle load may increase temporarily during speed changes.



The **jog handle** is used to jog one of the axes. Each step of the crank can be set to 0.0001, 0.001, 0.01 or 0.1 inch (0.001, 0.01, 0.1, or 1.0 degree per step for a rotary axis). When using metric units, the smallest handle step is 0.001 mm and the largest is 1.0 mm. The handle has 100 steps per rotation. It can also be used to move the screen cursor while in EDIT mode, or to change feed/spindle overrides by +/-1%.

An optional **remote jog handle**, with all the capabilities of the standard jog handle, is also available. Refer to the "Manual Operation" section for more information.

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

CYCLE START will start a program running in MEM or MDI mode, continue motion after a FEED HOLD, or continue after a SINGLE BLOCK stop. The CYCLE START button on the optional remote jog handle performs exactly the same functions.

FEED HOLD will stop all axis motion until the CYCLE START is pressed. The FEED HOLD button on the optional remote jog handle will perform exactly the same functions.

WARNING!

FEED HOLD will not stop the spindle, the tool changer, or the coolant pump. It will not stop motion of any auxiliary axes.

The optional **Memory Lock Key Switch** will prevent the operator from editing programs and from altering settings when turned to the locked position.

The following describes the hierarchy of locks:

- Key switch locks Settings and all programs
- Setting 7 locks parameters - parameters 57, 209 and 278 lock other features
- Setting 8 locks all programs
- Setting 23 locks 9xxx programs
- Setting 119 locks offsets
- Setting 120 locks macro variables

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

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

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



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

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

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

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

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

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

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

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

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

REAL TIME CLOCK

Included in the software is a real time clock. The current date and time are displayed on the diagnostics screen. The date and time are also applied to any alarms that are generated. The alarm history will display the alarm, the date and time that it occurred (see the alarms section on how to access the alarm history). Floppy directory contains file creation date and time. When files are saved to the floppy disk, the directory will now show the file creation date and time. Note that this feature also requires floppy driver software rev. 2.0 or later. When outputting a parameter file to a floppy or the serial port, it will contain two comments near the top containing the current date and time.

Macro variables are used to set the date and time. Macro variable #3011 contains the date in the format yymmdd (two digit year* 10000+ month* 100+ day). Macro variable #3012 contains the time in the format hhmmss (hours* 10000 + minutes* 100 + seconds).



3.11 KEYBOARD

The control panel keyboard consists of 133 keys and is divided into nine separate regions. They are:

RESET keys	Three (3) keys
FUNCTION keys	Eight (8) keys
JOG keys	Fifteen (15) keys
OVERRIDES	Fifteen (16) keys
DISPLAYS	Eight (8) keys
CURSOR keys	Eight (8) keys
ALPHA keys	Thirty (30) keys
MODE keys	Thirty (30) keys
NUMERIC keys	Fifteen (15) keys

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

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

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

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

F1-F4	Used in editing, graphics, background edit, and the help/calculator to execute special functions.
-------	---



TOOL OFFSET MESUR	Used to record tool length offsets in the offset page during part setup.
NEXT TOOL	Used to select the next tool during part setup.
TOOL RELEASE	Releases the tool from the spindle when in MDI mode, zero return, or handle jog. (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.
<u>JOG KEYS:</u>	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 JOG LOCK key stops jogging motion.
To the left side of the jog keys are three keys to control the optional chip auger. If the auger is enabled with Parameter 209, these keys perform the following functions:	
CHIP FWD	Turns the auger in a direction that removes chips from the work cell.
CHIP STOP	Stops auger movement.
CHIP REV	Turns the chip auger in the reverse direction.



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

CLNT UP	Pressing this key positions the coolant stream direction one position higher, if possible.
CLNT DOWN	Pressing this key positions the coolant stream direction one position lower, if possible.
AUX CLNT	Pressing this key while in MDI mode will turn on the Through the Spindle Coolant system, and pressing it again will shut off the system.

OVERRIDES:

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

HANDLE CONTROL FEEDRATE Allows jog handle to be used to control feedrate in +/-1% increments (from 0 to 999%).

-10	Decreases current feed rate by 10% (from 10 to 200%).
100%	Sets control feed rate to programmed feed rate.
+10	Increases current feed rate by 10% (from 10 to 200%).

HANDLE CONTROL SPINDLE

Allows jog handle to be used to control spindle speed in +/-1% increments (from 0 to 999%).

-10	Decreases current spindle speed by 10% (from 10 to 150%).
100%	Sets spindle speed to programmed speed.
+10	Increases current spindle speed by 10% (from 10 to 150%).
CW	Starts the spindle in the clockwise direction. Except CE machines.
STOP	Stops the spindle.
CCW	Starts the spindle in the counterclockwise direction.
5% RAPID	Limits rapid traverse to 5 percent of maximum.
25% RAPID	Limits rapid traverse to 25 percent of maximum.
50% RAPID	Limits rapid traverse to 50 percent of maximum.
100% RAPID	Allows rapid traverse to feed at its maximum.



DISPLAYS:

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

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

CURSOR KEYS:

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

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



(UP ARROW)

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

PAGE UP

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



(LEFT ARROW)

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



(RIGHT ARROW)

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

END

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



(DOWN ARROW)

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

PAGE DOWN

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

ALPHA KEYS:

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

SHIFT

The shift key provides access to the white characters on the keyboard. Pressing SHIFT and then the white character will cause that character to be sent to the control. The SHIFT key can also be continuously held down while a number of other keys are pressed.

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

In some FANUC compatible controls, the block delete symbol can be used to choose between two options when the "/" symbol is not at the beginning of the line. For instance, in the following line, T2 is executed when the block delete option is off, and when the block delete option is on, T1 is executed.

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

This cannot be done on a HAAS control.

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

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

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

MODE KEYS:

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

EDIT Selects edit mode.

INSERT	Inserts the text in the input buffer after the current cursor location. Also used to copy blocks of code in a program.
ALTER	Changes the item that the cursor is on to the text in the input buffer. Places an MDI program in the program list.
DELETE	Deletes the item that the cursor is on.
UNDO	Backs out or undoes up to the last 9 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	<p>Turns on optional stops. If an M01 code is encountered in the program and OPT STOP is on, a stop is executed. Depending on the lookahead function, it may not stop immediately. If the program has been interpreted many blocks ahead, and OPT STOP is pressed, then the nearest M01 may not be commanded. See G103.</p> <ol style="list-style-type: none">1. OPT STOP will take effect on the line after the highlighted line when OPT STOP is pressed.2. M01 is not allowed during cutter compensation. Alarm 349 will be generated in this case, as for M02, M30, and M00.
BLOCK DELETE	<p>Blocks with a slash ("/") as the first item are ignored or not executed when this option is enabled. If a slash is within a block, address codes after the slash will be ignored until after the block, if this option is enabled.</p> <ol style="list-style-type: none">1. When not in cutter compensation, block delete will take effect two lines after BLOCK DELETE is pressed.2. When in cutter compensation, blocks must be processed earlier. Therefore, block delete will not take effect until at last four lines after the highlighted line when BLOCK DELETE is pressed.3. If BLOCK DELETE changes state during the processing of the first block of a chamfering/rounding pair, and at least one of the pair is block-deleted, the behavior is undefined.4. Processing will slow down for paths containing block deletes during high-speed machining, because the lookahead queue will be emptied as processing approaches the block-deleted line(s). This limits the speed at which the previous blocks can run.
MDI/DNC	Selects MDI or DNC mode.
COOLNT	Turns the coolant on and off.
ORIENT SPINDLE	Rotates the spindle to a known position and then locks the spindle. Can be used during setup to indicate parts.
ATC FWD	Rotates the tool turret forward to the next sequential tool. If Tnn is in the input buffer, the turret will advance to tool nn . Except on CE machines.
ATC REV	Rotates the tool turret backwards to the previous tool. If Tnn is in the input buffer, the turret will advance to tool nn . Except on CE machines.



HANDLE JOG

Selects Jogging mode.

.0001, .1	.0001 inches or .001 mm for each division on the jog handle. For dry run .1 inches/min.
.001, 1.	.001 inches or .01 mm for each division on the jog handle. For dry run 1. inches/min.
.01, 10.	.01 inches or .1 mm for each division on the jog handle. For dry run 10. inches/min.
.1, 100.	0.1 inches or 1.0 mm for each division on the jog handle. For dry run 100. inches/min.

ZERO RET

Selects Zero Return mode.

AUTO ALLAXES	Searches for all axes' machine zero.
ORIGIN	Zeros out various displays and timers.
ZERO SINGLAXIS	Searches for machine zero on the axis that is specified in the input buffer.
HOME G28	Returns all axes to machine zero in rapid motion. Does not search.
SINGLE AXIS HOME G28	Either the X, Y, Z, A, or B axis can be returned to zero alone. The operator enters 'X', 'Y', 'Z', 'A' or 'B', then presses the HOME G28 key.

Pressing HOME G28 without first entering an axis letter will cause all enabled axes to be returned to zero.

If the chosen axis is disabled, the message DISABLED AXIS will be generated.

CAUTION! There is no warning message to alert the operator of any possible collision. For example, if the Z-axis is down in amongst parts on the table when X or Y is zeroed a crash can result.

SECOND HOME

This is an optional feature. When this button is pressed, the control will rapid all axes (which have the 2ND HOME BTN bit =1) to the coordinates specified in Work Offset G129. The sequence is as follows: First, assuming the X or Y axis need to be moved, the Z axis is returned to zero, then the X and Y axes are moved to their final positions, then the Z axis is moved to its final position. G129 Work Offsets must be set to the desired values for this feature to work correctly. This feature will work in any mode except DNC, just like the HOME G28 button.



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 RS-232 serial port. If ALL is highlighted, all the programs will be sent with one "%" at the beginning and one at the end of the stream.
RECV RS232	Receives programs from the RS-232 serial port. Unless ALL is highlighted, enter a valid program name in the form Onnnnn before pressing RECV RS232. If ALL is highlighted, do not enter a program name. The program names will be entered automatically from the input stream data.
ERASE PROG	Erases the highlighted program or the program specified in the input buffer.
<u>NUMERIC KEYS:</u>	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 / ENTER	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.



3.12 POWER ON/OFF

POWER ON

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

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

CAUTION !!!! After power on, the machine does not know its home position or stored stroke limits until it has been zero returned by the POWER UP/RESTART key or the ZERO RET/AUTO ALL AXES key. It is possible to jog the machine with the handle or jog keys at the lower feeds. If it is jogged unchecked in either direction, 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 machine Position displays are reset to zero.

The HOME G28 key should be used any time after the initial power up. This will return the Z-axis first and then the **X**, **Y**, **A**, and **B** axes all at rapid rate. If the Z-axis is positioned above the machine zero, the **X**, **Y**, **A**, and **B** axes are moved first. This key will work in any of the operating modes. The manual G28 button does not use any intermediate return point the way the programmed G28 does. Any auxiliary axes (**C**, **U**, ...) are returned to home after **X**, **Y**, **Z**, **A**, and **B**.

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

POWER OFF

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

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

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



3.13 MANUAL OPERATION

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 **Onnnnn** (new program number), then pushing ALTER. This will add that name to the program list and clear MDI.

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

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

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

When the Parameter 57 flag DOOR STOP SP is set to 1, the maximum spindle speed is 750 RPM with the doors open.

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. The display is changed to the Position Display and the currently selected axis for jogging will blink.

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

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

In the center of the jog buttons is a key labeled JOG LOCK. This key will cause the axis you are jogging to continue jogging even after you release the key. Press this key and then press the selected axis motion key to start. Motion will stop as soon as the JOG LOCK button is pressed again, or RESET is pressed.

Note: Selecting another axis will cause that axis to move.



This feature is handy, for example, when you are slow milling the soft jaws of a vise.

In order to select another axis for jogging while using the handle, use +/- **X**, **Y**, **Z**, or **A** buttons. When one of these buttons is pressed, that axis is selected for HANDLE JOG but does not move unless the button is held down for more than 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 will select the **A** axis for jogging. The display will show "JOGGING AAXIS HANDLE .01" while you are jogging the **A** axis. The **B** axis can be selected by pressing the SHIFT key, and then either the '+A' or '-A' key. When this is done the control will switch to jogging the **B** axis and the display will change to "JOGGING BAXIS 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.

REMOTE JOG HANDLE OPTION

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

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

The CYCLE START and FEED HOLD buttons on the remote jog handle perform the same exact functions as the same buttons on the control. They cannot be turned off, and can be used at any time.



3.14 AUTOMATIC OPERATION

OPERATION MODE

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

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

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.

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 current program. This is the program that will be run when you press START in MEM mode. It is the program with the “*” on the LIST PROG display. It is also the one that you will see on the EDIT display.

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

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

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



STARTING AUTOMATIC OPERATION

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

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

To start a program other than at the beginning, scan to the block number using the **down** arrow or PAGE DOWN until you reach the desired starting place. Push the MEM key and CYCLE START to begin. The Program Restart function, selected from Setting 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, it is important to use coolant.

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.

PROGRAM RESTART

Program Restart is designed to help the operator start a program from the middle while still properly interpreting all of the preceding lines of the program. To use program restart, turn on setting 36, move the cursor pointer to where you want to start (the restart point), and press Cycle Start. You do this by using the CURSOR up and down keys in MEM mode. The control will begin "invisible" interpretation of the program from the beginning and you will see the cursor move through the program. When it gets to the line you wanted to start on, the control will establish all of the conditions that would normally have been true at the end of the previous line and then execute the highlighted line and the rest of your program. Most program interpretation alarms which you might have gotten in the invisible phase will not occur until after the line at your restart point is complete.

As an example, if the following program were to restart at the T2 line, the control would change to T1 and then change to T2 before start axes motion. If the H and T agreement setting (15) was on, you would still not get an alarm. The double tool change is probably the most difficult to understand. The control does that because it insures that everything that must be true on the previous line is established and that means go tool 1. It then executes the restart line which says go to tool 2

```
O0123 ;
T1 M06 ;
G00 G90 G54 X0 Y0 ;
G01 F20. Z-2. ;
T2 M06 H03 ;           (RESART HERE)
G00 G90 G54 X0 Y0 ;
G01 F20. Z-2 ;
G28 ;
M30 ;
```

**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. The program is continued by pressing CYCLE START again.
- 3) A SINGLE BLOCK stop when operator selected. The program is continued by pressing CYCLE START again.
- 4) Door Hold stop caused by operator opening the enclosure doors. The program continues when the doors are closed.

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.



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.

WORK BEACONS

The red and green work beacons located directly on top of the control arm allow the operator to monitor the machine status.

While a program is running normally, the GREEN beacon will be on.

- The beacon will *flash* GREEN if:
- the operator selects FEEDHOLD or SINGLE BLOCK stop.
 - the control is in a M00, M01, M02, M30. It will stop flashing when RESET is pressed. If the control is in an M02 or M30, and door hold override is not on, the beacon will stop flashing when the door is opened.
- The beacon will *flash* RED if:
- the control encounters an alarm, such as when EMERGENCY STOP is pressed. It will stop flashing when RESET is pressed to clear all alarms.



3.15 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.

CREATING PROGRAMS

To create a new program, you must be in the PRGRM/CONVRS display and LIST PROG mode. Enter **O** (letter, not number) and a five 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 **Onnnnn** and an EOB (;). All further entries are made by typing a letter followed by a numeric value and pressing INSERT, ALTER, or WRITE. All items entered into a program are either addressed data (a letter of the alphabet followed by a number), a comment (text surrounded by parenthesis), or the End-Of-Block (EOB or ;).

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

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

A comment can be edited without entering the entire comment again. Simply highlight the characters you wish to change, enter the new characters, and press ALTER. To add characters move the cursor to where text is to be added, enter the new characters, and press INSERT. To remove characters highlight the characters and press DELETE. Use the UNDO button to reverse any changes. The UNDO button will work for the last nine entries.

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



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 **Onnnnn** 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 **Onnnnn**, type in the letter "O" followed by a four digit number, such as O1234, and press the ALTER key. The upper right hand screen will display the new program number. Your data will first appear in the lower left screen and will be input to the upper screen upon pressing INSERT, ALTER, or WRITE.

To enter a program from the keypad, type in the data you wish and press the INSERT key. More than one code, such as **X**, **Y**, and **Z**, can be entered before you press INSERT. After a program is entered, you may wish to change the data. Highlight the characters you wish to change, enter the new characters, and press ALTER. To add characters move the cursor to where text is to be added, enter the new characters, and press INSERT. To remove characters highlight the characters and press DELETE. Use the UNDO button to reverse any changes. The UNDO button will work for the last nine entries.

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

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

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

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

Background edit is described in detail later on in this chapter.

Editing error messages:

Guarded Code	You tried to remove the Onnnnn 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 Onnnnn 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.



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 **Onnnnn** for the program you want to edit and pressing F4. If you do not enter the **Onnnnn**, you will instead get the PROGRAM REVIEW display.

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

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

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

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

DELETING PROGRAMS

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

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



SPECIAL FUNCTION KEYS

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

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

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

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

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.

**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 shows 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, the lower right of the screen shows how to manipulate the block; the INSERT key is used to duplicate the defined block wherever the cursor is positioned, the DELETE key is used to delete the block, the ALTER key is used to move the block, and the UNDO key cancels the block definition.

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

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

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



3.16 PART PROGRAM INPUT/OUTPUT

RS-232 DATA INPUT/OUTPUT

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

Cables for the RS- 232 must be shielded

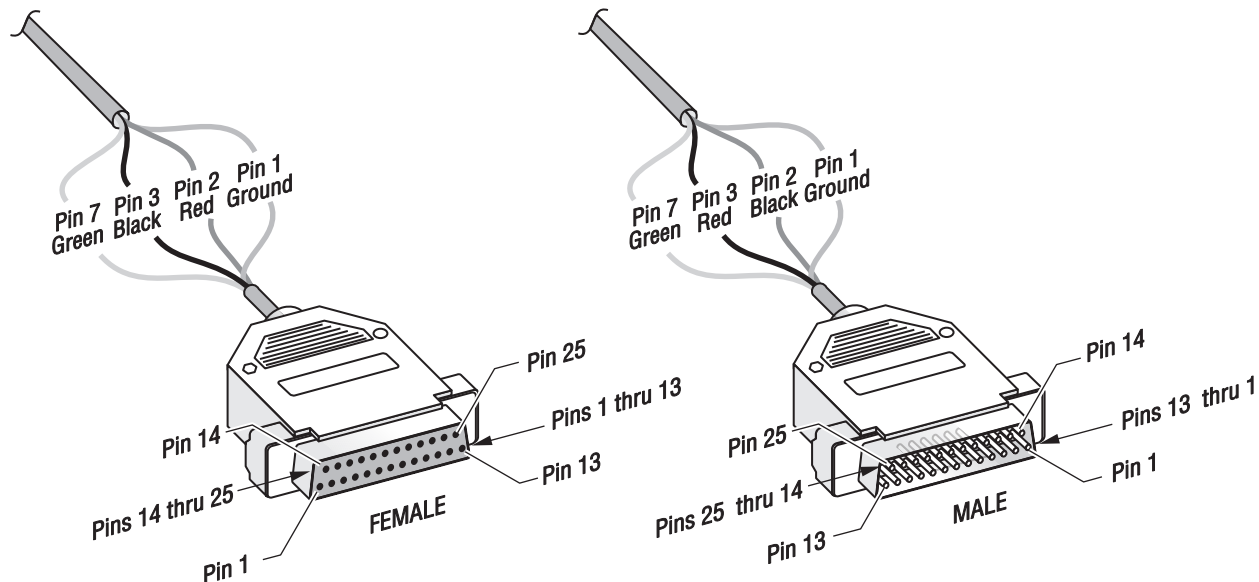
The following lists baud rate and the respective maximum cable length. This list assumes proper cable shielding and no signal boost.

9,600 baud rate: 100feet (30m) RS-232

38,400 baud rate: 25 feet (8m) RS-232

115200 baud rate: 6 feet (2m) RS-232

It is possible to use RS-232 to RS-422 converters on each end of the cable to accomplish longer cables at up to 115,200 baud. Proper twisted pair wire greatly improves reliability and increases maximum distance. A pentium processor should be used for baud speed of 115,200.



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 **Oxxxxx** 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 Onnnnn	When program name is received.
RS232 DONE	When complete and last % is received.
RS232 ABORT	When anything causes abnormal stop.

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

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

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

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

The 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



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

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

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

The format of data sent and received for macro variables is the same as above except there is a line N9999, and no line N0. The **N** number is the data number and **V** is the value. N0 is a CRC code that is computed by the control prior to sending the data. The N0 value is mandatory with parameters but is optional with settings and offsets. If you make a change to some saved data value and leave the old CRC, you will get an alarm when you try to load that data. With settings and offsets, you should delete the N0 line if you make changes to the saved data.

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

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

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

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

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



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

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

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

This version of XMODEM supports **checksum** verification only. Also, 512 bytes of memory must be available before using XMODEM with DNC. XMODEM must use 8 data bits and no parity.

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.

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

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.

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 CYCLE 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 CYCLE START before receiving a program, you will get the message: NO DNC PROG YET. The reason for not allowing the command of CYCLE 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 CYCLE 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 allowed, but it will not reset to the beginning of the program.

The program must begin with a % just like any other program sent over RS-232 and the program must end with a %. The data rate selected for the RS-232 port by settings must be fast enough to keep up with the rate of block execution of your program. If the data rate is too slow, the tool may be stopped in a cut when you might otherwise expect continuous cutter motion. The highest standard RS-232 data rate available is 115,200 bits per second (VF-E data rate is 38,400).

It is recommended that DNC be run with Xmodem or 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
RS-232 DATA BITS: 7
1 STOP BIT
XON/XOFF

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

To run DNC in graphics, you must select DNC first, then go to graphics display and send your program to the CNC

**FLOPPY OPERATION**

All files must be on MS DOS formatted 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 Onnnnn	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.

Note: To load or save a program numbered 9000 thru 9999, Setting 23 must be off.

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

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

On the List Prog page, type "DEL <filename>" where <filename> is the name of a file on the floppy disk. Press write. The message "FLOPPY DELETE" will appear, and the file will be deleted from the floppy disk.

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.

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 must enter the filename and press F3. You must do this before the START button can be pushed. After the beginning of the program is seen by the control, the display will show part of the program and a message at the bottom, left of the CRT will show DNC PROG FOUND. After the program is found, you may push CYCLE START just like running any other program from Memory.

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

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



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

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

When DNC is running, BACKGROUND EDIT is not available.

In order to run Graphics, you must first select DNC mode, then go to the graphics display, and enter a file name and press F3.



3.17 DRY RUN OPERATION

The DRY RUN function is used to check a program quickly without actually cutting parts. DRY RUN is selected by pressing the DRY RUN button while in MEM or MDI mode. When in DRY RUN, all rapids and feeds are run at the DRY RUN speed selected from the JOG speed buttons. The bottom of the screen will display the rate as 100, 10, 1.0 or 0.1 inches per minute.

DRY RUN cannot be turned on while a program is running. It can only be turned on or off when a program has completely finished or is reset. The first push of the DRY RUN button turns on this function and the second push will turn it off again. DRY RUN will still make all of the requested tool changes. The speed used in DRY RUN can be changed at any time and the operator can then check that the motions that are programmed are exactly what were intended. Note that Graphics mode is just as useful and may be even safer since it does not begin moving the machine before the program is checked.

3.18 DISPLAYS

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

PRGRM / CONVRS	To show or edit the program selected OR to select conversational mode (QuickCode)
POSIT	To show the axes positions.
OFFSET	To show or enter working offsets.
CURNT COMDS	To show current commands and times.
ALARM / MESGS	To show alarms and user messages.
PARAM / DGNOS	To show parameters and diagnostic data.
SETNG / GRAPH	To show or enter settings OR to select graphics simulation mode.
HELP / CALC	To show the help data and calculator.

In addition, when a program is running, you may press LIST PROG to select a list of the programs in memory. This is helpful in determining which programs can be edited in BACKGROUND EDIT, which is selected from the PROGRAM display.

**MESSAGES**

The screen will ALWAYS show some of the current conditions selected in the control. These are fixed status displays that describe the condition of the machine. The following conditions are displayed on the screen:

The selected display in the top left corner,	
The selected mode in parentheses,	
The presently selected program in the top right corner,	
The most recent line number in the top right corner,	
Up to 18 lines of variable display data,	
Any of the following conditions that apply:	
ALARM	Blinking in lower right corner when alarm occurs.
BLKDEL	BLOCK DELETE is turned on.
BUF	When next block is ready in continuous path.
DOOR HOLD	An open door has stopped the program.
DRYRUN	DRY RUN is selected.
DWELL	When a G04 is being performed.
FEED	When a feed motion is in progress.
FEED %	Feed rate override is active.
FEED HOLD	FEED HOLD is active.
OPTSTP	OPTIONAL STOP is turned on.
RAPID %	Rapid override is active.
RUNNING	When a program is running.
SINGBK	SINGLE BLOCK is turned on.
SINGBK STOP	When a program is stopped in SINGLE BLOCK.
SPIND %	Spindle speed override is active.
TOOL UNCLP	Reverse video when the tool is unclamped.
XYZA-MIR	These axes are set to mirror image.

The following error messages are received when the wrong button is pressed.

ALARM ON	Cannot start an operation until alarms are reset.
ALTER	The selected text can now be altered.
AUX AXIS BUSY	One or more auxiliary axes are busy in an operation.
BAD CODE	Code entered is not understood.
BAD FILE NAME	Not a valid file name.
BAD NAME	Name entered is not Onnnnn .
BLOCK TOO LONG	Block being edited would be too long.
CAN NOT COPY	The selected program can not be copied.
CAN'T RENAME	The selected program cannot be renamed.
CAN'T UNDO!	The last function can not be undone.
CNVEYR DISABLED	Chip conveyor has been disabled by parameters.
COOLANT OFF	Coolant pump is off.
COOLANT ON	Coolant pump is on.
DEL ALL (Y/N) ?	Delete all, yes or no ?
DELETE	Deleting text as requested.
DIR FULL	Maximum number of programs exceeded.
DIR NOT FOUND	Directory of floppy disk not found.
DISABLED AXIS	Requested axis has been disabled, and cannot be jogged.
DISK NOT IN DRV	No disk in floppy drive.
DISK NOT RDABLE	Floppy disk cannot be read by control.
DISK WRT PROTECT	Cannot save to floppy disk, it is write protected.



DISPLAYS OFF	Indicates that M76 was used to turn off displays.
DIVIDE BY ZERO	An attempt was made to divide by zero in calculator mode, or system error exists.
DNC END FOUND	The end of a DNC program has been found.
DNC PROG READY	The DNC program is ready to run.
DOOR IS OPEN	The door is open; some functions not allowed.
EMPTY PROG	No program found between the % and %.
END FOUND	End of program has been received.
END OF PROG	The program being run has completed.
EOF FOUND	End of file is found.
EXIT BG EDIT	Must exit Background Edit mode to perform this function.
FILE NOT FOUND	The file was not found.
FLOPPY ABORT	Something caused an abnormal stop.
FLOPPY DIR	The directory of files on the floppy disk.
FLOPPY DONE	When complete, and last % is received.
FLOPPY FOUND	Floppy drive is present.
FLOPPY READ	Reading from floppy drive.
FLOPPY WRITE	Writing to floppy drive.
FLPY NAME REQ	Floppy file must be named.
FLPY NOT ENBLED	Floppy drive has not been enabled by parameters.
FUNCTION ABORT	Requested function has been aborted.
FUNCTION LOCKED	Function attempted is locked from settings.
GUARDED CODE	Cannot remove Onnnnn at start of program.
INSERT	Selected text now being inserted.
INSUF DSK SPACE	Insufficient disk space to save the selected file.
INVALID AXIS	Selected axis is invalid.
INVALID NUMBER	Number entered is invalid.
JOG COMD	An axis jog has been commanded.
LOADING...	Reading programs or data from RS-232.
LOW AIR PR	Air pressure is low.
LOW COOLANT	Low level in coolant tank.
MACHINE LOCKED	Front panel has been locked by setting.
MACRO LOCKED	Macros 9000 to 9099 are locked by setting.
MEMORY FULL	Memory space is full.
MEMORY LOCKED	Memory lock is set in settings.
NEW PROGRAM	A new program may be entered.
NO DNC PROG YET	Attempted to start program before it was completely received.
NO FLOPPY FOUND	Cannot find the floppy drive.
NO INPUT	Cannot alter until something has been entered.
NO NAME ENTRY	No file name has been entered.
NO PROG YET	Cannot press Cycle Start until a program is received.
NO ZERO A	Cannot run machine until search for zero is complete on A-axis.
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.
NOT AVAILABLE	Function requested is not available at that time.
NOT FOUND	Item not found during search in editor.
NOT IN DRYRUN	The function requested applies to DRY RUN, but control is not presently in that mode.
ONE BIT ONLY	Only 0 or 1 is accepted to alter a parameter switch.
ONE PROG ONLY	Program name being selected cannot be ALL.
OVERWRITE (Y/N)	Do you want to overwrite the file, yes or no ?
PLEASE WAIT	Wait until spindle is stopped.
PROG EXISTS	Cannot rename to an existing program.
PROG NOT FOUND	Requested program not in memory.



PROG READY	Program has been received and is ready to run.
PROGRAM END	Cannot remove last EOB in program.
PROGRAM IN USE	Program is already in use.
RANGE ERROR	Data being entered is outside of the valid range.
RESET FIRST	Must press RESET before performing this function.
RIGID TAP	Rigid tapping is being performed.
RS-232 ABORT	RS-232 was aborted by operator action.
RS-232 DONE	RS-232 operation is complete.
RS-232 ERROR	RS-232 error (shown in alarms).
SEARCHING...	Searching program for requested text or G code.
SEL HI GEAR	High gear selected in program.
SEL LOW GEAR	Low gear selected in program.
SENDING OFFSET	Sending offsets via RS-232.
SENDING PARS	Sending parameters via RS-232.
SENDING SETTING	Sending settings via RS-232.
SENDING VARS	Sending variables via RS-232.
SENDING...	RS-232 output is in process.
SERVO IS OFF	When servos are off , you cannot start a program.
SERVO IS ON!	Parameter change was attempted with servo on. This is dangerous!
SHUTTLE IN	Tool changer is in position for a tool change.
SPEED COMD	A spindle speed must be commanded.
SPINDLE CCW	Spindle is turning counterclockwise.
SPINDLE CW	Spindle is turning clockwise.
SPINDLE HIGH	Spindle is in high gear.
SPINDLE IN USE	Spindle is being controlled by program - manual controls not available at this time.
SPINDLE LOCKED	Spindle is locked in place.
SPINDLE LOW	Spindle is in low gear.
SPINDLE ORI	Spindle is being oriented.
SPINDLE STOP	Spindle is not turning.
STRING TOO LONG	The text being entered is too long.
SYSTEM ERROR	Call your dealer.
TOOL CH LOCKED	Tool changer has been disabled by parameter 57 bit 1.
TOOL CLAMP	The tool is in the spindle.
TOOL OVERLOAD	Cutting tool is overloaded.
TOOL UNCLAMP	The tool has been unclamped.
TURRET IN	The tool changer is in position for a tool change.
TURRET OUT	The tool changer is out of position for a tool change.
WAIT OR RESET	Cannot perform requested function until program finishes or RESET is pressed.
WAIT...	Requested function is being performed.
WAITING...	Waiting for RS-232 input.
WRONG MODE	Function requested is available only in another mode.

And the following responses, only when in graphics mode:

CIRCULAR	A circular motion is being performed.
LINEAR	A linear motion is being performed.
M30 FOUND	End of program found and execution stopped.
M00 AFTER M06	Cannot have an M00 after M06.
NO ZOOM IN 3D	Zoom is not allowed in 3D graphics mode.
RAPID	A rapid motion is being performed.



In addition, the screen display can show one of the following eight types of data in the 18 lines of variable display:

Program Displays:

The PROGRAM DISPLAY is used to show your program while in either MEM, EDIT, or MDI modes.

Position Display:

The POSITION DISPLAY is used to select the **X**, **Y**, **Z**, or **A** axes positions in any of several coordinate systems. The PAGE UP and PAGE DOWN keys select between these.

Offsets Display:

The OFFSETS DISPLAY is used to enter and display tool length offsets, tool radius offsets, and work offsets. The PAGE UP and PAGE DOWN keys will scroll 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, Tool Load Monitor, and Axis Load Monitor. The PAGE UP and PAGE DOWN keys select between these.

Alarms / Messages Displays:

The ALARMS/MESSAGES DISPLAY is used to display alarms and to enter and display user messages. The second push of the ALARM button will select messages display. The CURSOR **up** and **down** buttons will display additional alarms if there is more than will fit on one page.

Parameters / Diagnostics Displays:

The PARAMETERS DISPLAY shows all of the machine dependent control parameters and the Diagnostic data. The second push of the PARAM / DGNOS button will select the diagnostic display. The PAGE UP, PAGE DOWN, up and down cursor keys, and the jog handle can be used to scroll through the parameter display screens in the control. The left and right cursor keys are used to scroll through the bits in a single parameter.

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.

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.



PROGRAM DISPLAYS

The PROGRAM DISPLAY is used to show a program being edited in EDIT mode or a program being run in MEM. In MEM mode, there is also a PROGRAM REVIEW display available.

The PROGRAM DISPLAY uses 18 lines of the text display area of the CRT to show the command blocks of a CNC program. The display is 40 positions wide and blocks that are longer than 40 positions are continued on the next line of the display.

The PROGRAM REVIEW function is available whenever a program is being run. This allows you to review the program that is running. This is selected by pressing F4 while in MEM mode and PROGRAM DISPLAY. The screen is changed to an 80 column display with the normal MEM display on the left and PROGRAM REVIEW on the right. The CURSOR and PAGE UP and PAGE DOWN keys can be used to change the display on the right to a different part of the program. The display on the left will show the progress of the running program. To exit PROGRAM REVIEW, select any other display.

While you are running a program, the BACKGROUND EDIT function is available as a standard feature. BACKGROUND EDIT allows you to edit any named program in memory while any program is being run in memory. BACKGROUND EDIT is selected from MEM mode in PROG display by entering **Onnnnn** 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 in your machine.

POSITION DISPLAYS

The following are the five position displays in this control:

Home Page

This display shows the four displays simultaneously in small characters. The other displays are shown in large characters. The PAGE UP and PAGE DOWN keys will change displays. The coordinates displayed (operator, work, machine, or distance to go) are selected using the cursor **up** and **down** keys. The last display selected will be shown in CURNT COMDS and SETNG / GRAPH displays when they are selected. In this display, any axis that is at the zero position will be highlighted. In the other four displays, the selected axis will be blinking.

Operator Display

This display is for the operator/setup person to use as desired, and is not used by the control for any position- ing functions. In JOG mode, and with this display selected, the ORIGIN button can be used to set the zero position. This display will then show position relative to the selected zero position.

Work Display

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

Machine Display

This display is the machine coordinate system that is automatically set on power up and the first ZERO RET. It cannot be changed by the operator or any work coordinate systems, and will always show the distance from machine zero. It can be used by a non-modal G53 command.

Distance To Go

This display is an incremental display that shows the travel distance remaining before the axes stop. When in the ZERO RET mode, this display shows a diagnostic value. When in JOG mode, this display shows the total distance jogged. In rigid tapping, this number decreases to zero at the bottom of the hole and then increases again as the reverse stroke occurs.

**OFFSETS DISPLAY**

Tool length and part zero offsets are displayed and can be entered in the Tool Offsets display. There are 100 **H** and **D** codes possible. The same offset number for both Z-axis offset (H1) and tool diameter compensation (D1) can be used because the offset contains separate values for each. Work offsets can also be specified for the fifth axis **B** address, in the work offset display. If the fifth axis is enabled (Setting 78), then additional data fields are made available for the fifth axis.

PAGE DOWN in the OFFSET display will go through all 100 possible tool offsets and then change to the work zero offsets. PAGE UP will go directly to work zero offsets. The jog handle can also be used to scroll through the offsets. Work zero offsets may be entered and displayed from this page or using the PART ZERO SET key. Refer to the "Setup Procedures" section for more information on using this key.

When entering offsets, pressing F1 will set the value to your entry. Pressing WRITE will cause the new value to be added to the old value. This allows small adjustments to be made to the offsets. Note that entering a negative entry and pressing WRITE will decrease the value of the offset. F2 will set the negative of the entered value into the offsets.

There is a geometry and wear value with each offset and these are added together by the control during operation. The initial value is entered into the geometry column by the setup person. During operation, the operator makes minor wear changes in the wear column. This method allows the operator to see actual tool wear by limiting it to the wear column. The geometry values can also be entered automatically by using the TOOL OFFSET MESUR key during setup procedures. Note that this automatic offset measurement works with G43 only. Refer to the "Setup Procedures" section for more information on using the TOOL OFFSET MESUR key.

The function of the offset display page has been modified slightly to accommodate multiple axes. Only the axes that are enabled are displayed on the work offset display. For example, if the fourth axis is enabled (Setting 30), and the fifth axis is not, then the display will show X,Y,Z,A. In this case, all axes can be displayed on the entire display. If only the fifth axis is enabled, then the display will show X,Y,Z,B. Since the fourth axis is disabled, there is no need to display it.

If both the **A** and **B** axes are enabled, then the last column of the offsets display serves a dual purpose. Either **A** or **B** can be accessed in the last column. By using the **left** or **right** arrow keys, the screen cursor can be moved into the axis field that is to be set. When the rightmost field is highlighted by the cursor and the top of the column indicates **A**, then the values in this column represent **A** axis offsets. If you press the **right** arrow key once, the cursor stays in the same place, but the column and its values will change to **B** axis parameters. You can view and modify work offsets in this manner using the **left** and **right** arrow keys when both axes are enabled.

Setting 15, "H & T Code Agreement", may be used to force the spindle tool number and offset number to be equal; otherwise an alarm will occur. This is the preferred setting as it avoids crashes.

Tool diameter may be entered as either radius or diameter. Setting 40 is used to select between these. The value used for cutter compensation is the sum of the geometry and wear values.

Cutter compensation is controlled by G41 and G42 and the selected tool diameter (D value). Positive values for cutter compensation work normally. Negative values for cutter compensation cause the opposite side cutter compensation to be used. This means that G41 with a negative number will be the same as G42 with the same but positive number.



Offsets may be sent and received with the RS-232 port. Refer to the "Data Input / Output" section for a description of how to do this.

When the COOLANT SPGT of Parameter 57 is enabled, the CLNT POS column of the offset display is accessible. The spigot can be positioned when an M08 is encountered in a program. The current **H** code is used by the M08 to determine where to position the coolant spigot. If offset #5 (H05) has **10** entered under the CLNT POS column, the spigot will be moved to position 10. When the cursor is positioned on the CLNT POS column, the current spigot position will be shown at the bottom of the display. If the spigot has not been zeroed, the position will be blank.

The permissible values that can be entered into this column are controlled by Parameter 206, SPIGOT POSITIONS. Entering a value of zero (0) indicates that the spigot will not be moved when an M08 code sequence is encountered.

The following code sequence demonstrates how the spigot can be commanded:

Note: That the tool length and radius offset numbers usually are the same as the tool number

```
O0001                (Sample coolant positioning)
T1 M06
G90 G54 G00 X_ Y_ S1400 M03
G43 H01 Z_ M08        (Moves spigot to H1 position)
.
.
.
T2 M06
G90 G54 X_ Y_ S900 M03
G43 H02 Z_ M08        (Moves spigot to H2 position)
.
.
.
G43 H42 Z.1M08        (Position to new location with a second offset H42)
                        (More than one offset number can be used for a tool if needed)
.
.
.
M30
```



CURRENT COMMANDS DISPLAY

The following are the seven current command displays in this control:

**Program Command Check (Home Page),
Current Display Command,
Operation Timers,
Macro Variables,
Tool Life Timers,
Tool Load Monitor,
and Axis Load Monitor.**

The PAGE UP and PAGE DOWN keys are used to select among these displays.

Program Command Check Display

This display, which is the Home Page for the Current Commands Display, shows a current overview of the important commands. It shows the programmed spindle speed (**Snnnnnn**), the spindle speed commanded to the spindle drive (**CMDnnnnnn**), and if Parameter 278 bit ACT DISPLAY is set to 1, the actual encoder spindle speed (**ACTnnnnnn**). In addition, this display shows the CW, CCW, or stopped command being sent to the spindle and the current gear position.

This display also shows the position of the axes. The coordinates displayed (operator, work, machine, or distance to go) are selected using the cursor **up** and **down** keys.

If the spigot is enabled, the current spigot position will be displayed.

Current display command

This display shows all of the alphabetical address codes (i.e. G, M, S, H, D) and their current value. These values may not be changed in this display. The default value is shown for the address codes that are not being used in the current program.

Macro Variables Display

This display shows a list of the macro variables and their present values. As the control interprets a program, the variable changes are displayed on this page and the results can be viewed. The variables may be modified in this display. For more information on this display, refer to the "Macros" section of this manual.

Operation Timers Display

This display shows the current power-on time, cycle start time, and the feed time. These times may be reset to zero by using the cursor **up** and **down** keys to highlight the desired title and pressing the ORIGIN button.

Listed below these times are two M30 counters that are used for counting completed parts. They may be set to zero independently to provide for the number of parts per shift and total parts. Both counters are increased when an M30 is operated.

Tool Life Display

This display shows the time the tool is in feed (Feed-Time), the time the tool is in the spindle (Total-Time), and the number of times the tool has been selected (Usage). This information can be used to assist in predicting tool life. The values in this display can be reset to zero using the Cursor and ORIGIN buttons. This is done by putting the cursor on the title line, and pressing ORIGIN to zero all of the data in that column.



This display may also be used to generate an alarm when a tool has been used a specific number of times. The last column is labeled "Alarm", and if the number for a tool is not zero, an alarm will be generated when that count is reached. This number can be changed by the operator. Alarm 362 is generated when the count is reached, and may be cleared with RESET.

The tool that is currently in the spindle is highlighted.

Tool Load Monitor and Display

With the tool load display, the operator can enter the maximum load that is expected for each tool, and when this load is exceeded in a feed, a certain action will be taken. This display provides for the entry of this alarm point and also displays the largest load that tool has seen in any previous feed.

The tool load monitor function operates whenever the machine is in a feed operation (G01, G02, or G03). The values entered into the tool load display are checked against the actual spindle motor load. If the limit is exceeded, the tool overload action specified in Setting 84 (alarm, feedhold, beep, or Autofeed) will be taken. If "alarm" is selected and the limit is exceeded, Alarm 174, "Tool Load Exceeded", will be generated. This alarm will stop the axis motors and the spindle motor, turn off the coolant, and disable the servos.

During a feed the load exceeds the tool limit the AUTOFEED feature, if selected, will automatically override the feed rate (reduce it) down to the percentage specified by parameter 301 (i.e. 1%) at the rate specified by parameter 300 (i.e. 10% per second). If the tool load later falls below 95% of the tool load limit percentage, the AUTOFEED feature will automatically override the feed rate (increase it) back to the feed rate that was in effect at the start of the feed at the rate specified by parameter 299 (i.e. 20% per second). These adjustments will be made in 0.1 second increments.

Axis Load Monitor

Loads are measured and displayed on the same screen for the X,Y and Z axes. Note that the MAX% and LIMIT% columns still apply to the spindle load.

ALARMS / MESSAGES 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.

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.

You may leave an electronic note to yourself or anyone else by using this feature. The note may be for the operator to change tools after running a number of parts or it may be a diary for machine maintenance intervals that are performed. Data is automatically stored and maintained even in a power off state. The message display page will be come up during power up if there are no alarms present.

To enter messages, press the ALARM / 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.



PARAMETER / DIAGNOSTIC DISPLAY

The **PARAMETER DISPLAY** can be selected at any time by pressing the PARAM DGNOS button. Changes to parameters can be made when in any mode except when running a program. The CURSOR **up** and **down** buttons move to different parameters and the PAGE UP and PAGE DOWN buttons move through groups of parameters. Parameters 1, 15, 29, 43, and 57 are displayed as a single page of discrete flags. Selecting among the flags is done with the CURSOR **left** and **right** buttons. Parameters cannot be changed with the servos on. 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. The following is a list of the parameter pages and the order of succession in the control:

PAGE TITLE	DATA DESCRIPTION
COMMON SWITCH	Non-axis bit switches.
COMMON PAGE1	First page of non-axis parameters.
COMMON PAGE2	Second page of non-axis parameters.
COMMON PAGE3	Third page of non-axis parameters.
MACRO M CALL	Parameters that alias M codes to subroutines.
MACRO G CALL	Parameters that alias G codes to macros.
X BIT SWITCH	Bit switches for the X axis.
X PARAMETERA	First page of X axis parameters.
X PARAMETERB	Second page of X axis parameters.
Y BIT SWITCH	Bit switches for the Y axis.
Y PARAMETERA	First page of Y axis parameters.
Y PARAMETERB	Second page of Y axis parameters.
Z BIT SWITCH	Bit switches for the Z axis.
Z PARAMETERA	First page of Z axis parameters.
Z PARAMETERB	Second page of Z axis parameters.
A BIT SWITCH	Bit switches for the A axis.
A PARAMETERA	First page of A axis parameters.
B BIT SWITCH	Bit switches for the B axis.
B PARAMETERA	First page of B axis parameters.
B PARAMETERB	Second page of B axis parameters.
X SCREW COMP	X axis screw compensation value.
Y SCREW COMP	Y axis screw compensation value.
Z SCREW COMP	Z axis screw compensation value.



The HOME key displays the first parameter page "COMMON SWITCH". Pressing the PAGE DOWN key will display the next page of parameters in the above list. The END key displays the last parameter page "B PARAMETERB". Pressing the PAGE UP key will display the preceding page of parameters in the above list. All other features on the parameters display have remained the same. So, if you are unfamiliar with the new format of the parameters, you can still search by parameter number. Enter the number of the parameter you want to see or view and press the **up** or **down** arrow key. The page that the parameter is on will be displayed and the parameter being searched for will be highlighted. Refer to the "Parameters" section for more information.

The **DIAGNOSTIC DATA DISPLAY** can be selected at any time by pressing the PARAM DGNOS button a second time. There are two pages of diagnostic data and the PAGE UP and PAGE DOWN buttons are used to select between them. After this, the current run time and the number of tool changes is displayed.

A five-axis control has additional diagnostic data to be aware of:

The first page of diagnostic data has two discrete outputs that control the rotary axes brakes; they are labeled "A BRAK" and "B BRAK". When motion is commanded to either the **A** or **B** axis, the brake for that axis must first be released. To release a brake on the HAAS control, a relay is activated. These two outputs represent the activation of the brake relays. If the **A** or **B** axis brakes are disengaged, then these outputs will read high (logical 1). Normally these bits will read low (logical 0).

The second page of diagnostic data shows the status of inputs from the motor interface board. Additional inputs for the expanded motor interface board are listed under "INPUTS4". These bits are monitored by the control to determine if the interface for the **B** axis is working correctly. Refer to the "Technical Reference" section of this manual for a description of the diagnostic page inputs.

SETTING / GRAPHIC DISPLAY FUNCTION

The **SETTINGS DISPLAY** can be selected at any time by pressing the SETNG / GRAPH button. When the settings are displayed, changes can be made to any of the settings. There are some special functions in the settings; refer to the "Settings" section for a more detailed description.

The **GRAPHICS FUNCTION** is a visual dry run of your part program without the need to move the axes and risk tool damage from programming errors. This function is far more powerful than using the DRY RUN mode because all of your work offsets, tool offsets, and travel limits can be checked before any attempt is made to move the machine. The risk of a crash during setup is greatly reduced.

To run a program in Graphics, you must be in either MEM or MDI mode.

After loading the program into memory, select MEM (or MDI) and press the SETNG/GRAPH key twice to select the Graphics Simulation mode. This function operates the same as if running a program on the machine except no physical machine action occurs.

The graphics screen is composed of the following areas:

DISPLAY TITLE AREA

The title area is on the top left line of the screen and indicates the display (GRAPHICS), the mode you are in (MEM or MDI), the program number, and the current program line being executed. It is the same as the top line of all displays.



KEY HELP AREA

The right side of the top line is the function key help area. Function keys that are currently available are displayed here with a brief description of their usage.

LOCATOR WINDOW

The lower right part of the screen has two functions: it can display the whole table area and indicate where the tool is currently located during simulation, or it can be used to display four lines of the program that is being executed. The F4 key can be used to toggle between these two modes.

TOOL PATH WINDOW

In the center of the display is a large window that represents a look down perspective of the X-Y axis. It displays tool paths during a graphics simulation of a CNC program. Rapid moves are displayed as coarse dotted lines, while feed motion is displayed as fine continuous lines. The rapid path can be disabled by Setting 4. The places where a drill can or canned cycle can be executed are marked with an X. The drill mark can be disabled by Setting 5.

The tool path window can be scaled. After running a program, you can scale any portion of the tool path by pressing F2 and then using the PAGE DOWN key and the ARROW keys to select the portion of the tool path that you want to see enlarged. During this process, a rectangle will appear within the TOOL PATH window and the Locator window indicating what the TOOL PATH window will represent when the zoom process is complete. The KEY HELP AREA will flash, indicating that the user is rescaling the view. The locator window always portrays the entire table with an outline of where the TOOL PATH window is zoomed to. The PAGE UP key unzooks the rectangle one step. After sizing or moving the rectangle, 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.

**HELP / CALCULATOR FUNCTION**

The **HELP FUNCTION** is selected by pressing the HELP display button. This will bring a mini-manual up on the CRT. There are 26 topic areas selectable with the A-Z keys. This also includes a directory of the topics. The areas covered are:

A	START UP AND RUNNING
B	PROG. REVIEW / DNC / 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; they are not displayed on the data entry line of the screen.

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

All of the Calculator Help functions have a calculator for simple add, subtract, multiply, and divide operations. When one of these functions (Trig, Circular, or Milling) is selected, a calculator window appears in the upper left corner of the screen, and below it the possible operations (**LOAD** + - * and /). **LOAD** is initially highlighted, and the other options can be selected with the left and right cursor arrows. Numbers are entered by typing them in at the cursor in the lower left corner of the screen and pressing the WRITE key. When a number is entered and **LOAD** is selected, that number will be entered into the calculator window directly. When a number is entered when one of the other functions (+ - * /) is selected, that calculation will be performed with the number just entered and any number that was already in the calculator window. The calculator will also accept a mathematical expression such as $23*4-5.2+6/2$. It will evaluate it (doing multiplication and division first) and place the result, 89.8 in this case, in the register (the window).



- F3** In EDIT and MDI modes the F3 key will copy the highlighted triangle/circular/milling/tapping value into the data entry line at the bottom of the screen. This is useful when you want to use a solution developed for a program input.

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

- F4** In the Calculator Help function, this button uses the highlighted Trig, Circular or Milling/Tapping data value to load, add, subtract, multiply, or divide with the calculator.

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.

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. Use the CURSOR **up** and **down** buttons to select the value to be entered with WRITE. 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.

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.

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.



3.19 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 POWER UP/RESTART 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.

Prior to establishing the home positions (POWER UP/RESTART 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.



3.20 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, from 10% to 150%, using the SPINDLE overrides as above. It 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.

By pressing the HANDLE CONTROL FEEDRATE key, the jog handle can be used to control feedrate from 0% to 999% in +/-1% increments. By pressing the HANDLE CONTROL SPINDLE key, the jog handle can be used to control spindle speed in +/-1% increments (from 0 to 999%).

Rapid moves (G00) may be limited to 5, 25, or 50 % of maximum using the keypad. If the 100% rapid is too fast, it may be set to 50% of maximum by Setting 10.

In the Setting page, it is possible to disable the override keys so that the operator cannot select them. This is Setting 19, 20 and 21.

The FEED HOLD button acts as an override button as it sets the rapid and feed rates to zero when it is pressed. The CYCLE START must be pressed to proceed after a FEED HOLD. When in a FEED HOLD, the bottom left of the screen will indicate this. The door switch on the enclosure also has a similar result but it will display "Door Hold" when the door is opened. When the door is closed, machine operation will continue normally. Door hold can be prevented with Setting 51. Door Hold and FEED HOLD do not stop any auxiliary axes.

When Parameter 57 flag DOOR STOP SP is set to 1, the door switch will stop the servos and the spindle. In addition, the maximum spindle speed is 750 RPM with the doors open.

There is also an override function for the coolant supply. This is done from the Setting 32. The "NORMAL" setting checks the low coolant alarm and turns the pump on and off with M codes. The "OFF" setting ignores the coolant alarm but will alarm if an attempt is made to turn the coolant on. The "IGNORE" setting is used to ignore all coolant commands and the low coolant alarm.

At any time a program is running, the operator may override the coolant setting by pressing the MDI Coolant button. The pump will remain either on or off until the next M command or operator action.

Overrides can now be reset through to defaults upon processing M30 and/or RESET. This feature is selected by Setting #83.



THIS PAGE INTENTIONALLY LEFT BLANK



This section contains the following:

- Alphabetical Address Codes
- Cutter Compensation
- Subroutines
- Functions

4. PROGRAMMING

4.1 WORK COORDINATE SYSTEM

These machines have three linear axes named **X**, **Y**, and **Z**. The **X**-axis moves the table left and right, the **Y**-axis moves it to and from the operator and the **Z** moves the milling head up and down. The machine zero position is the upper right corner of the mill table. All moves from this point are in a negative machine direction. If a rotary table is connected, an additional **A**-axis work offset is provided.

The work offset display is found on the offset display by pushing the PAGE UP key. You can display and manually enter work offsets from here. The work coordinate systems on a control with a fifth axis have all been expanded to accommodate **B**, the fifth axis. Work coordinate offsets can be set for the **B** axis in the offset display. Note that the auxiliary axes **C**, **U**, **V**, and **W** do not have any offsets; they are always programmed in machine coordinates.

The Home or Machine-Zero position is **X0**, **Y0**, **Z0**. Travel of these axes is limited in the negative direction by stored stroke limits defined in the parameters. Travel in the positive direction for the **X** and **Y** axes is limited simply to values less than zero. Positive travel for the **Z**-axis is limited to the highest position used for tool changing (about **Z4.5**). In addition, positive travel for all axes is limited by the home switch which acts as a limit switch.

Before a tool can machine your part, the control must know where your part is. The work coordinate system tells the control the distance from the work zero point of your part to the machine zero position. The work zero point of the part is decided by the programmer and usually is the common point where all print dimensions are referenced from. The machine zero position is fixed by the machine on power up and does not change. The operator must determine this distance and enter the value.

This control automatically chooses the **G54** system on power up. If you do not wish to use this system, zero out the values in the **G54 X**, **Y**, and **Z** or select another work offset.

The **G54** through **G59** or **G110** through **G129** offsets can be set by using the PART ZERO SET key. Position the axes to the work zero point of your part. Using the cursor, select the proper axis and work number. Press the PART ZERO SET key and the current machine position will be automatically stored in that address. This will work with only the work zero offsets display selected. Note that entering a nonzero **Z** work offset will interfere with the operation of an automatically entered tool length offset.

Work coordinate numbers are usually entered as positive numbers, except when Parameter 57, bit "Neg. Work Offset" is set to 1. In this case, the work coordinate numbers are entered as negative numbers.

Work coordinates are entered into the table as a number only. To enter an **X** value of **X2.00** into **G54**, you would cursor over to the **X** column and enter the number 2.0 only.



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

See the sections on G52, G92, and Setting 33 for more on work coordinate systems.

Offsets can be sent and received with the RS-232 port. See the "Data Input / Output" section for information on how to do this.



4.2 PROGRAM STRUCTURE

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

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

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

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

<pre>% O1234 (OP1 SAMPLE MILL PART) N1 (TOOL #1 IS A 1/2 INCH STUB DRILL) N5 G40 G49 T#1 M06 N100 G00 X0 Y0 Z.5 G43 H1 M3 S1400 T2 N101 G01 Z.2 F30. N102 G83 G98 Z-.625 R.03 Q.2 F5. N103 X1.5 Y1.5 N104 Y-1.5 N105 X-1.5 N106 Y1.5 N107 G00 G80 Z.5 N108 T2 M06 N2 (T #2 IS 5/8 90 DEG. COUNTERSINK) N200 G00 X0 Y0 Z.5 G43 H2 M3 S500 N201 G01 Z.2 F30. N202 G82 G98 Z-.27 R.0 F5. N203 X1.5 Y1.5 N204 Y-1.5 N205 X-1.5 N206 Y1.5 N207 G00 G80 Z.5 N208 G28 X0 Y0 Z2.0 N209 T#3 M06 N3 (TOOL #3 IS A 1/2 END MILL) (SET DIAMETER VALUE TOOL #3) N300 G00 X0 Y0 Z.5 G43 H3 M3 S1000 N301 G01 Z.2 F30. N302 Z-.625 F5. N303 G01 G41 X-1.00 N304 G03 I1.0 D1 N305 G00 G40 X00 N306 G00 Z.5 N307 G28 M30 %</pre>	<pre>:PROGRAM MUST BEGIN AND END WITH % :PROGRAM #AND COMMENT STATEMENT :(*****) NOTES TO OPERATOR : :RAPID TO POS, OFFSET 1, SPIN FWD :FEED 30 INCH/MINUTE TO Z DEPTH :PECK TO Z-.625 START .03 ABOVE :DRILL ANOTHER HOLE AT NEW X,Y :DRILL 3RD HOLE, PECK DEPTH IS .20 :DRILL FOURTH HOLE :DRILL FIFTH HOLE :CANCEL CANNED CYCLE :TOOL CHANGE TO TOOL #2 :N### ARE LINE NUMBERS :OFFSET 2, SPINDLE SPEED 500 RPM :FEED TO Z AT 30 INCH PER MINUTE :SPOT DRILL CYCLE, DRILL AT X0 Y0 :SEC HOLE R=START PLANE ABOVE ZERO :3RD HOLE G98=RETURN TO INIT POINT :FOURTH HOLE :FIFTH HOLE :RAPID TO Z.5 :ZERO RETURN AFTER MOVE TO X0, Y0 :TOOL CHANGE :N #'S ARE FOR YOUR CONVENIENCE :COMMENTS ARE IGNORED BY CONTROL :G43 = OFFSET Z IN MINUS DIRECTION :G01 CAN BE SPECIFIED AS G1 :FEED TO DEPTH :COMPENSATE CUTTER LEFT OF LINE :CUT CIRCLE CCW WITH TOOL DIA D1 :RAPID TO CENTER, G40 CANCELS COMP :RAPID OUT OF PART :ZERO RETURN, Z GOES FIRST THAN X,Y :RESET PROGRAM TO BEGINNING :END OF TAPE</pre>
---	--



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

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

4.3 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 -99999.000 degrees, and the largest number is 99999.000 degrees.

B Fifth axis rotary motion

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

C Auxiliary external rotary axis

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

D Tool diameter selection

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

E Contouring accuracy

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

F Feed rate

The **F** address character is used to select the feed rate applied to any interpolation functions, including pocket milling and canned cycles. It is either in inches per minute with four fractional positions or mm per minute with three fractional positions. When G93 (Inverse Time) is programmed, F is in blocks per minute, up to a maximum of 15400.0000 inches per minute (39300.000 millimeters per minute).



G Preparatory functions (G codes)

The **G** address character is used to specify the type of operation to occur in a block. The **G** is followed by a two or three digit number between 00 and 150. Each **G** code is part of a numbered group. 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. Multiple **G** codes can be placed in a block in order to specify all of the setup conditions for an operation, provided no two are from the same numbered group. 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 100. **H0** will cause no offset to be used and **Hn** will use the tool length entry **n** from the Offsets display. Note that **G49** is the default condition and will clear the tool length offsets; so you must select either **G43** or **G44** to activate tool length offsets. The TOOL OFFSET MESUR button will enter a value into the offsets to correspond to the use of **G43**.

I Canned cycle and circular optional data

The **I** address character is used to specify data used for some canned cycles and circular motions. It is either in inches with four fractional positions or mm with three fractional positions. It is followed by a signed number in inches between -15400.0000 and 15400.0000 or between -39300.000 and 39300.000mm 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. It is followed by a signed number in inches between -15400.0000 and 15400.0000 or between -39300.000 and 39300.000mm for metric.

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. It is followed by a signed number in inches between -15400.0000 and 15400.0000 or between -39300.000 and 39300.000mm for metric.

L Loop count for repeated cycles

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

M M Code miscellaneous functions

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

N Number of block

The **N** address character is entirely optional. It can be used to identify or number each block of a program. It is followed by a number between 0 and 99999. The **M97** function must 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 **Onnnnn** identification in the first block; it cannot be deleted. Altering the **O** in the first block causes the program to be renamed. An **Onnnnn** 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** in a program, 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), it may be a positive decimal with fraction between 0.001 and 1000.0. If it is used as a program name (for an M98), or a line number (for an M97), the value may be a positive number without decimal point up to 9999.

Q Canned cycle optional data

The **Q** address character is used in canned cycles and is followed by a signed number in inches between 0 and 8380.000 for inches or between 0 and 83800.00 for metric.

R Canned cycle and circular optional data

The **R** address character is used in canned cycles and circular interpolation. It is either in **inches** with four fractional positions or **mm** with three fractional positions. It is followed by a signed number in inches between -15400.0000 and 15400.0000 for inches or between -39300.000 and 39300.000 for millimeters. 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. It is either in inches with three or four fractional positions or millimeters with three fractional positions. It is followed by a signed number between -838.0000 and 838.0000 or -8380.000 and 8380.000 or in millimeters between -8380.000 and 8380.000. If no decimal point is entered, the last digit is assumed to be 1/1000 inches or 1/1000mm.

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. It is either in inches with three or four fractional positions or millimeters with three fractional positions. It is followed by a signed number between -838.0000 and 838.0000 or -8380.000 and 8380.000 or in millimeters between -8380.000 and 8380.000. If no decimal point is entered, the last digit is assumed to be 1/1000 inches or 1/1000mm.

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. It is either in inches with three or four fractional positions or millimeters with three fractional positions. It is followed by a signed number between -838.0000 and 838.0000 or -8380.000 and 8380.000 or in millimeters between -8380.000 and 8380.000. If no decimal point is entered, the last digit is assumed to be 1/1000 inches or 1/1000mm.

**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 -15400.0000 and 15400.0000 for inches or between -39300.000 and 39300.000 for millimeters. 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 -15400.0000 and 15400.0000 for inches or between -39300.000 and 39300.000 for millimeters. 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 -15400.0000 and 15400.0000 for inches or between -39300.000 and 39300.000 for millimeters. If no decimal point is entered, the last digit is assumed to be 1/10000 inches or 1/1000 mm.



4.4 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 **Onnnnn** 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);

```



4.5 TOOL FUNCTIONS (Tnn)

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 the 50 taper spindle option, the tool holders used are CT #50 taper, V flange, commonly called "CT 50".

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: 20-7165).

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 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.

CAUTION! If machine is equipped with the optional **50 taper spindle**, follow these guidelines:

- 25 lb. maximum per tool, and 300 lb. maximum total tool weight.
- Extremely heavy tool weights should be distributed evenly.



CAUTION! If machine is equipped with the **20 or 32 pocket tool changer**, follow these guidelines:

- 12 lb. maximum per tool, (200 lb. maximum total tool weight for the 32 pocket tool changer).|
- Extremely heavy tool weights should be distributed evenly.
- Ensure there is adequate clearance between tools in the tool changer before running an automatic operation. This distance is 3.6" for 20 pocket, and is 3.4" for 32 pocket.

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

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

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

If the shuttle should become jammed, the control will automatically come to an alarm state. To correct this, push the EMERGENCY STOP button and remove the cause of the jam. Push the RESET key to clear any alarms. Push the "Tool Changer Restore" button to reset the tool changer. **CAUTION!** Never put your hands near the tool changer when powered unless the EMERGENCY STOP button is pressed first.

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

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

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



4.6 SPINDLE SPEED FUNCTIONS

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 **Snnnn** should always be in the same block as the M41 or M42 as an unneeded double gear change might otherwise be performed.

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. Thread pitch can be from 5 to 100 TPI.

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

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

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

**WARMUP COMPENSATION**

When the machine is powered on, if setting 109, and at least one of Settings 110, 111 or 112 are set to a nonzero value, the following warning will be displayed:

CAUTION! Warm-up Compensation is specified!
Do you wish to activate
Warm-up Compensation (Y/N)?

If the operator responds 'Y', the control immediately applies the total compensation, (setting 110, 111 and/ or 112), and the compensation begins to decrease as the time elapses. For instance after 50% of the time in Setting 109 has elapsed, the compensation distance for the X-axis, in Setting 110, will be 50%.

As with other settings, the Warm-up Compensation Settings can be changed at any time. Updating the Warm-up Compensation Time may activate compensation, but changes to the X, Y or Z distance settings will not activate compensation. To "restart" the time period, it is necessary to power the machine off and on and answer "yes", to the compensation query at start-up.

WARNING!

Changing settings 110, 111 or 112 while compensation is in progress
can cause a sudden movement of up to 0.0044 inches.

The amount of remaining warmup time is displayed on the bottom right hand corner of the Diagnostics Inputs 2 screen using the standard hh:mm:ss format. The initial amount of warmup time to be used, starting when power is applied, is specified in Setting 109 WARMUP TIME IN MIN.



4.7 CUTTER COMPENSATION

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).

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.

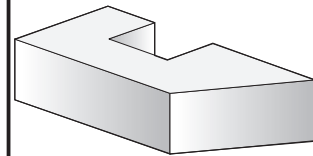
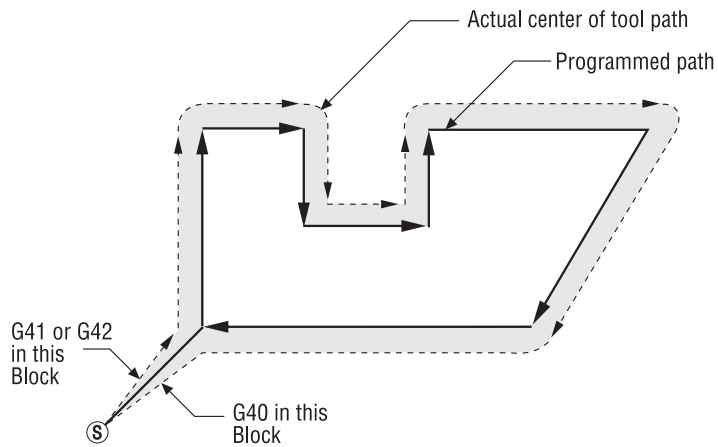
When Setting 58 is set to Fanuc, the control does not require that the tool cutting edge be placed along all programmed cuts. Overcutting, however, is still prevented and, if overcutting cannot be prevented, an alarm will still occur. Outside angles less than or equal to 270 degrees are joined by a square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

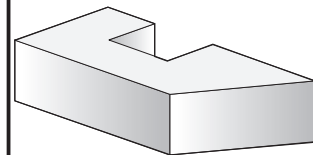
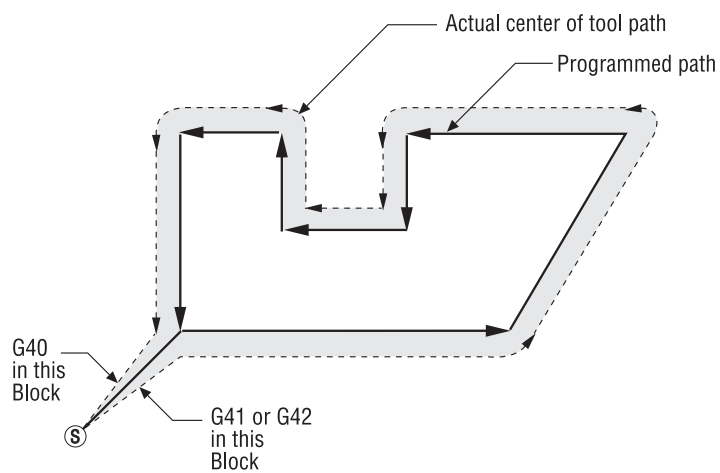
*G41 with Positive Tool Diameter
or G42 with Negative Tool Diameter*



- Center of tool Path
- Programmed Path
- Ⓢ Start Position
- ▭ Cutter Compensation

Cutter Compensation - (YASNAC Style)

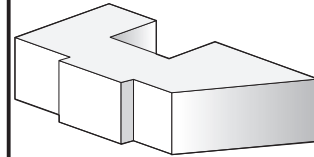
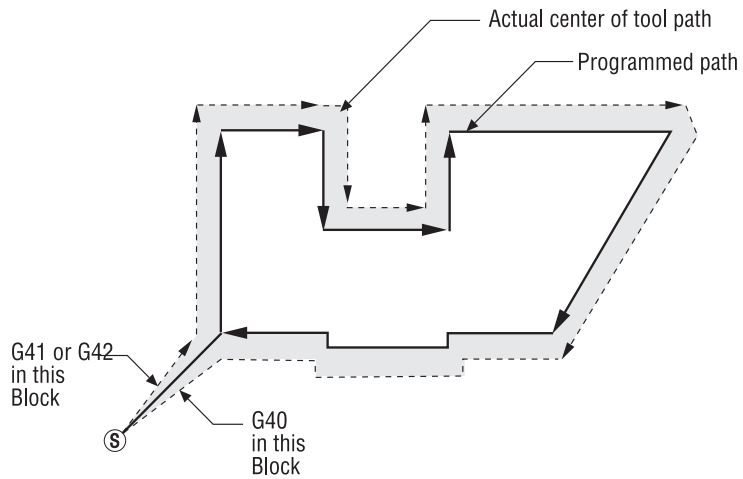
*G42 with Positive Tool Diameter
or G41 with Negative Tool Diameter*



- Center of tool Path
- Programmed Path
- Ⓢ Start Position
- ▭ Cutter Compensation

**Cutter Compensation - (FANUC Style)**

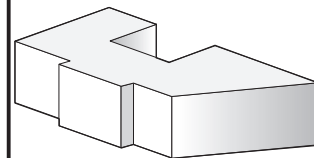
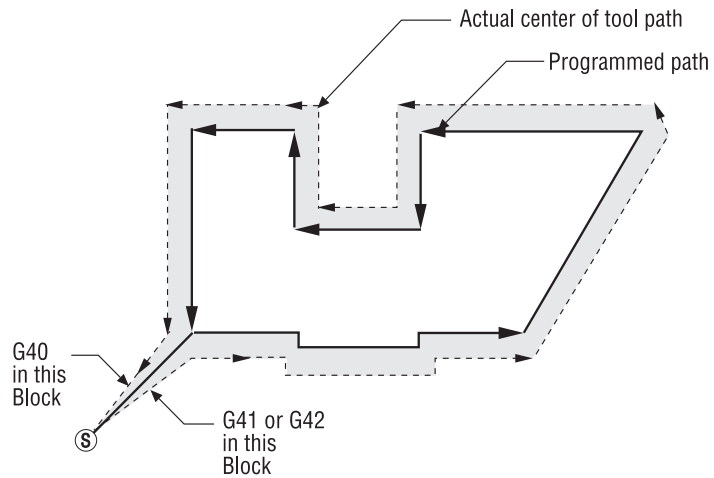
*G41 with Positive Tool Diameter
or G42 with Negative Tool Diameter*



- Center of tool Path
- Programmed Path
- Ⓢ Start Position
- ▭ Cutter Compensation

Cutter Compensation - (FANUC Style)

*G42 with Positive Tool Diameter
or G41 with Negative Tool Diameter*



- Center of tool Path
- Programmed Path
- Ⓢ Start Position
- ▭ Cutter Compensation



ENTRY AND EXIT FROM CUTTER COMPENSATION

When entering and exiting cutter compensation or when changing from left side to right side compensation, there are special considerations to be aware of. Cutting should not be performed during any of these three type of moves. In a block that turns on cutter compensation, the starting position of the move is the same as the programmed position but the ending position of the move will be offset by the cutter compensation size. In a block that turns off cutter compensation, the starting point is offset and the ending point is not offset. Similarly, when a block changes from left to right compensation, the starting point is shifted in one direction and the ending point is shifted in the other direction. The result of all of this is that the tool is moved through a path that may not be the same as the intended path or direction.

If cutter compensation is turned on or off in a block without any X-Y move, there is no change made to cutter compensation until the next **X** or **Y** move is encountered. To enter cutter compensation, a nonzero **D** code must be specified and either G41 or G42 specified. To exit from cutter compensation, you may either specify D0 or G40, or both.

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

An offset selection of D0 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 the previous section 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.

FEED ADJUSTMENTS IN CUTTER COMPENSATION

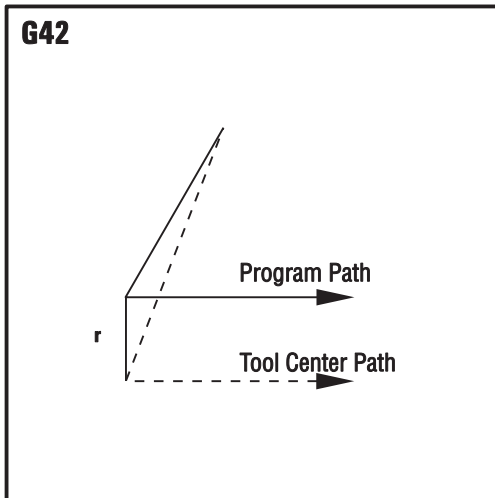
When using cutter compensation in circular moves, there is the possibility of speed adjustments to what has been programmed. If the intended finish cut is on the outside of a circular motion, the tool should be slowed down to ensure that the surface feed does not exceed what was intended by the programmer. There are problems, however, when the speed is slowed by too much. For this reason, Setting 44 is used to limit the amount by which the feed is adjusted in this case. It can be set between 1% and 100%. If set to 100%, there will be no speed changes. If set to 1% the speed can be slowed to 1% of the programmed feed.

When the cut is on the inside of circular motion, there is no speedup adjustment made to the feed rate.

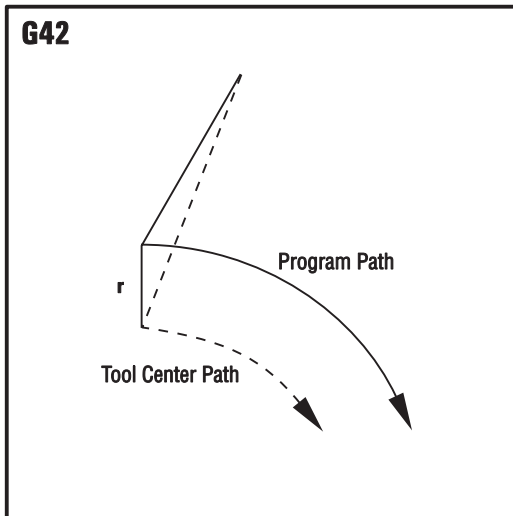


CUTTER COMPENSATION ENTRY (Yasnac)

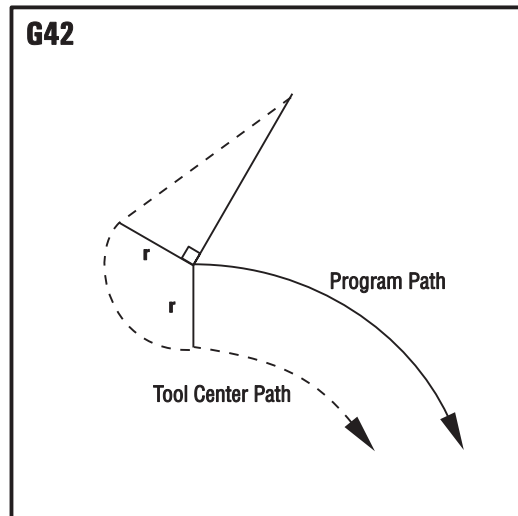
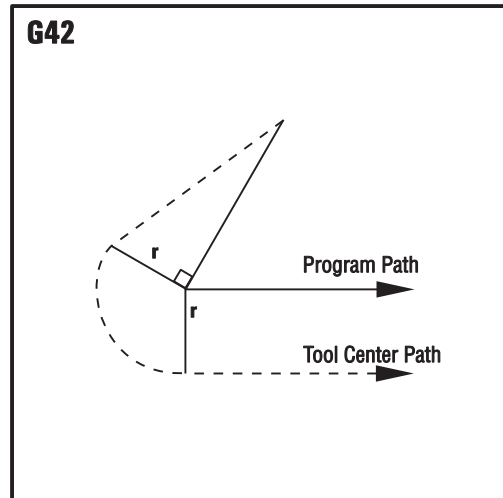
TYPE A



r Tool Radius
—→ Program Path
- - -→ Tool Center Path



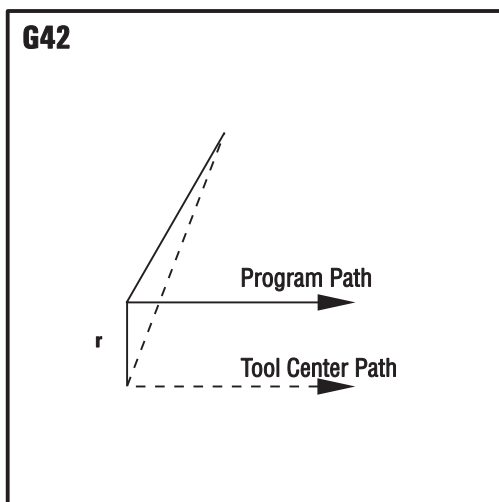
TYPE B



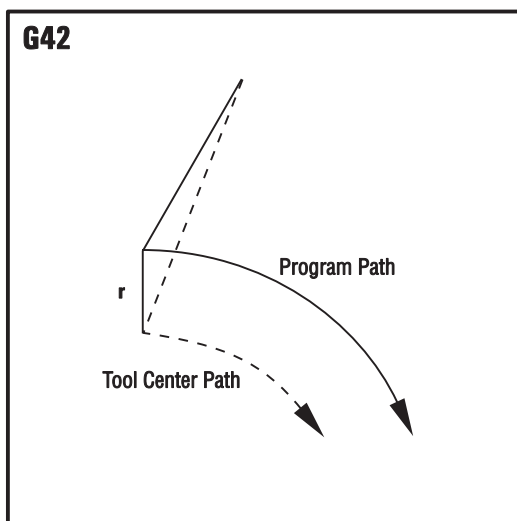


CUTTER COMPENSATION ENTRY (Fanuc style)

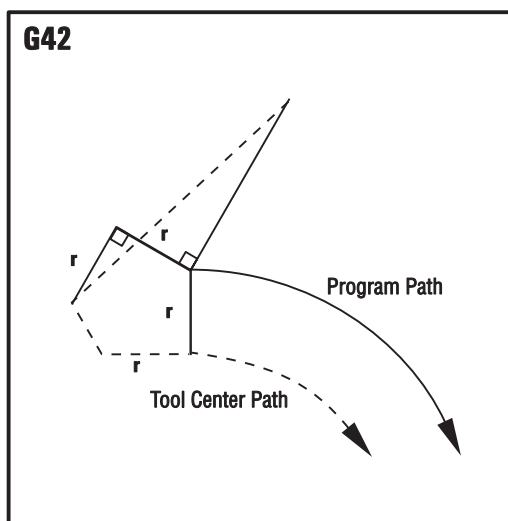
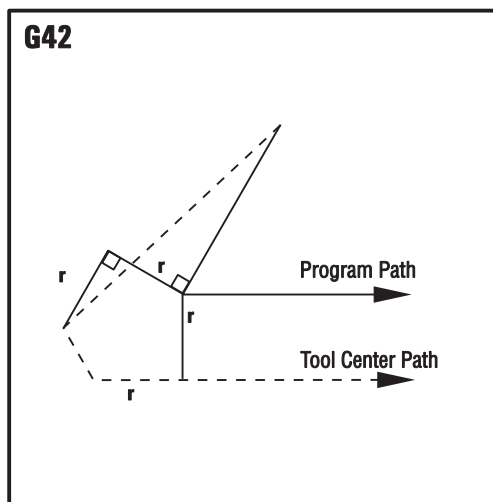
TYPE A



r Tool Radius
 ———→ Program Path
 - - - -> Tool Center Path



TYPE B



**FIVE AXES CUTTER LENGTH COMPENSATION**

The Haas control now has the capability for 5 axes cutter length compensation. This capability is most useful for the VR series of machines with A and B axes motion built into the machine head. It allows the user to correct for variations in the length of cutting tools without the need to revert to CAD/CAM or post processing steps. 5 axes length compensation applies only to machines where all rotary motion is movement of the cutting tool. It does not apply to machines where any of the rotary axes involves axes motion of the part or fixture.

Five axes cutter length compensation in the Haas CNC uses G143. An Hnn code is required to select the tool length from the existing length compensation tables. G143 is modal and is in the group 8 G codes. It is thus mutually exclusive with G43, G44 and G49. Selecting G49 or H00 will cancel 5 axes compensation. If only normal (Z axes) cutter length compensation is desired, simply program G43 or G44. For G143 to work correctly, there must be two rotary axes, A and B. G90 must be active for absolute positioning mode and G91 must not be used. Work position 0,0 for the A and B axis must be such that the tool is parallel with Z axes motion.

When G 143 is active, and commanded motion of X,Y or Z will have a vector component of the tool length added to the motion according to the work coordinates of the A and B axes. For a positive length of the tool, this will move Z upward or in the + direction. If one of X, Y or Z is not programmed, there will be no motion of that axes, even if motion of A or B cause a new vector for tool length. Thus a typical program would use all 5 axes on one block of data. If G91, incremental, is used, positioning results are incorrect.

Feed rate in G143 is complicated by the vector offset added to XYZ. Thus inverse feed (G93) is strongly recommended. If the inverse feed rate was correct before applying G143, it will still be correct with any length of compensation unless the maximum speed of an axes is exceeded.

An example follows:

```
T1 M06 ;
G00 G90 G54 X0 Y0 Z0 A0 B0 ;
G143 H01 X0. Y0. Z0. A-20. B-20. (RAPID POSIT W. 5 AX COMP) ;
G01 H01 X0. Y0. Z0. A-19.9 B-19.9 F300. (FEED INV TIME) ;
X0.02 Y0.03 Z0.04 A-19.7 B-19.7 F300. ;
X0.02 Y0.055 Z0.064 A-19.5 B-19.6 F300. ;
X2.345 Y.1234 Z-1.234 A-4.127 B12.32 F200. (LAST MOTION) ;
G94 F50. (CANCEL G93) ;
G0 G90 G49 Z0 (RAPID TO ZERO, CANCEL 5 AXIS COMP) ;
X0 Y0 ;
M30 ;
```

The following parameters have been added for G143:

Common switch SW 3:

5 AX TOFS -X	Set to 1 to negate X Comp. direction
5 AX TOFS -Y	Set to 1 to negate Y Comp. direction
5 AX A MOV B	Set to 1 if A axis moves B axis

When G143 is active, rotation of the A axes causes length compensation motion of the Y axes and rotation of the B axes causes length compensation motion of the X axes. The direction of these motions can be reversed with the above parameters. The B axes normally moves the A axes, but if this is not true, the "A MOV B" bit can be set to change which is the inner axes.



5. G CODES (PREPARATORY FUNCTIONS)

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

Code:	Group:	Function:	Description On Page:
G00	01*	Rapid Motion	161
G01	01	Linear Interpolation Motion	161
G02	01	CW Interpolation Motion	162
G03	01	CCW Interpolation Motion	163
G04	00	Dwell	164
G09	00	Exact Stop	164
G10	00	Set Offsets	164
G12	00	CW Circular Pocket Milling (Yasnac)	165
G13	00	CCW Circular Pocket Milling (Yasnac)	165
G17	02*	XY Plane Selection	167
G18	02	ZX Plane Selection	167
G19	02	YZ Plane Selection	167
G20	06	Select Inches	168
G21	06	Select Metric	168
G28	00	Return To Reference Point	168
G29	00	Return From Reference Point	168
G31	00	Feed Until Skip (optional)	168
G35	00	Automatic Tool Diameter Measurement (optional)	169
G36	00	Automatic Work Offset Measurement (optional)	169
G37	00	Automatic Tool Offset Measurement (optional)	169
G40	07*	Cutter Comp Cancel	170
G41	07	2D Cutter Compensation Left	170
G42	07	2D Cutter Compensation Right	171
G43	08	Tool Length Compensation +	171
G44	08	Tool Length Compensation -	171
G47	00	Text Engraving	171
G49	08*	G43/G44/G143 Cancel	171
G50	11	G51 Cancel	174
G51	11	Scaling (optional)	174
G52	12	Set Work Coordinate System G52 (Yasnac)	179
G52	00	Set Local Coordinate System (Fanuc)	179
G52	00	Set Local Coordinate System (HAAS)	179
G53	00	Non-Modal Machine Coordinate Selection	179
G54	12*	Select Work Coordinate System 1	179
G55	12	Select Work Coordinate System 2	179
G56	12	Select Work Coordinate System 3	179
G57	12	Select Work Coordinate System 4	179
G58	12	Select Work Coordinate System 5	179
G59	12	Select Work Coordinate System 6	179
G60	00	Unidirectional Positioning	180
G61	13	Exact Stop Modal	180
G64	13*	G61 Cancel	180
G65	00	Macro Subroutine Call (optional)	361
G68	16	Rotation (optional)	176
G69	16	G68 Cancel (optional)	179
G70	00	Bolt Hole Circle (Yasnac)	180
G71	00	Bolt Hole Arc (Yasnac)	180



G72	00	Bolt Holes Along an Angle (Yasnac)	181
G73	09	High Speed Peck Drill Canned Cycle	184
G74	09	Reverse Tap Canned Cycle	186
G76	09	Fine Boring Canned Cycle	187
G77	09	Back Bore Canned Cycle	188
G80	09*	Canned Cycle Cancel	189
G81	09	Drill Canned Cycle	189
G82	09	Spot Drill Canned Cycle	190
G83	09	Normal Peck Drill Canned Cycle	191
G84	09	Tapping Canned Cycle	193
G85	09	Boring Canned Cycle	194
G86	09	Bore/Stop Canned Cycle	195
G87	09	Bore/Stop/Manual Retract Canned Cycle	196
G88	09	Bore/Dwell/Manual Retract Canned Cycle	197
G89	09	Bore/ Dwell Canned Cycle	198
G90	03*	Absolute	198
G91	03	Incremental	199
G92	00	Set Work Coordinates - FANUC or HAAS	199
G92	00	Set Work Coordinates - YASNAC	199
G93	05	Inverse Time Feed Mode	199
G94	05	Feed Per Minute Mode	200
G98	10*	Initial Point Return	200
G99	10	R Plane Return	200
G100	00	Cancel Mirror Image	200
G101	00	Enable Mirror Image	200
G102	00	Programmable Output To RS-232	201
G103	00	Limit Block Buffering	202
G107	00	Cylindrical Mapping	202
G110	12	Select Work Coordinate System 7	206
G111	12	Select Work Coordinate System 8	206
G112	12	Select Work Coordinate System 9	206
G113	12	Select Work Coordinate System 10	206
G114	12	Select Work Coordinate System 11	206
G115	12	Select Work Coordinate System 12	206
G116	12	Select Work Coordinate System 13	206
G117	12	Select Work Coordinate System 14	206
G118	12	Select Work Coordinate System 15	206
G119	12	Select Work Coordinate System 16	206
G120	12	Select Work Coordinate System 17	206
G121	12	Select Work Coordinate System 18	206
G122	12	Select Work Coordinate System 19	206
G123	12	Select Work Coordinate System 20	206
G124	12	Select Work Coordinate System 21	206
G125	12	Select Work Coordinate System 22	206
G126	12	Select Work Coordinate System 23	206
G127	12	Select Work Coordinate System 24	206
G128	12	Select Work Coordinate System 25	206
G129	12	Select Work Coordinate System 26	206
G136	00	Automatic Work Offset Center Measurement	170
G141	07	3D+ Cutter Compensation	206
G143	08	5 AX Tool Length Compensation (optional)	206
G150	00	General Purpose Pocket Milling	208
G174/184	00	General-Purpose Rigid Tapping (optional)	210
G187	00	Accuracy Control for High Speed Machining	210



Each **G** code defined in this control is part of a group of **G** codes. The Group 00 codes are non-modal; that is, they specify a function applicable to the block they are in and do not affect other blocks. The other groups are modal and apply to all subsequent blocks until the end of the program or until replaced by a different **G** code from the same group. A modal **G** code means blocks between **G** code group members do not need to re-specify the same **G** code for each block.

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

RAPID POSITION COMMANDS (G00)

G00	Rapid Motion Positioning	Group 01
X	Optional X-axis motion command	
Y	Optional Y-axis motion command	
Z	Optional Z-axis motion command	
A	Optional A axis motion command	

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

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

INTERPOLATION COMMANDS (G01, G02, G03)

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 axes. All axes will start and finish motion at the same time. The rotary axis may also be commanded and this will provide a helical motion. The speed of all axes is controlled so that the feed rate specified is achieved along the actual path. Rotary axis feed rate 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 commands (G90 or G91) will change how those values are interpreted. The auxiliary axes **B**, **C**, **U**, **V**, and **W** can also be moved with a G01 but only one axis is moved at a time.

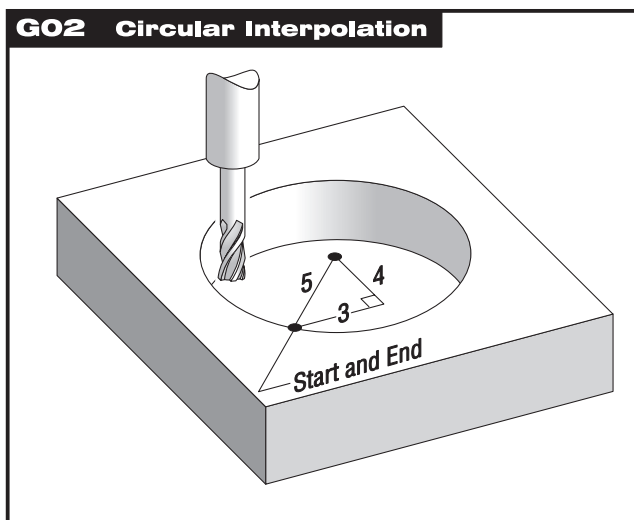
**G02 CW Circular Interpolation Motion****Group 01**

F	Feed rate in inches (mm) per minute
I	Optional distance along X-axis to center of circle
J	Optional distance along Y-axis to center of circle
K	Optional distance along Z-axis to center of circle
R	Optional radius of circle
X	Optional X-axis motion command
Y	Optional Y-axis motion command
Z	Optional Z-axis motion command
A	Optional A axis motion command

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

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

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



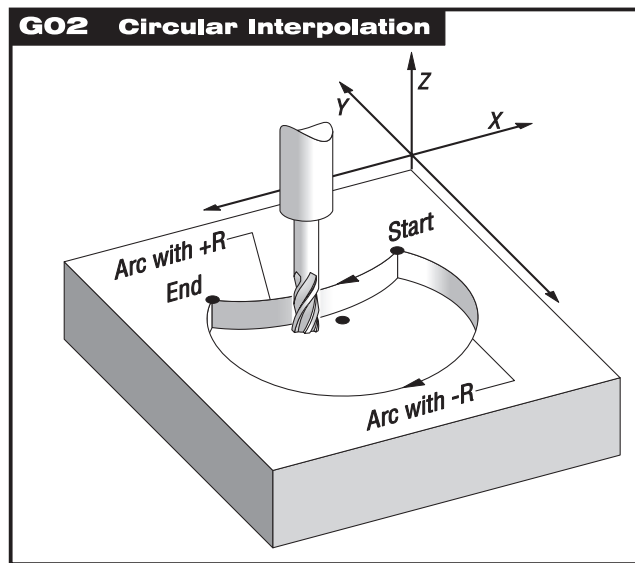
G02 I3.0 J4.0 (Assumes G17; XY plane)



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

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

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



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

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

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

G03 CCW Circular Interpolation Motion

Group 01

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

HELICAL

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

**MISCELLANEOUS G CODES (G04, G09)****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 non-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.

PROGRAMMABLE OFFSET SETTING (G10)**G10 Set 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.};

CIRCULAR POCKET MILLING (G12, G13)

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. Both are only functional in the default XY circular plane selection mode (G17).

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. To remove all the material within the circle use an **I** and **Q** value less than tool diameter and a **K** value equal to circle radius. To cut circle radius only use an **I** value set to circle radius and no **K** or **Q** value. 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.

G13 Circular Pocket Milling Counterclockwise

Group 00

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



%

O0100 (SAMPLE G12 AND G13)

(OFFSET D01 SET TO APPROX. TOOL SIZE)

(TOOL 1 IS A 0.5 INCH DIAMETER END MILL)

T1 M06

G54 G00 G90 X0. Y0.

G43 Z.1 H01

S2000 M03

G42 D01 X-5. Y-5.

G12 I.4 K1.4 Q.4 F10. Z-.25

G00 Z.1

G12 I1.5 F10. Z-.25

G41 D01 X-9. Y-5.

G13 I.4 K1.4 Q.4 F10. Z-.25

G00 Z.1

G13 I1.5 F10. Z-.25

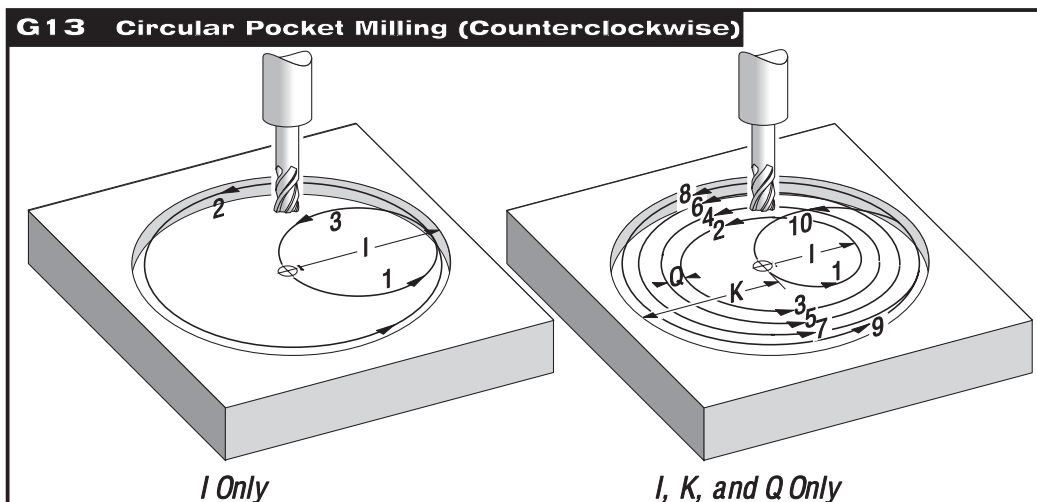
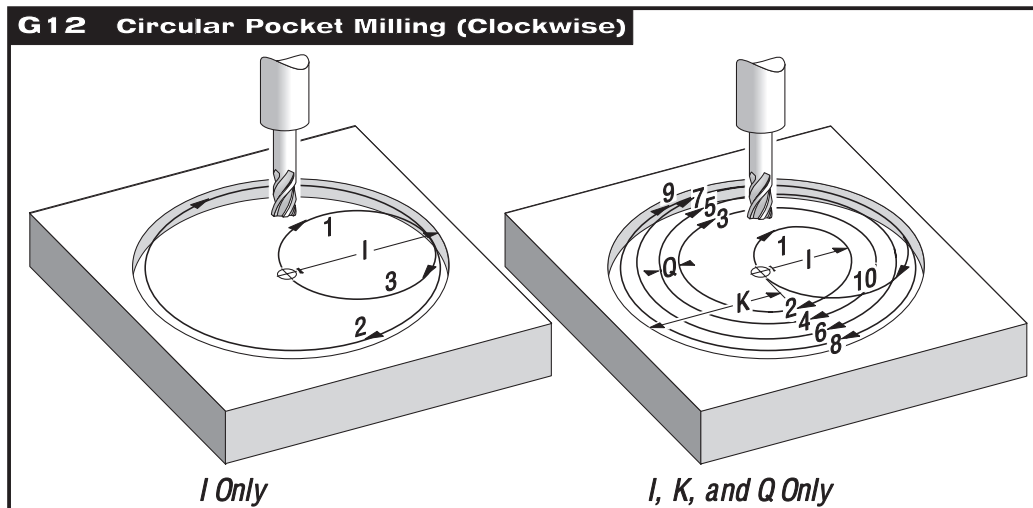
G00 Z.1

G28 Y0. G40 M30

%

(SET CUTTER COMP AND MOVE TO CIRCLE CENTER)
(REMOVE ALL MATERIAL IN CIRCLE TO 2.8 DIA)(FINISH CUT CIRCLE O.D. TO 3.0 DIA)
(SET CUTTER COMP AND MOVE TO CIRCLE CENTER)
(REMOVE ALL MATERIAL IN CIRCLE TO 2.8 DIA)

(FINISH CUT CIRCLE O.D. TO 3.0 DIA)





CIRCULAR PLANE SELECTION (G17, G18, G19)

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 ZX, and G19 for YZ. Each are modal and apply to all following circular motions until another Group 02 code is found.

G17 XY Plane Selection

Group 02

The G17 code is used to select the XY plane for circular motion. It is the default plane selection mode. In this plane, circular motion is defined as clockwise for the operator looking down onto the X-Y table from above. This is the motion of the tool relative to the table.

G18 ZX Plane Selection

Group 02

The G18 code is used to select the ZX plane for circular motion. In the X-Z plane (G18), circular motion is defined as clockwise for the operator looking towards the rear of the machine from the front control panel.

G19 YZ Plane Selection

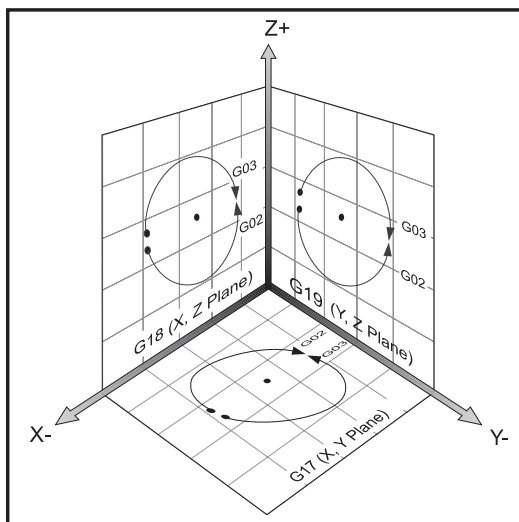
Group 02

The G19 code is used to select the YZ plane for circular motion. 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.

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

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

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



**INCH / METRIC SELECTION (G20, G21)**

G20 **Select Inches**
G21 **Select Metric**

Group 06
Group 06

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

Selection between inch and metric programming can only be done from the Setting 9.

REFERENCE POINT DEFINITION AND RETURN

G28 Return To Reference Point, set optional G29 REFERENCE point

Group 00

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

G29 **Return From REFERENCE point**

Group 00

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

SKIP FUNCTION (G31)

G31 **Feed Until Skip** (This G-code is optional and requires a probe)

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. The keypad will beep when the end of travel is reached. 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.

**AUTOMATIC TOOL MEASUREMENT (G35, G37)****G35 Automatic Tool Diameter Measurement** (This G-code is optional and requires a probe)**Group 00****G37 Automatic Tool Offset Measurement** (This G-code is optional and requires a probe)**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 offset measurement operation (G37) is a non-modal operation that causes a linear move of the Z-axis until the skip signal is received or the end of Z travel limits. A nonzero H code must be active, G43 or G44 must be active, a Z value must be specified, and a feed rate must be defined. No X, Y, or A code is allowed. When the move is terminated, the specified Z and the final Z positions are used to set the specified (Hnn) tool offset. The active coordinate system is taken into account.

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

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

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

**AUTOMATIC WORK OFFSET MEASUREMENT (G36, G136)**

G36 Automatic Work Offset Center Measurement (This G-code is optional and requires a probe) **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 in Settings 59 through 62.

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

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

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

CUTTER COMPENSATION (G40, G41, G42)

G40 **Cutter Comp Cancel** **Group 07**

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

G41 **2D 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 2D Cutter Compensation Right

Group 07

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

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

TOOL LENGTH COMPENSATION (G43, G44, G49)

G43 Tool Length Compensation + (plus)

Group 08

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

G44 Tool Length Compensation - (minus)

Group 08

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

G49 G43/G44/G143 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.

ENGRAVING (G47)

G47 Text Engraving

Group 00

Will not work with G code 91

E	=	Plunge rate (units/min)
F	=	Engraving feed rate (units/min)
I	=	Angle of rotation (-360. to +360.), default is 0
J	=	Scaling factor in inches (minimum = 0.001 inches), default is 1.0 inch
P	=	1 for Sequential Serial Number Engraving 0 for Literal String Engraving
R	=	Return plane
X	=	X start of engraving
Y	=	Y start of engraving
Z	=	Depth of cut

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

**SEQUENTIAL SERIAL NUMBER ENGRAVING**

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


G47 P1 (####)

will limit the serial number to four digits.

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

G47 P1 (1234)

will set the initial serial number to "1234".

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

G47 P1 (####)

will produce this:

1234

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

LITERAL STRING ENGRAVING

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

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

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

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



G47 P0 (ENGRAVE THIS)

will produce

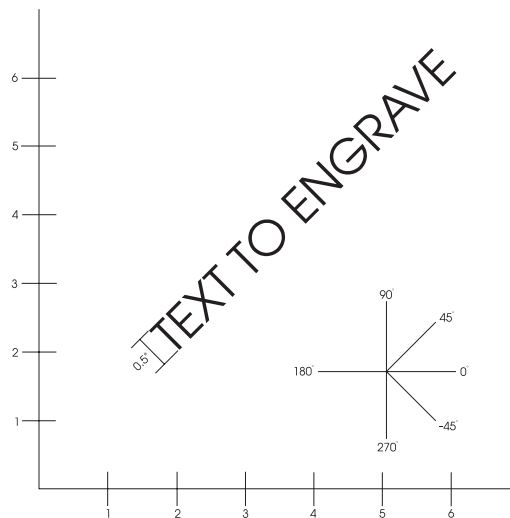
ENGRAVE THIS

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

EXAMPLE

This example will create the figure shown.

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



In this example:

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

**COORDINATE ROTATION AND SCALING (G50, G51, G68, G69)**

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

G50 Cancel Scaling**Group 11**

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

G51 Scaling (This G-code is optional and requires Rotation and 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.

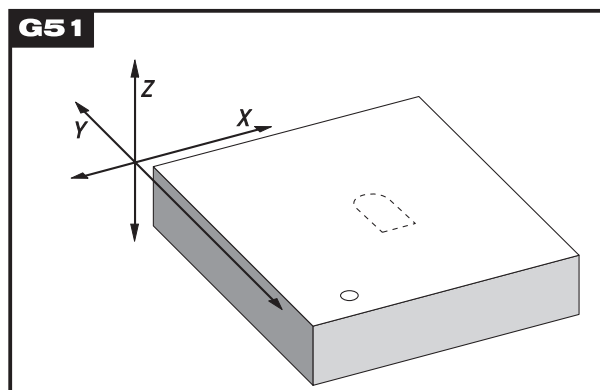
G51 is modal and modifies appropriate positional values in the blocks following the G51 command. It does not change or modify values in the block from which it is called. Axes X, Y, and Z are all scaled when the P code is used. If the P code is not used, the scaling factor currently in Setting 71 is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

```
0001 (GOTHIC WINDOW) ;
F20. S500 ;
G00 X1. Y1. ;
G01 X2. ;
Y2. ;
G03 X1. R0.5;
G01 Y1. ;
G00 X0 Y0 ;
M99 ;
```

○ = Work coordinate origin
No Scaling

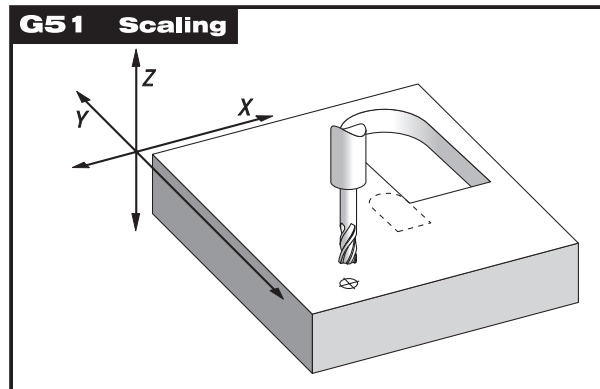


The first example illustrates how the control uses the current work coordinate location as a scaling center. Here, it is X0 Y0 Z0.



```
00010 ;
G59 ;
G00 G90 X0 Y0 Z0 ;
M98 P1 ;
G51 P2. (scaling center is X0 Y0 Z0) ;
M30 ;
```

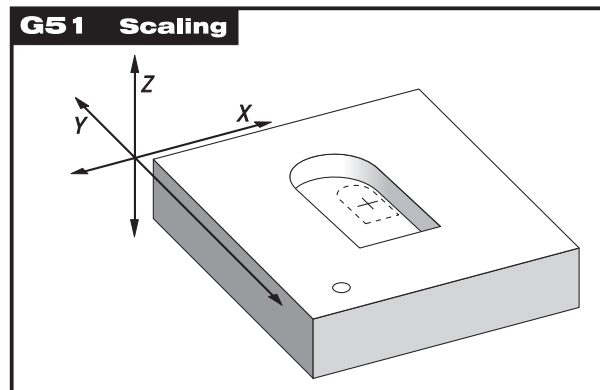
○ = Work coordinate origin
 + = Center of scaling



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

```
00011 ;
G59 ;
G00 G90 X0 Y0 Z0 ;
M98 P1 ;
G51 X1.5 Y1.5 P2. ;
M98 P1 ;
M30 ;
```

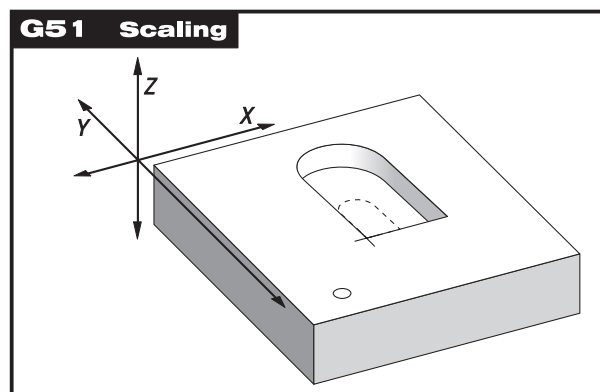
○ = Work coordinate origin
 + = Center of scaling



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.

```
00011 ;
G59 ;
G00 G90 X0 Y0 Z0 ;
M98 P1 ;
G51 X1.0 Y1.0 P2 ;
M98 P1 ;
M30 ;
```

○ = Work coordinate origin
 + = Center of scaling



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 (This G-code is optional and requires Rotation and Scaling)

Group 16

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

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

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

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

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

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

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

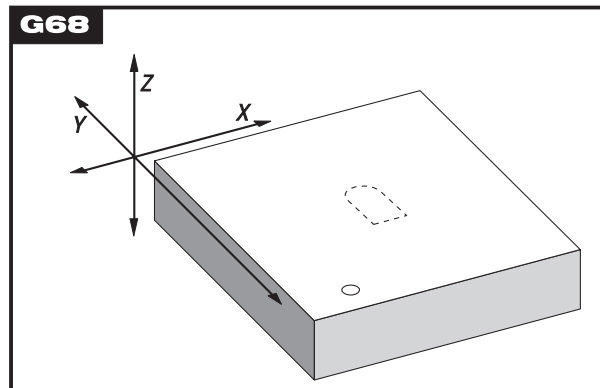
In G90 mode (absolute), the rotation angle takes on the value specified in **R**. When Setting 73 (G68 INCREMENTAL R) is set to ON, then the rotational value can be incremented on each call to G68. In G91 mode (Incremental), the rotation angle is incremented by the value in **R**. Each G68 command block,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



The following examples illustrate rotation using G68.

```
0001 (GOTHIC WINDOW) ;
F20, S500 ;
G00 X1. Y1. ;
G01 X2. ;
Y2. ;
G03 X1. R0.5
G01 Y1. ;
M99 ;
```

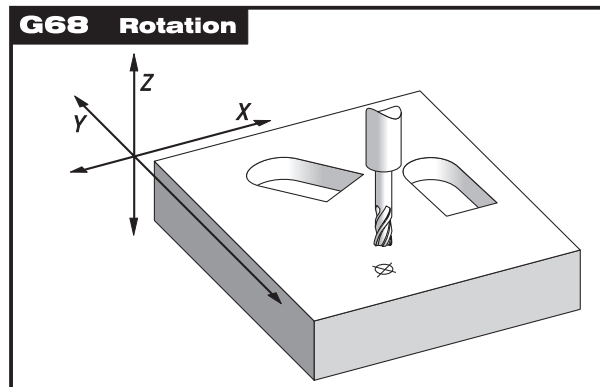
○ = Work coordinate origin
No Rotation



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

```
00002 ;
G59 ;
G00 G90 X0 Y0 Z0 ;
M98 P1 ;
G90 G00 X0 Y0 ;
G68 R60. ;
M98 P1 ;
G69 G90 G00 X0 Y0 ;
M30 ;
```

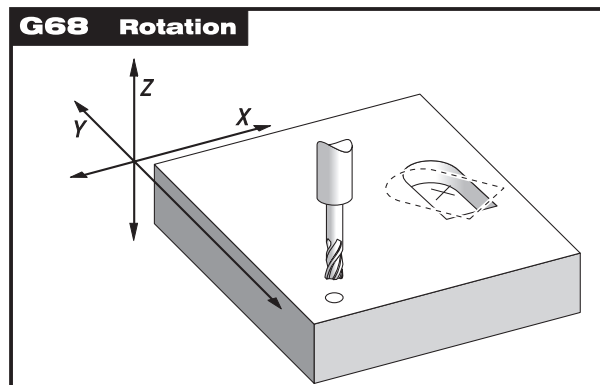
○ = Work coordinate origin
+ = Center of rotation



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

```
00003 ;
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 ;
```

○ = Work coordinate origin
+ = Center of rotation



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.



```

00004 ;
G59 ;
G00 G90 X0 Y0 Z0 ;
M98 P10 L8 (SUBROUTINE 00010) ;
M30 ;

```

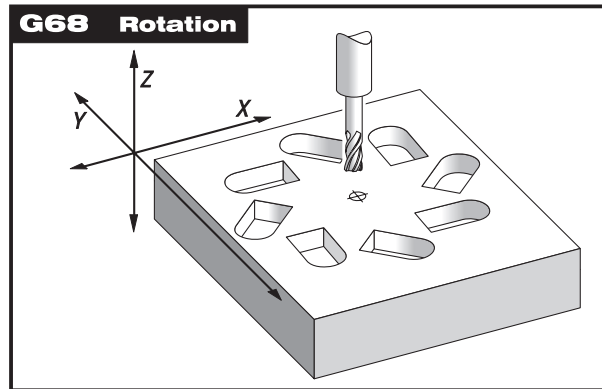
```

00010 ;
G91 G68 R45. ;
G90 M98 P1 ;
G90 G00 X0 Y0 ;
M99 ;

```

○ = Work coordinate origin

⊕ = Center of rotation



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 (This G-code is optional and requires Rotation and Scaling) **Group 16**

G code 69 cancels any rotation specified previously.

WORK COORDINATE SYSTEM SELECTION (G52, G53, G54-59)

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

G52 Set Work Coordinate System G52 YASNAC **Group 12**

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

YASNAC compatible.

G52 Set Local Coordinate System FANUC **Group 00**

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

G52 Set Local Coordinate System HAAS **Group 00**

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

G53 Non-Modal Machine Coordinate Selection **Group 00**

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

G54-59 Select Coordinate System #1 - #6 **Group 12**

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

**MORE MISCELLANEOUS G CODES (G60, G61, G64)****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 affect 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.

BOLT HOLE PATTERNS (G70, G71, G72)

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

G70 Bolt Hole Circle**Group 00**

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

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

G71 Bolt Hole Arc**Group 00**

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

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

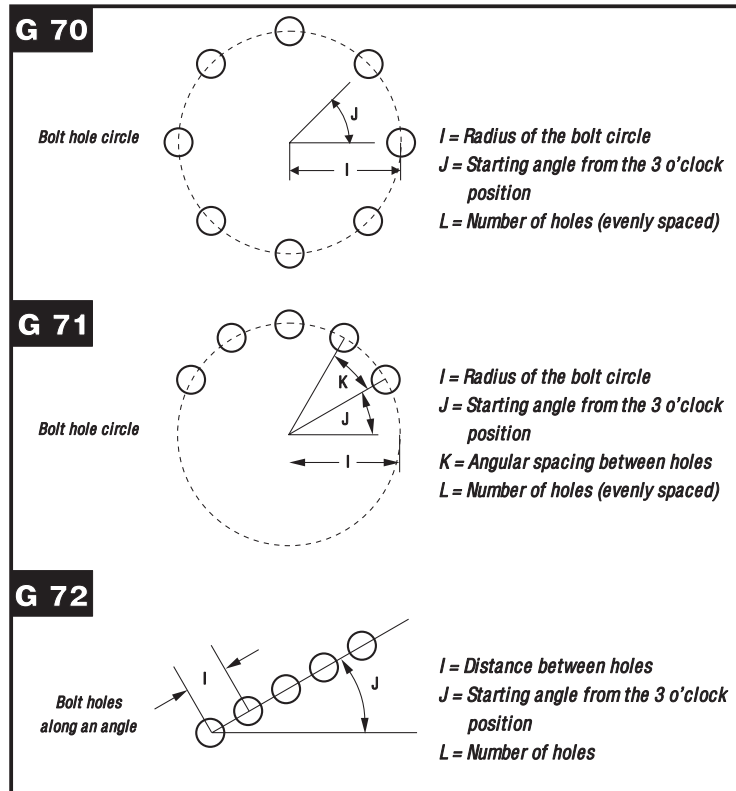


G72 Bolt Holes Along An Angle

Group 00

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

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



**CANNED CYCLES (G73, G74, G76, G81, G82, G83, G84, G85, G86, G87, G88, G89)**

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 X-Y motions are executed as rapid commands (G00) and the canned cycle operation is performed after the X-Y motion. There are six operations involved in every canned cycle:

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

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

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

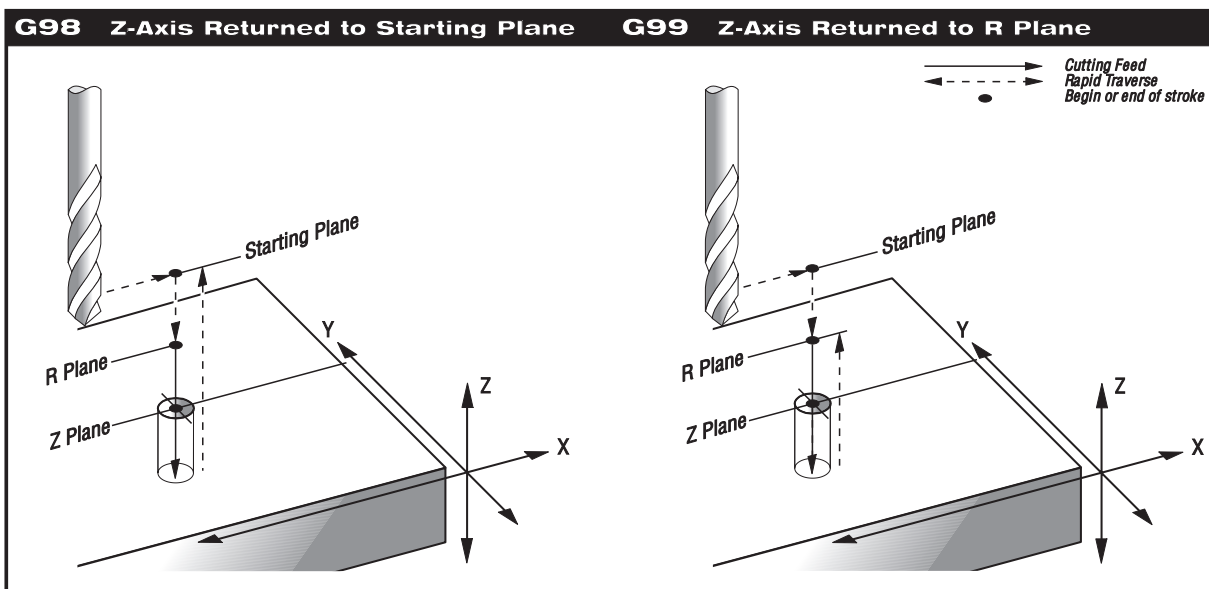
G Code	Z Drilling direction	Operation at bottom of hole	Retraction Z Direction	Application
G73	intermittent feed	none	rapid	high speed peck drilling
G74	feed	spindle CW	feed	left hand tapping
G76	feed then stop	orient spindle	rapid	fine boring
G81	feed	none	rapid	spot drilling
G82	feed	dwel	rapid	counter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

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

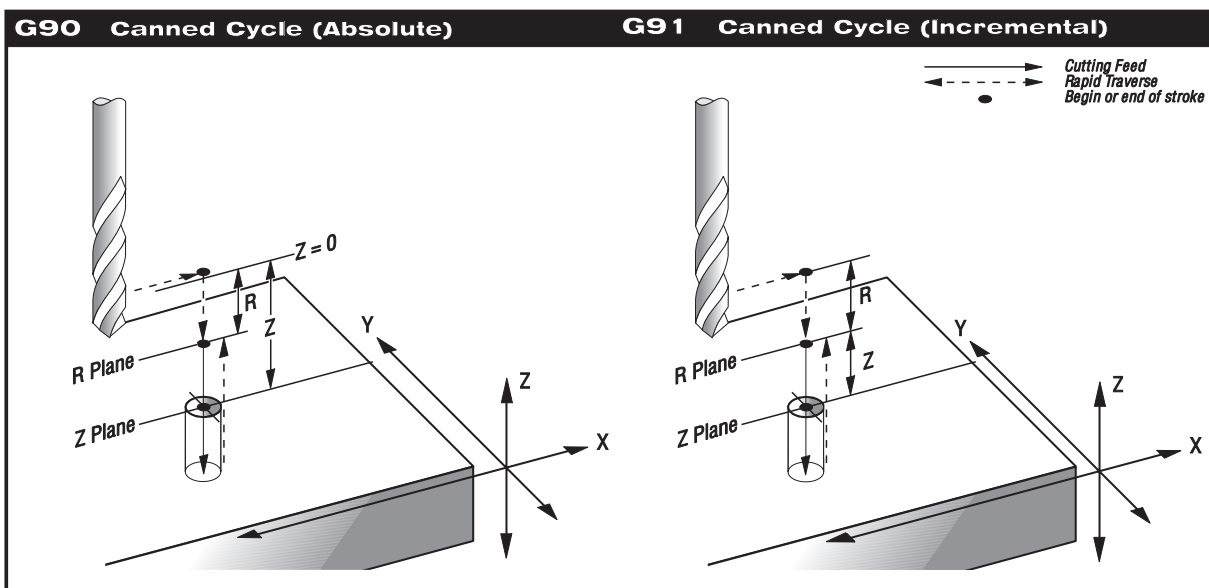


If a canned cycle is defined in a block without an **X** or **Y** motion, there are two common actions taken by other controls; some will execute the canned cycle at that time and some will not. With the VF Series Mill, these two options are selectable from Setting 28. In addition to this, if a canned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

Incremental motion in a canned cycle is often useful as a loop (L) count, it 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 in subsequently listed blocks. 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.

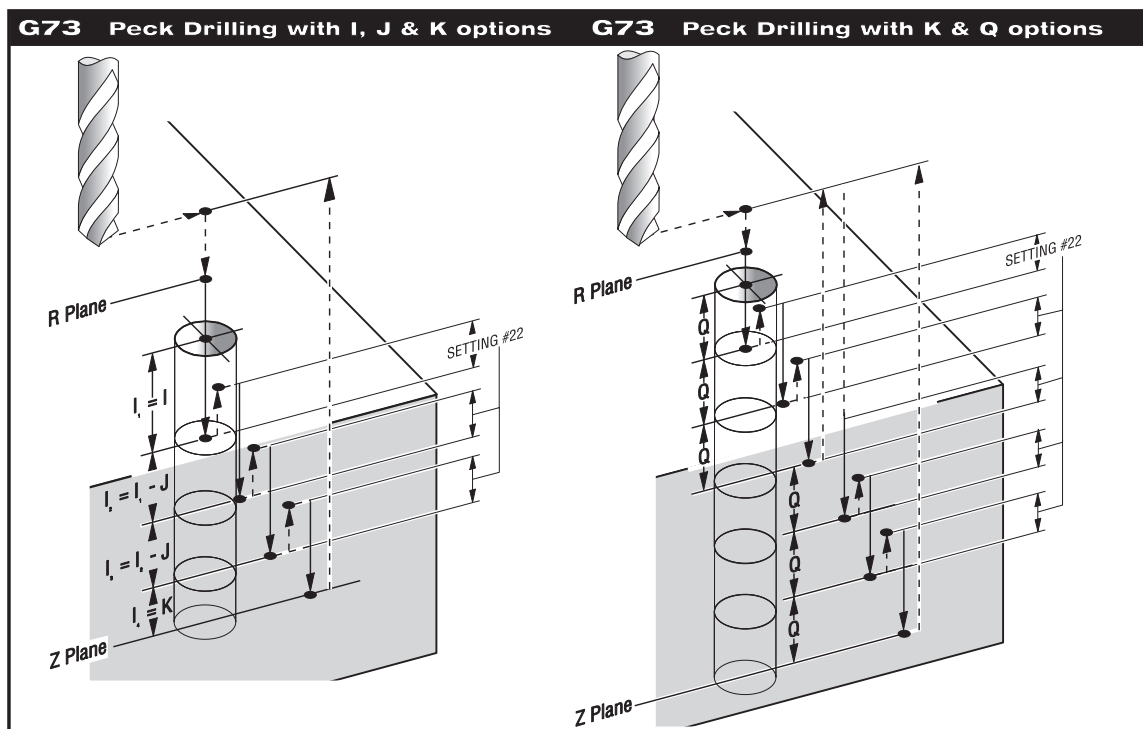
If these are listed in a block with an X-Y, the X-Y move is done and all subsequent canned cycles are performed with the new **R** or **Z** value.

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 cycles.

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 / number of pecks between retract
- L Number of repeats
- P Optional pause at end of last peck, in seconds
- 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





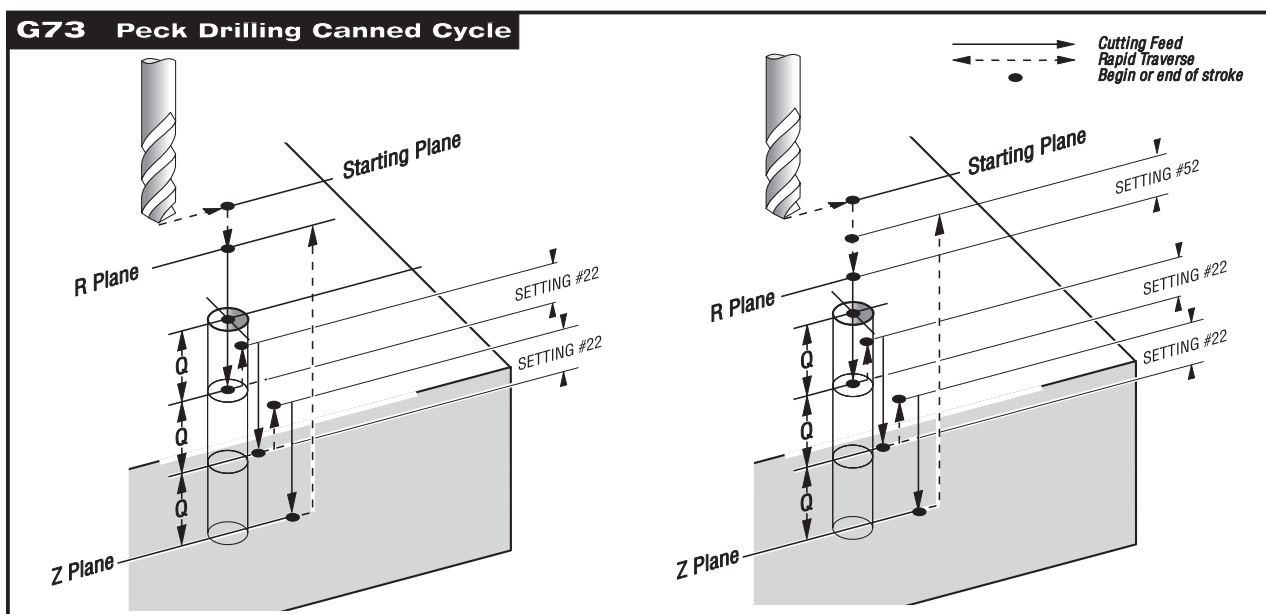
This **G** code is modal in that once activated, every motion of **X** and/or **Y** will cause this canned cycle to be executed until it is canceled or another canned cycle is selected. This cycle is a high speed peck cycle where the retract distance is set by Setting 22.

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 **P** is specified, the tool will pause at the bottom of the hole after the last peck for that amount of time.

The same dwell time applies to all subsequent blocks that do not specify a dwell time. When the canned cycle is cancelled (i.e. G00, G01, G80, RESET) the dwell time will be reset to zero. This dwell cannot be used in the same block as an M97, M98, M99, or G65, because these codes use **P** for different purposes.

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.



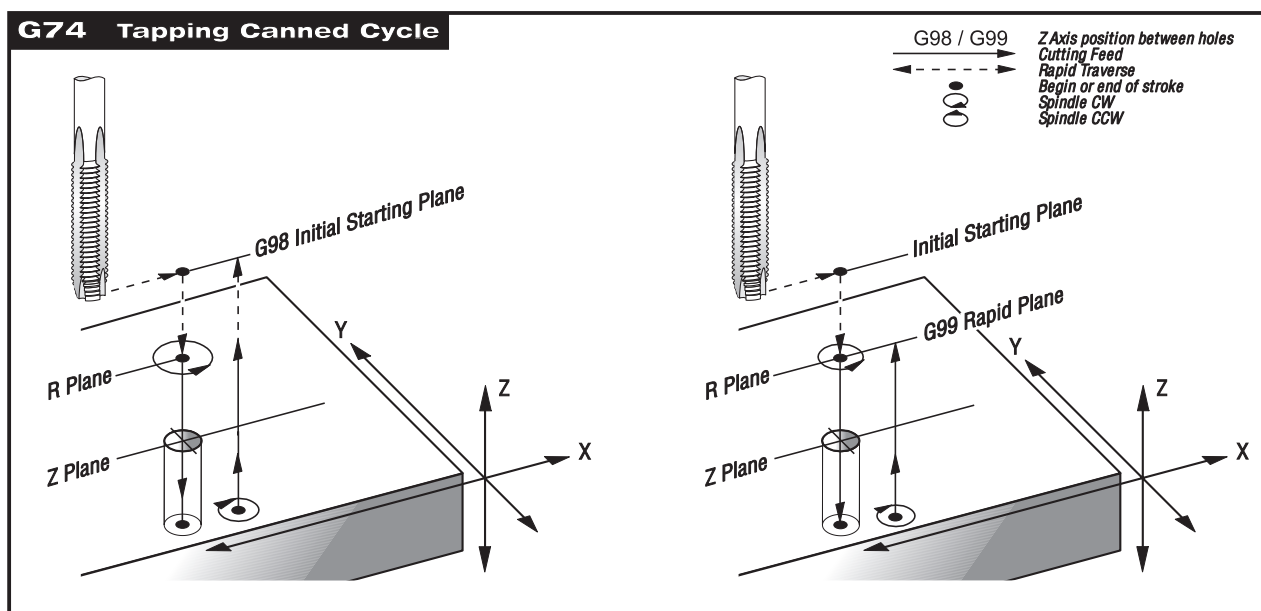
Setting 52 also changes the way G73 works when it returns to the **R** plane. Most programmers set the **R** plane well above the cut to ensure that the chip clear motion actually allows the chips to get out of the hole but this causes a wasted motion when first drilling through this “empty” space. If 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.

**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 tap

This **G** code is modal in that once activated, every motion of **X** and/or **Y** will cause this canned cycle to be executed until it is canceled or another canned cycle is selected. Note that operation of this cycle is different if the rigid tapping option is installed and selected. 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.

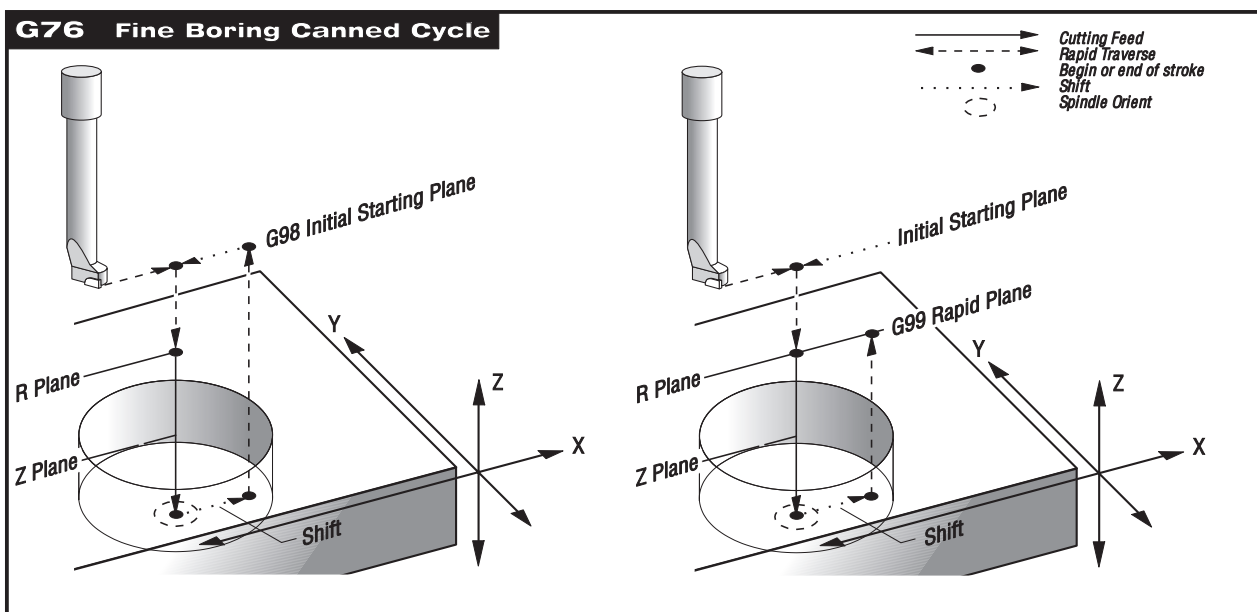




G76 Fine Boring Canned Cycle

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 tap



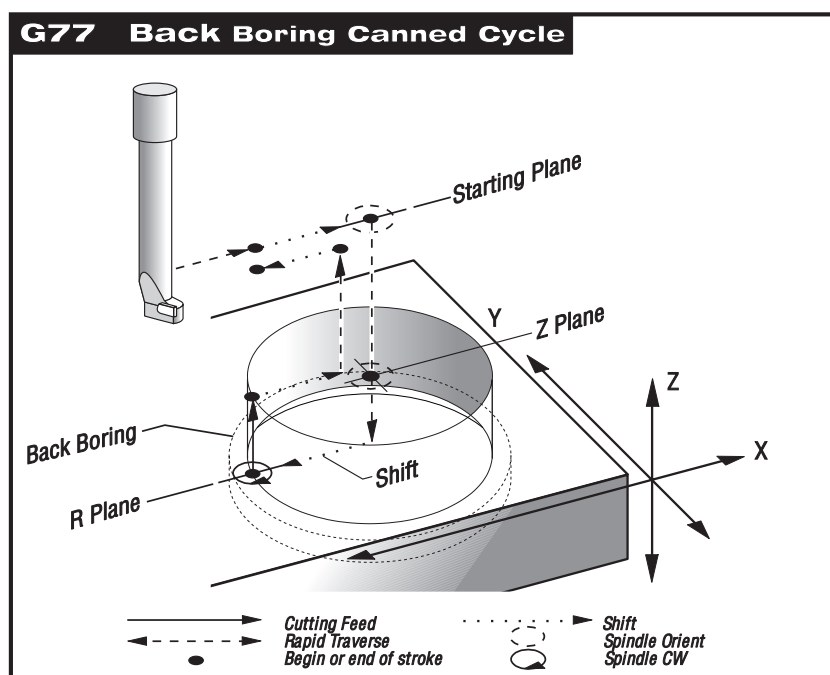
This **G** code is modal in that once activated, every motion of **X** and/or **Y** will cause this canned cycle to be executed until it is canceled or another canned cycle is selected. This cycle will shift the **X** and/or **Y**-axis prior to retracting in order to clear the tool while exiting the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

**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 once activated, every motion of **X** and/or **Y** will cause this canned cycle to be executed until it is canceled or another canned cycle is selected. This cycle will shift the X and/or Y axis prior to and after cutting in order to clear the tool while entering and exiting the part. If Setting 57 is on, the tool will perform an exact stop between rapids. This will prevent breaking a tool, or any nicking at the bottom of the hole. 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

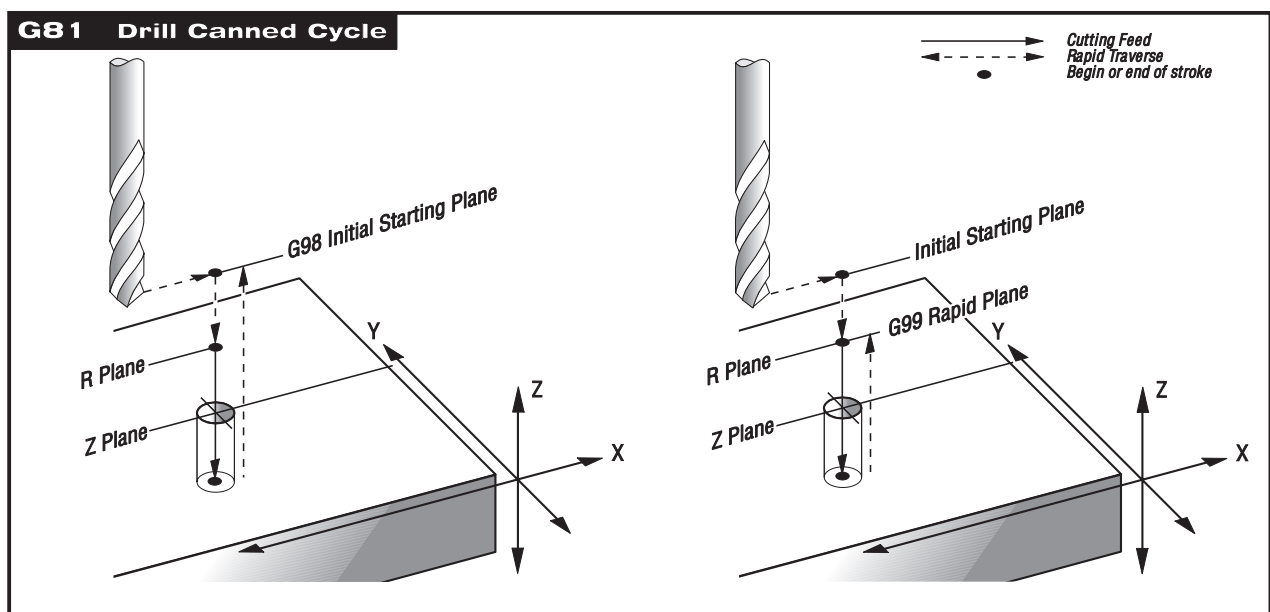
Group 09

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

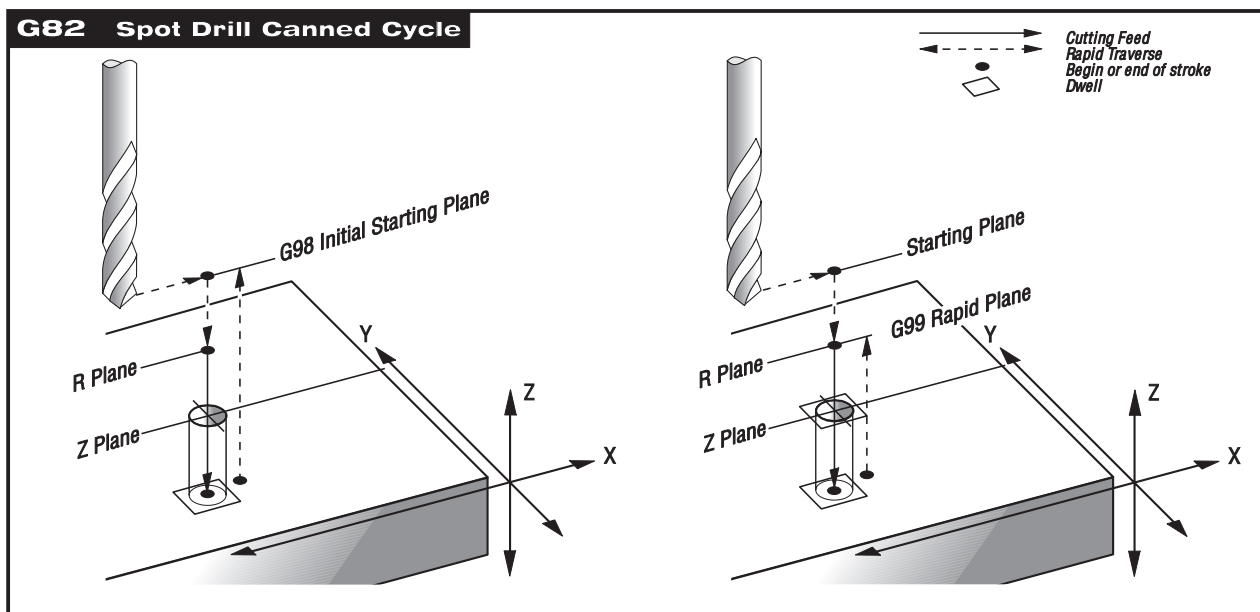


This **G** code is modal in that once activated, every motion of **X** and/or **Y** will cause this canned cycle to be executed until it is canceled or another canned cycle is selected.

**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 once activated, every motion of **X** and/or **Y** will cause this canned cycle to be executed until it is canceled or another canned cycle is selected.





G83 Normal 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
P	Optional pause at end of last peck, in seconds
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

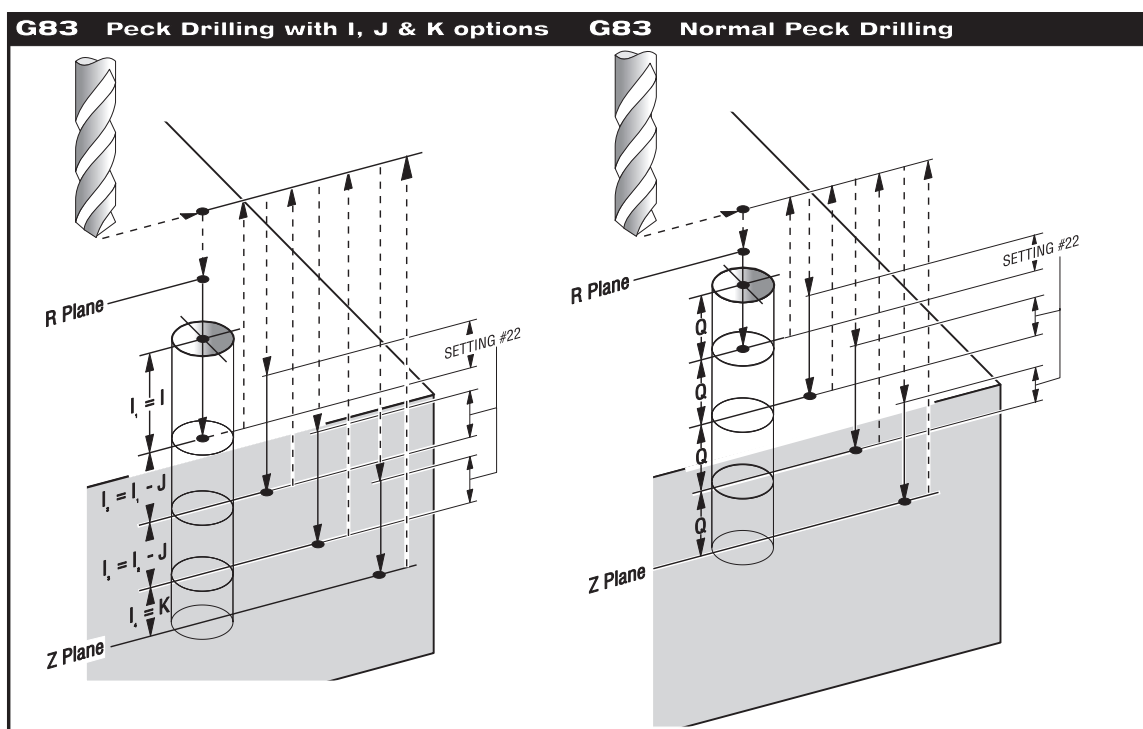
This **G** code is modal in that once activated, every motion of **X** and/or **Y** will cause this canned cycle to be executed until it is canceled or another canned cycle is selected.

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 **P** is specified, the tool will pause at the bottom of the hole after the last peck for that amount of time. The following example will peck several times and dwell for five seconds at the end:

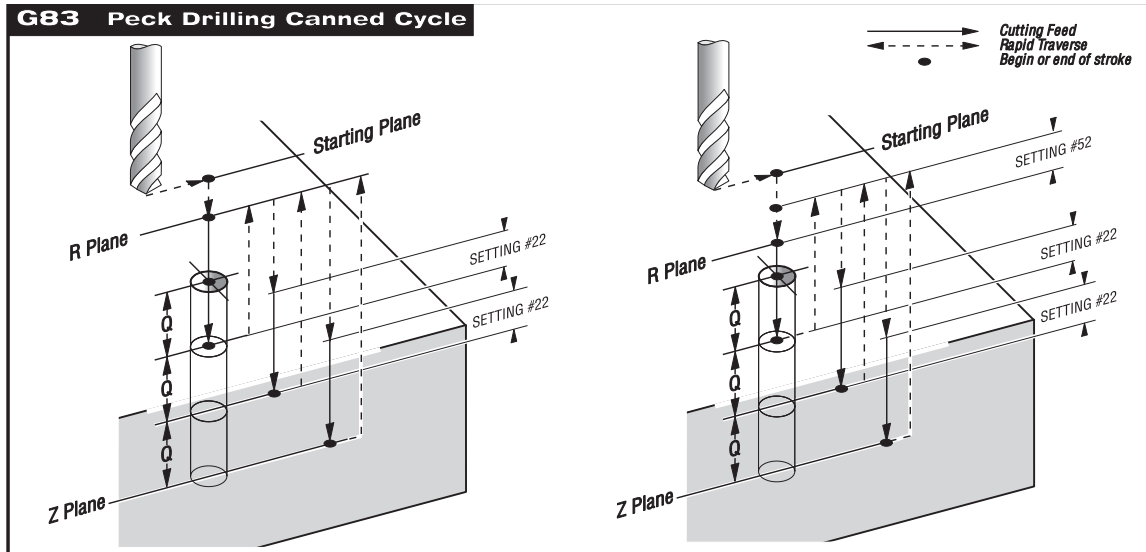
G83 Z-0.62 F15. R0.1 Q0.175 P5.

The same dwell time applies to all subsequent blocks that do not specify a dwell time. When the canned cycle is cancelled (i.e. G00, G01, G80, RESET) the dwell time will be reset to zero. This dwell cannot be used in the same block as an M97, M98, M99, or G65, because these codes use **P** for different purposes.





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.

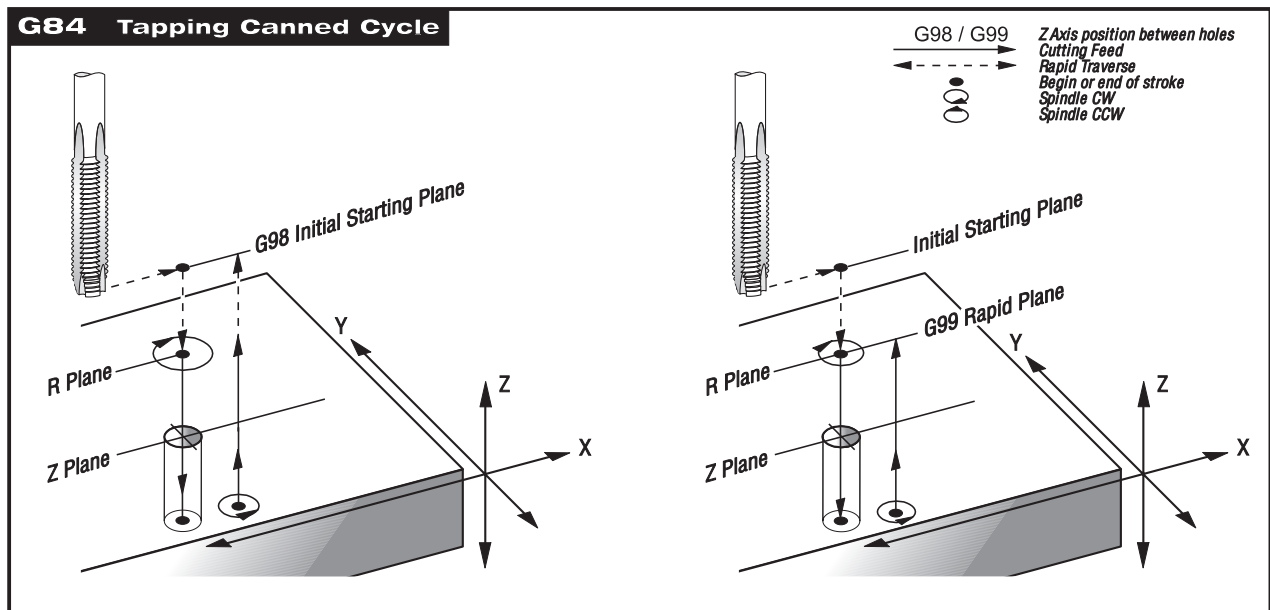




G84 Tapping Canned Cycle

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 tap

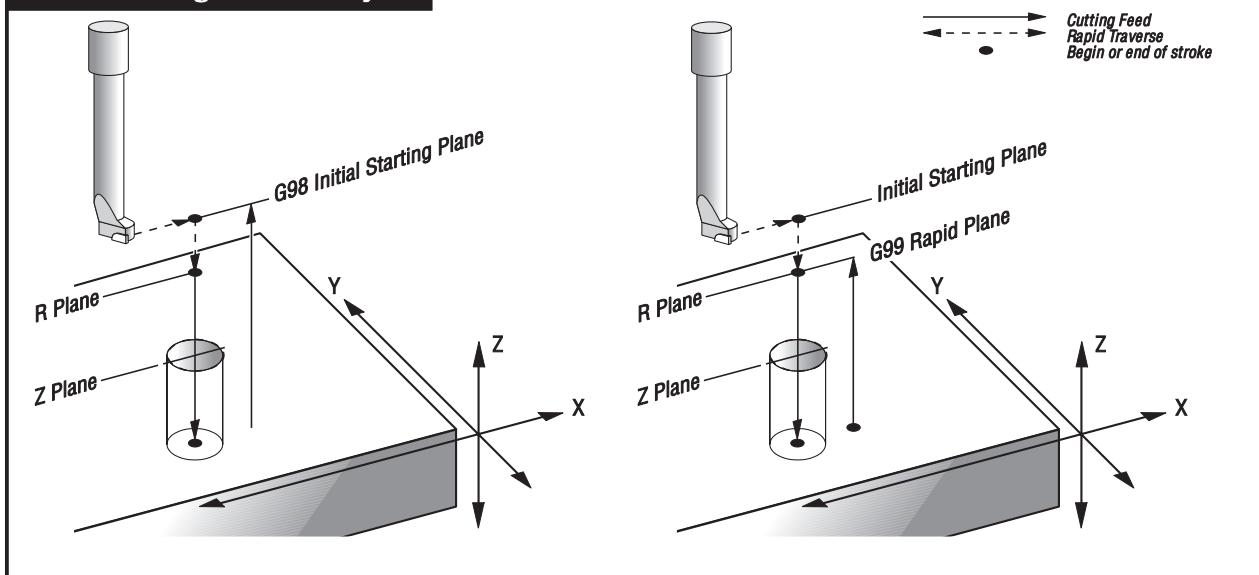


This **G** code is modal in that once activated, every motion of **X** and/or **Y** will cause this canned cycle to be executed until it is canceled or another canned cycle is selected. Note that operation of this cycle is different if the rigid tapping option is installed and selected (See Section 7.2). 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

G85 Boring Canned Cycle

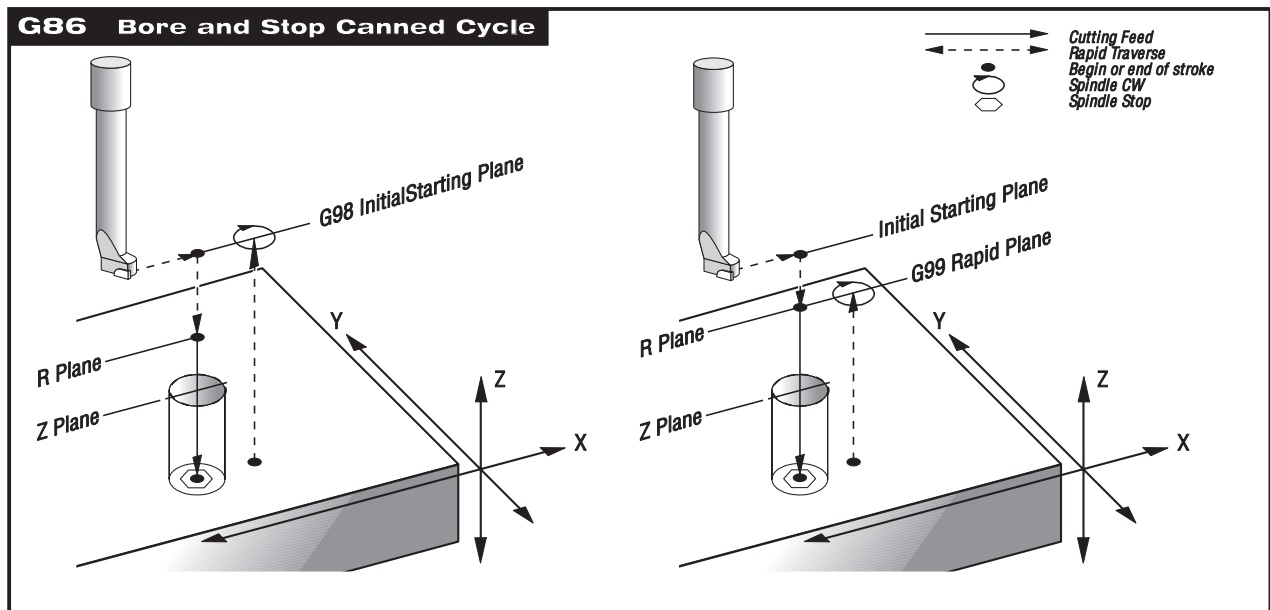
This **G** code is modal in that once activated, every motion of **X** and/or **Y** will cause this canned cycle to be executed until it is canceled or another canned cycle is selected.



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

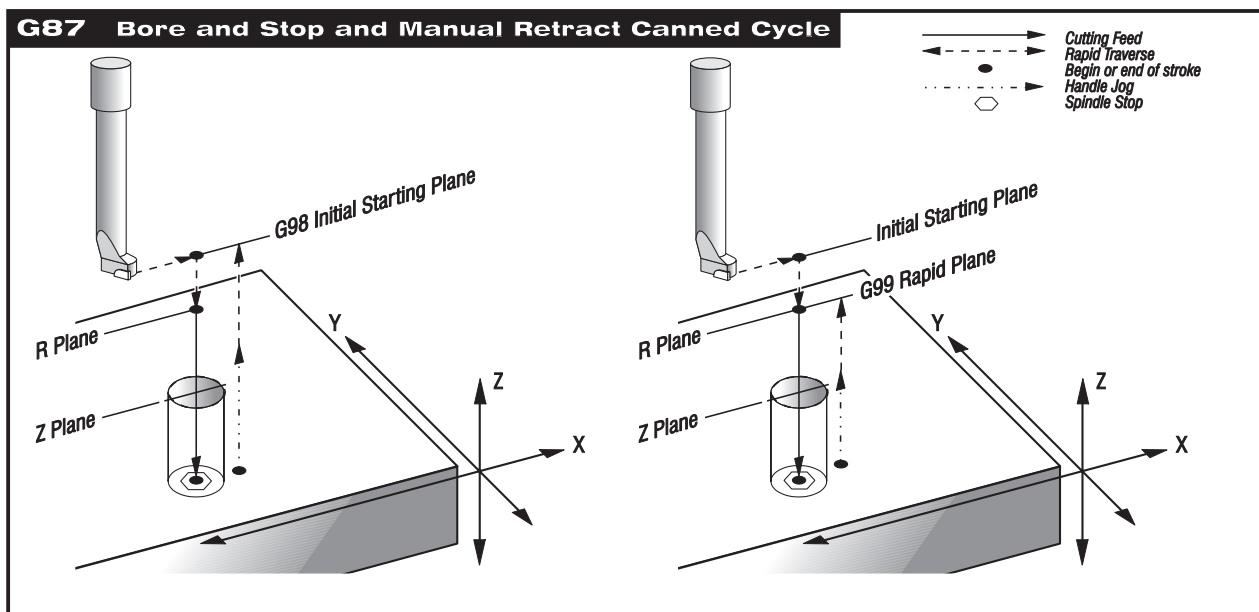


This **G** code is modal in that once activated, every motion of **X** and/or **Y** will cause this canned cycle to be executed until it is canceled or another canned cycle is selected.

**G87 Bore and Stop 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 once activated, every motion of **X** and/or **Y** will cause this canned cycle to be executed until it is canceled or another canned cycle is selected. After manual jog press cycle start to resume program.



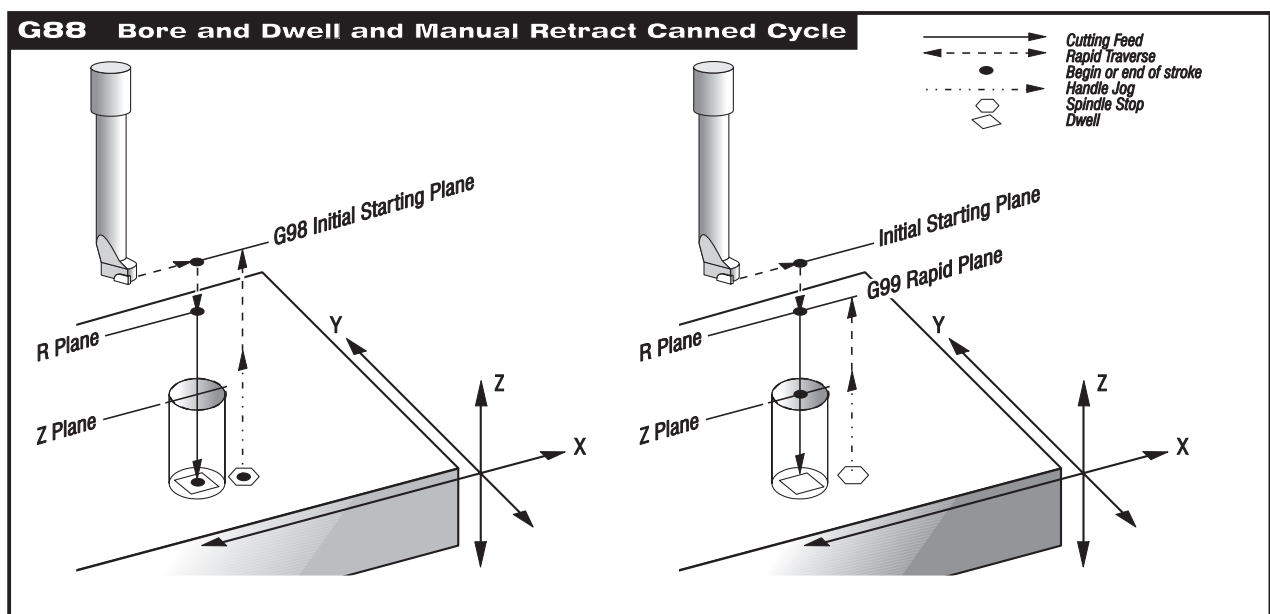


G88 Bore and Dwell and Manual Retract Canned Cycle

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

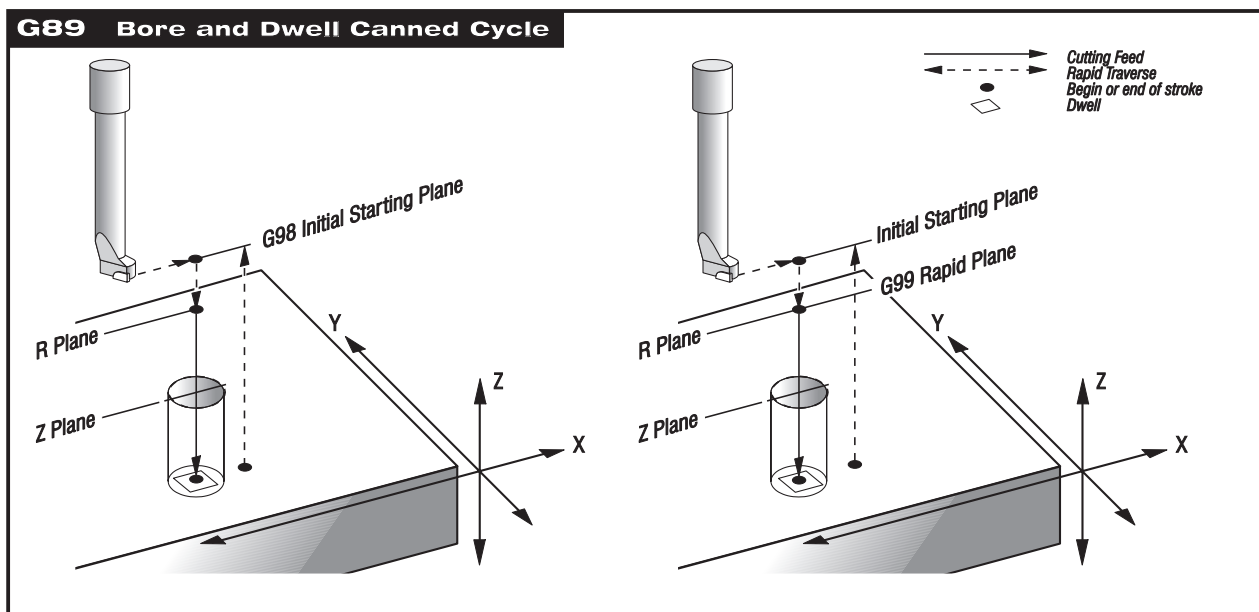
This **G** code is modal in that once activated, every motion of **X** and/or **Y** will cause this canned cycle to be executed until it is canceled or another canned cycle is selected. After manual jog press cycle start to resume program.



**G89 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 once activated, every motion of **X** and/or **Y** will cause this canned cycle to be executed until it is canceled or another canned cycle is selected.

**ABSOLUTE / INCREMENTAL SELECTION (G90, G91)****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. This code is not compatible with G143 (5AX Tool Length Compensation).



MORE WORK COORDINATE SELECTION (G92)

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

G92 Set Work Coordinate Systems Shift Value - FANUC OR HAAS Group 00

A G92 command effectively shifts all work coordinate systems (G54-59, G110-129) so that the command position becomes the 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.

INVERSE TIME (G93, G94)

G93 Inverse Time Feed Mode Group 05

This **G** code specifies that all F (feedrate) values are to be interpreted as **strokes per minute**. This is equivalent to saying that the 'F' code value, when DIVIDED INTO 60, is the number of seconds that the motion should take to complete.

G93 activates Inverse Time Feed Mode and a G94 deactivates it.

Any interpolated motion that involves only the auxiliary axes is NOT affected by G93 - the 'F' code specified will still be interpreted as Feed per Minute.

When G93 is active, the Feed Rate specification is MANDATORY for all interpolated motion blocks. i.e.: Each non-rapid motion block MUST have its own Feed Rate specification. If it doesn't, a **"NO FEED RATE"** alarm is generated. Mixing auxiliary axes with regular axes in a G01/02/03 motion in G93 mode will generate the alarm:

"AUX AXIS IN G93 BLOCK"

* All Group 9 motion commands, as well as any G12, G13, G70, G71, G72, or G150 command, will generate a syntax alarm when in G93 mode.

* Pressing RESET will reset the machine to G94 (Feed per Minute) mode.



* The diameter settings (4th & 5th axis) are meaningless in inverse time.

* Alarm 309, "EXCEEDED MAX FEEDRATE", will not be generated by G93, because the machine will automatically be limited by the slowest axis.

G94 Feed Per Minute Mode

Group 05

This code deactivates G93 (Inverse Time Feed Mode) and returns the control to Feed Per Minute Mode. Pressing RESET returns the machine to G94 mode.

CANNED CYCLE AUXILIARY FUNCTIONS (G98, G99)

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.

PROGRAMMABLE MIRROR IMAGE (G100, G101)

G100 Cancel 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

At least one of these is required

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 they do 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. G100 or G101 by itself will have 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:**

G41 X1.0 Y1.0
 G01 X2.0 Y2.0
 G101 X0
 G00 Z1.0
 G00 X2.0 Y2.0

 G40

Correct:

G41 X1.0 Y1.0
 G01 X2.0 Y2.0
 G101 X0
 G00 Z1.0
 G00 X1.0
 G00 X2.0 Y2.0

 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.

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 affects the block in which it is programmed. This **G** code should be used in a command block without any other **G** codes and it 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.

**LIMIT BLOCK BUFFERING FUNCTION (G103)****G103 Limit Block Bufering****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 387 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.

CYLINDRICAL MAPPING (G107)**G107 Cylindrical Mapping****Group 00**

X	Optional X-axis command	A	Optional A-axis command
Y	Optional Y-axis command	Q	Optional diameter of cylindrical surface
Z	Optional Z-axis command	R	Optional radius of rotary axis

This G-code translates all programmed motion occurring in a specified linear axis into the equivalent motion along the surface of a cylinder (attached to a rotary axis). It is a Group 0 G-Code, but its default operation is subject to Setting 56 (M30 RESTORE DEFAULT G). G107 is used to either activate or deactivate cylindrical mapping.

* Any linear axis can be cylindrically mapped to any rotary axis (Only one at a time).

* Existing linear-axis G-Code programs can be cylindrically mapped without modification; all that is required is the prior execution of a single G107 command, which is either placed at the beginning of the G-Code program itself or, if Setting 56 is set to OFF, can even be specified in a previous G-Code program, provided a RESET has not been issued in the interim.



* The radius (or diameter) of the cylindrical surface can be redefined in a G-Code program, allowing cylindrical mapping to occur along surfaces of different diameters without the operator having to change the program.

* The radius (or diameter) of the cylindrical surface can either be synchronized with, or be independent of, the rotary axis diameter(s) specified in the settings page.

* G107 can also be used to set the default diameter of a cylindrical surface, independently of any cylindrical mapping that may be in effect.

G107 DESCRIPTION

1) Three optional parameters (Address codes) can accompany G107:

X, Y, or Z - Either an X, Y, or Z address parameter can be specified. Its presence denotes that cylindrical mapping is to be initiated, and specifies the linear axis that will be mapped to the specified rotary axis (A or B). Any associated address value for this parameter will be ignored.

If one of these linear axes are specified, a rotary axis must also be specified, or an alarm will be generated.

A or B - Either an A or a B address parameter can be specified, and must be specified if any of the other parameters are specified. It serves to identify which rotary axis is being used to house the cylindrical surface; any associated address value for this parameter will be ignored.

Q - Diameter of the cylindrical surface. If this parameter is specified, a rotary axis must also be specified, or an alarm will be generated at run time. If neither this parameter or the R parameter are specified, then the diameter assumed for the cylindrical surface will be whatever was last specified for this rotary axis in a previously issued G107 since the machine was powered up.

If no G107 was issued since the machine was powered up, or if the last value specified was zero, then the diameter will be whatever is currently defined in the CNC Settings for this rotary axis ("4th Axis Diameter", etc.)

The value specified for this parameter will become the new G107 default for the specified rotary axis until the machine is turned off.

If this parameter was not specified in a previous G107 since the machine was powered up, or if its last specified value was zero, or if its last specified value was numerically equal to that defined for the rotary axis in the CNC Settings page, then any change made to the rotary axis diameter in the CNC Settings page will also change the default value for this parameter. In such a situation, the default G107 diameter is said to be "in sync" with the CNC Settings.

If the last non-zero G107 diameter value specified since the machine was powered up was not numerically equal to the Settings page value however, then the G107 diameter is said to be "out of sync" with the Settings page, and the default value thus established will be retained regardless of any changes made to the Settings page, until or unless they become equal in value.

This means that the rotary axis diameter as specified in the CNC Settings page is always used as the default cylindrical surface diameter, unless a previously issued G107 has established a different value. This allows the default cylindrical surface diameter to either be synchronized with, or operate independently from, the rotary axis diameter defined in the CNC Settings.



To force the default cylindrical surface diameter to be “in sync” with the Settings page Rotary Axis Diameter, which may be desirable if the current default cylindrical surface diameter status is unknown, the following G107 command can be used:

G107 A0 Q0

This command will reset the A-axis default cylindrical surface diameter to zero, which forces the Settings page value to be used as the default for that axis.

R - This parameter specifies the radius of the rotary axis. If Q has already been specified, the R will be ignored. Except as noted above, the conditions and/or restrictions associated with the use of this parameter are exactly the same as the Q parameter.

- 2) The above parameters can be entered in any order.
- 3) Pressing the RESET key will always turn off any Cylindrical Mapping that is currently in effect, regardless of the status of Setting 56. Cylindrical Mapping will also be turned off automatically whenever the G-Code program ends, but only if Setting 56 is set to ON. If Setting 56 is set to OFF, any Cylindrical Mapping that is in effect when the program ends will remain in effect.
- 4) A G107 code issued without any parameters will turn off any cylindrical mapping currently in effect. This should normally be done before a program ends. If Setting 56 is set to OFF however, it may be desirable not to turn Cylindrical Mapping off, so that the mapping established can be used by other programs.
- 5) Re-issuing a G107 with either the same or a different linear axis specification will turn off any cylindrical mapping that is currently in effect, before the new mapping is initiated.
- 6) Issuing a G107 code with only a rotary axis specification and either a Q or an R will only change the default diameter for the specified rotary axis; it will not turn on or off any cylindrical mapping.

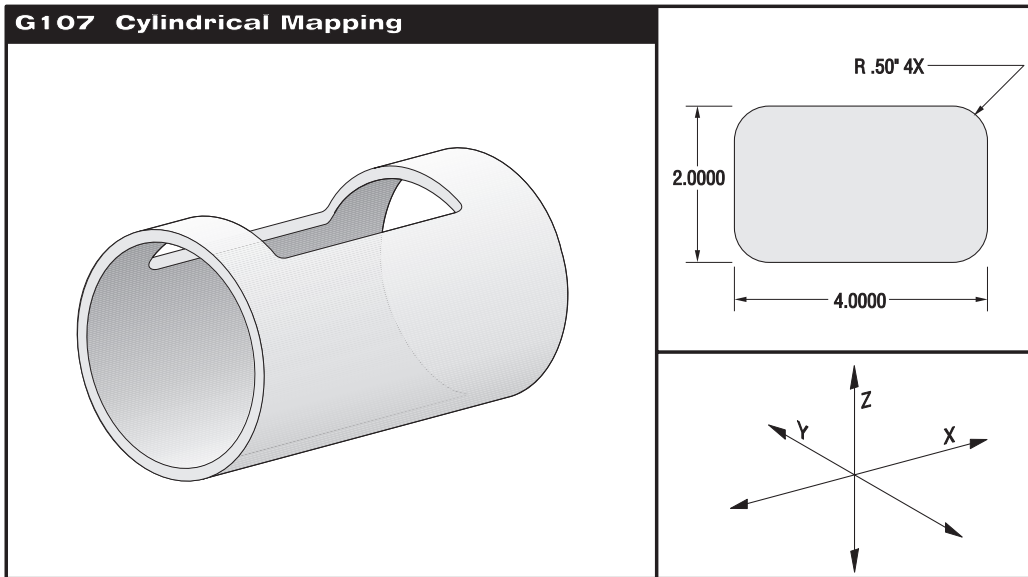
Note: Due to the fact that a rotary axis's maximum acceleration (as parameterized in steps/seconds/second in the parameters page) is different from a linear axis's maximum acceleration, there is a maximum achievable Feed Rate that can be programmed for a Circular Interpolated Move, when one of the Circular axes has been Cylindrically Mapped. Refer to the next section for a description of FEED RATE ALTERATION.

FEED RATE ALTERATION TO ENSURE CONCURRENT AXIS MOTION

The actual FeedRate achieved in any interpolated motion in the CNC Mill is limited by the maximum speed achievable in each axis of motion. The FeedRate may be lowered from its programmed value if that value would cause the Mill to command an axis to move **faster** than its parameterized maximum speed.

This means that the actual FeedRate achieved may be less than or equal to the programmed FeedRate, ensuring that all axes of interpolated motion move concurrently. When this condition occurs, the warning 'LIM' will be displayed on the screen immediately to the left of the 'FEED' display.

Note also that the **MAX FEED RATE EXCEEDED** alarm will still be issued if the programmed feed rate exceeds parameter 59 MAX FEEDRATE, (currently 300 ipm), but **ONLY** if the mill is not in Inverse Time mode.



SAMPLE PART PROGRAM

```

%
O0079 (G107 TEST)
G00 G40 G49 G80 G90
G28 G91 A0
G90
G00 G54 X1.75 Y0 S5000 M03
G107 A0 Y0 R2. (IF NO R OR Q VALUE, MACHINE WILL USE VALUE IN SETTING 34)
G43 H01 Z0.25
G01 Z-0.25 F25.
G41 D01 X2. Y0.5
G03 X1.5 Y1. R0.5
G01 X-1.5
G03 X-2. Y0.5 R0.5
G01 Y-0.5
G03 X-1.5 Y-1. R0.5
G01 X1.5
G03 X2. Y-0.5 R0.5
G01 Y0.
G40 X1.75
G00 Z0.25
M09
M05
G91 G28 Z0.
G28 Y0.
G90
G107
M30
%
```

**MORE WORK COORDINATE SELECTION (G110-G129)****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.

COMPENSATION (G141, G143)**G141 3D+ CUTTER COMPENSATION****Group 07**

This feature performs 3d+ cutter diameter compensation. The form is:

G141 Xnnn Ynnn Znnn Dnn Innn Jnnn Knnn

Subsequent lines can be of the form:

G01 Fnnn Xnnn Ynnn Znnn Innn Jnnn Knnn

or:

G00 Xnnn Ynnn Znnn Innn Jnnn Knnn

The 3d+ G141 cutter compensation is not just for 5 axes work. Any CAD system can output the I, J, K values to shift the tool by the amount in the offsets memory of the control, even if the motions are only in 2 or 3 axes. In the Haas version, only G00 and G01 will get G141 cutter compensation. No other functions or canned cycles will get the offset. G91 incremental motion also cannot be used. The G141 is used to indicate without any doubt, what type of compensation is being requested. G40 will cancel 3d+ cutter compensation. The Dnn code selects which radius of diameter offset to use. G141 is modal with G40, G41, and G42. Inverse time is usually used with this type of motion but is not required. The I, J, and K values point in the direction that the cutter compensation is to be applied. When G141 is active, commanded motion of X, Y, or Z will have a vector component of the tool diameter added to the motion according to the direction vector defined by I, J, and K. For example:

```
T1 M06
G00 G90 G54 X0 Y0 Z0 A0 B0
G141 D01 X0.Y0. Z0. (RAPID POSIT WITH 3 AX C COMP)
G01 G93 X.01 Y.01 Z.01 I.1 J.2 K.9747 F300. (FEED INV TIME)
X.02 Y.03 Z.04 I.15 J.25 K.9566 F300.
X.02 Y.055 Z.064 I.2 J.3 K.9327 F300
.
.
.
X2.345 Y.1234 Z-1.234 I.25 J.35 K.9028 F200. (LAST MOTION)
G94 F50. (CANCEL G93)
G0 G90 G40 Z0 (RAPID TO ZERO, CANCEL 3 AXIS C COMP)
X0 Y0
M30
```

Note: G141 is a group 7 G code, G40 cancels G141, G91 is not compatible with G141, G141 uses a D code



G143 5 Axes Tool Length Compensation +

Group 08

(This G-code is optional it is only used on the bridge mill and VR-11)

This allows the user to correct for variations in the length of cutting tools without the need to revert to CAD/CAM or post- processing steps. 5 axes length compensation applies only to machines where all rotary motion is movement of the cutting tool. It does not apply to machines where any of the rotary axes involve motion of the part or fixture. An H code is required to select the tool length from the existing length compensation tables. Selecting G49 or H00 will cancel 5 axes compensation. If only normal (Z axes) cutter length compensation is desired, simply program G43 or G44. For G143 to work correctly, there must be two rotary axis, A and B. G90 must be active for absolute positioning mode and G91 must not be used. Work position 0,0 for the A and B axes must be such that the tool is parallel with Z axes motion.

G143 tool length compensation works only with rapid (G00) and feed (G01) motions. No other feed functions (G02 or G03) or canned cycles (drilling, tapping, etc.) can be used. When G143 is active, a commanded motion of X,Y, or Z will have a vector component of the tool length added to the motion according to the work coordinates of the A and B axes. For a positive length of the tool, this will move Z upward or in the + direction. If one of X,Y, or Z is not programmed, there will be no motion of that axes, even if the motion of A or B causes a new vector for the tool length. Thus a typical program would use all 5 axes on one block of data.

Feed rate in G143 is complicated by the vector offset added to XYZ. Thus inverse feed (G93) is strongly recommended. If the inverse feed rate was correct before applying G143, it will still be correct with any length of compensation unless the maximum speed of an axes is exceeded. An example follows:

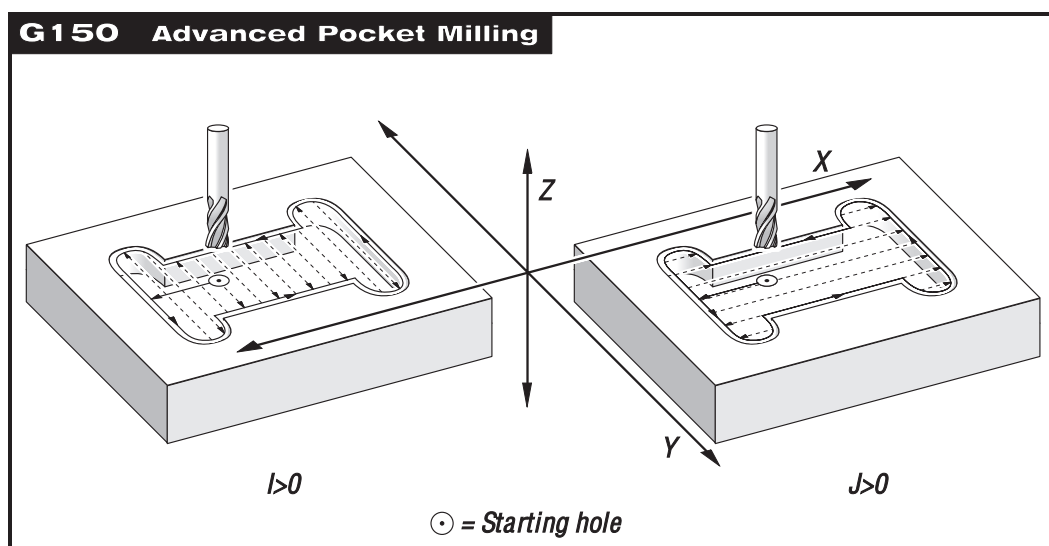
```
T1 M06
G00 G90 G54 X0 Y0 Z0 A0 B0
G143 H01 X0. Y0. Z0. A-20. B-20. (RAPID POSIT W. 5AX COMP)
G01 G93 X.01 Y.01 Z.01 A-19.9 B-19.9 F300. (FEED INV TIME)
X0.02 Y0.03 Z0.04 A-19.7 B-19.7 F300.
X0.02 Y0.055 Z0.064 A-19.5 B-19.6 F300
X2.345 Y.1234 Z-1.234 A-4.127 B-12.32 F200. (LAST MOTION)
G94 F50. (CANCEL G93)
G0 G90 G49 Z0 (RAPID TO ZERO, CANCEL 5 AXS COMP)
X0 Y0
M30
```

5 Axes Tool Length Compensation is canceled by G49

**GENERAL PURPOSE POCKET MILLING FUNCTION (G150)****G150 General Purpose Pocket Milling****Group 00**

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

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



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



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

Pocket milling should begin from a hole which has been previously drilled to the **Z** depth in order to clear the tool on entry to the pocket. The G150 block must specify this hole location with **X** and **Y**.

The first motion in the subroutine should move from this clear hole to the starting point of the block shape. The final motion in the subroutine should return to the same point as the starting motion of the subroutine, ie., in the example on the previous page, the start point of the G150 line is X3.25 Y4.5 and the first move of the 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. D02 J0.35 K.01 Q0.5 R.1 X3.25 Y4.5 Z-2. P200
G40 X3.25 Y.45
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)
```

**RIGID TAPPING****G174/ G184 General-Purpose Rigid Tapping** (This G-code is optional and requires a probe)**Group 00**

F	Feed rate in inches per minute
X*	Optional X position at bottom of hole
Y*	Optional Y position at bottom of hole
Z*	Optional Z position at bottom of hole

This G code is used to perform rigid tapping for non-vertical holes. It may be used with a right-angle head to perform rigid tapping in the X or Y axis on a three axis mill, or to perform rigid tapping along an arbitrary vector with a five-axis mill. When performing a five-axis rigid tap, the machinist must ensure that the head is positioned correctly before the G184 command is given. If the head is not aligned with the direction of motion, the tool will break. The ratio between the feed rate and spindle speed must be precisely the thread pitch being cut. Because only the end point of the tap is specified with this canned cycle, it is not modal. A motion to the next starting position will normally occur between G174/G184 canned cycles. G184 commands clockwise motion of the spindle during entry, and G174 command counter-clockwise motion. You do not have to start the spindle before this canned cycle.

IN POSITION ACCURACY (G 187)**G187 Accuracy Control****Group 00**

Programming G187 is as follows:

```
G187 E0.01          (to set value)
G187                (to revert to setting 85 value)
```

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



6. M CODES (MISCELLANEOUS FUNCTIONS)

M CODE SUMMARY

Only one **M** code may be programmed per block of a program. All **M** codes are effective or cause an action to occur at the end of the block. However, when Parameter 278 bit "CNCR SPINDLE" is set to 1, an M03 (spindle start) will occur at the beginning of a block.

M00	Stop Program
M01	Optional Program Stop
M02	Program End
M03	Spindle Forward
M04	Spindle Reverse
M05	Spindle Stop
M06	Tool Change
M08	Coolant On
M09	Coolant Off
M10	Engage 4th Axis Brake
M11	Release 4th Axis Brake
M12	Engage 5th Axis Brake
M13	Release 5th Axis Brake
M16	Tool Change (same as M06)
M19	Orient Spindle
M21-M24	Optional Pulsed User M Function with Fin
M30	Prog End and Rewind
M31	Chip Conveyor Forward
M32	Chip Conveyor Reverse
M33	Chip Conveyor Stop
M34	Increment Coolant Spigot Position
M35	Decrement Coolant Spigot Position
M36	Pallet Rotate
M39	Rotate Tool Turret
M41	Low Gear Override
M42	High Gear Override
M51-M54	Set Optional User M
M61-M64	Clear Optional User M
M75	Set Measure point
M76	Disable Displays
M77	Enable Displays
M78	Alarm if skip signal found
M79	Alarm if skip signal not found
M82	Tool Unclamp
M86	Tool Clamp
M88	Through the Spindle Coolant ON
M89	Through the Spindle Coolant OFF
M95	Sleep Mode
M96	Jump if no Input
M97	Local Sub-Program Call
M98	Sub Program Call
M99	Sub Program Return Or Loop

**M CODE DETAILED DESCRIPTION****M00 Stop Program**

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

M01 Optional Program Stop

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

M02 Program End

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

M03 Spindle Forward

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

When Parameter 278 bit 10, "CNC SPINDLE", is set to 1, the machine will not wait for the spindle to come up to speed immediately after an M03 command. Instead, it will check and/or wait for the spindle to come up to speed immediately before the next interpolated motion is initiated.

M04 Spindle Reverse

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

When Parameter 278 bit 10, "CNC SPINDLE", is set to 1, the machine will not wait for the spindle to come up to speed immediately after an M04 command. Instead, it will check and/or wait for the spindle to come up to speed immediately before the next interpolated motion is initiated.

M05 Spindle Stop

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

M06 Tool Change

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



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

M08 Coolant On

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

M09 Coolant Off

The M09 code will turn off the coolant supply.

M10 Engage 4th Axis Brake

The M10 code is used to apply the brake to the 4th axis. The brake is normally engaged, so M10 is only required when M11 is used to release the brake.

M11 activates a relay that releases the brake. M10 deactivates this relay, engaging the brake.

M11 Release 4th Axis Brake

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

M12 Engage 5th Axis Brake

The M12 code is used to apply the brake to the 5th axis. The brake is normally engaged, so M12 is only required when M13 is used to release the brake.

M13 activates a relay that releases the brake. M12 deactivates this relay, engaging the brake.

M13 Release 5th Axis Brake

The M13 code will "pre-release" the 5th axis brake. This is useful in preventing the delay that otherwise occurs when a 5th axis is used with a brake and a motion is commanded in that axis. It is not required but, without a prior M13, 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. The spindle is oriented electronically on machines with a Vector drive. Machines without a vector drive the spindle is oriented with a shot pin. A P value can be added that will cause the spindle to be oriented to a particular angle (in degrees). For example, M19 P270 will orient the spindle to 270 degrees. An R value will recognize up to four places to the right of the decimal point. An M19 R123.4567 will position the spindle to the angle specified by the R value.

M21-M24 Optional User M

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

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

M30 Prog End and Rewind

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

M31 Chip Conveyor Forward

M31 starts the chip conveyor motor in the forward direction. The forward direction is defined as the direction that the conveyor must move to transport chips out of the work cell. The conveyor will not turn if the door is open. This may be overridden by setting bit 17 of parameter 209 (CNVY DR OVRD).

M32 Chip Conveyor Reverse

M32 starts the chip conveyor motor in the reverse direction. The reverse direction is defined as the direction opposite of forward. The conveyor will not turn if the door is open. This may be overridden by setting bit 17 of parameter 209 (CNVY DR OVRD).

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 places the spigot at the most positive Z axis location the spigot can attain. Incrementing the spigot then lowers the coolant stream direction.



M35 Decrement Coolant Spigot Position

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

M36 Pallet Rotate

This is used on **Horizontal mills only**. Flashes PART READY button on the front switch box. It delays the pallet change execution until the PART READY button is depressed.

M39 Rotate Tool Turret

The M39 code is used to rotate the tool turret without performing a tool change. The desired tool pocket number (**Tn**) must be programmed previous to the M39.

This M code may be useful to move an empty pocket to face the spindle. 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.

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.

M50

Horizontal Mills Only. Signals the calling program and executes a pallet change sequence.

M51-M54 Set Optional User M

The M51 through M54 codes are optional for user interfaces. They will activate one of relays 25 through 28 and leave it active. These are the same relays used for M21-M24. Use M61-M64 to turn these off. The RESET key will turn off all of these relays. When Through the Spindle Coolant is enabled, M54 is disabled.

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



M61-M64 Clear Optional User M

The M61 through M64 codes are optional for user interfaces. They will deactivate one of relays 25 through 28. These are the same relays used for M21-M24. When Through the Spindle Coolant is enabled, M64 is disabled.

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

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. It is not necessary for machine performance.

M77 Enable Displays

This code is used to enable the updating of the screen displays. It is only used when M76 has been used to disable the displays.

M78 Alarm if Skip Signal Found

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

M79 Alarm if Skip Signal Not Found

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

M82 Tool Unclamp

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

M86 Tool Clamp

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

M88 Thru Spindle Coolant ON

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



M89 Thru Spindle Coolant OFF

This code is used to turn off the Through the Spindle Coolant (TSC) option. When M89 is called, the spindle will stop and the TSC pump will shut off. Turns on purge for the amount of time specified in parameter 237 then turns off purge.

M95 Sleep Mode

Sleep mode is essentially a long dwell but with power to the servo motors turned off. Sleep mode can be used when the user wishes his machine to begin warming itself up early in the morning and be ready for use upon his arrival. The format of the M95 command is:

M95 (hh:mm)

The comment immediately following the M95 must contain the hours and minutes that the machine is to sleep for. For example, if the current time were 6pm and the user wanted the machine to sleep until 6:30am the next morning, the following command would be used:

M95 (12:30)

Up to 99 hours can be specified thus allowing the machine to sleep for over 4 days. If the time is specified using an incorrect format, alarm 324 DELAY TIME RANGE ERROR will be generated. When the machine enters sleep mode, it will turn off power to the servos motors and display the following message:

HAAS
SLEEP MODE
REMAINING TIME nnn MIN.

The message will be re-displayed in a different position on the screen each second so that the user can see at a glance that the machine is sleeping. This has the advantage of preventing the message from being "burned" into one spot on the screen.

When less than one minute of sleep time remains, the message will change to:

REMAINING TIME nn SEC.

If the user presses any key or opens the door, sleep mode will be cancelled, power will be restored to the servo motors and the active program will wait at the block following the M95 until the user presses the Cycle Start key.

For the last 30 seconds of the sleep time, the machine will beep and display an additional message:

WAKE UP IN nn SECONDS

When the sleep time has elapsed, power will be restored to the servo motors and the active program will continue at the block following M95.

**M96 Jump If No Input**

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

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

The following is an M96 example:

```
N05 M96 P5 Q8 (TEST INPUT DOOR S, UNTIL CLOSED);  
N10 (START OF SOME PROGRAM LOOP);  
.  
(PROGRAM THAT MACHINES PART);  
.  
N85 M21 (EXECUTE AN EXTERNAL USER FUNCTION)  
N90 M96 P10 Q27 (LOOP TO N10 IF SPARE INPUT IS 0);  
N95 M30 (IF SPARE INPUT IS 1 THEN END PROGRAM);
```

M97 Local Sub-Program Call

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

M98 Sub Program Call

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

M99 Sub Program Return Or Loop

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

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



You can simulate FANUC behavior by using the following code.

calling program: HAAS	FANUC
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.



THIS PAGE INTENTIONALLY LEFT BLANK



7. PROGRAMMING EXAMPLES

G81 DRILLING CANNED CYCLE

FORMAT:

G81 Z___ F___ R___

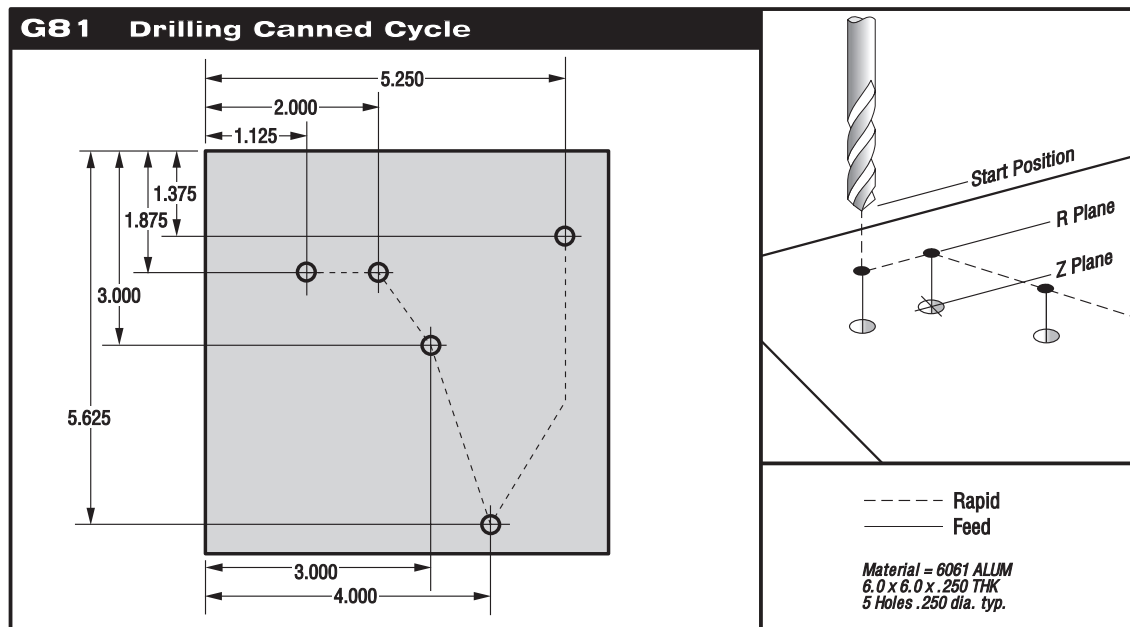
Z = Position of the bottom of the hole being drilled.

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.



Canned cycle programming example using aluminum block.

The following is the program to drill through the aluminum plate:

```

T1 M06
G00 G90 G54 X1.125 Y-1.875 S4500 M03
G43 H01 Z.1
G81 Z-.35 F45. 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
  
```

**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 USING G82, G83 AND G84

Helpful notes are listed in parentheses ().



PROGRAM EXAMPLE

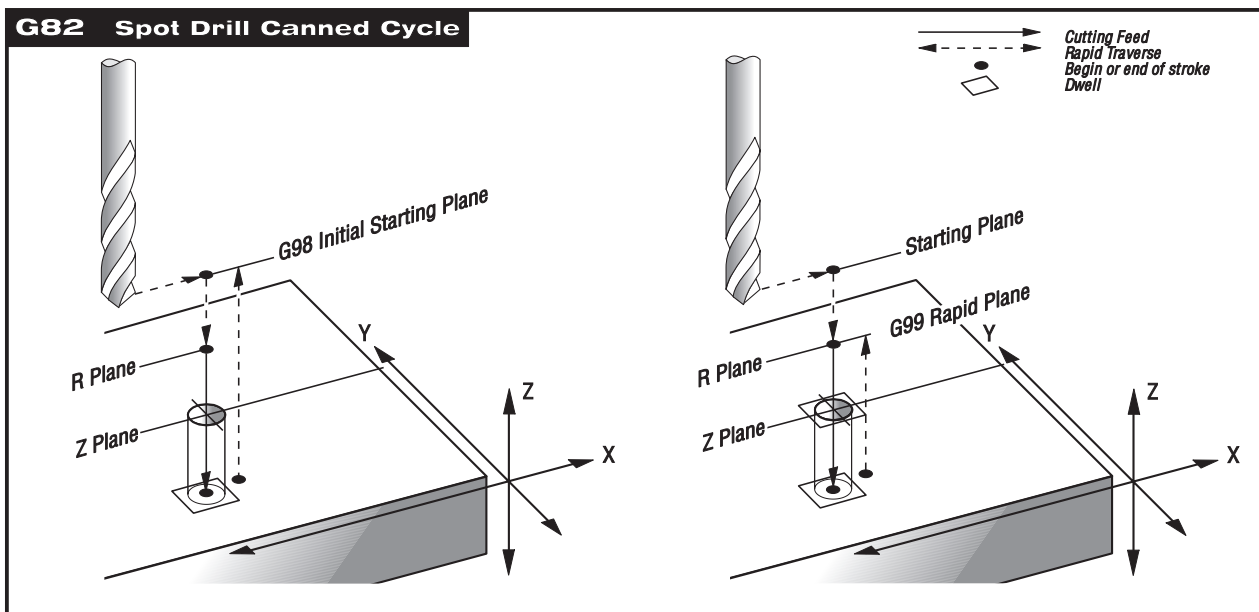
```
%
O1234
T1 M06
G00 G90 G54 X.565 Y-1.875 S1275 M03
G43 H01 Z.1 M08
G82 Z-.175 F10. R.1 P300
X1.115 Y-2.750
X3.365 Y-2.875
X4.188 Y-3.313
X5.0 Y-4.0
G00 G80 Z1.0 M09
```

DESCRIPTION

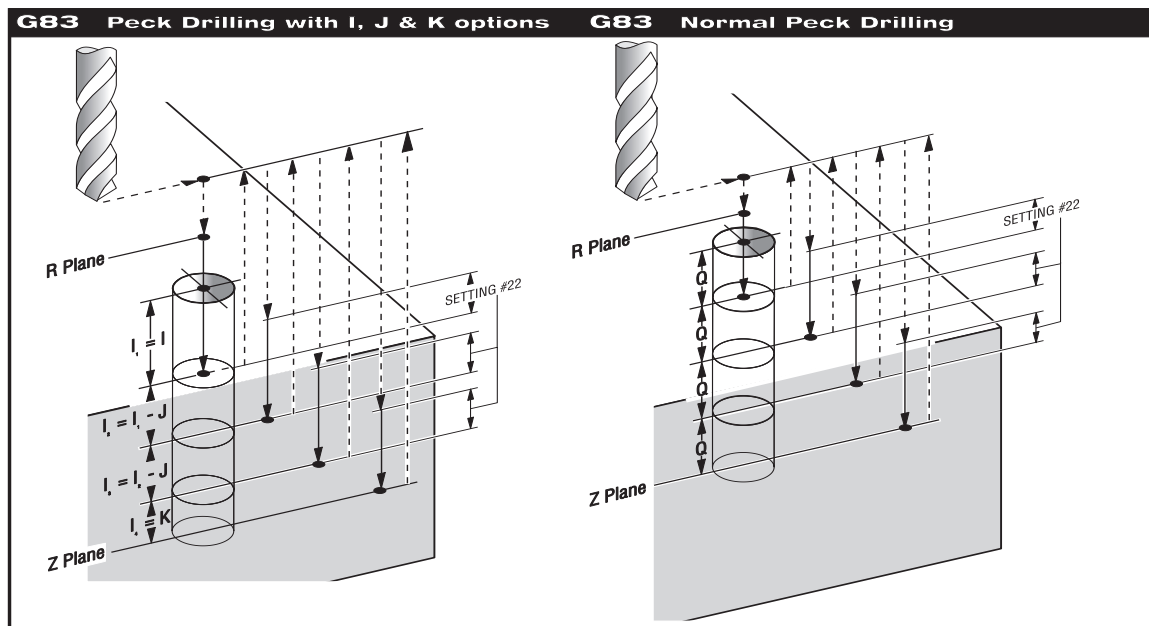
(Exercise program)

(TOOL #1 IS A .5 X 90 degree spot drill)

>(90 degree spot drill, the depth is)
(half of the chamfer diameter)



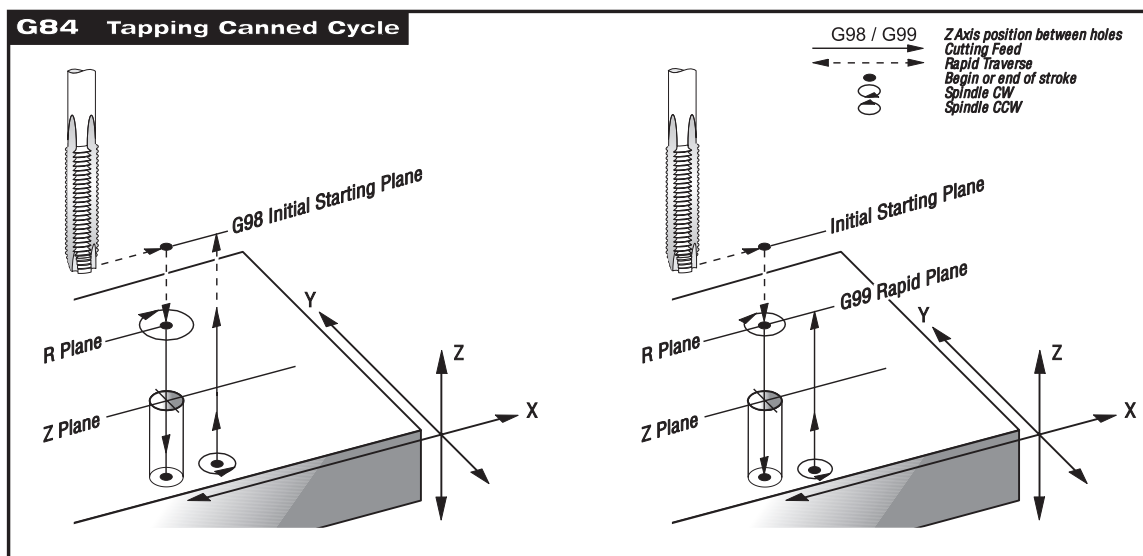
G82 Spot drilling example

*G83 Peck Drilling Canned Cycle Example*

```
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  
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
```

(Tool #2 IS A .3125 stub drill)

>(Drill point is 1/3 of the drill diameter.)



G 84 Tapping Canned Cycle Example

```

T3 M06
G00 G90 G54 X.565 Y-1.875 S900 M03
G43 H03 Z.2 M08
G84 Z-.600 F56.25 R.2
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
%
```

(Tool #3 is a 3/8-16 tap)

>(900 RPM divided by 16TPI = 56.25 IPM)

**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
%
```




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.

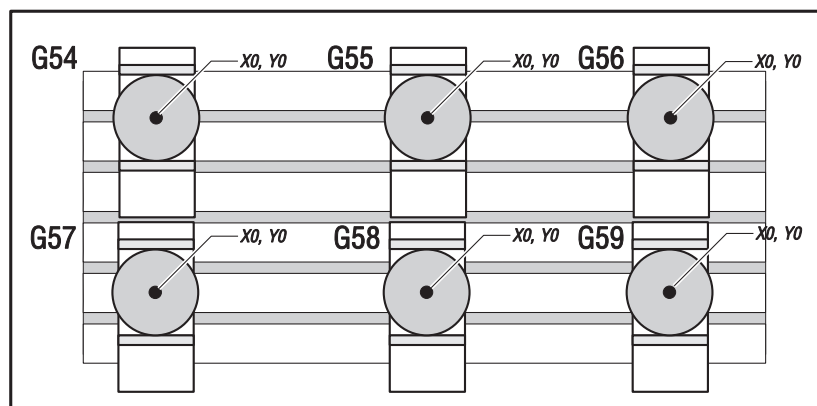
By looking at the next page, we can get a good idea of what this setup would look like on the machine table.

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 figure 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 positions.

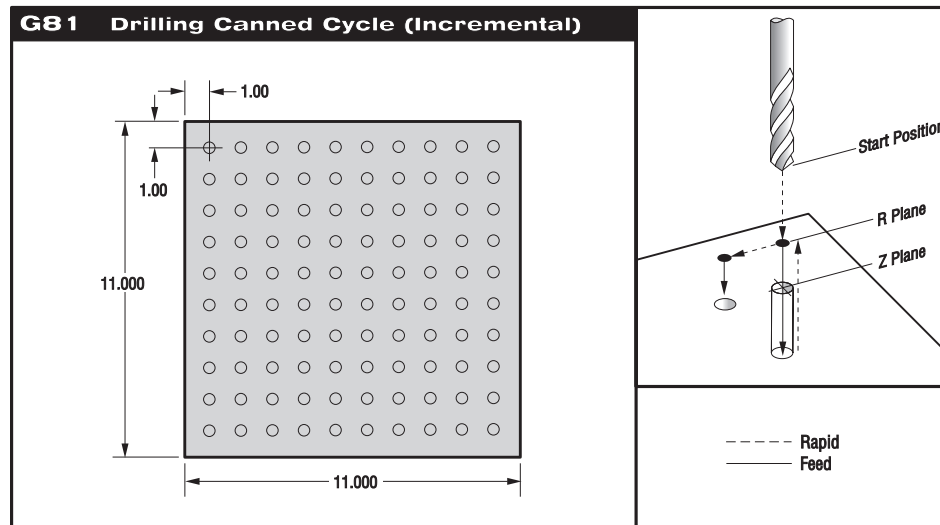


Multiple fixture setup.

**LOOPING CANNED CYCLES**

The following is an example of a program using a drilling canned cycle that is incrementally looped. Compare the grid plate drawing to the program.

Note: The sequence of drilling used here is designed to save time and to follow the shortest path from hole to hole.



Grid plate for multiple-fixturing subprogram exercise.

PROGRAM EXAMPLE

```
%
O3400
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
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
%
```

DESCRIPTION

(Drilling grid plate)

(Or stay in G91 and repeat Y-1.0)



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.

PROGRAM EXAMPLE

DESCRIPTION

```
%
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
X6.0 G99
X8.0
X10.0
X12.0 G98
X16.0 G99
X18.0 G98
G00 G80 Z2.0 M09
G28 G91 Y0 Z0
M30
%
```

(Will return to starting point after executing cycle)

(Will return to reference plane after executing cycle)

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.

**PROGRAM EXAMPLE****DESCRIPTION**

```
%  
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  
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  
%
```

(angular corner avoidance)

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.

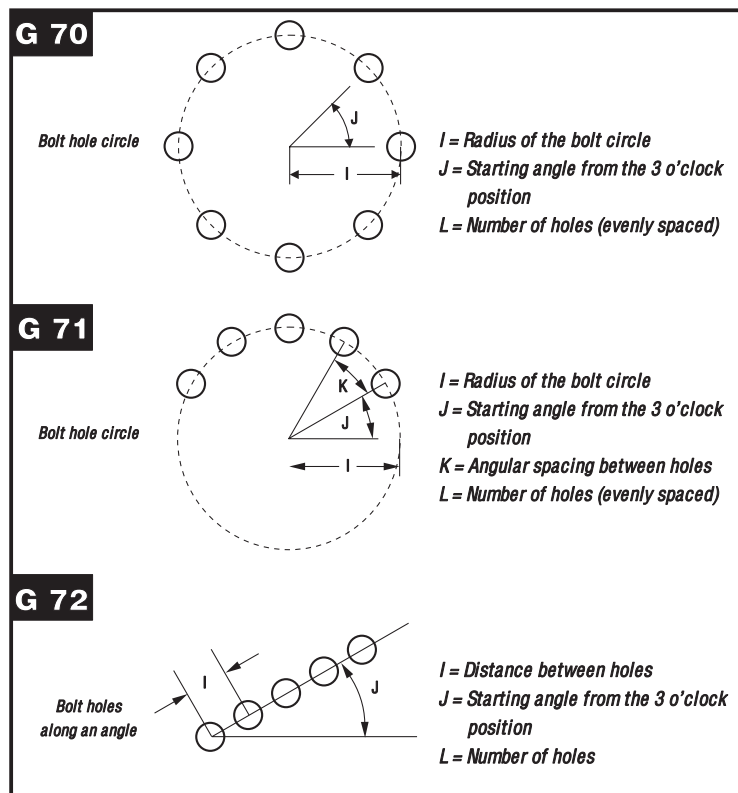
```
%  
O5000  
T1 M06  
G00 G90 G54 X0 Y0 S1500 M03  
G43 H01 Z.1 M08  
G70 I1.5 J0 L8 G81 Z-1.0 F15. R.1  
G00 G80 Z1.0 M09  
G28 G91 Y0 Z0  
M30  
%
```



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:



Bolt Hole Pattern Canned Cycles

```
%
O5000
T1 M06
G00 G90 G54 X0 Y0 S1500 M03
G43 H01 Z.1 M08
G83 R.1 Z-1.0 I.25 J.03 K.15 F15. L0 (L0 PREVENTS DRILLING AT CENTER)
G70 I1.5 J0 L8
G00 G80 Z1. M09
G28 G91 Y0 Z0
M30
%
```

**CIRCULAR INTERPOLATION AND CUTTER COMPENSATION**

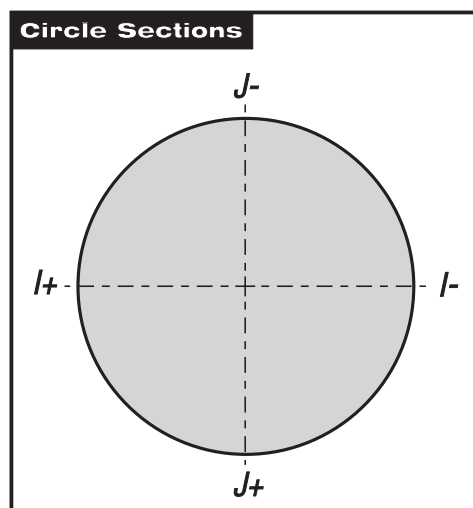
In this section, we will cover the usage of G02 (Circular Interpolation Clockwise) and G03 (Circular Interpolation Counterclockwise) and Cutter Compensation (G41: Cutter Compensation Left, G42: Cutter Compensation Right).

Using G02 and G03, we can program the machine to cut circular moves and radii. Generally, when programming a profile or a contour, the easiest way to describe a radius between two points is with an **R** and a value. For complete circular moves (360°), an **I** or a **J** with a value must be specified. The circle section illustration below will describe the different sections of a circle.

By using cutter compensation in this section we, the programmers, will be able to shift a cutter by the amount of the cutter radius and be able to program a profile or a contour to the exact print dimensions. By shifting the cutter radius, the programming time and the likelihood of calculation error is reduced.

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

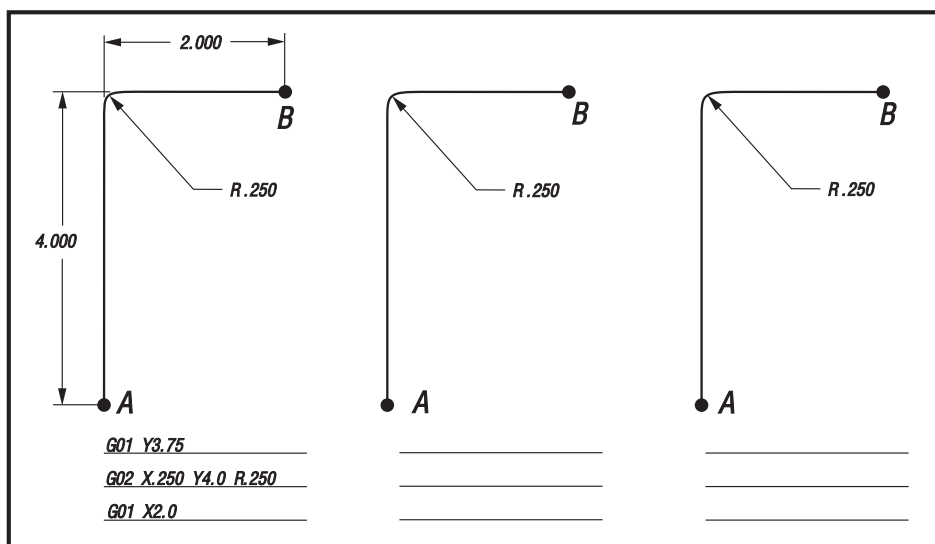
- 1) Cutter compensation must be turned ON during a G00 or G01 X,Y move that is equal to or greater than the cutter radius, or the amount being compensated for.
- 2) When an operation using cutter compensation is done, the cutter compensation will need to be turned OFF, using the same rules as the turn ON process, i.e., what is put in must be taken out.
- 3) In most machines, during cutter compensation, a linear X,Y move that is smaller than the cutter radius may not work. (Setting 58 - set to Fanuc - for positive results.)
- 4) Cutter compensation cannot be turned ON or OFF in a G02 or G03 arc movement.





DO THE FOLLOWING EXERCISE FOR PRACTICE.

WRITE THE CODE NECESSARY TO GET FROM POINT A TO POINT B



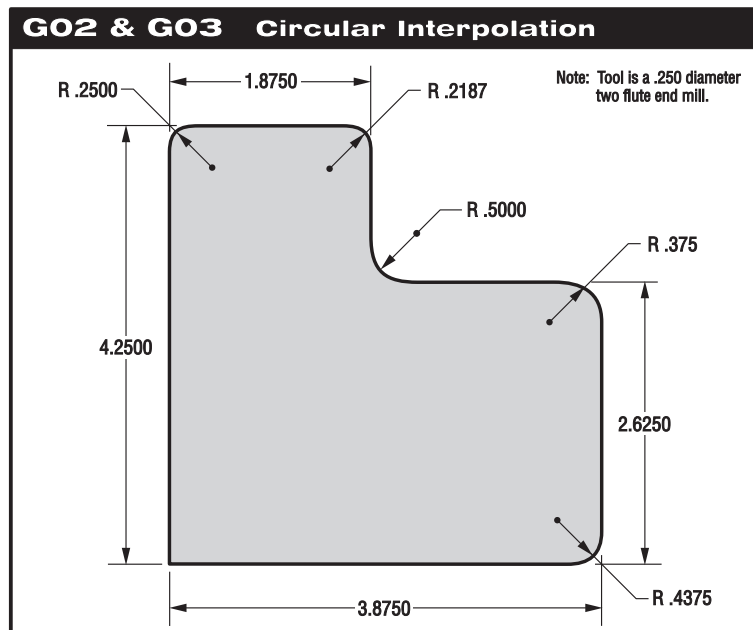
FIRST MOVE: Overall dimension and subtract the given radius.

SECOND MOVE: Add radius to current **X** and **Y** values. Always program the ending points in circular interpolation.

THIRD MOVE: Linear move to the next point.



WRITE A PROGRAM TO PRODUCE THE PART SHOWN BELOW.



Part for G02, G03, and R practice.

The correct program for the part is as follows:

PROGRAM EXAMPLE

```
%
O600
T1 M06
G00 G90 G54 X-.2 Y-.2 S5000 M03
G43 H01 Z.1 M08
Z-1.0
G41 D01 X0
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 Y-.3
Z1.0 M09
G28 G91 Y0 Z0
M30
%
```

DESCRIPTION

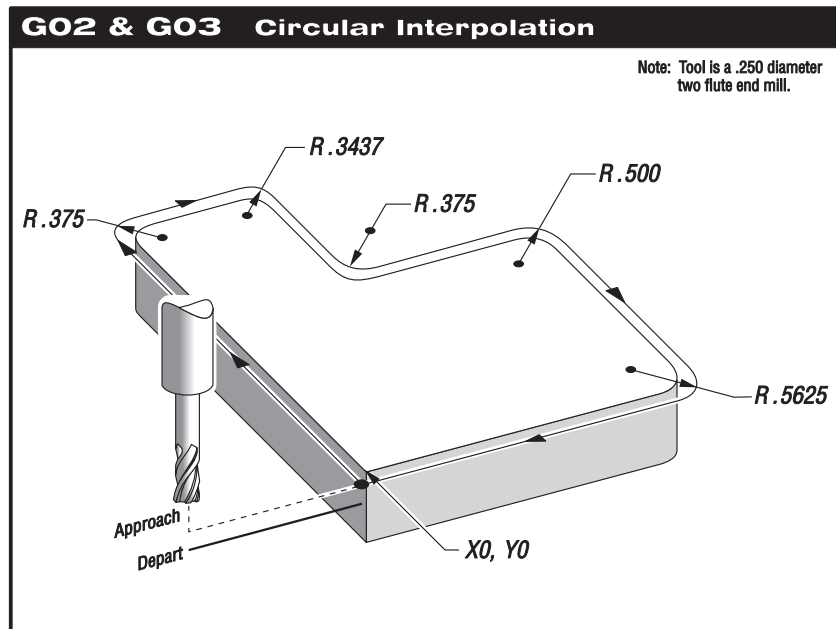
(Tool is a .250 diameter two-flute end mill)

(Turn **ON** cutter compensation, .200 move)

(Turn **OFF** cutter compensation, .200 move)



The following illustration shows how the tool path is calculated for the cutter compensation.



Programming exercise showing tool path.

The following 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.

```
%
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
%
```

**PROGRAMMING USING I AND J**

Most contour machining will use a radius value **R** for circular interpolation moves less than 360°. An **I** and **J** can also be used in the place of **R**, but this can be more confusing at times. The **I** and **J** are signed distances from the starting point to the center of the circle.

Referring back to the previous section, we can see the program below using the **R** or the **I** and **J**:

Using **R**:

```
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: Compare the +1.250 move with the "Circle Sections" illustration shown earlier.

Note: The G02 or G03 line will always need the X,Y end points, whenever **R** or **I** and **J** are used.

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; J is assumed 0 if left out)
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.



CIRCULAR POCKET MILLING

The Haas control has included in its software a Yasnac style circular pocket milling program (G12 clockwise circular pocket, G13 counterclockwise pocket). These G codes imply the use of cutter compensation, i.e., a G41 or G42 is not required to be stated in the program line. However, a D____ offset number for cutter radius or diameter is required for the ability to adjust the circle diameter.

In this section, we will cover the G12 and G13 format, as well as the different ways these programs can be written for many various applications:

SINGLE PASS: Using **I** only.

APPLICATIONS: One-pass counterboring; rough and finish pocketing of smaller holes; I.D. keyway cutting; "O"-ring grooves.

MULTIPLE PASS: Using **I**, **K**, and **Q**.

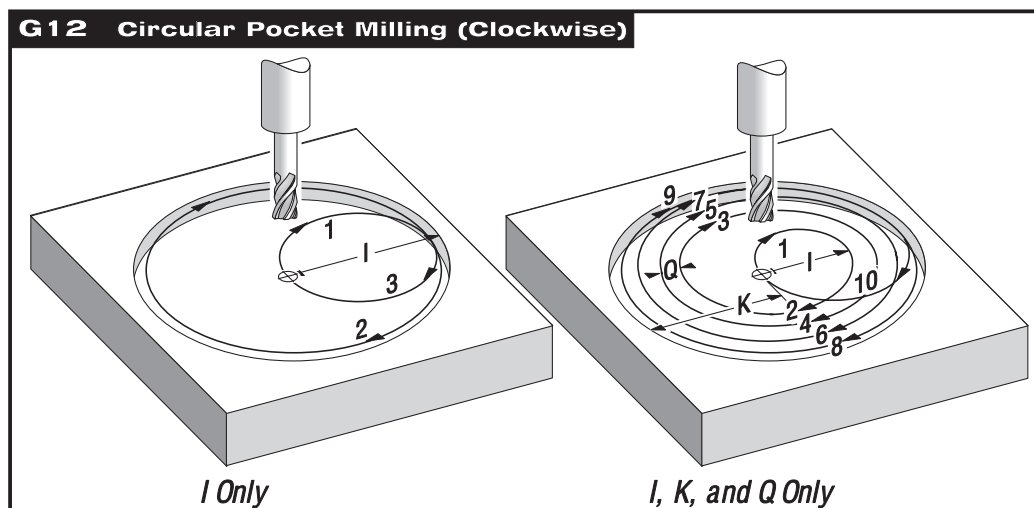
APPLICATIONS: Multiple-pass counterboring, rough and finish pocketing of large holes with cutter overlap.

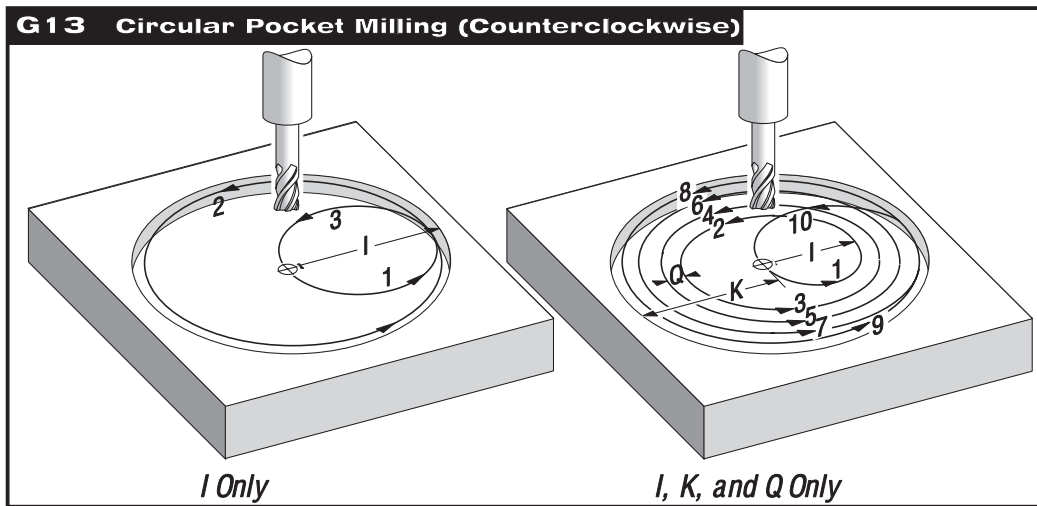
MULTIPLE Z DEPTH PASS: Using **I** only, or, **I**, **K**, and **Q**. (G91 and L)

APPLICATIONS: Deep rough and finish pocketing; incremental **Z** depth stepping.

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

The figure below shows the tool path during the G12 and G13 cycles. One uses **I** only and the other uses **I**, **K**, and **Q**.





Tool path during G12 and G13 cycles.

PROGRAM LINE REQUIREMENTS:

Z = Depth of cut or increment

F = Feed Rate

I = Radius of first circle

K = Radius of finished circle

Q = Radius increment or cutter overlap

D = Tool geometry offset number

L = Loop count for incremental Z depth stepping

(Finished radius if no K specified)

(If using I, K, and Q)

(Must use with K)

(Not required)

(Optional)

EXAMPLE G13 single-pass using I only:

PROGRAM EXAMPLE

```
%
O2000
T1 M06
G00 G90 G54 X0 Y0 S4000 M03
G43 H01 Z.1 M08
G13 Z-1.0 F20. I.500 D01
G00 Z1.0 M09
G28 G91 Y0 Z0
M30
%
```

DESCRIPTION

*(.500 entered in the Radius/Diameter offset column)
(Tool #1 is a .500 diameter end mill)*

(Will complete a one-inch diameter hole one-inch deep)



EXAMPLE G13 multiple-pass using I, K, and Q:

This example will complete a three-inch diameter hole, one inch deep, with a cutter overlap of .100 thousandths of an inch.

PROGRAM EXAMPLE

DESCRIPTION

%	
O3000	(.500 entered in the Radius/Diameter offset column)
T1 M06	(Tool #1 is a .500 diameter end mill)
G00 G90 G54 X0 Y0 S4000 M03	
G43 H01 Z.1 M08	
G13 Z-1.0 F20. D01 I.400 K1.5 Q.400	
G00 Z1.0 M09	
G28 G91 Y0 Z0	
M30	
%	

EXAMPLE G13 Multiple-pass using I, K, Q, L, and G91:

This 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 of the offsets display page 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.

PROGRAM EXAMPLE

DESCRIPTION

%	
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	
%	

**GENERAL PURPOSE POCKET MILLING**

The general purpose pocket milling program is 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.

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**

G150 FORMAT EXAMPLE:

```
%  
O4500  
T1 M06  
G00 G90 G54 X0 Y0 S3500 M03  
G43 H01 Z1.1 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  
%
```

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.

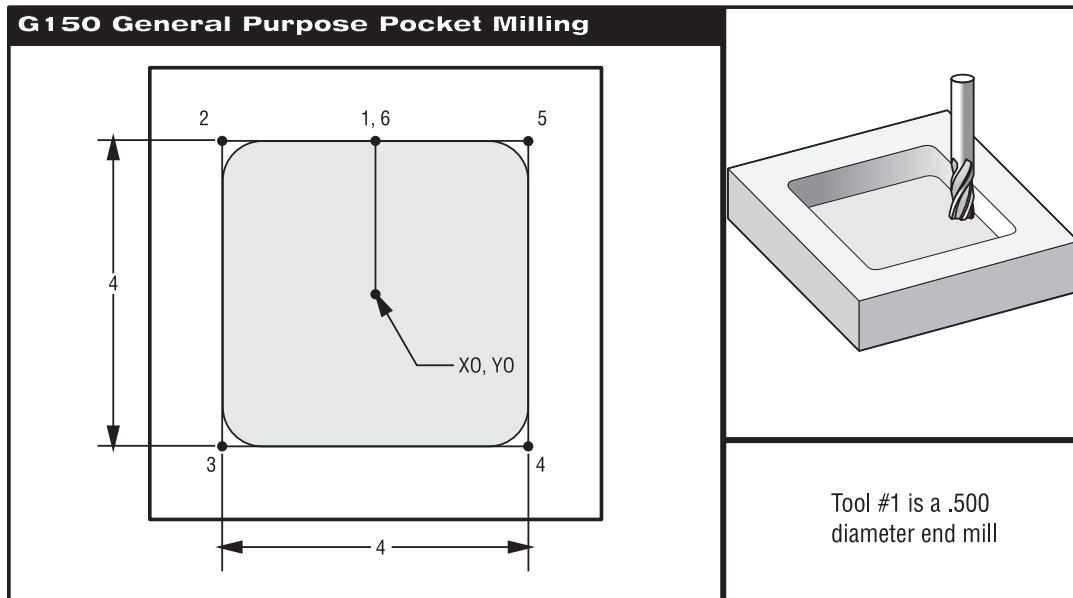
G150 EXAMPLES -

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  
G90  
M99  
%
```

**SQUARE POCKET:**

Pocket milling exercise for G150 operation.

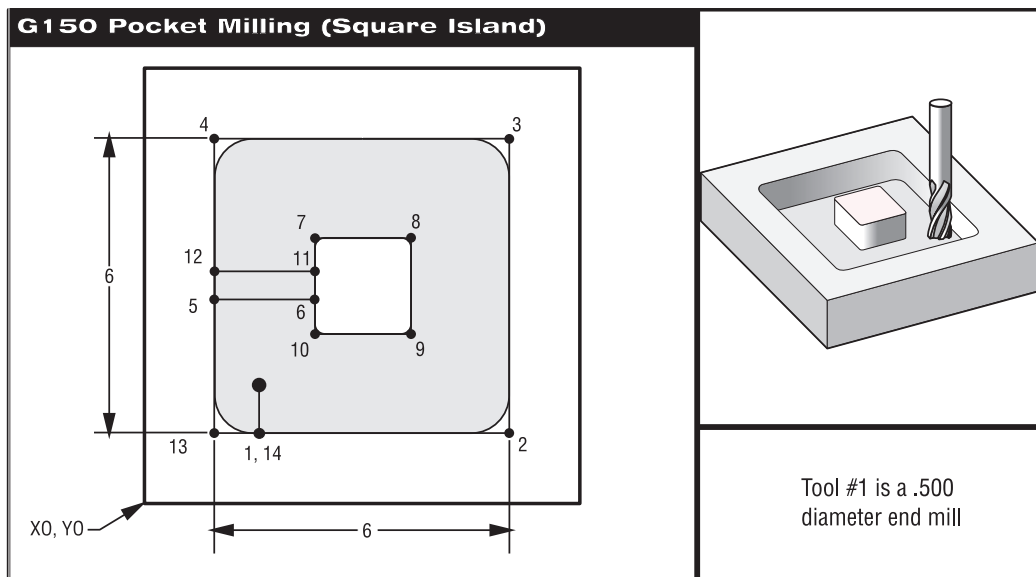
4.0 x 4.0 x .500 DP. SQUARE POCKET:

```
%
O1000
T1 M06
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
%
```

(Tool #1 is a .500 diameter end mill)



SQUARE ISLAND:



Square island programming exercise using G150.

```

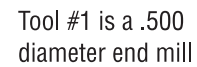
%
O1000
T1 M06
G00 G90 G54 S2500 M03
G43 H01 Z.1 M08
G01 Z0 F10.
G150 X1.0 Y1.0 Z-.5 F15. R.1 Q.25 I.4 K.01 P500 D01 G41
G00 Z1.0 M09
G40 G28 G91 Y0 Z0
M30
%

%
O0500
G01 X0 Y0
X6.0
Y6.0
X0
Y3.0
X2.0
Y4.0
X4.0
Y2.0
X2.0
Y3.0
X0
Y0
M99
%
```

(Tool is a .500 diameter end mill)



G150 Pocket Milling (Round Island)



```
%
O1000
T1 M06
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
%

**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 on mirror image for any axis listed in that block. The actual value given for the **X**, **Y**, **Z**, or **A** code has no effect and should be entered as zero value.

Example:

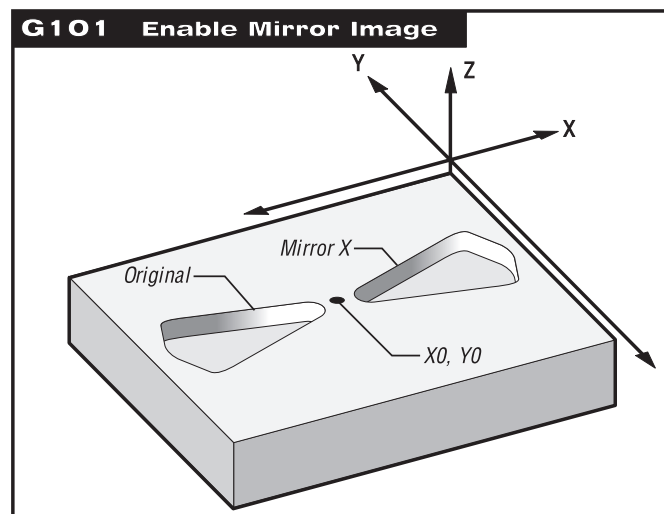
G101 X0 = Will turn on mirror image for the X-axis.

G100 X0 = Will turn off mirror image for the X-axis.

Most mirror image applications would consist of irregular pockets and contours and would most likely be set up in subprograms for convenience.

Note: 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.



Mirror image and pocket milling exercise for O3600.

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:

PROGRAM EXAMPLEDESCRIPTION

```
%  
O3600  
T1 M06  
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  
%
```

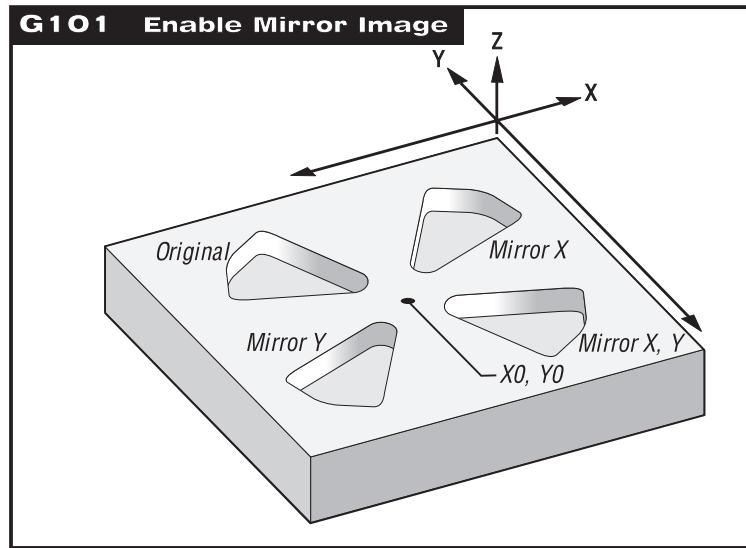
(Mirror image X-axis)
(Tool #1 is a .250 diameter end mill)

```
%  
O3601  
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  
%
```

(Contour subprogram)



PROGRAM CODE FOR MIRROR IMAGE IN THE X, Y, AND XY AXES:



Mirror image (X, Y, and X-Y) and pocket milling exercise for 03700.

PROGRAM EXAMPLE

```
%
O3700
T1 M06
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.
X-.2923 Y.3658
G01 Z-.25 F5.
F20.
M98 P3701
G00 Z.1
G100 X0.
G101 Y0.
X-.2923 Y.3658
G01 Z-.25 F5.
F20.
M98 P3701
G00 Z.1
G100 Y0.
G101 X0. Y0.
X-.2923 Y.3658
G01 Z-.25 F5.
F20.
```

DESCRIPTION

*(Mirror image X, Y, and XY axes)
(Tool #1 is a .250 diameter end mill)*

*(Turn on mirror image X-axis)
(Position to original coordinates)
(Feed to Z depth)
(Pocket feed rate)
(Pocket contour subprogram call)
(Part clearance)
(Cancel mirror image X-axis)
(Turn on mirror image Y-axis)*

*(Cancel mirror image Y-axis)
(Turn on mirror image X and Y axes)*



```

M98 P3701
G00 Z.1
G100 X0. Y0.
G28 G91 Y0 Z0
M30

```

(Cancel mirror image X and Y axes)

```

O3701
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
%

```

*(Contour subprogram)***THREAD MILLING**

We will use the following example and go through the thread milling procedures step-by-step to get the desired result:

DATA:

- I.D. Thread milling a 1.5 x 8 TPI hole.
- Using .750 diameter x 1.0 thread hob.
- Take the hole diameter 1.500. Subtract cutter diameter .750 = .750 Then divide by 2 = .375.

STEP 1: Within this space we need to turn on cutter compensation and ramp on to the circle to be machined.

STEP 2: Perform complete circle while simultaneously moving in the Z-axis the amount of one full pitch of the thread. This is called **helical interpolation**.

STEP 3: Ramp off the circle and turn off the cutter compensation.

Note: Always climb cut the cutter.

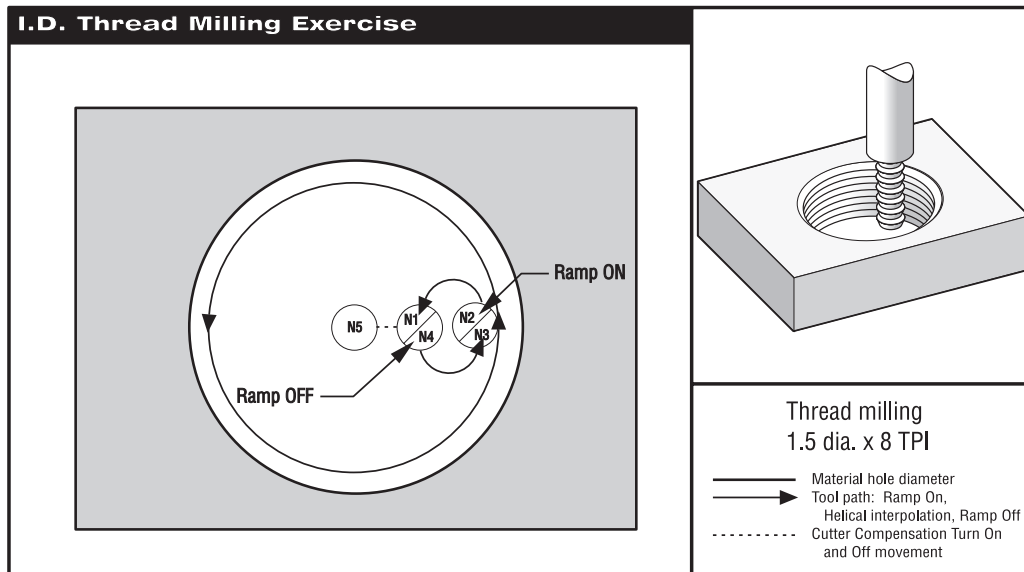
I.D. will be G03; **O.D.** will be G02. An **I.D.** right hand thread will move **up** in the Z-axis by the amount of one thread pitch. An **O.D.** right hand thread will move **down** in the Z-axis by the amount of one thread pitch.

PITCH = 1.0/Threads per inch

Example 1.0 divided by 8 TPI = .125



Cutter compensation cannot be turned off or on during an arc movement. A linear turn on and turn off movement must be made, either in the X- or Y-axis. This move will be the maximum compensation amount that can be adjusted.



Thread milling exercise

PROGRAM EXAMPLE

```
%
O2300
(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
N1 G01 G41 D01 X.175 F25.
N2 G03 X.375 R.100 F7.
N3 G03 I-.375 Z-.475
N4 G03 X.175 R.100
N5 G01 G40 X0 Y0
G00 Z1.0 M09
G28 G91 Y0 Z0
M30
%
```

DESCRIPTION

(Thread milling 1.5 diameter x 8 TPI)

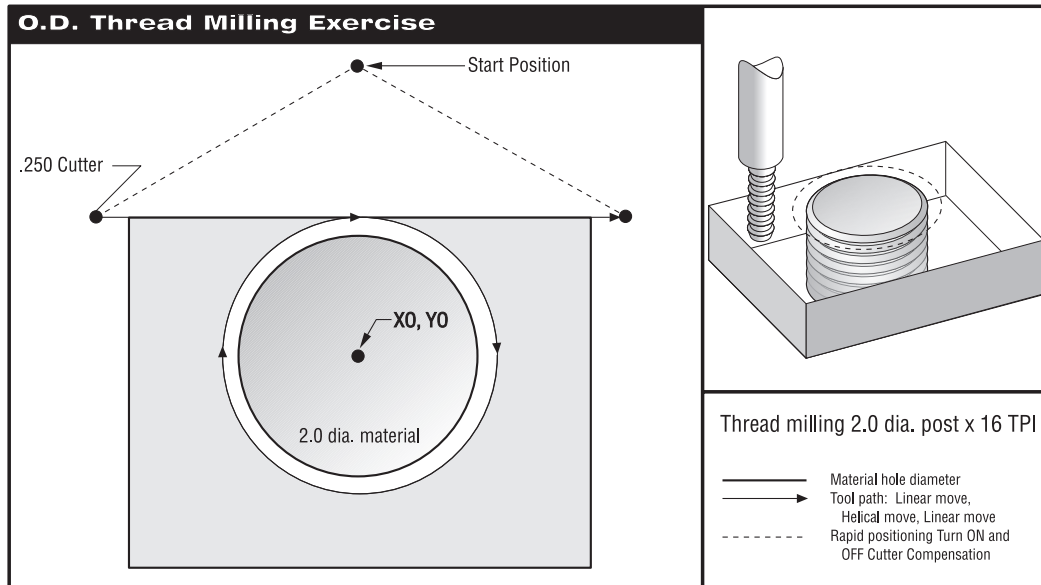
*(Turn on Cutter Comp)
(Ramp on move)
(One full revolution with Z moving up. 125)
(Ramp off move)
(Cancel Cutter Comp)*

Note: Maximum cutter compensation adjustability is .175, which is more than enough for this application.

Start with zero in the diameter offset column and enter a negative number to increase the thread diameter.



O.D. THREAD MILLING -



O.D. thread milling exercise.

PROGRAM EXAMPLE

```
%
O2400
(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
G01 X0. F15.
G02 J-1.125 Z-1.0625
G01 X1.5
G00 G40 X0 Y2.0
Z1.0 M09
G28 G91 Y0 Z0
M30
%
```

DESCRIPTION

(Thread milling a 2.0 diameter post x 16 TPI)

*(Turning on cutter compensation.)
(Linear interpolation onto the post.)
(360° helical circle; negative Z move.)
(Linear interpolation off the post.)
(Turning off cutter compensation.)*

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.



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

PROGRAM EXAMPLE

DESCRIPTION

%

O1000

(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.

G41 X.275 D1

(Radial value)

G3 X.875 I.3 F15.

G91 G3 I-.875 Z.0833 L14

(Multiply .0833 pitch x 14 passes = 1.1662 = total in Z-

axis)

G90 G3 X.275 I-.300

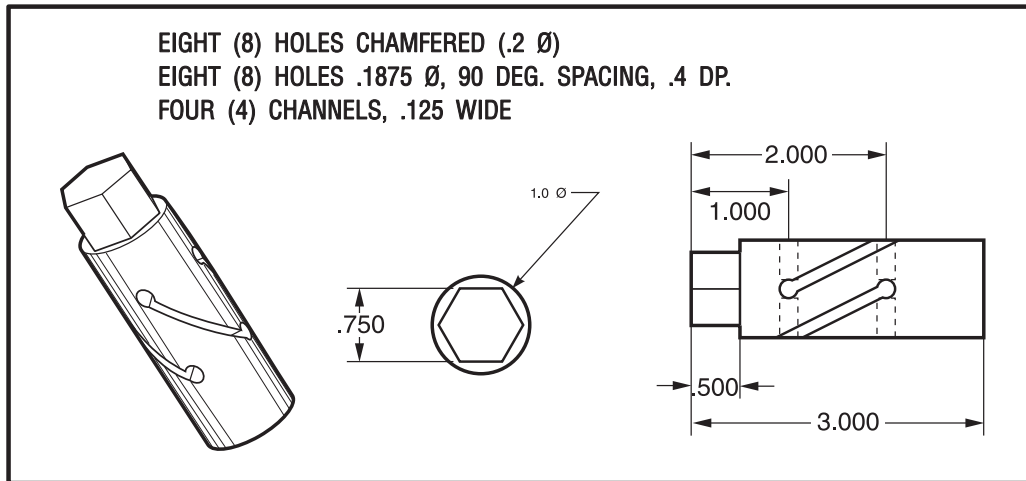
G00 G90 Z1.0 M09

G1 G40 X0 Y0

G28 G91 Y0 Z0

M30

%

**FOURTH AXIS PROGRAMMING***Fourth axis programming exercise.***PROGRAM EXAMPLE**

```
%
O1234
(Material is 1.0 Ø round stock x 3.0 L)
(Set material in collet to protrude 2.25.)
(Set fixture parallel with the table T-slots on the right side.)
(X0 is the front of the material.)
(Y0 is the centerline of the spindle and material.)
(Z0 is the top of the part.)
T1 M06
G00 G90 G54 X.250 Y-.500 A0 S4500 M03
G43 H01 Z.1 M08
M98 P1235 L6
G00 G90 Z.1 M09
T2 M06
G00 G90 G54 X1.0 Y0 A0 S5000 M03
G43 H02 Z.1 M08
G82 Z-.1 F10. R.1 P300
X2.0
A90.
X1.0
A180.
X2.0
A270.
X1.0
G00 G80 Z.1 M09
(Cont'd.)
```

DESCRIPTION*(Fourth axis program using a Haas Servo 5C)**(Tool #1 is a .500 end mill to mill hex.)**(Tool #2 is a .375 Ø NC spot drill.)*



```
T3 M06
G00 G90 G54 X1.0 Y0 A0 S5000 M03
G43 H03 Z.1 M08
G83 Z-1.125 F12. R.1 Q.25
X2.0
A90.
X1.0
```

(Tool #3 is a .1875 Ø stub twist drill.)

```
G00 G80 Z.1 M09
T4 M06
G00 G90 G54 X1.0 Y0 A0 S5000 M03
G43 H04 Z.1 M08
M98 P1236
G00 G90 Z.1 M09
G28 G91 Y0 Z0
M30
%
```

(Tool #4 is a .125 Ø end mill.)

```
%
O1235
G01 Z-.125 F50.
Y.5 F35.
G00 Z.1
G91 Y-1.0 A60.
G90
M99
%
```

(Subprogram to mill hex.)

The following subprogram can be written in absolute or incremental programming. Examine each program and determine which style would be faster and easier to understand and to program in the future.

ABSOLUTE:

```
%
O1236 (Subprogram to mill channels.)
G01 Z-.25 F15.
X2.0 A90.
G00 Z.1
A180.
G01 Z-.25
X1.0 A90.
G00 Z.1
A180.
G01 Z-.25
X2.0 A270.
G00 Z.1
A360.
G01 Z-.25
X1.0 A270.
G00 Z.1
M99
%
```

INCREMENTAL:

```
%
O1236 (Subprogram to mill channels.)
G91
G01 Z-.35 F15.
X1.0 A90.
G00 Z.35
A90.
G01 Z-.35
X-1.0 A-90.
G00 Z.35
A90.
G01 Z-.35
X1.0 A90.
G00 Z.35
A90.
G01 Z-.35
X-1.0 A-90.
G00 G90 Z.1
M99
%
```

**FORMULAS****TAPPING -**

STANDARD thread formula:

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

$$\text{RPM/TPI} = F$$

METRIC thread formula:

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

$$(P \times .03937) \times \text{RPM} = F$$

SPEED AND FEEDS -

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

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

$$.262 \times \text{Cutter Diameter} \times \text{RPM} = \text{SFM}$$

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

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

$$(3.82 \times \text{SFM}) / \text{Cutter Diameter} = \text{RPM}$$

I.P.M. (Inch Per Minute):

Feed per tooth multiplied by the number of cutter teeth multiplied by the RPM = Feed rate in inches per minute.

$$(\text{Feed/tooth} \times n) \times \text{RPM} = \text{IPM or F}$$

CUBIC INCH PER MINUTE: MATERIAL REMOVAL RATE

Effective diameter of cut multiplied by the depth of cut multiplied by the inch per minute feed rate = cubic inch per minute.

$$(E \text{ Diameter} \times d) \times \text{IPM} = \text{CIPM}$$



8. SETTINGS

The setting pages contain values 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.

The settings are organized into pages of functionally similar groupings. This will make it easier for the user to remember where the settings are located and reduces the amount of time spent maneuvering through the settings display. The list below is separated into page groups with the page title as the heading.

Use the vertical cursor keys to move to the desired setting. Depending on the setting, you may change it by entering a new number or, if the setting has specific values, press the horizontal cursor keys to display the choices. Press the WRITE key to enter or change the value. The message near the top of the screen tells you how to change the selected setting.

The serial number is Setting 26 on this page and is protected from user change. If you need to change this setting, contact HAAS or your dealer.

One of the more commonly adjusted settings will be number 34, the "Rotary Axis Diameter". This setting is used to control the surface feed rate when the fourth axis is used in a cutting feed. Feeding with the **X**, **Y**, or **Z** and the **A** axes assumes that the linear motion is along the axis of the rotary motion. When this is true and the diameter setting is correct, the programmed surface feed rate will be correct for helical cuts. In addition, feeds of just the A-axis depend on this setting to determine the correct angular rate.

If you have a fourth axis 5C, fifth axis, or rotary table, it may be disabled from the setting page and removed from the machine. Do not connect or disconnect any cables with the control on. If you do not disable the fourth axis when it is disconnected, you will get an alarm.

Settings may be sent and received with the RS-232 port. See the "Data Input / Output" section for a description of how to do this.

The settings are listed here with a description of each. The page title will precede each page of settings and the settings will appear in order as shown on the screen.

Page Name:	Setting No:	Description:	Range of Value:
GENERAL			
	26	SERIAL NUMBER	1000 to 99999
	82	LANGUAGE	English, German, French, Spanish, Italian
	1	AUTO POWER OFF TIMER	0 to 9999 minutes
	81	TOOL AT POWER DOWN	0 to 20
	9	DIMENSIONING	INCH or METRIC
	77	SCALE INTEGER F	ON or OFF
	33	COORDINATE SYSTEM	FANUC, YASNAC, or HAAS
	53	JOG W/O ZERO RETURN	ON or OFF
	40	TOOL OFFSET MEASURE	RADIUS or DIAMETER
	64	T. OFS MEAS USES WORK	ON or OFF



Page Name:	Setting No:	Description:	Range of Value:
---------------	-------------	--------------	-----------------

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 8379.999
72	DEFAULT G68 ROTATION	0.0000 to 3599.999
73	G68 INCREMENTAL ANGLE	ON or OFF

PROGRAM 2

30	4TH AXIS ENABLE	OFF, S5C, 7RT, 9RT, SRT, 11RT, HA5C, HRT160, HRT210, HRT310, 5CN, HRT450, HRT310HA, HRT450HA, VR-A, VR-B
34	4TH AXIS DIAMETER	0 to 29.9999 inches
78	5TH AXIS ENABLE	OFF, S5C, 7RT, 9RT, SRT, 11RT, HA5C, HRT160, HRT210, HRT310, 5CN, HRT450, HRT310HA, HRT450HA, VR-A, VR-B
79	FIFTH AXIS DIAMETER	0 to 29.9999 inches
38	AUX AXIS NUMBER	0 to 5
22	CAN CYCLE DELTA Z	0 to 29.9999 inches
28	CAN CYCLE ACT W/O X/Y	ON or OFF
52	G83 RETRACT ABOVE R	0.0 to 9.9999 inches
57	EXACT STOP CANNED X-Y	ON or OFF
43	CUTTER COMP TYPE	A or B
44	MIN F IN RADIUS CC %	1 to 100
58	CUTTER COMPENSATION	FANUC or YASNAC
85	MAX CORNER ROUNDING	0 to 0.25

PROGRAM 3

15	H & T CODE AGREEMENT	ON or OFF
27	G76/G77 SHIFT DIR.	X+, X-, Y+, or Y-
29	G91 NON-MODAL	ON or OFF
32	COOLANT OVERRIDE	NORMAL, OFF, or IGNORE
35	G60 OFFSET	0 to 0.9999 inches
42	M00 AFTER TOOL CHANGE	ON or OFF
49	SKIP SAME TOOL CHANGE	ON or OFF
86	M39 LOCKOUT	ON or OFF
108	QUICK ROTARY G28	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



Page Name:	Setting No:	Description:	Range of Value:
RS-232 PORTS			
	11	BAUD RATE SELECT	50, 110, 200, 300, 600, 1200, 2400, 4800, 7200, 9600, 19200, 38400, or 115200(optional)
	12	PARITY SELECT	NONE, ODD, EVEN, ZERO
	13	STOP BIT	1 or 2
	14	SYNCHRONIZATION	XON/XOFF, RTS/CTS, DC CODES, or XMODEM
	37	RS-232 DATA BITS	7 or 8
	24	LEADER TO PUNCH	NONE, BLANK, or NUL CHAR
	25	EOB PATTERN	CR LF, LF ONLY, CR ONLY, or LF CR CR
	41	ADD SPACES RS232 OUT	ON or OFF
	50	AUX AXIS SYNC	XON/XOFF, RTS/CTS, DC CODES, or XMODEM
	54	AUX AXIS BAUD RATE	50, 110, 200, 300, 600, 1200, 2400, 4800, 7200, 9600, 19200, 38400, 115200(optional)
	69	DPRNT LEADING SPACES	ON or OFF
	70	DPRNT OPEN/CLOS DCODE	ON or OFF
CONTROL PANEL			
	6	FRONT PANEL LOCK	ON or OFF
	55	ENABLE DNC FROM MDI	ON or OFF
	76	TOOL RELEASE LOCK OUT	ON or OFF
	16	DRY RUN LOCK OUT	ON or OFF
	17	OPT STOP LOCK OUT	ON or OFF
	18	BLOCK DELETE LOCK OUT	ON or OFF
	10	LIMIT RAPID AT 50%	ON or OFF
	103	CYC START/FH SAME KEY	ON or OFF
	104	JOG HANDL TO SNGL BLK	ON or OFF
	84	TOOL OVERLOAD ACTION	ALARM, FEEDHOLD, BEEP, PAUSE, AUTOFEED
	114	CONVEYOR CYCLE	0 - 1440
	115	CONVEYOR ON-TIME	0 - 1440
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	RESERVED	N/A
	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



Page Name:	Setting No:	Description:	Range of Value:
------------	-------------	--------------	-----------------

OVERRIDES

19	FEEDRATE OVERRIDE LOCK	ON or OFF
20	SPINDLE OVERRIDE LOCK	ON or OFF
21	RAPID OVERRIDE LOCK	ON or OFF
87	M06 RESET OVERRIDES	ON or OFF
83	M30 RESET OVERRIDE	ON or OFF
88	RESET RESET OVERRIDE	ON or OFF

The following is a detailed description of each of the settings:

1 AUTO POWER OFF TIMER

This is a numeric setting. When it is set to a number other than zero, the machine will be automatically turned off after that many minutes of idle operation. This will not occur while a program is running and will not occur while the operator is pressing any keys. The auto off sequence gives the operator a 30 second warning and pressing any key will interrupt the sequence.

2 POWER OFF AT M30

This is an On/Off setting. If it is set to ON, the machine will begin an automatic power down when an M30 ends a program. The auto off sequence gives the operator a 30 second warning and pressing any key will interrupt the sequence.

3 RESERVED

This setting is reserved for future use.

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 PROG MEMORY LOCK

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 change program axis values 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. When this setting is changed from inches to metric, or vice versa, all offset values will be converted accordingly.

	INCH	METRIC
Feed	inches/min.	mm/min.
Max Travel	+/- 15400.0000	+/-39300.000
Min. Programmable Dimension	.0001	.001
Feed Range	.0001 to 300.000 in/min.	.001 to 1000.000

Axis Jog Keys		
.001 Key	.001 in/jog click	.001 mm/jog click
.001	.001 in/jog click	.001 mm/jog click
.01	.01 in/jog click	.01 mm/jog click
.1 Key	.1 in/jog click	.1 mm/jog click

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 maximum override will automatically be limited to 50%.

11 BAUD RATE SELECT

This setting allows the operator to change the serial data rate for the first serial port. This applies to program, settings, offsets, and parameters upload and download and to DNC functions.

12 PARITY SELECT

This setting allows the setting of parity for the first serial port. The possible values are: NONE, ODD, EVEN, ZERO. When set to none, no parity bit is added to the serial data. When set to zero, a 0 bit is added in the place of parity. Even and odd work like normal parity functions. Make sure you know what your system needs. XMODEM must use 8 data bits and no parity.

13 STOP BIT

This setting changes the number of stop bits for the first serial port. It can be selected to be 1 or 2.

14 SYNCHRONIZATION

This changes the synchronization protocol between sender and receiver for the first serial port. When set to RTS/CTS, the signal wires in the serial data cable are used to tell the sender to temporarily stop sending data while the receiver catches up. When it is set to XON/XOFF, those ASCII character codes are used by the receiver to tell the sender to temporarily stop. XON/XOFF is the most common setting.

DC CODES is like XON/XOFF but the paper tape punch or reader start/stop codes are sent. XMODEM is a receiver-driven communications protocol that sends data in blocks of 128 bytes. XMODEM gives the RS-232 communication added reliability because each block is checked for integrity. Refer to "Data Input/Output" section for more information. XMODEM must use 8 data bits and no parity.

**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. In program restart this check is not done until motion begins.

16 DRY RUN LOCK OUT

This is an On/Off setting. When it is **off**, the machine operates normally. When it is **on**, the DRY RUN function cannot be turned on.

17 OPT STOP LOCK OUT

This is an On/Off setting. When it is **off**, the machine operates normally. When it is **on**, the OPTIONAL STOP function cannot be turned on.

18 BLOCK DELETE LOCK OUT

This is an On/Off setting. When it is **off**, the machine operates normally. When it is **on**, the BLOCK DELETE function cannot be turned on.

19 FEED RATE OVERRIDE LOCK

This is an On/Off setting. When it is **off**, the machine operates normally. When it is **on**, the feed rate override buttons are locked out.

20 SPINDLE OVERRIDE LOCK

This is an On/Off setting. When it is **off**, the machine operates normally. When it is **on**, the spindle speed override buttons are locked out.

21 RAPID OVERRIDE LOCK

This is an On/Off setting. When it is **off**, the machine operates normally. When it is **on**, the rapid speed override buttons are locked out.

22 CAN CYCLE DELTA Z

This is a decimal numeric entry. It must be in the range of 0.0 to 29.9999 inches. This setting specifies the delta **Z** used in the G73 canned cycle when the Z-axis is moved up to clear chips.

23 9xxx PROGS EDIT LOCK

This is an On/Off setting. When it is **off**, the machine operates normally. When it is **on**, the 9000 series programs (usually macro programs) are invisible to the operator and cannot be uploaded or download. They also cannot be listed, edited, or deleted.

24 LEADER TO PUNCH

This setting is used to control the leader sent to a paper tape punch device connected to the first RS-232 port. The values that can be selected are: NONE, BLANK, or NUL CHAR. None causes no extra data to be sent as a leader. Blank causes two feet of blanks to be punched at the start of a program and one foot of blanks at the end. Null causes the same thing as blanks but uses the ASCII code null which is all zero.

25 EOB PATTERN

This setting controls what is sent out and expected as input to represent the EOB (end of block) on serial port one. The possible selections are: CR LF, LF only, CR only, or LF CR CR.

**26 SERIAL NUMBER**

This is a numeric entry. It is the serial number of your machine. It cannot be changed.

27 G76/G77 SHIFT DIR.

This setting controls the shift direction used to clear a boring tool during a G76 or G77 canned cycle. The possible selections are: X+, X-, Y+, or Y-.

28 CAN CYCLE ACT W/O X/Y

This is an On/Off setting. When it is **off**, an initial canned cycle definition without an **X** or **Y** motion will not cause the canned cycle to be executed. When it is **on**, the initial canned cycle definition will cause one cycle to be executed even if there is no **X** or **Y** motion in that command block. Note that if an L0 is in that block, it will never execute the canned cycle on the definition line.

29 G91 NON-MODAL

This is an On/Off setting. When it is **off**, the machine operates normally. When it is **on**, G91 is not modal and applies only to the command block on which it occurs.

30 4TH AXIS ENABLE

When it is **off**, the fourth axis is disabled and no commands can be sent to that axis. When **off** is not selected, the selected rotary table type parameters are called up. A change to rotary parameters is saved under the selected table type for later recall. In order to change this setting the servos must be turned off.

31 RESET PROGRAM POINTER

This is an On/Off setting. When it is **off**, the RESET button will not change the execution program pointer. When it is **on**, a RESET will change the program execution pointer to the beginning of the program.

32 COOLANT OVERRIDE

This setting controls how the coolant pump operates. The possible selections are: NORMAL, OFF, or IGNORE. The "NORMAL" setting allows the operator to turn the pump on and off manually or with **M** codes. The "OFF" setting will generate an alarm if an attempt is made to turn the coolant on manually or from a program. The "IGNORE" setting will ignore all coolant commands, but the pump can be turned on manually.

33 COORDINATE SYSTEM

This setting changes the way the G92/G52 offset system works. It can be set to FANUC, HAAS, or YASNAC. The Fanuc and HAAS class of controls uses the G52 offset differently than the Yasnac class of controls. In a Fanuc or HAAS control, G52 sets a local coordinate system. In a Yasnac control, the G52 stands on its own as another work offset.

34 4th AXIS DIAMETER

This is a numeric entry. It is used to set the angular feed rate of the **A**-axis. It must be in the range of 0.0 to 29.9999 inches. Since the feed rate specified in a program is always inches per minute (or mm per minute), the control must know the diameter of the part being worked in the **A**-axis in order to compute the angular feed rate. When this setting is set correctly, the surface feed rate on a rotary cut will be exactly the feed rate programmed into the control. The feed rate will be correct only as long as the axis remains orthogonal (at right angles) to all other axes.

35 G60 OFFSET

This is a numeric entry in the range of 0.0 to 0.9999 inches. It is used to specify the amount of overshoot when unidirectional positioning (G60) is programmed.

**36 PROGRAM RESTART**

This is an On/Off setting. When it is **off**, starting a program from anywhere other than the beginning may produce inconsistent results. When it is **on**, starting a program from the middle causes the entire program to be scanned to ensure that the correct tools, offsets, **G** codes, and axes positions are set correctly before starting at the block where the cursor is positioned. Some alarm conditions are not detected prior to motion starting.

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. XMODEM must use 8 data bits and no parity.

38 AUX AXIS NUMBER

This is a numeric entry between 0 and 4. 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 C-axis. If it is set to 2, there are **C** and **U** 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 TOOL CHANGE

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 TOOL CHANGE is displayed as a message at the bottom left. This affects only programmed tool changes.

43 CUTTER COMP TYPE

This setting controls how an entry to cutter compensation occurs. It can be selected to be A or B. It affects only the first stroke that begins cutter compensation and changes the way the tool is cleared from the part being cut.

44 MIN F IN RADIUS CC %

This setting is a numeric entry between 1 and 100. It affects the feed rate when cutter compensation moves the tool towards the inside of a circular cut. In order to maintain a constant surface feed rate, such a cut will be slowed down. This setting specifies the minimum feed rate as a percentage of the programmed feed rate.

**45 MIRROR IMAGE X-AXIS****46 MIRROR IMAGE Y-AXIS****47 MIRROR IMAGE Z-AXIS****48 MIRROR IMAGE A-AXIS**

These are On/Off settings. When it is **off**, axes motions occur normally. When it is **on**, the specific axis motion is mirrored (or reversed) around the work zero point.

49 SKIP SAME TOOL CHANGE

This is an On/Off setting. When it is **off**, an M16 will always cause a tool change sequence to occur; even if the same tool is put back into the spindle. When it is **on**, a tool change to the same tool as is in the spindle will cause no action.

50 AUX AXIS SYNC

This changes the synchronization protocol between sender and receiver for the second serial port. When set to RTS/CTS, the signal wires in the serial data cable are used to tell the sender to temporarily stop sending data while the receiver catches up. When it is set to XON/XOFF, those ASCII character codes are used by the receiver to tell the sender to temporarily stop. XON/XOFF is the most common setting. Make sure that the Haas servo control is set to the same condition.

DC CODES is like XON/XOFF but the paper tape punch or reader start/stop codes are sent. XMODEM is a receiver-driven communications protocol that sends data in blocks of 128 bytes. XMODEM gives the RS-232 communication added reliability because each block is checked for integrity. Refer to "Data Input/Output" section for more information.

51 DOOR HOLD OVERRIDE

This is an On/Off setting. When it is **off**, a program cannot be started when the doors are open and opening the doors will cause a running program to stop just like in FEED HOLD. When it is **on**, and Parameter 57 bits DOOR STOP SP and SAFETY CIRC are set to zero, the door condition is ignored. When the control is turned on, this setting is set to Off.

52 G83 RETRACT ABOVE R

This is a numeric entry in the range of 0.0 to 9.9999 inches. This setting changes the way G83 and 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 **C**, **U**, **V**, and **W** axes. The possible values include: 50, 110, 200, 300, 600, 1200, 2400, 4800, 7200, 9600, 19200, 38400. Note that 4800 is standard in Haas servo controls and this should be set to the same value.

**55 ENABLE DNC FROM MDI**

This is an On/Off setting. When it is **off**, DNC cannot be selected. When it is turned **on**, DNC is selected by pressing MDI while already in MDI. The DNC option must be enabled in the control.

56 M30 RESTORE DEFAULT G

This is an On/Off setting. When it is **off**, no change to the modal **G** codes occurs at the end of a program (normally M30). When it is **on**, an M30 will reset all of the modal **G** codes to their defaults. If this setting is **on**, RESET will also reset defaults.

57 EXACT STOP CANNED X-Y

This is an On/Off setting. When it is **off**, the rapid X-Y motion associated with a canned cycle may not get exact stop; according to other conditions. When it is **on**, the X-Y motion always gets exact stop. This will make canned cycles slower but less likely to run into a close tolerance fixture.

58 CUTTER COMPENSATION

This setting controls the type of cutter compensation used in the control. The types are similar to the method of cutter compensation available in other classes of controls.

59 PROBE OFFSET X+**60 PROBE OFFSET X-****61 PROBE OFFSET Y+****62 PROBE OFFSET Y-**

Settings 59 through 62 are used to define the displacement and size of the spindle probe. These numbers only apply to the probing option. These four numbers specify the travel distance in four directions from where the probe is triggered to where the actual sensed surface is located. They are used by G31, G36, G136, and M75. They can be both positive and negative numbers. If the probe width were 0.23 inches in diameter and the probe was set exactly at the center of the spindle, these four settings would all be 0.115 inches.

63 TOOL PROBE WIDTH

This setting is used to specify the width of the probe that is used to test tool diameter. This setting only applies to the probing option. It is used by G35.

64 T. OFS MEAS USES WORK

This is an on/off setting. It changes the way the TOOL OFFSET MEASURE 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 the height of the work area that is displayed on the graphics screen. The maximum size is automatically limited to default height. The default shows the machine's entire work area. A specific scale can be set by using the following formula. The default of the setting is the total Y travel.

$$\begin{aligned}\text{Total Y travel} &= \text{Parameter 20} / \text{Parameter 19} \text{ (16.25 VF-0 through VF-2)} \\ \text{Scale} &= \text{Total Y travel} / \text{Setting 65}\end{aligned}$$

66 GRAPHICS X OFFSET

This setting locates the right side of the scaling window relative to the machine X zero position, (See "Displays" section). Its default is zero.



67 GRAPHICS Y OFFSET

This setting locates the top of the scaling window relative to the machine Y zero position, (See "Displays" section). Its default is zero.

68 GRAPHICS Z OFFSET

Reserved for future use.

69 DPRNT LEADING SPACES

This is an on/off setting. It suppresses leading spaces that are generated by a macro DPRNT format statement. In a DPRNT statement the format specifies the number of characters printed to the serial port for the whole portion of a variable. If the number is smaller than the space allowed for, then leading spaces are sent to the serial port. When this setting is OFF, then no leading spaces are generated. The following example illustrates control behavior when this setting is OFF or ON.

#1= 3.0 ;	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 is an on/off setting. It controls whether the POPEN and PCLOS statements in macros send DC control codes to the serial port. When the setting is ON, these statements will send DC control codes. When it is OFF, the control codes are suppressed. Its default value is ON.

71 DEFAULT G51 SCALING

Specifies the scaling for a G51 command when the P address is not contained in the same block. It must be in the range of .001 to 8380.000. This settings default is 1.000.

72 DEFAULT G68 ROTATION

Specifies the rotation, in degrees, for a G68 command when the R address is not contained in the same block. It must be in the range of 0.0000 to 360.0000. This settings default is 0.0000.

73 G68 INCREMENTAL ANGLE

This is an on/off setting. It is a switch that allows the internal variable that controls rotation to be incremented for each call to a G68 command. When this switch is ON, and a G68 command is executed in the absolute mode (G91), then the value specified in the R address is added to the internal variable. Otherwise, the internal variable is set to the rotation value specified by R. The default setting is OFF.

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 9xxx PROGS SINGLE BLK

When Setting 75 is set to ON and the control is operating in SINGLE BLOCK mode, then the control will stop at each block in a 9000 series program and wait for the operator to press CYCLE START. When Setting 75 is set to OFF, then all blocks in a 9000 series program are executed in a continuous manner even if SINGLE BLOCK is ON. The default setting is ON.



When Setting 74 and Setting 75 are both ON, the control acts normally. That is, all blocks executed are highlighted and displayed and when in single block mode there is a pause before each block is executed.

When Setting 74 and Setting 75 are both OFF, the control will execute 9000 series subroutines without displaying the blocks contained in that subroutine. If the control is in single block mode, no single block pause will occur within a 9000 series subroutine.

When Setting 75 is ON and Setting 74 is OFF, then 9000 series subroutines will be displayed as they are executed.

76 TOOL RELEASE LOCK OUT

When this setting is set to ON, the tool release key is disabled. The default setting is OFF.

77 SCALE INTEGER F

This setting aids those wishing to run programs developed on a control other than HAAS. It allows the operator to select how the control interprets an F address code that does not contain a decimal point, (It is recommended that the programmer always use a decimal point). The setting can be set to the following values:

DEFAULT	-	F12 is interpreted as	.0012 units/minute.
INTEGER	-	" " "	12.0 " "
.1	-	" " 1.2	" " "
.01	-	" " .12	" " "
.001	-	" " "	.012 " "
.0001	-	" " "	.0012 " "

The default setting is DEFAULT.

78 FIFTH-AXIS ENABLE

When this setting is **off**, the fifth axis is disabled and no commands can be sent to that axis. When **off** is not selected, the selected rotary table type parameters are called up. A change to rotary parameters is saved under the selected table type for later recall. In order to change this setting the servos must be turned off.

79 FIFTH-AXIS DIAMETER

This is a numeric entry. It is used to set the angular feed rate of the **B**-axis. It must be in the range of 0.0 to 29.9999 inches. Since the feed rate specified in a program is always inches per minute (or mm per minute), the control must know the diameter of the part being worked in the **B**-axis in order to compute the angular feed rate. When this setting is set correctly, the surface feed rate on a rotary cut will be exactly the feed rate programmed into the control. The feed rate will be correct only as long as the axis remains orthogonal (at right angles) to all other axes.

80 MIRROR IMAGE B-AXIS

This is an On/Off setting. When it is off, axes motions occur normally. When it is on, the **B** axis motion is mirrored (or reversed) around the work zero point.

81 TOOL AT POWER DOWN

When the POWER DOWN key is pressed, the control will change to the tool specified in this setting. If zero (0) is specified, no tool change occurs at power down. 1 is the default.

82 LANGUAGE

This setting allows the user to change between available languages. If the language selected does not reside in the control, NOT AVAILABLE will be displayed in the message area when that language is selected.



83 M30 /RESETS OVERRIDES

When on, an M30 causes feed rate override, rapid override, and spindle override to be reset to default values.

84 TOOL OVERLOAD ACTION

Causes the specified action to occur anytime a tool becomes overloaded (ALARM, FEEDHOLD, BEEP, AUTOFEED). When set to FEEDHOLD, the message "Tool Overload" will be displayed whenever this condition occurs. Pressing any key will clear the message. When set to AUTOFEED, the mill automatically limits the feed rate based on the tool load.

NOTES: When tapping (rigid and floating), the feed and spindle overrides will be locked out, so the AUTOFEED feature will be ineffective (although the display will appear to respond to the override buttons.)

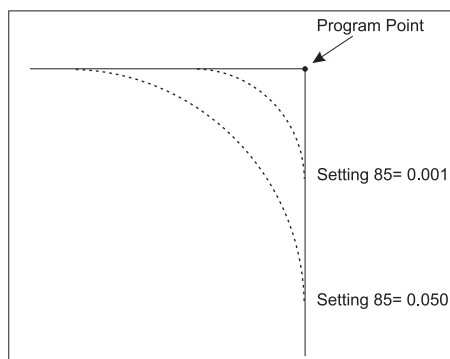
The AUTOFEED feature should not be used when doing thread milling or using autoreversing tapping head as it may cause unpredictable results or even a crash.

The last commanded feed rate will be restored at the end of the program execution, or when the operator presses RESET or turns off the AUTOFEED feature.

The operator may use the feed rate override buttons while the AUTOFEED feature is active. As long as tool load limit is not exceeded, these buttons will have the expected effect and the overridden feed rate will be recognized as the new commanded feed rate by the AUTOFEED feature. However, if the tool load limit has already been exceeded, the control will ignore the feed rate override buttons and the commanded feed rate will remain unchanged.

85 MAX CORNER ROUNDING

Defines the accuracy of corners within a selected tolerance. Initial default value is set to .05 inch. If this setting is zero, the control acts as if exact stop is commanded on each motion block. Parameter 134 is used as a floor so the machine will not slow down to extremely slow speeds. Alternatively, a G187 can be used in the program to alter the effective value of Setting 85 without permanently changing that setting. This method likewise takes advantage of the floor, but does not require that the machine be rebooted.



Setting 85 Example

86 M39 LOCKOUT

This is an on/off setting. Locks out the rotation of the tool changer.

**87 M06 RESETS OVERRIDE**

This is an on/off setting. When M06 is executed and this setting is on, any overrides are canceled and set to their programmed values.

88 RESET RESETS OVERRIDES

When the reset key is pressed and this is on, any overrides are cancelled and set to their programmed values.

95 THREAD CHAMFER SIZE

Used in G76 and G92 threading cycles, this setting represents the number of threads (lead) that will be used for a chamfer at the end of the thread. Values can range from 0 to 29.999 threads. Usually this is 1.000 thread.

103 CYC START/FH SAME KEY

This is an ON/OFF setting. When it is OFF, the machine operates normally. When it is ON, CYCLE START must be pressed and held to run a program. When CYCLE START is released, a FEED HOLD is generated.

This setting cannot be ON while Setting 104 is ON. When one of them is set to ON, the other will automatically turn OFF. This setting can be changed while running a program.

104 JOG HANDL TO SNGL BLK

This is an ON/OFF setting. When it is OFF, the machine operates normally. When it is ON, and SINGLE BLOCK is selected, the jog handle can be used to single step through a program.

Reversing the jog handle direction will generate a FEED HOLD. This can be useful when an unexpected long motion block is encountered.

CYCLE START must be used to begin running a program.

This setting cannot be ON while Setting 103 is ON. When one of them is set to ON, the other will automatically turn OFF. This setting can be changed while running a program.

108 QUICK ROTARY G28

This is an ON/OFF setting. When it is ON, the G28 command will return a rotary axis only back to zero when it has been commanded to greater than 360 degrees. For example, if a rotary axis has been commanded 361 degrees, and this setting is on, a G28 command will move it back only one degree instead of "unwinding" it all 361 degrees. Note that other G codes, such as G00 and G01, are unaffected by this setting.

109 WARM-UP TIME IN MIN.

This is the number of minutes (maximum= 300 minutes from the time the machine is powered on) to apply the compensation. If it is set to zero no compensation will be applied.

110 WARMUP X DISTANCE**111 WARMUP Y DISTANCE****112 WARMUP Z DISTANCE**

Setting 110, 111 and 112 specify the amount of compensation (maximum= +/- .0020" or +/- .051mm) applied to the X, Y and Z axis respectively. No compensation is applied if the setting is zero.

114 Conveyor cycle time (minutes)

See next entry



115 Conveyor On- Time (minutes)

The above two settings control the intermittent chip conveyor function. If setting 114 is zero, the chip conveyor will behave normally. If it is set to some number of minutes, the chip conveyor will automatically turn itself off after the number of minutes specified by setting 115, then turn itself back on later.

Setting 114 controls how often the cycle is to be repeated, that is, if setting 114 is set to 30 and setting 115 is set to 2, the chip conveyor will turn itself on every half hour, run for two minutes, then turn itself off. On-time should be set no greater than 80% of cycle time.

Note: The CHIP FWD button (or M31) will start the conveyor in the forward direction and activate the cycle.

Note: The CHIP REV button (or M32) will start the conveyor in the reverse direction and activate the cycle.

Notes: The CHIP STOP button (or M33) will stop the conveyor and cancel the cycle. If Parameter 209 CNVY DR OVRD bit 16 is set to 0 and the chip conveyor is cycling, opening the door will cause the conveyor to stop and suspend cycling. When the door is closed, the cycle will resume.

If Parameter 209 CNVY DR OVRD is set to 1 and the chip conveyor is cycling, the conveyor will continue to run when the door is open but will stop at the end of the cycle and cancel the cycle. When the door is closed, the cycle will resume.

Under no circumstances will the chip conveyor automatically start running when the door is open.

116 PIVOT LENGTH only changeable in DEBUG mode

This is to prevent accidental loss of the setting value. Setting 116 is intended to be set once when the machine is first built, then never changed unless the machine is physically altered. A setting file can be loaded from disk or RS-232 and the values of all settings will be replaced. However, now, if the control is not in DEBUG mode at the time, and the value of setting 116 is to be changed, all setting except 116 will be changed and alarm 202 SETTING CRC ERROR will be generated.

117 G143 GLOBAL OFFSET

This setting is provided for customers who have several VR-Series mills and want to transfer the programs and tool from one to another. The user can enter the pivot length difference into this setting, and it will be applied to the G143 tool length compensation.

118 M99 BUMPS M30 CNTRS

When this setting is turned ON, an M99 will increment the M30 counters that are visible by pressing CURNT COMNDS and PAGE DOWN twice. Note that an M99 will only increment the counters in loop mode in a main program, not a sub program. An M99 used as a subprogram return or used with a P value to jump to another part of the program will not increment the M30 counters.

119 OFFSET LOCK

This is an ON / OFF setting. When it is OFF, no special functions occur. When it is ON, the user is prevented from altering any of the offsets. However, programs which alter offsets will still be able to do so.

120 MACRO VAR LOCK

This is an ON / OFF setting. When it is OFF, no special functions occur. When it is ON, the user is prevented from altering any of the macro variables. However, programs which alter macro variables will still be able to do so.



THIS PAGE INTENTIONALLY LEFT BLANK



9. PARAMETERS

Parameters are seldom-modified values that change the operation of the machine. These include servo motor types, gear ratios, speeds, stored stroke limits, lead screw compensations, motor control delays and macro call selections. These are all rarely changed by the user and should be protected from being changed by the parameter lock setting. If you need to change parameters, contact HAAS or your dealer. Parameters are protected from being changed by Setting 7.

The Settings page lists some parameters that the user may need to change during normal operation and these are simply called "Settings". Under normal conditions, the parameter displays should not be modified. A complete list of the parameters is provided here.

The PAGE UP, PAGE DOWN, up and down cursor keys, and the jog handle can be used to scroll through the parameter display screens in the control. The left and right cursor keys are used to scroll through the bits in a single parameter.

PARAMETER LIST

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.
	REV PHASING	Used to reverse motor phasing.
	DISABLED	Used to disable any axis.
	Z CH ONLY	With A only, indicates that no home switch.
	AIR BRAKE	With A only, indicates that air brake is used.
	DISABLE Z T	Disables encoder Z test (for testing only).
	SERVO HIST	Graph of servo error (for diagnostics only).
	INV HOME SW	Inverted home switch (N.C. switch).
	INV Z CH	Inverted Z channel (normally high).
	CIRC. WRAP.	(Future Option - Not Yet Implemented) With A only, causes 360 wrap to return to 0.
	NO I IN BRAK	With A only, removes I feedback when brake is active.
	LOW PASS +1X	Adds 1 term to low pass filter.
	LOW PASS +2X	Adds two terms to low pass filter.
	OVER TEMP NC	Selects a normally closed overheat sensor in motor.
	CABLE TEST	Enables test of encoder signals and cabling.



Z TEST HIST	History plot of Z channel test data.
SCALE FACT/X	If set to 1, the scale ratio is interpreted as divided by X; where X depends on bits SCALE/X LO and SCALE/XHI.
INVIS AXIS	Used to create an invisible axis.
ROT ALM LMSW	Rotary alarms at the limit switch.
ROT TRVL LIM	Rotary travel limits are used.
UNDEFINED	
D FILTER X8	Enables the 8 tap FIR filter. Used to eliminate high frequency vibrations, depending on the axis motor.
D FILTER X4	Enables the 4 tap FIR filter. Used to eliminate high frequency vibrations, depending on the axis motor.
TORQUE ONLY	For HAAS diagnostic use only.
3 EREV/MREV	For HAAS diagnostic use only.
2 EREV/MREV	For HAAS diagnostic use only.
BRUSH MOTOR	Enables the brushless motor option.
LINEAR DISPL	This bit changes the display from degrees to inches (or millimeters) on the A and B axes.
SCALE/X LO	With SCALE/X HI bit, determines the scale factor used in bit SCALE FACT/X,
SCALE/X HI	With SCALE/X LO bit, determines the scale factor used in bit SCALE FACT/X. See below:

HI	LO	
0	0	3
0	1	5
1	0	7
1	1	9

Parameter 2 X P GAIN
Proportional gain in servo loop.

Parameter 3 X D GAIN
Derivative gain in servo loop.

Parameter 4 X I GAIN
Integral gain in servo loop.



Parameter 5 X RATIO (STEPS/UNIT)
The number of steps of the encoder per unit of travel. Encoder steps supply four (4) times their line count per revolution. Thus, an 8192 line encoder and a 6mm pitch screw give:

$$8192 \times 4 \times 25.4 / 6 = 138718$$

(5 steps per unit inch/mm ratio)

Parameter 6 X MAX TRAVEL (STEPS)
Max negative direction of travel from machine zero in encoder steps. Doesnot apply to A-axis. Thus a 20 inch travel, 8192 line encoder and 6 mm pitch screw give:

$$20.0 \times 138718 = 2774360$$

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. This is the maximum allowable error in Hz between the commanded speed and the actual speed. The purpose of this parameter is to prevent "motor runaway" in case of phasing reversal, or bad parameters. If this parameter is set to 0, it defaults to 1/4 of parameter 183 Max Frequency.

Parameter 10 X FUSE LEVEL
Used to limit average power to motor. If not set correctly, this parameter can cause an "overload" alarm.

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, an 8192 line encoder gives:

$$8192 \times 4 = 32768$$

Parameter 13 X BACKLASH
Backlash correction in encoder steps.

Parameter 14 X DEAD ZONE
Dead zone correction for driver electronics. Units are 0.0000001 seconds.

Parameter 15 Y SWITCHES
See Parameter 1 for description.



Parameter	16	Y P GAIN	See Parameter 2 for description.
Parameter	17	Y D GAIN	See Parameter 3 for description.
Parameter	18	Y I GAIN	See Parameter 4 for description.
Parameter	19	Y RATIO (STEPS/UNIT)	See Parameter 5 for description.
Parameter	20	Y MAX TRAVEL (STEPS)	See Parameter 6 for description.
Parameter	21	Y ACCELERATION	See Parameter 7 for description.
Parameter	22	Y MAX SPEED	See Parameter 8 for description.
Parameter	23	Y MAX ERROR	See Parameter 9 for description.
Parameter	24	Y FUSE LEVEL	See Parameter 10 for description.
Parameter	25	Y BACK EMF	See Parameter 11 for description.
Parameter	26	Y STEPS/REVOLUTION	See Parameter 12 for description.
Parameter	27	Y BACKLASH	See Parameter 13 for description.
Parameter	28	Y DEAD ZONE	See Parameter 14 for description.
Parameter	29	Z SWITCHES	See Parameter 1 for description.
Parameter	30	Z P GAIN	See Parameter 2 for description.



Parameter	31	Z D GAIN	See Parameter 3 for description.
Parameter	32	Z I GAIN	See Parameter 4 for description.
Parameter	33	Z RATIO (STEPS/UNIT)	See Parameter 5 for description.
Parameter	34	Z MAX TRAVEL (STEPS)	See Parameter 6 for description.
Parameter	35	Z ACCELERATION	See Parameter 7 for description.
Parameter	36	Z MAX SPEED	See Parameter 8 for description.
Parameter	37	Z MAX ERROR	See Parameter 9 for description.
Parameter	38	Z FUSE LEVEL	See Parameter 10 for description.
Parameter	39	Z BACK EMF	See Parameter 11 for description.
Parameter	40	Z STEPS/REVOLUTION	See Parameter 12 for description.
Parameter	41	Z BACKLASH	See Parameter 13 for description.
Parameter	42	Z DEAD ZONE	See Parameter 14 for description.
Parameter	43	A SWITCHES	See Parameter 1 for description AND make sure that this parameter is set to enable the fourth axis before you try to enable the fourth axis from settings.
Parameter	44	A P GAIN	See Parameter 2 for description.
Parameter	45	A D GAIN	See Parameter 3 for description.



- Parameter 46 A I GAIN
See Parameter 4 for description.
- Parameter 47 A RATIO (STEPS/UNIT)
This parameter defines the number of encoder steps required to complete one full rotation of the platter. For example an HRT 210 with a 90:1 gear ratio, a final drive ratio of 2:1, and an encoder count of 2000 lines would be:
$$2000 \times 4 \times (90 \times 2) / 360 = 2000 \text{ steps}$$

for a brushless HRT 210 with a 90:1 gear ratio, a final drive ratio of 2:1 and an encoder count of 8192 the formula would be:
$$8192 \times 4 \times (90 \times 2) / 360 = 16384 \text{ steps}$$

If for example 16384 ended up being 13107.2 (non integer) the user must make sure the single bits SCALE FACT/X and the COMBINATION OF SCALE/X LO and SCALE/X HI are turned on in parameter 43. When the scale factor/x bit is 1 the scale ratio is interpreted as divide by X: where X depends on scale/ x lo and scale/ x hi (see parameter 1 for scale/ x lo and scale x hi values). For example:
$$8192 \times 4 \times (72 \times 2) / 360 = 13107.2$$

You would then turn on the scale fact/x bit and the scale/ x lo bit which would give you a factor of 5 thus:
$$13107.2 \times 5 = 65536 \text{ encoder steps}$$
- Parameter 48 A MAX TRAVEL (STEPS)
See Parameter 6 for description. Normally this parameter would not apply to the A axis, however this parameter is used on mills with a gimbaled spindle (VR-Series mills). On a VR-series mill this parameter is used to limit the amount of angular movement of the spindle (A and B axes). The A and B axes are limited in movement to a distance between negative MAX TRAVEL, and positive TOOL CHANGE OFFSET. On VR-Series mills A and B axes ROT TRVL LIM must be set to 1, MAX TRAVEL and TOOL CHANGE OFFSET must be calibrated and set correctly.
- 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	Stops spindle then turns the power off at EMERGENCY STOP
RIGID TAP	Indicates hardware option for rigid tap.
REV SPIN ENC	Reverses sense direction of spindle encoder.
REPT RIG TAP	Selects repeatable rigid tapping.
EX ST MD CHG	Selects exact stop in moves when mode changes.
SAFETY CIRC.	This enables safety hardware, if machine is so equipped.
SP DR LIN AC	Selects linear deceleration for rigid tapping. 0 is quadratic.
PH LOSS DET	When enabled, will detect a phase loss.
COOLANT SPGT	Enables coolant spigot control and display.
OVER T IS NC	Selects control over temp sensor as N.C.
SKIP OVERSHT	Causes Skip (G31) to act like Fanuc and overshoot sense point.
NONINV SP ST	Non-inverted spindle stopped status.
SP LOAD MONI	Spindle load monitor option is enabled.
SP TEMP MONI	Spindle temperature monitor option is enabled.
ENA ROT & SC	Enables rotation and scaling.
ENABLE DNC	Enables DNC selection from MDI.
ENABLE BGEDT	Enables BACKGROUND EDIT mode.
ENA GRND FLT	Enables ground fault detector.
M19 SPND ORT	This bit makes the P and R codes a protected feature which can only be enabled with an unlock code. The unlock code will be printed on the parameter listing of all new machines. If this bit is set to 0, an M19 will orient the spindle to 0 degrees regardless of the value of any P or R code in the same block. If this is set to 1, a P code in the block will cause the spindle to be oriented to the specified angle such as P180. Alternately, a decimal R code can be used, such as R180.53. Note that the P and R codes only work on a vector drive machine.



ENABLE MACRO	Enables macro functions.
INVERT SKIP	Invert sense of skip to active low=closed.
HANDLE CURSR	Enable use of jog handle to move cursor.
NEG WORK OFS	Selects use of work offsets in negative direction.
SPIN COOLANT	Enables spindle low oil pressure detection.
ENA QUIKCODE	Enables conversational programming.
OILER ON/OFF	Enables oiler power when servos or spindle is in motion.
NC OVER VOLT	Inverts sense of over voltage signal.
DOOR STOP SP	Enables functions to stop spindle and manual operations 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 MAXIMUM FEED 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	61 TURRET STOP DELAY Maximum delay allowed in motion of tool turret. Units are milliseconds. After this time, an alarm is generated.
Parameter	62 SHUTTLE START DELAY This parameter is used to specify the time (in milliseconds) needed to allow the tool pocket to settle (stop bouncing) after being lowered in preparation for a tool change.
Parameter	63 SHUTTLE STOP DELAY Maximum delay allowed in motion of tool shuttle. Units are milliseconds. After this time, an alarm is generated.
Parameter	64 Z TOOL CHANGE OFFSET On Vertical mills: For Z-axis; displacement from home switch to tool change position and machine zero. About 4.6 inches, so for an 8192 line encoder this gives: $4.6 \times 138718 = 638103$ On Horizontal mills, this parameter is not used. It should be set to zero.
Parameter	65 NUMBER OF TOOLS Number of tool positions in tool changer. This number must be set to the configuration machine. The maximum number of tool positions is 32.



Parameter	66 SPINDLE ORI DELAY Maximum delay allowed when orienting spindle. Units are milliseconds. After this time, an alarm is generated.
Parameter	67 GEAR CHANGE DELAY Maximum delay allowed when changing gears. Units are milliseconds. After this time, an alarm is generated.
Parameter	68 DRAW BAR MAX DELAY Maximum delay allowed when clamping and unclamping tool. Units are milliseconds. After this, time an alarm is generated.
Parameter	69 A AIR BRAKE DELAY Delay provided for air to release from brake on A-axis prior to moving. Units are milliseconds.
Parameter	70 MIN SPIN DELAY TIME Minimum delay time in program after commanding new spindle speed and before proceeding. Units are milliseconds.
Parameter	71 DRAW BAR OFFSET Offset provided in motion of Z-axis to accommodate the tool pushing out of the spindle when unclamping tool. Units are encoder steps.
Parameter	72 DRAW BAR Z VEL UNCL Speed of motion in Z-axis to accommodate tool pushing out of the spindle when unclamping tool. Units are encoder steps per second.
Parameter	73 SP HIGH G/MIN SPEED Command speed used to rotate spindle motor when orienting spindle in high gear. Units are maximum spindle RPM divided by 4096. This parameter is not used in machines equipped with a Haas vector drive.
Parameter	74 SP LOW G/MIN SPEED Command speed used to rotate spindle motor when orienting spindle in low gear. Units are maximum spindle RPM divided by 4096. This parameter is not used in machines equipped with a Haas vector drive.
Parameter	75 GEAR CHANGE SPEED Command speed used to rotate spindle motor when changing gears. Units are maximum spindle RPM divided by 4096.
Parameter	76 LOW AIR DELAY Delay allowed after sensing low air pressure before alarm is generated. Alarm skipped if air pressure returns before delay. Units are 1/50 seconds.
Parameter	77 SP LOCK SETTLE TIME Required time in milliseconds that the spindle lock must be in place and stable before spindle orientation is considered complete.



Parameter	78	GEAR CH REV TIME	Time in milliseconds before motor direction is reversed while in a gear change.
Parameter	79	SPINDLE STEPS/REV	Sets the number of encoder steps per revolution of the spindle. Applies only to rigid tapping option.
Parameter	80	MAX SPIN DELAY TIME	The maximum delay time control will wait for spindle to get to commanded speed or to get to zero speed. Units are milliseconds.
Parameter	81	M MACRO CALL O9000	M code that will call O9000. Zero causes no call.
Parameter	82	M MACRO CALL O9001	Same as 81
Parameter	83	M MACRO CALL O9002	Same as 81
Parameter	84	M MACRO CALL O9003	Same as 81
Parameter	85	M MACRO CALL O9004	Same as 81
Parameter	86	M MACRO CALL O9005	Same as 81
Parameter	87	M MACRO CALL O9006	Same as 81
Parameter	88	M MACRO CALL O9007	Same as 81
Parameter	89	M MACRO CALL O9008	Same as 81
Parameter	90	M MACRO CALL O9009	Same as 81
Parameter	91	G MACRO CALL O9010	G code that will call O9010. Zero causes no call.
Parameter	92	G MACRO CALL O9011	Same as 91



Parameter	93	G MACRO CALL O9012 Same as 91
Parameter	94	G MACRO CALL O9013 Same as 91
Parameter	95	G MACRO CALL O9014 Same as 91
Parameter	96	G MACRO CALL O9015 Same as 91
Parameter	97	G MACRO CALL O9016 Same as 91
Parameter	98	G MACRO CALL O9017 Same as 91
Parameter	99	G MACRO CALL O9018 Same as 91
Parameter	100	G MACRO CALL O9019 Same as 91
Parameter	101	IN POSITION LIMIT X How close motor must be to endpoint before any move is considered complete when not in exact stop (G09 or G61). Units are encoder steps. This parameter does not apply to feeds.
Parameter	102	IN POSITION LIMIT Y Same definition as Parameter 101.
Parameter	103	IN POSITION LIMIT Z Same definition as Parameter 101.
Parameter	104	IN POSITION LIMIT A Same definition as Parameter 101.
Parameter	105	X MAX CURRENT Fuse level in % of max power to motor. Applies only when motor is stopped.
Parameter	106	Y MAX CURRENT Same definition as Parameter 105.
Parameter	107	Z MAX CURRENT Same definition as Parameter 105.



Parameter	108 A MAX CURRENT Same definition as Parameter 105.
Parameter	109 D*D GAIN FOR X Second derivative gain in servo loop.
Parameter	110 D*D GAIN FOR Y Second derivative gain in servo loop.
Parameter	111 D*D GAIN FOR Z Second derivative gain in servo loop.
Parameter	112 D*D GAIN FOR A Second derivative gain in servo loop.
Parameter	113 X ACC/DEC T CONST Exponential acceleration time constant. Units are 1/10000 seconds. This parameter provides for a constant ratio between profiling lag and servo velocity. It is also the ratio between velocity and acceleration.
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 200ths of a step/ms/ms for spindle motor.
Parameter	121 X PHASE OFFSET The motor phase offset for X motor. This is arbitrary units.
Parameter	122 Y PHASE OFFSET See Parameter 121 for description.



Parameter	123 Z PHASE OFFSET See Parameter 121 for description.
Parameter	124 A PHASE OFFSET See Parameter 121 for description.
Parameter	125 X GRID OFFSET This parameter shifts the effective position of the encoder Z pulse. It can correct for a positioning error of the motor or home switch.
Parameter	126 Y GRID OFFSET See Parameter 125 for description.
Parameter	127 Z GRID OFFSET See Parameter 125 for description.
Parameter	128 A GRID OFFSET See Parameter 125 for description.
Parameter	129 GEAR CH SETTLE TIME Gear change settle time. This is the number of one millisecond samples that the gear status must be stable before considered in gear.
Parameter	130 GEAR STROKE DELAY This parameter controls the delay time to the gear change solenoids when performing a gear change.
Parameter	131 MAX SPINDLE RPM This is the maximum RPM available to the spindle. When this speed is programmed, the D-to-A output will be +10V and the spindle drive must be calibrated to provide this.
Parameter	132 Y SCREW COMP. COEF. This is the coefficient of heating of the lead screw and is used to decrease or shorten the screw length.
Parameter	133 Z SCREW COMP. COEF. This is the coefficient of heating of the lead screw and is used to decrease or shorten the screw length.
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 $34/138718 = 0.00025$ inch.

Note: To change the values of parameters 134-137 permanently the machine must be rebooted.

Parameter 138 X FRICTION COMPENSATION

Parameter 139 Y FRICTION COMPENSATION

Parameter 140 Z FRICTION COMPENSATION

Parameter 141 A FRICTION COMPENSATION
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. Units are encoder counts.

Parameter 145 X ACCEL FEED FORWARD

Parameter 146 Y ACCEL FEED FORWARD

Parameter 147 Z ACCEL FEED FORWARD

Parameter 148 A ACCEL FEED FORWARD
These parameters set the feed forward gain for the axis servo. They have no units.

Parameter 149 PRE-CHARGE DELAY
This parameter sets the delay time from pre-charge to tool release. Units are milliseconds.

Parameter 150 MAX SP RPM LOW GEAR
Max spindle RPM in low gear.

Parameter 151 B SWITCHES
See Parameter 1 for description.



Parameter	152 B P GAIN See Parameter 2 for description.
Parameter	153 B D GAIN See Parameter 3 for description.
Parameter	154 B I GAIN See Parameter 4 for description.
Parameter	155 B RATIO (STEPS/UNIT) See Parameter 47 for description.
Parameter	156 B MAX TRAVEL (STEPS) See Parameter 6 for description. Normally this parameter would not apply to the A axis, however this parameter is used on mills with a gimbaled spindle (VR-Series mills). On a VR-series mill this parameter is used to limit the amount of angular movement of the spindle (A and B axes). The A and B axes are limited in movement to a distance between negative MAX TRAVEL, and positive TOOL CHANGE OFFSET. On VR-Series mills A and B axes ROT TRVL LIM must be set to 1, MAX TRAVEL and TOOL CHANGE OFFSET must be calibrated and set correctly.
Parameter	157 BACCELERATION See Parameter 7 for description.
Parameter	158 B MAX SPEED See Parameter 8 for description.
Parameter	159 B MAX ERROR See Parameter 9 for description.
Parameter	160 B FUSE LEVEL See Parameter 10 for description.
Parameter	161 B BACK EMF See Parameter 11 for description.
Parameter	162 B STEPS/REVOLUTION See Parameter 12 for description.
Parameter	163 B BACKLASH See Parameter 13 for description.
Parameter	164 B DEAD ZONE See Parameter 14 for description.
Parameter	165 IN POSITION LIMIT B Same definition as Parameter 101.



Parameter	166	B MAX CURRENT	Same definition as Parameter 105.
Parameter	167	D*D GAIN FOR B	Second derivative gain in servo loop.
Parameter	168	B ACC/DEC T CONST	Same definition as Parameter 113.
Parameter	169	B PHASE OFFSET	See Parameter 121 for description.
Parameter	170	B GRID OFFSET	See Parameter 125 for description.
Parameter	171	B EXACT STOP DIST.	See Parameters 134 for description.
Parameter	172	B FRICTION COMPENSATION	See Parameter 138 for description.
Parameter	173	B ACCEL FEED FORWARD	Same description as Parameter 145.
Parameter	174	X SCREW COMP. COEF.	This is the coefficient of heating of the lead screw and is used to decrease or shorten the screw length.
Parameter	175	B AIR BRAKE DELAY	Delay provided for air to release from brake on B-axis prior to moving. Units are milliseconds.

Note: The C-axis parameters (176-200) are used to control the Haas Vector Drive. Parameter 278 bit HAAS VECT DR must be set to 1 for these parameters to be available.

Parameter	176	C SWITCHES	See Parameter 1 for description.
Parameter	177	C P GAIN	See Parameter 2 for description.
Parameter	178	C D GAIN	See Parameter 3 for description.
Parameter	179	C I GAIN	See Parameter 4 for description.



- Parameter 180 SLIP GAIN
This name is used when a Vector Drive is installed. The slip rate calculated depends on two other variables: speed and current.
$$\text{Slip rate} = \text{slip gain} \times (\text{speed}/\text{max speed}) \times (\text{current}/\text{max current})$$

The slip gain value is the value that slip rate would assume at maximum speed, and maximum current (16.384=1 Hz). If a Vector Drive is not installed, this parameter is called: C AXIS RATIO (STEPS/UNIT) and is not used.
- Parameter 181 MIN SLIP
This name is used when a Vector Drive is installed. The minimum value allowed from the slip rate. From the equation:
$$\text{Slip rate} = \text{slip gain} \times (\text{speed}/\text{max speed}) \times (\text{current}/\text{max current})$$

it can be seen that at a zero speed, the slip rate would become zero. Therefore a minimum value for slip rate is required. (16.384 = 1Hz). If a Vector Drive is not installed, this parameter is called: C AXIS MAX TRAVEL (STEPS) and is not used.
- Parameter 182 CACCELERATION
This parameter is not used unless a Vector Drive is installed. See Parameter 7 for description. If a Vector Drive is installed and the name of the Parameter is ACCELERATION. The value is the units of encoder steps / s² at the motor.
- Parameter 183 C MAX FREQ
This parameter is not used unless a Vector Drive is installed. The frequency at which the motor will be run when maximum spindle RPM is commanded. Units: 0.01 Hz (two implied decimal places). If a Vector Drive is not installed, this parameter is called C axis MAX SPEED and is not used.
- Parameter 184 C MAX ERROR
The maximum allowable error (in Hz) between commanded spindle speed and actual speed. If set to zero, it will default to 1/4 of Parameter 183.
- Parameter 185 C FUSE LEVEL
See Parameter 10 for description.
- Parameter 186 C DECELERATION
This parameter is used when a Vector Drive is installed. Maximum deceleration of axis in encoder steps per second per second. If a vector Drive is not installed, this parameter is called C axis BACK EMF, and is not used.
- Parameter 187 C HIGH GEAR STEPS/REV
This parameter is used when a Vector Drive is installed. The value is the number of encoder steps per revolution of the motor when the transmission is in high gear. If the machine does not have a transmission, this is simply the number of encoder steps per revolution of the motor. If a Vector Drive is not installed, this parameter is called C axis STPS / REVOLUTION and is not used.
- Parameter 188 C ORIENT GAIN
This parameter is used if a Vector Drive is installed. The value is the proportional gain used in the position control loop when performing a spindle orientation. If a Vector Drive is not installed, this parameter is called C axis BACKLASH, and is not used.



- Parameter 189 C BASE FREQ
This parameter is used when a Vector Drive is installed. This is the rated frequency of the motor. If a Vector Drive is not installed, this parameter is called C axis DEAD ZONE, and is not used.
- Parameter 190 C HI SP CURR LIM
This parameter is used when a Vector Drive is installed. At speeds higher than the base frequency, the maximum current that is applied to the motor must be reduced. This is done linearly from base frequency to max frequency. this value is the max current at the max frquncy. If a Vector Drive is not installed, this parameter is called C axis IN POSITION LIMIT, and is not used.
- Parameter 191 C MAX CURRENT
Same definition as Parameter 105.
- Parameter 192 C MAG CURRENT
This parameter is used when a Vector Drive is installed. This is the magnetization component of the current in the motor, also called the flux or field current. If a Vector Drive is not installed, this parameter is called C axis D*D GAIN, and is not used.
- Parameter 193 C SPIN ORIENT MARGIN
This parameter is used when a Vector Drive is installed. When a spindle orientation is done, if the actual position of the spindle is within this value (plus or minus), the spindle will be considered locked. Otherwise, the spindle will not be locked. If a Vector Drive is not installed, this parameter is called C axis ACC / DEC T CONST, and is not used.
- Parameter 194 SPINDLE STOP FREQ
This parameter is used when a Vector Drive is installed. The spindle is considered to be stopped (discrete input SP ST*=0) when the speed drops below this value. Units are encoder steps/millisecond. If a Vector Drive is not installed, this parameter is called C axis PHASE OFFSET, and is not used.
- Parameter 195 C START/STOP DELAY
This parameter is used when a Vector Drive is installed. This delay is used at the start of motion to magnetize the rotor before acceleration starts. When the motor comes to a stop it remains energized for this amount of time. Units are in milliseconds. If a Vector Drive is not installed, this parameter is called C axis GRID OFFSET, and is not used.
- Parameter 196 C ACCEL LIMIT LOAD
This parameter is used when a Vector Drive is installed. This is the % load limit during acceleration. If the load reaches this limit during acceleration the control slows down the acceleration. If a Vector Drive is not installed, this parameter is called C axis EXACT STOP DISTANCE, and is not used.
- Parameter 197 SWITCH FREQUENCY. Unit: Hz.
This parameter is used when a Vector Drive is installed. This is the frequency at which the spindle motor windings are switched. Note that there is a hysteresis band around this point, defined by parameter 198. If a Vector Drive is not installed, this parameter is called C axis FRICTION FACTOR, and is not used.



Parameter	198 SWITCH HYSTERESIS. UNIT:Hz. This parameter is used when a Vector Drive is installed. This defines the \pm hysteresis band around parameter 197. For example if par. 197 is 85 Hz, and par. 198 is 5Hz, the switching will take place at 90Hz when the spindle is speeding up, and at 80 Hz when the spindle is slowing down. If a Vector Drive is not installed, this parameter is called C axis ACCEL FEED FORWARD, and is not used.
Parameter	199 PRE-SWITCH DELAY. UNIT: ms. This parameter is used when a Vector Drive is installed. This is the amount of time allowed for the current in the motor to drop before the winding change contactors are switched. If a Vector Drive is not installed, this parameter is called C axis THERMAL COMP. COEF., and is not used.
Parameter	200 POST- SWITCH DELAY. UNIT: ms This parameter is used when a Vector Drive is installed. This is the amount of time allowed for the contactors to stabilize after a switch is commanded, before current is applied to the motor. If a Vector Drive is not installed, this parameter is called C axis AIR BRAKE DELAY, and is not used.
Parameter	201 X SCREW COMP. COEF. This is the coefficient of heating of the lead screw and is used to shorten the screw length.
Parameter	205 A SCREW COMP. COEF. This parameter should be set to 0.
Parameter	206 SPIGOT POSITIONS Vertical mills only. Maximum number of spigot positions.
Parameter	207 SPIGOT TIMEOUT (MS) Vertical mills only. 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: <div style="display: flex; justify-content: space-between;"> <div style="width: 30%;">HORIZONTAL</div> <div>When set to (1), the control identifies the machine as a horizontal mill. The control will then make the necessary adjustments, such as enabling the horizontal tool changer.</div> </div> <div style="display: flex; justify-content: space-between;"> <div style="width: 30%;">RST STOPS T.C.</div> <div>Tool changer can be stopped with RESET button.</div> </div> <div style="display: flex; justify-content: space-between;"> <div style="width: 30%;">BRIDGE</div> <div>This bit should be set to 1 for the Bridge Mill, and 0 for all other models.</div> </div> <div style="display: flex; justify-content: space-between;"> <div style="width: 30%;">M21-28 @ 540</div> <div>When enabled (1), M21-28 is installed at cable 540.</div> </div> <div style="display: flex; justify-content: space-between;"> <div style="width: 30%;">ENA CONVEYOR</div> <div>Enables chip conveyor, if machine is so equipped.</div> </div>



50% RPD KBD	When (1) the control will support the new style keyboards with the 50% rapid traverse key. For controls without a 50% rapid keypad set this bit to (0).
FRONT DOOR	When enabled the control will look for an additional door switch and will generate an operator message.
TC Z NO HOME	In Horizontal mills only. This bit prevents Z-axis motion to machine zero prior to a tool change.
M36 AUTO MOT	In Horizontal only. When set to (1), an M36 rotates the A-axis after the PART READY button is pressed.
AUX AXIS TC	In Horizontal mills only. When enabled, means the tool changer carousel is driven by an aux. axis.
SPIGOT KEY INV	This bit controls the direction the spigot moves when the Coolant Up and Coolant Down buttons are pressed. Changing this bit reverses the direction the spigot moves when the buttons are pressed. It has no effect on the direction the spigot moves when commanded by the M34 and M35 codes.
T SUBROUTINE	Reserved for future use.
SPIN Y ENCDR	For Lathe only. When enabled, spindle encoder input is to the Y-axis.
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 and M22. When this bit is set, the control expects to see the conveyor hooked up to M27 and M28.
GREEN BEACON	When (1) user relay M25 is used to flash a beacon. If the control is in a reset state, the beacon will be off. If the control is running normally, the beacon will be steadily on. If the control is in a M00, M01, M02, M30 feedhold, or single block state, then the beacon will flash.
RED BEACON	When (1) user relay M26 is used to flash a beacon. The beacon flashes if the control is experiencing an alarm or emergency stop condition.
CONVY DR OVRD	When (1) the conveyor will continue to run with the door open. When (0) the conveyor will stop when the door is open, but will resume when the door is closed. For safety it is recommended that the bit be set to (0).
DSBL CLNT IN	If set to 1 low coolant input will not be used.
DSC INP PR	Discrete pallet rotate/part ready; inputs enabled if set to 1.
RMT TOOLS RLS	If set to 1, allows use of remote tool release button on spindle head.
FLOPPY ENABL	If set to 1, enables the optional floppy drive.
TCR KEYPAD	If set to 1, enables tool changer restore button on keypad.



MCD RLY BRD	If set to 1, adds 8 additional relays, for a total of 40. These additional relays (M21-M28) become available on a secondary board, and are shown on the discrete outputs page.
TSC ENABLE	When set to 1, "DSBL CLNT IN" bit is ignored, M24, M54 and M64 are disabled, and TSC will operate. When set to zero, the control functions normally.
AUX JOG NACC	If the jog handle is moved rapidly the auxillary axis will not develop extremely large lags.
ALISM PRGRST	Alias M codes during program restart.
DSBL JOG TST	Disables the encoder test for the jog handle.
AIR DR @ M24	Used on horizontal mills only.
P RDY @ Y160	Used on horizontal mills only.
APC ENABLED	Mills with an Automatic Pallet Changer, this bit must be set to 1. For all other mills, it must be set to 0
SPNDL NOWAIT	When (1), the machine will not wait for the spindle to come up to speed immediately after an M03 or M04 command. Instead, it will check and/or wait for the spindle to come up to speed immediately before the next interpolated motion is initiated. This bit does not affect rigid tapping or the TSC option.

Parameter	<p>210 X AXIS TOOL CHANGE OFFSET</p> <p>Used on the HS-2RP mill for X axis displacement from the home position to tool change position.</p> <p>If this parameter contains an incorrect value, a horizontal mill will crash when it does a tool change.</p>
Parameter	<p>211 Y AXIS TOOL CHANGE OFFSET</p> <p>Used on the HS-2RP mill for Y axis displacement from the home position to tool change position.</p> <p>If this parameter contains an incorrect value, a horizontal mill will crash when it does a tool change.</p>
Parameter	<p>214 D:Y CURRENT RATIO %. UNIT: %.</p> <p>This parameter is used with the Vector Drive. This defines the ratio between the two winding configurations. This default winding is Y, and the parameters are set for the Y winding. This number is used to adjust the parameters for the delta winding when the windings are switched. If a Vector Drive is not installed, this parameter is called C axis TOOL CHANGE OFFSET, and is not used.</p>
Parameter	<p>215 CAROUSEL OFFSET</p> <p>Used on horizontal mills only. Parameter used to align tool 1 of tool changing carousel precisely. Units are encoder steps.</p>
Parameter	<p>216 CNVYR RELAY DELAY</p> <p>Delay time in 1/50 seconds required on conveyor relays before another action can be commanded. Default is 50.</p>



- 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 2000.
- 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 5.
- Parameter 220 CONVYR RETRY TIMEOUT
Amount of time in 1/50 seconds between consecutive overcurrents in which the overcurrents is considered another retry. If this amount of time passes between overcurrents, then the retry count is set to (0). Default is 1500, 30 seconds.
- Parameter 221 MAX TIME NO DISPLAY
The maximum time (in 1/50 sec.) between screen updates.
- Parameter 222 ROTARY AXIS INCRMNT
For Horizontal mills only. This parameter sets the degrees of rotation of the A-axis at an M36 or Pallet Rotate.
- Parameter 223 AIR TC DOOR DELAY
For Horizontal mills only. This parameter sets the delay to open the tool changer door (in milliseconds). If the tool changer does not have a pneumatic door, this parameter is set to zero.
- Parameter 224 ROT AXIS ZERO OFFSET
This parameter shifts the zero point of A for a wheel fixture or tombstone.
- Parameter 225 MAX ROT AXIS ALLOW
For Horizontal mills with a wheel fixture only. This parameter sets the maximum rotation (in degrees) allowed before stopping at front door.
- Parameter 226 EDITOR CLIPBOARD
This parameter assigns a program number (nnnnn) to the contents of the clipboard (for the advanced editor).
- Parameter 227 FLOPPY DIR NAME
When the floppy drive is enabled and a directory is read the directory listing is placed into a program as comments. The program is then made the current program so the user can read the contents of the floppy drive. This parameter designates where to write the directory listing. Program 08999 is the default value.



Parameter	228 QUICKCODE FILE This parameter set the program numbers to store in the Quick Code definition program. Usually, this is 9999.
Parameter	229 X LEAD COMP 10E9 This parameter sets the X-axis lead screw compensation signed parts per billion.
Parameter	230 Y LEAD COMP 10E9 This parameter sets the Y-axis lead screw compensation signed parts per billion.
Parameter	231 Z LEAD COMP 10E9 This parameter sets the Z-axis lead screw compensation signed parts per billion.
Parameter	232 A LEAD COMP 10E9 This parameter sets the A-axis lead screw compensation signed parts per billion.
Parameter	233 B LEAD COMP 10E9 This parameter sets the B-axis lead screw compensation signed parts per billion.
Parameter	235 TSC PISTON SEAT With the 50 TSC option, the amount of time given for the piston to seat during system start-up. The default is 500 milliseconds. If machine has a 50 Taper spindle and the TSC option, this parameter must be set to 0 .
Parameter	236 TSC LOW PR FLT After the TSC system has stabilized following startup, Alarm 151 is generated if coolant pressure falls below 40 psi for the amount of time set in this parameter. The default is 1000 milliseconds.
Parameter	237 TSC CLNT LINE PURGE The amount of time given for the coolant to purge when the TSC system is shut off. This parameter may be increased by the user to a higher value to help purge coolant from small orifice tooling. The minimum (default) value is 2500 milliseconds.
Parameter	238 MAX TSC SPINDLE RPM When TSC is enabled and in use, this parameter limits the maximum spindle speed. Default value is 7500 RPM. On 50 taper machines, TSC can be run at the maximum speed of 5000 RPM
Parameter	239 SPNDL ENC STEPS/REV This parameter sets the number of encoder steps per revolution of the spindle encoder.
Parameter	240 1ST AUX MAX TRAVEL This parameter sets the maximum travel of the first auxillary (C) axis in the positive direction.
Parameter	241 2ND AUX MAX TRAVEL This parameter sets the maximum travel of the second auxillary (U) axis in the positive direction.



- Parameter 242 3RD AUX MAX TRAVEL
This parameter sets the maximum travel of the third auxillary (V) axis in the positive direction.
- Parameter 243 4TH AUX MAX TRAVEL
This parameter sets the maximum travel of the fourth auxillary (W) axis in the positive direction.
- Parameter 244 1ST AUX MIN TRAVEL
This parameter sets the maximum travel of the first auxillary (C) axis in the negative direction.
- Parameter 245 2ND AUX MIN TRAVEL
This parameter sets the maximum travel of the second auxillary (U) axis in the negative direction.
- Parameter 246 3RD AUX MIN TRAVEL
This parameter sets the maximum travel of the third auxillary (V) axis in the negative direction.
- Parameter 247 4TH AUX MIN TRAVEL
This parameter sets the maximum travel of the fourth auxillary (W) axis in the negative direction.
- Parameter 248 SMTCLY ON / OFF DLY
Vertical mills with sidemount tool changers only. For the Pasma serpentine type tool changer, it specifies the time needed (in milliseconds) between activating or deactivating the Direction relay and the Motor Enable relay. For the Disk type 1 tool changer, it specifies the time needed (in milliseconds) between turning off one relay and turning on the other one, when reversing the carousel.
- Parameter 249 TOOL CLAMP DELAY
This parameter provides a delay after the tool has been clamped and before retraction of the tool carousel at the end of a tool change. For most mills, this parameter should be set to zero. Units are milliseconds.
- Parameter 250 TOOL UNCLAMP DELAY
This parameter provides a delay after the tool has been unclamped and before the spindle is backed away at the beginning of a tool change. For most mills, this parameter should be set to zero. Units are in milliseconds.
- Parameter 255 CONVEYOR TIMEOUT
The amount of time (in minutes) the conveyor will operate without any machine motion or keyboard action. After this time, the conveyor will automatically shut off.
- Parameter 256 PALLET LOCK INPUT
Used in horizontal mills only.
- Parameter 257 SPINDL ORIENT OFFSET
If the machine is equipped with a spindle vector drive (as set in bit 7 of Parameter 278), this bit sets the spindle orientation offset. The offset is the number of encoder steps between the Z pulse and the correct spindle orientation position. It is used to orient the spindle properly anytime it needs to be locked, such as prior to a tool change, or orient spindle command.



Parameter	266	X SWITCHES
	Parameter 266 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:	
	X LIN SCALE EN	Used to enable linear scales for the X axis.
	X INVRT LN SCL	Used to invert the X-axis linear scale.
	X DSBL LS ZTST	Used to disable the linear scale Z test.
	X ZERO AXIS TC	Used to return axis to zero prior to tool change (VR-series) .
	X 2ND HOME BTN	Used to move axis to coordinate specified in Work Offset G129.
	X NEG COMP DIR	Used to negate the direction of thermal compensation.
	X DELAY AXIS 0	Used with an APL to ensure X axis is zeroed before A axis of APL
	X MAX TRAVEL INP	This bit is set to 1 on five axes machines. This bit indicates that there is a switch (visible through MOCON) that detects if the axis has rotated all the way round. It is used to tell the control to skip the first zero switch when zeroing so it can unwrap the cables.
Parameter	267	Y SWITCHES
	Parameter 267 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:	
	Y LIN SCALE EN	Used to enable linear scales for the Y axis.
	Y INVRT LN SCL	Used to invert the Y-axis linear scale.
	Y DSBL LS ZTST	Used to disable the linear scale Z test.
	Y ZERO AXIS TC	Used to return axis to zero prior to tool change (VR-series).
	Y 2ND HOME BTN	Used to move axis to coordinate specified in Work Offset G129.
	Y NEG COMP DIR	Used to negate the direction of thermal compensation.
	Y DELAY AXIS 0	Used with an APL to ensure Y axis is zeroed before A axis of APL.
	Y MAX TRAVEL INP	This bit is set to 1 on five axes machines. This bit indicates that there is a switch (visible through MOCON) that detects if the axis has rotated all the way round. It is used to tell the control to skip the first zero switch when zeroing so it can unwrap the cables.
Parameter	268	Z SWITCHES
	Parameter 268 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:	
	Z LIN SCALE EN	Used to enable linear scales for the Z axis.
	Z INVRT LN SCL	Used to invert the Z-axis linear scale



Z DSBL LS ZTST	Used to disable the linear scale Z test.
Z ZERO AXIS TC	Used to return axis to zero prior to tool change (VR-series) .
Z 2ND HOME BTN	Used to move axis to coordinate specified in Work Offset G129.
Z NEG COMP DIR	Used to negate the direction of thermal compensation.
Z DELAY AXIS 0	Used with an APL to ensure Z axis is zeroed before A axis of APL
Z MAX TRAVEL INP	This bit is set to 1 on five axes machines. This bit indicates that there is a switch (visible through MOCON) that detects if the axis has rotated all the way round. It is used to tell the control to skip the first zero switch when zeroing so it can unwrap the cables.

Parameter 269 A SWITCHES
Parameter 269 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:

A LIN SCALE EN	Used to enable linear scales for the A axis.
A INVRT LN SCL	Used to invert the A-axis linear scale.
A DSBL LS ZTST	Used to disable the linear scale Z test.
A ZERO AXIS TC	Used to return axis to zero prior to tool change (VR-series).
A 2ND HOME BTN	Used to move axis to coordinate specified in Work OPffset G129.
A DELAY AXIS 0	Used with an APL to ensure A axis is zeroed before B axis of APL.
AMAX TRAVEL INP	This bit is set to 1 on five axes machines. This bit indicates that there is a switch (visible through MOCON) that detects if the axis has rotated all the way round. It is used to tell the control to skip the first zero switch when zeroing so it can unwrap the cables.

Parameter 270 B SWITCHES
Parameter 269 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:

B LIN SCALE EN	Used to enable linear scales for the B axis.
B INVRT LN SCL	Used to invert the B-axis linear scale.
B DSBL LS ZTST	Used to disable the linear scale Z test.
B ZERO AXIS TC	Used to return axis to zero prior to tool change (VR-series).
B 2ND HOME BTN	Used to move axis to coordinate specified in Work Offset G129.
B DELAY AXIS 0	Used with an APL to ensure B axis is zeroed before A axis of APL.
B MAX TRAVEL INP	This bit is set to 1 on five axes machines. This bit indicates that there is a switch (visible through MOCON) that detects if the axis has rotated all the way round. It is used to tell the control to skip the first zero switch when zeroing so it can unwrap the cables.



Parameter	<p>271 C SWITCHES</p> <p>Parameter 271 is a collection of single-bit flags used to turn servo related functions on and off. This parameter is not used when machine is equipped with a Haas vector drive. The left and right cursor arrows are used to select the function being changed. All values are 0 or 1 only. The function names are:</p> <table> <tr> <td>C LIN SCALE EN</td><td>Used to enable linear scales for the C axis.</td></tr> <tr> <td>C INVRT LN SCL</td><td>Used to invert the C-axis linear scale.</td></tr> <tr> <td>C DSBL LS ZTST</td><td>Used to disable the linear scale Z test.</td></tr> <tr> <td>C ZERO AXIS TC</td><td>Used to return axis to zero prior to tool change (VR-series).</td></tr> <tr> <td>C 2ND HOME BTN</td><td>Used to move axis to coordinate specified in Work Offset G129.</td></tr> <tr> <td>C DELAY AXIS 0</td><td>Used with an APL to ensure C axis is zeroed before A axis of APL.</td></tr> </table>	C LIN SCALE EN	Used to enable linear scales for the C axis.	C INVRT LN SCL	Used to invert the C-axis linear scale.	C DSBL LS ZTST	Used to disable the linear scale Z test.	C ZERO AXIS TC	Used to return axis to zero prior to tool change (VR-series).	C 2ND HOME BTN	Used to move axis to coordinate specified in Work Offset G129.	C DELAY AXIS 0	Used with an APL to ensure C axis is zeroed before A axis of APL.
C LIN SCALE EN	Used to enable linear scales for the C axis.												
C INVRT LN SCL	Used to invert the C-axis linear scale.												
C DSBL LS ZTST	Used to disable the linear scale Z test.												
C ZERO AXIS TC	Used to return axis to zero prior to tool change (VR-series).												
C 2ND HOME BTN	Used to move axis to coordinate specified in Work Offset G129.												
C DELAY AXIS 0	Used with an APL to ensure C axis is zeroed before A axis of APL.												
Parameter	<p>272 X SCREW COMP T. CONST.</p> <p>This parameter is the thermal compensation time constant, and is the time constant governing the rate of cool down of the screw.</p>												
Parameter	<p>273 Y SCREW COMP T. CONST.</p> <p>This parameter is the thermal compensation time constant, and is the time constant governing the rate of cool down of the screw.</p>												
Parameter	<p>274 Z SCREW COMP T. CONST.</p> <p>This parameter is the thermal compensation time constant, and is the time constant governing the rate of cool down of the screw.</p>												
Parameter	<p>275 A SCREW COMP T. CONST.</p> <p>This parameter should be set to 0.</p>												
Parameter	<p>276 B SCREW COMP T. CONST.</p> <p>Parameter 276 is used in conjunction with parameter 205 for spindle head thermal compensation. This parameter should be set to -10,000</p>												
Parameter	<p>278 COMMON SWITCH 3</p> <p>Parameter 278 is a collection of general purpose single bit flags used to turnsome functions on and off. This bit will cause the machine to use discrete outputs 21 and 26 to command the shuttle to move in and out. On mills with the Air Driven Shuttle it must be set to 1. On all other mills it must be set to 0. The left and right cursor arrows are used to select the function being changed. All values are 0 or 1 only. The function names are:</p> <table> <tr> <td>INVERT G.B.</td><td>This bit allows an alternate gearbox configuration. It inverts the sense of the gearbox inputs. Used for 50 taper option.</td></tr> <tr> <td>DPR SERIAL</td><td>Causes the main serial inputs/outputs to go through the floppy video board.</td></tr> <tr> <td>CHECK PALLET IN</td><td>This bit is used on horizontal mills only.</td></tr> <tr> <td>CHECK HIDN VAR</td><td>This bit is used on horizontal mills only.</td></tr> </table>	INVERT G.B.	This bit allows an alternate gearbox configuration. It inverts the sense of the gearbox inputs. Used for 50 taper option.	DPR SERIAL	Causes the main serial inputs/outputs to go through the floppy video board.	CHECK PALLET IN	This bit is used on horizontal mills only.	CHECK HIDN VAR	This bit is used on horizontal mills only.				
INVERT G.B.	This bit allows an alternate gearbox configuration. It inverts the sense of the gearbox inputs. Used for 50 taper option.												
DPR SERIAL	Causes the main serial inputs/outputs to go through the floppy video board.												
CHECK PALLET IN	This bit is used on horizontal mills only.												
CHECK HIDN VAR	This bit is used on horizontal mills only.												



DISPLAY ACTUAL	When set to 1, displays the actual spindle speed on the Current Commands display page.
TSC PRG ENBL	Enables purge output on TSC option.
RESERVED	Reserved for later use.
SPND DRV LCK	This bit must be set to 1 if machine is equipped with a non-Haas vector spindle drive. This bit must be set to 1 if the machine has a 50 taper spindle or a non-Haas vector drive.
RESERVED	Reserved for later use
CNCR SPINDLE	(Concurrent Spindle) When set to 1, the spindle will be commanded to start concurrently with other commands in the same block. In the following example, with this bit set to 1, the spindle will start at the same time as the rapid move: G0 X-1. S7500 M3;
RESERVED	Reserved for later use.
HAAS VECT DR	(Haas Vector Drive) This bit must be set to 1 if machine is equipped with a HAAS vector spindle drive. When set to 1, voltage to the Haas vector drive is displayed in the diagnostics display as DC BUSS.
UP ENCL TEMP	(Microprocessor Enclosure Temperature) When set to 1, the enclosure temperature will be displayed on INPUTS2 screen of the diagnostics display.
HAAS RJH	(Haas Remote Jog Handle) This bit must be set to 1 if the machine is equipped with a Haas 5-Axis Remote Jog Handle.
SPIN TEMP NC	(Spindle Temperature Sensor Normally Closed) This bit specifies the type (normally open or normally closed) of the spindle temperature sensor. This bit should be set to 1 for machines with a Haas vector drive, and 0 for machines that do not have a vector drive.
AIR DRV SHTL	This bit will cause the machine to use discrete outputs 21 and 26 to command the shuttle to move in and out. On mills with the Air Driven Shuttle it must be set to 1. On all other mills it must be set to 0.
GIMBAL SPNDL	Used on VR-Series mills. This bit will cause the machine to check that the Z,A and B axes are at zero before a tool change is started. If one is not, alarm 150 will be generated. On mills with the gimbaled Spindle it must be set to 1. On all other mills it must be set to 0.
NO MFIN CKPU	When this bit is set, it will prevent checking of MFIN at power-up. It should be set for 1 for all machines that have the new Haas Automatic Pallet Changer attached, and 0 for all other machines.
D:Y SW ENABLE	(Delta Wye switch enabled). This bit is used for the Vector Drive. The bit enables the switching of spindle motor windings, provided the hardware ENABLE is installed, and the proper parameters are set. If this switch is set, but bit 19 is not, then the winding switching will only be done when the spindle is at rest, depending on the target speed of the spindle.



D:Y SW ON FLY	This bit enables switching on the fly, as the spindle motor is accelerating or decelerating through the switch point. If bit 18 is not set, this switch will be ignored.																																						
5 AX TOFS -X	This bit is used with the G143 (modal 5 axes tool length compensation) on machines with a Gimbaled Spindle. If it is set to 1, this means that when the corresponding rotary axes is moved, the sign of the X Position must be inverted. Normally, this bit should be set to 0.																																						
5 AX TOFS -Y	This bit is used with the G143 (modal 5 axes tool length compensation) on machines with a Gimbaled Spindle. If it is set to 1, this means that when the corresponding rotary axes is moved, the sign of the Y Position must be inverted. Normally, this bit should be set to 0.																																						
5 AX A MOV B	This bit is used with the G143 (modal 5 axes tool length compensation) on machines with a Gimbaled Spindle. The B axes normally moves the A axes, but if this is not true, this bit can be set to change which is the inner axes. Normally, this bit should be set to 0.																																						
S MNT BIT 1	Bits 26, 27, and 28 work together to specify the type of sidemount tool changer that is installed on a vertical mill. The following table shows the bit combinations that must be used: <table> <tr> <th>Bit</th><th>28</th><th>27</th><th>26</th></tr> <tr> <td>0</td><td>0</td><td>0</td><td>No side-mount tool changer installed</td></tr> <tr> <td>0</td><td>0</td><td>1</td><td>Serpentine 1</td></tr> <tr> <td>0</td><td>1</td><td>0</td><td>Serpentine 2</td></tr> <tr> <td>0</td><td>1</td><td>1</td><td>Serpentine 3</td></tr> <tr> <td>1</td><td>0</td><td>0</td><td>Disk 1</td></tr> <tr> <td>1</td><td>0</td><td>1</td><td>Disk 2</td></tr> <tr> <td>1</td><td>1</td><td>0</td><td>Disk 3</td></tr> <tr> <td>1</td><td>1</td><td>1</td><td>Disk 4</td></tr> </table>			Bit	28	27	26	0	0	0	No side-mount tool changer installed	0	0	1	Serpentine 1	0	1	0	Serpentine 2	0	1	1	Serpentine 3	1	0	0	Disk 1	1	0	1	Disk 2	1	1	0	Disk 3	1	1	1	Disk 4
Bit	28	27	26																																				
0	0	0	No side-mount tool changer installed																																				
0	0	1	Serpentine 1																																				
0	1	0	Serpentine 2																																				
0	1	1	Serpentine 3																																				
1	0	0	Disk 1																																				
1	0	1	Disk 2																																				
1	1	0	Disk 3																																				
1	1	1	Disk 4																																				
S MNT BIT 2	Bits 26, 27, and 28 work together to specify the type of sidemount tool changer that is installed on a vertical mill.																																						
S MNT BIT 3	Bits 26, 27, and 28 work together to specify the type of sidemount tool changer that is installed on a vertical mill.]																																						
TL DR SWITCH	This bit specifies the too carousel door configuration. If it is set to 0, this indicates the configuration where the door id driven open by a timed operation. If it is set to 1, this indicates the door is spring loaded closed and is driven open by a timmed operation against the door open switch.																																						
HS2 SDMTCRSL	This parameter bit is for the HS-2 sidemount tool changer. It must be set to 1 on all HS-2 mills, and 0 on all other mills.																																						



HS3 SDMTCRSL	This parameter bit is for the HS-3 sidemount tool changer. It must be set to 1 on all HS-3 mills, and 0 on all other mills.
SAFETY INVERT	This bit supports the CE door interlock that locks when power is turned off. For machines that have the regular door lock that locks when power is applied, this bit must be set to 0. For machines that have the inverted door lock, this bit must be set to 1.
INV SPD DCEL	Inverese Spindle Speed Deceleration. When this parameter is set to 1, the spindle decelerates faster at lower speeds, resulting in a shorter deceleration time.
SWAP A & C	This parameter causes the A and C axes to be swapped internally. This parameter bit should be set to 1 for the bridge mill. All other mills should set this bit to 0.
INV SPD DCEL	Inverse Spindle Speed Deceleration When this parameter is set to 1, the spindle decelerates faster at lower speeds, resulting in shorter deceleration time.

Parameter	279 X LSC GAIN MULTIPLIER This parameter is used on machines with linear scales. Linear scales are used to continuously correct any errors in the encoder position. The parameter determines the gain of the correction factor, that is, how fast it corrects. This parameter should be set to 40.
Parameter	280 Y MAX 3rd DERIV See parameter 279 for description
Parameter	281 Z MAX 3rd DERIV See parameter 279 for description
Parameter	282 A MAX 3rd DERIV See parameter 279 for description
Parameter	283 B MAX 3rd DERIV See parameter 279 for description
Parameter	284 RESERVED
Parameter	285 X LSC OFFSET This parameter is used on machines with linear scales. This parameters account for the unused portion of the lead screw between zero and the actual motor. This parameters should be a positive value (400000) unless the NEG COMP DIR bit for the axis is set, in which case this parameter should be a negative value (-400000.)
Parameter	286 Y LSC OFFSET See parameter 285 for description.
Parameter	287 Z LSC OFFSET See parameter 285 for description.



- Parameter 288 A LSC OFFSET
See parameter 285 for description.
- Parameter 289 B LSC OFFSET
See parameter 285 for description.
- Parameter 294 MIN BUSS VOLTAGE
This parameter specifies the minimum Haas Vector Drive buss voltage. When a Haas Vector Drive is installed, it should be set to 200 (the units are volts). If there is not a Vector Drive installed, the parameter should be set to zero. Alarm 160 will be generated if the voltage falls below this value.
- Parameter 295 SHTL SETTLE TIME
Used on the VR series mills. This is for the air driven shuttle. This parameter allows settling time for the shuttle after it has moved toward the spindle and before a tool change is performed. It should be set to approximately half a second (500) on all mills with the Air Driven Shuttle. This may vary. All other mills can be set to 0 as they are unaffected by it.
- Parameter 296 MAX OVER VOLT TIME
Specifies the amount of time (in 50ths of a second) that an overvoltage condition (alarm 119 OVER VOLTAGE) will be tolerated before the automatic shut down process is started.
- Parameter 297 MAX OVERHEAT TIME
Specifies the amount of time (in 50ths of a second) that an overheat condition (alarm 122 REGEN OVERHEAT) will be tolerated before the automatic shut down process is started.
- Parameter 298 MAX FEED (DEG/MIN)
Used on VR-Series mills. This parameter specifies the maximum rotary feed rate in degrees per minute. Any attempt at cutting faster than this will result in "LIM" being displayed next to the FEED message on the Program Command Check screen.

On mills with a Gimbaled Spindle, this parameter must be set to 200. For all other mills, this bit should be set to 99999.
- Parameter 299 AUTOFEED-STEP-UP
This parameter works with the AUTOFEED feature. It specifies the feed rate step-up percentage per second and should initially be set to 10.
- Parameter 300 AUTOFEED STEP-DOWN
This parameter works with the AUTOFEED feature. It specifies the feed rate step-down percentage per second and should initially be set to 20.
- Parameter 301 AUTOFEED-MIN-LIMIT
This parameter works with the AUTOFEED feature. It specifies the minimum allowable feed rate override percentage that the AUTOFEED feature can use and should initially be set to 1.
- Parameter 302 FEED ACCELERATION
This parameter supports the motion control feature. It is the base 2 exponent of the feed time constant in milliseconds. It should be set to 3.



- Parameter 305 SEVO PO BRK DLY
The SRV PO (Servo Power On) discrete output is used to engage and disengage an axis brake. This parameter is used to specify a time in milliseconds that the control should wait after activating the SRV PO output and turning off power to the servo motors via the MOCON. This parameter also specifies the time to wait after deactivating the SRV PO output and reactivating the servo motors via the MOCON.
- Parameter 306 POCKET UP / DN DELAY
This parameter supports the Pasma serpentine type and the Disk type 1 tool changers. It specifies the time allowed (in milliseconds) for the tool pocket to be raised or lowered. If the pocket does not move to its commanded position within the time allowed by this parameter and by parameter 62, alarm 626 TOOL POCKET SLIDE ERROR is generated. For mills without a side mount tool changer, this parameter should be set to 0.
- Parameter 307 POCK UN / LOCK DELAY
This parameter supports the Pasma serpentine type and the Disk type 1 tool changers. It specifies the time allowed (in milliseconds) to lock or unlock a tool pocket. For mills without a side mount tool changer, this parameter should be set to 0.
- Parameter 308 ARM ROTATE TIME
This parameter supports the Pasma serpentine type and the Disk type 1 tool changers. It specifies the time allowed (in milliseconds) for the arm to rotate to the next position. The positions are, Clamp, Unclamp, and Origin. If the arm does not move to the commanded position within the allowed time, alarm 622 TOOLARM FAULT is generated. For mills without a side mount tool changer, this parameter should be set to 0.
- Parameter 309 MOTOR COAST TIME
This parameter supports the Pasma serpentine type and the Disk type 1 tool changers. It specifies the time allowed (in milliseconds) for the tool arm to start and stop. If the arm has not started or stopped moving after the allowed time, alarm 627 ATC ARM POSITION TIMEOUT is generated. For mills without a side mount tool changer, this parameter should be set to 0.
- Parameter 310 CAM LOCK DELAY
This parameter supports the Pasma serpentine type and the Disk type 1 tool changers. It specifies the time allowed (in milliseconds) to lock the cam by pushing the shot pin in, or to unlock the cam by pulling the shot pin out. If the shot pin has not moved to its commanded position within the allowed time, alarm 625 INVALID TC START CONDITION is generated.
- Parameter 311 ARM BUMP TIME
This parameter supports the Pasma serpentine type and the Disk type 1 tool changers. During tool change recovery, the arm may be moved a small amount by pressing the ATC FWD or ATC REV key. Each press of the key will cause the arm motors to run for the amount of time (in milliseconds) specified by this parameter. For mills without a side mount tool changer, this parameter should be set to 0.
- Parameter 312 CAROUSEL BUMP TIME
This parameter supports the Pasma serpentine type and the Disk type 1 tool changers. During tool change recovery, the carousel may be moved a small amount by pressing the Left Arrow or Right Arrow key. Each press of the key will cause the carousel motors to run for the amount of time (in milliseconds) specified by this parameter. For mills without a side mount tool changer, this parameter should be set to 0.



- Parameter 313 POCKET INCREMENT
This is a parameter for the bridge mill. Under normal circumstances it should be set to 1. If it is set to 2, for example, the control will only recognize every other pocket. That is, it will treat the tools and pockets as follows:
 Tool 1 is in pocket 1
 Tool 2 is in pocket 3
 Tool 3 is in pocket 5
 Tool 4 is in pocket 7
 etc...
If this parameter is set to 3 the control will only recognize every third pocket and so on. It is the operator's responsibility to ensure that the total number of pockets in the tool changer is evenly divisible by this parameter value. If not, the control will pick the wrong pocket after the carousel has exceeded a full revolution.
- Parameter 314 FEED DELTA V
This parameter supports the motion control feature. It is the maximum change in velocity in encoder steps per millisecond. It should be set to 10.
- Parameter 315 ALIS M GRPHC
When this bit is set to 0, all user defined M codes (such as M50 normally used to do a pallet change on a horizontal mill) will be ignored when a program is run in graphics mode. If it is necessary to have graphics recognize such M codes, this bit should be set to 1.
- Parameter 327 X LS PER INCH
This parameter is used on machines equipped with linear scales. This parameter should be set to 25,400 on mills fitted with linear scales. On all other mills, they should be set to zero.
- Parameter 328 Y LS PER INCH
This parameter is used on machines equipped with linear scales. This parameter should be set to 25,400 on mills fitted with linear scales. On all other mills, they should be set to zero.
- Parameter 329 Z LS PER INCH
This parameter is used on machines equipped with linear scales. This parameter should be set to 25,400 on mills fitted with linear scales. On all other mills, they should be set to zero.
- Parameter 330 A LS PER INCH
This parameter is used on machines equipped with linear scales. This parameter should be set to 0 on mills with or without linear scales.
- Parameter 331 B LS PER INCH
This parameter is used on machines equipped with linear scales. This parameter should be set to 0 on mills with or without linear scales.
- Parameter 333 X LS PER REV
This parameter is used on machines equipped with linear scales. This parameter should be set to 50,000 on mills fitted with linear scales. On all other mills, they should be set to zero.
- Parameter 334 Y LS PER REV
This parameter is used on machines equipped with linear scales. This parameter should be set to 50,000 on mills fitted with linear scales. On all other mills, they should be set to zero.



Parameter	335 Z LS PER REV This parameter is used on machines equipped with linear scales. This parameter should be set to 50,000 on mills fitted with linear scales. On all other mills, they should be set to zero.
Parameter	336 A LS PER REV This parameter is used on machines equipped with linear scales. This parameter should be set to 0 on mills with or without linear scales.
Parameter	337 B LS PER REV This parameter is used on machines equipped with linear scales. This parameter should be set to 0 on mills with or without linear scales.
Parameter	339 X SPINDLE THERM COEF. This parameter supports the Spindle Head Thermal Compensation feature, and should be set to 0.
Parameter	340 Y SPINDLE THERM COEF. See parameter 339 for description.
Parameter	341 Z SPINDLE THERM COEF. See parameter 339 for description.
Parameter	342 A SPINDLE THERM COEF. See parameter 339 for description.
Parameter	343 B SPINDLE THERM COEF. See parameter 339 for description.
Parameter	345 X SPINDLE THERM T.C. This parameter supports the Spindle Head Thermal Compensation feature, and should be set to 0.
Parameter	346 Y SPINDLE THERM T.C. See parameter 345 for description.
Parameter	347 Z SPINDLE THERM T.C. See parameter 345 for description.
Parameter	348 A SPINDLE THERM T.C. See parameter 345 for description.
Parameter	349 B SPINDLE THERM T.C. See parameter 345 for description.



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.

ELECTRONIC THERMAL COMPENSATION

When ballscrews rotate they generate heat. Heat causes the ballscrews to expand. In constant duty cycles as in mold making the resultant ball screw growth can lead to cutting errors on the next morning start up. Haas' new ETC algorithm can accurately model this heating and cooling effect and electronically expand and contract the screw to give near glass scale accuracy and consistency.

This compensation is based on a model of the lead screw which calculates heating based on the distance traveled and the torque applied to the motor. This compensation does not correct for thermal growth due to changes in ambient temperature or due to part expansion.

Electronic thermal compensation works by estimating the heating of the screw based on the total amount of travel over its length and including the amount of torque applied to the screw. This heat is then turned into a thermal coefficient of expansion and the position of the axis is multiplied by the coefficient to get a correction amount.

If the machine is turned off when there is some compensation applied (due to motion and heating of screw), when the machine is turned back on, the compensation will be adjusted by the clock indicated elapsed time.

SPINDLE HEAD THERMAL COMPENSATION

This feature integrates spindle speed over time and builds a model of thermal growth. As the model shows the spindle head warming up, the control adjusts the Z axes to compensate for thermal growth.



THIS PAGE INTENTIONALLY LEFT BLANK



10. ALARMS

Any time an alarm is present, the lower right hand corner of the screen 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.

The **ALARMS DISPLAY** can be selected at any time by pressing the ALARM MESSAGES button. When there are no alarms, the display will show NO ALARM. If there are any alarms, they will be listed with the most recent alarm at the bottom of the list. The CURSOR and PAGE UP and PAGE DOWN buttons can be used to move through a large number of alarms. The CURSOR **right** and **left** buttons can be used to turn on and off the ALARM history display.

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.

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:

101	Comm. Failure with MOCON	During a self-test of communications between the MOCON and main processor, the main processor does not respond, one of them is possibly bad. Check cable connections and boards.
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 alarm 103.
105	Z Servo Error Too Large	Same as alarm 103.
106	A Servo Error Too Large	Same as alarm 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. This alarm will also be generated if there is a low pressure condition in the hydraulic counterbalance system. In this case, the alarm will not reset until the condition has been corrected.



- | | | |
|-----|---------------------------|--|
| 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 alarm 108. |
| 110 | Z Servo Overload | Same as alarm 108. |
| 111 | A Servo Overload | Same as alarm 108. |
| 112 | No Interrupt | Electronics fault. Call your dealer. |
| 113 | Shuttle In Fault | Tool changer is 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. Check relays K9-K12, and fuse F1 on IOPCB. |
| 114 | Shuttle Out Fault | Tool changer not completely to left. During a tool change operation the tool in/out shuttle failed to get to the OUT position. Parameters 62 and 63 can adjust the time-out times. This alarm can be caused by anything that jams the motion of the slide or by the presence of a tool in the pocket facing the spindle. A loss of power to the tool changer can also cause this. Check relays K9-K12, and fuse F1 on IOPCB. |
| 115 | Turret Rotate Fault | Tool carousel motor not in position. During a tool changer operation the tool turret failed to start moving, failed to stop 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. Check relays K9-K12, and fuse F1 on IOPCB. |
| 116 | Spindle Orientation Fault | Spindle did not orient correctly. This is either a vector drive problem or a mechanical problem on machines without a vector drive. During a spindle orientation function, the spindle is rotated until the lock pin drops in; but the lock pin never dropped. 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, circuit breaker CB4, the circuit breaker for the air pressure solenoids, 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 low 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. The servos will be turned off and the spindle, tool changer, and coolant pump will stop. If this condition remains for 4.5 minutes, an automatic shutdown will begin. |
| 120 | Low Air Pressure | Air pressure dropped below 80 PSI for a period defined by Parameter 76. The LOW AIR PR alarm will appear on the screen as soon as the pressure gets low, and this alarm appears after some time has elapsed. Check your incoming air pressure for at least 100 PSI and ensure that the regulator is set at 85 PSI. |
| 121 | Low Lube 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 on the side of the control cabinet. Check that the lube lines are not blocked. |
| 122 | Regen Overheat | The regenerative load temperature is above a safe limit. This alarm will 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 high for 4.5 minutes, an automatic shutdown will begin. It can also be caused by a high start/stop duty cycle of spindle. |
| 123 | Spindle Drive Fault | Failure of spindle drive, motor or regen load. This can be caused by a shorted motor, overvoltage, overcurrent, undervoltage, failure of drive, or shorted or open regen load. Undervoltage and overvoltage of DC bus are also reported as alarms 160 and 119, respectively. |
| 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. |



- | | | |
|-----|--------------------|---|
| 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 | The tool appeared to be unclamped during spindle orientation, a gear change, a speed change, or TSC start-up. The alarm will also be generated if the tool release piston is energized during Power Up. This can be caused by a fault in the air solenoids, relays on the I/O assembly, the drawbar assembly, or in the wiring. |
| 131 | Tool Not Clamped | When clamping or powering up the machine, the Tool Release Piston is not HOME. This is a possible fault in the air solenoids, relays on the IO Assembly, the drawbar assembly, or wiring. |
| 132 | Power Down Failure | Machine did not turn off when an automatic power-down was commanded. Check wiring to POWIF card on power supply assembly, relays on the IO assembly, and the main contactor K1. |
| 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 K16, the wiring to the sense switch, and the switch. |
| 134 | Tool Clamp Fault | When UNCLAMPING, the tool did not release from spindle when commanded. Check air pressure and solenoid circuit breaker CB4. Can also be caused by misadjustment of drawbar assembly. |
| 135 | X Motor Over Heat | Servo motor overheat. The temperature sensor in the motor indicates over 150 degrees F. This can be caused by an extended overload of the motor such as leaving the slide at the stops for several minutes. |
| 136 | Y Motor Over Heat | Same as alarm 135. |
| 137 | Z Motor Over Heat | Same as alarm 135. |
| 138 | A Motor Over Heat | Same as alarm 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 encoder connectors. |
| 140 | Y Motor Z Fault | Same as alarm 139. |
| 141 | Z Motor Z Fault | Same as alarm 139. |
| 142 | A Motor Z Fault | Same as alarm 139. |
| 143 | Spindle Not Locked | Vector drive orientation lost or 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 alarm 145 |
| 147 | Z Limit Switch | Same as alarm 145 |
| 148 | A Limit Switch | Normally disabled for rotary axis. |
| 149 | Spindle Turning | Spindle not at zero speed for tool change. A signal from 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 | Changer not at home and either the Z or A or B axis (or any combination) is not at zero. If RESET, E-STOP, or POWER OFF occurs during tool change, Z-axis motion and tool changer motion may not be safe. Check the position of the tool changer and remove the tool if possible. Re-initialize with the AUTO ALL AXES button but be sure that the pocket facing the spindle afterwards does not contain a tool. |
| 151 | Low Thru Spindle Coolant | For machines with Through the Spindle Coolant only. This alarm will shut off the spindle, feed, and pump all at once. It will turn on purge, wait for the amount of time specified in parameter 237 for the coolant to purge, and then turn off the purge. Check for low coolant tank level, any filter or intake strainer clogging, or for any kinked or clogged coolant lines. If no problems are found with any of these, and none of the coolant lines are clogged or kinked, call your dealer. |
| 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 encoder connectors. |
| 154 | Y-axis Z Ch Missing | Same as alarm 153. |
| 155 | Z-axis Z Ch Missing | Same as alarm 153. |
| 156 | A-axis Z Ch Missing | Same as alarm 153. |
| 157 | MOCON Watchdog Fault | The self-test of the MOCON has failed. Replace the MOCON. |



- | | | |
|-----|-----------------------------|---|
| 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 more than 10% below nominal. |
| 161 | X-Axis Drive Fault | Current in X servo motor beyond limit. Possibly caused by a stalled or overloaded motor. The servos are turned off. This can be caused by running a short distance into a mechanical stop. It can also be caused by a short in the motor or a short of one motor lead to ground. |
| 162 | Y-Axis Drive Fault | Same as alarm 161. |
| 163 | Z-Axis Drive Fault | Same as alarm 161. |
| 164 | A-Axis Drive Fault | Same as alarm 161. |
| 165 | X Zero Ret Margin Too Small | <p>This alarm will occur if the home/limit switches move or are misadjusted.</p> <p>This alarm indicates that the zero return position may not be consistent from one zero return to the next. The encoder Z channel signal must occur between 1/8 and 7/8 revolution of where the home switch releases. This will not turn the servos off but will stop the zero return operation.</p> |
| 166 | Y Zero Ret Margin Too Small | Same as alarm 165. |
| 167 | Z Zero Ret Margin Too Small | Same as alarm 165. |
| 168 | A Zero Ret Margin Too Small | This alarm will occur if the home/limit switches move or are misadjusted. This alarm indicates that the zero return position may not be consistent from one zero return to the next. The encoder Z channel signal must occur between 1/8 and 7/8 revolution of where the home switch releases. This will not turn the servos off but will stop the zero return operation. |
| 169 | Spindle Direction Fault | Problem with rigid tapping hardware. The spindle started turning in the wrong direction. |
| 170 | Phase Loss | Problem with incoming line voltage. This usually indicates that there was a transient loss of input power to the machine. |
| 173 | Spindle Ref Signal Missing | The Z channel pulse from the spindle encoder is missing for hard tapping synchronization. |



174	Tool Load Exceeded	The tool load monitor option is selected and the maximum load for a tool was exceeded in a feed. This alarm can only occur if the tool load monitor function is installed in your machine.
175	Ground Fault Detected	A ground fault condition was detected in the 115V AC supply. This can be caused by a short to ground in any of the servo motors, the tool change motors, the fans, or the oil pump.
176	Over Heat Shutdown	An overheat condition persisted for 4.5 minutes and caused an automatic shutdown.
177	Over Voltage Shutdown	An overvoltage condition persisted for 4.5 minutes and caused an automatic shutdown.
178	Divide by Zero	Software Error; Call your dealer.
179	Low Pressure Transmission Oil	Spindle coolant oil is low or low pressure condition in lines.
180	Pallet Not Clamped	The APC pallet change was not completed for some reason (pressing E-stop, reset, or feedhold), and an attempt was made to run the spindle. Run M50 pallet change to reset the machine.
182	X Cable Fault	Cable from X-axis encoder does not have valid differential signals.
183	Y Cable Fault	Same as alarm 182.
184	Z Cable Fault	Same as alarm 182.
185	A Cable Fault	Same as alarm 182.
186	Spindle Not Turning	Status from spindle drive indicates it is not at speed when expected.
187	B Servo Error Too Large	Same as alarm 103.
188	B Servo Overload	Same as alarm 108.
189	B Motor Overheat	Same as alarm 135.
190	B 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 encoder connectors.
191	B Limit Switch	Same as alarm 148.
192	B Axis Z Ch Missing	Broken wires or encoder contamination. All servos are turned off. This Ch Missing can also be caused by loose encoder connectors.



- | | | |
|-----|-----------------------------|---|
| 193 | B Axis Drive Fault | Same as alarm 161. |
| 194 | B Zero Ret Margin Too Small | This alarm will occur if the home/limit switches move or are misadjusted. This alarm indicates that the zero return position may not be consistent from one zero return to the next. The encoder Z channel signal must occur between 1/8 and 7/8 revolution of where the home switch releases. This will not turn the servos off but will stop the zero return operation. |
| 195 | B Cable Fault | Same as alarm 182. |
| 196 | Coolant Spigot Failure | Vertical mills only. Spigot failed to achieve commanded location after two (2) attempts. |
| 197 | 100 Hours Unpaid Bill | Call your dealer. |
| 198 | Precharge Failure | During TSC operation, the precharge failed for greater than 0.1 seconds. It will shut off the feed, spindle and pump all at once. If received, check all air lines and the air supply pressure. |
| 199 | Negative RPM | A negative spindle RPM was sensed. |
| 201 | Parameter CRC Error | Parameters lost maybe by low battery. Check for a low battery and low battery alarm. |
| 202 | Setting CRC Error | Settings lost maybe by low battery. Check for a low battery and low battery alarm. |
| 203 | Lead Screw CRC Error | Lead screw compensation tables lost maybe by low battery. Check for 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 | Possible corrupted program. Save all programs to floppy disk, delete all, then reload. Check for a low battery and low battery alarm. |
| 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	Possible corrupted program. Save all programs to floppy disk, delete all, then reload.
212	Program Integrity Error	Possible corrupted program. Save all programs to floppy disk, delete all, then reload. Check for a low battery and low battery alarm.
213	Program RAM CRC Error	Electronics fault; possibly with main processor.
214	No. of Programs Changed	Indicates that the number of programs disagrees with the internal variable that keeps count of the loaded programs. Possible processor board problem.
215	Free Memory PTR Changed	Indicates the amount of memory used by the programs counted in the changed system disagrees with the variable that points to free memory. Possible processor board problem.
216	EPROM Speed Failure	Possible processor board problem.
217	X Axis Phasing Error	Error occurred in phasing initialization of brushless motor. This can be caused by a bad encoder, or a cabling error.
218	Y Axis Phasing Error	Same as alarm 217.
219	Z Axis Phasing Error	Same as alarm 217.
220	A Axis Phasing Error	Same as alarm 217.
221	B Axis Phasing Error	Same as alarm 217.
222	C Axis Phasing Error	Same as alarm 217.
223	Door Lock Failure	In machines equipped with safety interlocks, this alarm occurs when the control senses the door is open but it is locked. Check the door lock circuit.
224	X Transition Fault	Illegal transition of count pulses in X axis. This alarm usually indicates that the encoder has been damaged and encoder position data is unreliable. This can also be caused by loose connectors at the MOCON or MOTIF PCB.
225	Y Transition Fault	Same as alarm 224.
226	Z Transition Fault	Same as alarm 224.
227	A Transition Fault	Same as alarm 224.
228	B Transition Fault	Same as alarm 224.
229	C Transition Fault	Same as alarm 224.



231	Jog Handle Transition Fault	Same as alarm 224.
232	Spindle Transition Fault	Same as alarm 224.
233	Jog Handle Cable Fault	Cable from jog handle encoder does not have valid differential signals.
234	Spindle Enc. Cable Fault	Cable from spindle encoder does not have valid differential signals.
235	Spindle Z Fault	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 encoder connectors.
236	Spindle Motor Overload	This alarm is generated in machines equipped with a Haas vector drive, if the spindle motor becomes overloaded.
237	Spindle Following Error	The error between the commanded spindle speed and the actual speed has exceeded the maximum allowable (as set in Parameter 184).
240	Empty Prog or No EOB	DNC program not found, or no end of program found.
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. See MESSAGE PAGE for input line.
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	Possible corrupted program. Save all programs to floppy disk, delete all, then reload.
250	Program Data Error	Same as alarm 249.
251	Prog Data Struct Error	Same as alarm 249.
252	Memory Overflow	Same as alarm 249.



253	Electronics Overheat	The control box temperature has exceeded 135 degrees F. This can be caused by an electronics problem, high room temperature, or clogged air filter.
254	Spindle Overheat	The motor driving spindle is too hot. The spindle motor temperature sensor sensed a high temperature for greater than 1.5 seconds. This alarm is only generated in machines equipped with a Haas vector drive.
257	Program Data Error	Possible corrupted program. Save all programs to floppy disk, delete all, then reload. Possible processor board problem.
258	Invalid DPRNT Format	Macro DPRNT statement not structured properly.
259	Language Version	Possible processor board problem.
260	Language CRC	Indicates FLASH memory has been corrupted or damaged. Possible processor board problem.
261	Rotary CRC Error	Rotary table saved parameters (used by Settings 30, 78) have a CRC error. Indicates a loss of memory - possible processor board problem.
262	Parameter CRC Missing	RS-232 or floppy read of parameter had no CRC when loading from floppy or RS-232.
263	Lead Screw CRC Missing	Lead screw compensation tables have no CRC when loading from floppy or RS-232.
264	Rotary CRC Missing	Rotary table parameters have no CRC when loading from floppy or RS-232
265	Macro Variable File CRC Error	Macro variable file has a CRC error. Indicates a loss of memory. Possible processor board problem.
267	Tool Door Out of Position	Horizontal mills only. Alarm will be generated during a tool change when parameter 278 TL DR SWITCH is set to 1, and the tool carousel air door and the tool carousel air door switch indicates that the door is open after commanded to be closed, or closed after it was commanded to be open. This alarm will most likely be caused by a stuck or broken switch.
268	Door open @ M95 Start	Generated whenever an M95 (Sleep Mode) is encountered and the door is open. The door must be closed in order to start sleep mode
270	C Servo Error Too Large	Same as alarm 103.
271	C Servo Overload	Same as alarm 108.
272	C Motor Overheat	Same as alarm 135.



273	C 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 encoder connectors.
274	C Limit Switch	Same as alarm 145.
275	C Axis Z Ch Missing	Broken wires or encoder contamination. All servos are turned off. This Ch Missing can also be caused by loose encoder connectors.
276	C Axis Drive Fault	Same as alarm 161.
277	C Zero Ret Margin Too Small	Same as alarm 165.
278	C Cable Fault	Same as alarm 182.
279	X Axis Linear Scale Z Fault	Encoder marker pulse count failure. This alarm usually indicates that the Z Fault encoder has been damaged and encoder position data is unreliable. This can also be caused by loose scale connectors.
280	Y Axis Linear Scale Z Fault	Encoder marker pulse count failure. This alarm usually indicates that the Z Fault encoder has been damaged and encoder position data is unreliable. This can also be caused by loose scale connectors.
281	Z Axis Linear Scale Z Fault	Encoder marker pulse count failure. This alarm usually indicates that the Z Fault encoder has been damaged and encoder position data is unreliable. This can also be caused by loose scale connectors.
282	A Axis Linear Scale Z Fault	Encoder marker pulse count failure. This alarm usually indicates that the Z Fault encoder has been damaged and encoder position data is unreliable. This can also be caused by loose encoder connectors.
283	X Axis Linear Scale Z CH Missing	Broken wires or encoder contamination. All servos are turned off. This Z Channel Missing can also be caused by loose scale connectors.
284	Y Axis Linear Scale Z CH Missing	Broken wires or encoder contamination. All servos are turned off. This Z Channel Missing can also be caused by loose encoder connectors.
285	Z Axis Linear Scale Z CH Missing	Broken wires or encoder contamination. All servos are turned off. This Z Channel Missing can also be caused by loose encoder connectors.
286	A Axis Linear Scale Z CH Missing	Broken wires or encoder contamination. All servos are turned off. This Z Channel Missing can also be caused by loose encoder connectors.
287	X Axis Linear Scale Cable Fault	Cable from X-axis scale does not have valid differential signals.



288	Y Axis Linear Scale Cable Fault	Cable from Y-axis scale does not have valid differential signals.
289	Z Axis Linear Scale Cable Fault	Cable from Z-axis scale does not have valid differential signals.
290	A Axis Linear Scale Cable Fault	Cable from A-axis scale does not have valid differential signals.
291	Low Air Volume/Pressure During ATC	An automatic tool change was not completed due to insufficient volume or pressure of compressed air. Check air supply line.
302	Invalid R In G02 or G03	Check your geometry. 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.
304	Invalid I, J, or K In G02 or G03	Check your geometry. 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.
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	Program contained a line of code that is not understood.
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	In M97, M98 or G65 a subprogram number must be put in the P code. G47 must have P0 for text engraving or P1 for sequential serial numbers.
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. G47 must have P0 for text engraving or P1 for sequential serial numbers.
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 alarm 316.
318	Z Over Travel Range	Same as alarm 316.



- | | | |
|-----|------------------------------------|--|
| 319 | A Over Travel Range | A-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. |
| 320 | No Feed Rate Specified | Must have a valid F code for interpolation functions. |
| 321 | Auto Off Alarm | 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 greater than or equal to 1000 seconds (over 999999 milliseconds). |
| 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 the value in Parameter 65. |
| 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 alarm 333. |
| 335 | Z-Axis Disabled | Same as alarm 333. |
| 336 | A-Axis Disabled | An attempt was made to program the A-axis while it was disabled (DISABLED bit in Parameter 43 set to 1). |
| 337 | GOTO or P line Not Found | Subprogram is not in memory, or P code is incorrect. P not found |
| 338 | Invalid IJK and XYZ in G02 or G03 | There is a problem with circle definition; check your geometry. |
| 339 | Multiple Codes | Only one M, X, Y, Z, A, Q etc. allowed in any block, only one G codes in the same group. |



340	Cutter Comp Begin With G02 or G03	Select cutter compensation earlier. Cutter comp. must begin on a linear move.
341	Cutter Comp End With G02 or G03	Disable cutter comp later.
342	Cutter Comp Path Too Small	Geometry not possible. Check your geometry.
343	Display Queue Record Full	Software error. Call your dealer.
344	Cutter Comp With G18 and G19	Cutter comp only allowed in XY plane (G17).
348	Motion Not Allowed In G93 Mode	This alarm is generated if the mill is in Inverse Time Feed mode, and a G12, G13, G70, G71, G72, G150, or any Group 9 motion command is issued.
349	Prog Stop W/O Cancel Cutter Comp	An X/Y cutter compensation exit move is required before a program stop.
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	Invalid P Code	In a block with G103 (Block Lookahead Limit), a value between 0 and 15 must be used for the P code.
352	Aux Axis Power Off	Aux 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	Aux axes not responding. Check auxiliary axes and RS-232 connections.
355	Aux Axis Position	Mismatch between machine and aux axes position. Check aux axes and Mismatch 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.
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 RESET.	Tool life limit was reached. To continue, reset the usage count in the Current Commands display and press
363	Coolant Locked Off	Override is off and program tried to turn on coolant.
364	No Circ Interp Aux Axis	Only rapid or feed is allowed with aux axes.
367	Cutter Comp Interference	G01 cannot be done with tool size.
368	Groove Too Small	Tool too big to enter cut.
369	Tool Too Big	Use a smaller tool for cut.
370	Pocket Definition Error	Check geometry for G150.
371	Invalid I, J, K, OR Q	Check G150.
372	Tool Change In Canned Cycle	Tool change not allowed while canned cycle is active.
373	Invalid Code in DNC	A code found in a DNC program could not be interpreted because of restrictions to DNC.
374	Missing XYZA in G31 or G36	G31 skip function requires an X , Y , Z , or A move.
375	Missing Z or H in G37	G37 automatic tool length measurement function requires H code, Z value, and tool offset enabled. X , Y , and A values not allowed.
376	No Cutter Comp In Skip	SkipG31 and G37 functions cannot be used with cutter compensation.
377	No Skip in Graph/Sim	Graphics mode cannot simulate skip function.
378	Skip Signal Found	Skip signal check code was included but skip was found when it was not expected.
379	Skip Signal Not Found	Skip signal check code was included but skip was not found when it was expected.
380	X, Y, A, or G49 Not Allowed in G37	G37 may only specify Z-axis and must have tool offset defined.
381	G43 or G44 Not Allowed in G36 or G136	Auto work offset probing must be done without tool offset.
382	D Code Required in G35	A Dnn code is required in G35 in order to store the measured tool diameter.
383	Inch Is Not Selected	G20 was specified but settings have selected metric input.
384	Metric Is Not Selected	G21 was specified but settings have selected inches.



385	Invalid L, P, or R	G10 was used to changes offsets but L , P , or R code is missing or Code In G10 invalid.
386	Invalid Address Format	An address A...Z was used improperly.
387	Cutter Comp Not Allowed With G103	If block buffering has been limited, Cutter comp cannot be used.
388	Cutter Comp Not Allowed With G10	Coordinates cannot be altered while cutter comp is active. Move G10 outside of cutter comp enablement.
389	G17, G18, G19 Illegal in G68	Planes of rotation cannot be changed while rotation is enabled.
390	No Spindle Speed	S code has not been encountered. Add an S code.
391	Feature Disabled	An attempt was made to use a control feature not enabled by a parameter bit. Set the parameter bit to 1.
392	B Axis Disabled	Same as alarm 336.
393	Invalid Motion In G74 or G84	Rigid Tapping can only be in the Z minus G74 or G84 direction. Make sure that the distance from the initial position to the commanded Z depth is in the minus direction.
394	B Over Travel Range	Same as alarm 316.
395	No G107 Rotary Axis	A rotary axis must be specified in order to perform cylindrical mapping Specified (G107).
396	Invalid G107 Rotary Axis Specified	The rotary axis specified is not a valid axis, or has been disabled.
397	Aux Axis In G93 Block	This alarm is generated if a G-code block specifies any form of interpolated motion that involves BOTH one or more of the regular axes (X, Y, Z, A, B, etc...) AND one or more of the auxiliary axes (C, U, V, W).
398	Aux Axis Servo Off	Aux. axis servo shut off due to a fault.
403	RS-232 Too Many Progs	Cannot have more than 200 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 bestored 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 Load 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 115200 bits per second. Computer sending data may not respond to X-OFF
415	RS-232 Overrun	Data sent too fast to CNC. This alarm is not normally possible as this control can keep up with even 115200 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	Floppy Dir Insufficient Memory	Floppy memory was almost full when an attempt was made to read the floppy directory.
430	Floppy Unexpected End of Input	Check your program. An ASCII EOF code was found in the input data before program receive was complete. This is a decimal code 26.
431	Floppy No Prog Name	Need name in programs when receiving ALL; otherwise has no way to store them.



432	Floppy Illegal Prog Name	Check files being loaded. Program must be Onnnn and must be at the beginning of a block.
433	Floppy Empty Prog Name	Check your program. Between % and % there was no program found.
434	Floppy Load Insufficient Memory	Program received doesn't fit. Check the space available in the LIST PROG mode and possibly delete some programs.
435	Floppy Abort	Could not read disk.
436	Floppy File Not Found	Could not find floppy file.
501	Too Many Assignments In One Block	Only one assignment macro assignment is allowed per block. Divide block into multiple blocks.
502	[Or = Not First Term In Expressn	An expression element was found where it was not preceded by "[" or "=", that start expressions.
503	Illegal Macro Variable Reference	A macro variable number was used that is not supported by this control, use another variable.
504	Unbalanced Paren. In Expression	Unbalanced brackets, "[" or "]", were found in an expression. Add or delete a bracket.
505	Value Stack Error	The macro expression value stack pointer is in error. Call your dealer.
506	Operand Stack Error	The macro expression operand stack pointer is in error. Call your dealer.
507	Too Few Operands On Stack	An expression operand found too few operands on the expression stack. Call your dealer.
508	Division By Zero	A division in a macro expression attempted to divide by zero. Re-configure expression.
509	Illegal Macro Variable Use	See "Macros" section for valid variables.
510	Illegal Operator or Function Use	See "Macros" section for valid operators.
511	Unbalanced Right Brackets	Number of right brackets not equal to the number of left brackets.
512	Illegal Assignment Use	Attempted to write to a read-only macro variable.
513	Var. Ref. Not Allowed With N Or O	Alphabetic addresses N and O cannot be combined with macro variables. Do not declare N#1, etc.
514	Illegal Macro Address Reference	A macro variable was used incorrectly with an alpha address. Same as 513.



515	Too Many Conditionals In a Block	Only one conditional expression is allowed in any WHILE or IF-THEN block.
516	Illegal Conditional Or No Then	A conditional expression was found outside of an IF-THEN, WHILE, or M99 block.
517	Exprsn. Not Allowed With N Or O	A macro expression cannot be concatenated to N or O. Do not declare O[#1], etc.
518	Illegal Macro Exprsn Reference	An alpha address with expression, such as A[#1+#2], evaluated incorrectly. Same as 517.
519	Term Expected	In the evaluation of a macro expression an operand was expected and not found.
520	Operator Expected	In the evaluation of a macro expression an operator was expected and not found.
521	Illegal Functional Parameter	An illegal value was passed to a function, such as SQRT[or ASIN[.
522	Illegal Assignment Var Or Value	A variable was referenced for writing. The variable referenced is read only.
523	Conditional Req'd Prior To THEN	THEN was encountered and a conditional statement was not processed in the same block.
524	END Found With No Matching DO	An END was encountered without encountering a previous matching DO. DO-END numbers must agree.
525	Var. Ref. Illegal During Movement	Variable cannot be read during axis movement.
526	Command Found On DO/END Line	A G-code command was found on a WHILE-DO or END macro block. Move the G-code to a separate block.
527	= Not Expected Or THEN Required	Only one Assignment is allowed per block, or a THEN statement is missing.
528	Parameter Precedes G65	On G65 lines all parameters must follow the G65 G-code. Place parameters after G65.
529	Illegal G65 Parameter	The addresses G, L, N, O, and P cannot be used to pass parameters.
530	Too Many I, J, or K's In G65	Only 10 occurrences of I, J, or K can occur in a G65 subroutine call. Reduce the I, J, or K count.
531	Macro Nesting Too Deep	Only four levels of macro nesting can occur. Reduce the amount of nested G65 calls.
532	Unknown Code In Pocket Pattern	Macro syntax is not allowed in a pocket pattern subroutine.



533	Macro Variable Undefined	A conditional expression evaluated to an UNDEFINED value, i.e. #0. Return True or False.
534	DO Or END Already In Use	Multiple use of a DO that has not been closed by and END in the same subroutine. Use another DO number.
535	Illegal DPRNT Statement	A DPRNT statement has been formatted improperly, or DPRNT does not begin block.
536	Command Found On DPRNT Line	A G-code was included on a DPRNT block. Make two separate blocks.
537	RS-232 Abort On DPRNT	While a DPRNT statement was executing, the RS-232 communications failed.
538	Matching END Not Found	A WHILE-DO statement does not contain a matching END statement. Add the proper END statement.
539	Illegal Goto	Expression after "GOTO" not valid.
540	Macro Syntax Not Allowed	A section of code was interpreted by the control where macro syntax is not permitted. In lathe controls, PQ sequences describing part geometry cannot use macro statements in the part path description.
541	Macro Alarm	This alarm was generated by a macro command in a program.
613	Command Not Allowed In Cutter Comp.	A command (M96, for example) in the highlighted block cannot be executed while cutter comp. is invoked.
620	C Axis Disabled	Parameters have disabled this axis
621	C Over Travel Range	C-axis will exceed stored 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.

The following alarms apply only to the the sidemount tool changer on the vertical mills:

622	Tool Arm Fault	This alarm supports the serpentine type 1 and the Disk type 1 tool changers. It is generated if the arm is not at the Origin position, or the arm motor is already on when a tool change process is started.
623	Side Mount Carousel Error	<p>This alarm supports the serpentine type 1 and the Disk type 1 tool changers. It is generated if the carousel motor is still on when the tool pocket is unlocked and lowered prior to a tool change.</p> <p>On the serpentine type 1 tool changer, during tool change recovery, this alarm is also generated if the carousel does not start to rotate after the allowed time specified by parameter 60 TURRET START DELAY, or does not stop rotating after the allowed time specified by parameter 61 TURRET STOP DELAY.</p>



- 624 Invalid Tool
- This alarm supports the serpentine type 1 and the Disk type 1 tool changers. It is generated if the tool specified by the G-code program is not found in the POCKET- TOOL table.
- 625 Invalid TC Start Condition
- This alarm supports the serpentine type 1 and the Disk type 1 tool changers. It is generated if conditions are not correct when a tool change is started. Incorrect conditions include:
- Tool not found in POCKET- TOOL table
 - Unable to move carousel to specified tool
 - Shot pin not engaged (serpentine type 1).
 - Tool carousel not at TC mark (Disk type 1).
 - Arm not in Origin position.
 - Arm motor already on.
- 626 Tool Pocket Slide Error
- This alarm supports the serpentine type 1 and the Disk type 1 tool changers. It is generated if the tool pocket has not moved to its commanded position (and settled) within the time allowed by parameters 306 and 62. This alarm can also be generated if the tool changer is in an invalid position at power-up, the start of a gear change or tool change.
- 627 ATC Arm Position Timeout
- This alarm supports the serpentine type 1 and the Disk type 1 tool changers. It is generated if the tool arm has not moved after the allowed time or has not stopped after the allowed time. Refer to Parameter 309 MOTOR COAST TIME.
- 628 ATC ARM Positioning Error
- This alarm supports the serpentine type 1 and the Disk type 1 tool changers. It is generated if:
- The arm was being moved from the ORIGIN position to the CLAMP position and it coasted past the MOTOR STOP point or could not get to the CLAMP point.
 - The arm was being moved from the CLAMP position to the UNCLAMP position and it coasted past the MOTOR STOP point or could not get to the UNCLAMP point (same physical point as CLAMP).
 - The arm was being moved back to the ORIGIN position and it coasted past the MOTOR STOP point or could not get to the ORIGIN point.
- 631 APC-PALLET NOT CLAMPED OR HOME
- DO NOT ATTEMPT TO MOVE X OR Y AXES OR MILL UNTIL APC IS IN A SAFE CONDITION. CAUTION- The APC is not in a safe operating condition. One pallet is at home but the other pallet is neither clamped nor at home. Locate the unclamped pallet, go to the lube/air panel at rear of mill and continuously press both white buttons in center of solenoid air valves while an assistant pulls the pallet off the receiver.
- 632 APC-UNCLAMP ERROR
- The pallet did not unclamp in the amount of time allowed. This can be caused by a bad air solenoid, a blocked or kinked air line, or a mechanical problem.



- | | |
|-----------------------------------|---|
| 633 APC-CLAMP ERROR | The pallet did not clamp in the amount of time allowed by the M50 macro. This alarm is most likely caused by the VMC table not being in the correct position. This can be adjusted using the macro variables for the X position (#500 and 504) as described in the "installation section". If the pallet is clamped, but not correctly, run an M17 to unclamp, push the pallet to the correct position, and run an M18 to clamp the pallet. Less common causes could be that the slip clutch is slipping, the motor is at fault, or an air line is blocked or kinked. |
| 634 APC-MISLOCATED PALLET | A pallet is not in the proper place on the APC. The pallet must be pushed back against the hard stop by hand. |
| 635 APC-PAL NUM CONFLICT REC & CH | Pallet Number Conflict Receiver and Pallet Changer. The incorrect pallet number is entered in Macro variable #510. Run an M50 to reset this variable. |
| 636 APC-SWITCH MISSED PAL 1 | Pallet #1 did not return from the receiver to the APC in the allowed time. This can be caused by the chain switch block missing the limit switch, or from another mechanical problem, such as clutch slippage. |
| 637 APC-SWITCH MISSED PAL 2 | Pallet #2 did not return from the receiver to the APC in the allowed time. This can be caused by the chain switch block missing the limit switch, or from another mechanical problem, such as clutch slippage. |
| 638 APC- DOOR NOT OPEN | The automatic door did not open (in the allowable time), when necessary to perform an APC function. This can be caused by a bad air solenoid, a blocked or kinked air line, or a mechanical problem. |
| 639 APC- DOOR NOT CLOSED | The automatic door did not close (in the allowable time), when necessary after an APC function has been performed. This can be caused by a bad air solenoid, a blocked or kinked air line, or a mechanical problem. |
| 640 APC- MISSING PALLET @ REC | Pallet change sequence was halted because receiver switch was not activated. Pallet is either unclamped or not on the receiver. Ensure the pallet is correctly located on the receiver (against the hard stop) then run M18 to clamp the pallet. |
| 641 APC-UNKNOWN CHAIN LOCATION | Neither chain location switch is tripped, so the control cannot locate the chain position. This can occur if a pallet change is interrupted for any reason, such as an alarm or an E-stop. To correct this problem, the pallets and chain must be moved back into a recognized position, such as both pallets home or one pallet home and one on the receiver. The chain position adjustment tool must be used to rotate the chain into position. The pallets must be pushed into place by hand. |
| 642 APC- INCORRECT CHAIN LOCATION | Chain not in position to load or unload pallets when necessary. To correct this, the mislocated pallet must be moved back into the proper position by hand. |

End Of List



Note: Alarms 1000-1999 are user defined by macro programs.

The following alarms only apply to vertical mills with a pallet changer:

- | | |
|--------------------------------|--|
| 1002 Unclamp Error | The pallet did not unclamp in the amount of time allowed. This can be caused by a bad air solenoid, a blocked or kinked air line, or a mechanical problem. |
| 1003 Clamp Error | <p>The pallet did not clamp in the amount of time allowed by the M50 macro. This alarm is most likely caused by the VMC table not being in the correct position. This can be adjusted using the macro variables for the X position (#500, 504) as described in the "Installation" section. If the pallet is in the correct position but not clamped, push the pallet against the hard stop and run an M18.</p> <p>If the pallet is clamped, but not correctly, run an M17 to unclamp, push the pallet to the correct position, and run an M18 to clamp the pallet. Less common causes could be that the slip clutch is slipping, the motor is at fault, an air solenoid is bad, or an air line is blocked or kinked.</p> |
| 1004 Mislocated Pallet @ APC | A pallet is not in the proper place on the APC. The pallet must be pushed back against the hard stop by hand. |
| 1005 Pal No Conflict Rec & Ch | (Pallet Number Conflict Receiver and Pallet Changer)The incorrect pallet number is entered in macro variable #510. Run an M50 to reset this variable. |
| 1006 Switch Missed Pal 1 | Pallet #1 did not return from the receiver to the APC in the allowable amount of time. This can be caused by the chain switch block missing the limit switch, or from another mechanical problem, such as clutch slippage. |
| 1007 Switch Missed Pal 2 | Pallet #2 did not return from the receiver to the APC in the allowable amount of time. This can be caused by the chain switch block missing the limit switch, or another mechanical problem, such as clutch slippage. |
| 1008 Door Not Open | The automatic door did not open (in the allowable time) when necessary to perform an APC function. This can be caused by a bad air solenoid, a blocked or kinked air line, or a mechanical problem. |
| 1009 Door Not Closed | The automatic door did not close (in the allowable time) when necessary after an APC function has been performed. This can be caused by a bad air solenoid, a blocked or kinked air line, or a mechanical problem. |
| 1010 Missing Pallet @ Receiver | Pallet change sequence was halted because receiver switch was not activated. Pallet is either unclamped or not on the receiver. Ensure the pallet is correctly located on receiver (against hardstop) then run M18 to clamp the pallet. |



1011 Unknown Chain Location

Neither Chain Location switch is tripped, so the control cannot locate the chain position. This can occur if a pallet change is interrupted for any reason, such as an alarm or an E-STOP. To correct this problem, the pallets and chain must be moved back into a recognized position, such as both pallets home or one pallet home and one on the receiver. The chain position adjustment tool must be used to rotate the chain into position. The pallets must be pushed into place by hand.

WARNING!

Do not move the limit switches for any reason.

CAUTION! The pallets weigh 300 lbs. each, and can cause serious injury. Use extreme caution when moving them.

1012 Incorrect Chain Location

Chain not in position to load or unload pallets when necessary. To correct this, the mislocated pallet must be moved back into the proper position by hand.

CAUTION! The pallets weigh 300 lbs. each, and can cause serious injury. Use extreme caution when moving them.

The following alarms only apply to horizontal mills with a pallet changer:

1001 Index St Unlocked	The index station is not in the correct orientation for a pallet change.
1002 Pallet Locked Down	The pallet did not begin to lift within two seconds of command, or did not complete lifting within six seconds.
1003 Pallets Jammed	The lift cylinder has not moved from the clockwise position within three seconds, or has not reached the counter clockwise position within twelve seconds.
1004 CW/CCW Switch Illegal Condition	One or both of the switches that sense the rotational position of the pallet changer has failed its self-test.
1007 Up/Down Switch Illegal Condition	One or both of the switches that sense the lifted/lowered position of the pallet changer has failed its self-test.
1008 Main Drawbar Locked In Up Position	The main drawbar will not disengage from the pallet nut.
1009 Main Drawbar Locked In Down Position	The main drawbar will not move upward to the pallet nut.
1010 Main Drawbar Switch Illegal Condition	One or both of the switches that sense the up/down position of the main drawbar has failed its self-test.
1011 Main Drawbar Unclamp Timeout	The main drawbar has disengaged from the pallet nut, but did not reach the main drawbar down switch.
1012 Main Drawbar Clamp Timeout	The main drawbar has begun to travel upward, but did not reach the fully raised position within 15 seconds.



THIS PAGE INTENTIONALLY LEFT BLANK



This section contains the following:

- Quick Code
- Advanced Editor
- Macros
- Programmable Coolant
- Through the Spindle Coolant
- Automatic Pallet Changer

11. OPTIONS

11.1 QUICK CODE

INTRODUCTION

This programming option can be activated by contacting your local HAAS dealer.

Quick Code 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 menu selection you can replace a large number of individual keystrokes, with just a few. 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 handle cursor over to the drill cycle you want and press the write button and all the code you need to drill the hole is inserted with default values for all necessary "I,K,Q,P's". And you can edit those values to suit your individual needs.

How It Works

Quick Code reverses the G code encryption confusion. On the right side of the screen you have English commands that describe the operation to perform. By selecting the operation and with one button push, the code is inserted in your program on the left side of the screen. A program is constructed by selecting English commands that are then changed over to machine language or G codes. In doing this you will learn quickly the G code format without studying any manual. Another feature is the ability to cursor through a program and Quick Code will tell you what all the G and M codes mean, shown at the bottom of the screen, a great help in learning the code.



An Open System

One of the neatest features of Quick Code is that it is adaptable to the way you program. Everybody programs a little differently and have special preferences, such as, do you put the "T" command on the same line as the tool change command 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 letting you define innovative new ways to program complex parts using Quick Code.

What it is Not

Quick Code is not a CAD/CAM package for generating complex moves on 3 dimensional parts. With most CAM packages you have to draw a drawing much like you would in AUTO CAD and then indicate the moves around the drawing and finally generate the code through the post processor. Not a simple task. Nor is it a conversational program with icons where you are asked to fill in the blanks. The difference with these packages is that they require training and much like learning a second language you have to have the time and determination to learn them. They have a tremendous amount of power but you don't always need it. Quick Code is a bridge between high end CAD/CAM and slow and cumbersome G code programming. It is our expectation that it can be used by anyone with very minimal training. For most simple parts we believe that Quick Code is an ideal choice.

Conversational Quick code

Conversational Quick Code makes programming with Quick Code even easier. This feature can be used to "prompt" the operator for the information necessary to create a program. Refer to the " Conversational Quick Code" section for a description of how to use this feature.

QUICK CODE TERMINOLOGY

Before describing the QUICK CODE environment you need to know the terms listed below. Following this brief list is an illustration of the QUICK CODE display and how the terms are related to the display.

EDIT WINDOW	Portion of the display that shows the currently edited program.
GROUP WINDOW	Portion of the display which presents a list of groups and items.
GROUP	A list of items that usually have something in common so that they can be grouped together.
ITEM	A line of text representing code that can be added to the edit window when it is selected.
HELP WINDOW	Portion of the display which presents user created help, address code help, and warning messages.

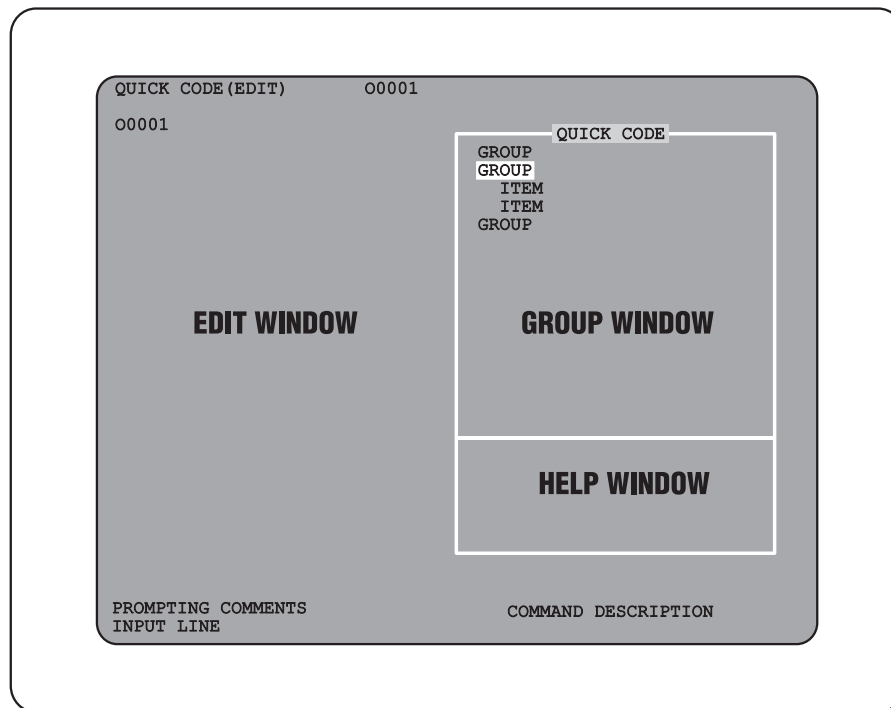


Figure 11-1. The QUICK CODE display.

USAGE AND FEATURES

ACCESSING QUICK CODE

Before QUICK CODE 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 QUICK CODE screen. Enter QUICK CODE 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 QUICK CODE screen. Each additional press of this key will switch between the Advanced editor, the standard editor, and QUICK CODE modes.

THE EDIT WINDOW

The QUICK CODE 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 QUICK CODE, the jog handle is reserved to maneuver through 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 QUICK CODE. 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. QUICK CODE is not available while in BACKGROUND EDIT MODE.



THE GROUP WINDOW

The group window displays a list of groups that are defined in the QUICK CODE 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 QUICK CODE source file help, address code help, and warning messages to the user.

The QUICK CODE 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 QUICK CODE 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

QUICK CODE makes use of the jog handle to select from the group list and group items. This is described in the group window section above. QUICK CODE 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 QUICK CODE to take the following action:

-If the currently highlighted Quick Code item is designed as a text help item only, the edit window is not modified.

-If numeric program code is found associated with the highlighted Quick Code 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 QUICK CODE block that was inserted.



CONVERSATIONAL QUICK CODE

Quick Code is used to "prompt" the operator for the information necessary to create a program. The "prompting comments" are created by placing a '?' as the first character of a comment in the Quick Code source file (O9999). A comment is any text, up to 34 characters, that is contained in parentheses. When a program is being written using Quick Code, the prompting comments will appear on the screen, requiring a response from the operator. The numeric value entered by the operator will be assigned to the G-code item that immediately precedes the prompting comment in the source file. For example, the following line of code:

```
G01 F25. (?WHAT IS THE FEED RATE ?) X2. Y5. ;
```

Will produce the prompt with the default feed rate value displayed below the prompt, as shown:

```
WHAT IS THE FEED RATE ?
```

```
F25. 
```

The operator must then enter a numeric value and press the WRITE key to change the default feed rate, or simply press the WRITE key to accept the default feed rate. The control will wait for an operator response before entering the block to the program. Unacceptable responses, such as those containing too many digits or an unnecessary decimal, will cause the control to flash an error message and wait for another response.

Once the operator has entered a value for all of the 'variable' G-code items in a block of code, the entire (revised) block is displayed on the input line, as shown:

```
CORRECT (Y/N) ?
```

```
G01 F25. X2. Y5. ; 
```

If the block of code is too long to fit on the screen, the operator can scroll to view the entire line using the right or left arrow keys, the HOME key, or the END key. The operator then must enter 'Y', or 'enter' to accept the block, or 'N', or 'cancel' to cancel it. If it is accepted, the block is written to the edit file, and the Quick Code processing resumes with the next block (if there is one). If it is not accepted, the prompting process is repeated for the same block.

Pressing the UNDO key will exit the current block at any time while in Quick code, and it will also undo any previous blocks done. in Quick Code.

Pressing RESET will exit Quick Code and send the cursor back to the beginning.

**A SAMPLE QUICK CODE SESSION**

Quick Code screens may vary slightly from what is displayed on the control.

The following illustrates how QUICK CODE 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 QUICK CODE 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 QUICK CODE 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

QUICK CODE 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 QUICK CODE.

The following screen is presented.

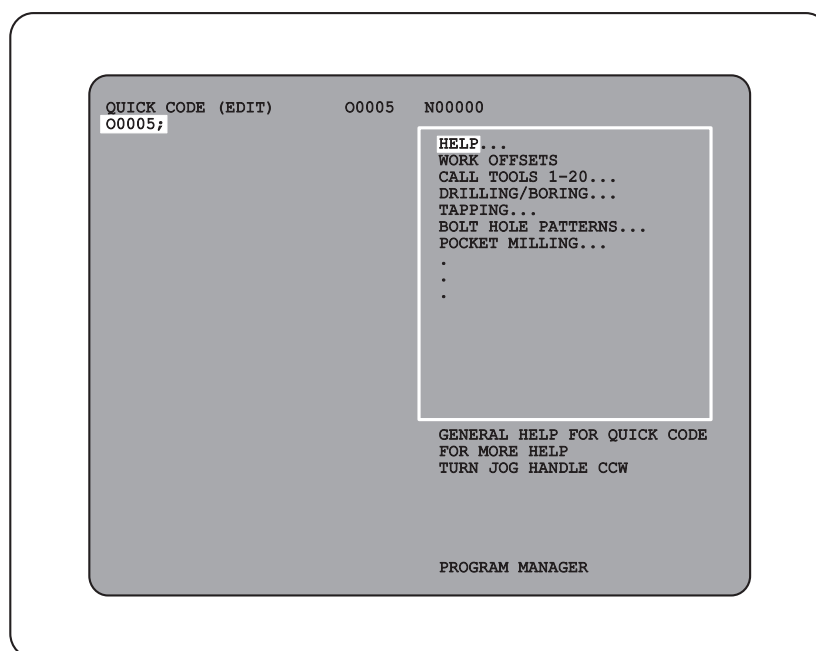


Figure 11-2. Empty Program.



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 QUICK CODE 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.

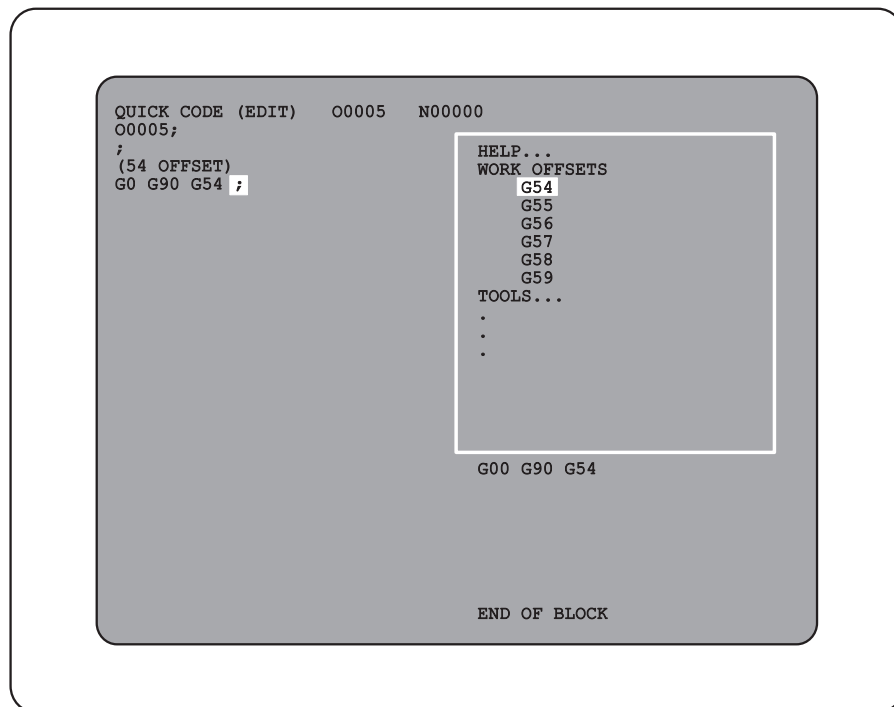


Figure 11-3. Program with G54 added.



- 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.

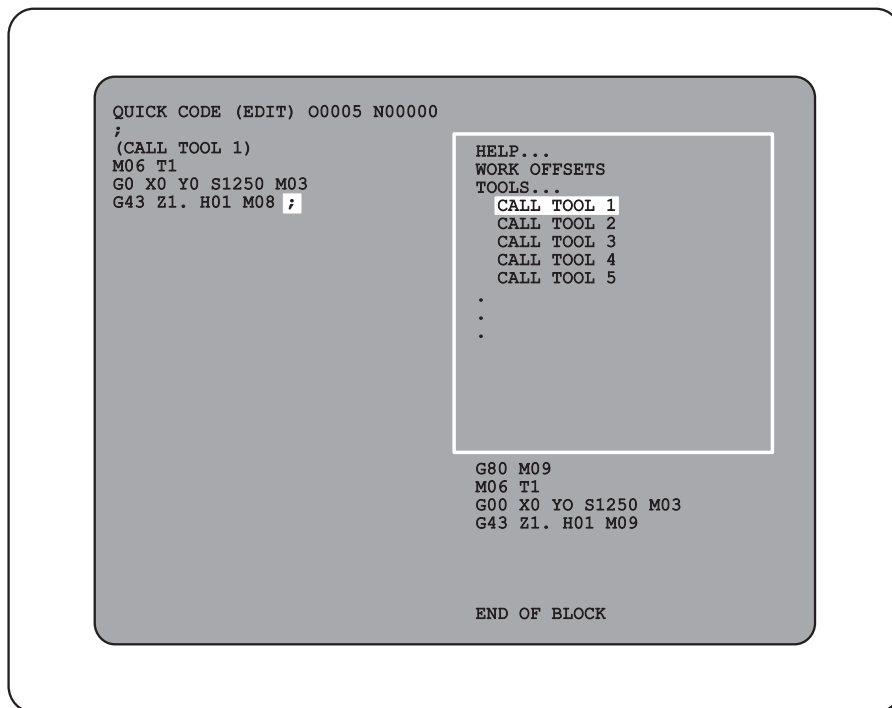


Figure 11-4. Program with CALL TOOL 1 added.

Remember that after pressing WRITE you can always edit the program to make minor adjustments to code that QUICK CODE inserts into your program. You do not have to leave the QUICK CODE display to do this. But you must remember to reposition the cursor back to the block where you want to add the next item. QUICK CODE 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 QUICK CODE source file was set up with average spindle speeds in mind. You could have different QUICK CODE files for different materials. By changing parameter 228 you can quickly change the file that QUICK CODE 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 QUICK CODE 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.

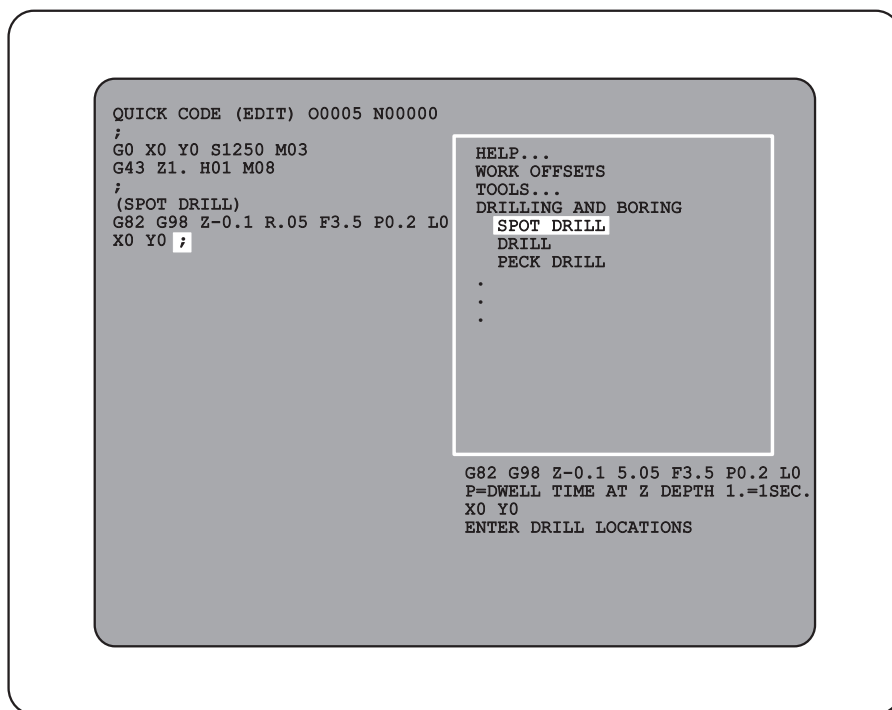


Figure 11-5. Program with spot drilling invoked.

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 QUICK CODE 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.

11.5 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.

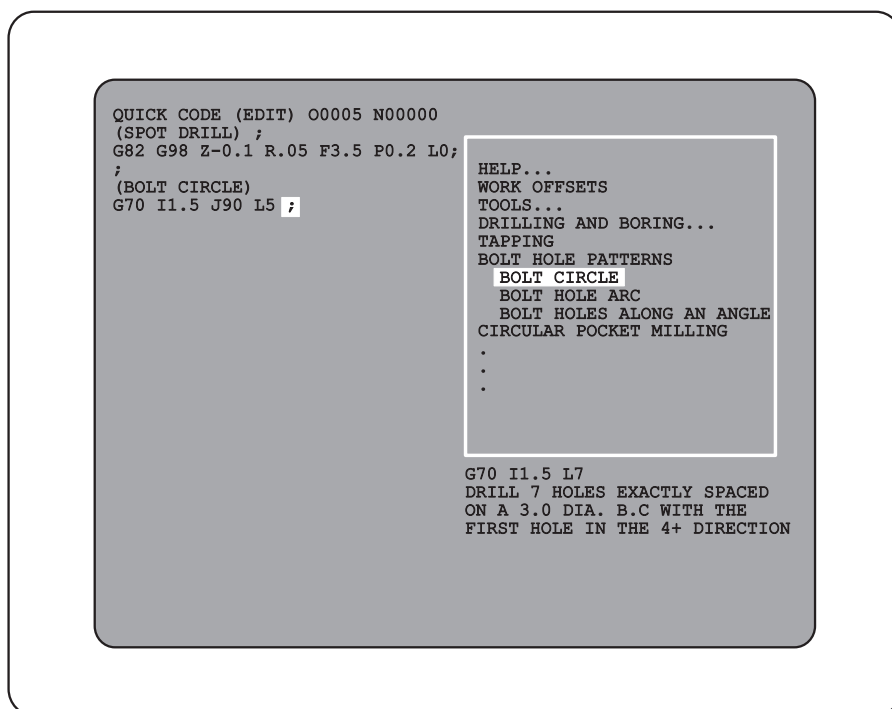


Figure 11-6. Program with a bolt hole circle added.

By now you should have a good idea of how your program changes after selecting a group item and pressing WRITE. To save space we will not show you each display as a selection is made. Instead we will list the remaining actions needed to finish drilling and tapping the 5 holes. The remaining selections are very similar to what we have already done.



- CALL TOOL 2

- 1.) JHCW and highlight the group titled TOOLS.
- 2.) JHCCW two clicks. CALL TOOL 2 will be highlighted.
- 3.) Press the WRITE key.

- INVOKE A DRILLING CANNED CYCLE

- 1.) JHCW and highlight the group titled DRILLING / BORING.
- 2.) JHCCW two clicks. DRILL will be highlighted.
- 3.) Press the WRITE key.

For most drilling cycles you will have to edit the generated program to specify the depth of drilling. And since we are doing a bolt hole pattern, don't forget to remove the X0 Y0 supplied as a sample drill location.

- EXECUTE A CIRCULAR BOLT HOLE PATTERN

- 1.) JHCW and highlight the group titled BOLT HOLE PATTERNS.
- 2.) JHCCW one click. BOLT CIRCLE will be highlighted.
- 3.) Press the WRITE key.

And now for tool 3, the 10-32 tap.

- CALL TOOL 3

- 1.) JHCW and highlight the group titled TOOLS.
- 2.) JHCCW three clicks. CALL TOOL 3 will be highlighted.
- 3.) Press the WRITE key.

- INVOKE A TAPPING CYCLE

- 1.) JHCW and highlight the group titled TAPPING.
- 2.) JHCCW one click. 10-32 TAP will be highlighted.
- 3.) Press the WRITE key.

Note that for all tapping you will never have to calculate the spindle speed and feed again. Only the depth of the tap will have to be edited, and perhaps your X and Y coordinates.

Remove the supplied X0 Y0 to prepare for the bolt hole pattern as done above.



- EXECUTE A CIRCULAR BOLT HOLE PATTERN

- 1.) JHCW and highlight the group titled BOLT HOLE PATTERNS.
- 2.) JHCCW one click. BOLT CIRCLE will be highlighted.
- 3.) Press the WRITE key.

At this point you may decide to move the table forward to remove the part. You can do this in your program with the following.

- HOME THE Y AND Z AXES

- 1.) JHCW and highlight the group titled MISC. INSTRUCTIONS.
- 2.) JHCCW until HOME Y AND Z AXIS is highlighted.
- 3.) Press WRITE.

And finally, you can properly terminate your program with:

- PROGRAM RESTART

- 1.) JHCCW until END PROGRAM/RESET TO BEGINNING is highlighted.
- 2.) Press WRITE.

You now have a ready to run program. You should always verify everything in graphics to make certain that you have not forgotten any steps. Although this looks like a lot of steps, it is actually very easy once you become familiar with the QUICK CODE environment. The above program can be generated in less than a minute.

THE QUICK CODE 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 QUICK CODE source file. With this design, the user can modify QUICK CODE and tailor it to his specific needs. You can add or change groups and items. The user can develop his own QUICK CODE 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 QUICK CODE an extremely flexible tool.

SOURCE FILE PROGRAM DESIGNATION

Program number 9999 is the default QUICK CODE source file. Every HAAS control equipped with QUICK CODE 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 QUICK CODE 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 QUICK CODE groups. You can use the following skeleton as a start for defining a QUICK CODE source file.



```
%
O9999 (QUICK CODE - 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.      )
()
(QUICK CODE GROUP DEFINITIONS FOLLOW)
.
.
.
()
(END OF QUICK CODE)
%
```

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 QUICK CODE 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 QUICK CODE. 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 QUICK CODE. This is a method of documenting the source file for the developer of the QUICK CODE 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.

QUICK CODE 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 QUICK CODE source file, only one group is displayed in the group window when the jog handle is turned clockwise. When the jog handle is turned counter clockwise, the five items are displayed and traversed. The item titles are indented one space so that you can differentiate items from groups. Only 34 characters of the item definition comment are displayed in the group window. Additional characters are ignored. The group that items belong to will always be displayed on the screen. The only limit to the number of items in a group is the amount of control memory available.

ITEM HELP

Item help works the same way as group help. The first four comments after the item definition are displayed in the help window. If more than four lines are required, it is recommended that prior items contain the desired comments. In this case instructions would have to be added to indicate which item generates the

G code.

For example:

```
(*GROUP)
(**HELP FOR THE FOLLOWING ITEM)
(THESE LINES OF CODE ARE HELP)
(COMMENTS THAT REQUIRE MORE THAN)
(FIVE LINES OF COMMENTARY)
(THIS IS THE LAST LINE OF THIS ITEM)
(**ITEM THAT GENERATES CODE)
(AND HERE WE FINISH THE COMMENTARY)
(FOR CODE GENERATED BY THIS ITEM)
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.

**A SAMPLE QUICK CODE SOURCE FILE**

After developing or modifying a QUICK CODE file, it is recommended that you save an off-line copy in a computer. You can keep comments in the QUICK CODE source file prior to the first group indicating what version the file is and how it differs from other versions. Maintain this program as you would any other G code program in your control with a proper backup scheme. **Remember! This file operates the QUICK CODE feature in your HAAS machine.**

A sample QUICK CODE source file can be found on the floppy that comes with the control. It contains many examples of how QUICK CODE can be used.



11.2 ADVANCED EDITOR

The HAAS Advanced Editor gives the user the ability to view and edit two CNC programs at a time. This makes it easier to modify existing programs and to create new ones. The editor has an 80 column display, and includes pull-down menus that allow the user to access the features of the editor. Additionally, a context-sensitive help function is available to provide information on all of the editor's features.

The following terms are used throughout this addendum to describe the advanced editor:

CURRENT PROGRAM	The program that is expected to be run from MEM or MDI mode.
ACTIVE PROGRAM	The program that is altered by user input.
INACTIVE PROGRAM	The program opposite the active program in the editor.
CONTEXT-SENSITIVE HELP	A help function that provides information based on what the user is currently doing.
PULL-DOWN MENUS	Menus accessed, or "pulled down", via the "menu bar" at the top of the screen. Only one menu can be accessed at any one time. When a menu is pulled down, items appear that can be scrolled through and selected.
HOT KEY	A key that, when pressed, will immediately execute an editor function.

The advanced editor is entered by pressing the EDIT key. The old 40 column editor can be accessed by pressing the PRGRM/CONVRS key. Another press of PRGRM/CONVRS will place the user into the Quick Code display. The Quick Code application can also be accessed from within the F1:HELP pull-down menu. A third press of the PRGRM/CONVRS key will access the advanced editor. The user can alternate between the advanced editor, the 40 column editor, and Quick Code with successive presses of the PRGRM/CONVRS key.

All of the functions of the 40 column editor are still available in the advanced editor. However, the F1 key, which used to mark the beginning of selected text, now provides access to the pull-down menus and context-sensitive help. Pressing the F1 key activates a pull-down menu, which appears descending from the menu bar. If the pull-down menu system has not been accessed during the current editor session, the Help menu is pulled down. Otherwise the most recently accessed menu is pulled down.

Whenever the pull-down menu system is active, the current menu is pulled down and one item is highlighted. The user can then use the jog handle to scroll through the menu items. The user can also use the up and down arrow keys to scroll through the items of that menu, or use the left and right arrow keys to open other menus.

When a menu item is highlighted, the user can see a brief description of it in the lower right hand corner. This activates context-sensitive help, which describes what that menu item does. The context-sensitive help explains any prompts that may appear, what keys are available for action, and what the "hot key" (if one exists) is for that menu item.

The UNDO key is used to deactivate the pull-down menu system. Pressing RESET will also deactivate the pull-down menu, but UNDO is preferred. If UNDO is pressed after invoking an executing function from a pull-down menu, it will abort that function. A context-sensitive help session can also be ended by pressing the UNDO key. This will return the user to the active program.

The EDIT key can be used to “switch” between two programs that have been selected to edit.

Pressing the F4 key will open a copy of the current program in the inactive window. The user can quickly edit two different locations in the same program by pressing F4, moving to the second location, and then using the EDIT key to move back and forth between the two locations. If the user enters ‘Onnnn’ and then presses F4, program Onnnn is opened in the inactive window.

Figure 11-7 illustrates the layout of the advanced editor.

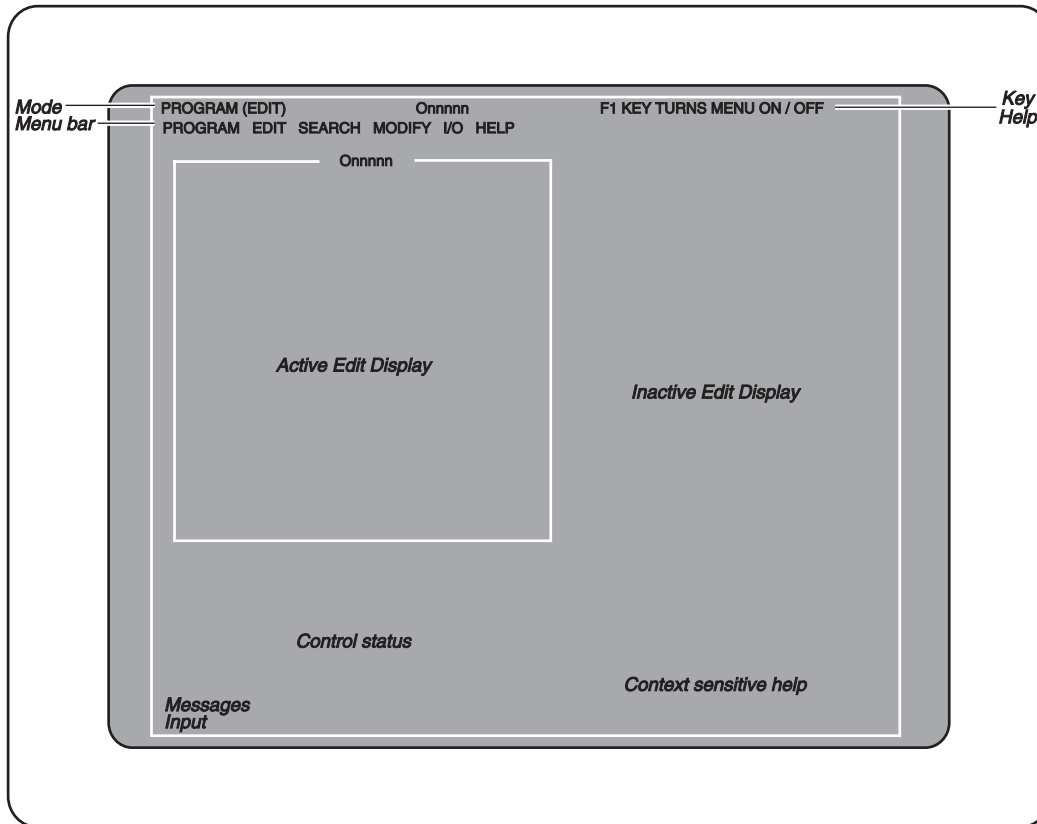


Figure 11-7. The advanced editor screen layout.



The advanced editor screen is divided into the following areas:

- **Mode and Control Status** - contain the Current Display page, Operating mode and Control status.
- **Messages** - where control status messages are displayed. This area is used to display prompts for user input in the advanced editor.
- **Input** - where the user's input is displayed.
- **Menu Bar** - contains the pull-down menu banner.
- **Key Help** - contains short immediate help messages. These are meant to show the user the most important keys that can be used in the current context or operation.
- **Left/Right Side Display** - shows the active and inactive programs. When first entering the editor, the current program will be displayed on the left side and the right side will be blank. These areas can also display program lists and help pages when the appropriate menu item is selected from the pull-down menu.
- **Context-Sensitive Help** - where context-sensitive help is displayed when a menu is active, or when the user is entering data in response to selecting a menu item.

ADVANCED EDITOR FEATURES

This section briefly describes each feature of the advanced editor, each item found in the pull-down menus, and any prompts that might appear.

CONTEXT SENSITIVE HELP (The F1 key)

While in the advanced editor, press the F1 key to get into the menus, use the arrow keys to highlight the menu item. The help text for that item will appear in the lower right corner of the screen. The jog handle, up and down arrow keys, and the PAGE UP and PAGE DOWN keys can then be used to view the help text. Press CANCEL or UNDO to go back to the menu, or RESET to exit the menus entirely.

THE PROGRAM MENU

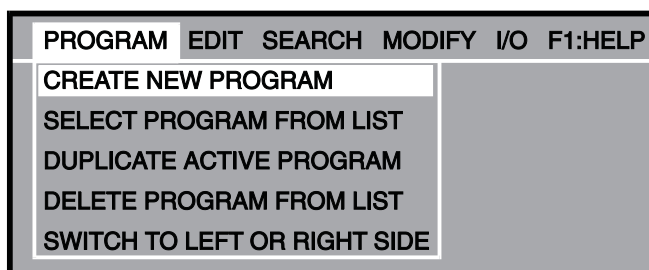


Figure 11-8. The PROGRAM menu items.



CREATE NEW PROGRAM

This menu item will create a new program, providing there is room in the program directory and enough memory is available. Enter a program name (Onnnn) in the range of 0 through 9999 that is not already in the program directory.

SELECT PROGRAM FROM LIST

The HAAS control maintains a directory of programs that the user can select. Select this menu item to edit a program that exists in the directory.

When this menu item is selected, a list of programs is presented for viewing. Scroll through the list by using the cursor keys or the jog handle. Pressing the WRITE or SELECT PROG key will select the program that is highlighted and will replace the selection list with the selected program. The selected program is now active, and the previously active program will appear on the inactive edit screen.

Program size and memory usage appear at the bottom of the display. SELECT PROG is the hot key for this item.

DUPLICATE ACTIVE PROGRAM

This menu item will create a new program, copy the contents of the current program into it, rename it as specified, and make it the active program.

ENTER NEW PROGRAM NUMBER: If no program code is present on the input line when this menu item is invoked, this prompt will appear. Enter a valid program number (Onnnn), then press the WRITE key. Only numeric inputs will be accepted.

DELETE PROGRAM FROM LIST

This menu item will delete a program from the program directory. A list of all programs is presented, with 'ALL' at the end.

To delete a single program, cursor to the program number and press the WRITE key. A prompt will ask for a confirmation of the deletion operation. Enter 'Y' to delete the highlighted program. If any other key is pressed, the program will not be deleted. After a program is deleted, the list of programs will again be presented.

To delete all programs, cursor to 'ALL' and press WRITE. Confirm deletion of all programs by pressing 'Y'. When all programs are deleted, program O0000 is created, and it is made the active program.

Press UNDO to exit this menu item and return to the active program. Program size and memory usage are displayed at the bottom of the screen.

ERASE PROG is the hot key for this item.



SWITCH TO LEFT OR RIGHT SIDE

This will make the active program inactive and the inactive program active. If there is no inactive program, then nothing happens. There are only two possible displays: one on the left and one on the right. The inactive display is used when a second program is selected or created.

The EDIT key is the hot key for this item.

THE EDIT MENU

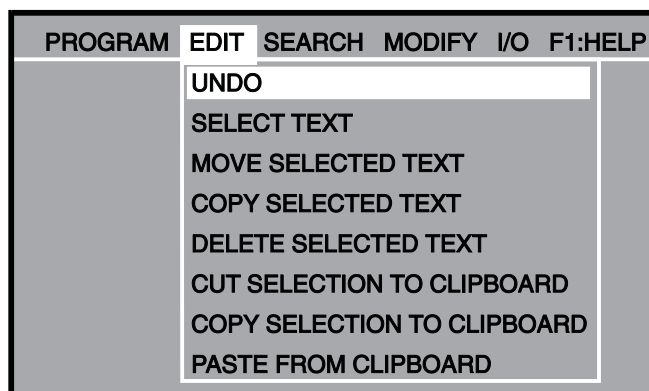


Figure 11-9. The EDIT menu items.

UNDO

The last insert, delete, or alter operation will be undone. Pressing UNDO again will restore the previous editing operation, up to the last 9 editing operations. If a block has been selected, choosing this item will simply exit block select mode without undoing anything. UNDO is the hot key for this item.

SELECT TEXT

This item will set the start point of the text selection. To set the end point, scroll up or down to the desired place, and press the F2 or WRITE key. The selected text will then be highlighted. To deselect the text, press UNDO. This function works the same as in the old editor, except the menu option is used to start selecting text, instead of the F1 key, and either the WRITE or F2 key can be used to end the selection. The following prompt will appear when this item is selected:

SCROLL TO END POINT, PRESS WRITE OR F2: Press the WRITE or F2 key to complete the text selection.

MOVE SELECTED TEXT

All selected text will be moved to the line following the current cursor position.

COPY SELECTED TEXT

All selected text will be copied to the line following the current cursor position.

DELETE SELECTED TEXT

This item deletes any selected text. If no text is selected, the currently highlighted item is deleted. This function works the same as in the old editor, except that if the cursor is in the middle of a comment, it will delete the entire comment instead of just the highlighted character. The UNDO key will restore any deleted comment. The DELETE key deletes individual characters from comments and is the hot key for this menu item.

CUT SELECTION TO CLIPBOARD

All selected text will be moved from the current program to a new program called the clipboard. Any previous contents of the clipboard are deleted. The program number for the clipboard is specified by Parameter 226 and can be altered, if necessary.

COPY SELECTION TO CLIPBOARD

All selected text will be copied from the current program to a new program called the clipboard. Any previous contents of the clipboard are deleted. The program number for the clipboard is specified by Parameter 226 and can be altered, if necessary.

PASTE FROM CLIPBOARD

The contents of the clipboard are copied into the current program at the line following the current cursor position.

THE SEARCH MENU

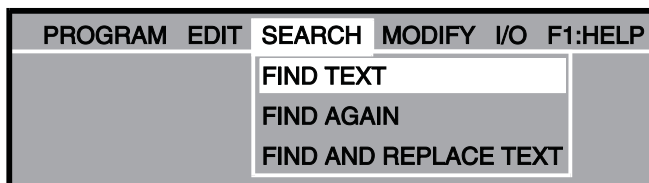


Figure 11-10. The SEARCH menu items.

FIND TEXT

This menu item will search for one or more G-Code items in the current program. The search can be performed in either the forward or backward direction from the current cursor location. If the item is found, the cursor will be positioned on it. The following prompts will appear when this item is selected:

ENTER TEXT/ITEM TO SEARCH FOR: Type in one or more G-code items, a single address code character, or a comment to be searched for. Press the WRITE key to enter this input. If a full G-Code item is specified, only items that exactly match will be found. If a single address code character is specified, the next matching address code will be found, regardless of the numeric value associated with it. And finally, if a comment is specified, the next comments that contain the text specified will be found.

FORWARD OR BACKWARD (F/B) ? Type in either 'F' or 'B'; all other input will be ignored. Entering 'F' will commence the search for the specified G-code item in the forward direction. Entering 'B' will commence the search in the backward direction; i.e. it will find the previous occurrence of the specified G-Code.



FIND AGAIN

This menu item will search the current program for the last text that was searched for. It will begin to search at the current cursor location, in the direction that was specified in the previous search. This function will search both selected and unselected blocks.

FIND AND REPLACE TEXT

This menu item will search the current program for one or more occurrences of a specified G-Code item and optionally replace each (or all) with another G-Code item. The search can be performed in either the forward or backward direction from the current cursor location. As each G-Code item is found, the cursor will be positioned on it, and a prompt will ask whether to replace the item, continue the search, both, or neither. This function affects both selected and unselected blocks. The following prompts will appear when this item is selected:

ENTER TEXT/ITEM TO SEARCH FOR: Enter either one or more G-code items, a single address code character, or a comment. Press the WRITE key to enter the input. If one or more G-code items are specified, only G-code items that exactly match will be found. If a single address code character is specified, all matching address codes will be found, regardless of their associated numeric value. And finally, if a comment is specified, all comments that contain the specified text will be found.

ENTER REPLACEMENT TEXT/ITEM: Enter one or more G-code items that will replace each occurrence of the G-Code item(s) found. If nothing is entered at this prompt, all occurrences of the found items will be deleted, upon verification.

FORWARD OR BACKWARD (F/B) ? Enter either an 'F' or a 'B' ; all other inputs will be ignored. Entering 'F' will commence the search for the specified G-code item in the forward direction. Entering 'B' will commence the search in the backward direction.

REPLACE (YES/NO/ALL/CANCEL) ? As each G-Code item is found, the cursor will be positioned on it, and this prompt will appear. Typing in a 'Y' (for 'Yes') will replace the item, and continue the search in the specified direction. Typing an 'N' (for 'No') will not replace the item, but will continue the search in the specified direction. Typing an 'A' (for 'All') will replace all occurrences of the item with the replacement text, and end the search process. Pressing any key while the editor is "replacing all" will abort the process. Typing a 'C' (for 'Cancel') will abort the search process.

THE MODIFY MENU

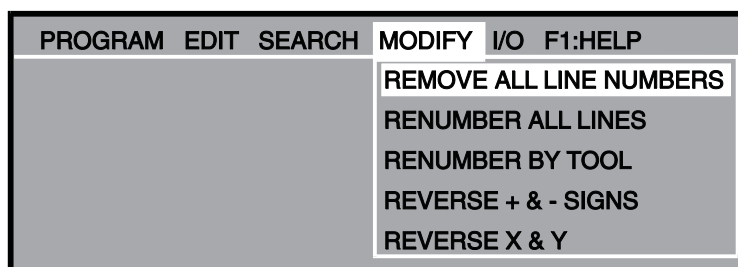


Figure 11-11. The MODIFY menu items.



REMOVE ALL LINE NUMBERS

This menu item will automatically remove all unreferenced N-Codes from the edited program. If a block is selected, only the G-Code blocks contained within it will be affected.

RENUMBER ALL LINES

This menu item will either renumber all G-Code blocks in the program or, if a block is selected, renumber only those G-Code blocks contained in that block. Below are the prompts that may be encountered while in this item, with a brief explanation of each:

ENTER STARTING N-CODE NUMBER: Type in the starting N-Code number, then press the WRITE key to enter the number. The maximum value accepted is 99999. Any non-numeric input will be ignored.

ENTER N-CODE INCREMENT: Type in the desired numeric difference between consecutive N-Codes, then press the WRITE key to enter the number.

RESOLVE OUTSIDE REFERENCES (Y/N) ? This prompt will appear only if a selected block was defined prior to execution of this menu item. Entering a 'Y' (for Yes) will cause G-Code items outside of the selected block that refer to N-Codes inside the block (such as a GOTO) to be changed to reference the new N-Codes correctly. If an 'N' (for No) is entered, G-Code references that exist outside the selected block will not be changed.

RENUMBER BY TOOL

Searches selected text, or the entire program, for T codes and rennumbers program blocks grouped by T code. The following prompts will appear when this item is selected:

ENTER STARTING N-CODE NUMBER: This prompt will appear when a T code is found. Blocks of code will be renumbered, starting with the code entered here, until the next T code is found.

ENTER N-CODE INCREMENT: Each block that is renumbered is incremented by the amount entered here.

REVERSE + & - SIGNS

This menu item will reverse the signs of all numeric values associated with one or more address codes in the program. If a certain block is selected, only the address codes in the selected block will be affected. The following prompt will appear when this item is selected:

ENTER ADDRESS CODE(S) TO CHANGE: Type in the valid address code characters whose associated values in the program are to have their signs reversed. They can be entered in any order, but duplicate entries will be ignored. Entries that don't make sense in the context of reversing signs, such as the 'G' character, will also be ignored.

REVERSE X & Y

When this menu item is selected, all X address codes in the program will be changed to Y address codes, and Y's to X. If a block is selected, only the address codes contained within it will be affected.



THE I/O MENU

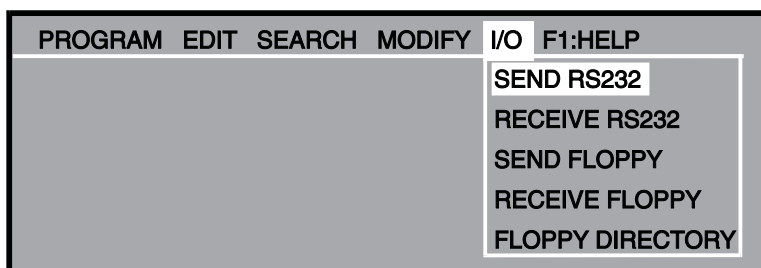


Figure 11-12. The I/O menu items.

SEND RS232

This menu will send program(s) that are selected from the program directory to the RS-232 port. When this menu item is selected, a list of all the programs in memory is presented, with 'ALL' at the end.

To select a program, cursor to the program number and press the INSERT key. A highlighted space will appear before the program to indicate it has been selected. Pressing INSERT again will deselect the program, and the highlighted space will disappear. The DELETE key can be used to deselect all selected programs. When the cursor is on "ALL", all the programs are selected regardless of highlighting.

To send the selected program(s), press the WRITE key. If more than one program or 'ALL' is selected, the data will be sent with one "%" at the beginning of the stream and one at the end.

When in the Advanced Editor, the SEND RS232 button activates the menu system and highlights the Send RS232 option.

RECEIVE RS-232

This menu item will receive program(s) from the RS-232 serial port. The program(s) will then be stored in the CNC memory with the corresponding Onnnnn program number(s).

On LISTPROG "ALL" must first be highlighted before using this menu item. The Onnnnn program numbers will be entered automatically from the input stream data. Note, "ALL" must be reselected on the LIST PROG screen after each file transfer.

SEND FLOPPY

This menu item will send program(s) to the floppy. When this menu item is selected, a list of all the programs in memory is presented, with 'ALL' at the end.

To select a program, cursor to the program number and press the INSERT key. A highlighted space will appear before the program to indicate it has been selected. Pressing INSERT again will deselect the program, and the highlighted space will disappear. The DELETE key can be used to deselect all selected programs. When the cursor is on "ALL", all programs are selected regardless of highlighting.



ENTER FLOPPY FILENAME: Type in the desired floppy filename (in standard PC DOS format) for the floppy file being sent, then press the **WRITE** key. If more than one program or "ALL" is selected, the data will be sent with one "%" at the beginning of the stream and one at the end. If a filename is not entered, the controller will send each selected file separately using the Onnnnn program number as the filename.

RECEIVE FLOPPY

This menu item will receive programs from the floppy. The program(s) will then be stored in the CNC memory with the corresponding Onnnnn program number(s).

ENTER FLOPPY FILENAME: Type in the filename (in standard PC DOS format) of the floppy file being received, then press the **WRITE** key.

FLOPPY DIRECTORY

This menu item will display the directory of the floppy, with the first entry highlighted. To select a file, press the up and down arrow buttons or use the jog handle. To load a file, select it and press the **WRITE** key. The G-code programs in the file will be loaded into memory.

If there is insufficient memory for the entire file, Alarm 429 will be generated, and only a partial directory will be displayed.

THE F1:HELP MENU

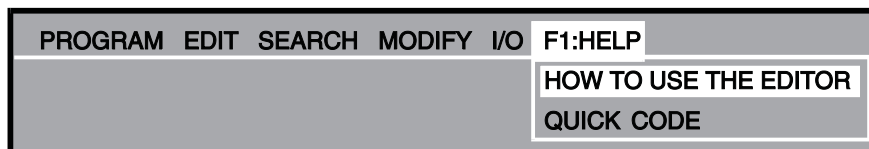


Figure 11-13. The F1:HELP menu items.

HOW TO USE THE EDITOR

The on-line help is displayed whenever a menu is accessed. The help manual gives a brief description of the editor and its features. The up and down arrows and the jog handle control the menus, and the Page Up, Page Down, Home and End keys are used to scroll through the help display. In addition, if the F1 key is pressed during the use of one of the menu options, the help is likewise displayed. Pressing F1 again will exit the help display. Pressing the **UNDO** key returns to the active program.

QUICK CODE

Selecting this menu item will place Quick Code on the inactive side of the editor. All Quick Code functions are now available to the user. Refer to the Operator's Manual for a full description of Quick Code. Pressing the **EDIT** key will exit Quick Code.

11.3 MACROS

INTRODUCTION

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

This is an introduction to macros as implemented on the HAAS CNC controls. MACROS adds capabilities and flexibility to standard G-code programming that allow the programmer to better define a tool path in a quicker and more natural way. With few exceptions, MACROS, as implemented on the HAAS controls, is compatible with FANUC 10M and 15M controls. Macro features not included in the current release are listed at the end of this section. 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.

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 GT 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)

**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:

```
G06 X.5 Y.25 Z.05 F10. T10 ;
```

Here, we have substituted an unused G code, G06 for G65 P9010. In order for the above block to work we must set the parameter associated with subroutine 9010 to 06 (Parameter 91).

Note that G00, G65, G66, and G67 can not be aliased. All other codes between 1 and 255 can be used for aliasing.

Program numbers 9010 through 9019 are reserved for G code aliasing. 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

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.

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	L	M
Variable:	1	2	3	7	8	9	-	11	4	5	6	-	13
Address:	N	O	P	Q	R	S	T	U	V	W	X	Y	Z
Variable:	-	-	-	17	18	19	20	21	22	23	24	25	26

Alternate Alphabetic Addressing

Address:	A	B	C	I	J	K	I	J	K	I	J
Variable:	1	2	3	4	5	6	7	8	9	10	11
Address:	K	I	J	K	I	J	K	I	J	K	I
Variable:	12	13	14	15	16	17	18	19	20	21	22
Address:	J	K	I	J	K	I	J	K	I	J	K
Variable:	23	24	25	26	27	28	29	30	31	32	33

Arguments accept any floating point value to four decimal places. If you are in metric, the control will assume thousandths (.000). In Example 3 below, local variable #7 will receive .0004.

If a decimal is not included in an argument value, such as:

G65 P9910 A1 B2 C3

The values are passed to macro subroutines according to the following table:

Integer Argument Passing (no decimal point)

Address:	A	B	C	D	E	F	G	H	I	J	K	L	M
Variable:	.001	.001	.001	1.	1.	1.	-	1.	.0001	.0001	.0001	1.	1.
Address:	N	O	P	Q	R	S	T	U	V	W	X	Y	Z
Variable:	-	-	-	.0001	.0001	1.	1.	.0001	.0001	.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.

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.

MACRO VARIABLES

There are three categories of macro variables: *system* variables, *global* variables, and *local* variables.

Variable Usage

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

Variables are decimal values that are represented as floating point numbers. If a variable has never been used, it can take on a special "undefined" value. This indicates that it has not been used. A variable can be set to undefined with the special variable #0. #0 has the value of undefined or 0.0 depending on the context it is used in. More about this later. Indirect references to variables can be accomplished by enclosing the variable number in brackets.

#[<expression>]

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

#1=3;

#[#1]=3.5 + #1;

This sets the variable #3 to the value 6.5.

Variables can be used in place of G-code address constants where "address" refers to the letters A..Z.

In the block

N1 G0 G90 X1.0 Y0 ;

the variables can be set to the following values:

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

and the block replaced by:

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

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

Local Variables

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

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 M97 or 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 M97 or 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

#0	Not a number (read only)
#1-#33	Macro call arguments
#100-#199	General purpose variables saved on power off
#500-#699	General purpose variables saved on power off
#700-#749	Hidden variables for internal use only.
#800-#899	General purpose variables saved on power off
#1000-#1031	32 discrete inputs (read only)
#1032-#1063	32 additional discrete inputs for expansion (read only)
#1080-#1087	Raw analog to digital inputs (read only)
#1090-#1097	Filtered analog to digital inputs (read only)
#1094	Spindle load with OEM spindle drive (read only)
#1098	Spindle load with Haas vector drive (read only)
#1100-#1131	32 discrete outputs
#1132-#1155	24 extra relay outputs via multiplexed output
#2000-#2099	Tool length offsets
#2200-#2299	Tool length wear
#2400-#2499	Tool diameter/radius offsets
#2600-#2699	Tool diameter/radius wear
#3000	Programmable alarm
#3001	Millisecond timer
#3002	Hour timer
#3004	Override control
#3006	Programmable stop with message
#3011	Year, month, day
#3012	Hour, minute, second
#3020	Power on timer (read only)
#3021	Cycle start timer (read only)
#3022	Feed timer (read only)
#3023	Present part timer (read only)
#3024	Last complete part timer (read only)
#3025	Previous part timer (read only)

#3026	Tool in spindle (read only)
#3027	Spindle RPM (read only)
#3901	M30 count 1
#3902	M30 count 2
#4001-#4020	Previous block group codes
#4101-#4126	Previous block address codes

NOTE: Mapping of 4101 to 4126 is the same as the alphabetic addressing of "Macro Arguments" section.

#5001-#5005	Previous block end position
#5021-#5025	Present machine coordinate position
#5041-#5045	Present work coordinate position
#5061-#5064	Present skip signal position
#5081-#5085	Present tool offset
#5201-#5206	Common offset
#5221-#5226	G54 work offsets
#5241-#5246	G55 work offsets
#5261-#5266	G56 work offsets
#5281-#5286	G57 work offsets
#5301-#5306	G58 work offsets
#5321-#5326	G59 work offsets
#5401-#5500	Tool feed timers (seconds)
#5501-#5600	Total tool timers (seconds)
#5601-#5699	Tool life monitor limit
#5701-#5800	Tool life monitor counter
#5801-#5900	Tool load monitor maximum load sensed so far
#5901-#6000	Tool load monitor limit
#6001-#6277	Settings (read only)
#6501-#6999	Parameters (read only)

Note: The low order bits of large values will not appear in the macro variables for settings and parameters

#7001-#7005	G110 additional work offsets
#7021-#7025	G111 additional work offsets
#7041-#7045	G112 additional work offsets
#7061-#7065	G113 additional work offsets
#7081-#7085	G114 additional work offsets
#7101-#7105	G115 additional work offsets
#7121-#7125	G116 additional work offsets
#7141-#7145	G117 additional work offsets
#7161-#7165	G118 additional work offsets
#7181-#7185	G119 additional work offsets
#7201-#7205	G120 additional work offsets
#7221-#7225	G121 additional work offsets
#7241-#7245	G122 additional work offsets
#7261-#7265	G123 additional work offsets
#7281-#7285	G124 additional work offsets
#7301-#7305	G125 additional work offsets
#7321-#7325	G126 additional work offsets
#7341-#7345	G127 additional work offsets
#7361-#7365	G128 additional work offsets
#7381-#7385	G129 additional work offsets

SYSTEM VARIABLES IN-DEPTH

This section fully describes system variables.

1-Bit Discrete Inputs

For a complete description of discrete inputs, refer to the "Technical Reference" section. Inputs designated as "spare" can be connected to external devices and used by the programmer.

#1000-#1020	Reserved for HAAS Controller use.
#1021	Spare
#1022	Spare
#1023	Spare
#1024-#1028, 1030, 1031	Reserved for HAAS Controller use.
#1029	Skip signal

1-Bit Discrete Outputs

The HAAS control is capable of controlling up to 56 discrete outputs. However, a number of these outputs are already reserved for use by the HAAS controller. The following list shows which outputs are "spare", or can be connected to external devices and used by the programmer, or "reserved" by the control, and cannot be used:

#1100-#1123	Reserved
#1124-#1126	Spare
#1127	Reserved
#1128-#1131	Reserved
#1132-#1139	M-code relay option, spare.

CAUTION!

Do not use outputs that are reserved by the system. Using these outputs may result in injury or damage to your equipment.

The user can change the state of these outputs by writing to variables designated as "spare". If the outputs are connected to relays, then an assignment of "1" sets the relay. An assignment of "0" clears the relay.

Referencing these outputs will return the current state of the output and this may be the last assigned value or it may be the last state of the output as set by some user M code. For example, after verifying that output #1108 is "spare":

#1108=1;	(Turns #1108 relay on)
#101=#3001+1000;	(101 is 1 second from now)
WHILE [[#101 GT #3001] AND [#1109 EQ 0]] D01	
END1	(Wait here 1 second or until relay #1109 goes high)
#1108=0;	(Turns #1108 relay off)

The number of outputs available to the user and where user M codes are mapped is model dependent. If your control is not equipped with the new M-code relay board, then M21 through M28 will be mapped from #1124-#1131. If you have equipment with the M-code relay board installed (Parameter 209 bit MCD RLY BRD is set to 1) then M21 through M28 will be mapped to #1132-#1139.

You should always test or dry run programs that have been developed for macros that is running with new hardware.

Tool Offsets

HAAS macros have been implemented with FANUC control memory C option in mind. This means that each tool offset has a length (H) and radius (D) along with associated wear values.

- #2001-#2100 H geometry offsets (1-100) for length.
- #2200-#2300 H geometry wear (1-100) for length.
- #2401-#2500 D geometry offsets (1-100) for diameter.
- #2601-#2700 D geometry wear (1-100) for diameter.

Programmable Messages

#3000 ALARMS can be programmed. A programmable alarm will act just like HAAS internal alarms. An alarm is generated by setting the macro variable #3000 to a number between 1 and 999.

#3000= 15 (MESSAGE PLACED INTO ALARM LIST) ;

When this is done, ALARM flashes in the lower right hand corner of the display and the text in the next comment is placed into the alarm list. The alarm number (in this example, 15) is added to 1000 and used as an alarm number. If an alarm is generated in this manner all motion stops and the program must be reset to continue. Programmable alarms can always be identified in alarm history because the alarm numbers range between 1000 and 1999.

The first 34 characters of the comment will be used for the alarm message.

Timers

HAAS macros supports access to two timers. These timers can be set to a value by assigning a number to the respective variable. A program can then later read the variable and determine the time passed since the timer was set. Timers can be used to emulate dwell cycles, determine part to part time or wherever time dependent behavior is desired.

#3001 MILLISECOND TIMER - The millisecond timer is updated every 20 milliseconds and thus activities can be timed with an accuracy of only 20 milliseconds. At POWER ON, the millisecond timer is reset. The timer has a limit of 497 days. The whole number returned after accessing #3001 represents the number of milliseconds.

#3002 HOUR TIMER - The hour timer is similar to the millisecond timer except that the number returned after accessing #3002 is in hours. The hour and millisecond timers are independent of each other and can be set separately.

System Overrides

#3004 Variable #3004 is a bitmapped variable that overrides specific control features during runtime.

The first bit disallows FEED HOLD from the keypad. If you do not want feed hold to be executed during any section of code, then bracket that code with assignments to variable #3004. Assigning "1" to #3004 disables the console's feed hold button. Assigning "0" to #3004 re-enables the FEED HOLD button. For example:

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

The following is a map of variable #3004 bits and the associated overrides. E=Enabled D=Disabled

#3004	FEED HOLD	FEED RATE OVERRIDE	EXACT STOP CHECK
0	E	E	E
1	D	E	E
2	E	D	E
3	D	D	E
4	E	E	D
5	D	E	D
6	E	D	D
7	D	D	D

Programmable Stop

#3006 Stops can be programmed. A programmable stop acts like an M00. In the following example, when the assignment statement is executed, the first 15 characters of the comment are displayed in the messaging area on the lower left part of the screen above the command input line. The control stops and waits for a cycle start from the operator. Upon cycle start, operation continues with the next block after the assignment statement.

```
IF [#1 EQ #0] THEN #3006=101(ARG.A REQUIRED);
```

Last Block (MODAL) Group Codes

#4001-#4021 The grouping of G codes permits more efficient processing. G codes with similar functions are usually under the same group. For instance, G90 and G91 are under group 3. Variables have been set aside to store the last or default G code issued for any of 21 groups. By reading the group code, a macro program can change its behavior based on the contents of the group code. If 4003 contains 91, then a macro program could determine that all moves should be incremental rather than absolute. There is no associated variable for group zero, group zero G codes are NON-modal.

Last Block (MODAL) Address Data

#4101-#4126 Address codes A..Z (excluding G) are also maintained as modal values. The modal information represented by the last block interpreted by the lookahead process is contained in variables 4101 through 4126. The numeric mapping of variable numbers to alphabetic addresses corresponds to the mapping under alphabetic addresses. For instance, the value of the previously interpreted **D** address is found in #4107 and the last interpreted **I** value is #4104.

Last Target Position

#5001-#5005 The final programmed point, target position, for the most recent motion block can be accessed through variables #5001-#5005, X, Y, Z, A, and B, respectively. Values are given in the current work coordinate system and can be used while the machine is in motion.

Current Machine Coord Position

#5021-#5025 The current position in machine coordinates can be obtained through #5021-#5025, X, Y, Z, A, and B, respectively. The values CANNOT be read while the machine is in motion. #5023 (Z) represents the value after tool length compensation has been applied.

Current Work Coord Position

#5041-#5045 The current position in the current work coordinates can be obtained through #5041-5045, X, Y, Z, A, and B, respectively. The values can NOT be read while the machine is in motion. #5043 (Z) represents the value after tool length compensation has been applied.

Current Skip Signal Position

#5061-#5065 The position where the last skip signal was triggered can be obtained through #5061-#5065, X, Y, Z, A, and B, respectively. Values are given in the current work coordinate system and can be used while the machine is in motion. #5063 (Z) represents the value after tool length compensation has been applied.

Tool Length Compensation

#5081-#5085 The current total tool length compensation that is being applied to the tool is returned. This includes tool length offset referenced by the current modal value set in H (#4008) plus the wear value.

Offsets

All tool work offsets can be read and set within a macro expression. This allows the programmer to preset coordinates to approximate locations, or to set coordinates to values based upon the results of skip signal locations and calculations. When any of the offsets are read, the interpretation lookahead queue is stopped until that block is executed.

#5201-#5205	G52 X, Y, Z, A, B OFFSET VALUES
#5221-#5225	G54 " " " " " " "
#5241-#5245	G55 " " " " " " "
#5261-#5265	G56 " " " " " " "
#5281-#5285	G57 " " " " " " "
#5301-#5305	G58 " " " " " " "
#5321-#5325	G59 " " " " " " "
#7001-#7005	G110 X, Y, Z, A, B OFFSET VALUES
" " " " " "	" " " " " " "
#7381-#7385	G129 X, Y, Z, A, B OFFSET VALUES

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.

MACRO STATEMENTS

Macro statements are lines of code that allow the programmer to manipulate the control with features similar to any standard programming language. Included are functions, operators, conditional and arithmetic expressions, assignment statements, and control statements.

Functions and operators are used in expressions to modify variables or values. The operators are essential to expressions while functions make the programmer's job easier.

Functions

Functions are built-in routines that the programmer has available to use. All functions have the form **<function_name>[argument]**. Functions can be passed any expression as arguments. Functions return floating point decimal values. The function provided with the HAAS control are as follows:

FUNCTION	ARGUMENT	RETURNS	NOTES
SIN[]	Degrees	Decimal	Sine
COS[]	Degrees	Decimal	Cosine
TAN[]	Degrees	Decimal	Tangent
ATAN[]	Decimal	Degrees	Arctangent Same as FANUC ATAN[]/[1]
SQRT[]	Decimal	Decimal	Square root
ABS[]	Decimal	Decimal	Absolute value
ROUND[]	Decimal	Decimal	Round off a decimal
FIX[]	Decimal	Integer	Truncate fraction
ACOS[]	Degrees	Decimal	Arccosine
ASIN[]	Degrees	Decimal	Arcsine
#[]	Integer	Integer	Variable Indirection
DPRNT[]	ASCII text		External Output



Notes on Functions

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

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

When round is used in an address expression, then the argument of round is rounded to the addresses significant precision. For *metric* and *angle* dimensions, three-place precision is the default. For *inch*, four-place precision is the default. Integral addresses such as D, T and H are rounded normally.

```
#1= 1.00333 ;
G0 X[ #1 + #1 ] ;
      (Table moves to 2.0067) ;
G0 X[ ROUND[ #1 ] + ROUND[ #1 ] ] ;
      (Table moves to 2.0066) ;
G0 A[ #1 + #1 ] ;
      (Axis moves to 2.007) ;
G0 A[ ROUND[ #1 ] + ROUND[ #1 ] ] ;
      (Axis moves to 2.006) ;
D[1.67] (Diameter 2 is made current) ;
```

Operators

Operators can be classified into three categories: Arithmetic operators, Logical operators and Boolean operators.

Arithmetic Operators

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

+	- Unary plus	+1.23
-	- Unary minus	-[COS[30]]
+	- Binary addition	#1=#1+5
-	- Binary subtraction	#1=#1-1
*	- Multiplication	#1=#2*#3
/	- Division	#1=#2/4
MOD	- Remainder	#1=27 MOD 20 (#1 contains 7)

Logical Operators

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

OR - logically OR two values together
 XOR - Exclusively OR two values together
 AND - Logically AND two values together

Examples:

```
#1=1.0; 0000 0001
#2=2.0; 0000 0010
#3=#1 OR #2    0000 0011
```

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

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

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

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 four 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
```

Explanation

Jump to block 100 if value in variable #1 equals 0.0.
 While variable #101 is less than 10 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 jumps to block 1.

Expressions

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

Conditional Expressions

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

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

```
N50 M99 P10 ;
```

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

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

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

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

Arithmetic Expressions

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

Examples of Arithmetic expressions: #101=#145*#30;

```
#1=#1+1;  
X[#105+COS[#101]];  
#[#2000+#13]=0;
```

Assignment Statements

Assignment statements allow the programmer to modify variables. The format of the assignment statement is:

<expression>=<expression>

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

O0300	(Initialize an array of variables) ;
N1 IF [#2 NE #0] GOTO2	(B=base variable) ;
#3000=1	(BASE VARIABLE NOT GIVEN) ;
N2 IF [#19 NE #0] GOTO3	(S=size of array);
#3000=2	(SIZE OF ARRAY NOT GIVEN) ;
N3 WHILE [#19 GT 0] DO1 ;	
#19=#19-1	(DECREMENT COUNT) ;
#[#2+#19]=#22	(V=value to set array to) ;
END1 ;	
M99 ;	

The above macro could be used to initialize three sets of variables as follows:

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

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

Control Statements

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

Unconditional Branch (GOTO_{nnn} and M99 P_{nnnn})

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

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

Computed branching allows the program to transfer control to another block in the same subprogram. The block can be computed on the fly, as in the case of the GOTO[expression] form, or the block can be passed in through a local variable, as in the GOTO#_n form.

The GOTO will round the variable or expression result that is associated with the Computed branch. For instance, if #1 contains 4.49 and GOTO#1 is executed, the control will attempt to transfer to a block containing N4. If #1 contains 4.5, then execution will transfer to a block containing N5.



The following code skeleton could be developed to make a program that serializes parts:

```
O9200          (Engrave digit at current location.)
;
(D=DECIMAL DIGIT TO ENGRAVE) ;
IF [[#7 NE #0] AND [#7 GE 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 branch processing based on the results of reading hardware inputs. 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 GOTO)

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

IF [<conditional expression>] GOTO

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>;

Note: To preserve compatibility with FANUC syntax "THEN" may not be used with GOTO.

This format is traditionally used for conditional assignment statements such as:

```
IF [#590 GT 100] THEN #590=0.0 ;
```

Here, variable #590 is set to zero when the value of #590 exceeds 100.0. In the HAAS control, if a conditional evaluates to FALSE (0.0), then the remainder of the IF block is ignored. This means that control statements can also be conditioned so that we could write something like:

```
IF [#1 NE #0] THEN G1 X#24 Y#26 F#9 ;
```

This executes a linear motion only if variable #1 has been assigned a value. You might try something like this:

```
IF [#1 GE 180] THEN #101=0.0 M99 ;
```

This says that if variable #1 (address A) is greater than or equal to 180, then set variable #101 to zero and return from the subroutine.

Here is an example of an IF statement that branches if a variable has been initialized to contain any value. Otherwise, processing will continue and an alarm will be generated. Remember, when an alarm is generated, program execution is halted.

```
N1 IF [#9NE#0] GOTO3 (TEST FOR VALUE IN F) ;  
N2 #3000=11(NO FEED RATE) ;  
N3 (CONTINUE) ;
```

Iteration/Looping (WHILE DO END)

Essential to all programming languages is the ability to execute a sequence of statements a given number of times or to loop through a sequence of statements until a condition is met. Traditional G coding allows this with the use of the L address. A subroutine can be executed any number of times by using the L address.

```
M98 P2000 L5 ;
```

This is limited since you can not terminate execution of the subroutine on condition. Macros allows more flexibility with the WHILE-DO-END construct. The syntax is as follows:

```
WHILE [<conditional expression>] DOn ;  
<statements> ;  
ENDn ;
```

This executes the statements between DOn and ENDn as long as the conditional expression evaluates to TRUE. The brackets in the expression are necessary. If the expression evaluates to FALSE, then the block after ENDn is executed next. WHILE can be abbreviated to WH. The DOn-ENDn portion of the statement is a matched pair. The value of n is 1..3. This means that there can be no more than three nested loops per subroutine. A nest is basically a loop within a loop. A good example of how nesting of WHILE loops can be used is in defining a matrix.

```
#101= 3 ;
#102= 4 ;
G0 X#101 Y4. ;
F2.5 ;
WH [ #101 GT 0 ] DO1 ;
  #102= 4 ;
  WH [ #102 GT 0 ] DO2 ;
    G81 X#101 Y#102 Z-0.5 ;
    #102= #102 - 1 ;
    END2 ;
  #101= #101 - 1 ;
  END1 ;
;
M30 ;
```

This program drills a 3 x 4 matrix hole pattern.

Although nesting of WHILE statements can only be nested to three levels, there really is no limit since each subroutine can have up to three levels of nesting. If there ever is a need to nest to a level greater than 3, then the segment containing the three lowest levels of nesting can be made into a subroutine thus overcoming the limitation.

If two separate WHILE loops are in a subroutine, they can use the same nesting index. For example:

```
#3001=0 (WAIT 500 MILLISECONDS) ;
WH [#3001 LT 500] DO1 ;
  END1 ;
  <other statements>
```

```
#3001=0 (WAIT 300 MILLISECONDS) ;
WH [#3001 LT 300] 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 ] DO1 ;
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).

COMMUNICATION WITH EXTERNAL DEVICES - DPRNT[]

Macros allow additional capabilities to communicate with peripheral devices. One can do digitizing of parts, provide runtime inspection reports, or synchronize controls with user provided devices. The commands provided for this are POPEN, DPRNT[] and PCLOS.

Communication preparatory commands

POPEN and PCLOS are not required on the HAAS mill. It has been included so that programs from different controls can be ported to the HAAS. On some controls POPEN is required prior to using a DPRNT statement. POPEN prepares the device on the serial port by sending it a DC2 code. PCLOS terminates communication with external devices by sending it a DC4 code.

Formatted output

The DPRNT statement allows the programmer to send formatted text to the serial port. Any text and any variable can be printed to the serial port. Variables can be formatted. The form of the DPRNT statement is as follows:

```
DPRNT[ <text> <#nnnn[wf]>... ] ;
```

DPRNT must be the only command in the block. In the above, <text> is any character from A to Z or the letters (+, -, /, *, and the space). When an asterisk is output, it is converted to a space. The <#nnnn[wf]> is a variable followed by a format. The variable number can be any legal macro variable. The format [wf] is required and consists of two digits within square brackets. Remember that macro variables are real numbers with a whole part and a fractional part. The first digit in the format designates the total places reserved in the output for the whole part. The second digit designates the total places reserved for the fractional part. The total places reserved for output cannot be equal to zero or greater than eight. Thus the following formats are illegal:

```
[00] [54] [45] [36] /* not legal formats */
```

A decimal point is printed out between the whole part and the fractional part. The fractional part is rounded to the least significant place. When zero places are reserved for the fractional part, then no decimal point is printed out. Trailing zeros are printed as necessary if there is a fractional part. At least one place is reserved for the whole part, even when a zero is used there. If the value of the whole part has fewer digits than have been reserved, then leading spaces are output. If the value of the whole part has more digits than has been reserved, then the field is expanded so that these numbers are printed.

A carriage return is sent out after every DPRNT block.

DPRNT[] Examples

	Code	Output
N1	#1= 1.5436 ;	
N2	DPRNT[X#1[44]*Z#1[03]*T#1[40]] ;	X1.5436 Z 1.544 T 1
N3	DPRNT[***MEASURED*INSIDE*DIAMETER***] ;	MEASURED INSIDE DIAMETER
N4	DPRNT[] ;	(no text, only a carriage return)
N5	#1=123.456789 ;	
N6	DPRNT[X-#1[25]] ;	X-123.45679 ;

EXECUTION

DPRNT statements are executed at block interpretation time. This means that the programmer must be careful about where the DPRNT statements appear in the program, particularly if the intent is to print out 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.

OPERATION NOTES

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

Macro variables can be saved and restored to RS-232 or the optional floppy, much like parameters, settings, and offsets. Refer to the "Part Program Input / Output" section for RS-232 sending and receiving of macro variables, or the "Floppy Operation" section for sending and receiving them with this method.

Variable Display Page

The macro variables are displayed and can be modified through the current commands display. The variable display is located after the operation timers display. To get to this page, press CURNT COMDS and use the page up/down key.

As the control interprets a program, the variable changes are displayed on the variable display page and results can be viewed.

Pages contain up to 32 variables and the display can be "paged" by pressing the left/right arrow keys.

Setting of a variable is accomplished by entering a value and then pressing the WRITE key. The variable that is highlighted on the screen is the variable that is affected.

Searching for a variable can be done by entering the variable number and pressing the up/down arrow. The page will change to the one that contains that variable and the entered variable will become the highlighted item.

The variables displayed represent the values of the variables at program interpretation time. At times, this may be up to 15 blocks ahead of the actual machine activity. Debugging of programs can be made easier by inserting a G103 at the beginning of a program to limit block buffering and then removing the G103 block after debugging is completed.

Editing

For the most part, the editing of macro programs from the control is the same as before. There are a few peculiarities to be aware of.

Editing macro statements is more open than previously. For instance, it is possible to place a floating point constant within a standard G-code block, but it doesn't make much sense, and the control will raise an alarm at runtime. For all instances of improperly structured or improperly placed macro statements, the control will raise an appropriate alarm. Most of these alarms have been put off until runtime so that operator editing can be more flexible. Be careful when editing expressions. Brackets must be balanced and you will not receive an alarm until runtime.

The DPRNT[] function can be edited much like a comment. You can delete it or move it as a whole item, or you can edit individual items within the brackets. Variable references and format expressions must be altered as a whole entity. If you wanted to change [24] to [44], place the cursor so that [24] is highlighted, enter [44] and press the write key. Remember, you can use the crank handle to maneuver through long

DPRNT[] expressions.

Addresses with expressions can be somewhat confusing. In this case, the alphabetic address stands alone. For instance, the following block contains an address expression in X:

```
G1 G90 X [ COS[ 90 ] ] Y3.0 (CORRECT) ;
```

Here, the **X** and brackets stand alone and are individually editable items. It is possible, through editing, to delete the entire expression and replace it with a floating point constant.

```
G1 G90 X 0 Y3.0 (!!! WRONG !!!) ;
```

The above block will result in an alarm at runtime. The correct form looks as follows:

G1 G90 X0 Y3.0 (CORRECT) ;

Note that the zero is attached to **X**. REMEMBER when you see an alpha character standing alone it is an address expression.

FANUC-STYLE MACRO FEATURES NOT INCLUDED IN HAAS CNC CONTROL

This section lists the FANUC macro features that have not yet been implemented.

M ALIASING	REPLACE G65 Pnnnn WITH Mnn PROGS 9020-9029.
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
#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 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.

**11.4 FOURTH AXIS SETUP**

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 press the right arrow cursor until the model of the fourth axis is shown. Then, press 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.

WARNING !

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 a POWER UP/RESTART or AUTO ALL AXES. When turning the control on after disconnecting the fourth axis, be sure to turn off Setting 30 BEFORE doing a POWER UP/RESTART or AUTO ALL AXES.

Setting 34 is used to specify the diameter of the part being turned by the rotary axis. This parameter must be set close to the correct value in order to get the correct surface feed rate when an A-axis motion is involved in a cut.

If the fourth axis is being set up for the first time, you will not be able to enable the fourth axis from settings if Parameter 43 is set to disable the fourth axis. Make sure Parameter 43 is set correctly.

The tilting rotary table (TRT) is a little more complicated in its setup. One of the two axes may be the A-axis that can be controlled in simultaneous interpolated moves but the other axis must be setup as an auxiliary (**B**) axis and can only be used for positioning or single axis feeds; see "Auxiliary Axis Control" section. Depending on which axis may be used in cutting, you can exchange these two.

The following table summarizes the fourth axis options:

Product:	Max speed: (deg/sec)	Max torque: (ft - lb)	Comments:
HA5C	270	45	Quick and easy to use for light loads only.
HA5C2,HA5C3 HA5C4	200	20/ Spindle	HA5C2 fits along Y and X axes; HA5C3, HA5C4 only along X axis.
TRT 160	60/80 (A/B)	210/100 (A/B)	Convenient compact size fits VF-1,2.
TRT 210	50/60 (A/B)	200/210 (A/B)	Fits VF-3 and VF-4 with some restrictions in tool length.



HRT 160 HRT 160 SP	80	100	Available with 5" spindle nose. For light loads.
HRT 210 HRT 210 SP	60	210	Available with 6" spindle nose and pneumatic drawbar.
HRT 310 HRT 310 SP	50	200	Large capacity. Well-suited to VF-3 and 4.
HRT 450 HRT 600	50	325	For very large or heavy parts. Large through-hole. Ideal for VF-6,8.
T5C	60/270 (A/B)	30/160 (A/B)	Used for small, complex geometry parts.

11.5 AUXILIARY AXIS CONTROL

Besides the five directly controlled axes in this control, up to four additional external positioning axes may be added. These axes may be commanded directly from the program using the **C**, **U**, **V**, and **W** axis codes. Commands to these axes are only allowed in a G00 or G01 block. Connection of these axes is done through the second RS-232 port to one or more HAAS single axis controls. In the Setting page, Setting 38 is used to select the number of auxiliary axes. The machine position display will show the present position of these axes.

Only one auxiliary axis is moved at a time. If a feed (G01) is programmed, the feed rate programmed in the CNC is sent to the auxiliary control without any changes. For a C-axis feed at F30.0, this means that the C-axis will move at 30 degrees per second. A G00 motion of an auxiliary axis will move at its maximum feed rate.

The FEED HOLD and RESET buttons will not stop an auxiliary axis. EMERGENCY STOP and SINGLE BLOCK will stop an auxiliary axis. When the CNC control is waiting for an auxiliary axis motion to complete, the bottom of the screen will display "C FIN". A failure in RS-232 communication with the auxiliary axes may cause this display to pause indefinitely. The RESET button will terminate any "hung-up" auxiliary axis communication.

If you add one auxiliary axis, Setting 38, then the auxiliary axis will be designated **C**. If you add two auxiliary axes they will be designated **C** and **U**; etc.

When interfacing to an auxiliary axis the Haas single axis servo control must have Parameter 21 set according to the following table.

Name in CNC:	Parameter 21:	Axis select
C	6	Z
U	1	U
V	2	V
W	3	W

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



Auxiliary axes can be jogged from the CNC front panel using the jog handle. Each step must start and stop before another one can be commanded.

There are no work offsets for these axes so all commands are in the machine coordinate system. But if a displaced zero position has been entered into the HAAS servo control, that position will be used as zero. On power-up of the CNC, the auxiliary axes control will also be initialized and zero will be shifted by the value set into the single axis control. To set a displaced zero, you must jog the single axis control to a new zero position and then press and hold the CLEAR key on the single axis control. This must only be done when the single axis control is otherwise idle.

Auxiliary axes communication is always seven data bits, even parity, two stop bits. The data rate is CNC Setting 54 and should be set to 4800. CNC Setting 50 must be set to XON/XOFF. Parameter 26 in the single axis control must be set to 5 for 4800 bit per second and Parameter 33 must be set to 1 for XON/XOFF. Parameter 12 in the single axis control should always be set to 3 or 4 to prevent circular wraparound.

The cable connecting the CNC to the single-axis control must be a DB-25 cable (male lead on both ends) and must wire at least pins 1, 2, 3, and 7 directly from the second (lower) serial port of the CNC to the upper connector of the servo control.



11.6 FIFTH AXIS CONTROL

This section describes the features of a HAAS five-axis control.

THE COORDINATE SYSTEM

The layout of the **A** and **B** axes on the HAAS five-axis control are depicted below. The **A** axis is rotary motion about the **X** axis, while the **B** axis determines rotary motion about the **Y** axis. The right hand rule can be used to determine axis rotation for the **A** and **B** axes. When placing the thumb of the right hand along the positive **X** axis, the fingers of the right hand will point in the direction of tool movement for a positive **A** axis command. Likewise, when placing the thumb of the right hand along the positive **Y** axis, the fingers of the right hand will point in the direction of tool movement for a positive **B** axis command. It is important to remember that the right hand rule determines direction of tool movement and not the table movement direction. For the right hand rule, the fingers will point opposite of the positive rotary table movement. Refer to Figures 11-14 and 11-15.

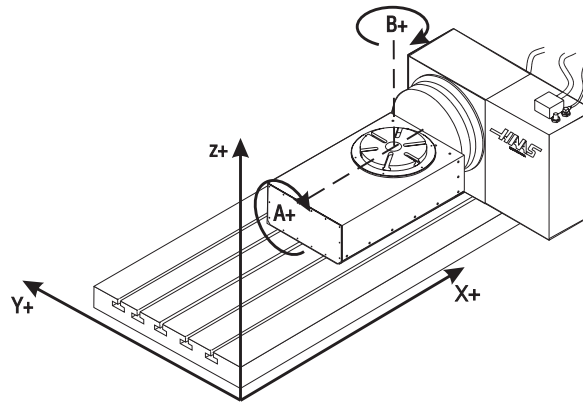


Fig. 11-14. Work coordinates (Positive direction).

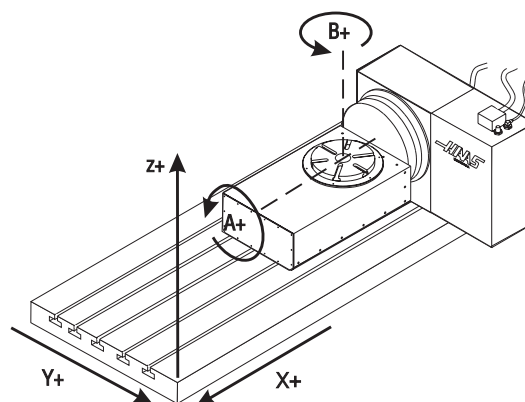


Fig. 11-15. Table movement (Positive command).

Note: Figures 11-14 and 11-15 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.

**ALPHABETICAL ADDRESS CODES****B Fifth axis rotary motion**

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

PREPARATORY FUNCTIONS (G CODES)

In a control configured with a fifth axis, all G codes that have an option for an **A** axis motion command can also simultaneously command fifth axis, **B**, motion. Of course, since address **B** is modal, it can be entered on any line. The following G codes are users of address B:

G00	G03	G29	G73	G77	G83	G86	G89	G101
G01	G10	G31	G74	G81	G84	G87	G92	G102
G02	G28	G36	G76	G82	G85	G88	G100	G136

INCH/METRIC SELECTION (G20,G21)

Fifth-axis programming is not effected by the selection of metric. It is always programmed in degrees.

WORK COORDINATE SYSTEMS

The work coordinate systems on a control with a fifth axis have all been expanded to accommodate **B**, the fifth axis. Work coordinate offsets can be set for the **B** axis in the offset display.

M CODES**M12 Engages the fifth axis brake.**

This M code waits until the M-fin signal is received through the secondary RS-232 port.

M13 Releases the fifth axis brake.

This code activates the fifth axis brake relay and waits for the M-fin signal, releases the relay upon receipt, and waits for the M-fin signal to cease. The RESET key will terminate any operation that is hung-up waiting for M-fin.

Both M12 and M13 are aliased by M10 and M11 on fourth-axis machines.



HANDLE/JOG

All aspects of handle jogging for the fifth axis work as they do for the other axes. The exception is the method of selecting jog between axis **A** and axis **B**.

By default the '+A' and '-A' keys, when pressed, will select the **A** axis for jogging. The display will show "JOGGING AAXIS 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.

SETTINGS

The fifth axis, or rotary table, may be disabled from the setting page and removed from the machine. Do not connect or disconnect any cables with the control on. If you do not disable the fifth axis when it is disconnected, you will get an alarm.

The settings listed here are settings that have been added to a fifth-axis control.

<u>Number:</u>	<u>Name:</u>	<u>Range of value:</u>
78	FIFTH-AXIS ENABLE	OFF,S5C,7RT,9RT,SRT,11RT, HA5C, HRT 160, HRT 210, HRT 310
79	FIFTH-AXIS DIAMETER	0 to 29.9999 inches
80	MIRROR IMAGE B AXIS	ON or OFF

The following is a detailed description of each of the settings.

78 FIFTH-AXIS ENABLE

This is an on/off setting. When it is off, the fifth axis is disabled and no commands can be sent to that axis. When it is on, the selected rotary table type parameters are called up. A change to rotary parameters is saved under the selected table type for later recall.

79 FIFTH-AXIS DIAMETER

This is a numeric entry. It is used to set the angular feed rate of the **B**-axis. It must be in the range of 0.0 to 29.9999 inches. Since the feed rate specified in a program is always inches per minute (or mm per minute), the control must know the diameter of the part being worked in the **B**-axis in order to compute the angular feed rate. When this setting is set correctly, the surface feed rate on a rotary cut will be exactly the feed rate programmed into the control. The feed rate will be correct only as long as the axis remains orthogonal (at right angles to) to the Y axis.

80 MIRROR IMAGE B-AXIS

This is an On/Off settings. When it is off, axes motions occur normally. When it is on, the B axis motion is mirrored (or reversed) around the work zero point.

**TOOL OFFSETS**

Work offsets can be specified for the fifth axis **B** address.

The work offset display is found on the offset display page. If the fifth axis is enabled, setting 78, then additional data fields are made available for the fifth axis.

The function of the offset display page has been modified slightly to accommodate multiple axes. Only the axes that are enabled are displayed on the work offset display. For example, if the fourth axis is enabled, setting 30, and the fifth axis is not, then display will show X,Y,Z,A. In this case, all axes can be displayed on the entire display. If only the fifth axis is enabled, then the display will show X,Y,Z,B. Since the fourth axis is disabled, there is no need to display it.

If both **A** and **B** axes are enabled, then the last column of the offsets display serves a dual purpose. Either **A** or **B** can be accessed in the last column. By using the left or right arrow keys, the screen cursor can be moved into the axis field that is to be set. When the right most field is highlighted by the cursor and the top of the column indicates A, then the values in this column represent **A** axis offsets. If you press the right arrow key once, the cursor stays in the same place, but the column and its values will change to **B** axis parameters. You can view and modify work offsets in this manner using the left and right arrow keys when both axes are enabled.



PARAMETER DISPLAY

The Parameter Display organization has been modified to accommodate the extra axis parameters. Parameter numbers have remained the same as in a four-axis control. Additional parameters have been added for the fifth.

The parameters have been reorganized, so that logically-associated parameters are grouped together. These logical groupings are placed together into contiguous screens called pages. The most commonly changed parameters have been placed at the beginning of the page list. A list of the parameter pages and the order of succession in the control are given below.

PAGE TITLE	DATA DESCRIPTION
COMMON SWITCH	Non-axis bit switches.
COMMON PAGE1	First page of non-axis parameters.
COMMON PAGE2	Second page of non-axis parameters.
COMMON PAGE3	Third page of non-axis parameters.
MACRO M CALL	Parameters that alias M codes to subroutines.
MACRO G CALL	Parameters that alias G codes to macros.
X BIT SWITCH	Bit switches for the X axis.
X PARAMETERA	First page of X axis parameters.
X PARAMETERB	Second page of X axis parameters.
Y BIT SWITCH	Bit switches for the Y axis.
Y PARAMETERA	First page of Y axis parameters.
Y PARAMETERB	Second page of Y axis parameters.
Z BIT SWITCH	Bit switches for the Z axis.
Z PARAMETERA	First page of Z axis parameters.
Z PARAMETERB	Second page of Z axis parameters.
A BIT SWITCH	Bit switches for the A axis.
A PARAMETERA	First page of A axis parameters.
B BIT SWITCH	Bit switches for the B axis.
B PARAMETERA	First page of B axis parameters.
B PARAMETERB	Second page of B axis parameters.
X SCREW COMP	X axis screw compensation value.
Y SCREW COMP	Y axis screw compensation value.
Z SCREW COMP	Z axis screw compensation value.

The HOME key displays the first parameter page "COMMON SWITCH". Pressing the PAGE DOWN key will display the next page of parameters in the above list. The END key displays the last parameter page "B PARAMETERB". Pressing the PAGE UP key will display the preceding page of parameters in the above list. All other features on the parameters display have remained the same. So, if you are unfamiliar with the new format of the parameters, you can still search by parameter number. Enter the number of the parameter you want to see or view and press the up or down arrow key. The page that the parameter is on will be displayed and the parameter being searched for will be highlighted.

**DIAGNOSTIC DATA DISPLAY**

The first page of diagnostic data has two discrete outputs that control the rotary axes brakes; they are labeled "4TH BK" and "5TH BK". When motion is commanded to either the **A** or **B** axis, the brake for that axis must first be released. To release a brake on the HAAS control, a relay is activated. These two outputs represent the activation of the brake relays. If the **A** or **B** axis brakes are disengaged, then these outputs will read high (logical 1). Normally these bits will read low (logical 0).

The second page of diagnostic data show the status of inputs from the motor interface board. Additional inputs for the expanded motor interface board are listed under "INPUTS4". These bits are monitored by the control to determine if the interface for the **B** axis is working correctly. Refer to the "Technical Reference" section for a description of the diagnostic page inputs.

PARAMETERS

Additional parameters specifying the fifth axis have been added to the parameter list **B** axis parameter numbers range from 151 through 175. The parameter descriptions and their number assignments are listed below in the order that they are arranged on the **B** parameter pages. Refer to the Parameters List in the "Operation" section of this manual for more information.

PARAMETERS LISTED HERE IN ORDER OF APPEARANCE ON DISPLAY PAGES.**B BIT SWITCH**

Parameter	151	B	SWITCHES
-----------	-----	---	----------

B PARAMETER A

Parameter	152	B	P GAIN
Parameter	153	B	D GAIN
Parameter	154	B	I GAIN
Parameter	155	B	RATIO (STEPS/INCH)
Parameter	156	B	MAX TRAVEL (STEPS)
Parameter	157	B	ACCELERATION
Parameter	158	B	MAX SPEED
Parameter	159	B	MAX ERROR
Parameter	160	B	FUSE LEVEL
Parameter	161	B	BACK EMF
Parameter	162	B	STEPS/REVOLUTION
Parameter	163	B	BACKLASH
Parameter	164	B	DEAD ZONE
Parameter	165	B	IN POSITION LIMIT B
Parameter	166	B	MAX CURRENT
Parameter	167	B	D*D GAIN FOR B

B PARAMETER B

Parameter	168	B	ACC/DEC T CONST
Parameter	169	B	PHASE OFFSET
Parameter	170	B	GRID OFFSET
Parameter	171	B	EXACT STOP DIST
Parameter	172	B	FRICTION FACTOR
Parameter	173	B	ACCEL FEED FORWARD
Parameter	175	B	AIR BRAKE DELAY



SETTING UP A FIFTH AXIS

Installing a fifth axis is the same as installing a fourth axis. Refer to the "Setting Up A Fourth Axis" section of this manual for detailed instructions on fifth axis installation.

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.

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 129.7 inches per minute in the following example.

EXPONENTIAL ACCELERATION

Exponential acceleration/deceleration is a type of motion where the speed is proportional to the distance remaining in a programmed travel. The exponential accel/decel time constant is set by Parameters 113, 114, 115, and 116. It has units of 0.0001 seconds. The speed limit at which exponential accel/decel is not available is defined by the relationship between Parameters 7 and 113 (for the X-axis). Thus, if Parameter 7 is 8000000 steps/sec/sec and Parameter 113 is 375 (0.0375 seconds); the maximum velocity for exponential accel/decel would be:

$$\begin{aligned} 8000000 \times 0.0375 &= 300000 \text{ steps/second} \\ \text{For a 8192 line encoder and 6 mm screw, this would be:} \\ 60 \times 300000 / 138718 &= 129.7 \text{ inches/minute} \end{aligned}$$

ACCELERATION IN FEED MOTIONS

In the normal feed cutting mode, with G64 active, giving continuous cutter motion, deceleration of the axes in motion begins at some distance away from the end point. If lookahead has buffered another motion, the acceleration for that motion will begin at the same instant. This means that two motions, at right angles to each other, will not produce a perfectly square corner. The corner will be rounded. In addition, two motions which smoothly blend one into the other will not cause the tool to pause.

If you use cutter compensation to cut an outside corner, there will be no rounding if the cutter compensation amount is close to the actual tool size. This is because the tool is moved beyond the end of the first programmed stroke before it is moved to the beginning of the second stroke. Note that in this machine, using the default parameter settings, rapid and feed moves will both be blended to provide continuous cutter path and rounded corners. Unless you specify exact stop, the following rapid or feed block will be started slightly before the completion of the previous block.



The end of a feed move is delayed until the following error is below an amount set in Setting 85. If Setting 85 is set to 0.1, with Parameter 113 set to 375 (0.0375 seconds), this means that the highest feed rate which will give continuous cutter motion is:

$$(0.1) * 60 / 0.0375 = 160 \text{ inches per minute}$$

ACCELERATION IN RAPID MOVES

Rapid moves have a slightly different operation when continuous cutter mode is active. Acceleration for the next motion is started when the axes being moved are all within the "In Position Limit" Parameters 101, 102, 103, and 104. These parameters have units of encoder steps. Rapid moves will also decelerate at the constant accel/decel limit until the speed drops below that for exponential accel/decel. An example of the "In Position Limit" values follows. If Parameter 101 (for X) is 8000 and Parameter 5 is 138718, a rapid move of will proceed to the next block when the X-axis is within a distance of:

$$8000 / 138718 = 0.0577 \text{ inches}$$

To prevent the rounding of corners, you can specify exact stop either with G09 (non-modal) or with G61 (modal). When either of these is active in a motion, all of the axes are brought to an exact stop, at zero speed, before the next motion is started.

Note that in this machine, using the default parameter settings, rapid and feed moves will both be blended to provide continuous cutter path and rounded corners. Unless you specify exact stop, the following rapid or feed block will be started slightly before the completion of the previous block.



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 129.7 inches per minute). However, the actual radius of a circular move will always be slightly smaller than the programmed value. The amount of change can be computed by the following equation:

$$Ra = \text{SQRT}(R * R - L * L)$$

Where **Ra** is the actual radius,
R is the programmed radius, and
L is the accel/decel lag in feed motion.

The lag amount is computed by:

$$L = (\text{Par. 113}) * (\text{feed in/min}) / 600000$$

As an example; if Par 113 is 375 (0.0375 sec) and the feed is 30 inches per minute and the programmed diameter is two inches, the actual radius will be:

$$L = 375 * 30 / 600000 = 0.0187 \text{ inches}$$

and

$$Ra = \text{SQRT}(1 - 0.000351) = 0.999824$$

or an error of 176 millionth's of an inch. This is an upper bound on the accuracy of this cut and many other factors could contribute additional errors.

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.



11.7 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. 600 blocks per second is standard and 1000 blocks per second is a purchased option. No special programming or setup is required to get these speeds.

There are a few points to consider at high feed rates.

- 1) At higher feed rates interpolation accuracy will be reduced.
- 2) The higher the resolution you specify on your CAD/CAM the longer the program and the slower the speed
- 3) When using DNC you must have a very fast computer and excellent DNC software.

In most CNC controls you have to set the maximum feedrate according to the most accurate detail you were machining. Just as in a car you cannot make a right turn at 100 MPH so to the CNC control must slow down in the corners in order to give the accuracy you require. Unfortunately the corners often dictate the maximum feed rate for the entire program when that may not be desired. The HAAS control uses a special function called Contouring Accuracy (G187) which allows you to specify the required accuracy. If the programmed feedrate is above what the control can accurately machine the control will automatically slow the feedrate to obtain the required accuracy. In straight line moves the feed rate will be at the programmed rate. See Contouring accuracy in the next section.

When machining a sculptured mold you obviously would like to have the smoothest surface possible. The higher the resolution you select (the distance between machined points) the more accurate the final part. The downside to this is that program length is longer and feedrates slower. Beware of the relationship between stroke length, feed rate, and block per second. If you have 1000 blocks per second and each block is 0.001 inches in length, this is 1 inch per second and 60 inches per minute. Trying to program more than 60 inches per minute in this case will not go any faster; you cannot get more than 1000 blocks per second. Similarly a resolution of .0001 will limit your feedrate to 6 IPM.

The best way to machine a part is from CNC memory. The HAAS control can be purchased with up to 8 megabytes of ram. This eliminates any communications problems and offers the highest blocks per second possible. If you choose to run the HAAS control directly from your computer, you must ensure that your computer can send data fast enough. A general purpose Windows based communication program will not work reliably. The serial port and wiring from your computer must be of the highest quality. Beware, that when running DNC, computer serial ports are not able to send data to the CNC fast enough to get the blocks per second rate which you require. The fastest serial rate of this control is 115200 bits per second. This translates to 115200 characters per second. If a typical block consists of about 20 characters, it is not possible to run DNC at more than 575 blocks per second. If you are not experienced with high-speed DNC get expert help from your dealer or have him recommend someone local with hands on experience.

In order to get maximum performance from large memory programs the following hints will help:

- 1) Eliminate all unnecessary codes such as line numbers, comments, redundant G codes, etc.
- 2) Eliminate all leading and trailing zeros, i.e. make X002.0000 to X2.
- 3) Incremental (G91) programming utilizes the least data i.e., (G91 X1 Y1 Vs G90 X2.0001 Y2.0001)



CONTOURING ACCURACY

When milling a part it is desirable to machine it at the fastest feedrate possible and yet obtain the highest accuracy. No machine can instantly accelerate or decelerate and this leads to positioning errors when axis directions change. The higher the feedrate and the sharper the turn the more pronounced the effect. A high feedrate selected for straight line milling will cause corner rounding at intersections when you actually want a sharp corner. Similarly a high feed milling cut blending into a radius will cause the radius to be smaller than programmed because of axis acceleration distance. To alleviate this the HAAS control uses a special function called Contouring Accuracy (G187) so the operator can select the required accuracy.

When using this function the control will move up to the programmed feedrate in straight line moves and slow down at intersections or radii to obtain the required accuracy you want. This accuracy is measured in true three dimensional motion. It uses setting 85 to define a default value and G187 to program a new value directly from your program. The amount of slow down depends on the accuracy specified and on how well one stroke blends into the next. If two strokes blend into each other exactly (in one line), there is never any slow down.

Programming G187 is as follows:

G187 E0.01	(to set value)
G187	(to revert to setting 85 value)

The first line will set the required accuracy to 0.01 inches. G187 must be programmed on a line by itself. If there is no E code, the accuracy reverts to setting 85. If MM programming is active, the units are millimeters. The range of values possible are 0.0001 to 0.25 inches and 0.001 to 2.5 mm.

The most important thing to remember with Contouring Accuracy is that normal, well blended strokes should not get a slowdown in feedrate. Only the sharp corners need this in order to achieve the programmed accuracy. However, if you set the accuracy to extremely small values (0.0005 and smaller) you may effect such a slow feedrate that the machine appears to pause when you don't think it should. Even well blended strokes can have a slight error.

When roughing out a pocket the default setting (85) should be used for maximum speed then on the final clean up passes you can specify a higher accuracy. A simple test part of milling a rectangle and varying the values for Contouring Accuracy will demonstrate the principles outlined above.



11.8 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.

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.

USING FLOATING TAP HOLDERS

Floating tap holders are probably the most common method of tapping holes. The tap is held in a quick change holder that can float up and down slightly. This is done to allow the tap to follow the hole it is tapping and compensate for differences in the accel and decel of the spindle versus the feed of the Z-axis. Upon reaching the bottom of the hole, the feed stops and the spindle reverses, if you watch closely, you will see the tap pull the floating holder out slightly. Upon reversal, the tap will be pushed back into the holder.

If the holder is pulled out or pushed in to its mechanical limits while tapping, you can break the tap, damage the threaded part, or pull the tap completely out of the holder. Carefully watch for this condition when setting up a job because it usually becomes a problem after the job has been running a while. Also tapping of diameters less than 5/16 of an inch while below 1201 rpm (low gear shift point par 142) should be done in high gear. Spindle reversal is quicker in high gear and will minimize tap pullout. This is done by putting an M42 code with the speed command such as: M42 S900. Tapping is done with G74 and G84 cycles which automatically reverse the spindle at **Z** depth. The feed rate can be calculated by using the HELP display and paging down to the tapping calculator and inputting your speed and tap pitch into the control to obtain your feed rate that is then input to the **F** command of the cycle. Example:

N100 G84 Z-1.0 R.3 F46.875 (for 32 pitch tap at 1500 RPM)



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

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)



11.9 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 by the CNC program.

OPERATING THE COOLANT SPIGOT

When the spigot is enabled it will 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.

OFFSET DISPLAY WITH SPIGOT ENABLE

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.

SPIGOT PROGRAM COMMANDS

There are two ways that the spigot can be moved under program control. The first, as just discussed, is by entering positional values into the CLNT POS fields on the tool offset display page. Having positional values entered into the CLNT POS field does not mean that the spigot will move. Only when the M08 command is executed and the current H code has a value in its CLNT POS field, will the spigot move to the designated position.

The second method of moving the spigot is by programming M34 or M35.

M34 moves the spigot in a positive direction. If the spigot is at the end of travel then no spigot movement occurs. For example, if the current spigot position is at 8 and M34 is executed, then the spigot will move to position 9. On a vertical mill this would lower the coolant stream.

M35 moves the spigot in a negative direction. If the spigot is at home position then no spigot movement occurs. For example, if the current spigot position is at 8 and M35 is executed, then the spigot will move to position 7. On a vertical mill this would raise the coolant stream.

It is important to note that each programming method requires the operator to specifically program the spigot in reference to the particular tool being used, while also taking into consideration the tool length, width and size of the part.



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

SPIGOT M CODES

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.



11.10 AUTOMATIC CHIP CONVEYOR OPERATION

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. If there is no axis motion or keyboard action within the time set in Parameter 255, the conveyor will automatically shut off.

Note: It is recommended that the chip conveyor be used intermittently. Continuous operation will cause the motor to overheat.

CONVEYOR KEYBOARD COMMANDS

The conveyor can be started at any time from the keyboard. The conveyor can be enabled in either direction by pressing the CHIP FWD or CHIP REV and stopped by pressing the CHIP STOP key. The conveyor will also stop by pressing the RESET key.

CONVEYOR PROGRAM COMMANDS

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

CONVEYOR PARAMETERS

The parameters that control the conveyor are below.

CNVYR RELAY DELAY	Parameter 216	CNVYR IGNORE OC TIM	Parameter 217
CNVYR RETRY REV TIM	Parameter 218	CNVYR RETRY LIMIT	Parameter 219
CNVYR RETRY TIMEOUT	Parameter 220	CONVEYOR TIMEOUT	Parameter 255

A complete description of conveyor parameters is given in the "Parameters" section of this manual.

CONVEYOR M CODES

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.



11.11 THROUGH THE SPINDLE COOLANT (TSC) - OPTION

The HAAS Through The Spindle Coolant (TSC) option includes an auxiliary coolant pump that is used to supply high pressure coolant to the cutting tool.

When TSC is commanded (M88), the precharge is activated and the tool release piston moves down to engage the drawbar, forming a seal. For 50 taper TSC, coolant or air pressure closes the seal, and it opens again when pressure is released.

NOTE: The wear surfaces of the seal are engaged only when TSC is in use.

The auxiliary pump turns on and begins sending coolant to the spindle. The control waits for coolant pressure to remain stable for 0.1 seconds before proceeding. If coolant pressure does not come on within 60 seconds, the system shuts down and gives Alarm 151 (Low Tool Coolant).

When the coolant system is turned off (M89), the spindle is stopped, the pump is shut off, and air flows through the spindle and the TSC drain line for 2-1/2 seconds to purge leftover coolant.

Refer to the "Maintenance" section for TSC system maintenance information.

OPERATION

GENERAL WARNINGS

The TSC pump is a precision gear pump and will wear out faster and lose pressure if abrasive particles are present in the coolant.

Shortened pump life, reduction of pressure and increased maintenance are normal and to be expected in abrasive environments and are not covered by warranty.

When machining castings, sand from the casting process and the abrasive properties of cast aluminum and cast iron will shorten pump life unless a special filter is used in addition to the 100 mesh suction filter. Contact HAAS for recommendations.

Machining of ceramics and the like voids all warranty claims for wear and is done entirely at customer's risk. Increased maintenance schedules are absolutely required with abrasive swarf. The coolant must be changed more often and the tank thoroughly cleaned of sediment on the bottom. An auxiliary coolant tank is recommended.

- Proper tooling, with a through-hole, must be in place before using the TSC system. **Failure to use proper tooling will flood the spindle head with coolant and void the warranty.**
- Use a pull stud with "45 Degree, P40T Type 1, inch threads" built to JMTBA standard "MAS 403-1982". If the machine is equipped with the optional BT tool changer, use BT tooling only. Contact the tool manufacturer for further information. **Pull studs are available through HAAS. Refer to the Technical Reference section of the manual for the proper tool part numbers and identification.**



- Coolant will be used more quickly when the TSC system is in use. Make sure to keep the coolant level up and to check the level more frequently (check after every eight hour shift). **Premature wear of the pump can result from running with a low coolant level in the tank.** The spindle will shut off automatically if the coolant level gets too low.
- The maximum spindle speed when using the TSC system is 7500 RPM.

Note: Follow the tool manufacturer's instructions for speed and feed rate restrictions.

PROGRAM COMMANDS

Through The Spindle Coolant is controlled mainly by two M Codes, M88 and M89; and four Parameters, 235, 236, 237 and 238 (See the "Parameters" section for a description of these). M88 is used to turn the system on, and M89 is used to turn it off. The AUX CLNT button on the control panel may also be used to control TSC. Pressing this button while in MDI mode will turn on the Through The Spindle Coolant system, and pressing it again will shut off the system.

When Through The Spindle Coolant is on, the RESET and EMERGENCY STOP buttons on the control panel perform slightly different functions. When RESET is pressed, the feed and spindle are shut off, as in a normal reset. However, with TSC on, the control also waits for the spindle to stop, turns off the TSC coolant pump, turns on purge, waits for [Parameter 237] turns off purge, and then turns off the precharge. When EMERGENCY STOP is pressed, the feed, spindle, and pump are shut off, as in a normal E-Stop. However, with TSC turned on, the control will turn on purge air for the time specified in Parameter 237 so coolant can be purged, and then shut off the precharge.

TSC PARAMETERS

The following parameters (and bits) apply only to the Through The Spindle Coolant system:

WARNING!

These are factory preset parameters. Changing them may void the warranty.

Parameter 235	TSC PISTON SEAT
Parameter 236	TSC LOW PR FLT
Parameter 237	TSC CLNT LINE PURGE

This can be increased by the machine user if desired to help purge coolant from small orifice tooling. The minimum setting is 2500 milliseconds.

Parameter 238	MAX TSC SPINDLE RPM
Parameter 209	COMMON SWITCH 2

The bit, "TSC ENABLE" in **Parameter 209**, is set to "1".

Parameter 278	COMMON SWITCH 3
----------------------	------------------------

The bit, "TSC PRG ENBL" in **Parameter 278**, is set to "1".

A complete description of TSC parameters is given in the "Parameters" section of this manual.

**TSC M CODES**

The following M codes apply only to the Through The Spindle Coolant system:

M88 Thru Spindle CInt ON

M88 performs the following operations:

- Stop the spindle
- Turn on the TSC pump
- Wait for coolant pressure
- Restart spindle

M89 Thru Spindle CInt OFF

M89 performs the following operations:

- Stop the spindle
- Turn off the TSC pump
- Turn on purge
- Wait for [Parameter 237] for coolant to purge
- Turn off purge

The following pre-existing M codes perform a slightly different function when Through the Spindle Coolant is turned ON:

M00 Stop Prog**M01 Optional Stop**

When TSC is ON, M00 and M01 will shut it OFF, as in M89.

M06 Tool Change

When TSC is ON, M06 causes the following operations to be performed:

- Orient the spindle and move Z-axis to tool change position
- Turn OFF the TSC pump
- Turn on purge
- Wait for [Parameter 237] for coolant to purge
- Turn off purge
- Perform tool change

TSC then remains OFF until M88 is used.

M24 Option User M

When TSC is enabled, this M code is disabled.

M30 Prog End Rew

When TSC is ON, M30 will shut it OFF as in M89, then perform an M30 operation.



M54 **Set User M**

M64 **Clear User M**

When TSC is enabled, these M codes are disabled.

ALARM DESCRIPTION

151 **Low Tool Coolant**

This alarm will shut off the spindle, feed and pump all at once. It will turn on purge, wait for the time specified in parameter 237 for the coolant to purge, and then turn off the purge. If this alarm is received, check the coolant tank level, the filter and intake strainer for any clogging. If no problems are found with any of these, and none of the coolant lines are clogged or kinked, call you dealer.

198 **Precharge Failure**

This alarm is received if the precharge fails for greater than 0.1 seconds. It will shut off the feed, spindle and pump all at once. If received, check all air lines and the air supply pressure. Regulated air pressure above 85 psi may cause this alarm. **This alarm does not apply to 50 taper.**

SAMPLE PROGRAM

IMPORTANT!

Note that the M88 command appears before the spindle speed command in this program. This is a good programming practice because the M88 command will stop the spindle, start TSC, then restart the spindle, slowing the cycle time.

T1 M6;	(TSC Coolant Through Drill)
G90 G54 G00 X0 Y0;	
G43 H06 Z.5;	
M88;	(Turn TSC on)
S4400 M3;	
G81 Z-2.25 F44. R.03;	
M89 G80;	(Turn TSC off)
G91 G28 Z0;	
M30;	

**11.12 SIDE MOUNT TOOL CHANGERS 40 TAPER DISK AND 50 TAPER SERPENTINE****INSTALLATION****50 Taper Spindle Machines**

1. Remove all tools from the carousel and the spindle.
2. Jog the spindle head all the way up in the 'Z' direction.
3. Check which pocket number is currently in the Pocket Drop position.
4. At the control press 'MDI', and then press 'ATC FWD', when the pocket drops into the tool change position (Pocket down), press 'FEED HOLD'.
5. Verify whether the carousel moved in proper direction.
6. If the carousel has moved to the next higher number (e.g. 5 to 6), then the phasing is correct.
7. If the carousel has moved to the next lower number (e.g.. 5 to 4) then the phasing is incorrect and you must switch any of the two leads from the main power line in the control cabinet.

Caution! Make sure that the main breaker is set to OFF before switching the leads.

8. Finish ATC operation by pressing 'CYCLE START'.
9. Verify that the arm rotates into the spindle with open section of the arm facing the spindle.
10. Perform a single tool change with a tool, to verify proper operation.

40 Taper Spindle Machines

40 taper spindle machines use a DC motor to rotate the carousel and arm, therefore the motor is not sensitive to incorrect phasing.

**OPERATION**

These tool changers allow the control to rotate the carousel to the next tool needed while the previous tool is still in the spindle. When a tool change is commanded, the two tools are swapped by the tool arm. Thus, a tool will be taken from one pocket and returned to another. This means that carousel pockets are not dedicated to any particular tool.

Tool Pocket Set-up

To initialize a pocket to a particular tool, scroll to that pocket, enter the tool number and press ENTER. To initialize the pocket as empty, press 0, ENTER.

Pressing ORIGIN at any time while on this screen will set the tool pockets to default values. For example, tool 1 in the spindle, tool 2 in pocket 1, tool 3 in pocket 2, and so on. Thus, a machine with a 24 pocket carousel can handle 25 tools.

Pressing 0, ORIGIN at any time while on this screen will set the tool pockets to all zeros.

To designate a pocket as "large" (meaning that the pockets on either side will remain empty,) scroll the highlight to that pocket on the screen and press "L", WRITE. An "L" will appear on the screen next to this pocket, and the tool numbers on either side will be changed to "-". To remove the "L" and change the pockets on either side to zeros, press SPACE, WRITE.

If the carousel contains a mixture of large and normal size tools, tool changes will often result in normal size tools being placed in pockets designated as large. When this happens, the control will automatically change the upper case "L" to a lower case "l". When another tool change results in a large tool being placed in that pocket, the control will automatically change the "l" back to "L".

If the operator loads all the large pockets with large tools and loads another large tool in the spindle, it will not be possible for the control to change to a normal size tool unless one pocket is designated as "1", meaning a large pocket containing a normal size tool. To designate such a pocket, scroll to that pocket and enter "1", WRITE. A lower case "l" will appear beside this pocket. To clear it, enter SPACE, WRITE. Note that only one pocket can be designated by the operator with "1". If a second pocket is set to "1", the previous one will be automatically changed to "L".

If the spindle currently contains a large tool and a normal size tool is needed next, the control will have to swap the large tool with a normal size tool that is in a large pocket, and then (assuming it is not the right tool) swap that tool for the one that is needed. Thus, in such a case, two tool changes will be performed.

The areas most affected by this feature are the tool change and tool changer restore functions. There are a number of parameters and alarms that support these tool changers.

Note: If alarms occur while using a Vertical Mill side-mount tool changer, the associated parameters should first be verified (and corrected if necessary) then the tool change recovery procedure should be performed



Note: The tool pockets in these tool changers are very close together and will accommodate "normal" size tools. If "large" tools are needed (meaning that when placed in the pocket, they will extend beyond the space allowed) then the pocket on either side must remain empty. Pockets can be designed to hold large tools as described in the POCKET AND TOOL screen in the Offsets display.

Faults which are caused by the carousel will not generate an alarm until an M6 (tool change) is commanded.

POWER UP PROCEDURE

1. Press the Green power On button and wait for the control to complete the boot up process.
2. Press the Power Up/ Restart button and wait for the machine to perform a Zero Return.

Setting 81 TOOLAT POWERDOWN will cause one of the following actions to occur during power up:

- A. If Setting 81 TOOLAT POWERDOWN contains zero the carousel will be rotated to pocket #1. No tool change will be performed.
- B. If Setting 81 contains a number greater than zero, the tool is listed in the pocket and tool table, and the tool is already in the spindle. Then the carousel will remain at the same pocket and no tool change will be performed.
- C. If Setting 81 contains a number greater than zero, the tool is listed in the pocket and tool table, and the tool is **not** already in the spindle. Then the carousel will be rotated to pocket #1, then to the pocket containing the tool specified by Setting 81 and a tool change will be performed that results in that tool in the spindle.

**POWER DOWN PROCEDURE**

1. Assuming the machine has been zeroed since it was powered up, press the Power Up / Restart button and wait for the machine to perform a Zero Return.

Setting 81 TOOLAT POWERDOWN will cause one of the following actions to occur during power down:

- A. If Setting 81 TOOLAT POWERDOWN contains zero the carousel will remain at the same pocket. No tool change will be performed.
- B. If Setting 81 contains a number greater than zero, the tool is listed in the pocket and tool table, and the tool is already in the spindle. Then the carousel will remain at the same pocket and no tool change will be performed.
- C. If Setting 81 contains a number greater than zero, the tool is listed in the pocket and tool table, and the tool is **not** already in the spindle. Then the carousel will be rotated to pocket #1, then to the pocket containing the tool specified by Setting 81 and a tool change will be performed that results in that tool in the spindle.
- D. If Setting 81 contains a number greater than zero, but that tool is not listed in the pocket and tool table. Then the carousel will be rotated to pocket #1, and no tool change will be performed.

2. Press the red Power Off button.

TOOL CHANGE RECOVERY

If a problem occurred during a tool change, a tool change recovery needs to be performed. Enter tool change recovery mode by pressing 'TOOL CHANGER RESTORE'. Once in tool change recovery mode, the operator is given instructions and questions for a proper tool change recovery. The entire tool change recovery process must be completed before exiting. If an exit was done prematurely the tool change recovery must be started from the beginning.

The logic tree (flow chart) for the tool change restore function is at the end of this section. This logic tree shows the questions, that will be asked of the operator, and the result of the answers.

FEATURES**POCKET AND TOOL Screen**

This screen has been added to support the serpentine type 1 and Disk type 1 tool changers. When a side mount tool changer has been specified by parameter 278 bits 26, 27 and 28, it will appear in the Offsets display screens (press Page-Up or Page -Down to scroll to it.) It lists the tool carousel pockets and the tools that they each contain, and the tool that is currently in the spindle.



PARAMETERS

Parameter 278 bit 26 S MNT BIT 1

Bits 26,27, and 28 work together to enable the Vertical mill side- mount tool changers. See below.

Parameter 278 bit 27 S MNT BIT 2

Bits 26,27, and 28 work together to enable the Vertical mill side- mount tool changers. See below.

Parameter 278 bit 28 S MNT BIT 3

Bits 26,27, and 28 work together to enable the Vertical mill side- mount tool changers.

The following table shows the bit combinations that must be used:

Bit 28	27	26	Decimal value	Type
0	0	0	0	No side-mount tool changer installed
0	0	1	1	Serpentine 1
0	1	0	2	Serpentine 2
0	1	1	3	Serpentine 3
1	0	0	4	Disk 1
1	0	1	5	Disk 2
1	1	0	6	Disk 3
1	1	1	7	Disk 4

Parameter 60 TURRET START DELAY

On mills with a serpentine type 1 tool changer, this parameter is used to specify the time (in milliseconds) allowed to move the carousel to the high cam position. If the carousel does not move to its commanded position within the time allowed by this parameter, alarm 625 INVALID TC START CONDITION is generated.

On mills with the Disk type 1 tool changer, this parameter is used to specify the time (in milliseconds) allowed for the carousel to move off the TC mark. If the carousel has not moved within the time allowed by this parameter, alarm 625 INVALID TC START CONDITION is generated.

Parameter 61 TURRET STOP DELAY

On mills with a serpentine type 1 tool changer, this parameter is used to specify the time (in milliseconds) allowed to move the carousel from the start of the high cam position until the shot pin falls into place at the low cam position. If the carousel does not move to its commanded position within the time allowed by this parameter, alarm 625 INVALID TC START CONDITION is generated.

On mills with the Disk type 1 tool changer, this parameter is used to specify the time (in milliseconds) allowed for the carousel to move to the TC mark. If the carousel has not moved within the time allowed by this parameter, alarm 625 INVALID TC START CONDITION is generated.

Parameter 62 SHUTTLE START DELAY

On mills with a serpentine type 1 or Disk type 1 tool changer, this parameter is used to specify the time (in milliseconds) needed to allow the tool pocket to settle (stop bouncing) after being lowered in preparation for a tool change.

**Parameter 248 SMTCLY ON/OFF DLY**

On mills with a serpentine type 1 tool changer it specifies the time needed (in milliseconds) between deactivating the DIRECTION relay and the MOTOR ENABLE relay.

On mills with a Disk type 1 tool changer it specifies the time needed (in milliseconds) between turning off one relay and turning on the other one when reversing direction of the carousel.

For Vertical Mills without a serpentine type 1 or disk type 1 side- mount tool changer, this parameter should be set to 0.

Parameter 306 POCKET UP/DOWN DELAY

This parameter supports the serpentine type 1 and the Disk type 1 tool changers. It specifies the time allowed (in milliseconds) for the tool pocket to be raised or lowered. If the pocket does not move to its commanded position within the time allowed by this parameter and parameter 62, alarm 626 TOOL POCKET SLIDE ERROR is generated. For vertical mills without a serpentine type 1 or Disk type 1 side- mount tool changer, this parameter should be set to 0.

Parameter 307 POCK UN/LOCK DELAY

This parameter supports the serpentine type 1 tool changer. It specifies the time needed (in milliseconds) to lock or unlock a tool pocket. For vertical mills without a serpentine type 1 side- mount tool changer, this parameter should be set to 0.

Parameter 308 ARM ROTATE TIME

This parameter supports the serpentine type 1 and the Disk type 1 tool changers. It specifies the time allowed (in milliseconds) for the tool arm to rotate to the next position. The positions are: Clamp, Unclamp and Origin. If the arm does not move to the commanded position within the allowed time, alarm 622 TOOLARM FAULT is generated. For vertical mills without a serpentine type 1 side- mount tool changer, this parameter should be set to 0.

Parameter 309 MOTOR COAST TIME

This parameter supports the serpentine type 1 and the Disk type 1 tool changers. It specifies the time allowed (in milliseconds) for the tool arm motor to start moving after being commanded to do so, and to stop moving after being commanded to do so. In either case, if the arm has not responded after the allowed time, alarm 627 ATC ARM POSITION TIMEOUT is generated. For vertical mills with out a serpentine type 1 or Disk type 1 side- mount tool changer, this parameter should be set to 0.

Parameter 310 CAM LOCK DELAY

This parameter supports the serpentine type 1 tool changers. It specifies the time allowed (in milliseconds) to lock the cam by pushing the shot in, or to unlock the cam by pulling the shot pin out. If the shot pin has not moved to its commanded position within the allowed time, alarm 625 INVALID TC START CONDITION is generated.

Parameter 311 ARM BUMP TIME

This parameter supports the serpentine type 1 and the Disk type 1 tool changers. During tool change recovery, the arm may be moved a small amount by pressing the ATC FWD or ATC REV key. Each press of the key will cause the arm motors to run for the amount of time (in milliseconds) specified by this parameter. For vertical mills without a serpentine type 1 or Disk type 1 side- mount tool changer, this parameter should be set to 0.



Parameter 312 CAROUSEL BUMP TIME

This parameter supports the serpentine type 1 and the Disk type 1 tool changers. During tool change recovery, the carousel may be moved a small amount by pressing the ATC FWD or ATC REV key. Each press of the key will cause the carousel motors to run for the amount of time (in milliseconds) specified by this parameter. For vertical mills without a serpentine type 1 or Disk type 1 side-mount tool changer, this parameter should be set to 0.

ALARMS

Alarm 622 TOOL ARM FAULT

This alarm supports the serpentine type 1 and the Disk type 1 tool changers. It is generated if the arm is not at the Origin position, or the arm motor is already on when a tool change process is started.

Alarm 623 SIDE MOUNT CAROUSEL ERROR

This alarm supports the serpentine type 1 and the Disk type 1 tool changers. It is generated if the carousel motor is still on when the tool pocket is unlocked and lowered prior to a tool change.

On the serpentine type 1 tool changer, during tool change recovery, this alarm is also generated if the carousel does not start to rotate after the allowed time specified by parameter 60 TURRET START DELAY, or does not stop rotating after the allowed time specified by parameter 61 TURRET STOP DELAY.

Alarm 624 INVALID TOOL

This alarm supports the serpentine type 1 and the Disk type 1 tool changers. It is generated if the tool specified by the G-code program is not found in the POCKET- TOOL table.

Alarm 625 INVALID TC START CONDITION

This alarm supports the serpentine type 1 and the Disk type 1 tool changers. It is generated if conditions are not correct when a tool change is started. Incorrect conditions include:

- Tool not found in POCKET- TOOL table
- Unable to move carousel to specified tool
- Shot pin not engaged (serpentine type 1).
- Tool carousel not at TC mark (Disk type 1).
- Arm not in Origin position.
- Arm motor already on.

Alarm 626 TOOL POCKET SLIDE ERROR

This alarm supports the serpentine type 1 and the Disk type 1 tool changers. It is generated if the tool pocket has not moved to its commanded position (and settled) within the time allowed by parameters 306 and 62. This alarm can also be generated if the tool changer is in an invalid position at power-up, the start of a gear change or tool change.

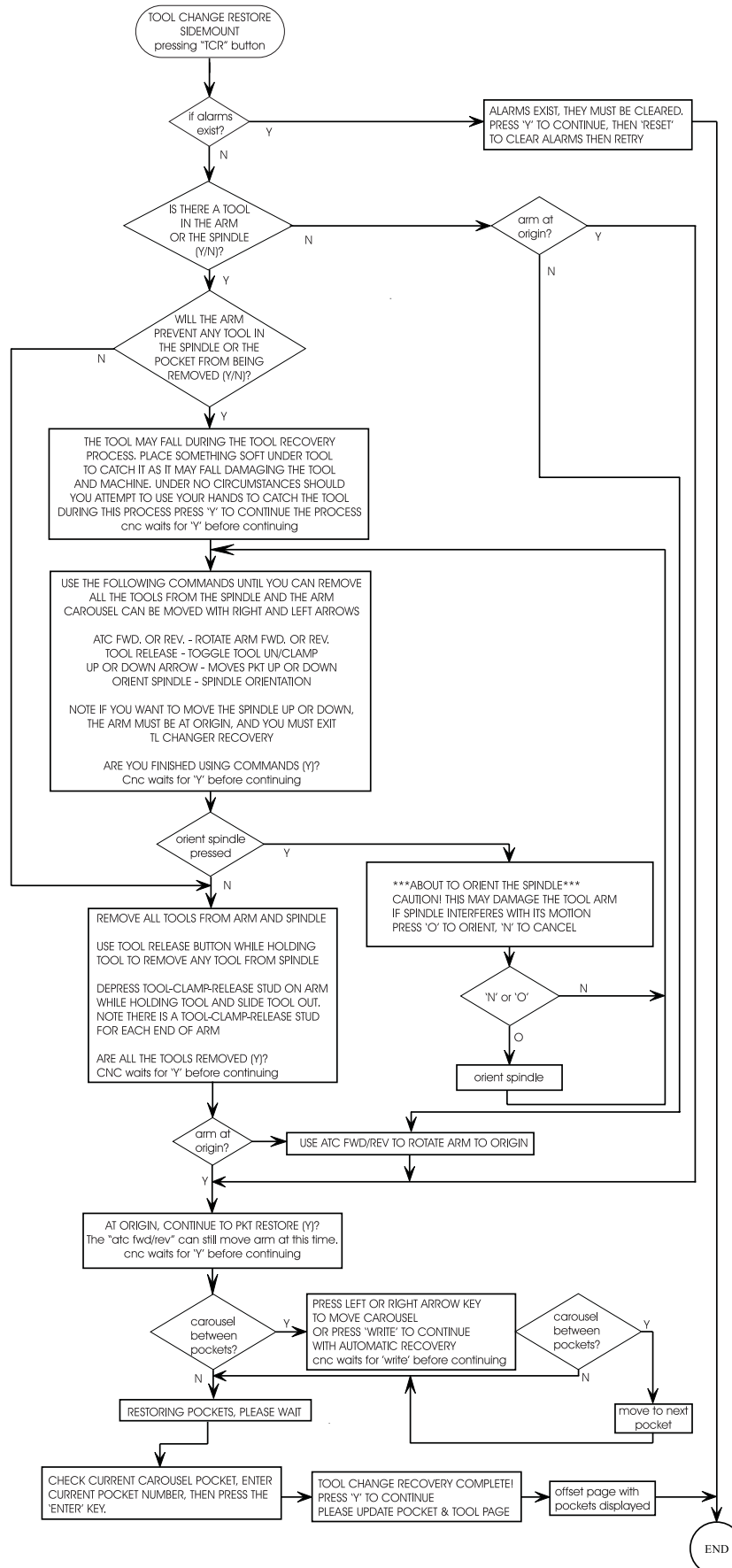
Alarm 627 ATC ARM POSITION TIMEOUT

This alarm supports the serpentine type 1 and the Disk type 1 tool changers. It is generated if the tool arm has not moved after the allowed time or has not stopped after the allowed time. Refer to Parameter 309 MOTOR COAST TIME.

Alarm 628 ATC ARM POSITIONING ERROR

This alarm supports the serpentine type 1 and the Disk type 1 tool changers. It is generated if:

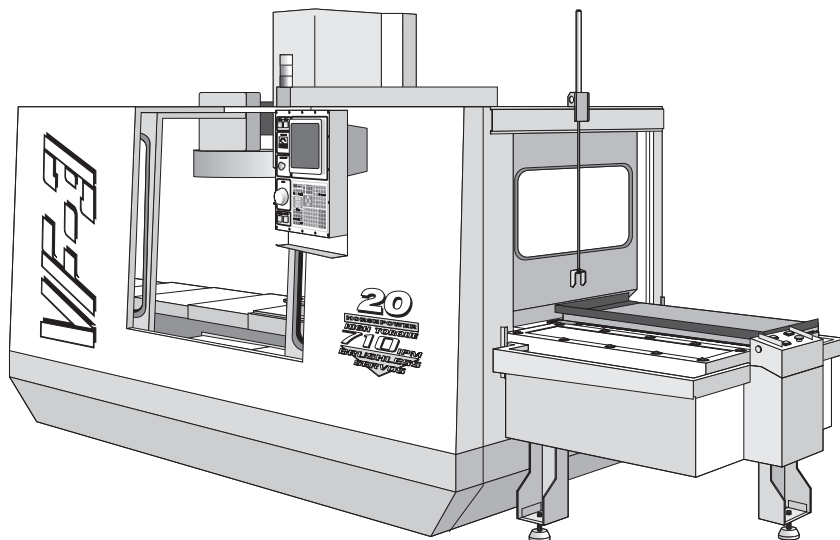
- The arm was being moved from the ORIGIN position to the CLAMP position and it coasted past the MOTOR STOP point or could not get to the CLAMP point.
- The arm was being moved from the CLAMP position to the UNCLAMP position and it coasted past the MOTOR STOP point or could not get to the UNCLAMP point (same physical point as CLAMP).
- The arm was being moved back to the ORIGIN position and it coasted past the MOTOR STOP point or could not get to the ORIGIN point.



SIDEMOUNT MILL TOOL RECOVERY
SCREEN TEXT = UPPER CASE
ACTIONS - DECISIONS = LOWER CASE
S = SITUATION
M = NEXT MODE



11.13 AUTOMATIC PALLET CHANGER (APC)



OVERVIEW

IMPORTANT!

Read this entire section before attempting to install or run the APC. Serious injury or damage to the machine could result from not following these procedures.

The HAAS Automatic Pallet Changer (APC) is used to automatically load and unload pallets into a HAAS VF-3 or VF-4. The APC is controlled by the VMC control, and uses the same air and power supplies. A simple M code (M50) is all that is required to change pallets.

The APC assembly consists of the base, which serves as the "home" position for the two 19" x 40" pallets, and a receiver assembly, which is bolted to the VMC table and holds the pallets in place during machining operations. The VMC unit contains air lines for clamping pallets, and a special automatic door in the side of the VMC enclosure allows for loading and unloading of pallets. The APC has its own operator's panel, which includes CYCLE START, FEED HOLD, PALLET READY, and EMERGENCY STOP buttons.

The APC can be programmed to load a pallet or change pallets by calling an M50. When a pallet change is programmed, the VMC table moves to the right and adjacent to the empty pallet position on the APC base. The receiver assembly then unclamps the pallet. The automatic door opens, the table moves to the unload position, and the pallet is pulled into its home position. The VMC table moves over to the other pallet position, the next pallet is pushed onto the receiver, and the pallet is clamped into position. The table then returns to its programmed position and the door closes.

Pallet movement is accomplished through the use of a drive motor and chain. The drive motor has a non-adjustable slip clutch that prevents damage to the motor if the pallet change is physically blocked.

Note: The machine will beep during a pallet change. This is an audible warning signal, and not an alarm.

**INSTALLATION****TOOLS REQUIRED:**

- Machinists' level (2)
- 6' straight-edge
- Allen wrenches
- Forklift capable of lifting 5,000 lbs, with forks at least 6' long
- **Included** - APC chain rotation tool (P/N 20-0102)
- **Included** - APC jumper

Prior to installing your VMC, make certain that there is enough floor space on the right-hand side of the VMC to place the APC. The APC will add approximately 52 $\frac{1}{2}$ " (57 $\frac{1}{2}$ " for the VF-4) to the width of the VMC. Ensure that all the air and power requirements for the VMC have been met.

Note: After unpacking the APC remove the pallet fixing bolts and take the pallets down from the top of the APC. Do not reinstall the pallets onto the APC until it has been properly attached to the VMC and leveled.

CAUTION! The pallets have wipers on the bottom of the clamp rails at the VMC end. These wipers should overhang any temporary storage position.

M CODES - INSTALLATION

The following **M** codes are used during the APC Installation procedure. These **M** codes are not intended to be used to program the APC at any other time.

- M14** **Go to pallet #1 load position**
- M15** **Go to pallet #2 load position**
- M17** **Unclamp pallet and open automatic door**
- M18** **Clamp pallet and close automatic door**

CAUTION! When commanding M Codes, always command M17 to open the door before commanding M14 or M15 to position the table to a pallet load/unload position. With a pallet present on the receiver the pallet drive leg will interfere with door movement.



ROUGH LEVELING / ALIGNMENT

In this section, the APC will be set in place and roughly aligned to the VMC. When the alignment procedure is complete, the rollers on the VMC receiver and on the APC base will be at the same height and aligned with each other. In the "Fine Tuning" section, the APC will be aligned more precisely by adjusting variables in the VMC control.

1. Remove the APC from it's shipping crate. Remove all tie-downs and strapping materials. Remove the pallet shipping bolts, lift off the pallets and set them aside. Remove the door air cylinder and the cable bundle from their storage positions in the APC pan.
2. **Important!** Before installing the APC, ensure the VMC has been installed and leveled. Also make sure the APC jumper is installed in the "5th Axis" port on the side of the VMC control cabinet (see Figure 11.16).

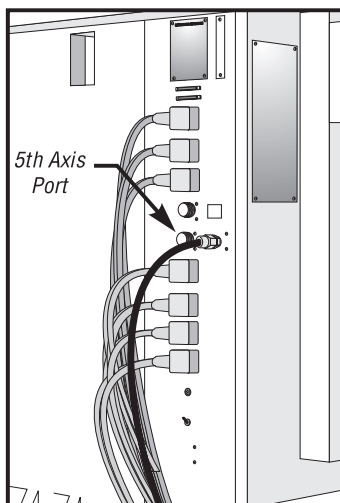


Figure 11.16 Fifth Axis port on the side of the VMC control cabinet

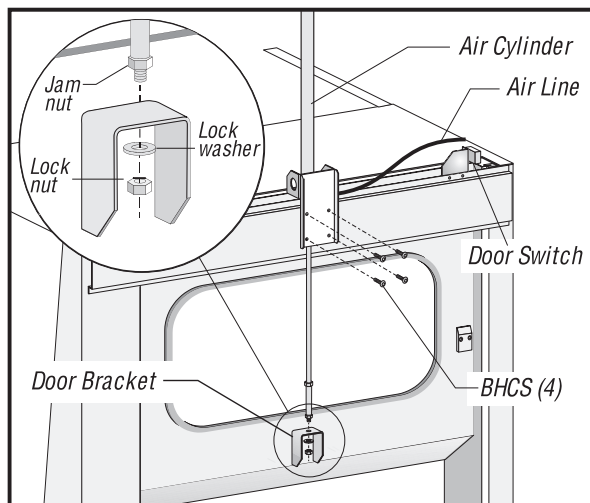


Figure 11.17 Air cylinder installation.

3. Remove the lock nut and washer from the end of the air cylinder rod, but leave the jam nut in place. Connect the automatic door air cylinder and bracket to the machine with four 1/4"-20 BHCS, as shown in Figure 11.17.
4. Insert the air cylinder rod into the door bracket, then attach the lock nut and washer to the bottom of the cylinder rod.
5. Install the door switch and bracket onto the VMC with two SHCS. Ensure the switch is pointing towards the door. Insert the door air line into the fitting at the base of the air cylinder.
6. POWER ON the VMC and ZERO RETURN all axes. Verify that the VMC operates normally.



7. In MDI mode, command an M17. The table will unclamp (raise) and the automatic door will open. The door and table will stay up. Clean all shipping protectant from the rollers, alignment pins and clamp bars.
8. In MDI mode, command an M14. The table will move to the pallet #1 load position. (The front right corner of the VMC).

Note: M14 will move the table slowly in case a pallet is present.

9. Raise the APC with the forklift and install the leveling screws into the legs. As a preliminary height setting, turn the screws until the length of screw extending above the APC leg is the same as the length of VMC leveling screw extending above the VMC base. Set the leveling pads (3) in place.
10. Set the APC in place on the leveling pads, so that it is centered in the VMC door opening (see Figure 11.18). Ensure that the lip of the APC enclosure is centered in the VMC drip trough (see Figure 11.19).

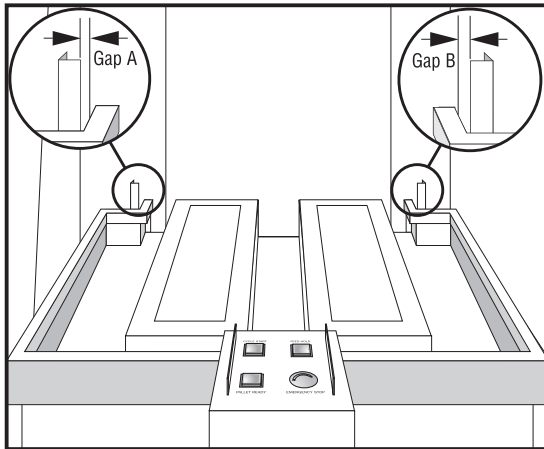


Figure 11.18 Gaps A and B should be equal.

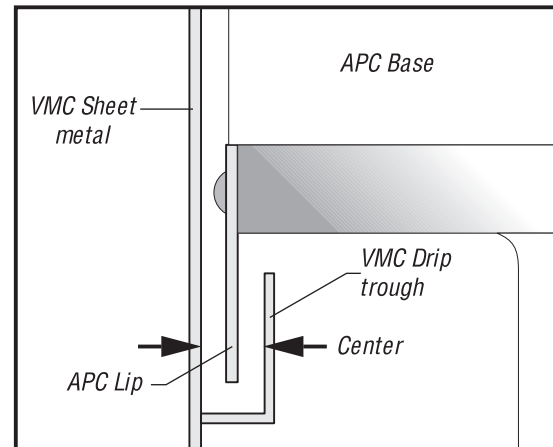


Figure 11.19 The APC lip should be centered in the VMC drip trough.

11. Attach the APC brace to the VMC and the APC with six screws (four 3/8" screws in the APC legs and two 1" screws in the VMC base), but DO NOT TIGHTEN (see Figure 11.20). The VF-3 uses the two holes in the brace that are closest to the VMC, while the VF-4 uses the two holes farthest from the VMC.

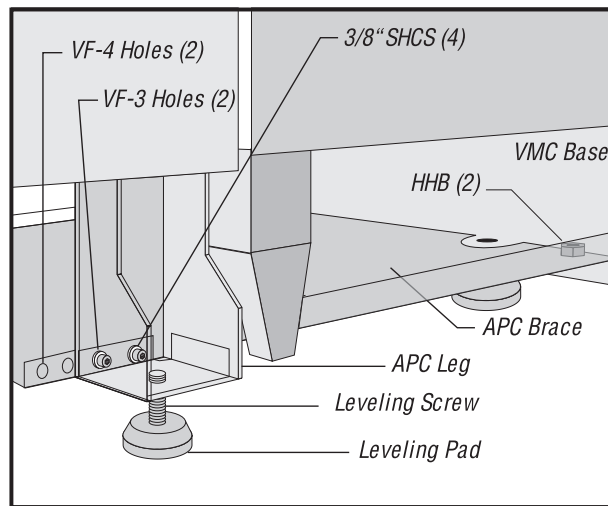


Figure 11.20 Installing the APC brace (VF-3 shown).

12. Place the straight-edge on top of the rollers of both the VMC receiver (unclamped) and the APC, to check the APC height (see Figure 11.21). Make sure that at least 12" of the straight-edge is on each in order to get an accurate reading. Adjust the leveling screws in the APC legs until the straight-edge lays completely flat on all rollers.

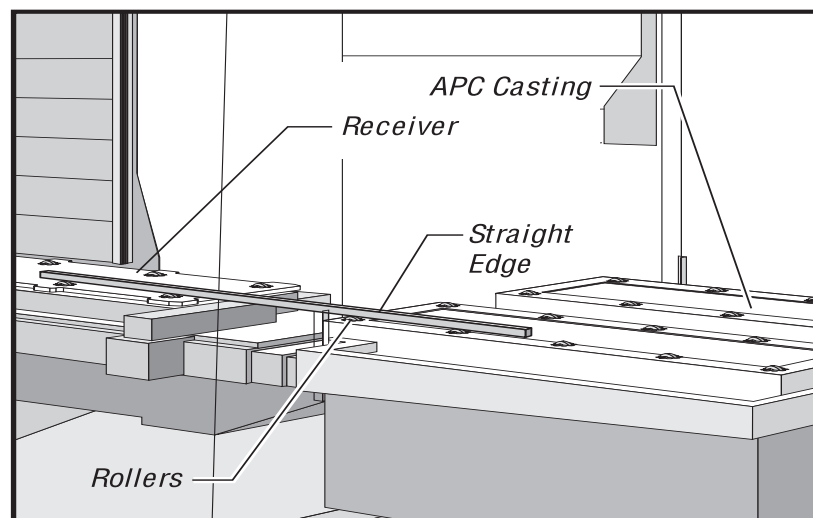


Figure 11.21 APC roller alignment setup.

13. Place the straight-edge against the side of the rollers on the VMC receiver and the APC, to ensure that the rollers are aligned in the Y-axis. The straight-edge must extend past at least two rollers on both the APC and VMC to get an accurate reading.



If the rollers are not parallel, the APC must be moved. The APC can be moved slightly by tapping on the leveling pads with a hammer. Leave the straight-edge against the edge of the rollers to ensure that the alignment isn't affected during APC installation.

Note: If the rollers are parallel, but are not aligned, the adjustments will be made later in the "Fine Tuning" section.

14. In MDI mode, command an M15. The table will move to the pallet #2 load position (right rear corner of the VMC). Repeat Steps 12 and 13 with the table in this position.
15. Tighten down the six SHCS on the APC brace **evenly**, so the APC is not pulled out of alignment. Recheck the level and all alignments.
16. Open the hinged APC operator's panel cover by removing the two BHCS (see Figure 11.22).

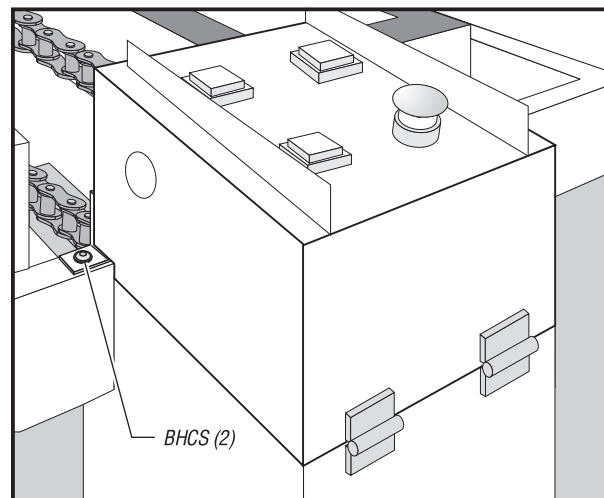


Figure 11.22 APC operator's panel cover.

17. Ensure that the drive roller on the APC chain is centered under the control box, as shown in Figure 11.23. If not, use the chain position adjustment tool to rotate it into this position.

To use this tool, place it into the center of the drive sprocket with the key in the sprocket's keyway (see Figure 11.24). Note that the tool only goes about 1/4" into the sprocket. Push down and turn the tool with a wrench in order to position the chain. **Important! Remember to remove the adjustment tool when finished.**

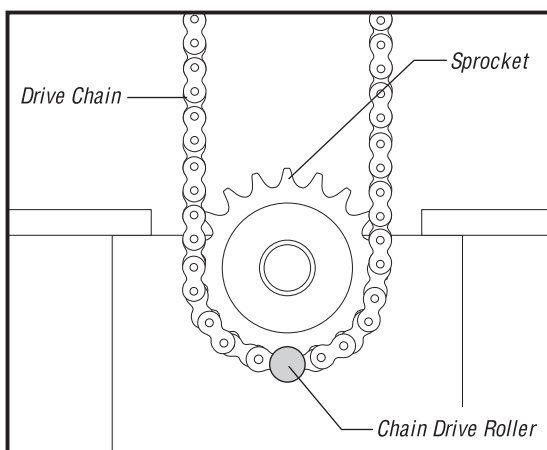


Figure 11.23 Chain drive roller positioning.

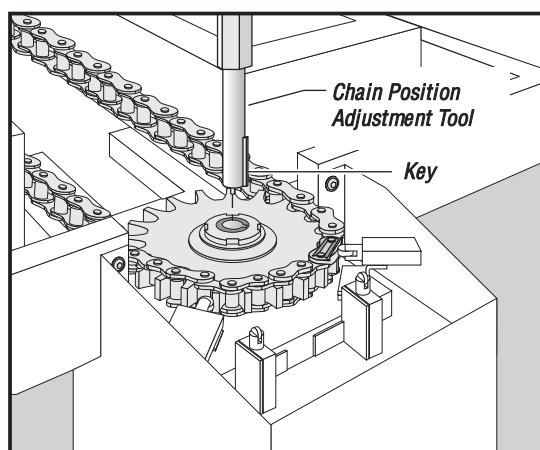


Figure 11.24 Chain position adjustment tool placement.

CAUTION! Use extreme caution when moving the pallets; each pallet weighs 300 lbs., and can cause serious injury. Be careful not to damage the pallets or the APC rollers.

18. Insert two 1/2"-13 eyebolts into each pallet (see Figure 11.25). Attach straps or chains to the eyebolts, and use a forklift or hoist to lift each of the pallets onto the APC. Ensure that each pallet is placed with its drive leg pawl under the APC operator's panel, and that the APC rollers fit into the grooves on the bottom of the pallets (see Figure 11.26).

Note: Be careful not to set the pallets on top of the chain drive roller. If necessary, rotate the chain slightly using the chain position adjustment tool.

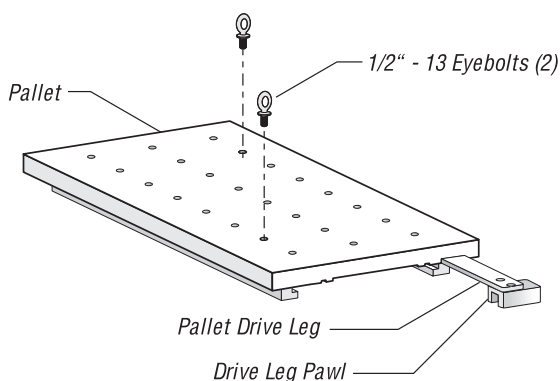


Figure 11.25 Lifting eyebolt locations.

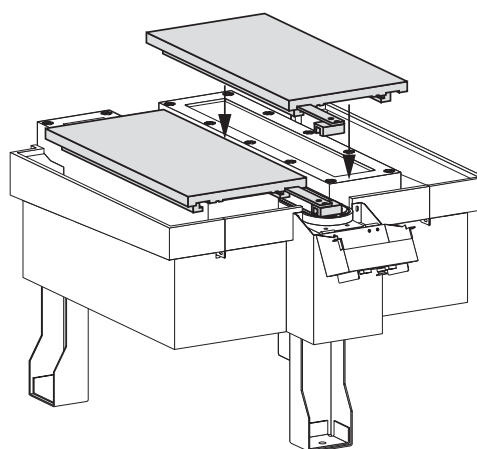


Figure 11.26 Pallet placement.



19. Pull the pallets onto the friction blocks and against the hard stop (see Figure 11.27). Rotate the chain slightly, using the chain position adjustment tool, to allow each drive leg pawl to clear the chain drive roller.

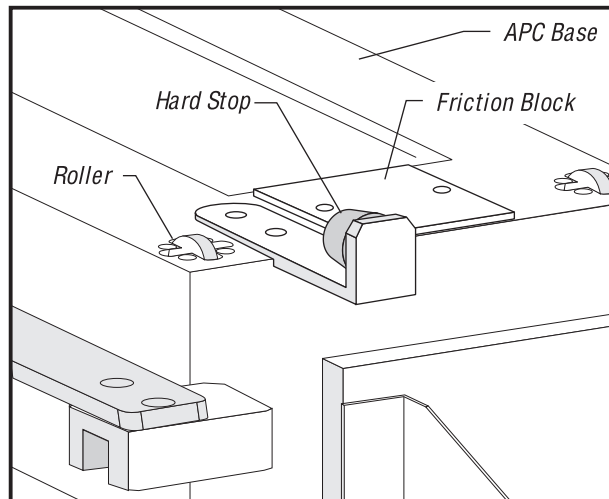


Figure 11.27 View of APC base showing friction block and hard stop.

20. Power OFF the VMC. **WARNING! This will cause the automatic door to fall. Keep hands clear.** Remove the jumper from the Fifth Axis (B) port on the side of the VMC control cabinet (see Figure 11.16), and insert the APC amphenol connector into this port. Tie-wrap the cable to the door switch cable and the air hoses at the rear of the VMC.
21. Screw the air blast hose into the APC. Remove the piece of tubing from the top fitting of the VMC lube/air panel cross fitting, and replace it with the APC air hose (see Figure 11.28).

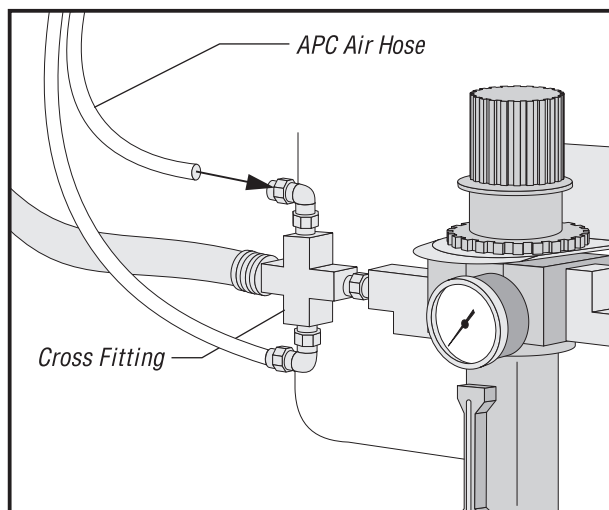


Figure 11.28 APC air hose connection to VMC air/lube cross fitting.



FINE TUNING

The APC was set up, run, and adjusted at the factory. Due to slight variations when disassembling and reassembling, it will be necessary to fine tune the APC to ensure that the pallets load and unload as smoothly as possible.

Note: It is normal to hear a 'clunking' sound during pallet loading and unloading. Unless it is excessive, do not mistake this noise for misalignment.

Macro Variables -

The following macro variables control the location of the VMC table during pallet changes. Fine tuning consists of adjusting variables #500-507 and the height of the APC. For instance, #500 controls the X position that the VMC table will move to in order to load pallet 1. If the receiver does not clamp the pallet due to misalignment of the locating pins and the pallet bushings, this value can be changed to move the receiver closer to or farther from the APC. The drive pin on the chain should drop the pallet directly over the locating pins. Similarly, if the pallet is misaligned in the Y-axis, causing the pallet to shift to one side during loading and unloading, variable #501 can be modified. The following macro variables can be adjusted:

# 500	=	X pallet 1 load position
# 501	=	Y pallet 1 load position
# 502	=	X pallet 1 unload position
# 503	=	Y pallet 1 unload position
# 504	=	X pallet 2 load position
# 505	=	Y pallet 2 load position
# 506	=	X pallet 2 unload position
# 507	=	Y pallet 2 unload position
# 508	=	Clamp Delay
# 509	=	Unclamp Delay
# 510	=	Next pallet number
# 511	=	Door Delay

Macros must be enabled in order to use these statements. The macro variables can be viewed on the CURNT COMDS (Current Commands) page of the VMC display. The PAGE UP and PAGE DOWN keys may then be used to locate the correct list of macro variables. To scroll through this list, use the Cursor Right and Left buttons. Refer to the VMC Operators Manual for instructions on how to modify these variables.

To stop a pallet change in order to adjust the macro variables, press FEED HOLD at any time. This will cause the pallet change to stop after the present step is completed. Pressing CYCLE START will restart the program. If E-STOP is pressed during a pallet change, the motors will stop instantly, and the machine will have to be reset. If one of the chain location switches is not tripped at this time, the chain position will also have to be reset using the chain position adjustment tool.

22. POWER ON the VMC. Release the E-STOP button and press RESET. WARNING! The automatic door may move when the VMC is powered on. Keep hands clear.

23. In MDI mode, command an M17, then an M14, to position the VMC table.



24. Manually push the pallet adjacent to the receiver off the APC and onto the receiver. If necessary, use the chain position adjustment tool to rotate the chain until the chain drive roller clears the drive leg pawl.

WARNING!

Do not push the wrong pallet off the APC. It can cause serious injury, and will damage the machine.

25. Push the pallet between the receiver and the APC a number of times. As the pallet goes over the receiver rollers, note if the Y or Z axis is out of alignment.

If the alignment is slightly off in the Y-axis, handle jog the table in the Y-axis (in increments of .001) until pallet loading and unloading is smooth. Record the final Y-axis position in macro variables #501 and #503. These two variables should be set to the same value initially, and may be adjusted separately later if either loading or unloading is not smooth.

If alignment appears to be incorrect in the Z-axis, adjust the APC height by turning the leveling screws until pallet loading and unloading is smooth.

26. Push the pallet back onto the APC. In MDI mode, command an M15 to move the pallet to load position #2. Repeat Steps 24 and 25 above for pallet #2.

The APC height and the Y-axis alignment (macro variables #501, 503, 505, and 507) are now set.

27. In MDI mode, command an M18 to clamp the pallet and close the automatic door.

Note: The message "PEND MOVE - CY/ST" will appear and motion will stop during an M18. The CYCLE START button must be pressed to continue motion.

28. The X-axis alignment must now be checked. To do this, perform the following steps :

Place a program in MDI that consists of an M36 (flash "Pallet Ready" light), an M50 (pallet change), and an M99 (repeat).

Press CYCLE START and the PALLET READY light will flash.

Press PALLET READY, and the pallet will start to load. **Important!** When the chain starts to move, immediately press FEED HOLD. This will cause the pallet to load, but not be clamped.

When the pallet has stopped, check that the bushing in the bottom of the pallet is directly over the locating pin in the receiver (within +/- .09). If not, change the appropriate macro variable and push the pallet against the hard stop.

Press CYCLE START. The pallet should go down fully.

Note: The machine will beep during a pallet change. This is an audible warning signal, and not an alarm.

29. To confirm the correct X-axis pallet load positions, run the program that consists of an M36, M50, and M99 again. Note any places that the process does not run smoothly, press FEED HOLD, and change one of the macro variables to adjust this misalignment.

Note: Make sure to confirm the alignment at both pallet load positions. After confirmation the variables should be written down or saved to disk.



PROGRAMMING

PROGRAM COMMANDS

The APC is controlled by a program in CNC memory. M codes are used in the parts program to control the APC. The pallet change routine is run by using an M50 code in the parts program, which accesses a macro program that cannot be viewed or edited. In order for M50 to work, Parameter 81 (M MACRO CALL 09000) must be set to 50.

The APC can be programmed to run one program on both pallets, or a different program on each pallet. The "Sample Programs" section consists of two basic programs: the first sample program runs the same part on both pallets, and the second program runs a different part on each pallet.

The APC has its own operator's panel, which includes CYCLE START, FEED HOLD, and EMERGENCY STOP buttons that perform the same functions as those on the VMC control panel. This panel also includes a PALLET READY button, which is used for APC functions (Refer to figure 11.29).

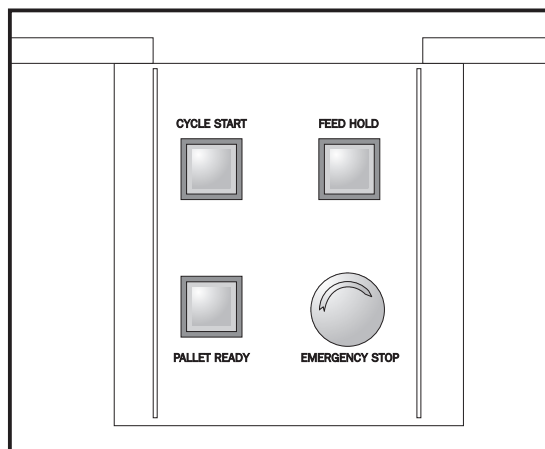


Figure 11.29 APC operator's control panel (top view).

When an M36 is programmed prior to an M50, the APC will flash the "PALLET READY" light at the completion of the user's part program, and wait until the PALLET READY button has been pressed before performing the pallet change. This M code is optional, but is recommended because it prevents a pallet change from occurring before the operator is ready. The PALLET READY button can be pressed at any time and will be recognized when the next program is finished and a pallet change is required.

Important! If any alignment adjustments are necessary during operation, refer to the "Installation" section and modify the appropriate macro variables.

Note: Use caution when changing macro variable #510 during operation. An alarm will be received if the control and the limit switches disagree on pallet position.

**M CODES**

Only one **M** code may be programmed per block of a program. All **M** codes are effective or cause an action to occur at the end of the block. The following **M** codes are used to program the APC:

M36 Hold until "Pallet Ready" button is pressed

M50 Perform pallet change

SAMPLE PROGRAMS**Example #1:**

Basic program using the "Pallet Ready" light on the APC.

```
Oxxxx
M36          (Flash "Pallet Ready" light - wait until PALLET READY button pressed to continue)
M50          (Perform pallet change after "Pallet Ready" button is pressed)
-
(USER's PART PROGRAM)
-
M99          (Repeat program)
```

Example #2:

Basic pallet change program that tracks which part is to be machined on each pallet.

```
Oxxxx
M36          (Flash "Pallet Ready" light - wait until PALLET READY button pressed to
              continue)
M50          (Perform pallet change after "Pallet Ready" button is pressed)
If [#510 EQ 1] M98 (USER's PART PROGRAM #1)
If [#510 EQ 2] M98 (USER's PART PROGRAM #2)
M99
```

**TROUBLESHOOTING****MACHINE SEQUENCE**

The machine performs the following sequences when these **M** codes are commanded.

M14 - GO TO PALLET #1 LOAD POSITION (Used for machine installation only)

Move table into pallet #1 load position.

M15 - GO TO PALLET #2 LOAD POSITION (Used for machine installation only)

Move table into pallet #2 load position.

M17 - UNCLAMP PALLET AND OPEN AUTOMATIC DOOR (Used for machine installation only)

If table is near door, move table away from door.

Unclamp pallet.

Open automatic door.

Check Door Open switch - If door not open, generate alarm #1008.

Check Receiver Unclamp switch - If receiver not unclamped, generate alarm #1002.

M18 - CLAMP PALLET AND CLOSE AUTOMATIC DOOR (Used for machine installation only)

Clamp pallet.

Check Receiver Clamp switch - If receiver not clamped within the amount of time set in macro variable #509, generate alarm #1003.

Close automatic door.

Check Door Open switch - If door not closed, generate alarm #1009.

M36 - HOLD UNTIL "PALLET READY" BUTTON IS PRESSED

When "PALLET READY" button is pressed, enable an M50 pallet change (see below).

M50 - PERFORM PALLET CHANGE**Prepare for Pallet Change:**

Turn coolant OFF.

Turn Through the Spindle Coolant OFF (if applicable).

Stop the spindle.

Enable APC motor.

Turn beeper ON. (The 'beeper' refers to the audible APC warning signal, and not to an alarm condition.)

Read all 3 pallet position switches (one switch for Receiver Clamp/Unclamp and two switches for Pallet Stop / Location) to determine empty position.

**Read Switches:**

If both pallets are home on APC, move to load pallet position # listed in macro variable #510.
If a pallet is on the receiver, determine the unload position.
Move the table to the correct unload position.
Open the automatic door.

Execute Pallet Change (with a pallet on the receiver):

Unclamp receiver.
Start chain drive (unload pallet).
Stop chain drive when pallet hits stop switch.
Move table to new position.
Start chain drive (load pallet).
Stop chain drive when Chain Switch Block trips the Chain Stop switch.
Clamp receiver.
Move table away from automatic door.

Return to Normal Operation:

Turn beeper off.
Disable chain drive.
Close automatic door.

Note: The M50 pallet change macro must be successfully completed before normal VMC operation can continue.

LIMIT SWITCHES

The APC contains the following six (6) limit switches that monitor machine operation:

Door Open Switch - Senses when the door is in the fully up position. Note that the door is not sensed in the down position.

Receiver Clamp/Unclamp Switch - Senses whether or not the receiver is in the down (clamped) position.

Pallet Stop / Location Switches (2) - Sense whether each pallet is home (on the APC). When a pallet is being unloaded, the chain drive will stop when this switch is tripped.

Chain Stop / Location Switches (2) - Sense the chain location, and whether it is in the correct position to perform an operation. After a pallet has been loaded, the chain drive will stop when this switch is tripped.

**ALARMS / FAULTS**

If a fault occurs during operation, the red beacon on the VMC will flash and a message will be displayed on the screen. An alarm code will be placed in the VMC's alarm history and will be available for future reference.

EMERGENCY STOP or a machine fault not associated with the APC inputs or program will halt all motors. Pressing the RESET button will clear these faults.

Important! If E-STOP was pressed during a pallet change, the control may not know the location of the pallets or chain, depending on when E-STOP was pressed, and alarm #1011 will be generated. In this case, refer to the troubleshooting flowchart at the end of this section.

When an alarm occurs, correct the problem, then run a single M50 in MDI mode to reset the APC.

The following is a list of the alarms associated with the APC:

180 Pallet Not Clamped

Before machining a part, the VMC checks to make sure the pallet is in the correct position and is fully clamped. This alarm will be generated if either of these is not true, or if the pallet change macro has not been completed. The pallet change macro can be interrupted by pressing E-stop, Reset, or Feed Hold.

This alarm indicates that the VMC table is not in the correct position, a pallet change was not completed, or E-STOP was pressed.

If the pallet is clamped, run an M50 to reset the machine. If the pallet is in the correct position but not clamped, push the pallet against the hard stop, run an M18 to clamp the pallet, then run an M50 to reset the machine. If the pallet is clamped, but not correctly, run an M17 to unclamp, push the pallet to the correct position, run an M18 to clamp the pallet, then run an M50 to reset the machine.

Less common causes for this alarm could be that the slip clutch is slipping, the motor is at fault, an air solenoid is bad, or an air line is blocked or kinked.

1002 Unclamp Error

The pallet did not unclamp in the amount of time allowed. This can be caused by a bad air solenoid, a blocked or kinked air line, or a mechanical problem.

1003 Clamp Error

The pallet did not clamp in the amount of time allowed by the M50 macro. This alarm is most likely caused by the VMC table not being in the correct position. This can be adjusted using the macro variables for the X position (#500, 504) as described in the "Installation" section. If the pallet is in the correct position but not clamped, push the pallet against the hard stop and run an M18.

If the pallet is clamped, but not correctly, run an M17 to unclamp, push the pallet to the correct position, and run an M18 to clamp the pallet. Less common causes could be that the slip clutch is slipping, the motor is at fault, an air solenoid is bad, or an air line is blocked or kinked.

**1004 Mislocated Pallet @ APC**

A pallet is not in the proper place on the APC. The pallet must be pushed back against the hard stop by hand.

1005 Pal No Conflict Rec & Ch (Pallet Number Conflict Receiver and Pallet Changer)

The incorrect pallet number is entered in macro variable #510. Run an M50 to reset this variable.

1006 Switch Missed Pal 1

Pallet #1 did not return from the receiver to the APC in the allowable amount of time. This can be caused by the chain switch block missing the limit switch, or from another mechanical problem, such as clutch slippage.

1007 Switch Missed Pal 2

Pallet #2 did not return from the receiver to the APC in the allowable amount of time. This can be caused by the chain switch block missing the limit switch, or another mechanical problem, such as clutch slippage.

1008 Door Not Open

The automatic door did not open (in the allowable time) when necessary to perform an APC function. This can be caused by a bad air solenoid, a blocked or kinked air line, or a mechanical problem.

1009 Door Not Closed

The automatic door did not close (in the allowable time) when necessary after an APC function has been performed. This can be caused by a bad air solenoid, a blocked or kinked air line, or a mechanical problem.

1010 Missing Pallet @ Receiver

Pallet change sequence was halted because receiver switch was not activated. Pallet is either unclamped or not on the receiver. Ensure the pallet is correctly located on receiver (against hardstop) then run M18 to clamp the pallet.

1011 Unknown Chain Location

Neither Chain Location switch is tripped, so the control cannot locate the chain position. This can occur if a pallet change is interrupted for any reason, such as an alarm or an E-STOP. To correct this problem, the pallets and chain must be moved back into a recognized position, such as both pallets home or one pallet home and one on the receiver. The chain position adjustment tool must be used to rotate the chain into position. The pallets must be pushed into place by hand.

WARNING!

Do not move the limit switches for any reason.

CAUTION!

The pallets weigh 300 lbs. each, and can cause serious injury. Use extreme caution when moving them.

1012 Incorrect Chain Location

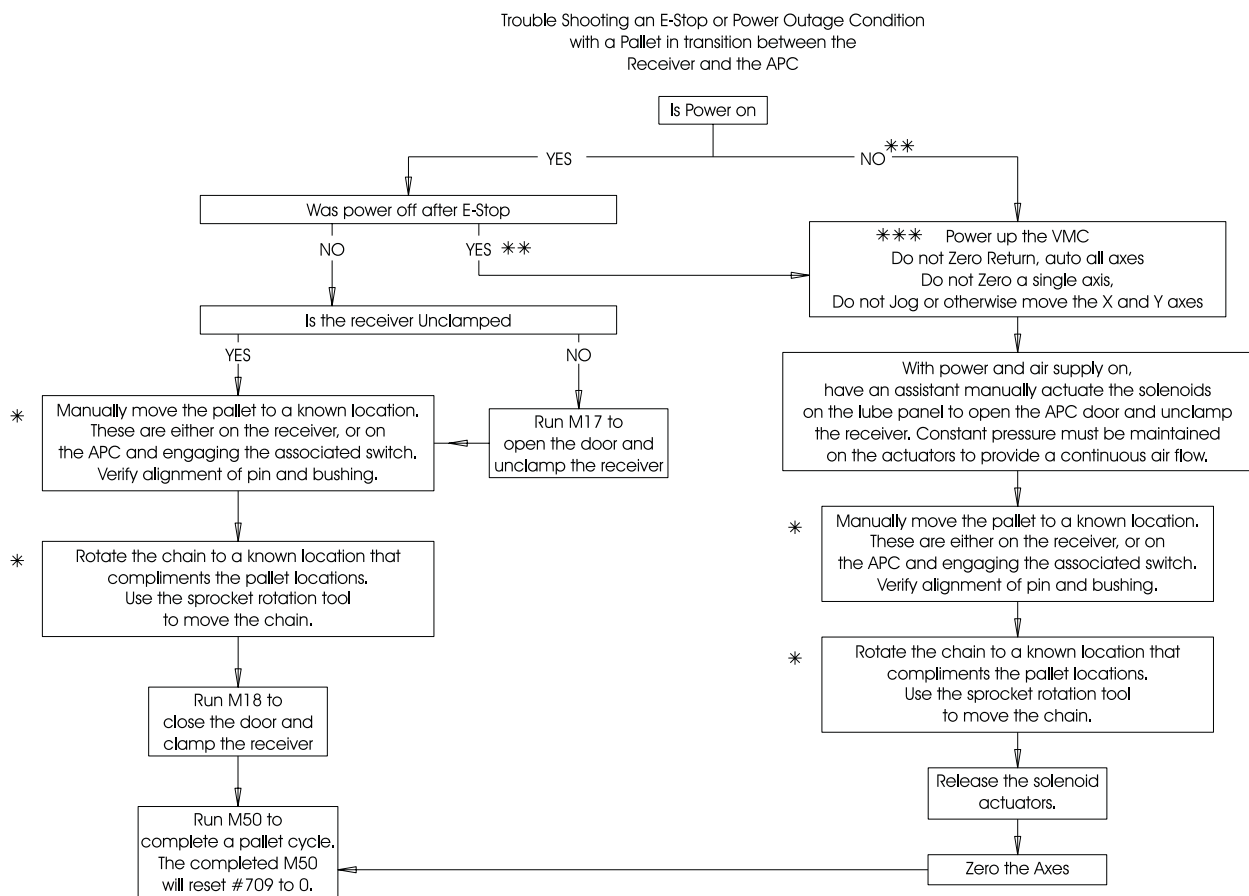
Chain not in position to load or unload pallets when necessary. To correct this, the mislocated pallet must be moved back into the proper position by hand.

CAUTION!

The pallets weigh 300 lbs. each, and can cause serious injury. Use extreme caution when moving them.



Recovery from an E-Stop initiated during a pallet change



* There are 5 switches involved in the location of the pallets and chain.
 1 pallet switch on the receiver
 2 pallet switches on the APC
 2 chain switches on the APC } Under the control panel
 See attached APC electrical notes.

** If the power to the VMC has been shut down either intentionally or by power outage, damage may occur to the APC pallet or receiver if the X or Y axes are moved in the VMC at power up.

*** At power up the VMC should investigate the condition of the pallet and chain location and alarm if either an unknown chain location or unknown pallet location are detected.

**TECHNICAL REFERENCE****PALLET SPECIFICATIONS**

Weight 300 lbs.
Thickness 1 1/2"
Bolt Holes 5 inch spacing, 1/2"-13 X 3/4 DP threads

CAUTION: Drilled holes in pallet should be less than 1" deep.

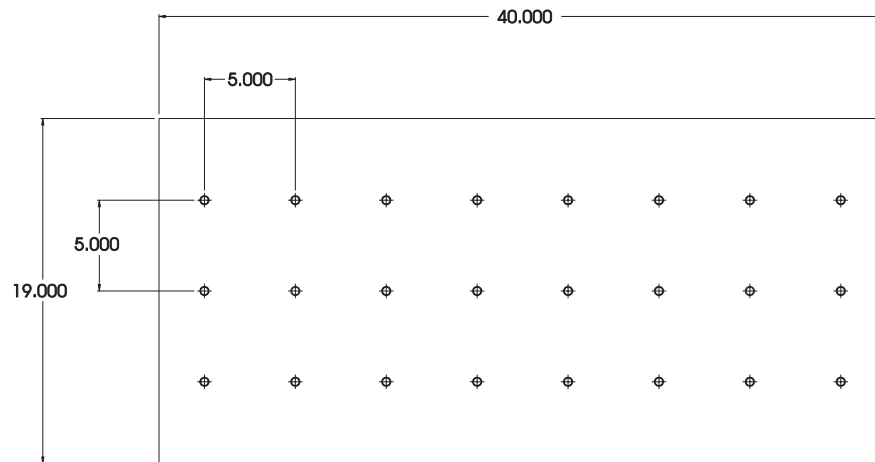
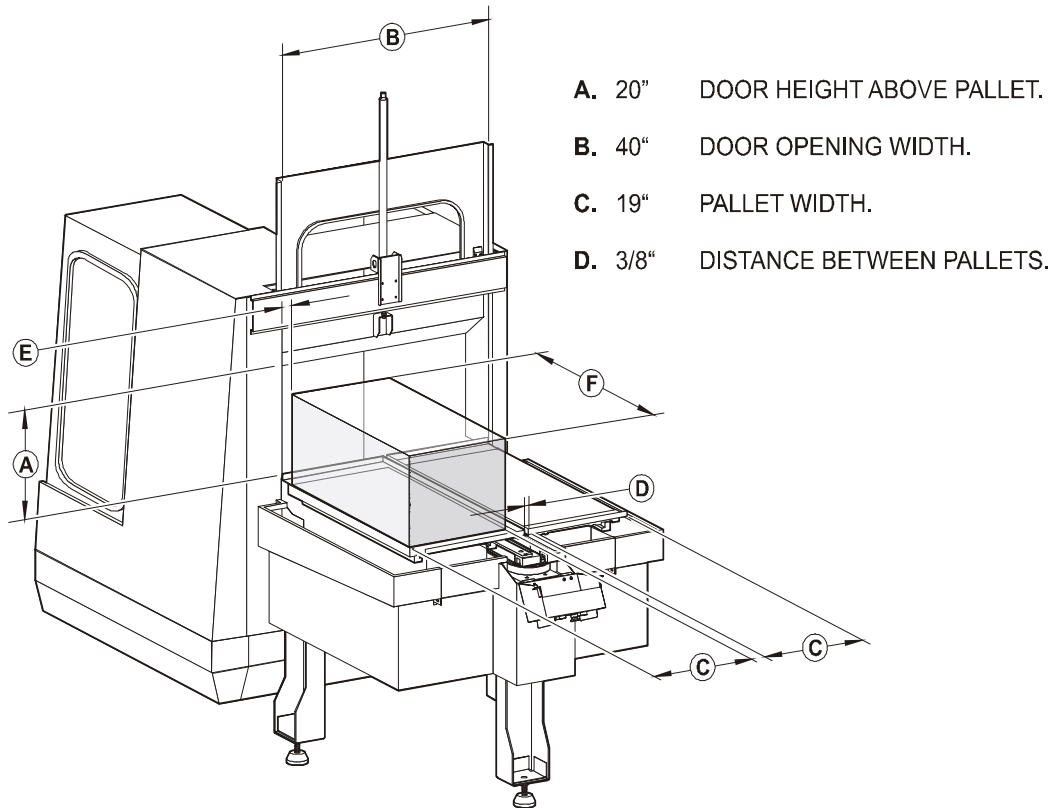


Figure 11.30 Pallet bolt hole pattern and spacing.

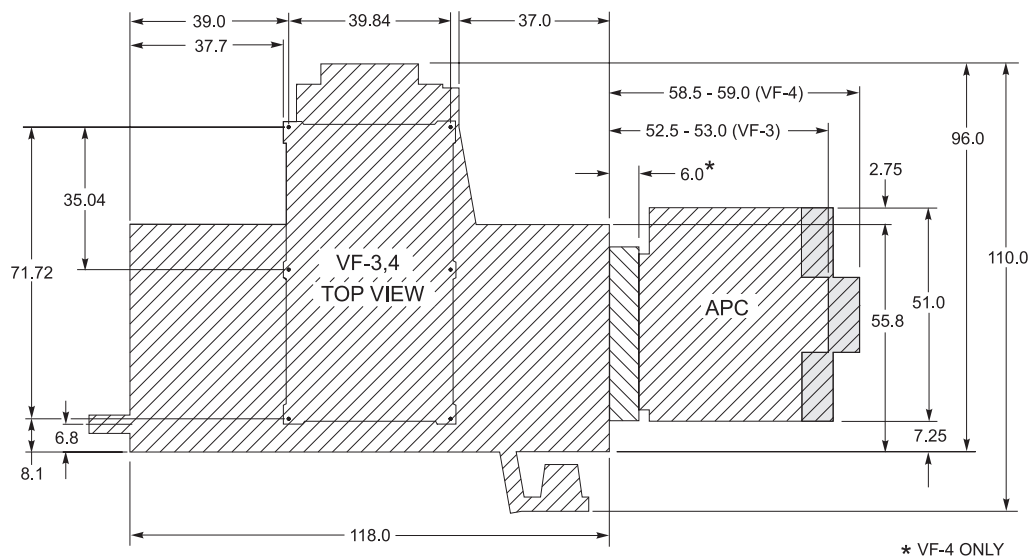
CAUTION! For pallet loads over 250 lbs, the load should be distributed evenly around the center of the pallet to avoid tipping the pallet during loading and unloading.



APC DIMENSIONS

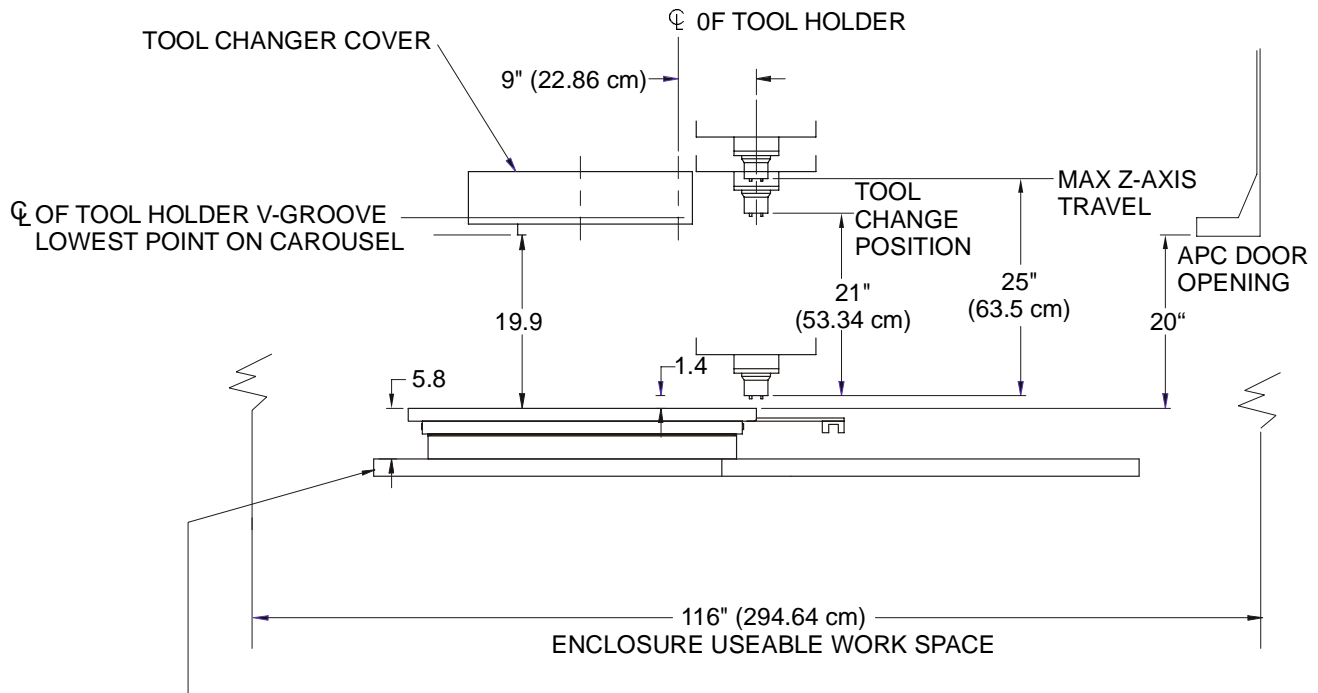


VF 3/4 WITH APC FOOTPRINT





APC WORK ENVELOPES

**MILL TABLE (AT HOME POSITION)**

SIZE: VF-3 48" x 18", (121.92 cm x 45.72 cm)

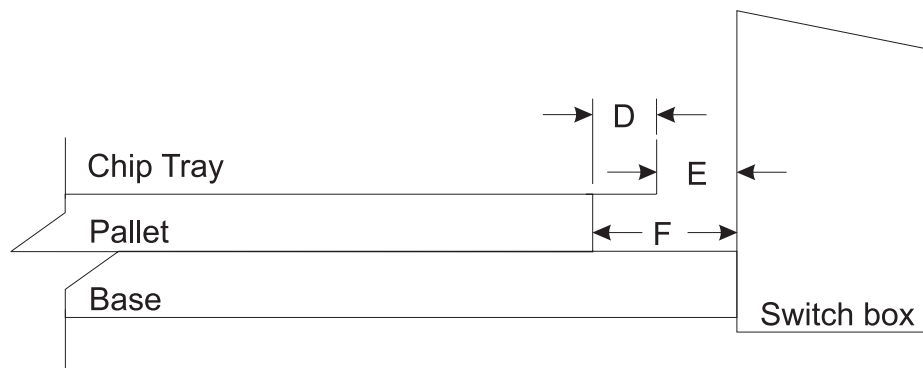
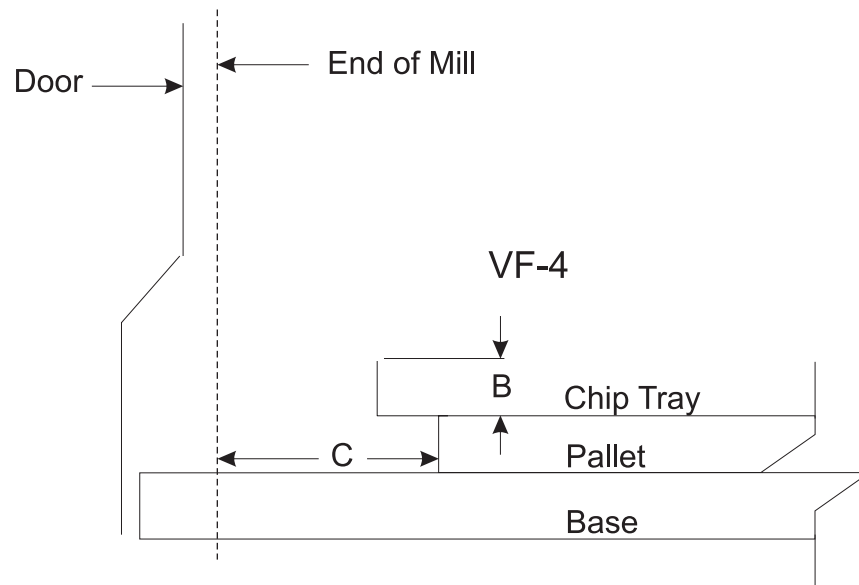
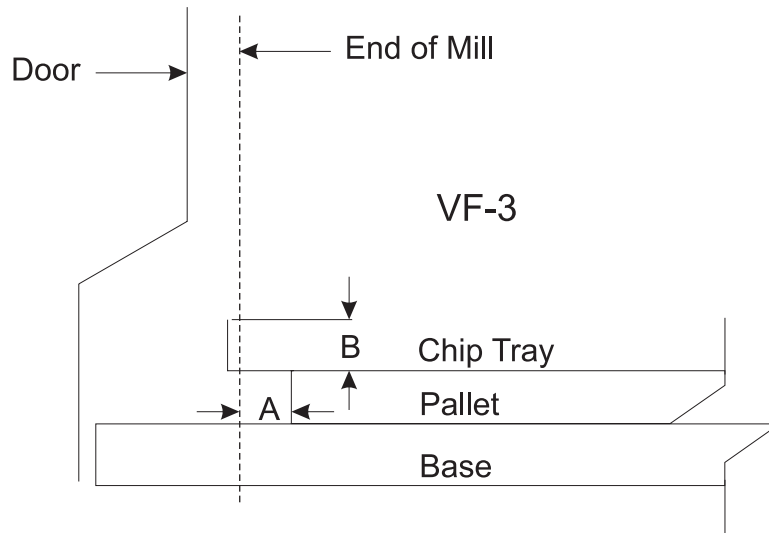
SIZE: VF-4 44" x 18", (111.76 cm x 45.72 cm)

TRAVEL: VF-3 40" x 20", (101.60 cm x 50.80 cm)

TRAVEL: VF-4 50" x 20", (127.00 cm x 50.80 cm)



APC PALLET CLEARANCES



**MAINTENANCE SCHEDULE**

The following is a list of required regular maintenance for the HAAS Automatic Pallet Changer. These specifications must be followed in order to keep your machine in good working order and to protect your warranty.

INTERVAL**MAINTENANCE PERFORMED****DAILY**

Clean chips, coolant, and other debris from the APC and receiver areas. Pay special attention to the receiver clamping bars, and the pallet clamping rails and wipers. Clean chip and coolant from receiver pallet clamped switch. Apply a light coat of grease and work into plunger. **Do not over grease.** Refer to the Figure. Brush off the drive chain to clear chips.

Clean the two locating pins in the receiver base and the corresponding bushings in the pallets. Apply a very light coat of general purpose grease to the pins and bushings, if needed.

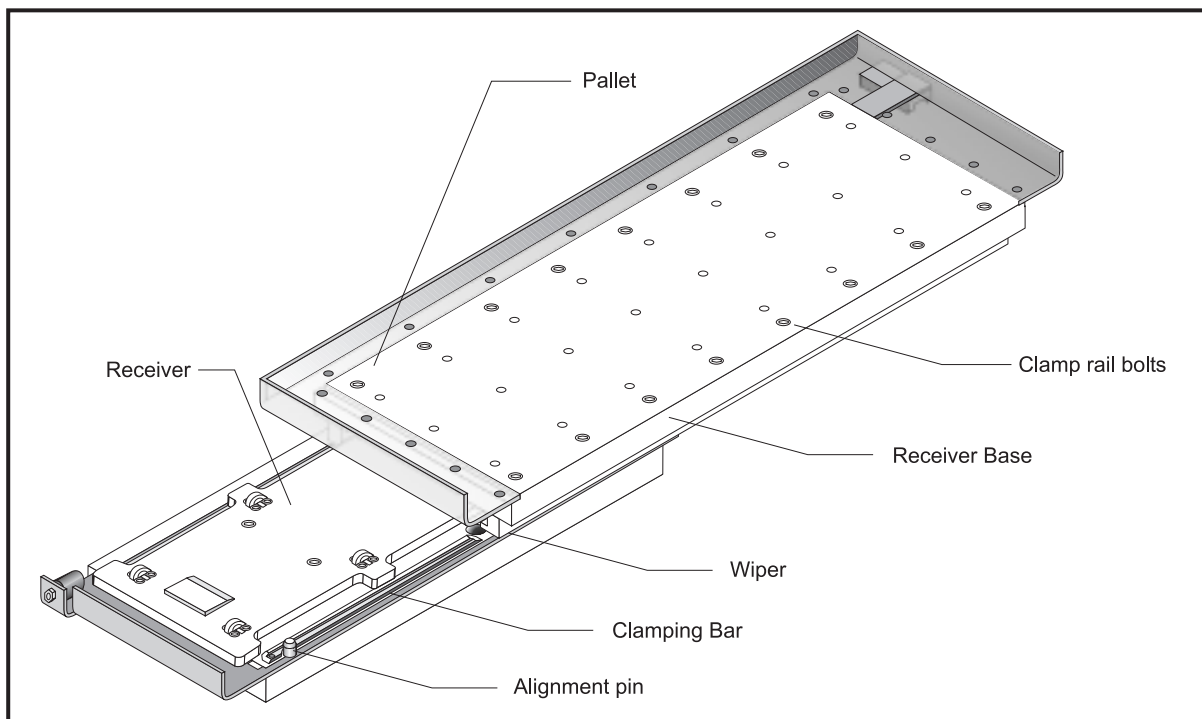
Clean excessive chips and coolant from pallet drive chain.

WEEKLY

Remove the two 1/4" screws from the control panel and rotate to open position. Clean chips and coolant from the drive sprocket and limit switch. Apply a light coat of general purpose grease and work into the switch plungers. Check air gauge/regulator at the VMC lube/air panel for 85 psi.

MONTHLY

Check door for proper operation, both up and down. Clean and grease door side guide rails as required.



12. TECHNICAL REFERENCE

12.1 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.

CAUTION! If machine is equipped with the optional **50 taper spindle**, follow these guidelines:

- 25 lb. maximum per tool, and 300 lb. maximum total tool weight.
- Extremely heavy tool weights should be distributed evenly.

CAUTION! If machine is equipped with the **20 or 32 pocket tool changer**, follow these guidelines:

- 12 lb. maximum per tool (200 lb. maximum total tool weight for 32 pocket tool changer).
- Extremely heavy tool weights should be distributed evenly.
- Ensure there is adequate clearance between tools in the tool changer before running an automatic operation. This distance is 3.6" for 20 pocket, and is 3.4" for 32 pocket.

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.

WARNING

AN INADEQUATE AIR SUPPLY
WILL CAUSE TOOL CHANGER FAULTS

FOLLOW THESE GUIDELINES:

MINIMUM AIR SUPPLY PRESSURE TO MACHINE IS 100 PSI.
OBSERVE GAGE DURING TOOL CHANGE - 10 PSI MAX. DROP.
USING THE AIR GUN DURING TOOL CHANGES MAY CAUSE
FAULTS IF THE AIR SUPPLY TO THE MACHINE IS MARGINAL.
ALLOW 2 HP OF AIR COMPRESSOR PER MACHINE,
(I.E., 5 MACHINES REQUIRE A 10 HP AIR COMPRESSOR).
USE MINIMUM 3/8 ID HOSE FOR 40 TAPER MACHINES,
MINIMUM 1/2 ID HOSE FOR 50 TAPER & HS MACHINES.
AVOID QUICK DISCONNECTS IN SUPPLY LINES - THEY ARE
RESTRICTIVE.

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. Press "Tool Changer Restore" button, to automatically reset the tool changer after a crash. Never put your hands near the tool changer when powered unless the EMERGENCY STOP button is pressed.

FU1 on the I/O PCB or the Power PCB is a fuse for the tool changer motors. It might be blown by an overload or jam of the tool changer. Operation of the tool changer can also be interrupted by problems with the tool clamp/unclamp and the spindle orientation mechanism. Problems with them can be caused by low air pressure or a blown solenoid circuit breaker CB4.

When a tool change operation is performed, the following sequence of events occurs:

- 1) Z axis moves up to machine zero,
- 2) If the spindle is turning, it is commanded to stop; coolant stopped,
- 3) Spindle oriented to Tool Changer,
- 4) Turn TSC pump off, (optional)
- 5) Turn purge on and off (optional)
- 6) Pre-charge is on (40 taper spindle only),
- 7) Shuttle moves in to receive tool,
- 8) Tool unclamps,
- 9) Z axis moves up,
- 10) Tool Changer rotates,
- 11) Z axis moves down,
- 12) Tool clamps,
- 13) Pre-charge off (40 taper spindle only),
- 14) Shuttle moves out.

PULL STUDS

The tool holders used are CT #40 taper, V flange, commonly called "CT 40". For the 50 taper spindle option, the tool holders used are CT #50 taper, V flange, commonly called "CT 50". Use A "45 Degree, P40T Type 1 (P50T Type 1 for 50 taper) 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. If the machine is equipped with the optional BT tool changer, use BT tooling only. **Pull studs are available through HAAS.**

Tool Holders/Pull Studs

CT CAT V-Flange						40T	20-7594 (TSC)		Kit # TPS24CT
								50T	20-7164 (non-TSC)
						40T	22-0075 (TSC)		
								50T	22-0039 (non-TSC)
A	B	C	D	E					
40T	2.69	2.50	.44	5/8"-11	1.75				
50T	4.00	3.87	.44	1"-8	2.75				

BT MAS 403						40T	20-7595 (TSC)		Kit # TPS24BT
								50T	20-7165 (non-TSC)
						40T	22-7171 (TSC)		
								50T	22-7170 (non-TSC)
A	B	C	D	E					
40T	2.57	2.48	.65	M16X2	1.75				
50T	4.00	3.94	.91	M24X3	2.75				

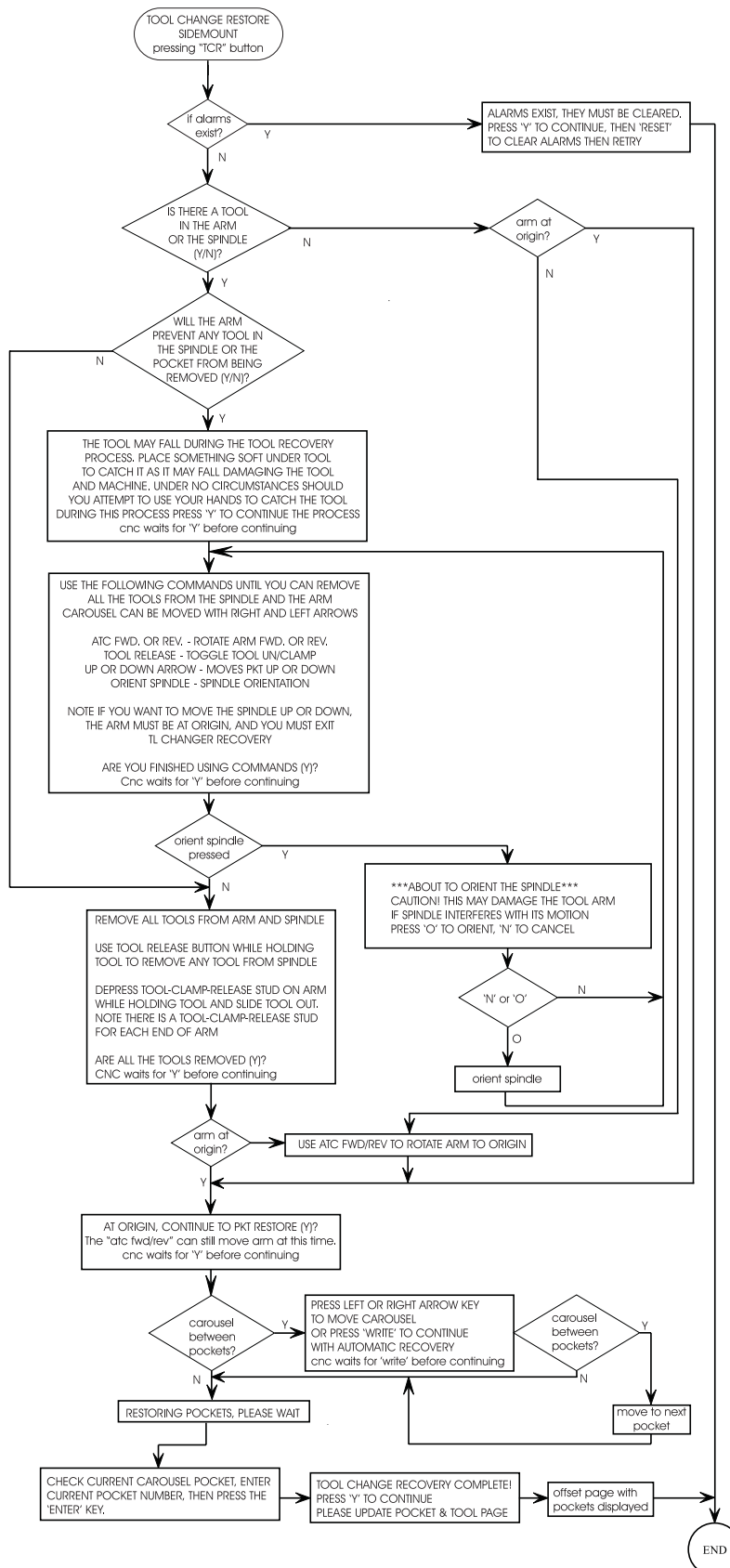
DIN-69871 (MIKRON) ISO-7388											
						40T				Kit # TPS24E	
						40T				Kit # PS24E	
						50T				Kit # TPS24E50	
						50T				Kit # PS24E50	
	A	B	C	D	E						
40T	2.69	2.50	.44	M16X2	1.75						
50T	4.00	3.84	.44	M24X3	2.75						

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.

TOOL CHANGER LUBRICATION

Place lubricating grease on the outside edge of the guide rails of the tool changer and run through all tools.

SIDE MOUNT TOOL CHANGER RECOVERY FLOW CHART



SIDEMOUNT MILL TOOL RECOVERY
SCREEN TEXT = UPPER CASE
ACTIONS - DECISIONS = LOWER CASE
S = SITUATION
M = NEXT MODE

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.

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. Each 1/2 revolution of the Geneva mechanism moves the tool turret one tool position forward or backward.

Note: This motor should never be disassembled.

12.2 TOOL CLAMP/UNCLAMP

The tool holder drawbar 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 side of the spindle head. The two manual buttons only operate in MDI or JOG modes.

TOOL CLAMP/UNCLAMP AIR SOLENOIDS

A single solenoid controls the air pressure to release the tool clamp. This corresponds to relay K15. When the relay is activated, 115V AC is applied to the solenoid. This applies air pressure to release the tool. Relay K15 is on the I/O PCB. Circuit breaker CB4 will interrupt power to this solenoid.

TOOL CLAMP/UNCLAMP SENSE SWITCHES

There are two switches used to sense the position of the tool clamping mechanism. They are both normally closed and one will activate at the end of travel during unclamping and the other during clamping. When both switches are closed, it indicates that the draw bar is between positions.

A tool change operation will wait until the unclamped switch is sensed before the Z-axis pulls up from the tool. This prevents any possibility of breaking the tool changer or its support mounts.

The diagnostic display can be used to display the status of the relay outputs and the switch inputs.

The Precharge and Through the Spindle Coolant system applies low air pressure and releases the clamped switch (with 40 taper spindle only).

REMOTE TOOL UNCLAMP SWITCH

The Remote Tool Unclamp switch is mounted on the side 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.

12.3 SPINDLE OPERATION

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!** When using the Through the Spindle Coolant option, the maximum spindle speed is 7500 RPM (5000 RPM for 50 taper spindles).

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. Spindle speed accuracy is best at the higher speeds and in low gear.

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

The spindle is hardened and ground to the precise tool holder dimensions providing an excellent fit to the holder.

SPINDLE WARM-UP / BREAK-IN

Spindle Warm-up Program

All spindles, which are to be run at speeds above 6000 RPM for extended periods of time, must be warmed up prior to use. A 20 minute warm-up program has been supplied with the machine, which will bring the spindle up to speed slowly and allow the spindle to thermally stabilize. The program number is O02020 (Spindle Warm-up Program).

Spindle Break-in Program

All spindles, which have been idle for more than 4 days, must be thermally cycled prior to running above 6000 RPM. This will prevent possible overheating of spindle due to settling of lubrication. A program has been supplied with the machine for spindle break-in at initial machine installation and for long periods of machine down time. The program number is O02021 (Spindle Break-in Program). Cycle Time: 2 hours

These programs can be used for all spindle types. For machines equipped with 7,500 RPM or 10,000 RPM spindles run the above programs with spindle speed override at 100%. For machines equipped with 15,000 RPM spindle, run spindle speed override at 150%. For machines equipped with a 50 taper spindle, run spindle speed override at 50%.

If either of these programs were not supplied with the machine, contact your dealer for a copy.

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.

On machines equipped with a **Haas vector drive**, orientation is performed electrically and no shot pin or solenoid is required for locking the motor in place. Orientation of the spindle is automatically performed for tool changes and can be programmed with M19 commands. Orientation is performed by turning the spindle until the encoder reference is reached, the spindle motor holds the spindle locked in position. If the spindle is oriented and locked, commanding spindle forward or reverse will release the lock.

SPINDLE ORIENTATION LUBRICATION

The spindle orientation mechanism does not require regular lubrication.

SPINDLE ORIENTATION AIR SOLENOID (WITH SHOT PIN)

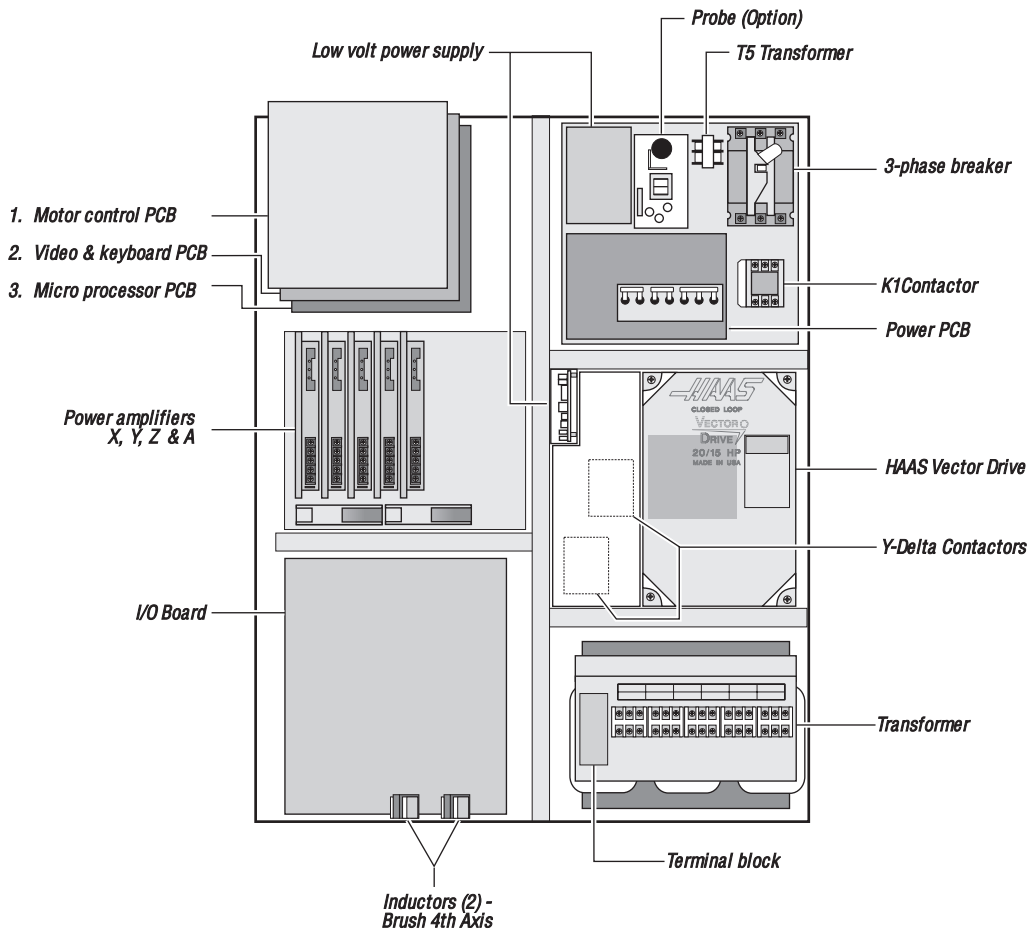
A solenoid controls the air valve supplying pressure to the orientation lock pin. The diagnostic display can be used to display the status of the relay output and the switch inputs. Circuit breaker CB4 will interrupt power to this solenoid.

SPINDLE ORIENTATION SEQUENCE

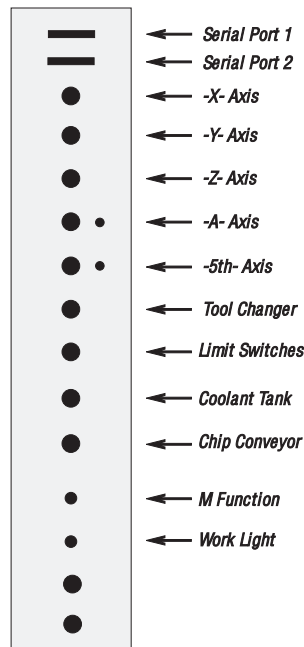
When spindle orientation is commanded, the following sequence of operations occurs:

- 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) **(Vector drive only)** Spindle encoder rotates past a reference mark,
- 6) **(Vector drive only)** The spindle drive stops and holds the spindle position at a parameter distance from the reference mark,
- 7) Command spindle lock air solenoid active,
- 8) Pause until spindle locked status is active and stable,
- 9) If not locked after time-out time, alarm and stop.

12.4 CONTROL PANEL



Control cabinet general overview.



Connectors on side of control cabinet.

12.5 SERVOS BRUSH / BRUSHLESS

SERVO ENCODERS

Haas machines are equipped with brushless motors, which provides for better performance, and no maintenance. In addition to the performance differences, these machines differ from brush type machines, which have already been discussed, in the following areas:

The brushless motors have 8192 line encoders built in, which result in differences in acceleration parameters 7, 21, 35, 49 and 157. The exponential accel/decel time is set by parameters 115, 116 and 168. "In Position" parameters 101, 102, 103, 104 and 165 also affect brushless motors.

The motor controller board has a dedicated processor which does all the servo control algorithm.

There is no servo distribution board anymore, therefore there is no CHARGE light present. Care should still be taken however, since there are high voltages present on the amplifiers, even when power is shut off. The high voltage comes from the spindle drive, which does have a CHARGE light.

The servo drive cards are replaced by Brushless Servo Amplifiers, and are controlled differently.

A low voltage power supply card is added to the servo drive assembly to supply the low voltage requirement to the amplifiers.

The CNC software is version 9.xx.

The user interface and motion profiling have not changed however, and the user should not see any functional differences between a brush type machine and a brushless machine.

SERVO CHARACTERISTICS

Servo characteristics are explained in detail in the previous chapter. The following is an example of how to achieve 130 inches/minute.

The exponential accel/decel time constant is set by Parameters 113, 114, 115, 116 and 168. It has units of 0.0001 seconds. The speed limit at which exponential accel/decel is not available is defined by the relationship between Parameters 7 and 113 (for the X-axis). Thus if Parameter 7 is 8000000 steps/sec/sec and Parameter 113 is 375 (0.0375 seconds); the maximum velocity for accurate interpolation should be:

$$8000000 \times 0.0375 = 300000 \text{ steps/second}$$

For an 8192 line encoder and 6 mm screw, this would be:

$$60 \times 300000 / 138718 = 130 \text{ inches/minute}$$

SERVO AMPLIFIERS

The brushless servo amplifier is a PWM based current source. The PWM outputs control the current to a three phase brushless motor. The PWM frequency is either 12.5 KHz or 16 KHz. The amplifiers are current limited to 30 amps peak (45A peak for a medium amplifier. However there are fuse limits both in hardware and software to protect the amplifiers and motors from over current. The nominal voltage for these amplifiers is 320 volts. Therefore the peak power is about 9600 watts or 13 H.P. The amplifiers also have short circuit, over temperature and over voltage protection.

There is a 15 amp (20A for a medium amplifier) supply fuse for failure protection. This fuse is relatively slow, therefore it can handle the 30 amp peak. Current limit to the motor is controlled by software.

Commands to the amplifier are +/-5 volts current in two legs of the motor and a digital enable signal. A signal from the amplifier indicates drive fault or sustained high current installed motor.

The connectors on the amplifiers are:

+H.V.	+320 volts DC
-H.V.	320 volts return
A	motor lead phase A
B	motor lead phase B
C	motor lead phase C
J1	Three pin Molex connector used for +/-12 and GND.
J2	Eight pin Molex connector used for input signals.

12.6 INPUT/OUTPUT ASSEMBLY

The IOPCB contains a circuit for electronically turning the tool changer power on and off. This prevents any arcing of the tool changer relays and increases their life tremendously. This includes an adjustable current limit to the tool changer. Potentiometer R45 adjusts the current limit to the tool changer motors. R45 should be set to limit current to between four and six amps.

The IOPCB also contains a circuit for sensing a ground fault condition of the servo power supply. If more than 0.5 amps is detected flowing through the grounding connection of the 160V DC buss, a ground fault alarm is generated and the control will turn off servos and stop.

Relay K6 is for the coolant pump 230V AC. It is a plug-in type and is double-pole. Relays K9 through K12 are also plug-in types for controlling the tool changer.

The Input/Output Assembly consists of a single printer circuit board called the IOPCB.

The connectors on the IOPCB are:

P1	16-pin relay drivers from MOCON 1 to 8 (510)
P2	16-pin relay drivers from MOCON 9 to 16 (520)
P3	16-pin relay drivers from MOCON 17 to 24 (M21-M24) (540)
P4	34-pin inputs to MOCON (550)
P5	Servo power on relay 1-1 (110)
P6	230V AC from CB3 (930)
P7	230V AC to coolant pump (940)
P8	Auto-off relay 1-7 (170)
P9	Spindle drive commands (710)
P10	Spindle fan and oil pump 115V AC (300)
P12	115V AC to spindle head solenoids (880A)
P13	Tool changer status inputs (820)
P14	Low TSC(900)
P15	Spindle head status inputs (890)
P16	Emergency stop input (770)
P17	Low Lube input (960)
P18	Over Voltage Input (970)
P19	Low Air Input (950)
P20	Overheat input (830)
P21	Spindle drive status inputs (780)
P22	M-FIN input (100)
P23	Remote Unclamp input (tool release) (190)
P24	Spare 2 (790)
P25	Spare 3 (200)
P26	Spare terminals for M21 to M24
P27	Door lock (1040)
P28	115V AC from CB4 (910)
P29	A-axis brake solenoid output (390)
P30	Tool changer shuttle motor output (810A)
P31	230 VAC for Chip Conveyor (160)
P33	115V AC three-phase input from power supply assembly (90)
P34	115V AC to CRT (90A)
P35	115V AC to heat exchanger (90B)

P36	115V AC to CB4 (90C)
P37	115V AC spare (870)
P38	Door open (1050)
P39	Tool changer turret motor output (810)
P40	(770A) A/B
P43	Ground fault sense signal input (1060) Axis Brake
P44	5TH axis brake (319)
P45	HTC Shuttle
P46	Chip Conveyor (140)
P47	Skip input signal (1070)
P48	spare 1
P49	spare 2
P50	Spigot Motor (200)
P51	16 PIN Relay drivers 17-24 (530)
P52	spare 1
P53	Spigot Sense (180)
P54	Servo Brake (350)
P55	Red/green lights (280)
P56	Thru spindle coolant pump (940A)
P57	115V spare
P58	115V spare

12.7 TWO-SPEED GEAR TRANSMISSION

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.

GEAR BOX LUBRICATION

Gear Box: Mobil DTE 25 oil.

The gear box uses an oil sump and is cooled by gear oil. The VF-0/E/OE does not have a gearbox and is air-cooled.

GEAR BOX AIR SOLENOIDS

There is a double solenoid valve controlling air to the gear box shifter. This solenoid sends air to select either the high gear or the low gear. When power is removed from the solenoids, the valve remains in its last state. Air is always required to ensure the gears are held in either high or low gear. Circuit breaker CB4 will interrupt power to these solenoids. Power is left on the solenoid which is commanded last.

On machines equipped with a **50 taper spindle**, an electric motor drives the gearbox shifter into high or low gear.

GEAR BOX SENSE SWITCHES

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".

GEAR CHANGE SEQUENCE

When a gear change is performed, the following sequence of events occurs:

- 1) If the spindle is turning, it is commanded to stop,
- 2) Pause until spindle is stopped,
- 3) Gear change spindle speed is commanded forward,
- 4) Pause until spindle is at speed,
- 5) Command high or low gear solenoid active,
- 6) Pause until in new gear or reversal time,
- 7) Alarm and stop if max gear change time elapsed,
- 8) If not in new gear, reverse spindle direction,
- 9) Turn off high and low gear solenoids

12.8 CONTROL PANEL

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.

POWER ON/OFF SWITCHES

The POWER ON switch engages the main contactor. The on switch applies power to the contactor coil and the contactor thereafter maintains power to its coil. The POWER OFF switch interrupts power to the contactor coil and will always turn power off. POWER ON is a normally open switch and POWER OFF is normally closed. The maximum voltage on the POWER ON and POWER OFF switches is 24V AC and this voltage is present any time the main circuit breaker is on.

SPINDLE LOAD METER

The Load meter measures the load on the spindle motor as a percentage of the rated continuous power of the motor. There is a slight delay between a load and the actual reflection of the meter. The eighth A-to-D input also provides a measure of the spindle load for cutter wear detection. The second page of diagnostic data will display % of spindle load. The meter should agree with this display within 5%. The spindle drive display #7 should also agree with the load meter within 5%.

There are different types of spindle drive that are used in the control. They are all equivalent in performance but are adjusted differently.

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.

KEYBOARD BEEPER

There is a speaker inside the control panel that is used as an audible response to pressing keyboard buttons and as a warning beeper. The beeper is a one kHz signal that sounds for about 0.1 seconds when any keypad key, CYCLE START, or FEED HOLD is pressed. The beeper also sounds for longer periods when an auto-shut down is about to occur and when the "BEEP AT M30" setting is selected.

If the beeper is not audible when buttons are pressed, the problem could be in the keypad, keyboard interface PCB or in the speaker. Check that the problem occurs with more than one button and check that the speaker volume is not turned down.

12.9 MICROPROCESSOR ASSEMBLY

The microprocessor assembly is in the rear cabinet at the top left position. It contains three large boards. They are: microprocessor, the keyboard and the MOCON. All three boards of the processor assembly receive power from the low voltage power supply. The three PCB's are interconnected by a local buss on dual 50-pin connectors. At power-on of the control, some diagnostic tests are performed on the processor assembly and any problems found will generate alarms 157 or 158. In addition, while the control is operating, it continually tests itself and a self test failure will generate Alarm 152.

MICROPROCESSOR PCB (68ECO30)

The Microprocessor PCB contains the 68ECO30 processor running at 40 MHz, one 128K EPROM; between 256K and 8MB of CMOS RAM and between 512K and 1MB of FAST STATIC RAM. It also contains a dual serial port, a five year battery to backup RAM, buffering to the system buss, and eight system status LED's.

Two ports on this board are used to set the point at which an NMI* is generated during power down and the point at which RESET* is generated during power down.

The eight LED's are used to diagnose internal processor problems. As the system completes power up testing, the lights are turned on sequentially to indicate the completion of a step. The lights and meanings are:

- | | |
|-------------|---|
| +5V | +5V logic power supply is present. (Normally On)
If this light does not come on, check the low voltage power supply and check that all three phases of 230V input power are present. |
| HALT | Processor halted in catastrophic fault. (Normally Off)
If this light comes on, there is a serious problem with the processor PCB. Check that the EPROM is plugged in. Test the card with the buss connectors off. |
| POR | Power-on-reset complete. (Normally On)
If this light does not come on, there is a serious problem with the processor PCB. Check that the EPROM is plugged in. Test the card with the buss connectors off. |
| SIO | Serial I/O initialization complete. (Normally On)
If this light does not come on, there is a problem with the serial ports. Disconnect anything on the external RS-232 and test again. |
| MSG | Power-on serial I/O message output complete. (Normally On)
If this light does not come on, there is a problem with serial I/O or interrupts. Disconnect anything on the external RS-232 and test again. |
| CRT | CRT/VIDEO initialization complete. (Normally On)
If this light does not come on, there is a problem communicating with the VIDEO PCB. Check the buss connectors and ensure the VIDEO PCB is getting power. |

PGM Program signature found in memory.(Normally On)

If this light does not come on, it means that the main CNC program package was not found in memory or that the auto-start switch was not set. Check that switch S1-1 is on and the EPROM is plugged in.

RUN Program Running Without Fault Exception. (Normally On)

If this light does not come on or goes out after coming on, there is a problem with the microprocessor or the software running in it. Check all of the buss connectors to the other two PCB's and ensure all three cards are getting power.

There is 1 two-position DIP switch on the processor PCB labeled S1. Switch S1-1 must be ON to auto-start the CNC operational program. If S1-1 is OFF, the PGM light will remain off.

Switch S2-1 is used to enable FLASH. If it is disabled it will not be possible to write to FLASH.

The processor connectors are:

J1	Address buss
J2	Data buss
J4	Serial port #1 (for upload/download/DNC) (850)
J5	Serial port #2 (for auxiliary 5th axis) (850A)
J3	Power connector
J6	Battery

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.

VIDEO KEYBOARD WITH FLOPPY

The VIDEO and KB PCB generates the video data signals for the monitor and the scanning signals for the keyboard. In addition, the keyboard beeper is generated on this board. There is a single jumper on this board used to select inverse video. The video PCB connectors are:

P1	LOW VOLTAGE POWER SUPPLY PCB (860)
P3*	KEYBOARD INFO. (700)
P4	ADDRESS BUSS
P5	DATA BUSS
P10	FLOPPY DR. POWER
P11	SPARE
P12	FLOPPY DR. SIGNAL
P13	VIDEO SIGNAL (760)
J9	RS422 B
J13	SERIAL DATA (850)

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 signal (660)
P7	Y-axis encoder, Z, home, and overheat signal (670)
P8	Z-axis encoder, Z, home, and overheat signal (680)
P9	A-axis encoder, Z, home, and overheat signal (690)
P10	32 discrete inputs (550)
P11	Relay drives 1 to 8 (510)
P12	Relay drives 9 to 16 (520)
P13	Relay drives 17 to 24 (530)
P14	Relay drives 25 to 32 (540)
P15	Power connector (+5,+12+)
P16	D-to-A output and -12V DC (720)
P17	A-to-D inputs for DC buss voltage (980)
P18	Jog Crank input and aux 1,2 (750)
P19	Address buss
P20	Spindle encoder inputs (1000)
P21	A-to-D input for spindle temperature (1020)
P22	A-to-D input for spindle load monitor (730B)
P24	Home switch inputs X, Y, Z (990)

MOTOR CONTROLLER (MOCON) - BRUSHLESS

The brushless machining centers are equipped with a microprocessor based brushless motor controller board (MOCON) that replaces the motor interface in the brush type controls. It runs in parallel with the main processor, receiving servo commands and closing the servo loop around the servo motors.

In addition to controlling the servos and detecting servo faults, the motor controller board, (MOCON), is also in charge of processing discrete inputs, driving the I/O board relays, commanding the spindle and processing the jog handle input. Another significant feature is that it controls 6 axes, so there is no need for an additional board for a 5 axis machine.

P1 Data Buss

- P2 X amplifier control and fault sensing (610)
- P3 Y amplifier control and fault sensing (620)
- P4 Z amplifier control and fault sensing (630)
- P5 A amplifier control and fault sensing (640)
- P32B amplifier control and fault sensing (640B)
- P33C amplifier control and fault sensing (640C)
- P6 X encoder input (660)
- P7 Y encoder input (670)
- P8 Z encoder input (680)
- P9 A encoder input (690)
- P30B encoder input (690B)
- P31C encoder input (690C)
- P18Jog encoder input (750)
- P20Spindle encoder input (1000)
- P10Inputs from I/O board (550)
- P11 I/O relays K1-8 (510)
- P12I/O relays K9-16 (520)
- P13I/O relays K17-24 (530)
- P14I/O relays K25-32 (540)
- P15Low Voltage Power (860)
- P16Spindle command output (720)
- P19Address bus
- P24Axis home switches (990)

12.10 SPINDLE DRIVE ASSEMBLY

The spindle drive is located in the main cabinet on the right side and halfway down. It operates from three-phase 200 to 240V AC. It has a 10 (or 20) H.P. continuous rating, and a 15 (or 30) H.P. one-minute rating. The spindle drive is protected by CB1. 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.

HAAS VECTOR DRIVE

The Haas vector drive is a current amplifier controlled by the Mocon software, using the C axis output. The vector drive parameters are a part of the machine parameters and are accessible through the Haas front panel. The spindle encoder is used for the closed loop control and spindle orientation, as well as rigid tapping if the option is available. Spindle speed is very accurate, since this is a closed loop control and the torque output at low speeds is superior to non vector drive spindles.

12.11 RESISTOR ASSEMBLY

The Resistor Assembly is located on top of the control cabinet. It contains the servo and spindle drive regen load resistors.

SPINDLE DRIVE REGEN RESISTOR

A resistor bank is used by the spindle drive to dissipate excess power caused by the regenerative effects of decelerating the spindle motor. If the spindle motor is accelerated and decelerated again in rapid succession repeatedly, this resistor will get hot. In addition, if the line voltage into the control is above 255V, this resistor will begin to heat. This resistor is overtemp protected at 100° C. At that temperature, an alarm is generated and the control will begin an automatic shutdown. If the resistor is removed from the circuit, an alarm may subsequently occur because of an overvoltage condition inside the spindle drive.

SERVO DRIVE REGEN RESISTOR

A 25-ohm, 300-watt resistor is used by the brush-type servo drives to dissipate excess power caused by the 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.

OVERHEAT SENSE SWITCH

There is an over-temperature sense switch mounted near the above-mentioned regen resistors. This sensor is a normally-closed switch that opens at about 100° C. It will generate an alarm and all motion will stop. After the time period, specified by parameter 297, of an overheat condition, an automatic shutdown will occur in the control.

12.12 POWER SUPPLY ASSEMBLY

All power to the control passes through the power supply assembly. It is located on the upper right corner of the control cabinet.

MAIN CIRCUIT BREAKER CB1

Circuit breaker CB1 (see chart for ratings) is primarily used to protect the spindle drive and to shut off all power to the control. The locking On/Off handle on the outside of the control cabinet will shut this breaker off when it is unlocked. A trip of this breaker indicates a **SERIOUS** overload problem and should not be reset without investigating the cause of the trip. The full circuit breaker ratings are listed in the following chart.

CIRCUIT BREAKER (CB1) AMP RATING		
HP RATING	195-260 VAC	354-488 VAC
20 - 15	40 AMP	20 AMP
40 - 30	80 AMP	40 AMP

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, auxiliary contacts 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.

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.

POWER PCB (POWER)

The low voltage power distribution and high voltage fuses and circuit breakers are mounted on a circuit board called the POWER PCB. The following connectors are on it:

P1	Brings incoming 230 VAC (460 VAC) 3-phase 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	Primary and secondary connections to transformer T5.
P6	230V AC from CB3 to coolant pump (930)
P7	115V AC from CB4 to IOPCB for solenoids (910)
P8	115V AC from IOPCB to CB4 for solenoids (90)
P9	+5/+12/-12 GND from low volt supply to logic boards (to MOCON) (860).
P10	+5/+12/Gnd from low volt supply to logic boards (860)
P11	+5/+12/Gnd from low volt supply to logic boards (860)
P12	+5/+12/Gnd from low volt supply to logic boards (860)
P13	+5/+12/Gnd from low volt supply to logic boards (860)
P14	12V AC to operator's lamp switch (800A) (not used)
P15	230V AC from main transformer to CB3 (70)
P16	Low voltage power from power supply (not used)
P17	+12V DC option connector (to MCD relay board) (860A)
P18	Not used
P19	Primary and Secondary connections to worklight transformer T4 (290) (not used)
P20	115V AC to low voltage power supply
P21	-12V DC option connector
P22	-12V DC option connector (to MOTIF)
P23	230 VAC from main transformer to CB5 (70A)
P24	230 VAC from CB3 to TSC coolant pump (930A)
P26	+5/+12/-12/ GND from low volt supply to logic boards (to processor) (860)
P27	+5/+12/-12/ GND from low volt supply to logic boards (860)
P28	+5/+12/-12 GND low volt power from power supply
P29	+5/-5 GND low volt power from power supply
P30	12V AC to operator's light (800) (not used)
P31	+12V option connector (to stack fan)(860A)
TB1	115VAC 3-phase from main transformer (94,95,96)
TB2	115 VAC 3-phase to IOPCB (91,92,93)

SECONDARY CIRCUIT BREAKERS

The following circuit breakers are located on the Power supply assembly.

CB2 controls the 3-phase 115volt distribution. It can be tripped only if there is a short in the control cables or on the IOPCB.

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.

CB5 Controls power to the TSC coolant pump only. It can be tripped by an overload of the TSC coolant pump motor or a short in the wiring to the motor.

POWER-UP LOW VOLTAGE CONTROL TRANSFORMER (T5)

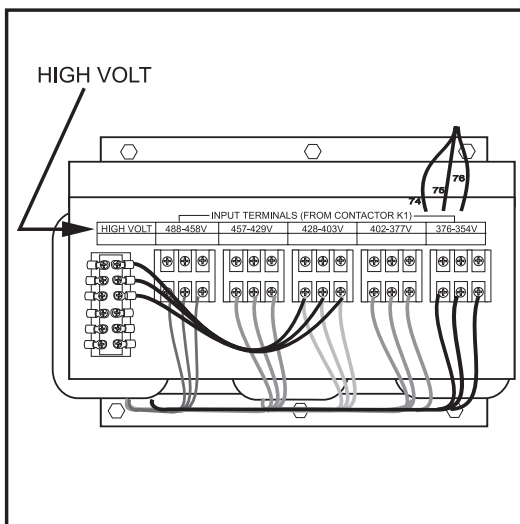
The low voltage control transformer, T5, supplies power to the coil of the main contactor K1. It guarantees that the maximum voltage leaving the Power Supply assembly when power is off is 12V AC to earth ground. It is connected via P5 to the POWER PCB.

12.13 POWER TRANSFORMER ASSEMBLY (T1)

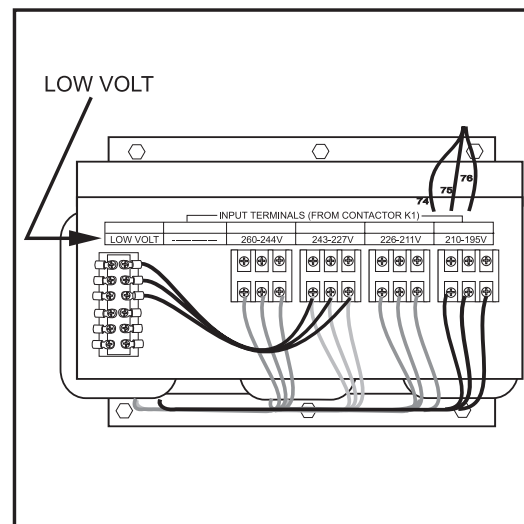
The power transformer assembly is used to convert three-phase input power (50/60Hz) to three phase 230V and 115V power. Two different transformers are used depending on the input voltage range. The low voltage transformer has four different input connections to allow for a range of voltages from 195 V RMS to 260 V RMS. The high voltage transformer has five different input connections and will accept a range of voltages from 354V RMS to 488 V RMS.

The 230 V is used to power the spindle drive, which also develops the 325 VDC power for the axis servo amplifiers. The 115 V is used by the video monitor, solenoids, fans and pumps, in addition to supplying power to the main LVPS used by the control electronics.

The transformer assembly is located in the lower right hand corner of the main cabinet. Besides the high/low voltage variations, two different power levels are available depending on the spindle motor used. The small and large transformers have power ratings of 14 KVA and 28 KVA, respectively. They are protected by the main circuit breaker to the levels shown in the preceding table.



Transformer with 354-488V range



Transformer with 195-260V range

PRIMARY CONNECTION TO T1

Input power to T1 is supplied through CB1, the 40 amp three-phase main circuit breaker. Three-phase 230 to T1 is connected to the first three terminals of TB10.

VOLTAGE SELECTION TAPS

There are four labeled plastic terminal blocks. Each block has three connections for wires labeled 74, 75, and 76. Follow the instructions printed on the transformer.

SECONDARY CONNECTION TO T1

The secondary output from T1 is 115V AC three-phase CB2 protects the secondary of transformer T1 and is rated at 25 amps.

OPTIONAL 480 TRANSFORMER

Voltage Selection Taps for the 480 Transformer:

Right to left:

354 to 376
377 to 402
403 to 428
429 to 457
458 to 488*

* 480 V transformer has additional terminal block

12.14 FUSES

Brush Motors

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 two fuses FU4 and FU6 are not used.

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	" "
(not used) Lamp	AGC	½	250V	" lower left
FU1	AGC	½	250V	SDIST pcb, right center
FU2	AGC	½	250V	" "
FU3	AGC	5	250V	" top center
F1	ABC	20	250V	SDRIVER pcb's (X, Y, Z, A)
F2	ABC	20	250V	"
F3	ABC	10	250V	"
FU1	ABC	5	250V	I/O PCB
FU2	ABC	5	250V	I/O PCB
FU3	ABC	5	250V	I/O PCB
FU4	ABC	5	250V	I/O PCB

Brushless Motors

The brushless amplifier has one fuse, F1 15 amps. This fuse protects the amplifier itself from drastic damage. If this fuse is ever blown, the associated motor will stop. This will only happen if there is a failure of the amplifier 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 two fuses FU4 and FU6 are not used.



FUSE NAME	TYPE	RATING	VOLTAGE (amps)	LOCATION
FU1	AGC	½	250V	POWER pcb, upper right
FU2	AGC	½	250V	" "
FU3	AGC	½	250V	" "
(not used) Lamp	AGC	½	250V	POWER pcb, lower left
FU1	ABC	5	250V	I/O PCB
FU2	ABC	5	250V	I/O PCB
FU3	ABC	5	250V	I/O PCB
FU4	ABC	5	250V	I/O PCB
F1	ABC	15	250V	Amplifier (X,Y,Z,A,B)

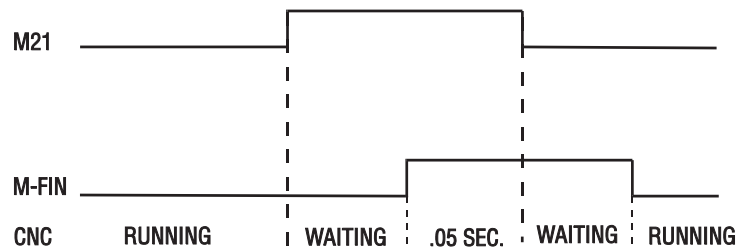
12.15 SPARE USER M CODE INTERFACE

The M code interface uses outputs M21-23 and one discrete input circuit. M codes M21 through M23 will activate relays labelled M21-23. 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.

Note: If the optional M code relay board is installed, relays M21-28 become available on the secondary board. These relays will be controlled by outputs M21-28.

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:

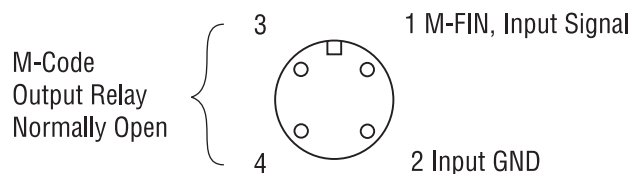


The Diagnostic Data display page may be used to observe the state of these signals.

M FUNCTION RELAYS

The IOPCB contains three relays (M21-M23) and the optional M code relay board contains eight (M21-M28), either one of these groups of relays may be available to the user. 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.

Note: If the optional M code relay board is installed, the relays on the IOPCB are to be left unused.



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.

TURNING M FUNCTIONS ON AND OFF

The M code relays can also be separately turned on and off using M codes M51- M53 and M61- M63. M51 to M53 will turn on one of the relays and M61 to M63 will turn the relays off. M51 and M61 correspond to M21, etc.

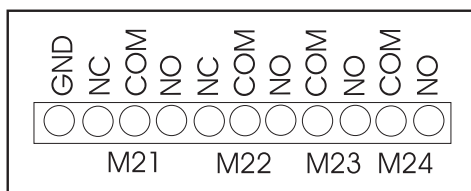
Note: If the M code relay board is installed M51-M58 will turn on the relays and M61-M68 will turn off the relays. M51 and M61 correspond to M21, etc. on the M code relay board.

WIRING THE RELAYS

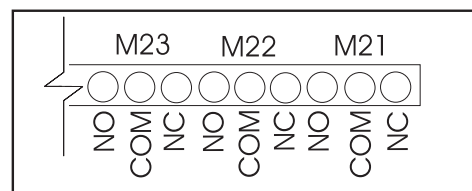
The relays are marked on both the IOPCB and the M code relay board, with their respective terminals forward of them. If the optional M code relay board is installed then the connections on the IOPCB are to be left unused as they are replaced by the relays on the optional board. Refer to the figures below, and the Probe Option figure in the Electrical Diagrams section for the terminal labeling. Maximum voltage for the relays is 125 VAC with a maximum amperage of 3 amps.

WARNING!

Power circuits and inductive loads must have snubber protection.



IOPCB Relays



M Code Relay Board

CAUTION! If a screw terminal is already in use DO NOT connect anything else to it. Call your dealer.

Note: Relay M24 on the IOPCB is reserved for Through the Spindle Coolant (AUXCLT).

12.16 LUBRICATION SYSTEM

The lubrication system is a resistance type system which forces oil through metering units at each of the 16 lubricating points within the machine. The system uses one metering unit at each of the lubricating points: one for each linear guide pad, one for each lead screw and one for spindle lubrication. A single oil pump is used to lubricate the system. The pump is powered only when the spindle and/or an axis moves. Once powered the pump cycles approximately 3.0 cc of oil every 30 minutes throughout the oil lines to the lube points. Every lube point receives approximately 1/16 of oil. The control monitors this system through an internal level switch in the reservoir and external pressure switch on the lube panel.

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.

12.17 SWITCHES

LAMP ON/OFF SWITCH

An on/off switch is supplied for the operator's lamp. It is located on the side of the operator's pendant.

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 routed through the front panel support arm and down through the top of the enclosure.

If the doors are open, you will not be able to start a program. Door Hold will not stop a tool change operation or a tapping operation, and will not turn off the coolant pump. Also, if the doors are open, the spindle speed will be limited to 750 RPM.

The Door Hold function can be temporarily disabled by turning Setting 51 **on**, if Parameter 57 bits DOOR STOP SP and SAFETY CIRC are set to zero, but this setting will return to OFF when the control is turned off.

LIMIT SWITCHES

There are a number of limit switches located on the VMC, and some are difficult to reach. Ensure the problem is the switch before beginning removal procedures. The following is a list of all switches, their general location, and a functional description:

CLAMP/UNCLAMP SWITCHES

[Tool Release Piston Assembly (2)]

There are two switches used to sense the position of the tool clamping mechanism. They are both normally closed and one will activate at the end of travel during unclamping and the other during clamping. When both switches are closed, it indicates that the draw bar is between positions.

A tool change operation will wait until the unclamped switch is sensed before the Z-axis pulls up from the tool. This prevents any possibility of breaking the tool changer or its support mounts.

The diagnostic display can be used to display the status of the relay outputs and the switch inputs.

SPINDLE ORIENT SWITCH

[Top rear of transmission]

Note: This switch does not exist on machines that have a Vector Drive.

A normally-open switch that is held closed is used to sense when the pin drops in to lock the spindle. When the pin drops the switch opens, indicating orientation is complete.

The normally-closed side of the same switch that is held open, is wired to the spindle drive and commands it into a "Coast Stop" condition. This is done to ensure the spindle motor is not powered when the pin is locking the spindle.

X, Y, AND Z LIMIT SWITCHES

Prior to performing an POWER UP/RESTART or an AUTO ALL AXES operation, there are no travel limits. Thus, you can jog into the hard stops in either direction for X, Y, or Z. After a ZERO RETURN has been performed, the travel limits will operate unless an axis hits the limit switch. When the limit switch is hit, the zero returned condition is reset and an AUTO ALL AXES must be done again. This is to ensure that if you hit the limit switch, you can still move the servo back away from it.

The limit switches are normally closed. When a search for zero operation is being performed, the X, Y, and Z axes will move towards the limit switch unless it is already active (open); then they will move away from the switch until it closes again; then they will continue to move until the encoder Z channel is found. This position is machine zero.

Auto search for zero in the Z-axis is followed by a rapid move from the limit switch position down to the tool change position. This makes the Z-axis a little different from the other axes. The position found with the limit switch is not machine zero but is the position used to pull tools out of the spindle. Machine zero for Z is below this by Parameter 64. Be careful during the Z zero search and stay clear of that rapid move.

What Can Go Wrong With Limit Switches?

If the machine is operated without connector P5, a LOW LUBE and DOOR OPEN alarm will be generated. In addition, the Home search will not stop at the limit switch and will instead run into the physical stops on each axis.

If the switch is damaged and permanently open, the zero search for that axis will move in the negative direction at about 0.5 in/min until it reaches the physical travel stops at the opposite end of travel.

If the switch is damaged and permanently closed, the zero search for that axis will move at about 10 in/min in the positive direction until it reaches the physical stops.

If the switch opens or a wire breaks after the zero search completes, an alarm is generated, the servos are turned off, and all motion stops. The control will operate as though the zero search was never performed. The RESET can be used to turn servos on but you can jog that axis only slowly.

TOOL CHANGER POSITION SWITCHES

[Inside of Tool Carriage (2)]

GENEVA WHEEL POSITION MARK

The turret rotation mechanism has a switch mounted so that it is activated for about 30° of travel of the Geneva mechanism. When activated, this switch indicates that the turret is centered on a tool position. This switch is normally closed. The diagnostic display will show this status of this input switch as "TC MRK". A "1" indicates the Geneva wheel is in position.

TOOL #1 SENSE SWITCH

The tool rotation turret has a switch that is activated when tool one is in position or facing towards the spindle. At POWER ON this switch can indicate that tool #1 is in the spindle. If this switch is not active at power-on, the first tool change will rotate the turret until the switch engages and then move to the selected tool. The diagnostic display will show the status of this input switch as "TOOL #1". A "1" indicates that tool #1 is in position.

SHUTTLE IN/OUT SWITCHES

[Tool Changer Holding Plate (2)]

Two switches are used to sense the position of the tool changer shuttle and the arm that moves it. One switch is activated when the shuttle is moved full travel inward and one is activated when it is full travel outward. These switches are normally closed so that both will be closed between in and out. The diagnostic display will show this status of the input switch. A "1" indicates the associated switch is activated or open.

TRANSMISSION HIGH/LOW GEAR POSITION SWITCHES

[Bottom of Gearbox Assembly (2)]

On machines with a two-speed transmission, there are two switches in the gear box used to sense the position of the gears. One switch indicates HIGH by opening and the other indicates LOW by opening. Between gears, both switches are closed indicating a between-gear condition. The diagnostic display shows the status of these switches and the CURNT COMDS display shows which gear is selected. If the switches indicate that the gear box is between gears, the display will indicate "No Gear".

Note: The Transmission High/Low Gear Position Switches are located at the bottom of the Gearbox Assembly and are extremely difficult to reach. Removal of this assembly is necessary to replace these switches. See Mechanical Service, for Spindle Motor and Transmission removal.

12.18 HYDRAULIC COUNTERBALANCE

The spindle head weight is balanced by the upward pull of a hydraulic cylinder. The hydraulic oil forces the piston to retract into the cylinder body. The oil is then pressurized by a nitrogen reservoir. The system is self contained and passive (no pump is required to maintain the lift). Normal Z-Axis of the gas/oil counter balance has the initial pressure to balance the weight at full system volume, plus an additional 50-75 psi overcharge for longevity.

12.19 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 210 for a complete list of alarms, their possible causes, and some corrective action.

If there is an electronics problem, the controller may not complete the power-up sequence and the CRT will remain blank. In this case, there are two sources of diagnostic data; these are the audible beeper and the LED's on the processor PCB. If the audible beeper is alternating a ½ second beep, there is a problem with the main control program stored in EPROM's on the processor PCB. If any of the processor electronics cannot be accessed correctly, the LED's on the processor PCB will or will not be lit.

If the machine powers up but has a fault in one of its power supplies, it may not be possible to flag an alarm condition. If this happens, all motors will be kept off and the top left corner of the CRT will have the message:

POWER FAILURE ALARM

and all other functions of the control will be locked out.

When the machine is operating normally, a second push of the PARAM/DGNOS key will select the diagnostics display page. The PAGE UP and PAGE DOWN keys are then used to select one of two different displays. These are for diagnostic purposes only and the user will not normally need them. The diagnostic data consists of discrete input signals, 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 described in the following section.

12.20 DISCRETE INPUTS / OUTPUTS

The inputs/outputs that are followed by an asterisk (*) are active when equal to zero (0).

DISCRETE INPUT

#	Name	#	Name
1000	TC Changer In	1016	Spindle Lock (Vector Drive: spare)
1001	TC Changer Out	1017	Spindle Fault (Vector Drive: spare)
1002	Tool One In Pos.	1018	Spindle Stopped* (Vector Drive: spare)
1003	Low TSC Pressure	1019	Spindle At Speed* (Vector Drive: spare)
1004	Tool In Position	1020	Low Trans Oil Prs
1005	Spindle High Gear	1021	Spare 1
1006	Spindle Low Gear	1022	Spare 2
1007	Emergency Stop	1023	Spare 3
1008	Door Switch	1024	Tool Unclmp Rmt*
1009	M Code Finish*	1025	Low Phasing 115V
1010	Over Voltage	1026	Spare 3A
1011	Low Air Pressure	1027	Spare 3B
1012	Low Lube Press.	1028	Ground Fault
1013	Regen Over Heat	1029	G31 Block Skip
1014	Draw Bar Open	1030	Spigot Position
1015	Draw Bar Closed	1031	Conveyr Overcmt

The inputs are numbered the same as the connections on the inputs printed circuit board.

DISCRETE OUTPUTS

#	Name	#	Name
1100	Powered Servos	1116	Move Spigot CW
1101	Spindle Forward (Vector Drive: spare)	1117	Move Spigot CCW
1102	Spindle Reverse (Vector Drive: spare)	1118	Pal Ready Light
1103	Reset Spind. Drv (Vector Drive: spare)	1119	TSC Purge
1104	Brake 4th Axis	1120	Unclamp Pre-Chrg
1105	Coolant Pump On	1121	HTC Shuttle Out (Air Drive Shuttle: Move shuttle in)
1106	Auto Power Off	1122	Brake 5TH Axis
1107	Spind. Motor Fan	1123	CE Door Lock
1108	Move T.C. In	1124	M21
1109	Move T.C. Out	1125	M22
1110	Rotate T.C. CW	1126	M23 (Air Drive Shuttle: Move Shuttle Out)
1111	Rotate T.C. CCW	1127	TSC Coolant
1112	Spindle Hi Gear	1128	Green Beacon On
1113	Spindle Low Gear	1129	Red Beacon On
1114	Unclamp Tool	1130	Enable Conveyor
1115	Lock Spindle (Vector Drive: spare)	1131	Reverse Conveyor

Note: If the machine is equipped with an APC the following inputs and outputs will change:

#	Name	#	Name
1009	Pallet Clamped SW	1121	PAL Clamp
1021	APC Door	1122	Door
1022	APC Pin CLR #1	1125	APC Motor
1023	APC Pin CLR #2	1126	Beeper
1026	APC PAL #2 Home		
1027	APC PAL #1 Home		

The second page of diagnostic data is displayed using the PAGE UP and PAGE DOWN keys. It contains:

INPUTS 2

Name	Name	Name
X Axis Z Channel	X Overheat	X Cable Input
Y Axis Z Channel	Y Overheat	Y Cable Input
Z Axis Z Channel	Z Overheat	Z Cable Input
A Axis Z Channel	A Overheat	A Cable Input
B Axis Z Channel	B Overheat	B Cable Input
X Home Switch	X Drive Fault	Spindle Z Channel
Y Home Switch	Y Drive Fault	
Z Home Switch	Z Drive Fault	
A Home Switch	A Drive Fault	
B Home Switch	B Drive Fault	

The following inputs and outputs pertain to the Haas Vector Drive. If it is not enabled, these will display a value of *. Otherwise, it will display a 1 or 0.

Spindle Forward
 Spindle Reverse
 Spindle Lock
 Spindle at Speed*
 Spindle Stopped
 Spindle Fault
 Spindle is Locked
 Spindle Over heat
 Spindle Cable Fault

The following Discrete Inputs / Outputs 2 are available when parameter 278 SMNT BIT 1,2 or 3 (Side Mount Tool Changer) is set and parameter 209 MCD RLY BRN (M-Code relay board) is ON.

DISCRETE INPUTS 2

Name	Name
Spare Input 4A	Spare Input 8A
Spare Input 4B	Serp. Shot Pin*
Spare Input 5A	Motor Stop
Spare Input 5B	Origin
Spare Input 6A	Clamp / Unclamp
Spare Input 6B	Serp. Cam Count
Spare Input 7A	Spare Input 11A
Spare Input 7B	Spare Input 11 B

DISCRETE OUTPUTS 2

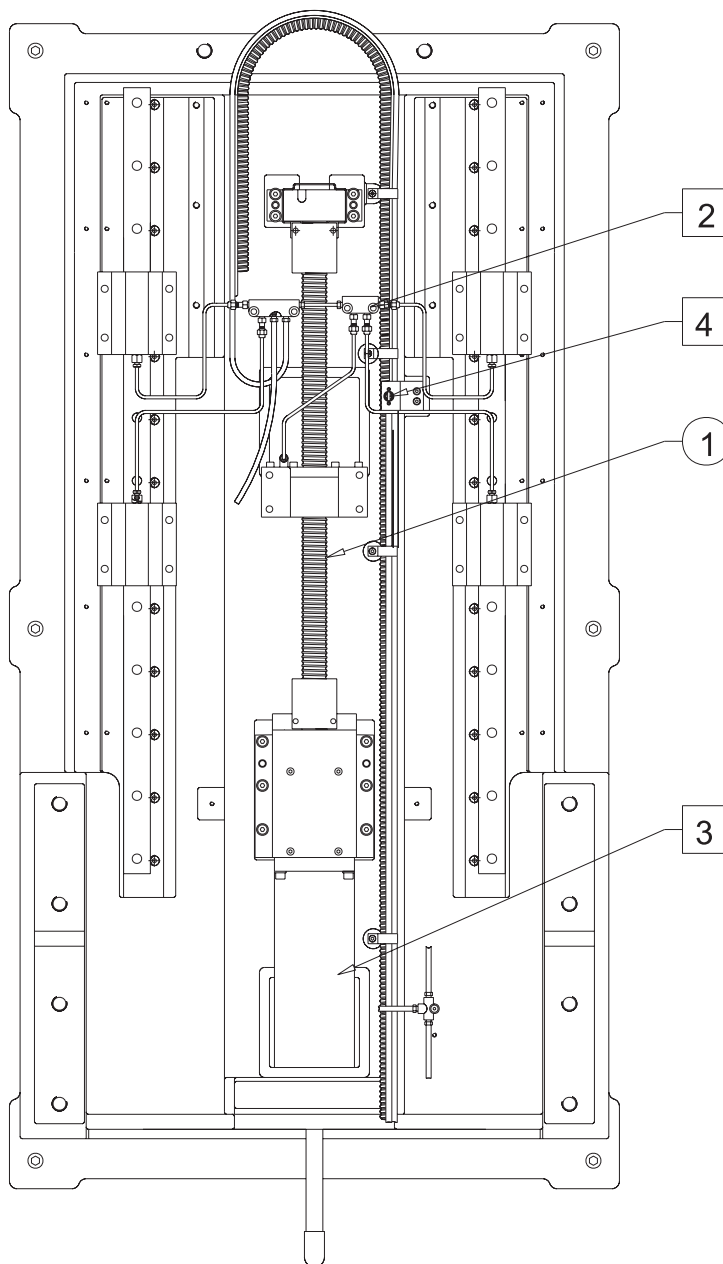
Name	Name
Spare Output 40	Spare Output 48 (SMTC: Serp. ATC Enable)
Spare Output 41	Spare Output 49 (SMTC: Serp. ATC Rev.)
Spare Output 42	Spare Output 50 (SMTC: Serp. Carsl CW)
Spare Output 43	Spare Output 51 (SMTC: Serp. Carsl CCW)
Spare Output 44	Spare Output 52 (SMTC: Serp. Carsl Ena.)
Spare Output 45	Spare Output 53
Spare Output 46	Spare Output 54
Spare Output 47	Spare Output 55

ANALOG DATA

Name	Description
DC BUSS	Voltage from Haas Vector Drive (if equipped)
uP TEMP	Microprocessor enclosure temperature (displayed only when Parameter 278 bit "uP ENCL TEMP" is set to 1)
SP LOAD	Spindle load in %
SP SPEED	Spindle RPM CW or CCW
RUN TIME	Machine total run time
TOOL CHANGES	Number of tool changes
VER X.XXX	Software version number
MOCON	MOCON software version
YY/MM/DD	Today's date
MDL HS__	Machine model

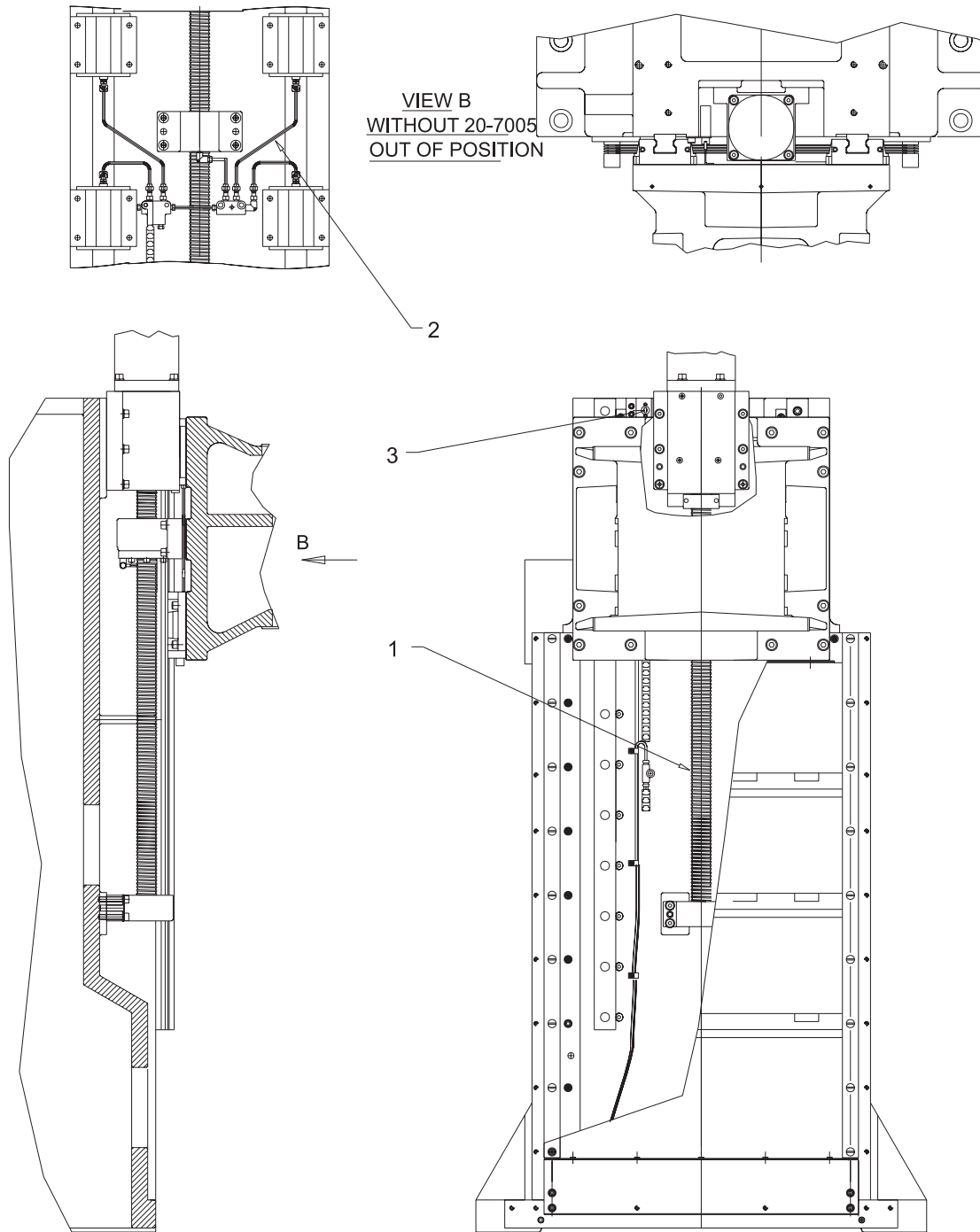


ASSEMBLY DRAWINGS



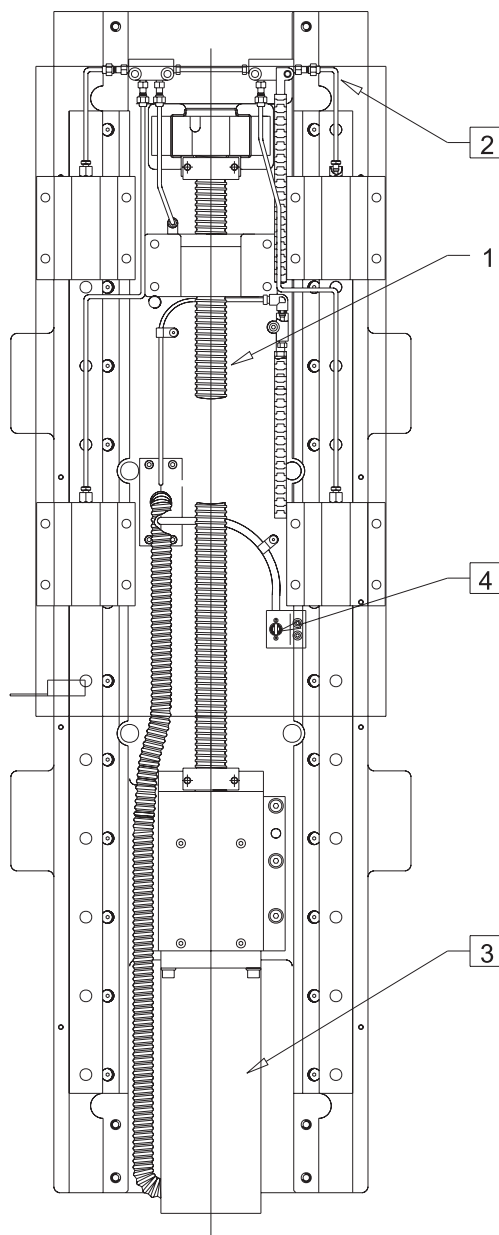
- 1 - 30-0157 - LEAD SCREW ASSEMBLY
- 2 - 30-7524 - BASE OIL LINE ASSEMBLY
- 3 - 32-1600 - Y AXIS MOTOR ASSEMBLY
- 4 - 32-2030 - TELEMECHANIQUE SWITCH ASSEMBLY

VF-0/1 Base

**30-1000C VERTICAL COLUMN ASSEMBLY**

- 1 - 30-0157 - LEAD SCREW ASSEMBLY
- 2 - 30-7525 - COLUMN OIL LINE ASSEMBLY
- 3 - 32-2040 - TELEMECHANIQUE SWITCH ASSEMBLY

VF-0/1 Column

**30-1000S SADDLE ASSEMBLY**

1 - 30-0157 - LEAD SCREW ASSEMBLY

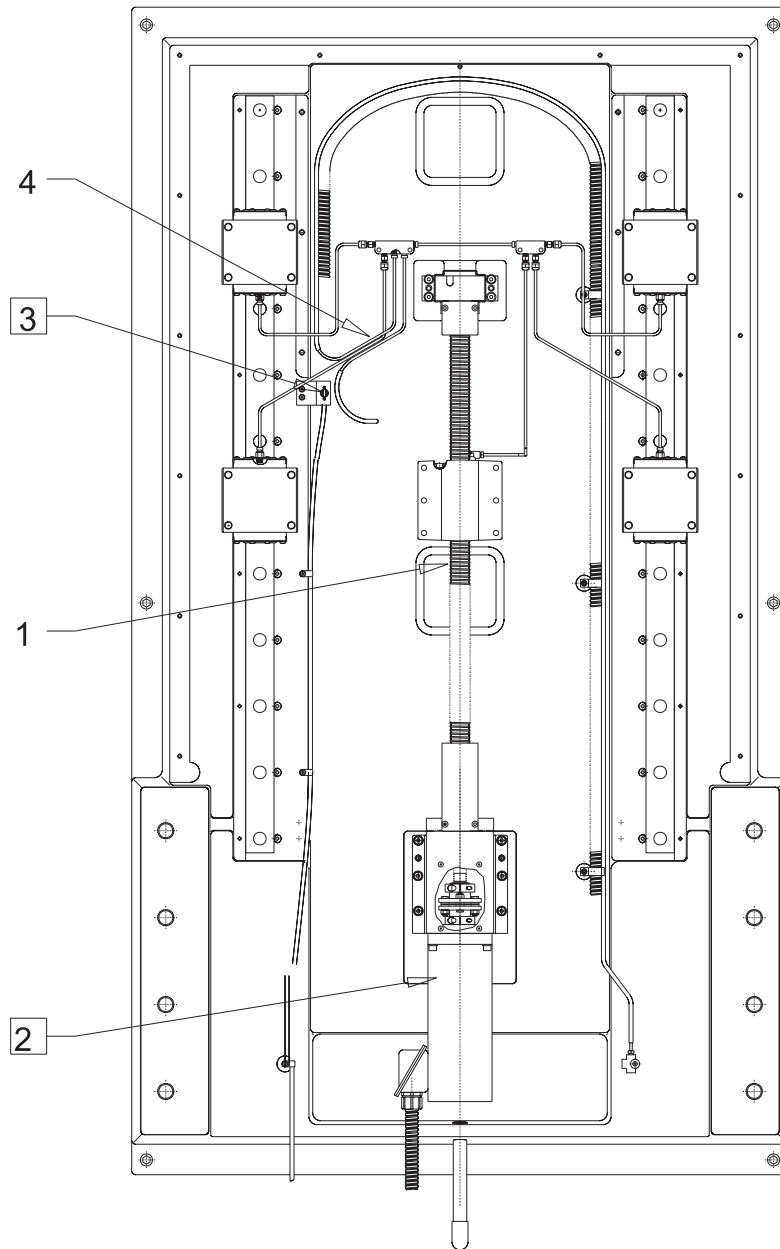
2 - 30-7523 - SADDLE OIL LINE ASSEMBLY

3 - 32-1400 - X AXIS MOTOR ASSEMBLY

4 - 32-2050 - TELEMECHANIQUE SWITCH ASSEMBLY

□ = DESIGNATES PARTS AVAILABLE FROM HAAS SERVICE DEPARTMENT

VF-0/1 Saddle

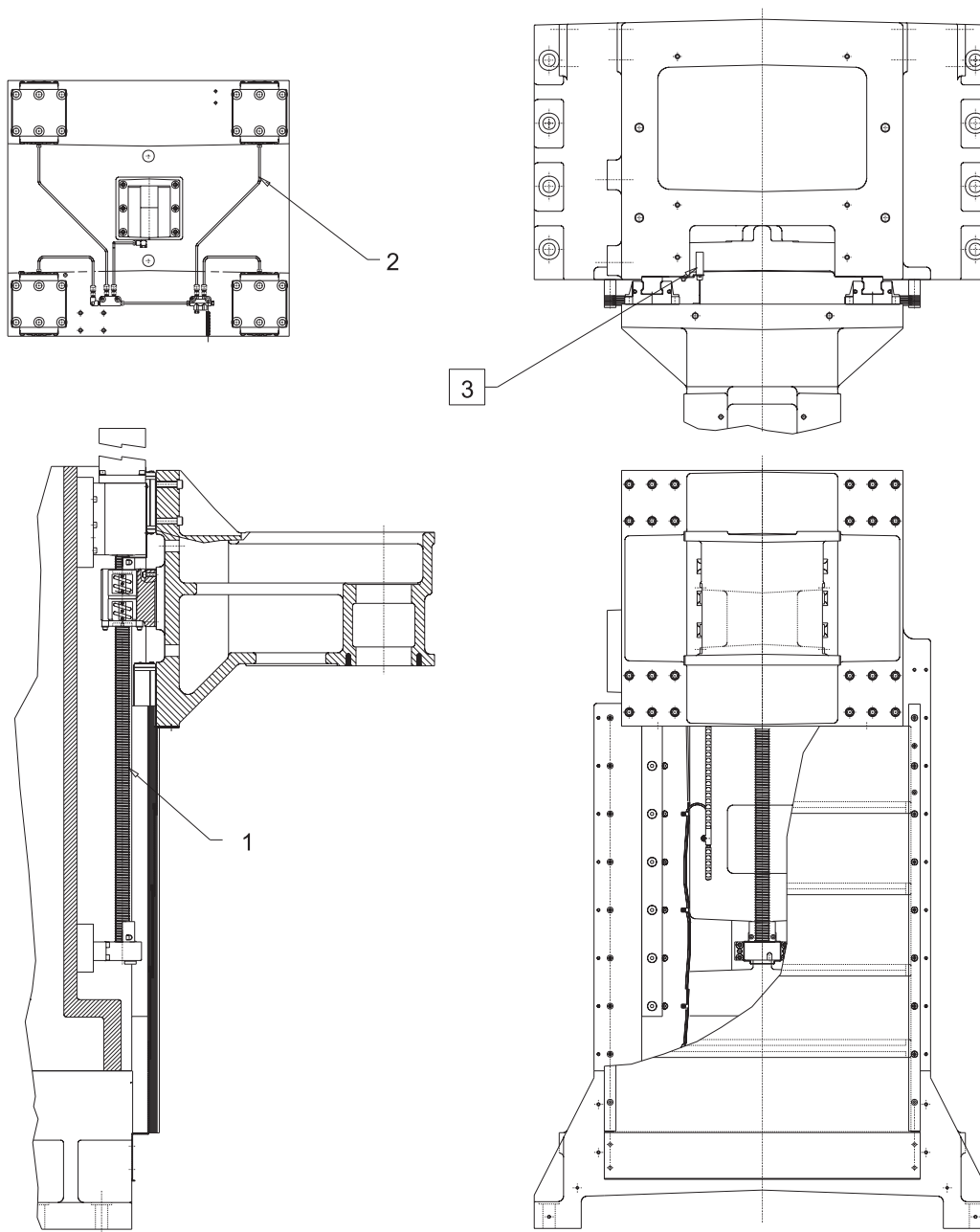
**30-1900A BASE ASSEMBLY**

- 1 - 30-0196 - LEAD SCREW ASSEMBLY
- 2 - 32-1600 - MOTOR ASSEMBLY
- 3 - 32-2031 - TELEMECHANIQUE SWITCH ASSEMBLY
- 4 - 30-7526 - BASE OIL LINE ASSEMBLY

☐ DESIGNATES PARTS AVAILABLE FROM HAAS SERVICE DEPT.

* DESIGNATES REMOVED PART

VF-3 Base

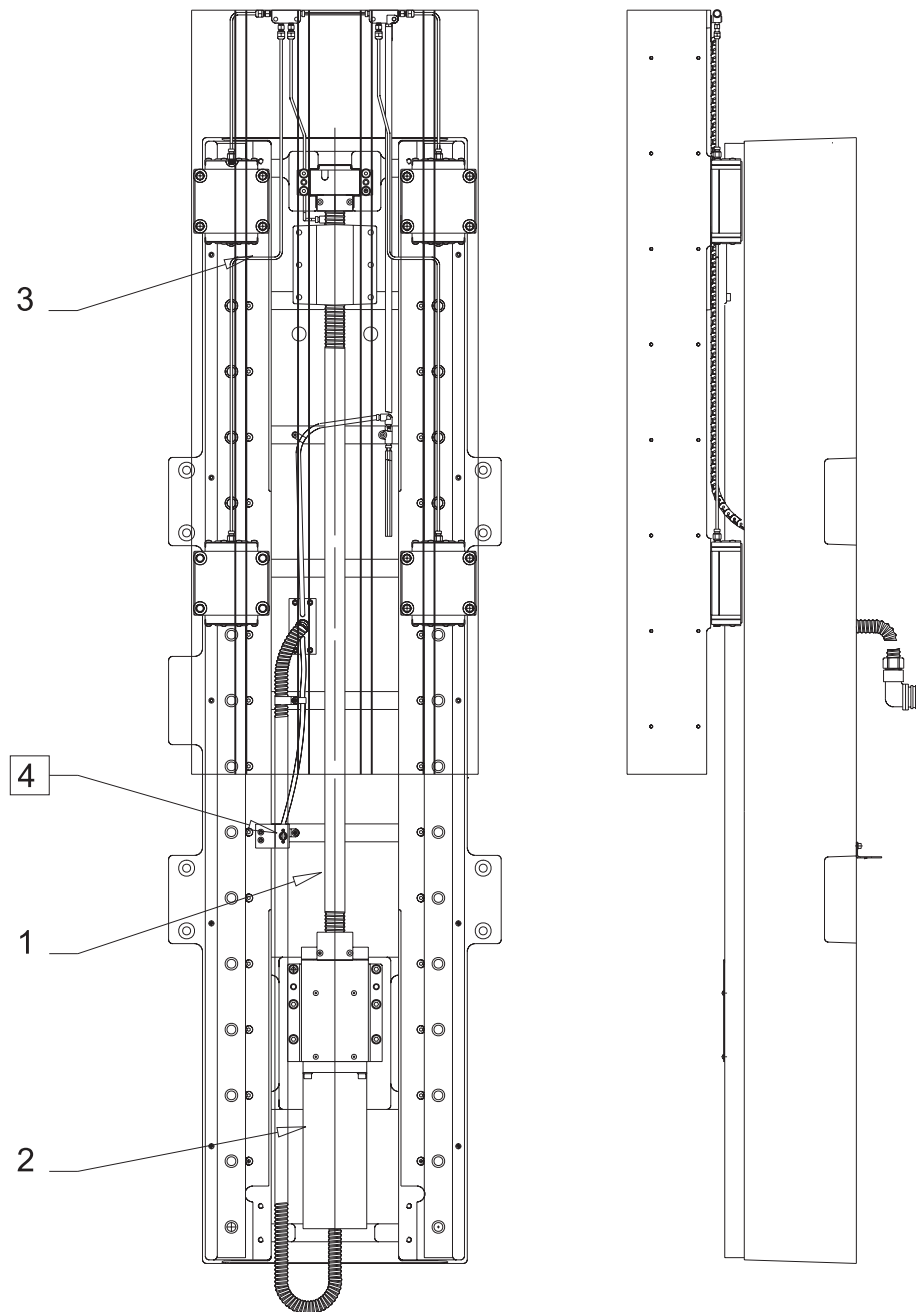


□ DESIGNATES PARTS AVAILABLE FROM HAAS SERVICE DEPT.

- 1 - 30-1210 - LEAD SCREW ASSEMBLY
- 2 - 32-7528 - COLUMN OIL LINE ASSEMBLY
- 3 - 32-2041 - TELEMECHANIQUE SWITCH ASSEMBLY

* DESIGNATES REMOVED PART

VF-3 Column

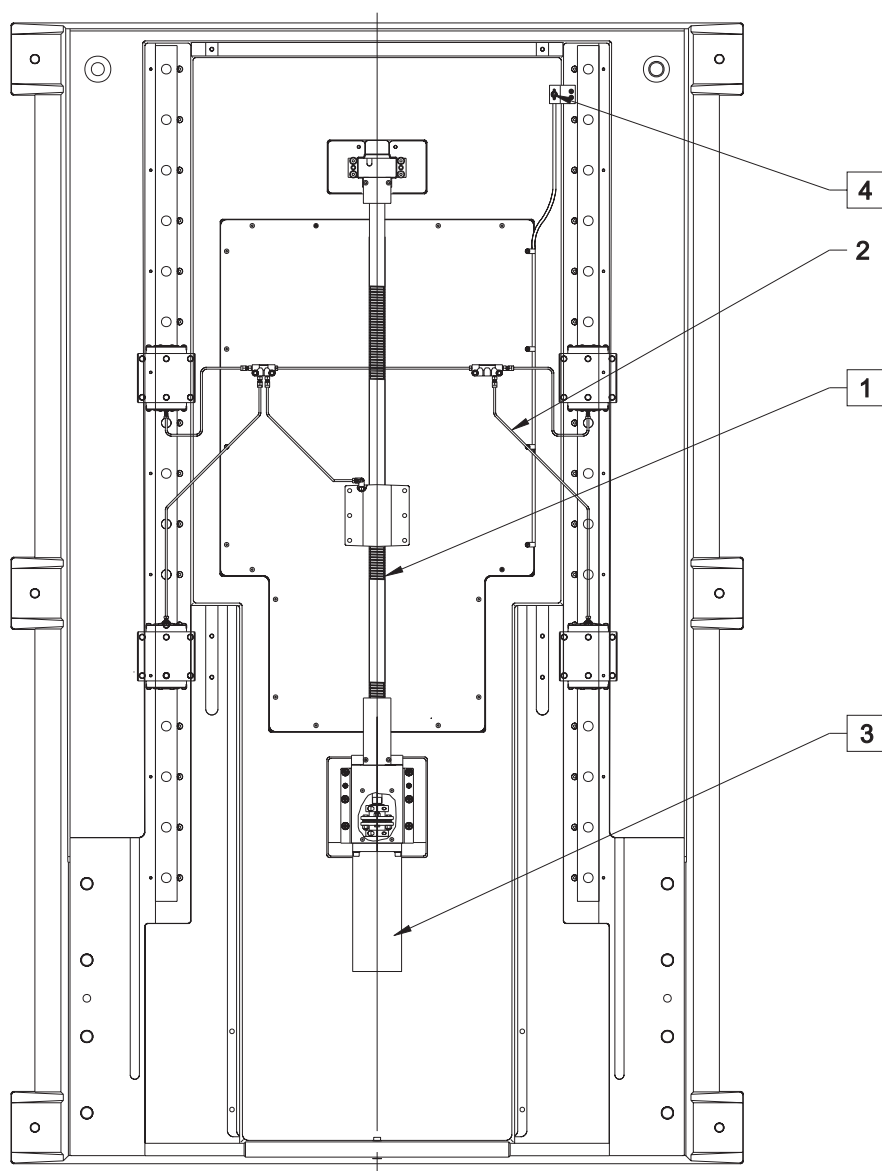


- 1 - 30-0195 - LEAD SCREW ASSEMBLY
 2 - 30-1401 - MOTOR ASSEMBLY
 3 - 30-7527 - SADDLE OIL LINE ASSEMBLY
 4 - 32-2050 - TELEMECHANIQUE SWITCH ASSEMBLY

□ DESIGNATES PARTS AVAILABLE FROM HAAS SERVICE DEPT.

* DESIGNATES REMOVED PART

VF-3 Saddle



 DESIGNATES PARTS AVAILABLE FROM HAAS SERVICE DEPT.

1 - 30-1200 - LEAD SCREW ASSEMBLY

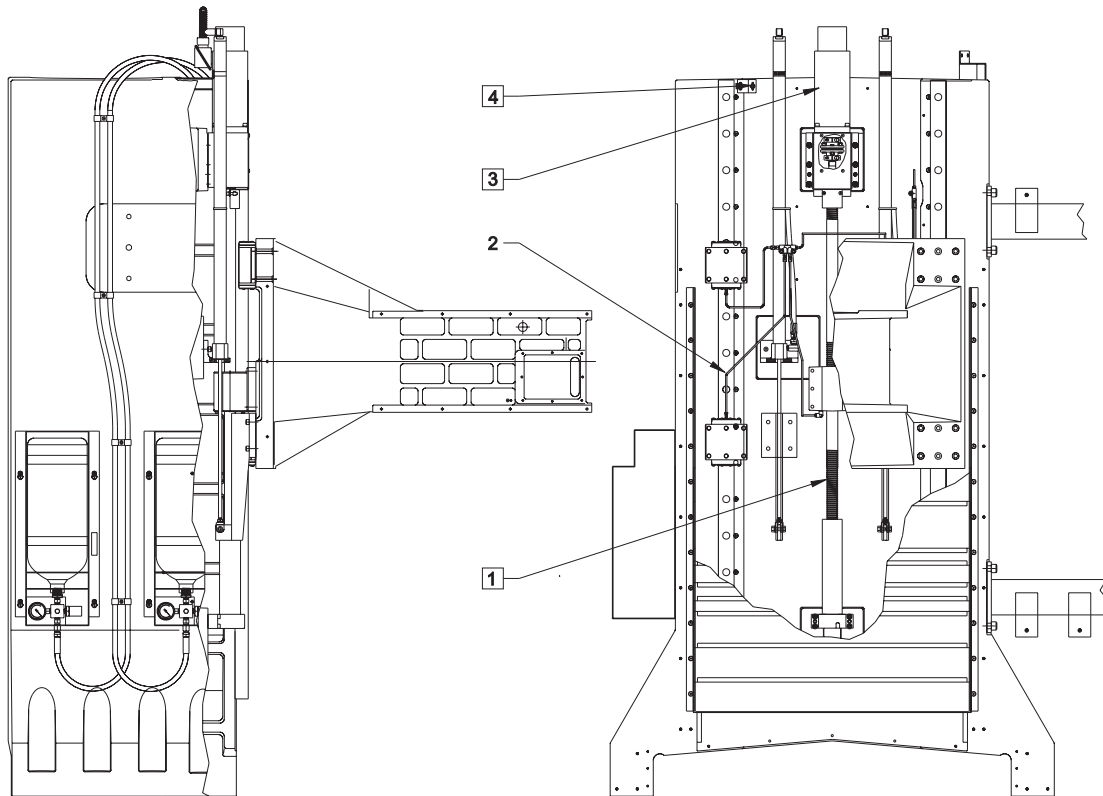
2 - 30-7420 - BASE OIL LINE ASSEMBLY

3 - 32-1780 - Z AXIS MOTOR ASSEMBLY

4 - 32-5056 - LIMIT SWITCH ASSEMBLY

* DESIGNATES REMOVED PART

VF-6 Base

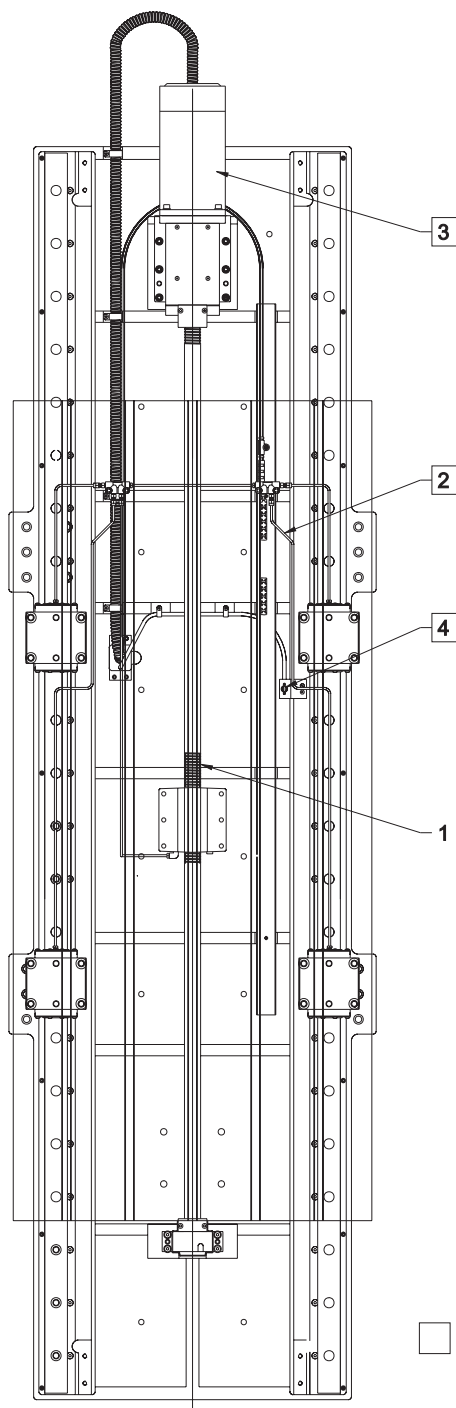


□ DESIGNATES PARTS AVAILABLE FROM HAAS SERVICE DEPT.

- 1 - 30-0195 - LEAD SCREW ASSEMBLY
- 2 - 30-7400 - COLUMN OIL LINE ASSEMBLY
- 3 - 32-1780 - Z AXIS MOTOR ASSEMBLY
- 4 - 32-2050 - TELEMCHANIQUE SWITCH ASSEMBLY

* DESIGNATES REMOVED PART

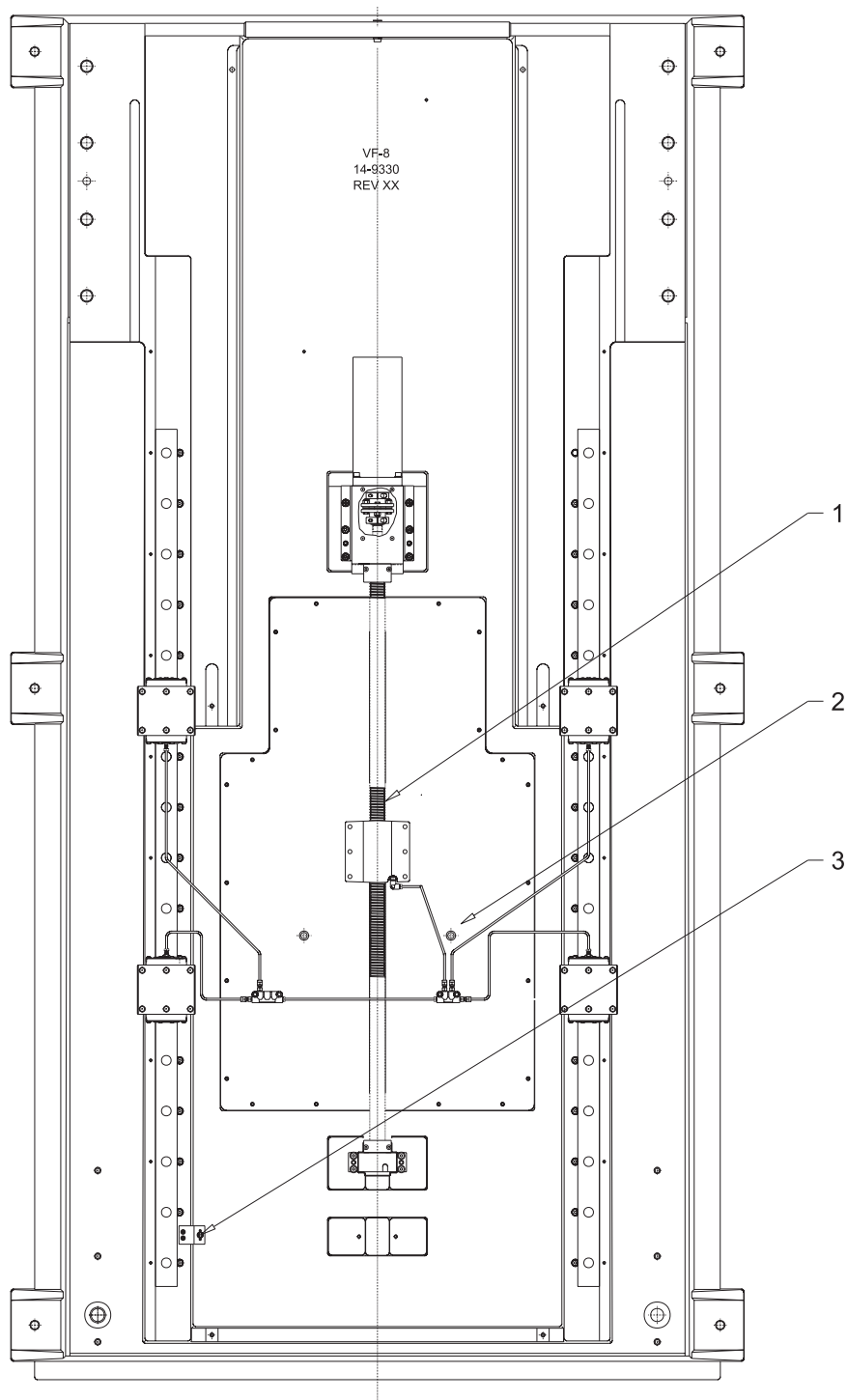
VF-6 Column



- 1 - 30-1240 - LEAD SCREW ASSEMBLY
- 2 - 30-7410 - SADDLE OIL LINE ASSEMBLY
- 3 - 32-1406 - X AXIS MOTOR ASSEMBLY
- 4 - 32-2051 - TELEMCHANIQUE SWITCH ASSEMBLY

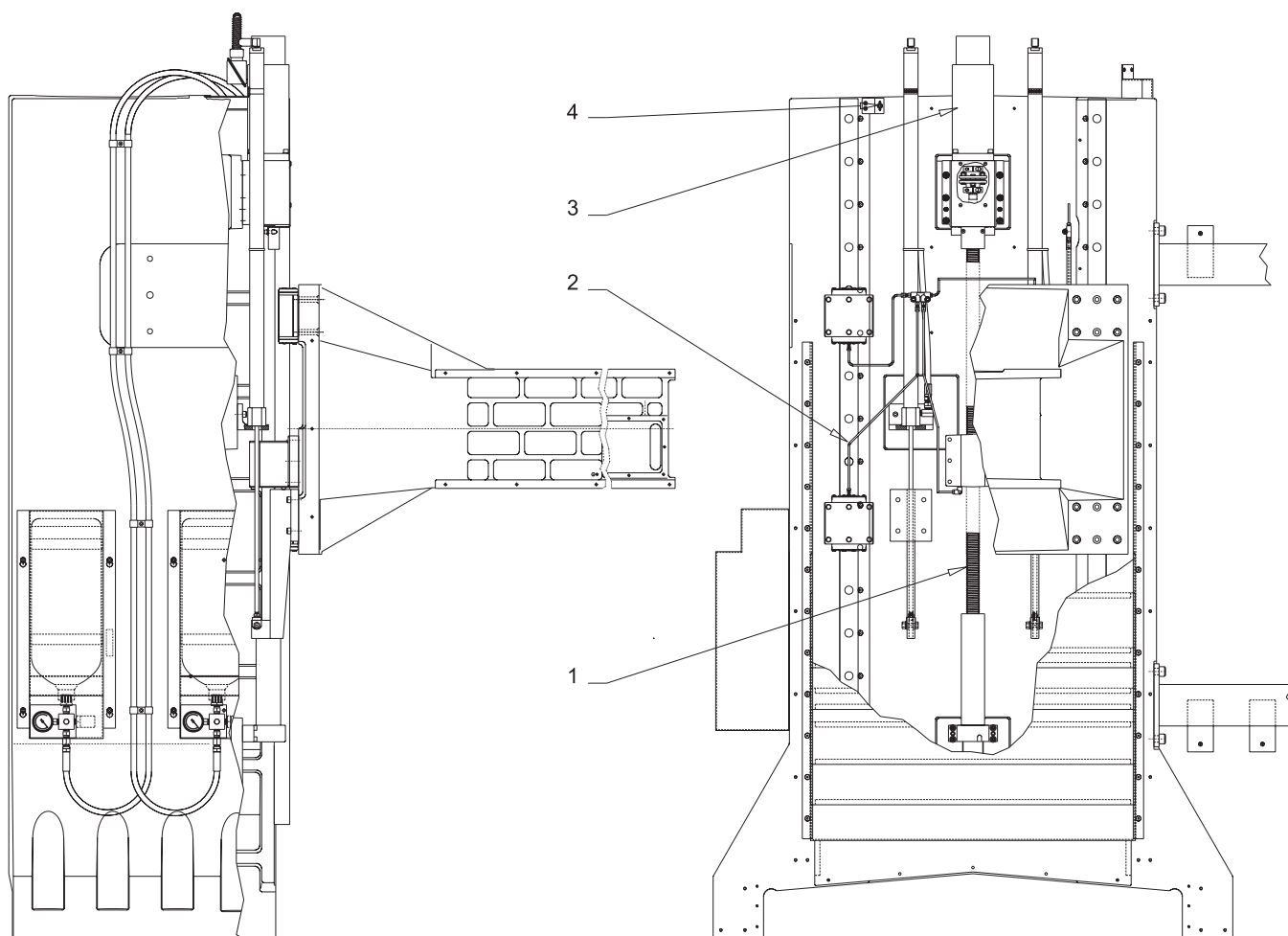
* DESIGNATES REMOVED PART

VF-6 Saddle



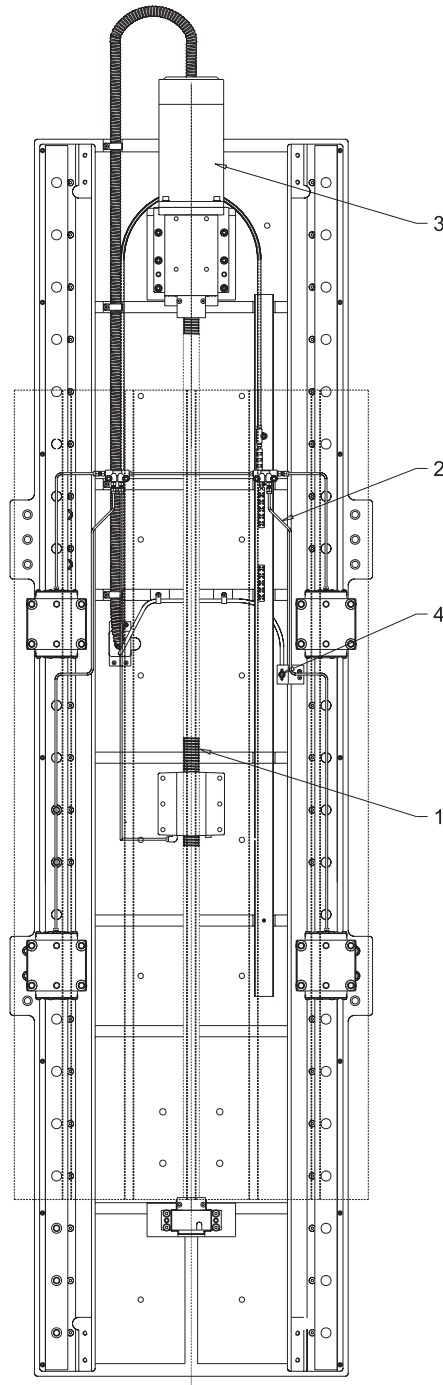
- 1 - 30-0195 - LEAD SCREW ASSEMBLY
2 - 30-7420 - BASE OIL LINE ASSEMBLY
3 - 32-5056 - LIMIT SWITCH ASSEMBLY

VF-8 Base



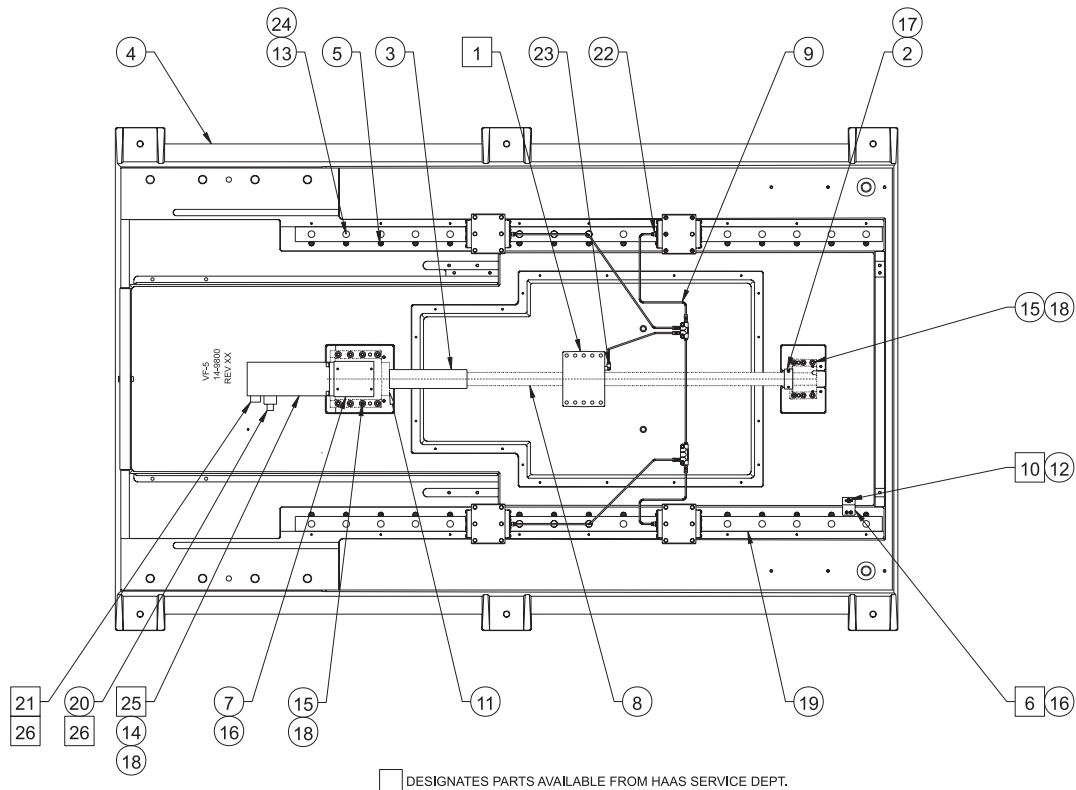
- 1 - 30-0195 - LEAD SCREW ASSEMBLY
- 2 - 30-7400 - COLUMN OIL LINE ASSEMBLY
- 3 - 32-1780 - Z AXIS MOTOR ASSEMBLY
- 4 - 32-2050 - TELEMECHANIQUE SWITCH ASSEMBLY

VF-8 Column

**30-2300S MACHINE CASTING ASSY (SADDLE)**

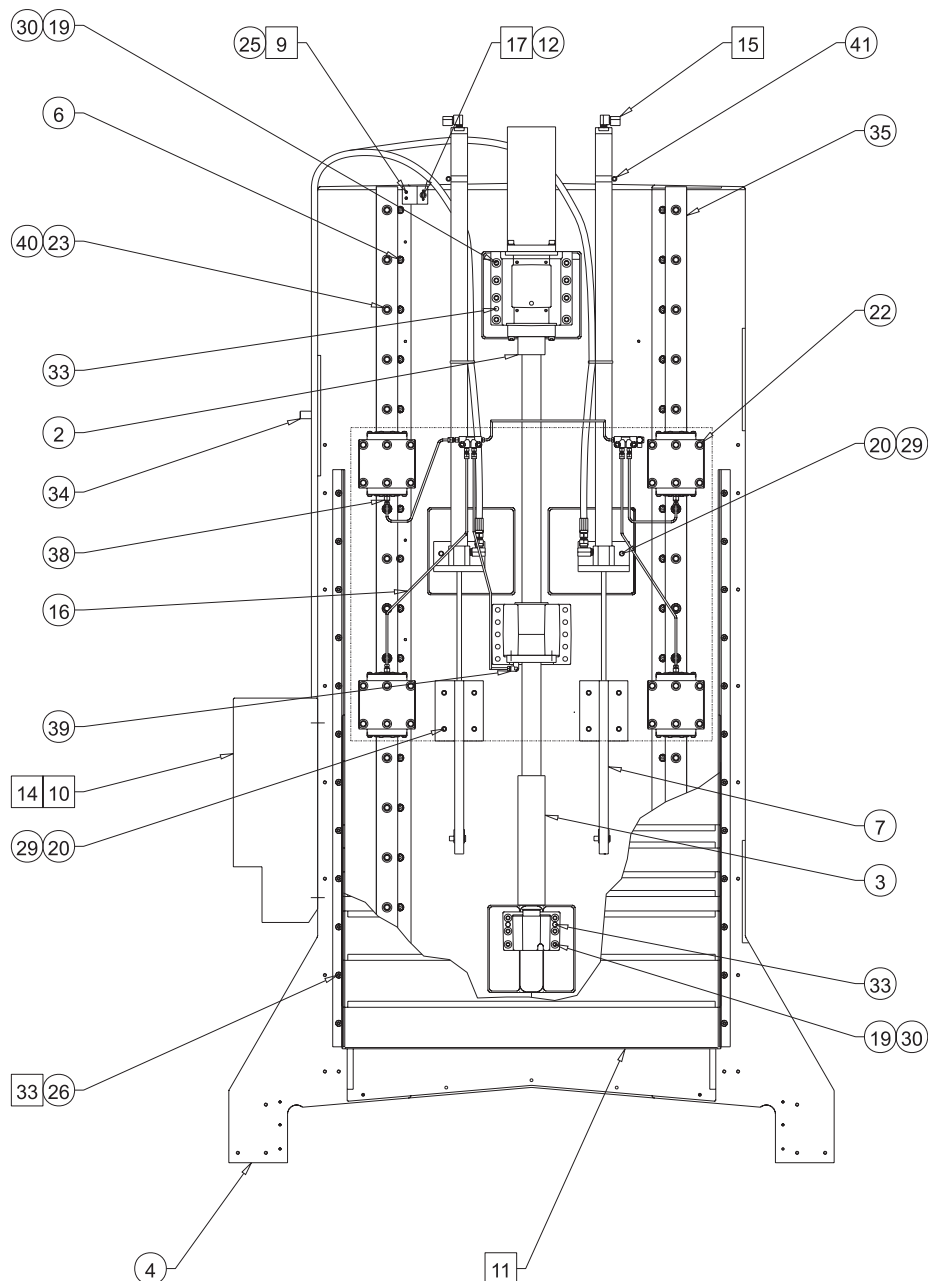
- 1 - 30-0470 - LEAD SCREW ASSEMBLY
- 2 - 30-7410 - SADDLE OIL LINE ASSEMBLY
- 3 - 32-1406 - X AXIS MOTOR ASSEMBLY
- 4 - 32-2051 - TELEMECHANIQUE SWITCH ASSEMBLY

VF-8 Saddle



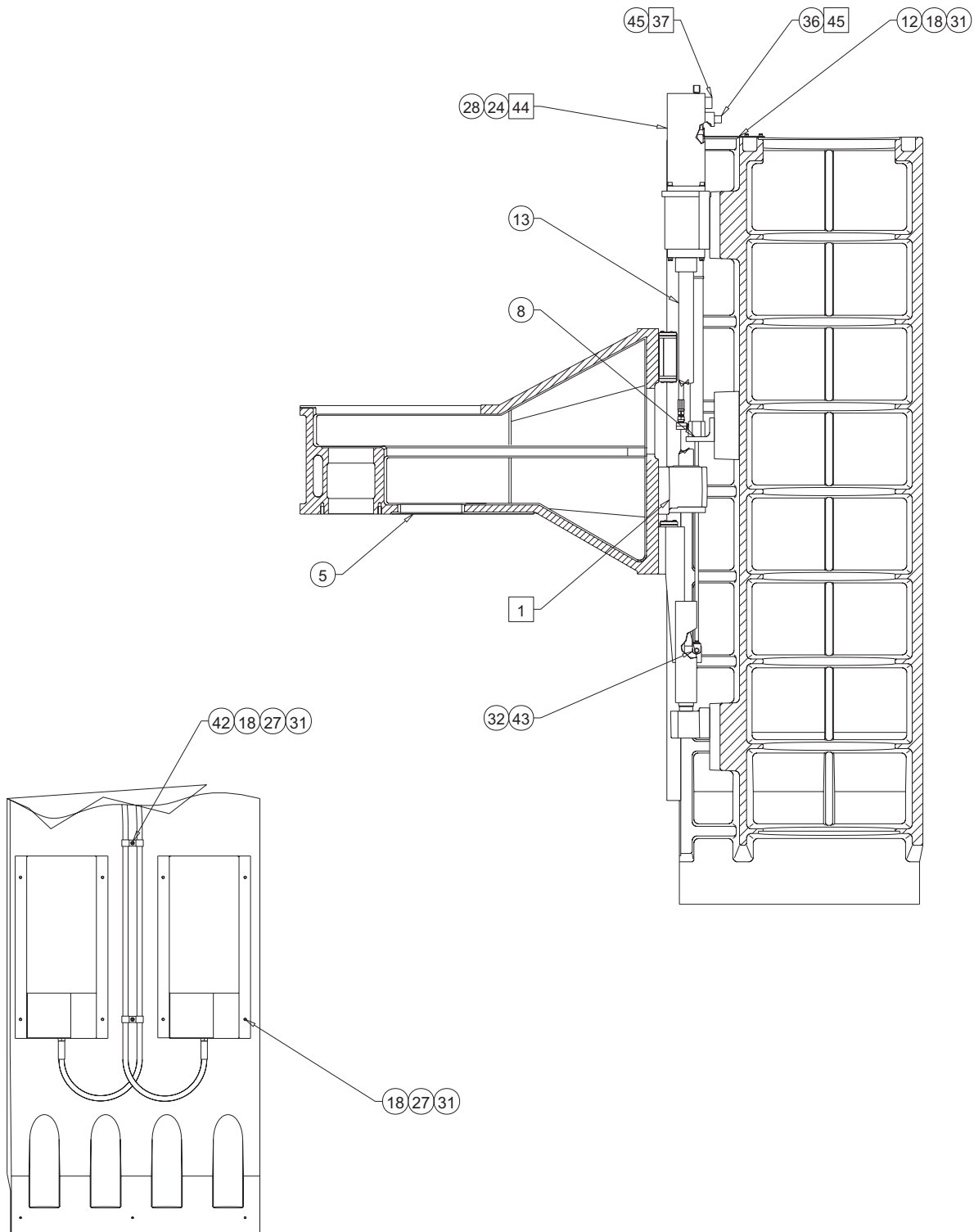
ITEM	QTY	PART NO.	DESCRIPTION
1	1	20-0150	NUT HOUSING 40/50mm BS MACH
2	1	20-9215	Y-AXIS BUMPER BRG END
3	1	20-9218	Y-AXIS BUMPER MTR END
4	1	20-9800	BASE, MACHINING
5	34	22-7458	CAM, LINEAR GUIDE
6	1	25-7267	Y-AXIS MOUNTING BRACKET
7	1	25-9203	COVER PLATE MOTOR MOUNT
8	1	30-1212A	LEAD SCREW Y-AXIS
9	1	30-7420A	BASE OIL LINE ASSEMBLY
10	1	32-5056	LIMIT SWITCH ASSEMBLY
11	5	40-0011	MSHCS M10 X 25mm
12	2	40-16413	MSHCS, M3 X 5
13	34	40-1660	SHCS, 1/2-13 X 1 1/2
14	6	40-1712	SHCS, 5/16-18 X 1 1/4
15	14	40-1715	SHCS, 5/16-18 X 1 1/2
16	6	40-1750	BHCS, 10-32 X 3/8
17	2	40-1950	SHCS, 10-32 X 3/4
18	20	45-1600	WASHER, LOCK
19	4	50-9010	LINEAR GUIDE, X-AXIS VF-3
20	1	57-0075	O-RING 2-021 BUNA
21	1	57-0080	O-RING 2-023 BUNA
22	4	58-1560	ADPT 1/8 M BSPT TO 5/16 F
23	1	58-3031	BANJO ELBOW 5/16 F X M6 M
24	34	59-2033	1/2" CONDUIT STRAP
25	1	62-0013	SERVO MOTOR YASKAWA
26	.05	99-4521	ELECTRICAL GREASE

VF-10 Base



DESIGNATES PARTS AVAILABLE FROM HAAS SERVICE DEPT.

VF-10 Column



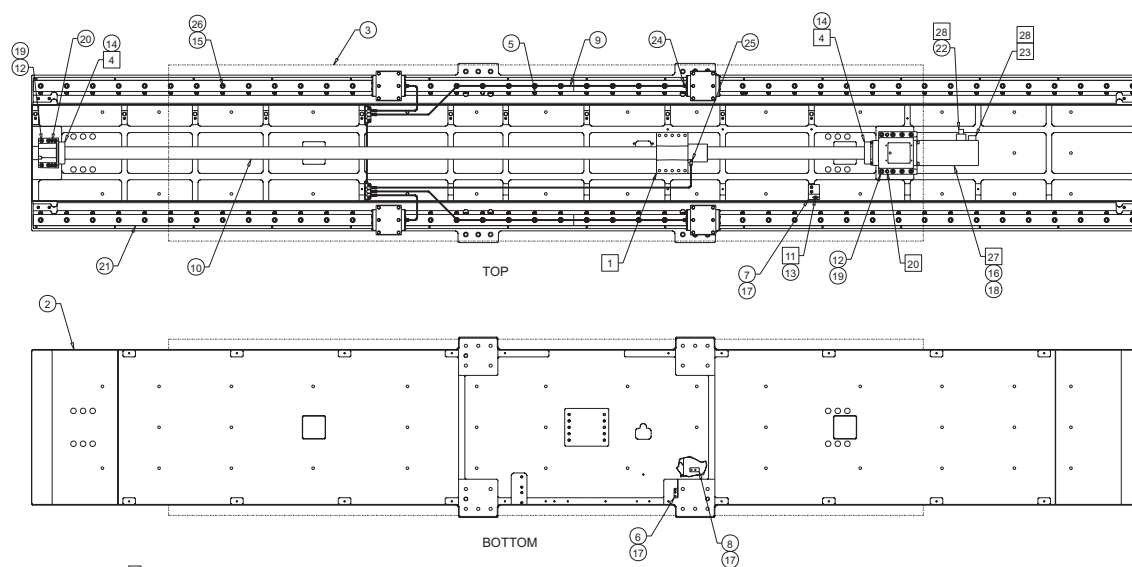
VF-10 Column



VF-10 Column

30-0038 BASIC MACHINE ASSEMBLY (COLUMN)

ITEM	QTY	DWG. NO.	TITLE
1.	1	20-0150	NUT HOUSING 40/50 mm BS
2.	1	20-9216	Z-AXIS BUMPER MTR END 40MM
3.	1	20-9217	Z-AXIS BUMPER SPRT END 40MM
4.	1	20-9801	COLUMN, MACHINED
5.	1	20-9802	SPINDLE HEAD, MACHINED
6.	34	22-7458	CAM, LINEAR GUIDE
7.	2	22-9826A	COUNTER WEIGHT HEAD BRACKET
8.	2	22-9927	CYL. BRKT. COUNTER BALANCE
9.	1	25-7267	Y-AXIS MOUNTING BRACKET
10.	2	25-7560B	HYD. FLUID TANK MOUNT
11.	1	25-9813	WAY COVER
12.	2	25-9929	STABILIZER BRKT. HYD. CYL.
13.	1	30-0473	BL LEADSCREW ASSEMBLY Z AXIS
14.	1	30-3250A	FLUID TANK ASSEMBLY
15.	2	30-3980A	HYD. CYLINDER ASSEMBLY
16.	1	30-7400	COLMUMN OIL LINE ASSEMBLY
17.	1	32-2050	TELEMECHANIQUE ASSEMBLY
18.	15	40-1628	SHCS, 1/4-20 X 1/4
19.	14	40-16372	SHCS, 3/8-16 X 1 1/2
20.	12	40-16391	SHCS, 3/8-16 X 1/2
21.	2	40-16413	MSHCS, M3 X 5
22.	16	40-1655	MSHCS, M12 X 65
23.	34	40-1660	SHCS, 1/2-13 X 1 1/2
24.	6	40-1712	SHCS, 5/16-18 X 1 1/4
25.	2	40-1750	BHCS, 10-32 X .38
26.	24	40-2021	FHCS, 1/4-20 X 3"
27.	11	45-0045	WASHER, BLACK HARD 1/4 X 1/8 THK
28.	6	45-1600	WASHER, LOCK 5/16
29.	12	45-1665	WASHER, FLAT 3/8 I.D.
30.	14	45-1681	WASHER, SPLIT LOCK 3/8 MED.
31.	15	45-1800	WASHER, SPLIT LOCK 1/4 MED.
32.	2	46-1810	JAM NUT, HEX 3/8-24
33.	6	48-0045	PIN, PULL 3/8 X 1 1/2
34.	2	48-1699	PIN, DOWEL 5/8 X 2 1/4
35.	4	50-0001	LINEAR GUIDE
36.	1	57-0075	O-RING 2-021 BUNA
37.	1	57-0080	O-RING 2-023 BUNA
38.	4	58-1560	ADPT 1/8 M BSPT TO 5/16 F
39.	1	58-3031	BANJO ELBOW 5/16 F X M6 M
40.	34	59-2033	1/2" CONDUIT STRAP
41.	2	59-4002	HOSE CLAMP
42.	6	59-4016	HYD. HOSE CLAMP
43.	2	59-9829	CLEVIS COUNTER WEIGHT
44.	1	62-0013	SERVO MOTOR YASKAWA
45.	.05	99-4521	ELECTRICAL GREASE



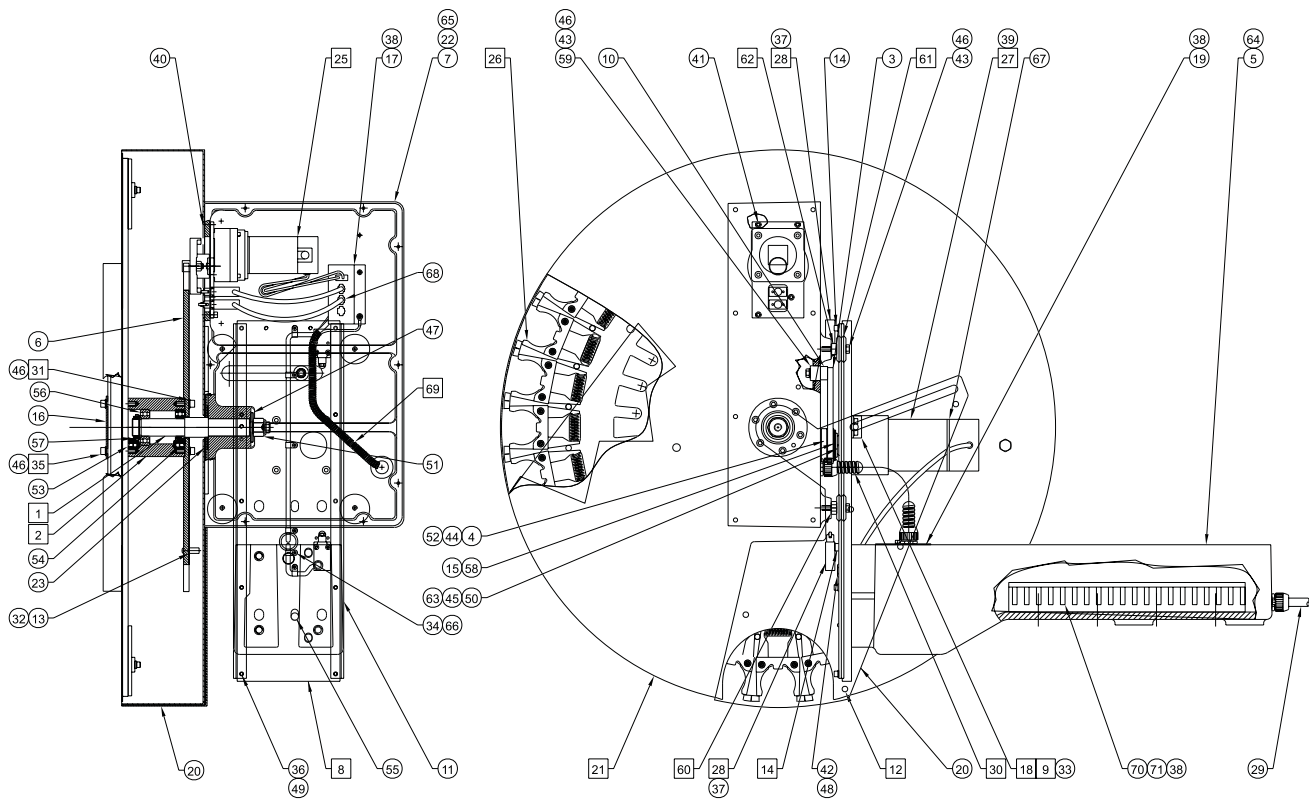
□ DESIGNATES PARTS AVAILABLE FROM HAAS SERVICE DEPT.

30-0038 BASIC MACHINE ASSEMBLY (SADDLE)

ITEM	QTY	DWG. NO.	TITLE
1.	1	20-0150	NUT HOUSING 40/50 mm BS
2.	1	20-0153	SADDLE, VF-10 MACHINED
3.	1	20-0154	TABLE, VF-10 MACHINED
4.	2	20-0156	BUMPER, 1" 40 & 50 mm LD SCREW
5.	86	22-7458	CAM, LINEAR GUIDE
6.	1	25-7459	TRIP BRACKET, TABLE
7.	1	25-9219	LIMIT SWITCH BRACKET X- AXIS
8.	1	25-9220	TRIP BRACKET X-AXIS
9.	1	30-0036	X-AXIS OIL LINE ASSEMBLY
10.	1	30-0516	BL LEADSCREW ASSEMBLY X AXIS
11.	1	32-2051	LIMIT SWITCH X HOME
12.	14	40-16372	SHCS, 3/8-16 X 1 1/2
13.	2	40-16413	MSHCS, M3 X 5
14.	4	40-16455	SHCS, 10-32 X .88
15.	86	40-1660	SHCS, 1/2-13 X 1 1/2
16.	6	40-1712	SHCS, 5/16-18 X 1 1/4
17.	6	40-1750	BHCS, 10-32 X .38
18.	6	45-1600	WASHER, LOCK
19.	14	45-1681	WASHER, SPLIT LOCK
20.	4	48-0045	PULL PIN 3/8 X 1 1/2
21.	4	50-0001	LINEAR GUIDE
22.	1	57-0075	O-RING 2-021 BUNA
23.	1	57-0080	O-RING 2-023 BUNA
24.	4	58-1560	ADPT 1/8 M BSPT TO 5/16 F
25.	1	58-3031	BANJO ELBOW 5/16 F X M6 M
26.	86	59-2033	1/2" CONDUIT STRAP
27.	1	62-0013	SERVO MOTOR YASKAWA
28.	.05	99-4521	ELECTRICAL GREASE

* DESIGNATES REMOVED PART

VF-10 Saddle



□ DESIGNATES PARTS AVAILABLE FROM HAAS SERVICE DEPT.

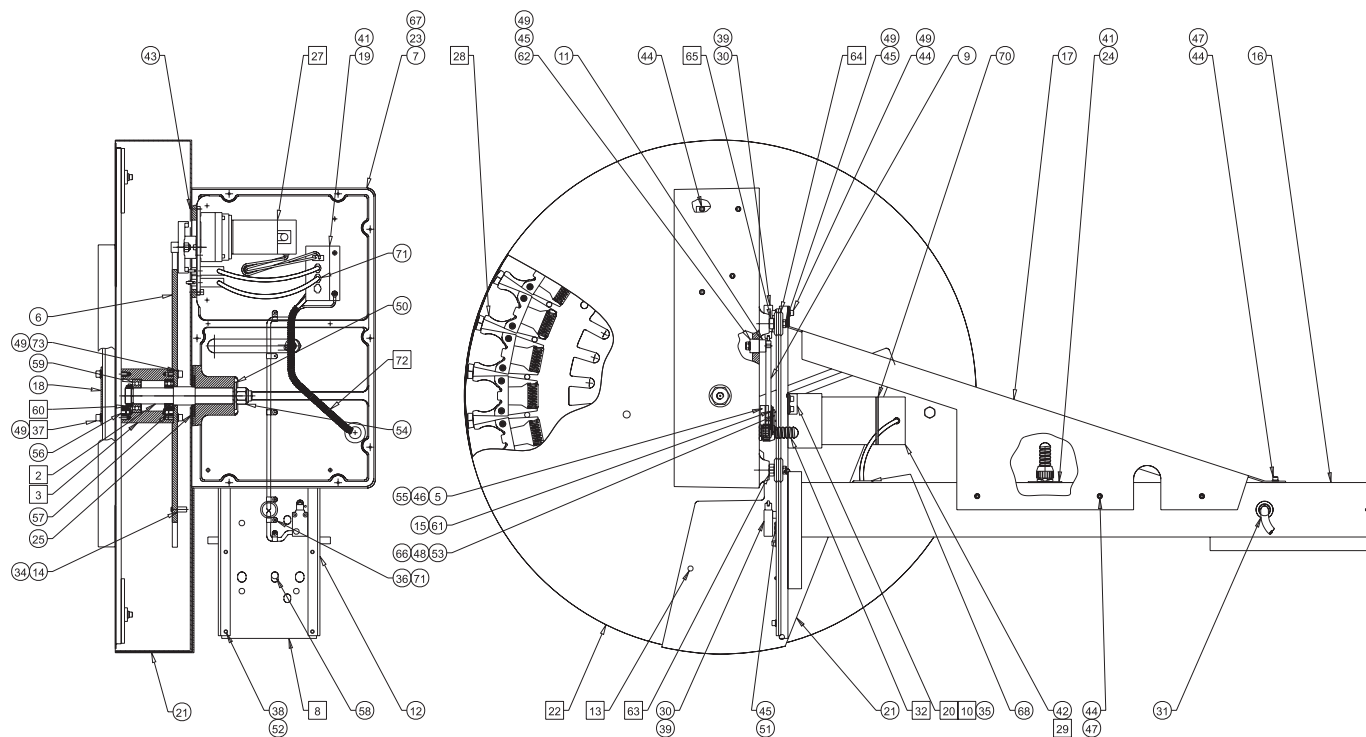
Tool Changer Assembly VF-3/4



Tool Changer Assembly VF-3/4

ITEM	QTY	PART NO.	DESCRIPTION
1	1	20-7035G	VERTICAL AXLE
2	1	20-7038A	BEARING HOUSING
3	1	20-7475	ARM, SLIP CLUTCH
4	1	20-7476	HUB, SLIP CLUTCH
5	1	20-9008	TOOL HOLDING ARM
6	1	20-9325	32 TOOL GENEVA STAR, 2 PIN
7	1	20-9326	TOOL CARRIAGE, MACHINING
8	1	20-9330	32 T/C HOLDING PLATE
9	1	22-2065	LOCATING PIN
10	1	22-7034	SPACER, CAM FOLLOWER
11	2	22-7106	'V' TRACK, T/C
12	3	22-7163	RIDER, TRAP DOOR
13	1	22-7255A	TOOL #1 STAND OFF
14	2	22-7263	SWITCH MOUNTING BLOCK
15	1	22-7477	PRESSURE PLATE
16	1	25-7036	CAP, TOOL CHANGER
17	1	25-7162	CONNECTOR BRACKET
18	1	25-7168	DOOR OPENER BRACKET
19	1	25-9085	CONDUIT MTG PLATE
20	1	25-9329	DOOR T/C COVER
21	1	25-9331	TOOL CHANGER COVER
22	1	25-9334	SHUTTLE COVER PLATE
23	2	26-7239	SPACER RING
24	1	29-7612	CT TOOLING DECAL
25	1	30-0005	GENEVA DRIVER ASSY
26	1	30-0006	CAROUSEL ASSY, 32 TOOL
27	1	32-1800	SHUTTLE MOTOR ASSY
28	2	32-2010	24" LIMIT SWITCH
29	1	32-7011	CONDUIT ASSY, T/C
30	1	32-7611	CONDUIT ASSY, TOOL CARRIAGE
31	6	40-1500	SHCS, 5/16-18 x 1"
32	1	40-16091	BHCS, 10-32 x 1"
33	3	40-1632	SHCS, 1/4-20 x 1/2"
34	6	40-1669	BHCS, 8-32 x 3/8"
35	6	40-1676	SHCS, 5/16-18 x 2"
36	10	40-1697	SHCS, 1/4-20 x 3/4"
37	4	40-1803	SHCS, 8-32 x 1 1/4"
38	12	40-1850	SHCS, 10-32 x 3/8"
39	4	40-1970	FHCS, 1/4-28 x 1"
40	8	40-1980	BHCS, 1/4-20 x 1/2"
41	4	40-2000	SHCS, 1/4-20 x 5/8"
42	4	43-1602	HHB, 1/2-13 x 3"
43	5	43-7000	HHB, 5/16-18 x 1 3/4"
44	1	44-1710	SSS, CUP PT 1/4-20 x 3/8"
45	1	45-0050	WASHER, 5702-313-120
46	17	45-1600	WASHER, SPLIT LOCK, 5/16 MED.
47	1	45-1725	WASHER, FLAT CUT 3/4"
48	4	45-1740	WASHER, BLACH HARD 1/2"
49	10	45-1800	WASHER, SPLIT LOCK 1/4" MED.
50	2	45-2020	WASHER, NYLON
51	1	46-1705	LOCK-NUT, ELASTIC, 3/4-10
52	1	48-0005	PIN, DOWEL 3/16 x 3/8"
53	1	48-0019	PIN, DOWEL 1/4 x 5/8"
54	1	48-0020	PIN, DOWEL 1/4 x 1"
55	2	48-1750	PIN, DOWEL 1/2 x 1 1/2"
56	2	51-0010	BEARING DEEP GROOVE
57	1	51-0012	BEARING LOCK NUT, BH-06
58	1	51-6000	BEARING LOCK NUT, NT-05
59	1	54-0010	CAM FOLLOWER, TOOL CHANGER
60	2	54-0020	BUSHING, GUIDE WHEEL
61	4	54-0030	GUIDE WHEEL
62	2	54-0040	STANDARD BUSHING, GD. WHEEL
63	2	55-0010	SPRING WASHER, B2500-080
64	1	57-9139	GASKET, TOOL HOLD ARM
65	1	57-9335	SHUTTLE COVER GASKET
66	6	63-1031	CABLE CLAMP, 1/4"
67	1	70-0050	PLT4S-M CABLE TIES
68	1	75-15721	MOLEX BSNG. 2 PIN MALE
69	1	78-1996	SPLIT FLEX TUBING 1/2" I.D.
70	1.75'	79-1000	WIRE CHANNEL, 1" x 2"
71	1.70'	79-1001	COVER, 1" WIRE CHANNEL

* DESIGNATES REMOVED PART



□ DESIGNATES PARTS AVAILABLE FROM HAAS SERVICE DEPT.

Tool Changer Assembly VF-6..10

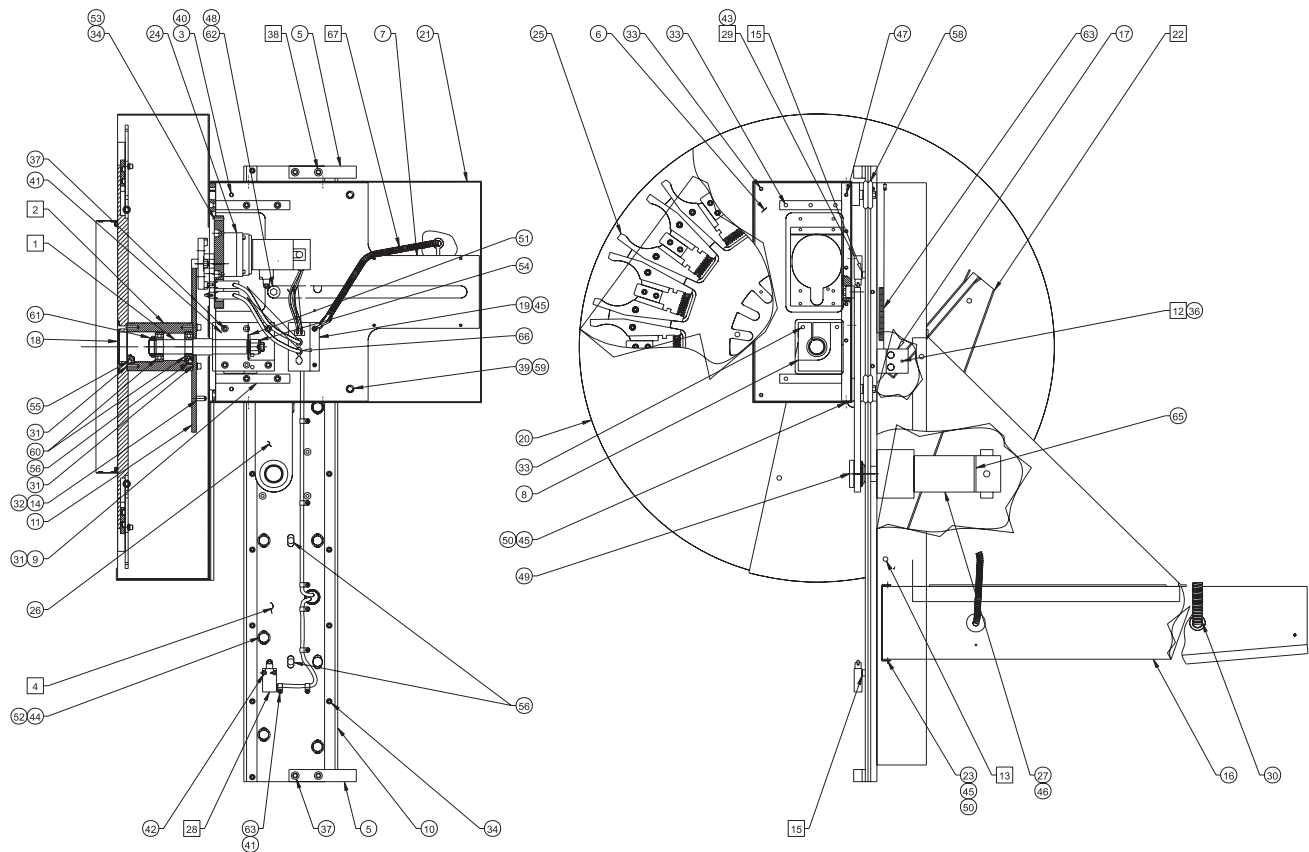


Tool Changer Assembly VF-6..10

IT	QTY	PART NO	TITLE	IT	QTY	PART NO	TITLE
1	2	20-0031	ADJ. SWITCH BLOCK	38	10	40-1697	SHCS, 1/4-20 x 3/4"
2	1	20-7035G	VERTICAL AXLE	39	8	40-1800	SHCS, 8-32 X 3/4" LG.
3	1	20-7038A	BEARING HOUSING	40	4	40-1830	HHB, 1/2-13 x 1 3/4"
4	1	20-7475	ARM, SLIP CLUTCH	41	12	40-1850	SHCS, 10-32 x 3/8"
5	1	20-7476	HUB, SLIP CLUTCH	42	4	40-1970	FHCS, 1/4-28 x 1"
6	1	20-9325	32 TOOL GENEVA STAR, 2 PIN	43	8	40-1980	BHCS, 1/4-20 x 1/2"
7	1	20-9326	TOOL CARRIAGE, MACHINING	44	13	40-2000	SHCS, 1/4-20 x 5/8"
8	1	20-9330	32 T/C HOLDING PLATE	45	5	43-7000	HHB, 5/16-18 x 1 3/4"
9	1	20-9834	TOOL CHANGER CLUTCH ARM	46	1	44-1710	SSS, CUP PT 1/4-20 x 3/8"
10	1	22-2065	LOCATING PIN	47	9	45-0045	WASHER, BLK HRD, 1/4 X 1/8 THK
11	1	22-7034	SPACER, CAM FOLLOWER	48	1	45-0050	WASHER, 5702-313-120
12	2	22-7106	'V' TRACK, T/C	49	17	45-1600	WASHER, SPLIT LOCK, 5/16 MED.
13	4	22-7163	RIDER, TRAP DOOR	50	1	45-1725	WASHER, FLAT CUT 3/4"
14	1	22-7255A	TOOL #1 STAND OFF	51	4	45-1740	WASHER, BLACK HARD 1/2"
15	1	22-7477	PRESSURE PLATE	52	10	45-1800	WASHER, SPLIT LOCK 1/4" MED.
16	1	22-9805	HOLDING ARM	53	2	45-2020	WASHER, NYLON
17	1	25-0014	BRACE	54	1	46-1705	LOCK-NUT, ELASTIC, 3/4-10
18	1	25-7036	CAP, TOOL CHANGER	55	1	48-0005	PIN, DOWEL 3/16 x 3/8"
19	1	25-7162	CONNECTOR BRACKET	56	1	48-0019	PIN, DOWEL 1/4 x 5/8"
20	1	25-7168	DOOR OPENER BRACKET	57	1	48-0020	PIN, DOWEL 1/4 x 1"
21	1	25-9329	DOOR T/C COVER	58	2	48-1750	PIN, DOWEL 1/2 x 1 1/2"
22	1	25-9331	TOOL CHANGER COVER	59	2	51-0010	BEARING DEEP GROOVE
23	1	25-9334	SHUTTLE COVER PLATE	60	1	51-0012	BEARING LOCK NUT, BH-06
24	1	25-9912	CONDUIT MOUNTING PLATE, VF-6C	61	1	51-6000	BEARING LOCK NUT, NT-05
25	2	26-7239	SPACER RING	62	1	54-0010	CAM FOLLOWER, TOOL CHANGER
26	1	29-7612	CT TOOLING DECAL	63	2	54-0020	BUSHING, GUIDE WHEEL
27	1	30-0005	GENEVA DRIVER ASSY	64	4	54-0030	GUIDE WHEEL
28	1	30-0006	CAROUSEL ASSY, 32 TOOL	65	2	54-0040	STANDARD BUSHING, GD. WHEEL
29	1	32-1800	SHUTTLE MOTOR ASSY	66	2	55-0010	SPRING WASHER, B2500-080
30	2	32-2013	TELMECH 44" CABLE ASS'Y	67	1	57-9335	SHUTTLE COVER GASKET
31	1	32-7012B	MOLDED ATC CABLE ASSY	68	1	59-7222	GROMMET
32	1	32-7611	CONDUIT ASSY, TOOL CARRIAGE	69	6	63-1031	CABLE CLAMP, 1/4"
*33	6	40-1500	SHCS, 5/16-18 X 1"	70	1	70-0050	PLT4S-M CABLE TIES
34	1	40-16091	BHCS, 10-32 x 1"	71	1	75-15721	MOLEX BSNG. 2 PIN MALE
35	8	40-1632	SHCS, 1/4-20 x 1/2"	72	1	78-1996	SPLIT FLEX TUBING 1/2" I.D.
36	6	40-1669	BHCS, 8-32 x 3/8"	73	6	40-16385	SHCS, 5/16-18 X 3/4"
37	6	40-1676	SHCS, 5/16-18 x 2"				

NOTE: 1. ON VF-8,9 USE 22-9949
FOR ITEM 5.

* DESIGNATES REMOVED PART



□ DESIGNATES PARTS AVAILABLE FROM HAAS SERVICE DEPT.

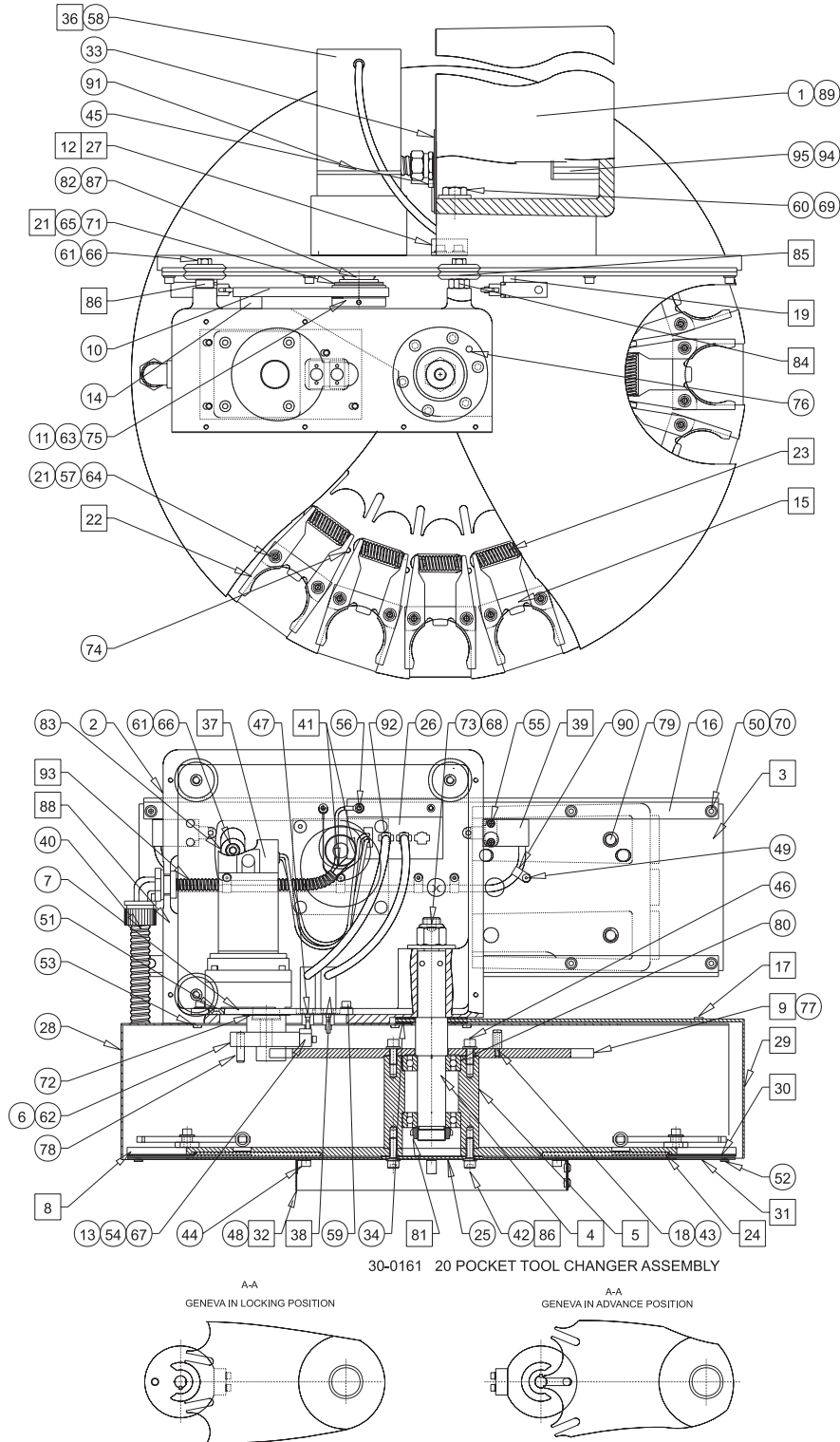
50 Taper Tool Changer Assembly



50 Taper Tool Changer Assembly

IT	QTY	PART NO	DESCRIPTION	IT	QTY	PART NO	DESCRIPTION
1	2	20-0031	ADJ. SWITCH BLOCK	37	6	40-1676	SHCS, 5/16-18 x 2"
2	1	20-7035G	VERTICAL AXLE	38	10	40-1697	SHCS, 1/4-20 x 3/4"
3	1	20-7038A	BEARING HOUSING	39	8	40-1800	SHCS, 8-32 X 3/4" LG.
4	1	20-7475	ARM, SLIP CLUTCH	40	4	40-1830	HHB, 1/2-13 x 1 3/4"
5	1	20-7476	HUB, SLIP CLUTCH	41	12	40-1850	SHCS, 10-32 x 3/8"
6	1	20-9325	32 TOOL GENEVA STAR, 2 PIN	42	4	40-1970	FHCS, 1/4-28 x 1"
7	1	20-9326	TOOL CARRIAGE, MACHINING	43	8	40-1980	BHCS, 1/4-20 x 1/2"
8	1	20-9330	32 T/C HOLDING PLATE	44	13	40-2000	SHCS, 1/4-20 x 5/8"
9	1	20-9834	TOOL CHANGER CLUTCH ARM	45	5	43-7000	HHB, 5/16-18 x 1 3/4"
10	1	22-2065	LOCATING PIN	46	1	44-1710	SSS, CUP PT 1/4-20 x 3/8"
11	1	22-7034	SPACER, CAM FOLLOWER	47	9	45-0045	WASHER, BLK HRD, 1/4 X 1/8 THK
12	2	22-7106	'V' TRACK, T/C	48	1	45-0050	WASHER, 5702-313-120
13	4	22-7163	RIDER, TRAP DOOR	49	17	45-1600	WASHER, SPLIT LOCK, 5/16 MED.
14	1	22-7255A	TOOL #1 STAND OFF	50	1	45-1725	WASHER, FLAT CUT 3/4"
15	1	22-7477	PRESSURE PLATE	51	4	45-1740	WASHER, BLACK HARD 1/2"
16	1	22-9805	HOLDING ARM	52	10	45-1800	WASHER, SPLIT LOCK 1/4" MED.
17	1	25-0014	BRACE	53	2	45-2020	WASHER, NYLON
18	1	25-7036	CAP, TOOL CHANGER	54	1	46-1705	LOCK-NUT, ELASTIC, 3/4-10
19	1	25-7162	CONNECTOR BRACKET	55	1	48-0005	PIN, DOWEL 3/16 x 3/8"
20	1	25-7168	DOOR OPENER BRACKET	56	1	48-0019	PIN, DOWEL 1/4 x 5/8"
21	1	25-9329	DOOR T/C COVER	57	1	48-0020	PIN, DOWEL 1/4 x 1"
22	1	25-9331	TOOL CHANGER COVER	58	2	48-1750	PIN, DOWEL 1/2 x 1 1/2"
23	1	25-9334	SHUTTLE COVER PLATE	59	2	51-0010	BEARING DEEP GROOVE
24	1	25-9912	CONDUIT MOUNTING PLATE, VF-6C	60	1	51-0012	BEARING LOCK NUT, BH-06
25	2	26-7239	SPACER RING	61	1	51-6000	BEARING LOCK NUT, NT-05
26	1	29-7612	CT TOOLING DECAL	62	1	54-0010	CAM FOLLOWER, TOOL CHANGER
27	1	30-0005	GENEVA DRIVER ASSY	63	2	54-0020	BUSHING, GUIDE WHEEL
28	1	30-0006	CAROUSEL ASSY, 32 TOOL	64	4	54-0030	GUIDE WHEEL
29	1	32-1800	SHUTTLE MOTOR ASSY	65	2	54-0040	STANDARD BUSHING, GD. WHEEL
30	2	32-2013	TELMECH 44" CABLE ASS'Y	66	2	55-0010	SPRING WASHER, B2500-080
31	1	32-7012B	MOLDED ATC CABLE ASSY	67	1	57-9335	SHUTTLE COVER GASKET
32	1	32-7611	CONDUIT ASSY, TOOL CARRIAGE	68	1	59-7222	GROMMET
33	6	40-1500	SHCS, 5/16-18 X 1"	69	6	63-1031	CABLE CLAMP, 1/4"
34	1	40-16091	BHCS, 10-32 x 1"	70	1	70-0050	PLT4S-M CABLE TIES
35	8	40-1632	SHCS, 1/4-20 x 1/2"	71	1	75-15721	MOLEX BSNG. 2 PIN MALE
36	6	40-1669	BHCS, 8-32 x 3/8"	72	1	78-1996	SPLIT FLEX TUBING 1/2" I.D.

* DESIGNATES REMOVED PART



□ DESIGNATES PARTS AVAILABLE FROM HAAS SERVICE DEPT.

20 Pocket Tool Changer VF- 0/1/2

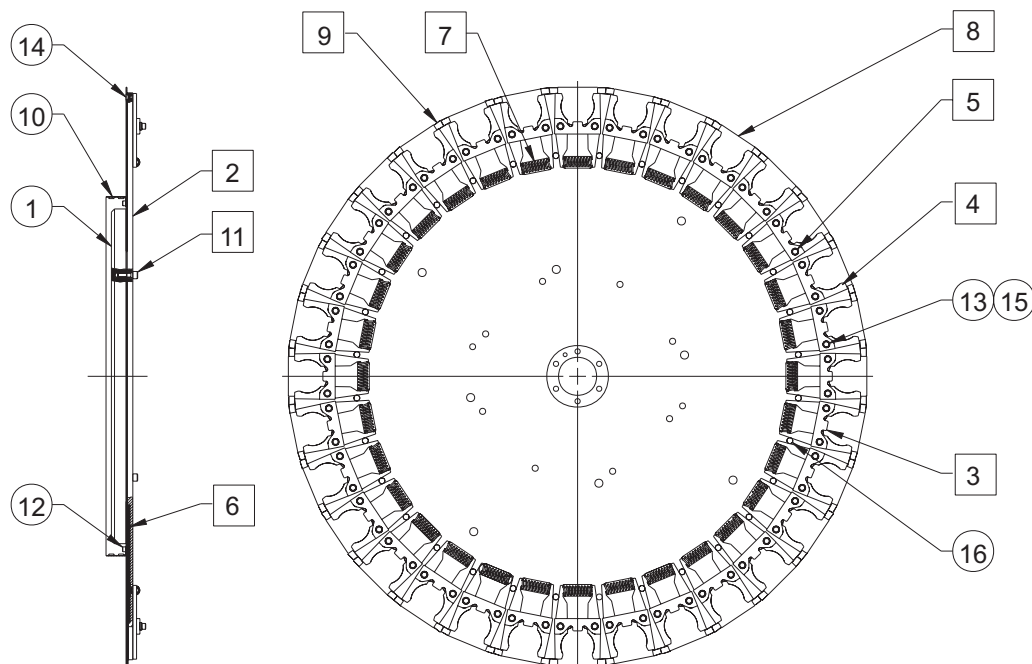


20 Pocket Tool Changer VF- 0/1/2

30-0161 20 POCKET TOOL CHANGER ASSEMBLY

IT	QTY	PART No.	DESCRIPTION	IT	QTY	PART No.	DESCRIPTION
1	1	20-7029B	TOOL HOLDING ARM, MACHINED	48	2	40-1650	BHCS, 10-32 x 1/4
2	1	20-7030E	TOOL CARRIAGE	49	6	40-1669	BHCS, 8-32 x 3/8"
3	1	20-7033F	HOLDING PLATE	50	10	40-1697	SHCS, 1/4-20 x 3/4"
4	1	20-7035G	VERTICAL AXLE	51	4	40-1702	FHCS, 10-32 x 5/8"
5	1	20-7038A	BEARING HOUSING, TOOL CHANGER	52	40	40-1704	FHCS, 10-32 x 1/4"
6	1	20-7039T	GENEVA DRIVE	53	6	40-1750	BHCS, 10-32 x 3/8"
7	1	20-7236	MOTOR MOUNTING PLATE	54	2	40-1800	SHCS, 8-32 x 3/4"
8	1	20-7352B	20 TOOL CAROUSEL	55	4	40-1803	SHCS, 8-32 x 1 1/4"
9	1	20-7353	20 POCKET GENEVA STAR	56	12	40-1850	SHCS, 10-32 x 3/8"
10	1	20-7475	ARM, SLIP CLUTCH	57	40	40-1860	SHCS, 1/4-20 x 7/8"
11	1	20-7476	HUB, SLIP CLUTCH	58	4	40-1970	FHCS, 1/4-28 x 1"
12	1	22-2065	LOCATING PIN	59	4	40-2000	SHCS, 1/4-20 x 5/8"
13	1	22-7026A	CAM, GENEVA DRIVER	60	4	43-1602	HBB, 1/2-13 x 3"
14	1	22-7034	SPACER, CAM FOLLOWER	61	5	43-7000	HBB, 5/16-18 x 1 3/4"
15	20	22-7067F	KEY, EXTRACTOR	62	1	44-1622	SSS, K CUP PT 1/4-20 x 1/4"
16	2	22-7106	'V' TRACK, TOOL CHANGER	63	1	44-1710	SSS, CUP PT 1/4-20 x 3/8"
17	3	22-7163	RIDER - TRAP DOOR	64	40	45-0045	WASHER, BLK HRD 1/4 x 1/8"THK
18	1	22-7255A	TOOL #1 STANDOFF	65	1	45-0050	WASHER, 5702-313-120
19	2	22-7263	SWITCH MOUNTING BLOCK	66	17	45-1600	WASHER, SPLIT LOCK 5/16 MED.
20	1	22-7477	PRESSURE PLATE	67	2	45-1603	WASHER, SPLIT LOCK #8 MED.
21	40	22-9256	BUSHING, EXTRACTOR	68	1	45-1725	WASHER, FLAT CUT 3/4"
22	40	22-9574A	CT-EXTRACTOR	69	4	45-1740	WASHER, BLACK HARD 1/2"
23	20	24-2010	COMPRESSION SPRING	70	10	45-1800	WASHER, SPLIT LOCK 1/4" MED.
24	20	24-9257	EXTRACTOR SPRING	71	2	45-2020	WASHER, NYLON
25	1	25-7036	CAP, TOOL CHANGER	72	1	45-2021	WASHER, DELRIN 2"
26	1	25-7162	CONNECTOR BRACKET	73	1	46-1705	LOCK NUT, ELASTIC, 3/4-10
27	1	25-7168	DOOR OPENER BRACKET	74	20	48-0002	PIN, SPRING 7/32 x 7/8"
28	1	25-7237C	TOOL CHANGER COVER	75	1	48-0005	PIN, DOWEL 3/16 x 3/8"
29	1	25-7238C	TRAP DOOR, TOOL CHANGER	76	1	48-0019	PIN, DOWEL 1/4 x 5/8"
30	20	25-7249	SLIDING PANEL	77	1	48-0020	PIN, DOWEL 1/4 x 1"
31	20	25-7250B	SLIDING PANEL COVER	78	1	48-1661	PIN, DOWEL 5/16 x 1 1/4"
32	1	25-7570	TOOL CHANGER, NUMBER RING	79	2	48-1750	DOWEL PIN, 1/2 x 1 1/2"
33	1	25-9253	CONDUIT MOUNTING PLATE	80	2	51-0010	BEARING, DEEP GRV #6206-2NSL
34	2	26-7239	SPACER RING	81	1	51-0012	BRNG LOCKNUT, BH-06
35	1	29-7612	CT TOOLING DECAL	82	1	51-6000	BRNG LOCKNUT, NT-05
36	1	32-1800	SHUTTLE MOTOR ASSM.	83	1	54-0010	CAM FOLLOWER, TOOL CHANGER
37	1	32-1900A	TURRET MOTOR ASSM.	84	2	54-0020	BUSHING, GUIDE WHEEL
38	2	32-2000	TELEMECHANIQUE 8 INCHES	85	4	54-0030	GUIDE WHEEL
39	2	32-2010	TELEMECHANIQUE 24 INCHES CABLE	86	2	54-0040	STANDARD BUSHING, GUIDE WHEEL
40	1	32-7011	TOOL CHANGER CONDUIT ASSM.	87	2	55-0010	SPRING WASHER, B2500-080
41	1	32-7610A	CONDUIT ASSM., TOOL CARRIAGE	88	1	57-7378	GASKET, TOOL CARRIAGE
42	6	40-1500	SHCS, 5/16-18 x 1"	89	1	57-7379	GASKET, TOOL HOLDING ARM
43	1	40-16091	BHCS, 10-32 x 1"	90	6	63-1031	CABLE CLAMP, 1/4"
44	40	40-16095	SHCS, 10-32 x 1/4"	91	1	70-0050	PLT4S-M CABLE TIES, BLACK
45	3	40-1632	SHCS, 1/4-20 x 1/2"	92	1	75-15721	LARGE MOLEX HOUSING MALE
46	6	40-16385	SHCS, 5/16-18 x 3/4"	93	1	78-1996	SPLIT FLEX TUBING 1/2" I.D.
47	4	40-16413	MSHCS, M3 x 5	94	1	79-1000	WIRE CHANNEL, 1" x 2", 1.75 FT.
				95	1	79-1001	COVER, 1" WIRE CHANNEL, 1.75 FT.

* DESIGNATES REMOVED PART

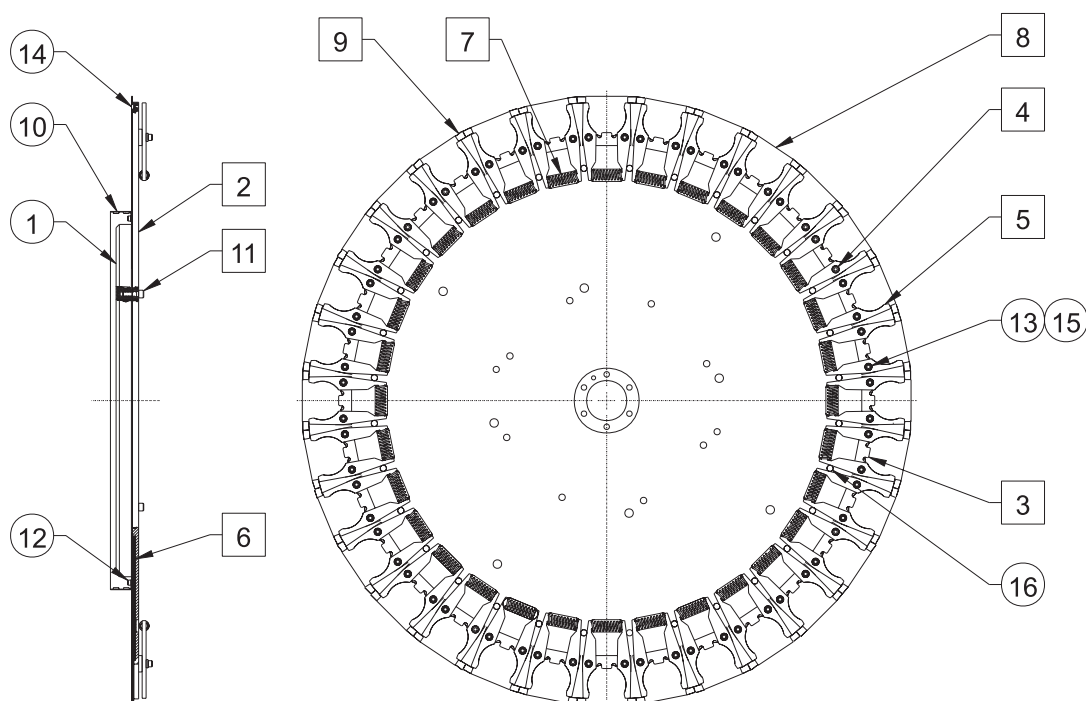


IT	QTY	PART NO.	TITLE
1	1	20-9193	CAROUSEL SUPPORT PLATE
2	1	20-9324	CAROUSEL, 32 TOOL
3	32	22-7067F	KEY, EXTRACTOR
4	64	22-7166A	EXTRACTOR, BT-40 TOOL CHN
5	64	22-9256	BUSHING, EXTRACTOR
6	32	24-2010A	COMPRESSION SPRING
7	32	24-9257	SPRING, EXTRACTOR
8	32	25-7249	SLIDING PANEL
9	32	25-9328	32 TOOL SLIDING PANEL COVER
10	1	25-9333	NUMBER RING, 32 T/C
11	8	40-1500	SHCS, 5/16-18 X 1"
12	64	40-16095	SHCS, 10-32 x 1/4"
13	64	40-1697	SHCS, 1/4-20 x 3/4"
14	64	40-1704	FHCS, 10-32 x 1/4"
15	64	45-0045	WASHER, BLK HARD 1/4" x 1/8" THK.
16	32	48-0004	SPRING PIN, 3/8" x 1"

☐ DESIGNATES PARTS AVAILABLE FROM HAAS SERVICE DEPT.

* DESIGNATES REMOVED PART

32 Tool Carousel Assembly (BT) VF 3-10

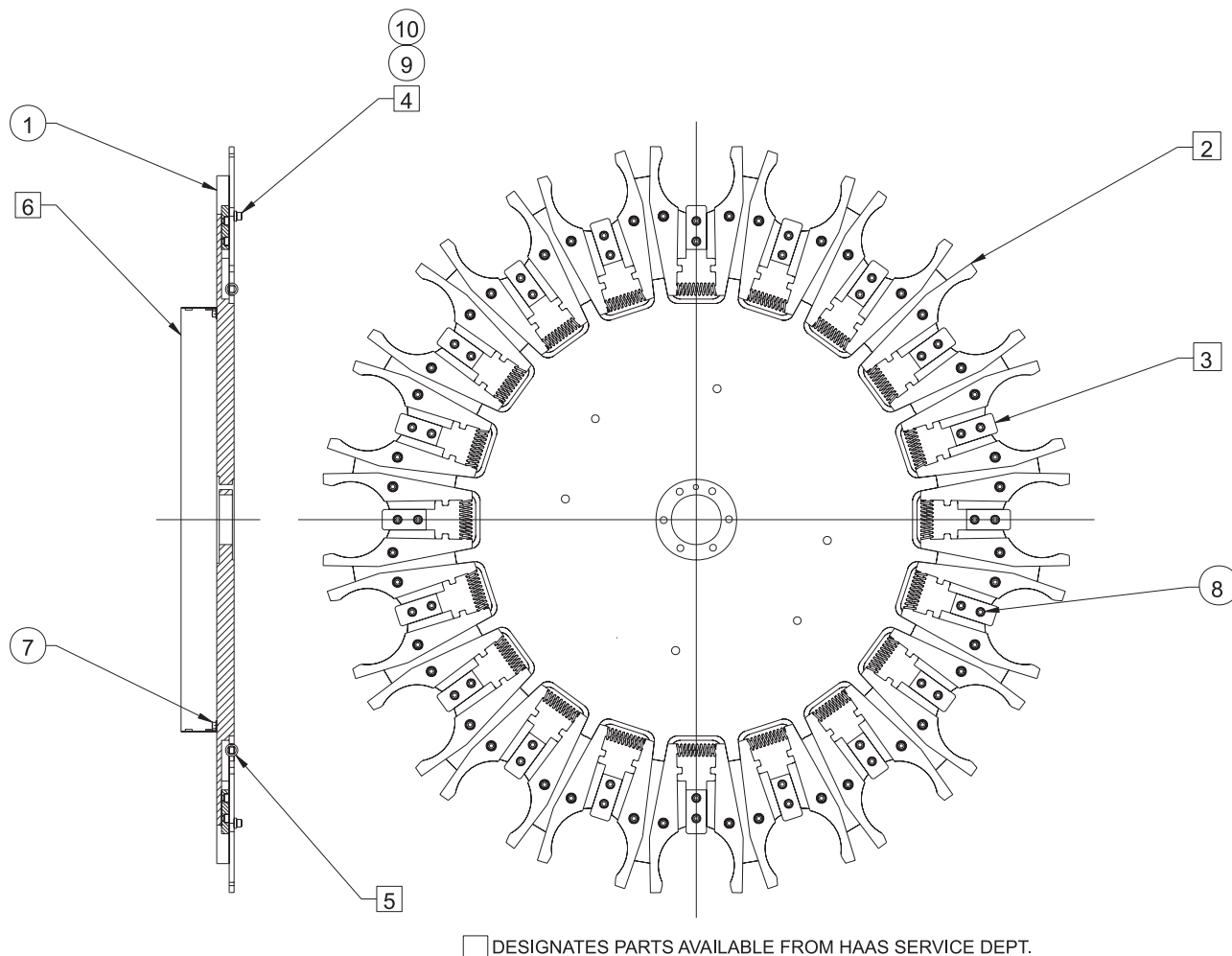


☐ DESIGNATES PARTS AVAILABLE FROM HAAS SERVICE DEPT.

IT	QTY	PART NO.	TITLE
1	1	20-9193	CAROUSEL SUPPORT PLATE
2	1	20-9324	CAROUSEL, 32 TOOL
3	32	22-7067F	KEY, EXTRACTOR
4	64	22-9574A	CT EXTRACTOR
5	64	22-9256	BUSHING, EXTRACTOR
6	32	24-2010A	COMPRESSION SPRING
7	32	24-9257	SPRING, EXTRACTOR
8	32	25-7249	SLIDING PANEL
9	32	25-9328	32 TOOL SLIDING PANEL COVER
10	1	25-9333	NUMBER RING, 32 T/C
11	8	40-1500	SHCS, 5/16-18 X 1"
12	64	40-16095	SHCS, 10-32 x 1/4"
13	64	40-1697	SHCS, 1/4-20 x 3/4"
14	64	40-1704	FHCS, 10-32 x 1/4"
15	64	45-0045	WASHER, BLK HARD 1/4" x 1/8" THK.
16	32	48-0004	SPRING PIN, 3/8" x 1"

* DESIGNATES REMOVED PART

32 Tool Carousel Assembly Assembly (CT) VF 3-10

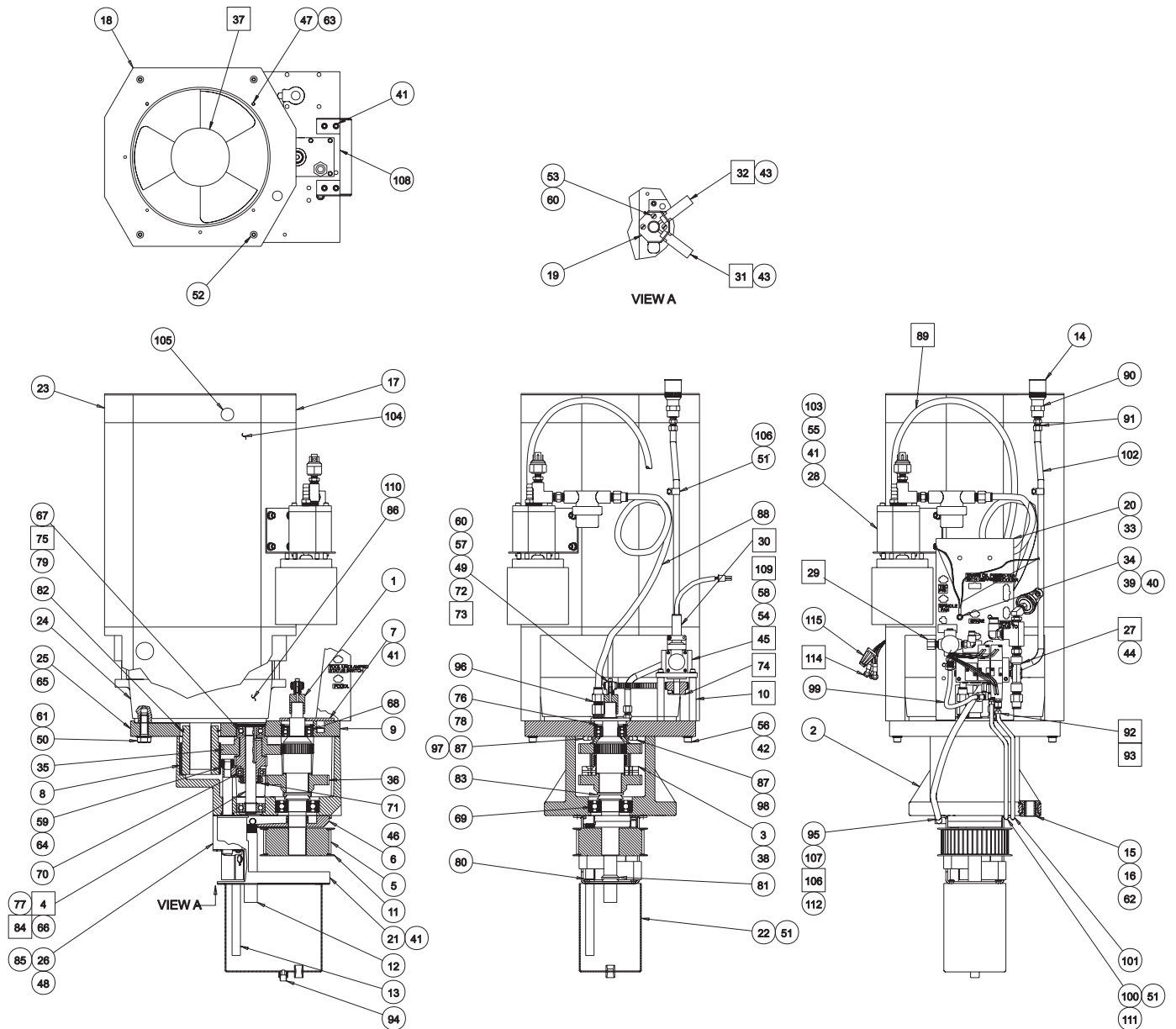


50T CAROUSEL ASSEMBLY - CT

ITEM	QTY	DWG. NO.	DESCRIPTION
1	1	20-9296	50 TAPER CAROUSEL - 20 TOOL
2	40	20-9297	EXTRACTION FINGER 50 TAPER
3	20	20-9298	ALIGNMENT KEY 50 TAPER
4	40	22-9256	BUSHING, EXTRACTOR,
5	20	24-9257	SPRING, EXTRACTOR, VF-ALL
6	1	25-9349	20 TOOL NUMBER RING 50 T
7	5	40-16095	SHCS, 10-32 X 1/4
8	40	40-1631	SHCS, 1/4-20 X 3/8
9	40	40-1860	SHCS, 1/4-20 X 7/8
10	40	45-0045	WSHR, BLK HRD 1/4 X 1/8 THK

* DESIGNATES REMOVED PART

50 Taper Carousel Assembly (CT) VF 6-10



□ DESIGNATES PARTS AVAILABLE FROM HAAS SERVICE DEPT.

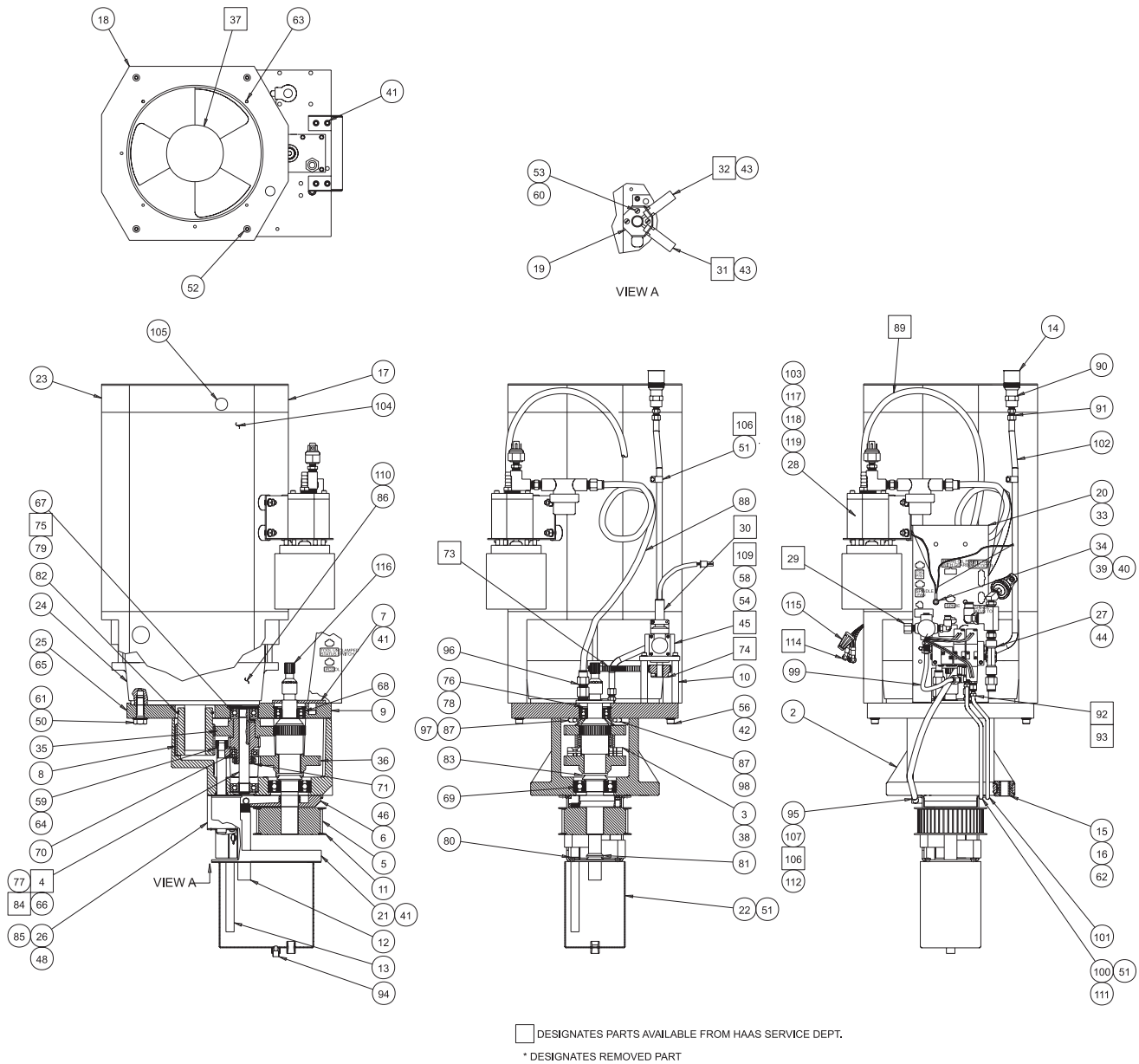
VF 1-11 Gearbox Assembly 15 HP



VF 1-11 Gearbox Assembly 15 HP

ITEM	QTY	PART	DESCRIPTION	ITEM	QTY	PART	DESCRIPTION
1	1	20-0064	ADAPTER ENCODER PULLEY	59	1	45-1682	WASHER, SPLIT LOCK 7/16MED
2	1	20-7011D	HOUSING GEARBOX MACHINING	60	4	45-1700	WASHER, INTERNAL LOCK #8
3	1	20-7062	BRG FORK, GEAR CLUSTER	61	3	45-1740	WASHER, BLACK HARD 1/2
4	1	20-7064	TRANSFER SHAFT	62	6	45-1851	WASHER TRANS
5	1	20-7374	1 1/8 SPROCKET	63	4	46-1617	NUT, HEX 8-32
6	1	20-7430A	OIL CATCH PAN	64	1	46-1654	NUT, HEX 7/16-20
7	1	20-7435	OILER PLATE	65	2	48-0020	PIN, DOWEL 1/4 X 1
8	1	20-9125A	MOTOR SHAFT GEAR	66	1	48-0050	PIN, DOWEL 1/8 X 7/16
9	1	20-9126	TOP PLATE GEARBOX	67	2	51-2031	BRNG. RADIAL OPEN 6303
10	4	22-7260	ENCODER STANDOFF	68	1	51-2032	BRNG. RADIAL OPEN 6205
11	2	22-7376	SPROCKET FLANGE	69	1	51-2033	BRNG. RADIAL OPEN 6306
12	1	22-7445A	DRAIN TUBE DRY SUMP	70	1	51-2034	BRNG. RADIAL OPEN 6005
13	1	22-7446	PICK UP TUBE DRY SUMP	71	1	51-2041	BRNG LOCKNUT BH-05
14	1	22-7487	OIL FILL CAP MODIFIED	72	1	54-1013	DRIVE SPROCKET .250 RTAP
15	12	22-7520A	ISOLATOR TRANS	73	1	54-2125	DRIVE BELT HTD 300-3M-09
16	6	22-7521A	SPACER TRANS	74	1	54-7127	DRIVE SPROCKET.375 RTAP
17	1	25-0107	MOTOR SHROUD VMC	75	2	55-0035	SPRING WASHER, BS-204
18	1	25-0108	FAN BRACKET MOTOR SHROUD	76	2	55-0036	SPRING WASHER, BS-205
19	1	25-7264	SWITCH MOUNTING BRACKET	77	1	56-0060	SNAP RING, N5100-66
20	1	25-7336	SOLENOID MOUNTING BRACKET	78	1	56-0070	SNAP RING N5000-187
21	1	25-7433	SUMP BRACKET	79	1	56-2087	SNAP RING, N5000-206
22	1	25-7434	SUMP TANK	80	1	57-0001	OIL SEAL
23	1	29-0022	SHROUD CAUTION DECAL	81	1	57-0002	OIL SEAL
24	1	29-7399	TRANSMISSION MOTOR LABEL	82	1	57-0006	O-RING 2-328 BUNA
25	1	29-9128	LABEL, TRANSMISSION	83	1	57-0013	V-RING SEAL CR 400280
26	1	30-3130C	SHIFTER ASSY	84	2	57-0058	O-RING, 2-014 BUNA
27	1	30-3146	AIR SOLENOID ASSY MAC TP	85	1	57-0105	QUAD RING, Q4-114 VITON
28	1	30-3260B	OIL GEAR PUMP ASSY	86	1	57-7573A	TRANS MOTOR GASKET
29	1	30-3270A	PRECHARGE REGULATOR ASSY	87	2	58-16752	90 DEG. COMPRESSION TILT
30	1	32-1455D	RTAP ENCODER CABLE	88	5.75FT	58-2001	POLYU HOSE 1/2OD X 3/8ID
31	1	32-2010	24 LIMIT SWITCH	89	2.5FT	58-2020	3/8 OD NATURAL TUBING
32	1	32-2011	TELMECH. 30 IN CABLE ASSY	90	1	58-2065	COUPLING, 1/4NPT
33	1	33-3200	SOLENOID BRKT CABLE ASSY	91	2	58-2070	1/4NPT MALE TO 3/8 COMP
34	1	33-5008	GRND STRP SPNDL MTR SHRD	92	4	58-2100	SLEEVE LUBE ASSY
35	1	35-7065A	TRANSFER GEAR ASSY	93	4	58-2110	SLEEVE NUTS LUBE ASSY
36	1	35-7170A	DRIVE SHAFT ASSY	94	1	58-2745	MAGNETIC OIL PLUG
37	1	36-3035	SPINDLE FAN ASSY	95	1	58-3616	3/8 90 DEG. ELBOW 1/4 NPT
38	2	40-1602	FHCS, 1/4-28 X 5/8	96	1	58-3657	1/4 FEMALE 1/8 MALE ADPT
39	1	40-1628	SHCS, 1/4-20 X 1/4	97	1	58-7357	TOP PLATE TUBE - A
40	1	40-1630	SHCS, 1/4-20 X 5/16	98	1	58-7358A	TOP PLATE TUBE - B
41	20	40-1632	SHCS 1/4-20 X 1/2	99	1	58-7377	AIR REG / SOLENOID TUBE
42	8	40-16385	SHCS 5/16-18 X 3/4	100	1	58-7635	LOW GEAR TUBE VF-3
43	4	40-16413	MSHCS, M3 X 5	101	1	58-7636	HIGH GEAR TUBE VF-3
44	2	40-1644	SHCS, 10-32 X 1 1/2	102	1	58-9114B	TRANS FILL TUBE
45	4	40-1645	SHCS, 10-32 X 5/8	103	1	59-0027	HOSE CLAMP 1/2 HOSE
46	5	40-16455	SHCS, 10-32 X 7/8	104	2	59-0046	SOUND COAT SHROUD RT/LT
47	4	40-1669	BHCS, 8-32 X 3/8	105	4	59-1482	NYLON FINISH PLUG, 13/16
48	4	40-1700	SHCS, 10-32 X 2	106	2	59-2040	CABLE CLAMP, 7/16
49	1	40-1800	SHCS 8-32 X 3/4	107	1	59-4006	HOSE CRIMP, 35/64
50	3	40-1830	HNB, 1/2-13 X 1 3/4	108	0.5FT	59-7130	PROTECTIVE STRIP
51	7	40-1850	SHCS 10-32 X 3/8	109	1	60-1810	SHAFT ENCODER 2000 LINE
52	4	40-1981	FBHCS 1/4-20 X 1/2	110	1	62-3010	SPINDLE MTR, 10 HP
53	3	41-1500	PPHS, 8-32 X 3/8	111	2	63-0001	NYLON CABLE CLAMP 1/2
54	4	44-0003	SSS HALF DOG PT 1/4-20X1	112	1	63-1031	CABLE CLAMP 1/4
55	4	45-0040	WASHER BLK HARD 1/4 A325	113	25	70-0020	PLT 1.5M CABLE TIES
56	4	45-1600	WASHER SPLIT LOCK 5/16MED	114	3	76-2420	CRIMP RING, 12-10 10 STUD
57	1	45-1603	WASHER, SPLIT LOCK #8 MED.	115	1	77-8001	WIRE NUT, IDEAL #30-076
58	4	45-1620	WASHER SPLIT LOCK #10 MED				

* DESIGNATES REMOVED PART

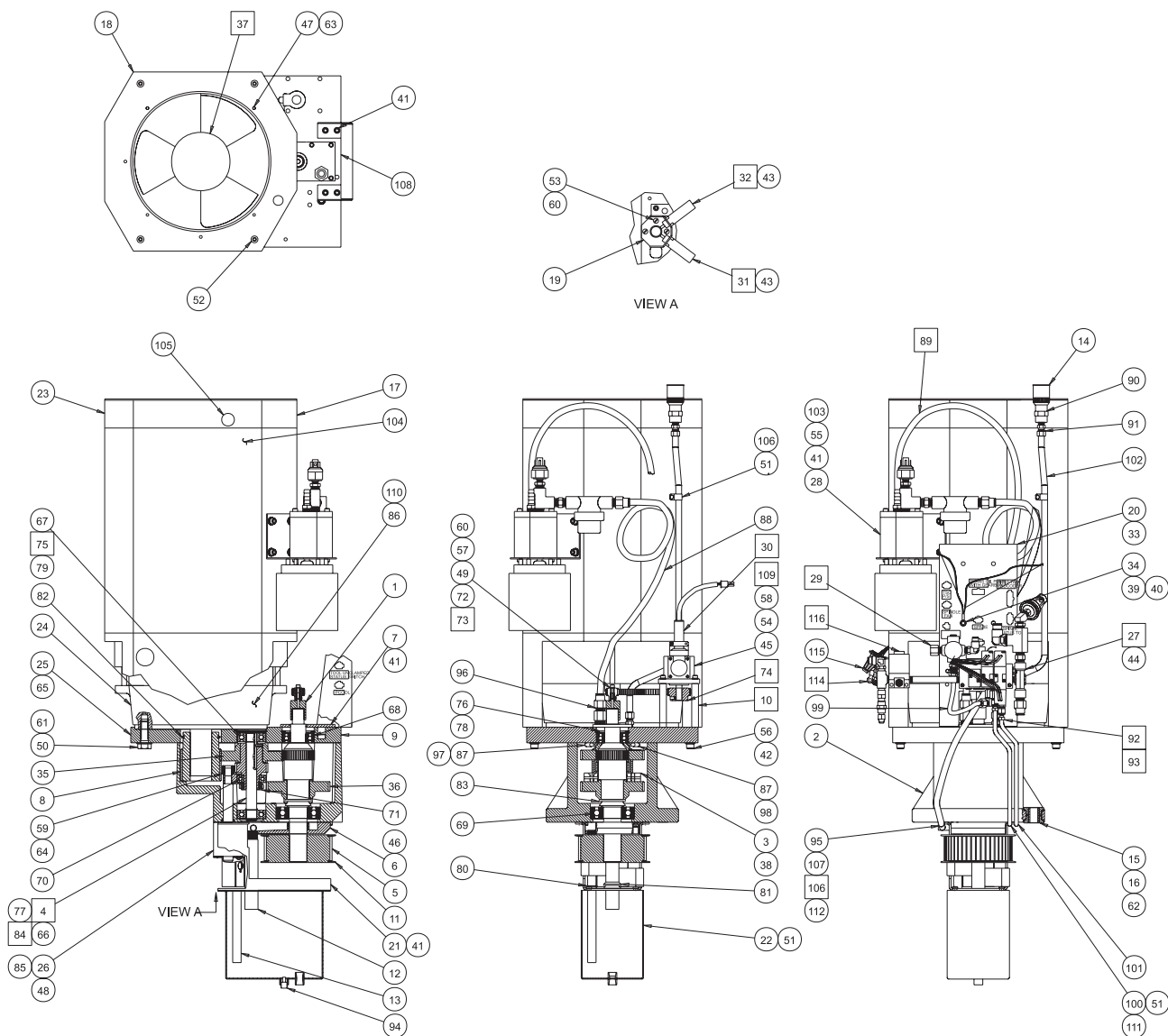


VF 1-11 Gearbox Assembly HT10K



VF 1-11 Gearbox Assembly HT10K

ITEM	QTY	PART	DESCRIPTION	ITEM	QTY	PART	DESCRIPTION
1*	1	20-0064	"REMOVED" ADAPTER ENCODER PULLEY	60	3	45-1700	WASHER, INTERNAL LOCK #8
2	1	20-7011D	HOUSING GEARBOX MACHINING	61	3	45-1740	WASHER, BLACK HARD 1/2
3	1	20-7062	BRG FORK, GEAR CLUSTER	62	6	45-1851	WASHER TRANS
4	1	20-7064	TRANSFER SHAFT	63	4	46-1617	NUT, HEX 8-32
5	1	20-7374	1 1/8 SPROCKET	64	1	46-1654	NUT, HEX 7/16-20
6	1	20-7430A	OIL CATCH PAN	65	2	48-0020	PIN, DOWEL 1/4 X 1
7	1	20-7435	OILER PLATE	66	1	48-0050	PIN, DOWEL 1/8 X 7/16
8	1	20-9125A	MOTOR SHAFT GEAR	67	2	51-2031	BRNG. RADIAL OPEN 6303
9	1	20-9126	TOP PLATE GEARBOX	68	1	51-2032	BRNG. RADIAL OPEN 6205
10	4	22-7260	ENCODER STANDOFF	69	1	51-2033	BRNG. RADIAL OPEN 6306
11	2	22-7376	SPROCKET FLANGE	70	1	51-2034	BRNG. RADIAL OPEN 6005
12	1	22-7445A	DRAIN TUBE DRY SUMP	71	1	51-2041	BRNG LOCKNUT BH-05
13	1	22-7446	PICK UP TUBE DRY SUMP	72*	1	54-1013	"REMOVED" DRIVE SPROCKET .250 RTAP
14	1	22-7487	OIL FILL CAP MODIFIED	73	1	54-2125	DRIVE BELT HTD 300-3M-09
15	12	22-7520A	ISOLATOR TRANS	74	1	54-7127	DRIVE SPROCKET.375 RTAP
16	6	22-7521B	SPACER TRANS	75	2	55-0035	SPRING WASHER, BS-204
17	1	25-0107	MOTOR SHROUD VMC	76	2	55-0036	SPRING WASHER, BS-205
18	1	25-0108	FAN BRACKET MOTOR SHROUD	77	3	56-0060	SNAP RING, N5100-66
19	1	25-7264	SWITCH MOUNTING BRACKET	78	1	56-0070	SNAP RING N5000-187
20	1	25-7336	SOLENOID MOUNTING BRACKET	79	1	56-2087	SNAP RING, N5000-206
21	1	25-7433	SUMP BRACKET	80	1	57-0001	OIL SEAL
22	1	25-7434	SUMP TANK	81	1	57-0002	OIL SEAL
23	1	29-0022	SHROUD CAUTION DECAL	82	1	57-0006	O-RING 2-328 BUNA
24	1	29-7399	TRANSMISSION MOTOR LABEL	83	1	57-0013	V-RING SEAL CR 400280
25	1	29-9128	LABEL, TRANSMISSION	84	2	57-0058	O-RING, 2-014 BUNA
26	1	30-3130C	SHIFTER ASSY	85	1	57-0105	QUAD RING, Q4-114 VITON
27	1	30-3146	AIR SOLENOID ASSY MAC TP	86	1	57-7573A	TRANS MOTOR GASKET
28	1	30-3260B	OIL GEAR PUMP ASSY	87	2	58-16752	90 DEG. COMPRESSION TILT
29	1	30-3270A	PRECHARGE REGULATOR ASSY	88	5.75FT	58-2001	POLYU HOSE 1/2OD X 3/8ID
30	1	32-1455D	RTAP ENCODER CABLE	89	2.5FT	58-2020	3/8 OD NATURAL TUBING
31	1	32-2010	24 LIMIT SWITCH	90	1	58-2065	COUPLING, 1/4NPT
32	1	32-2011	TELMECH. 30 IN CABLE ASSY	91	2	58-2070	1/4NPT MALE TO 3/8 COMP
33	1	33-3200	SOLENOID BRKT CABLE ASSY	92	4	58-2100	SLEEVE LUBE ASSY
34	1	33-5008	GRND STRP SPNDL MTR SHRD	93	4	58-2110	SLEEVE NUTS LUBE ASSY
35	1	35-7065A	TRANSFER GEAR ASSY	94	1	58-2745	MAGNETIC OIL PLUG
36	1	35-7170A	DRIVE SHAFT ASSY	95	1	58-3616	3/8 90 DEG.ELBOW 1/4 NPT
37	1	36-3035	SPINDLE FAN ASSY	96	1	58-3657	1/4 FEMALE 1/8 MALE ADPT
38	2	40-1602	FHCS, 1/4-28 X 5/8	97	1	58-7357	TOP PLATE TUBE - A
39	1	40-1628	SHCS, 1/4-20 X 1/4	98	1	58-7358A	TOP PLATE TUBE - B
40	1	40-1630	SHCS, 1/4-20 X 5/16	99	1	58-7377	AIR REG / SOLENOID TUBE
41	12	40-1632	SHCS 1/4-20 X 1/2	100	1	58-7635	LOW GEAR TUBE VF-3
42	8	40-16385	SHCS 5/16-18 X 3/4	101	1	58-7636	HIGH GEAR TUBE VF-3
43	4	40-16413	MSHCS, M3 X 5	102	1	58-9114B	TRANS FILL TUBE
44	2	40-1644	SHCS, 10-32 X 1 1/2	103	1	59-0027	HOSE CLAMP 1/2 HOSE
45	4	40-1645	SHCS, 10-32 X 5/8	104	2	59-0046	SOUNDcoat SHROUD RT/LT
46	5	40-16455	SHCS, 10-32 X 7/8	105	4	59-1482	NYLON FINISH PLUG, 13/16
47*	4	40-1669	"REMOVED" BHCS, 8-32 X 3/8	106	2	59-2040	CABLE CLAMP, 7/16
48	4	40-1700	SHCS, 10-32 X 2	107	1	59-4006	HOSE CRIMP, 35/64
49*	1	40-1800	"REMOVED" SHCS 8-32 X 3/4	108*	0.5FT	59-7130	"REMOVED" PROTECTIVE STRIP
50	3	40-1830	HBB, 1/2-13 X 1 3/4	109	1	60-1810	SHAFT ENCODER 2000 LINE
51	7	40-1850	SHCS 10-32 X 3/8	110	1	36-3078	10K 10HP MOTOR KIT
52	4	40-1981	FBHCS 1/4-20 X 1/2	111	2	63-0001	NYLON CABLE CLAMP 1/2
53	3	41-1500	PPHS, 8-32 X 3/8	112	1	63-1031	CABLE CLAMP 1/4
54	4	44-0003	SSS HALF DOG PT 1/4-20X1	113	25	70-0020	PLT1.5M CABLE TIES
55*	4	45-0040	"REMOVED" WASHER BLK HARD 1/4 A325	114	3	76-2420	CRIMP RING, 12-10 10 STUD
56	4	45-1600	WASHER SPLIT LOCK 5/16MED	115	1	77-8001	WIRE NUT, IDEAL #30-076
57*	1	45-1603	"REMOVED" WASHER,SPLIT LOCK #8 MED.	116	1	20-0125	DRIVE SPROCKET ENCODER
58	4	45-1620	WASHER SPLIT LOCK #10 MED	117	4	57-0049	RUBBER STUD BUMPER
59	1	45-1682	WASHER,SPLIT LOCK 7/16MED	118	4	46-1625	NUT HEX BLK OX 1/4-20
				119	4	45-1800	WASHER SPLIT LOCK 1/4 MED



□ DESIGNATES PARTS AVAILABLE FROM HAAS SERVICE DEPT.

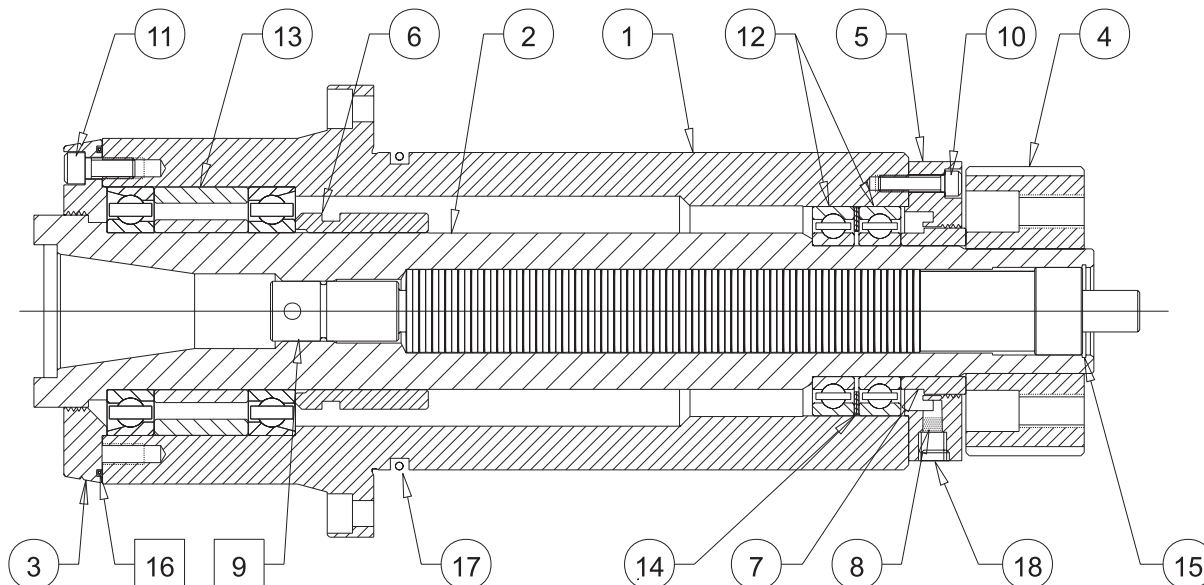
* DESIGNATES REMOVED PART

VF 1-11 Gearbox Assembly HT10K TSC



VF 1-11 Gearbox Assembly HT10K TSC

ITEM	QTY	PART	DESCRIPTION	ITEM	QTY	PART	DESCRIPTION
1	1	20-0064	ADAPTER ENCODER PULLEY	59	1	45-1682	WASHER, SPLIT LOCK 7/16MED
2	1	20-7011D	HOUSING GEARBOX MACHINING	60	4	45-1700	WASHER, INTERNAL LOCK #8
3	1	20-7062	BRG FORK, GEAR CLUSTER	61	3	45-1740	WASHER, BLACK HARD 1/2
4	1	20-7064	TRANSFER SHAFT	62	6	45-1851	WASHER TRANS
5	1	20-7374	1 1/8 SPROCKET	63	4	46-1617	NUT, HEX 8-32
6	1	20-7430A	OIL CATCH PAN	64	1	46-1654	NUT, HEX 7/16-20
7	1	20-7435	OILER PLATE	65	2	48-0020	PIN, DOWEL 1/4 X 1
8	1	20-9125A	MOTOR SHAFT GEAR	66	1	48-0050	PIN, DOWEL 1/8 X 7/16
9	1	20-9126	TOP PLATE GEARBOX	67	2	51-2031	BRNG. RADIAL OPEN 6303
10	4	22-7260	ENCODER STANDOFF	68	1	51-2032	BRNG. RADIAL OPEN 6205
11	2	22-7376	SPROCKET FLANGE	69	1	51-2033	BRNG. RADIAL OPEN 6306
12	1	22-7445A	DRAIN TUBE DRY SUMP	70	1	51-2034	BRNG. RADIAL OPEN 6005
13	1	22-7446	PICK UP TUBE DRY SUMP	71	1	51-2041	BRNG LOCKNUT BH-05
14	1	22-7487	OIL FILL CAP MODIFIED	72	1	54-1013	DRIVE SPROCKET .250 RTAP
15	12	22-7520A	ISOLATOR TRANS	73	1	54-2125	DRIVE BELT HTD 300-3M-09
16	6	22-7521A	SPACER TRANS	74	1	54-7127	DRIVE SPROCKET.375 RTAP
17	1	25-0107	MOTOR SHROUD VMC	75	2	55-0035	SPRING WASHER, BS-204
18	1	25-0108	FAN BRACKET MOTOR SHROUD	76	2	55-0036	SPRING WASHER, BS-205
19	1	25-7264	SWITCH MOUNTING BRACKET	77	1	56-0060	SNAP RING, N5100-66
20	1	25-7336	SOLENOID MOUNTING BRACKET	78	1	56-0070	SNAP RING N5000-187
21	1	25-7433	SUMP BRACKET	79	1	56-2087	SNAP RING, N5000-206
22	1	25-7434	SUMP TANK	80	1	57-0001	OIL SEAL
23	1	29-0022	SHROUD CAUTION DECAL	81	1	57-0002	OIL SEAL
24	1	29-7399	TRANSMISSION MOTOR LABEL	82	1	57-0006	O-RING 2-328 BUNA
25	1	29-9128	LABEL, TRANSMISSION	83	1	57-0013	V-RING SEAL CR 400280
26	1	30-3130C	SHIFTER ASSY	84	2	57-0058	O-RING, 2-014 BUNA
27	1	30-3146	AIR SOLENOID ASSY MAC TP	85	1	57-0105	QUAD RING, Q4-114 VITON
28	1	30-3260B	OIL GEAR PUMP ASSY	86	1	57-7573A	TRANS MOTOR GASKET
29	1	30-3270A	PRECHARGE REGULATOR ASSY	87	2	58-16752	90 DEG. COMPRESSION TILT
30	1	32-1455D	RTAP ENCODER CABLE	88	5.75FT	58-2001	POLYU HOSE 1/2OD X 3/8ID
31	1	32-2010	24 LIMIT SWITCH	89	2.5FT	58-2020	3/8 OD NATURAL TUBING
32	1	32-2011	TELMECH. 30 IN CABLE ASSY	90	1	58-2065	COUPLING, 1/4NPT
33	1	33-3200	SOLENOID BRKT CABLE ASSY	91	2	58-2070	1/4NPT MALE TO 3/8 COMP
34	1	33-5008	GRND STRP SPNDL MTR SHRD	92	4	58-2100	SLEEVE LUBE ASSY
35	1	35-7065A	TRANSFER GEAR ASSY	93	4	58-2110	SLEEVE NUTS LUBE ASSY
36	1	35-7170A	DRIVE SHAFT ASSY	94	1	58-2745	MAGNETIC OIL PLUG
37	1	36-3035	SPINDLE FAN ASSY	95	1	58-3616	3/8 90 DEGBOW 1/4 NPT
38	2	40-1602	FHCS, 1/4-28 X 5/8	96	1	58-3657	1/4 FEMALE 1/8 MALE ADPT
39	1	40-1628	SHCS, 1/4-20 X 1/4	97	1	58-7357	TOP PLATE TUBE - A
40	1	40-1630	SHCS, 1/4-20 X 5/16	98	1	58-7358A	TOP PLATE TUBE - B
41	20	40-1632	SHCS 1/4-20 X 1/2	99	1	58-7377	AIR REG / SOLENOID TUBE
42	8	40-16385	SHCS 5/16-18 X 3/4	100	1	58-7635	LOW GEAR TUBE VF-3
43	4	40-16413	MSHCS, M3 X 5	101	1	58-7636	HIGH GEAR TUBE VF-3
44	2	40-1644	SHCS, 10-32 X 1 1/2	102	1	58-9114B	TRANS FILL TUBE
45	4	40-1645	SHCS, 10-32 X 5/8	103	1	59-0027	HOSE CLAMP 1/2 HOSE
46	5	40-16455	SHCS, 10-32 X 7/8	104	2	59-0046	SOUND COAT SHROUD RT/LT
47	4	40-1669	BHCS, 8-32 X 3/8	105	4	59-1482	NYLON FINISH PLUG, 13/16
48	4	40-1700	SHCS, 10-32 X 2	106	2	59-2040	CABLE CLAMP, 7/16
49	1	40-1800	SHCS 8-32 X 3/4	107	1	59-4006	HOSE CRIMP, 35/64
50	3	40-1830	HCB, 1/2-13 X 1 3/4	108	0.5FT	59-7130	PROTECTIVE STRIP
51	7	40-1850	SHCS 10-32 X 3/8	109	1	60-1810	SHAFT ENCODER 2000 LINE
52	4	40-1981	FBHCS 1/4-20 X 1/2	110	1	36-3078	10K 10HP MOTOR KIT
53	3	41-1500	PPHS, 8-32 X 3/8	111	2	63-0001	NYLON CABLE CLAMP 1/2
54	4	44-0003	SSS HALF DOG PT 1/4-20X1	112	1	63-1031	CABLE CLAMP 1/4
55	4	45-0040	WASHER BLK HARD 1/4 A325	113	25	70-0020	PLT1.5M CABLE TIES
56	4	45-1600	WASHER SPLIT LOCK 5/16MED	114	3	76-2420	CRIMP RING, 12-10 10 STUD
57	1	45-1603	WASHER, SPLIT LOCK #8 MED.	115	1	77-8001	WIRE NUT, IDEAL #30-076
58	4	45-1620	WASHER SPLIT LOCK #10 MED	116	1	30-3276	PURGE SOLENOID VALVE ASSY

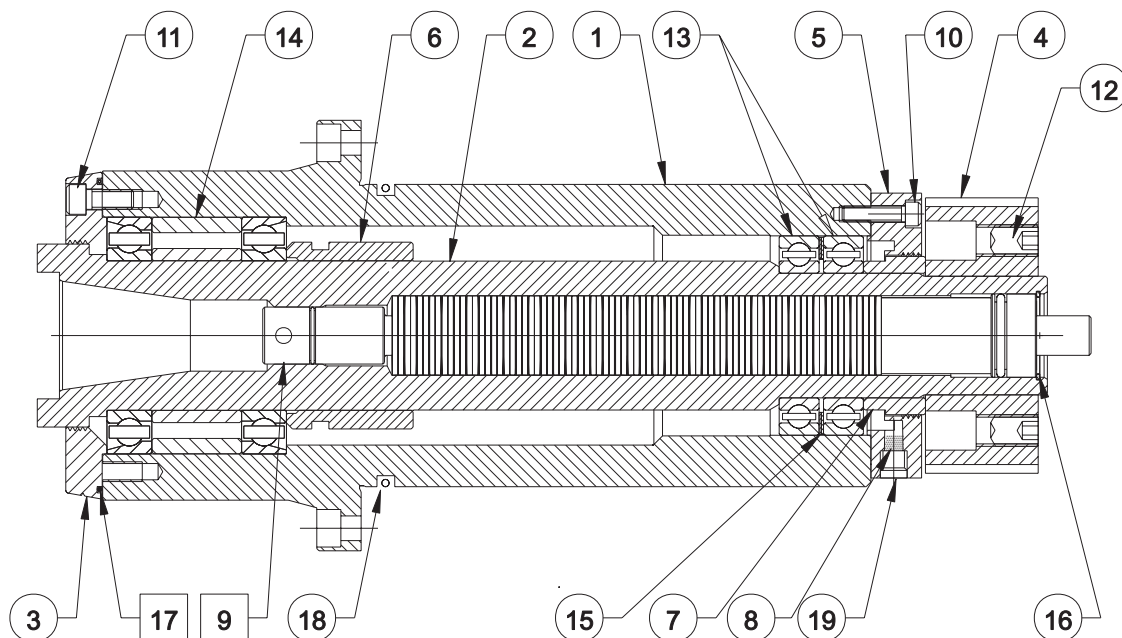


DESIGNATES PARTS AVAILABLE FROM HAAS SERVICE DEPT.

* DESIGNATES REMOVED PART

ITEM	QTY	PART NO.	DESCRIPTION
1	1	20-7016B	SPINDLE HOUSING
2	1	20-7018L	SPINDLE SHAFT
3	1	20-7022C	SPINDLE CAP TAPERED
4	1	20-7373	1 7/8 PULLEY
5	1	20-7442C	OIL INJECTION COVER
6	1	20-7530	LOCK, 60MM BEARING
7	1	20-7531	50MM LOCK 7500 SPINDLE
8	1	24-4200	BRONZE FILTER ELEMENT
9	1	30-3410D	DRAWBAR ASSY HIGH CLAMP
10	4	40-1610	SHCS 1/4-20 X 1
11	6	40-16385	SHCS 5/16-18 X 3/4
12	2	51-0021	BEARING 6010 OPEN
13	1	51-1012	BRNG DUPLEX MD 36MM SPCR
14	1	55-0020	WAVE WASHER W3118-035
15	1	56-0075	SNAP RING N5000-131
16	1	57-2984	O-RING 2-158 VITON
17	1	57-2990	O-RING 2-348 BUNA
18	1	58-1627	1/8-27 PIPE PLUG

Spindle Assembly 7.5

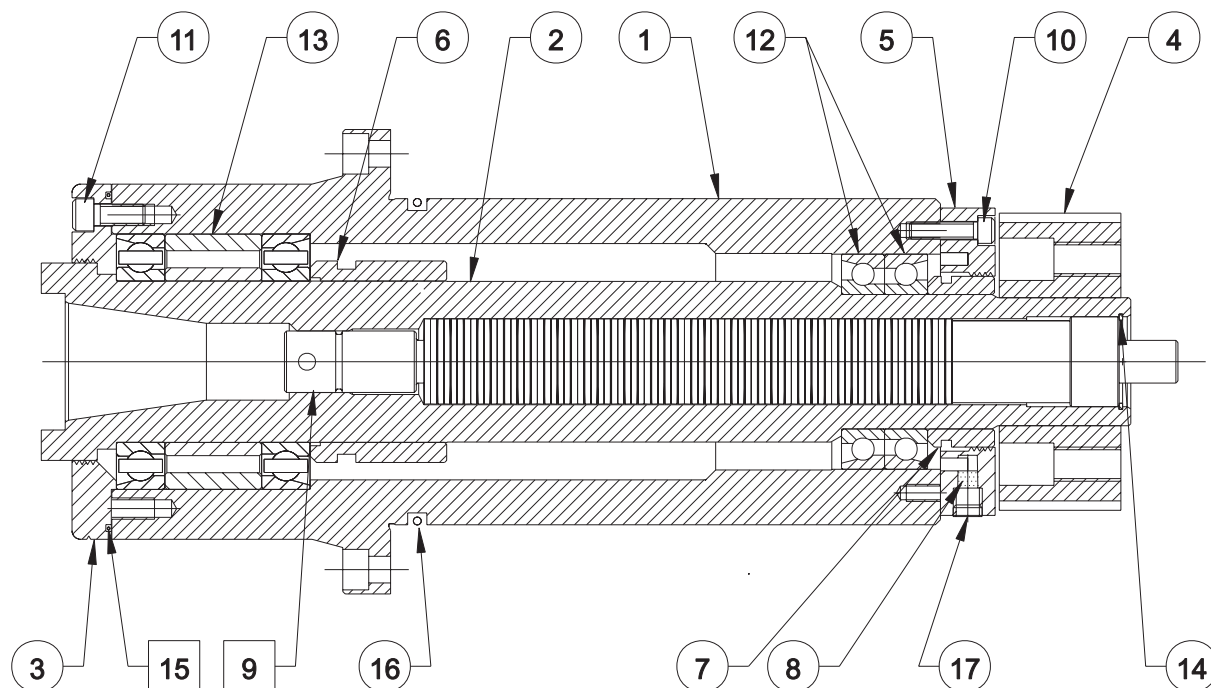


□ DESIGNATES PARTS AVAILABLE FROM HAAS SERVICE DEPT.

ITEM	QTY	PART NO.	DESCRIPTION
1	1	20-7016B	SPINDLE HOUSING
2	1	20-7018L	SPINDLE SHAFT
3	1	20-7022C	SPINDLE CAP TAPERED
4	1	20-7373	1 7/8 PULLEY
5	1	20-7442C	OIL INJECTION COVER
6	1	20-7530	LOCK 60MM BEARING
7	1	20-7531	50MM LOCK 7500 SPINDLE
8	1	24-4200	BRONZE FILTER ELEMENT
9	1	30-3415F	DRAWBAR ASSY HC TSCHP CARB
10	4	40-1610	SHCS 1/4-20 X 1
11	6	40-16385	SHCS 5/16-18 X 3/4
12	2	44-1698	SSS 1/2-13 X 3/4
13	2	51-0021	BEARING 6010 OPEN
14	1	51-1012	BRNG DUPLEX MD 36MM SPCR
15	1	55-0020	WAVE WASHER W3118-035
16	1	56-0075	SNAP RING N5000-131
17	1	57-2984	O-RING 2-158 VITON
18	1	57-2990	O-RING 2-348 BUNA
19	1	58-1627	1/8-27 PIPE PLUG

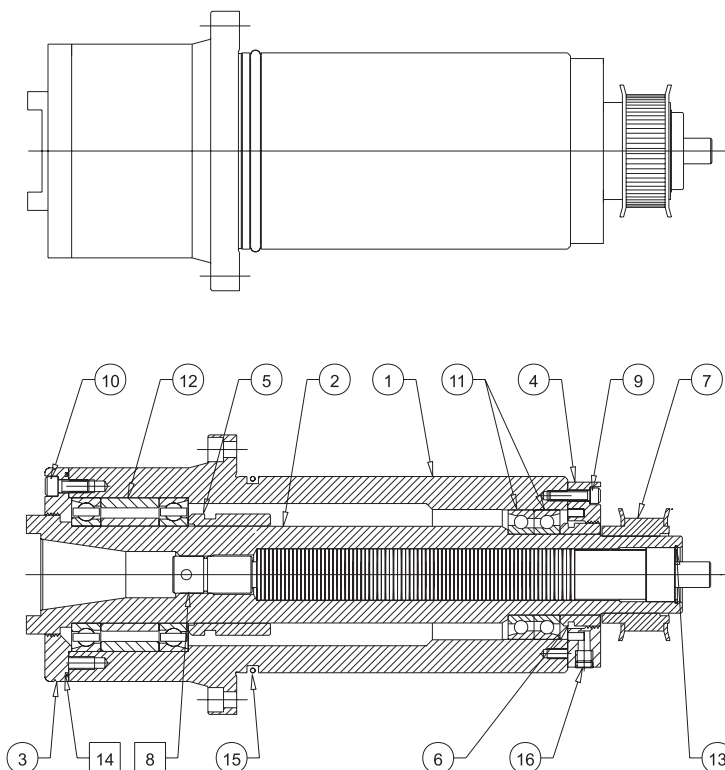
* DESIGNATES REMOVED PART

Spindle Assembly, 7.5K TSCHP



ITEM	QTY	PART NO.	DESCRIPTION
1	1	20-7016B	SPINDLE HOUSING
2	1	20-7018L	SPINDLE SHAFT
3	1	20-7022C	SPINDLE CAP TAPERED
4	1	20-7373	1 7/8 PULLEY
5	1	20-7442C	OIL INJECTION COVER
6	1	20-7530	LOCK 60MM BEARING
7	1	20-7532	LOCK 50MM ANG CNTACT BRNG
8	1	24-4200	BRONZE FILTER ELEMENT
9	1	30-3410D	DRAWBAR ASSY HIGH CLAMP
10	4	40-1610	SHCS 1/4-20 X 1
11	6	40-16385	SHCS 5/16-18 X 3/4
12	1	51-1002	LT ANG CONT DUPLEX
13	1	51-1012A	BRNG DUPLEX LT 36MM SPCR
14	1	56-0075	SNAP RING N5000-131
15	1	57-2984	O-RING 2-158 VITON
16	1	57-2990	O-RING 2-348 BUNA
17	1	58-1627	1/8-27 PIPE PLUG

10K Spindle Assembly

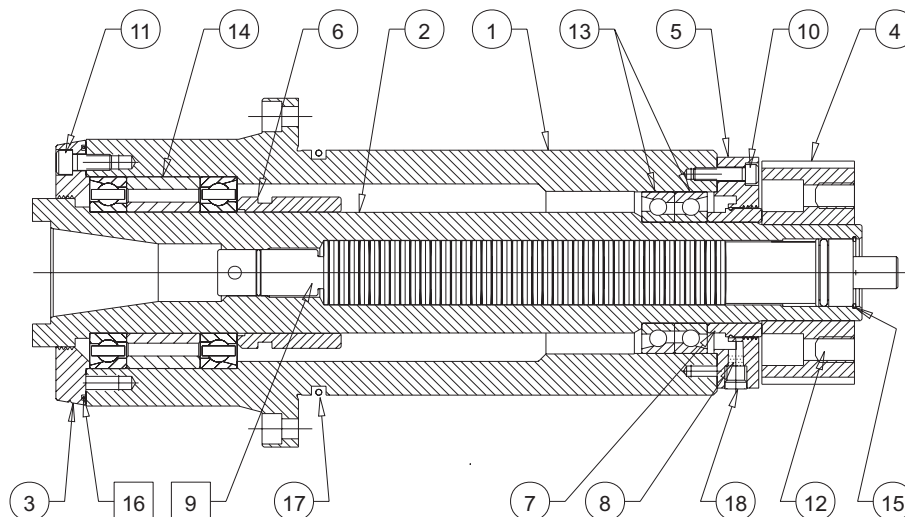


☐ DESIGNATES PARTS AVAILABLE FROM HAAS SERVICE DEPT.

ITEM	QTY	PART NO.	DESCRIPTION
1	1	20-7016B	SPINDLE HOUSING
2	1	20-7018L	SPINDLE SHAFT
3	1	20-7022C	SPINDLE CAP TAPERED
4	1	20-7442C	OIL INJECTION COVER
5	1	20-7530	LOCK, 60MM BEARING
6	1	20-7532	LOCK 50MM ANG CNTACT BRN
7	1	20-7539	28 TOOTH PULLEY MOD
8	1	30-3410D	DRAWBAR ASSY HIGH CLAMP
9	4	40-1610	SHCS 1/4-20 X 1
10	6	40-16385	SHCS 5/16-18 X 3/4
11	1	51-1002	LT ANG CONT DUPLEX
12	1	51-1012A	BRNG DUPLEX LT 36MM SPCR
13	1	56-0075	SNAP RING N5000-131
14	1	57-2984	O-RING 2-158 VITON
15	1	57-2990	O-RING 2-348 BUNA
16	1	58-1627	1/8-27 PIPE PLUG

* DESIGNATES REMOVED PART

10K/ 12K Spindle Assembly

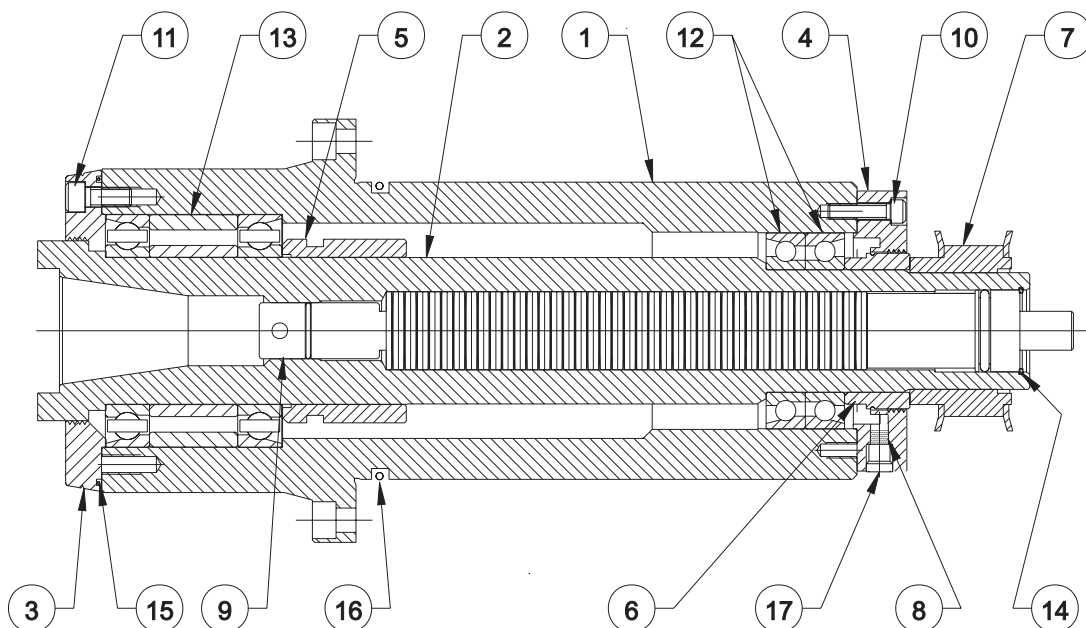


ITEM	QTY	PART NO.	DESCRIPTION	ITEM	QTY	PART NO.	DESCRIPTION
1	1	20-7016B	SPINDLE HOUSING	10	4	40-1610	SHCS 1/4-20 X 1
2	1	20-7018L	SPINDLE SHAFT	11	6	40-16385	SHCS 5/16-18 X 3/4
3	1	20-7022C	SPINDLE CAP TAPERED	12	2	44-1698	SSS 1/2-13 X 3/4
4	1	20-7373	1 7/8 PULLEY	13	1	51-1002	LT ANG CONT DUPLEX
5	1	20-7442C	OIL INJECTION COVER	14	1	51-1012A	BRNG DUPLEX LT 36MM SPCF
6	1	20-7530	LOCK 60MM BEARING	15	1	56-0075	SNAP RING N5000-131
7	1	20-7532	LOCK 50MM ANG CNTACT BRNG	16	1	57-2984	O-RING 2-158 VITON
8	1	24-4200	BRONZE FILTER ELEMENT	17	1	57-2990	O-RING 2-348 BUNA
9	1	30-3415F	DRAWBAR ASSY HC TSCHP CARB	18	1	58-1627	1/8-27 PIPE PLUG

☐ DESIGNATES PARTS AVAILABLE FROM HAAS SERVICE DEPT.

* DESIGNATES REMOVED PART

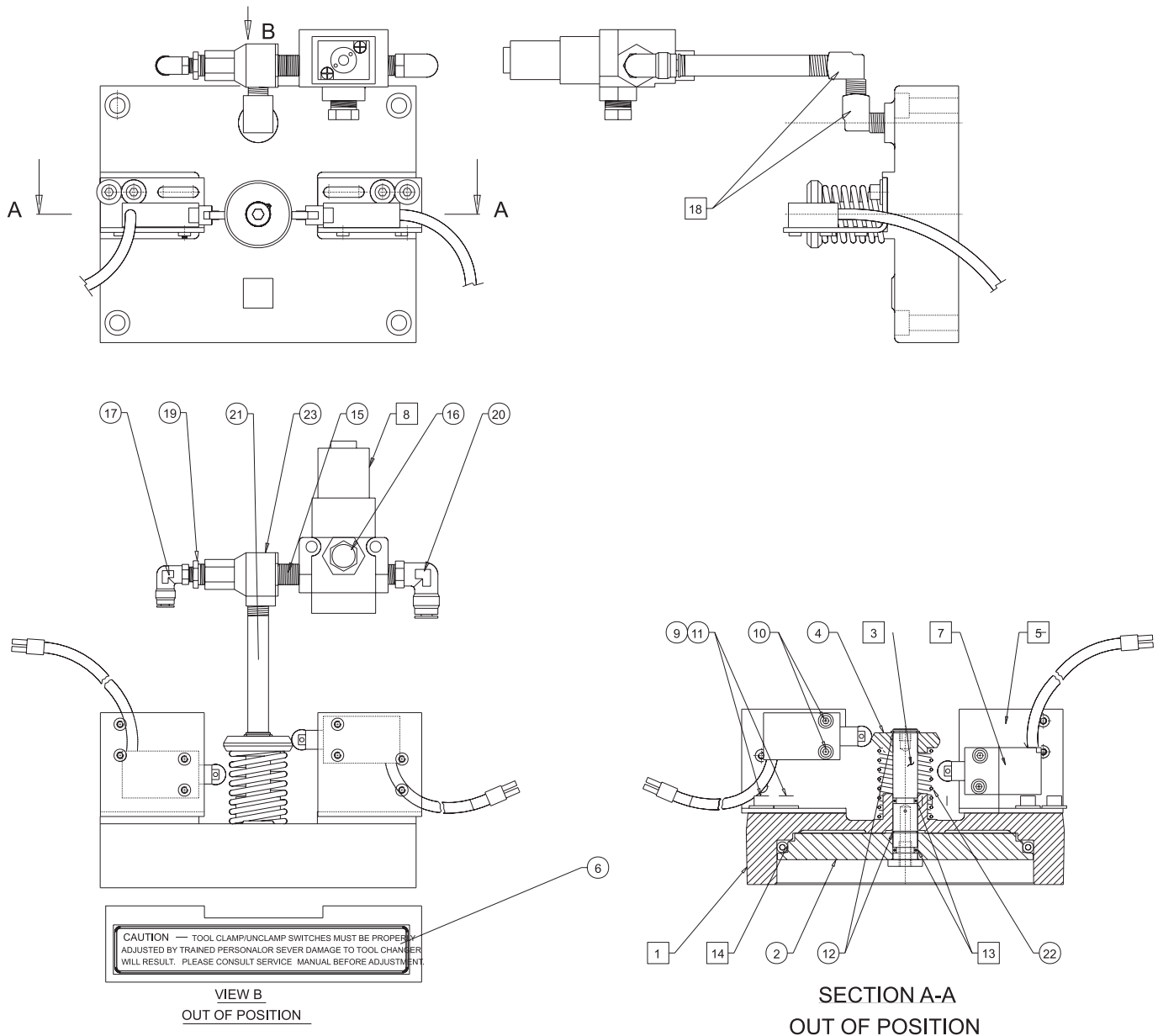
Spindle Assembly 10K TSCHP



ITEM	QTY	PART NO.	DESCRIPTION
1	1	20-7016B	SPINDLE HOUSING
2	1	20-7018L	SPINDLE SHAFT
3	1	20-7022C	SPINDLE CAP TAPERED
4	1	20-7442C	OIL INJECTION COVER
5	1	20-7530	LOCK 60MM BEARING
6	1	20-7532	LOCK 50MM ANG CNTACT BRNG
7	1	20-7539	28 TOOTH PULLEY MOD
8	1	24-4200	BRONZE FILTER ELEMENT
9	1	30-3415F	DRAWBAR ASSY HC TSCHP CARB
10	4	40-1610	SHCS 1/4-20 X 1
11	6	40-16385	SHCS 5/16-18 X 3/4
12	1	51-1002	LT ANG CONT DUPLEX
13	1	51-1012A	BRNG DUPLEX LT 36MM SPCR
14	1	56-0075	SNAP RING N5000-131
15	1	57-2984	O-RING 2-158 VITON
16	1	57-2990	O-RING 2-348 BUNA
17	1	58-1627	1/8-27 PIPE PLUG

* DESIGNATES REMOVED PART

Spindle Assembly VF0, 10K, TSCHP



□ = DESIGNATES PARTS AVAILABLE FROM HAAS

VF-1,2,3,4,6,HS-1

TOOL RELEASE PISTON ASSY

Tool Release Piston Assembly

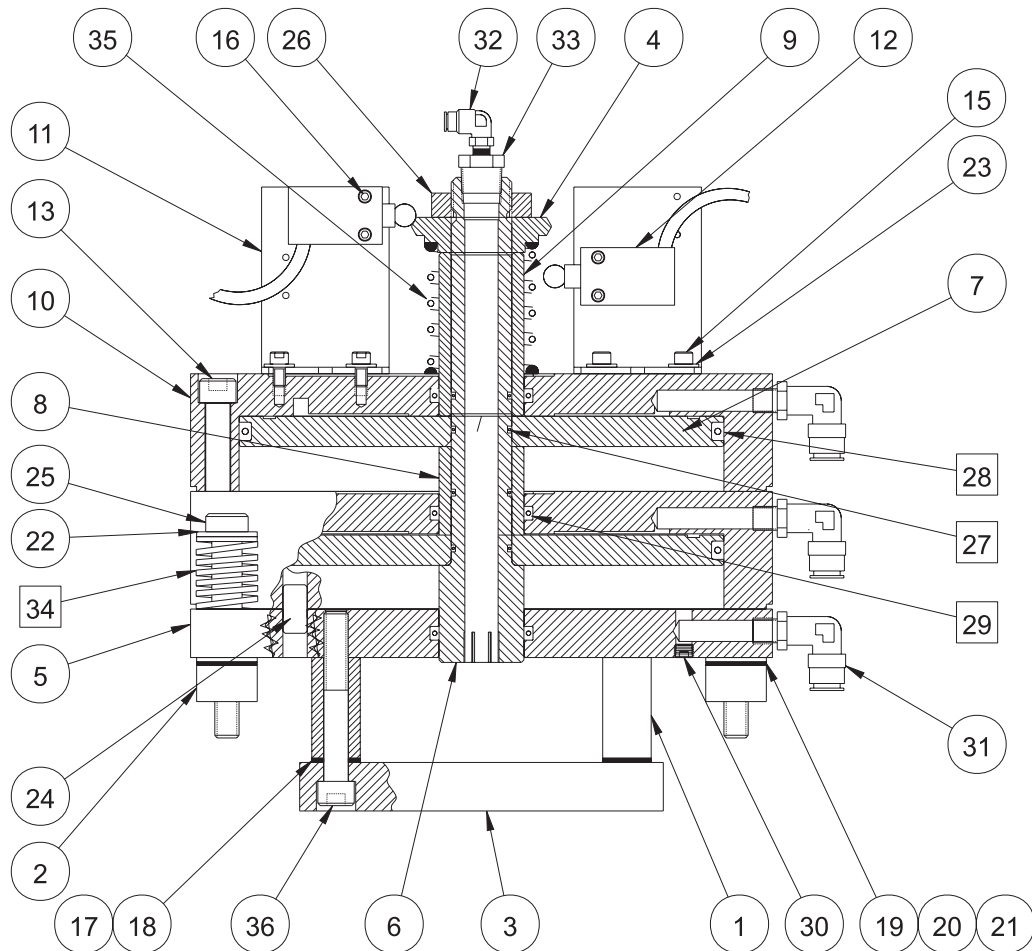


Tool Release Piston Assembly

30-3292 TOOL RELEASE PISTON ASSEMBLY

ITEM	PART NO.	DESCRIPTION	QTY
1.	20-7007A	CYLINDER HOUSING MACHINED	1
2.	20-7043A	PISTON TRP RECTANGLE	1
3.	20-7044C	SHAFT TRP	1
4.	22-7045A	SPRING RETAINER TRP 30DEG	1
5.	25-7050B	SWITCH MNT., TOOL RELEASE	2
6.	29-7397	LABEL, TOOL RELEASE PISTN	1
7.	32-2010	24 LIMIT SWITCH	2
8.	32-5620	TRP SOLENOID VALVE ASSY	1
9.	40-1632	SHCS 1/4-20 X 1/2	4
10.	40-1800	SHCS 8-32 X 3/4	4
11.	45-0040	WASHER BLK HARD 1/4 A325	4
12.	56-0040	SNAP RING N5100-62	2
13.	57-0040	O-RING 2-111 BUNA	2
14.	57-2155	O-RING 2-441 BUNA	1
15.	58-2165	FITTING CLOSE NIPPLE 1/4	1
16.	58-2265	AIR MUFFLER 3/8 FLAT	1
17.	58-3050	ELBOW 1/4 NYLON TUBING	1
18.	58-3618	1/4 STREET ELBOW, 90 DEG	2
19.	58-3670	1/4 NPT M - 1/8 F REDUCER	1
20.	58-3685	1/4NPT M-3/8 TUBE-SVL LBO	1
21.	58-3727A	1/4 NPT X 4 NIPPLE BRASS	1
22.	59-2760	COMP. SPRING/LARGE WIRE	1
23.	59-2832B	QUICK EXHAUST 1/4	1

☐ SERVICE DEPARTMENT



 DESIGNATES PARTS AVAILABLE FROM HAAS SERVICE DEPT.

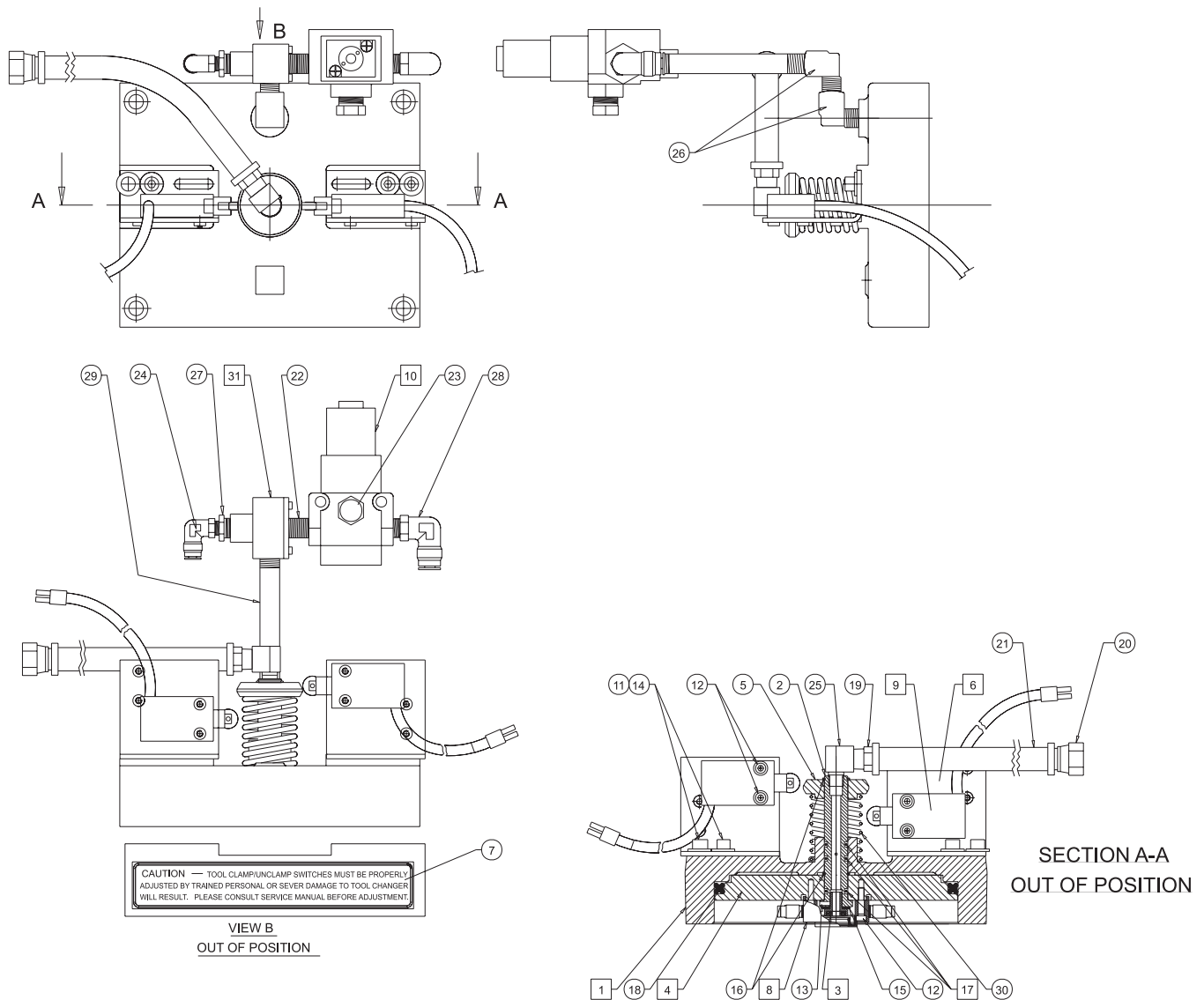
50 Taper Tool Release Piston



50 Taper Tool Release Piston

ITEM	QTY.	PART NO.	DESCRIPTION
1.	4 EA.	20-0013A	SPACER, FORK, SPINDLE
2.	4 EA.	20-0014A	SPACER, TRP
3.	1 EA.	20-0015	FORK, LIFT, SPINDLE
4.	1 EA.	20-0016B	SWITCH TRIP
5.	1 EA.	20-0017A	SUB PLATE, TRP 50T
6.	1 EA.	20-0018A	SHAFT, TRP 50T
7.	2 EA.	20-0019A	PISTON, TRP 50T
8.	1 EA.	20-0020A	SPACER, LOWER, TRP 50T
9.	1 EA.	20-0021B	SPACER, UPPER, TRP 50T
10.	2 EA.	20-0022A	HOUSING, TRP 50T
11.	2 EA.	25-0009	SWITCH MOUNTING BRACKET
12.	2 EA.	32-2013	TELEMECH. 44 IN. CABLE ASSY
13.	8 EA.	40-0006	SHCS, 1/2-13 X 5
*14.	4 EA.	40-0007	SHCS, 1/2-13 X 3 1/2
15.	4 EA.	40-1632	SHCS, 1/4-20 X 1/2"
16.	4 EA.	40-1800	SHCS, 8-32 X 3/4"
17.	4 EA.	45-0014	WASHER STL .505X1.00X.01
18.	28 EA.	45-0015	WASHER STL .505X1.00X.018
19.	8 EA.	45-0017	WASHER STL .625X1.239X.01
20.	12 EA.	45-0018	WASHER STL .625X1.239X.015
21.	4 EA.	45-0019	WASHER NYLON .625X1.25X.093
22.	8 EA.	45-0046	WASHER, SAE, 5/8
23.	4 EA.	45-16390	WASHER, 1/4 ID X 5/8 OD SAE
24.	6 EA.	48-1662	PIN, DOWEL 1/2 X 1"
25.	4 EA.	49-0003	SHLDR SCR HX HD 5/8X3-1/2
26.	1 EA.	52-0003	SHAFT CLAMP, 1 1/4-7
27.	4 EA.	57-0027	O-RING 2-121 BUNA
28.	2 EA.	57-0092	O-RING 2-448 BUNA
29.	3 EA.	57-0095	O-RING 2-327 VITON
30.	1 EA.	58-1627	1/8-27 PIPE PLUG
31.	3 EA.	58-1695	CONN, ELBOW 1/2 PUSH-1/4MP SVL
32.	1 EA.	58-3050	ELBOW 1/4 NYLON TUBING
33.	1 EA.	58-3631	REDUCER BUSH 1/2M-1/8F
34.	4 EA.	59-0016	SPRING, COMPRESSION
35.	1 EA.	59-0049	SPRING, COMPRESSION
36.	4 EA.	40-16627	SHCS, 1/2-13 X 3 1/2

* DESIGNATES REMOVED PART

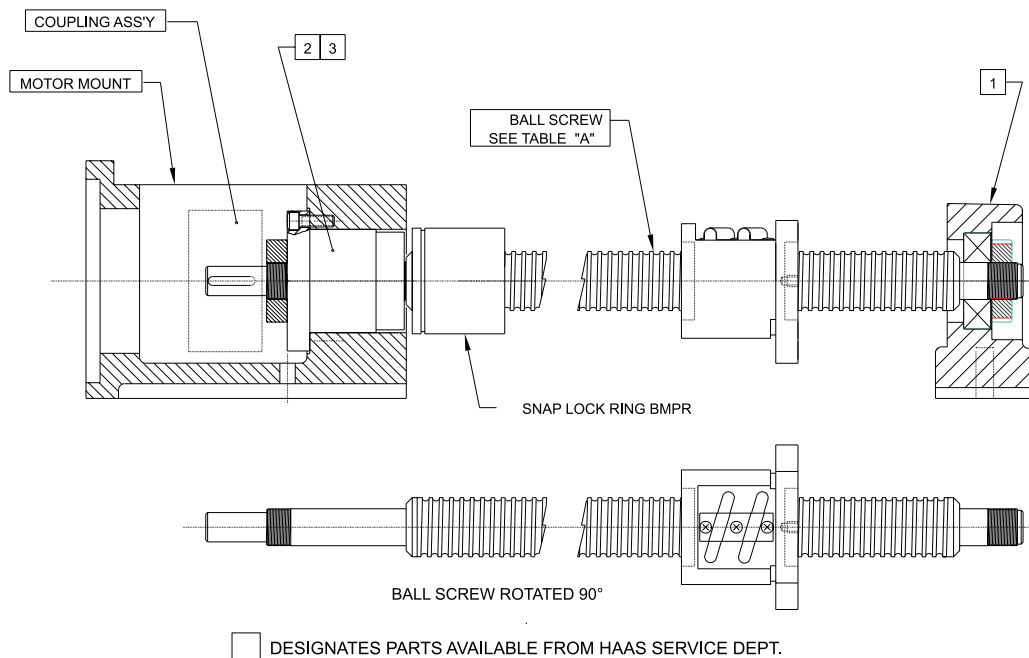


TSCHP Tool Release Piston Assembly



TSCHP Tool Release Piston Assembly

ITEM	PART NO.	DESCRIPTION	QTY
1	20-7007A	CYLINDER HOUSING MACHINED	1
2	20-7626	SHAFT TRP TSC	1
3	20-7627A	COOLANT TIP CARBIDE	1
4	20-7630A	TR PISTON RECTANGLE TSC	1
5	22-7045A	SPRING RETAINER TRP 30DEG	1
6	25-7050B	SWITCH MNT., TOOL RELEASE	2
7	29-7397	LABEL, TOOL RELEASE PISTN	1
8	30-3286A	SEAL HOUSING ASSY	1
9	32-2010	24 LIMIT SWITCH	2
10	32-5620	TRP SOLENOID VALVE ASSY	1
11	40-1632	SHCS 1/4-20 X 1/2	4
12	40-1800	SHCS 8-32 X 3/4	8
13	44-1614	SSS FLAT PT 6-32 X 1/4	3
14	45-0040	WASHER BLK HARD 1/4 A325	4
15	45-2000	WASHER, SHIM 1/4 ,010 THK	5
16	56-0040	SNAP RING N5100-62	2
17	57-0040	O-RING 2-111 BUNA	3
18	57-2156	QUAD-RING Q4-440 BUNA	1
19	58-0028	HOSE BARB 3/8PL-1/4MP	1
20	58-0032	HOSE BARB 3/8PL-3/8SAE-F	1
21	58-2046	HOSE 3/8ID PUSHLOC 300PSI	.75'
22	58-2165	FITTING CLOSE NIPPLE 1/4	1
23	58-2265	AIR MUFFLER 3/8 FLAT	1
24	58-3050	ELBOW 1/4 NYLON TUBING	1
25	58-3614	1/4 F-1/8 M STREET ELBOW	1
26	58-3618	1/4 STREET ELBOW, 90 DEG	2
27	58-3670	1/4 NPT M - 1/8 F REDUCER	1
28	58-3685	1/4NPT M-3/8 TUBE-SVL LBO	1
29	58-3727A	1/4 NPT X 4 NIPPLE BRASS	1
30	59-2760	COMP. SPRING/LARGE WIRE	1
31	59-2832B	QUICK EXHAUST 1/4	1

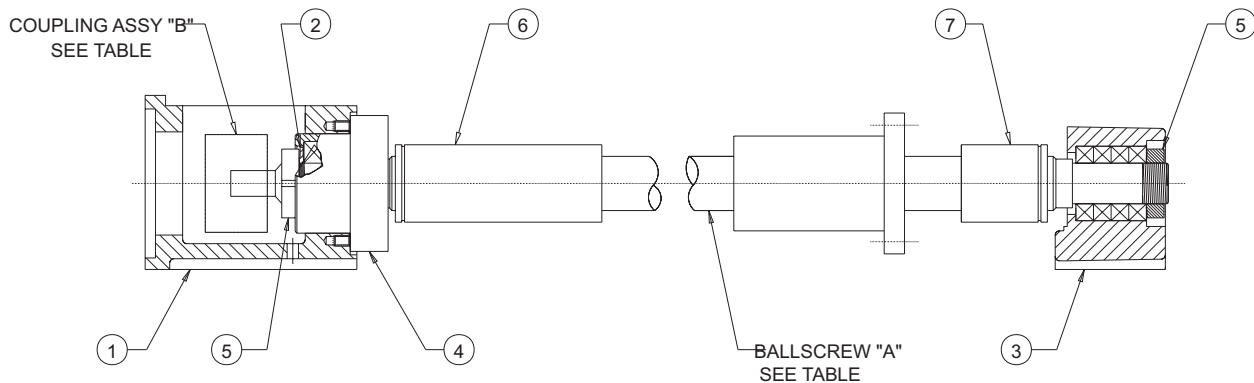




ITEM	QTY	PART NO.	TITLE
1.	1	30-0153	SUPPORT BEARING ASS'Y
2.	1	30-0154	MOTOR HOUSING, BEARING
3.	1	30-0156	BSREW FSTNR KIT VF-E/4,H


BALL SCREW ASS'Y "A"		BALL SCREW		SNAP LOCK RING BMPR	MOTOR MOUNT	COUPLING ASS'Y	APPLICATION
30-0157	BS ASS'Y 32mm (1.26) X 25.650	24-7146	BALLSCR 32mm (1.26) X 25.650	NONE	20-7010A	30-0211	SL40 (X)
30-0615	BS ASS'Y 32mm (1.26) X 33.268	24-9013	BALLSCR 32mm (1.26) X 33.268	20-0142 SNAP LOCK RING BMPR 6.00	20-7010A	30-0211	SL20 (Z)
30-0617	BS ASS'Y 32mm (1.26) X 48.228	24-9012	BALLSCR 32mm (1.26) X 48.228	20-0143 SNAP LOCK RING BMPR 7.00	20-7010A	30-0211	SL30 (Z)
30-0451	BS ASS'Y 32mm (1.26) X 25.650	24-0118	BALLSCR 32mm (1.26) X 25.650	20-0141 SNAP LOCK RING BMPR 4.00	20-7010A	30-0211	SL40 (X)
30-0192	BS ASS'Y 32mm (1.26) X 25.650	24-7146	BALLSCR 32mm (1.26) X 25.650	NONE	20-7010	30-1220P	VF-2, OEB (X)
30-0193	BS ASS'Y 32mm (1.26) X 35.650	24-7147	BALLSCR 32mm (1.26) X 35.650	NONE	20-7010	30-1220P	VF-3 (Y) (Z)
30-0194	BS ASS'Y 32mm (1.26) X 35.650	24-7147	BALLSCR 32mm (1.26) X 35.650	NONE	20-7010A	30-0211	SL20 (X), (Y)
30-0195	BS ASS'Y 32mm (1.26) X 48.228	24-9012	BALLSCR 32mm (1.26) X 48.228	NONE	20-7010A	30-0211	SL30 (X)
30-0196	BS ASS'Y 32mm (1.26) X 33.268	24-9013	BALLSCR 32mm (1.26) X 33.268	NONE	20-7010A	30-0211	SL20 (X)
30-0197	BS ASS'Y 32mm (1.26) X 58.485	24-9306	BALLSCR 32mm (1.26) X 58.485	NONE	20-7010A	30-0211	SL20 (X)
30-0618	BS ASS'Y 32mm (1.26) X 16.475	24-8765	BALLSCR 32mm (1.26) X 16.475	NONE	20-7010A	30-0211	SL20 (X)
30-0616	BS ASS'Y 32mm (1.26) X 13.525	24-9548	BALLSCR 32mm (1.26) X 13.525	NONE	20-7010A	30-0211	SL20 (X)

* DESIGNATES REMOVED PART

32mm Ball Screw Assembly



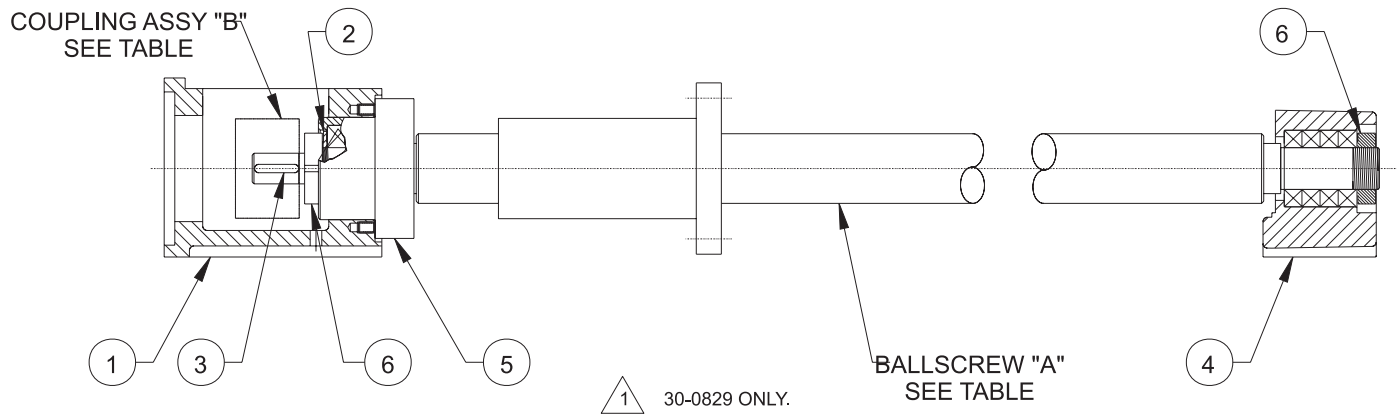
ITEM	QTY	PART NO.	TITLE
1	1	20-0151	MOTOR MOUNT 40/50mm BS MACH.
2	1	20-9213	SPACER RING 40mm BALL SCREW
3	1	30-0472	SUPPORT BEARING ASSY
4	1	30-1222	BALL SCREW SUPPORT BEARING ASSY
5	2	51-0008	BEARING LOCKNUT TCN-06-F
 6	1	20-0142	SNAP LOCK RING BMPR
 7	1	20-0146	SNAP LOCK RING BMPR

 FOR 30-0450 ONLY

BALL SCREW ASS'Y		BALL SCREW "A"		COUPLING ASSY "B"	APPLICATION
30-0450	BS ASS'Y 40mm (1.57) X 57.897	24-0003A	BALLSCR 40mm (1.57) X 57.897	30-1219	SL40 (Z)
30-0198	BS ASS'Y 40mm (1.57) X 57.897	24-0003A	BALLSCR 40mm (1.57) X 57.897	30-1215	VF-5/50 (X)
30-0202	BS ASS'Y 40mm (1.57) X 32.696	24-0006A	BALLSCR 40mm (1.57) X 32.696	30-1215	VF-5/50 (Y) (Z)
30-0470	BS ASS'Y 40mm (1.57) X 71.935	24-9961C	BALLSCR 40mm (1.57) X 71.935	30-1219	VF-6,8 (X)
30-0473	BS ASS'Y 40mm (1.57) X 90.859	24-9970C	BALLSCR 40mm (1.57) X 90.859	30-1219	VF-7,9 (X)
30-0474	BS ASS'Y 40mm (1.57) X 47.711	24-9960C	BALLSCR 40mm (1.57) X 47.711	30-1219	VF-6,7,8,10,11 VR-11 (Y) (Z)
30-0832	BS ASS'Y 40mm (1.57) X 90.859	24-9970C	BALLSCR 40mm (1.57) X 90.859	30-1215	VB-1 (Y)
30-0835	BS ASS'Y 40mm (1.57) X 47.711	24-9960C	BALLSCR 40mm (1.57) X 47.711	30-1215	VB-1 (Z)

* DESIGNATES REMOVED PART

40mm Ballscrew Assembly

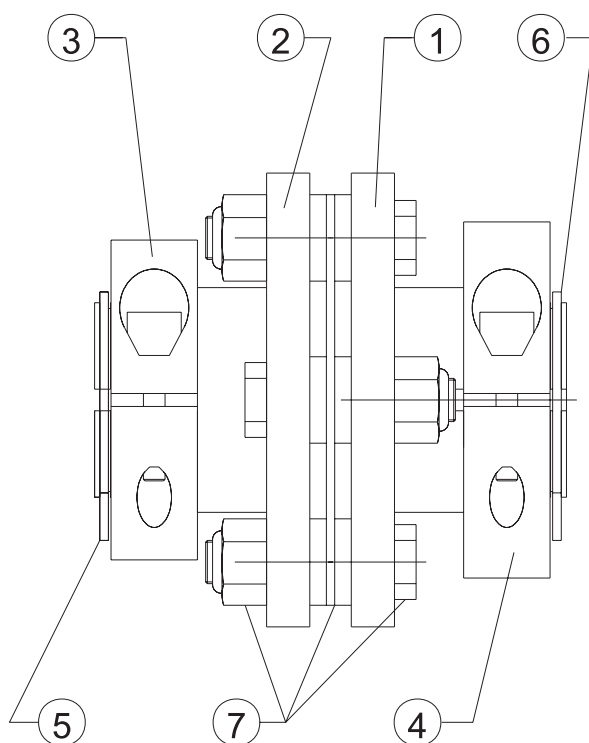


BALL SCREW ASS'Y		BALL SCREW "A"		COUPLING ASSY "B"	APPLICATION
30-0516	BS ASS'Y 50mm (1.57) X 129.42	24-0002C	BALLSCR 50mm (1.57) X 129.42	30-1225A	VF-10/11 (Y) (Z)
30-0829	BS ASS'Y 50mm (1.57) X 217.09	24-9962	BALLSCR 50mm (1.57) X 217.09	30-1225A	VB-1 (X)

* = REMOVED PART

ITEM	QTY	PART No.	DESCRIPTION
1	1	20-0151	MTR MOUNT 40/50MM BS MACH
2	1	20-9213	SPACER RING 40MM BALLSCRW
1 3	1	22-5170A	KEY X-AXIS BALLSCREW
4	1	30-0472	SUPPORT BEARING ASSY
5	1	30-1222	BALL SCREW SUPP BRNG ASSY
6	2	51-0008	BEARING LOCK NUT TCN-06-F

50mm Ball Screw Assembly

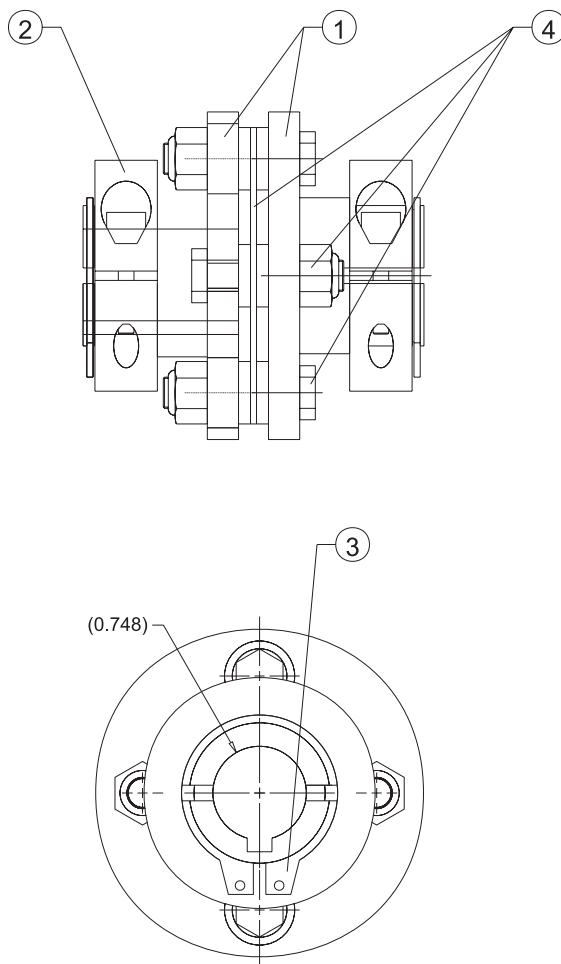


ITEM QTY PART No. TITLE

1	1	20-0105A	COUPLER, SERVO BRUSHLESS
2	1	20-7615	COUPLER, SERVO DRIVE BRUSHLESS
3	1	51-2014	BEARING LOCKNUT, CL18F
4	1	51-2019	CLAMP COLLAR 1 1/4 BORE
5	1	56-0065	SNAP RING, (5100-112)
6	1	56-0076	SNAP RING 5100-125
7	1	59-2060	FLEXPAC FOR AJ05

WHERE USED	
PART NO.	DESCRIPTION
50T-10	50 TAPER OPTION VF-10
50T-6	50 TAPER OPTION VF-6
50T-7	50 TAPER OPTION VF-7
50T-8/9/11	50 TAPER OPTION VF-8/9/11
93-8015	50 T AXS MTR UPGRD W/4AXIS
30-0832	BSCREW ASSY VB-Y-AXIS
30-0202	BSCREW ASSY VF-5/50
30-0198	BSCREW ASSY VF-5/50

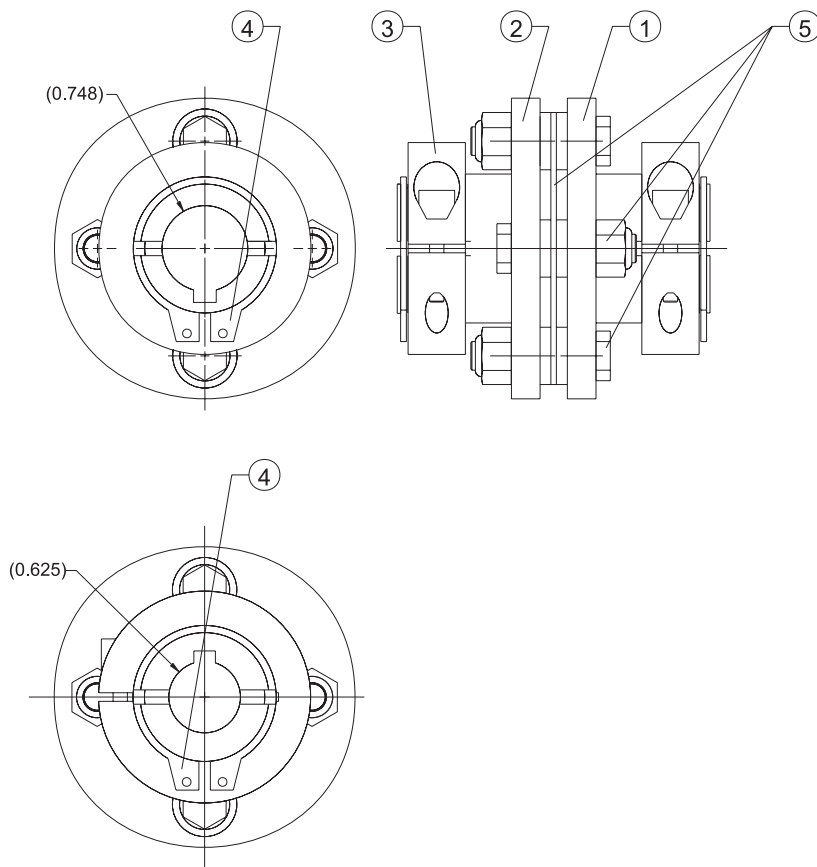
30-1215 Coupling Assembly



ITEM	QTY	PART No.	TITLE
1	2	20-7615	COUPLER, SERVO DRIVE BRUSHLESS
2	2	51-2014	BEARING LOCKNUT, CL18F
3	2	56-0065	SNAP RING, (5100-112)
4	1	59-2060	FLEXPAC FOR AJ05

WHERE USED		APPLICATION
30-0470	BS ASSY 40mm(1.57) x 71.94	VF-6B, 8B (X)
30-0450	BS ASSY 40mm(1.57) x 57.90	SL40 (Z)
30-0473	BS ASSY 40mm(1.57) x 90.86	VF-7B, 9B (X)
30-0474	BS ASSY 40mm(1.57) x 47.71	VF-6B, 7B, 8B, 9B, 10B, 11B & VR-11B (Y) (Z)
30-0198	BS ASSY 40mm(1.57) x 32.70	VF-5/50 (Y) (Z)
30-0202	BS ASSY 40mm(1.57) x 57.90	VF-5/50 (X)

30-1219 Coupling Assembly

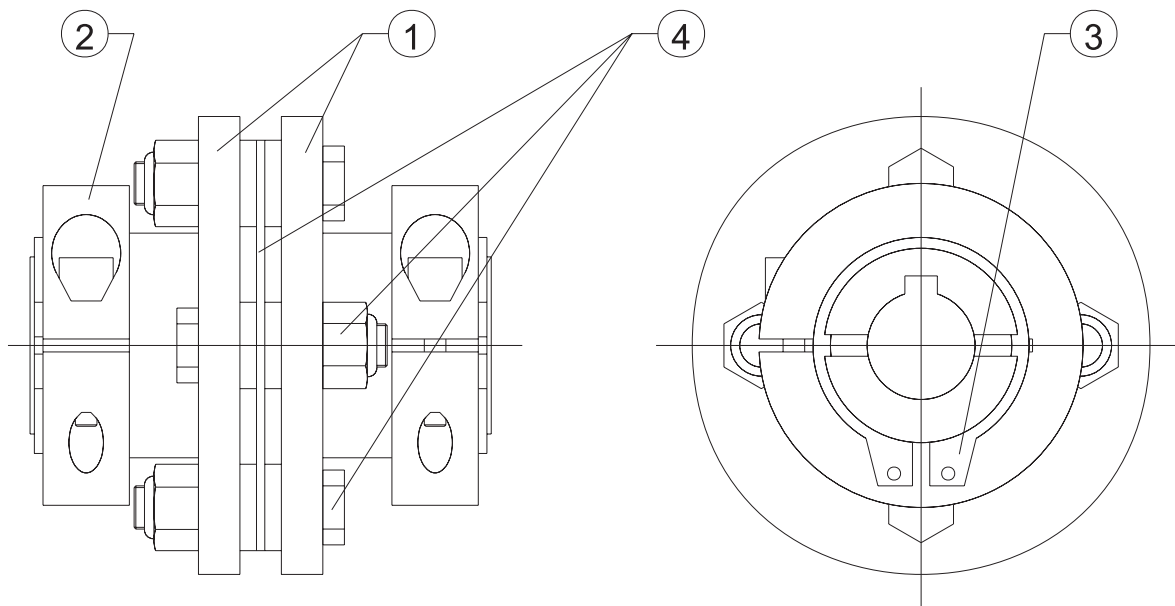


ITEM QTY PART No. TITLE

1	1	20-7403	COUPLING, SERVO DRIVE
2	1	20-7615	COUPLER, BRUSHLESS
3	2	51-2014	BEARING LOCKNUT, CL18F
4	2	56-0065	SNAP RING, (5100-112)
5	1	59-2060	FLEXPAC FOR AJ05

WHERE USED		APPLICATION
30-0615	BS ASSY 32mm(1.26) x 33.27	SL20 (Z)
30-0617	BS ASSY 32mm(1.26) x 48.23	SL30 (Z)
30-0451	BS ASSY 32mm(1.26) x 25.65	SL40 (X)
30-0194	BS ASSY 32mm(1.26) x 35.65	VF-0EB, 2B (X)
30-0195	BS ASSY 32mm(1.26) x 48.23	VF-3B, (X) (Z)
30-0196	BS ASSY 32mm(1.26) X 33.27	VF-3B, 4B, SL20 (Y) (Z), HS-1/RP
30-0197	BS ASSY 32mm(1.26) x 58.47	VF-4B (X)
30-0616	BS ASSY 32mm(1.26) x 13.53	SL20 (X)
30-0618	BS ASSY 32mm(1.26) x 16.78	SL30 (X)
30-0157	BS ASSY 32mm(1.26) x 25.65	SL40, VF-1,2,O

30-1220A Coupling Assembly

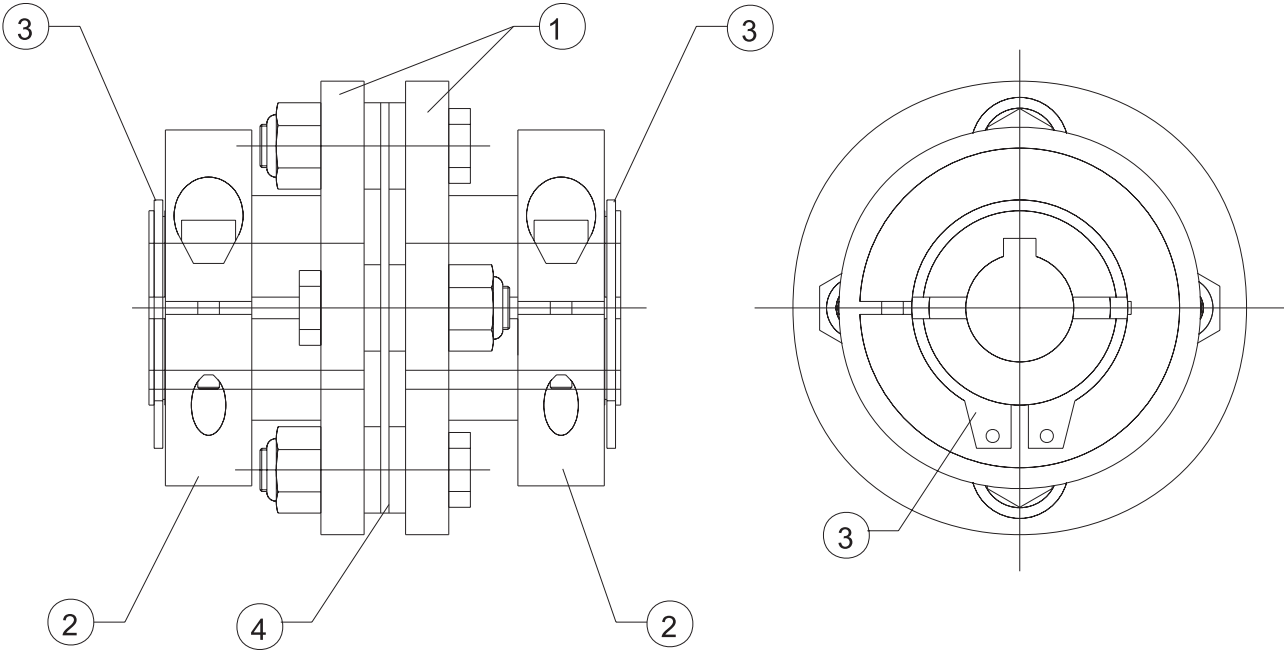


ITEM QTY PART No. TITLE

1	1	20-7403	COUPLING, SERVO DRIVE
2	2	51-2014	BEARING LOCKNUT, CL18F
3	2	56-0065	SNAP RING, (5100-112)
4	1	59-2060	FLEXPAC FOR AJ05

WHERE USED		APPLICATION
30-0192	BS ASSY 32mm(1.26) x 25.65	VF-E, EXT (X) (Y) (Z)
30-0193	BS ASSY 32mm(1.26) x 35.65	VF-EXT (X)

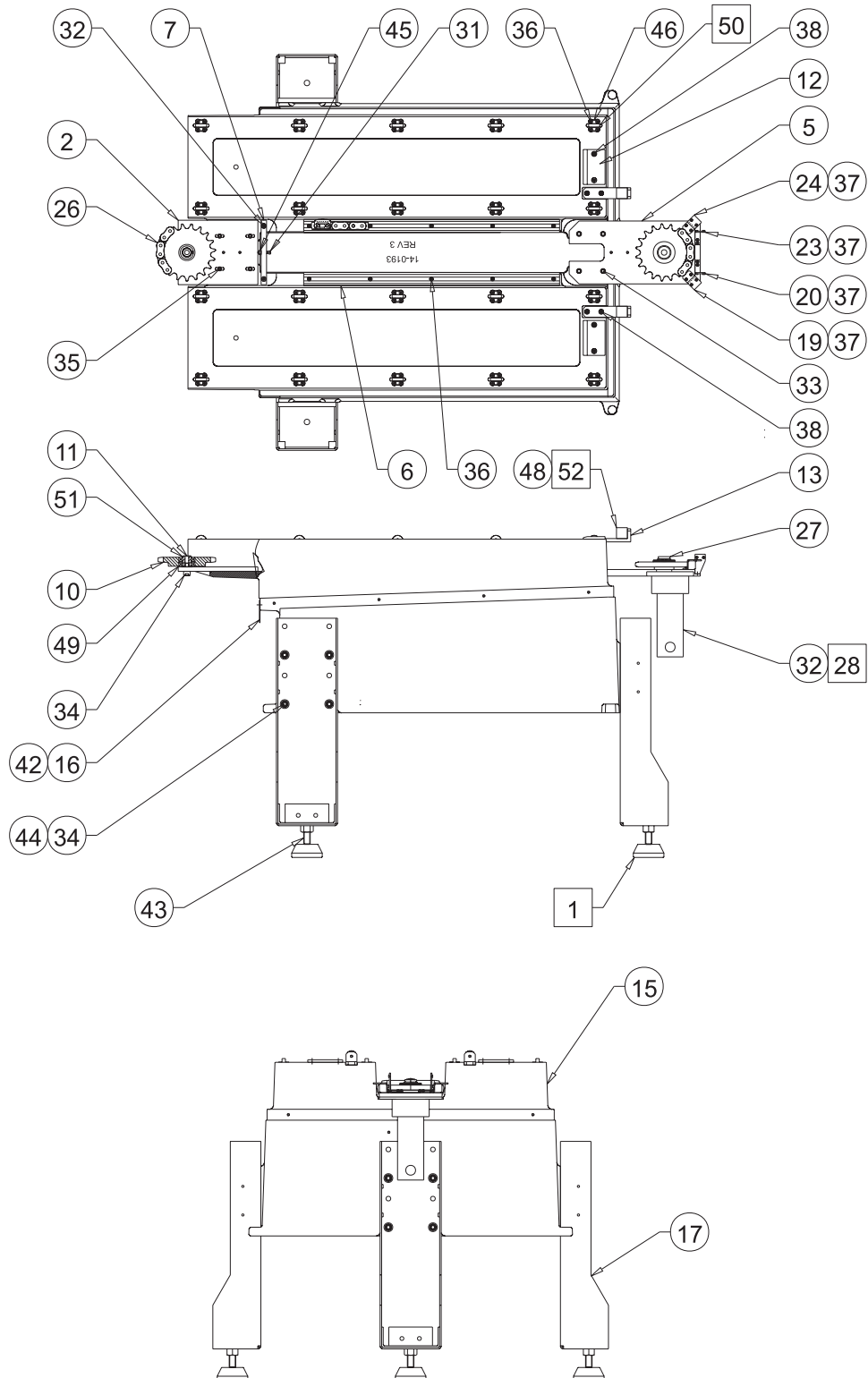
30-1220P Coupling Assembly



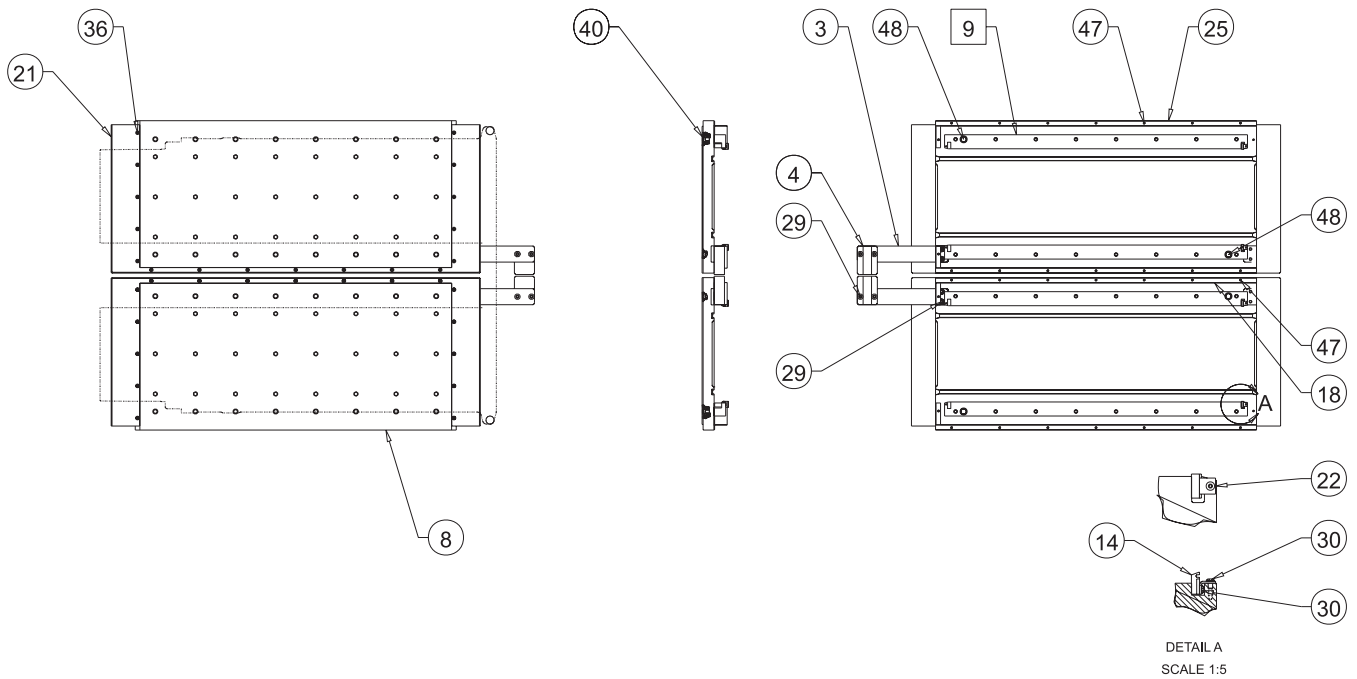
WHERE USED		APPLICATION
30-0516	BS ASSY 50mm(1.97) x 129.42	VF-10B, 11B, VR-11B (X)
30-0829	BS ASSY 50mm(1.97) x 217.09	VB-1 (X)

ITEM	QTY	PART No.	TITLE
1	2	20-0105A	COUPLING SERVO/BRUSHLESS
2	2	51-2019	CLAMP COLLAR 1 1/4 BORE
3	2	56-0076	SNAP RING 5100-125
4	1	59-2060	FLEXPAC FOR AJ05

30-1225 Coupling Assembly



APC Assembly

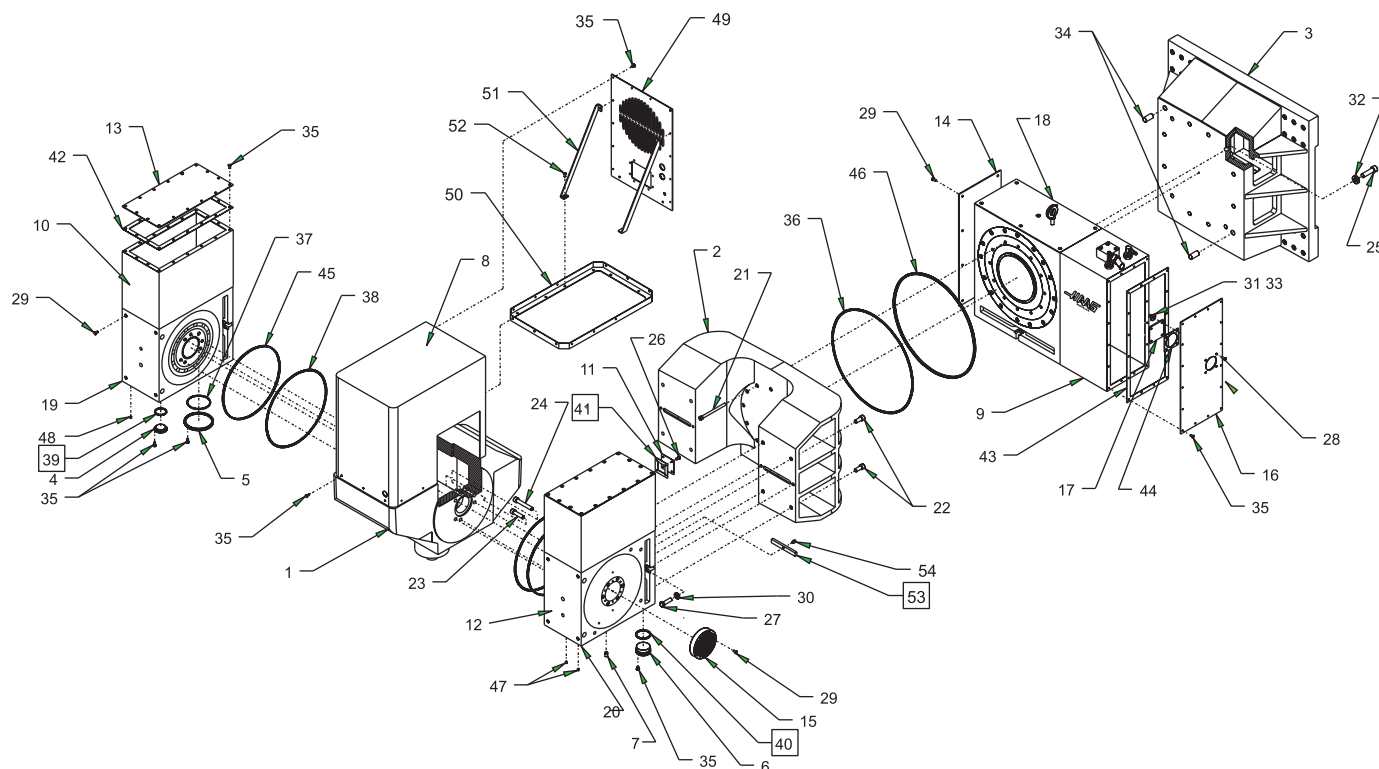


APC Assembly



APC Assembly

ITEM	QTY.	PART NO.	TITLE
1.	3	14-7068	CASTING, LEVEL PAD
2.	1	20-0046	SUPPORT, IDLER SPROCKET
3.	2	20-0048	DRIVE LEG, APC
4.	2	20-0049	DETENT, APC
5.	1	20-0050	SUPPORT, MOTOR, APC
6.	2	20-0051	GUIDE, CHAIN, APC
7.	1	20-0052	TENSIONER BLOCK
8.	2	20-0053	PALLET
9.	4	20-0054	CLAMPING RAIL
10.	1	20-0057	IDLER SPROCKET
11.	1	20-0060	JOURNAL, IDLER SPROCKET
12.	2	20-0065	FRICTION BLOCK
13.	2	20-0066	PALLET STOP, APC
14.	4	20-0071	WIPER, APC
15.	1	20-0193	BASE, MACHINED
16.	1	25-0066	SHIELD, SPLASH, LOW PROFILE
17.	3	25-0072	LEG, APC
18.	2	25-0077	PALLET, SKIRT, REAR
19.	1	25-0082	SWITCH BRACKET, CHAIN, LOW
20.	1	25-0085	SWITCH BRACKET, ARM #1
21.	2	25-0095	PALLET DRIP PAN
22.	4	25-0100	BRACKET, WIPER
23.	1	25-0101	SWITCH BRACKET ARM #2
24.	1	25-0102	SWITCH BRACKET, CHAIN, HIGH
25.	2	25-0105	PALLET SKIRT, FRONT
26.	1	30-0054	CHAIN ASSEMBLY, APC
27.	1	30-0055	SLIP CLUTCH ASSEMBLY
28.	1	32-1800	SHUTTLE MOTOR, 507-01-110AH
29.	8	40-0017	FHCS, 5/16-18 X 3/4"
30.	8	40-16081	BHCS, 6-32 X 5/16"
31.	1	40-1614	SHCS, 1/4-20 X 1 1/4
32.	2	40-1617	FHCS, 1/4-20 X 1"
33.	4	40-1636	SHCS, 3/8-16 X 1 1/4
34.	13	40-1654	SHCS, 1/2-13 X 1"
35.	4	40-1667	SHCS, 5/16-18 X 1 1/4
36.	124	40-1703	FHCS, 10-32 X 1/2
37.	8	40-1850	SHCS, 10-32 X 3/8"
38.	8	40-1920	FHCS, 1/4-20 X 5/8
39.	2	40-1950	SHCS, 10-32 X 3/4
40.	32	40-1961	SHCS, 3/8-16 X 2"
41.	4	40-1970	FHCS, 1/4-28 X 1"
42.	3	40-1981	FBHCS, 1/4-20 X 1/2
43.	3	44-1700	SSS, CUP PT. 3/4-10 X 4:"
44.	12	45-1666	WASHER, FLAT 1/2 I.D.
45.	1	46-1625	NUT, HEX, BLACK OX, 1/4-20
46.	20	48-0012	DOWEL PIN, 12mm X 30 mm LG.
47.	32	49-16201	BHCS, 10-32 X .38
48.	4	51-0300	BUSHING, DRILL .6260 I.D.
49.	2	51-2836	BEARING, RADIAL, #60052RS
50.	20	51-4000	BEARING, RADIAL 12 X 32 X 10MM
51.	1	56-0085	RETAINING RING 5100-100
52.	2	59-1057	BUMPER, PALLET



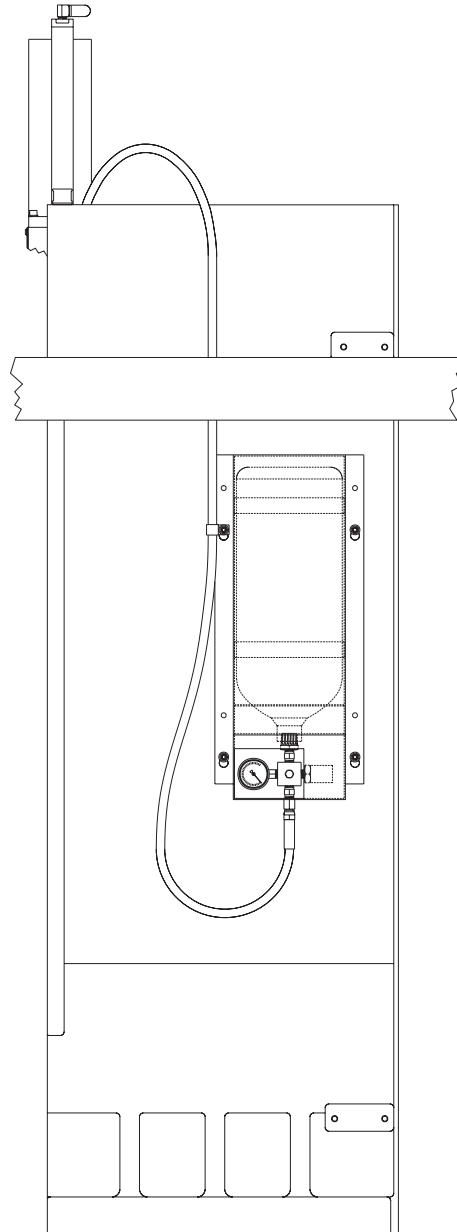
TORQUE SETTINGS	
ITEM	VALUE
21	45 FT LBS
22	88 FT LBS
23	100 FT LBS
24	100 FT LBS
25	160 FT LBS

VR-Series Head Assembly



VR-Series Head Assembly

ITEM	QTY	DWG #	DESCRIPTION
1	1	20-4360	5AX SPINDLE HEAD (MACHINED)
2	1	20-4361	5AX YOKE, (MACHINED)
3	1	20-4367	5AX HEAD SPACER (MACHINED)
4	1	20-4381	SIGHT GLASS PLUG
5	1	20-4382	WORM HOUSING COVER, 5AX
6	1	20-4388	WORM PLUG 310 PULLEY SIDE
7	1	22-4040	MICRO SWITCH, PLUG
8	1	25-4363	5AX HEAD COVER
9	1	25-4366	MOTOR ENCLOSURE 450
10	2	25-4371	MOTOR ENCLOSURE 310/5AX
11	1	25-4372	BLOCK OFF PLATE 310/5AX
12	2	25-4373	TOP COVER 310/5AX
13	2	25-4375	ENCLOSURE COVER 310/5AX
14	1	25-4377	SIDE COVER 450/5AX
15	2	25-4380	PORT CHIP GUARD 5AX
16	1	25-4386	ENCLOSURE COVER 450/5AX
17	1	28-4278	SIGHT GLASS, PRESS GAGE
18	1	30-1070	HRT450 ASSY W/ 5AX MODS
19	1	30-1071	HRT310 DRIVE ASSY 5AX
20	1	30-1072	HRT310 DRIVEN ASSY 5AX
21	12	40-164391	SHCS, 3/8-16 X 5 1/4
22	8	40-16575	SHCS, 1/2-13 X 1 1/4
23	8	40-1661	SHCS, 1/2-13 X 2
24	4	40-16626	SHCS, 1/2-13 X 3 1/4.
25	12	40-16643	SHCS, 5/8-11 X 2 1/4
26	4	40-1669	BHCS, 8-32 X 3/8
27	4	40-1830	HHB, 1/2-13 X 1 3/4
28	4	40-1976	BHCS, 1/4-20 X 3/4
29	22	40-1980	BHCS, 1/4-20 X 1/2
30	4	45-1740	WASHER, BLACK HARD 1/2
31	4	45-1850	WASHER, FENDER 1/4 IDX1 OD
32	12	45-2011	HARD WASHER 5/8
33	4	46-1625	NUT HEX BLK OX 1/4-20
34	2	48-1757	DOWEL PIN 3/4 X 1 1/2.
35	85	49-1750	BHCS, 10-32 X 3/8
36	1	57-0093	O RING, 2-385 BUNA
37	1	57-2250	O-RING, 2-156 VITON
38	2	57-2252	O RING, 2-381 VITON
39	1	57-2831	O-RING, 2-130 BUVA
40	1	57-4120	O-RING, 2-226 VITON
41	1	57-4133	J-BOX GASKET
42	2	57-4223	GASKET MOTOR ENCLOSURE
43	1	57-4261	ENCLOSURE COVER GASKET 450
44	1	57-4279	GASKET, SIGHT GLASS
45	2	57-4384	HRT310 TEFLON SEAL
46	1	57-4385	HRT450 TEFLON SEAL
47	3	58-1627	1/8-27 PIPE PLUG
48	1	58-3105	1/4 NPT PIPE PLUG
49	1	25-4362	5AX HEAD COVER, BACK PLATE
50	1	25-4364	HEAD COVER MOUNTING ANGLE
51	2	25-4383	HEAD COVER BRACE, 5AX
52	10	40-1975	BHCS 1/4-20 X 5/19
53	2	20-4230	KEY, BODY
54	20	40-1630	SHCS 1/4-20 X 5/16



(HYDRAULIC PRESSURE SENSOR CABLE NOT SHOWN)

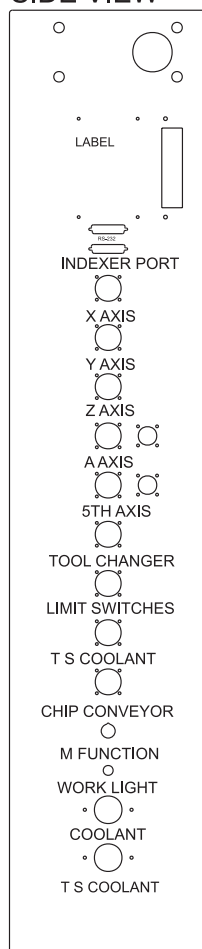
VF-Series Hydraulic Couterbalance System



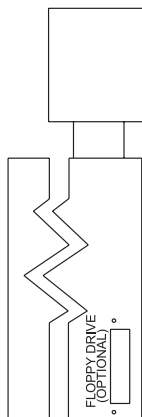
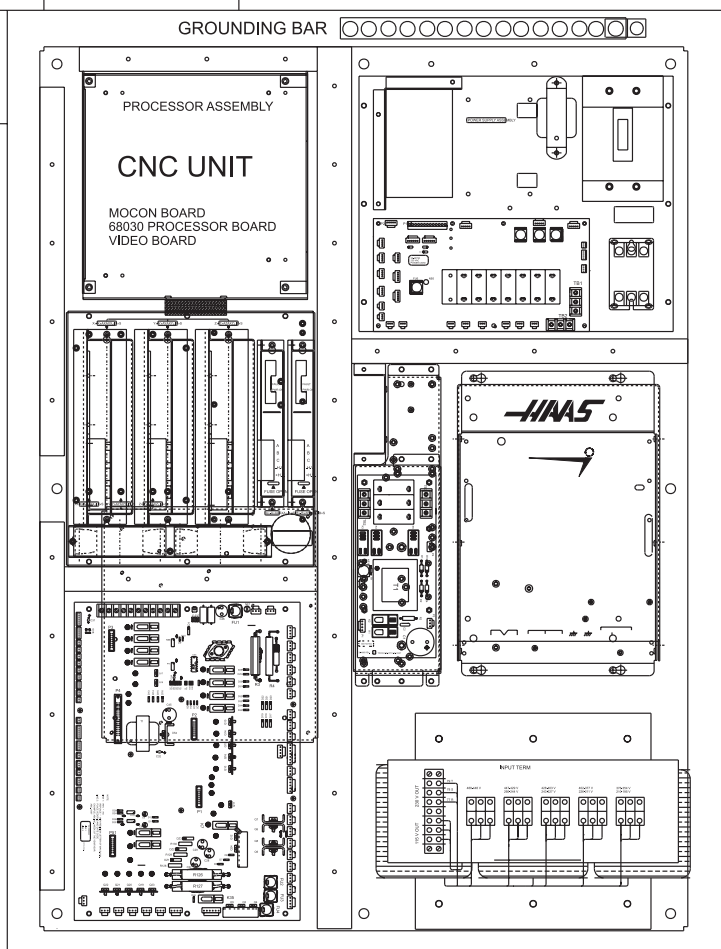
ELECTRICAL WIRING DIAGRAMS



SIDE VIEW



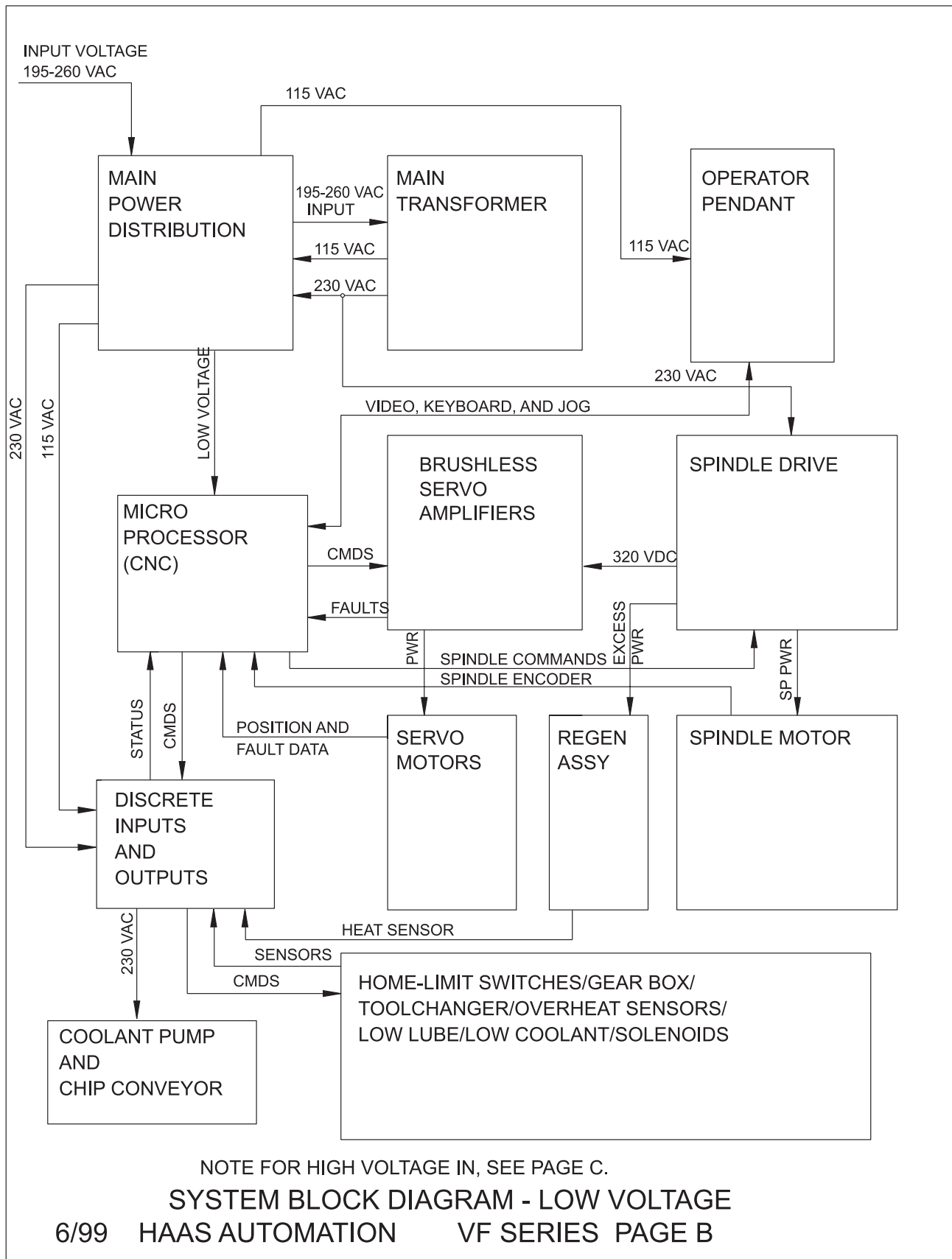
OPERATOR PENDANT

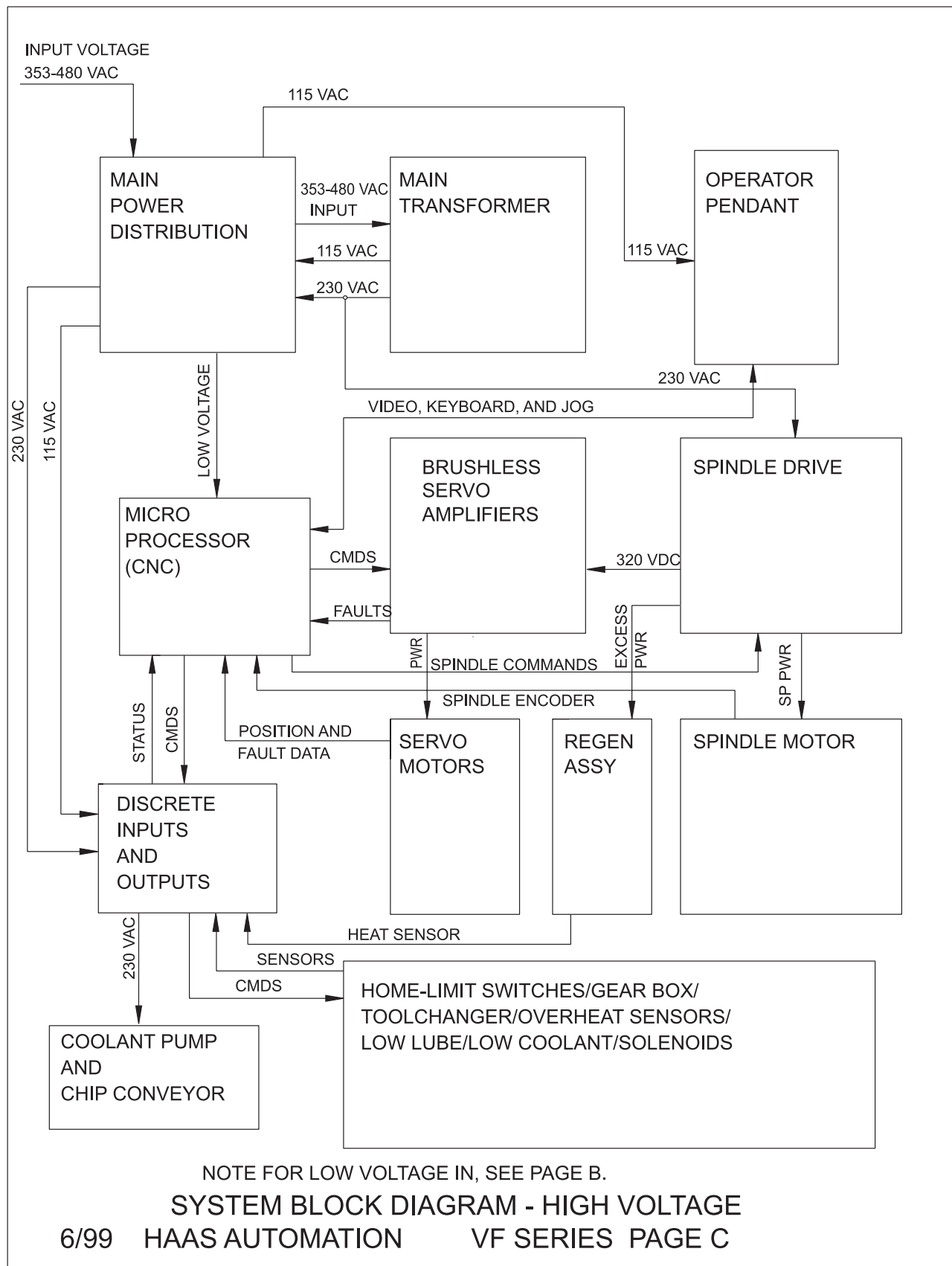
SPINDLE
REGEN RESISTORSCONTROL CABINET
REAR OF MACHINE

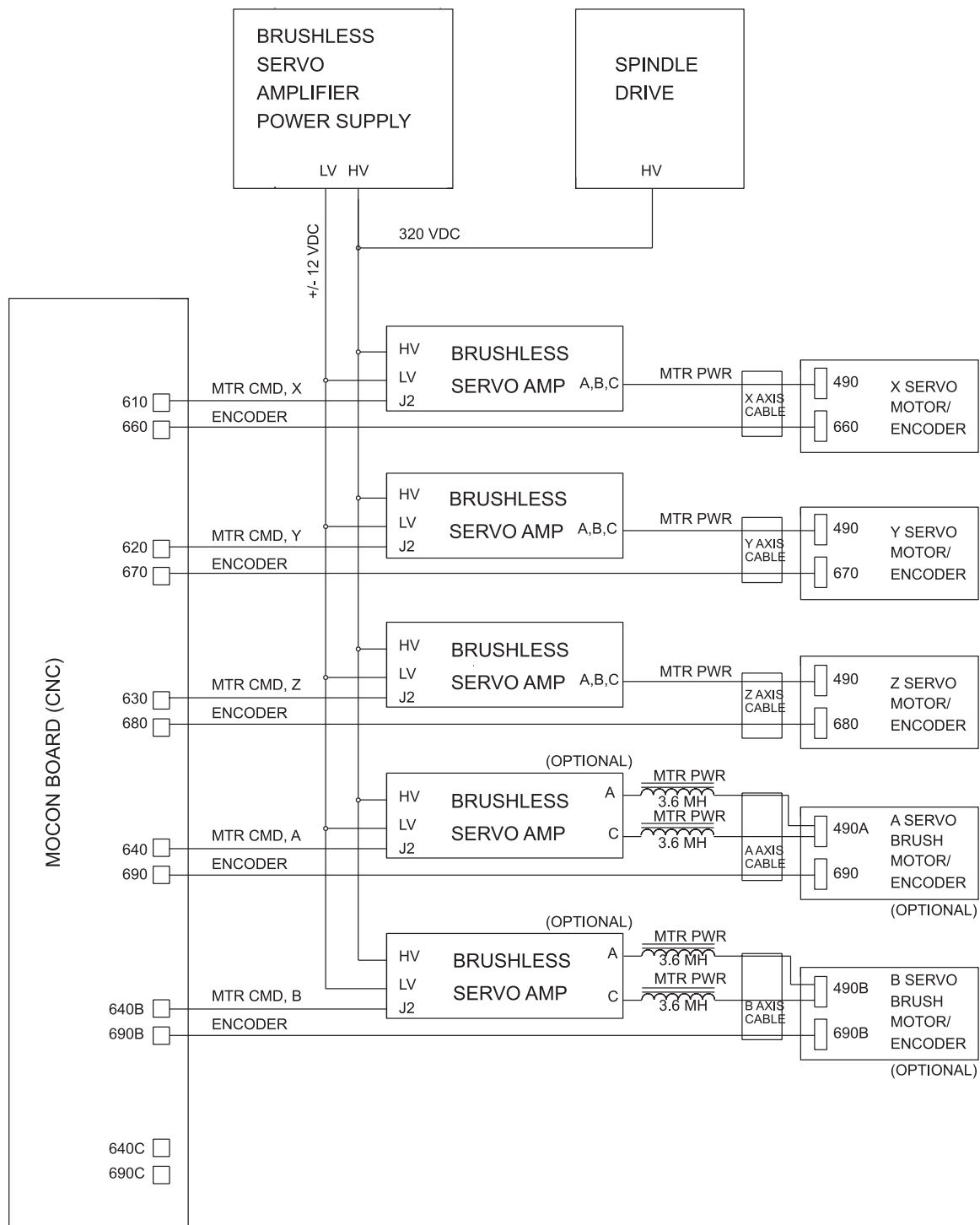
ITEM DESCRIPTION	PAGE #	ITEM DESCRIPTION	PAGE #
CNC LAYOUT	A	RELAY COIL DRIVERS, IOPCB	8-11
SYSTEM BLOCK DIAGRAM	B,C	SPINDLE DRIVE UNIT	12
CABLE INTERCONNECT DIAGRAM	D,E	AXIS MOTOR & ENCODER	14,15
SERVO SYSTEM	1	CABINET CONNECTORS	16
MAIN TRANSFORMER	2,3	TOOL CHANGE MOTORS	17
CNC UNIT	4	CHIP CONVEYOR/SPIGOT MOTOR	18
115VAC CIRCUITS	5	OPERATOR PENDANT	19
INPUTS IOPCB	6,7	ELECTRICAL SYMBOLS	21

CONTROL LAYOUT DIAGRAM

6/99 HAAS AUTOMATION VF SERIES PAGE A





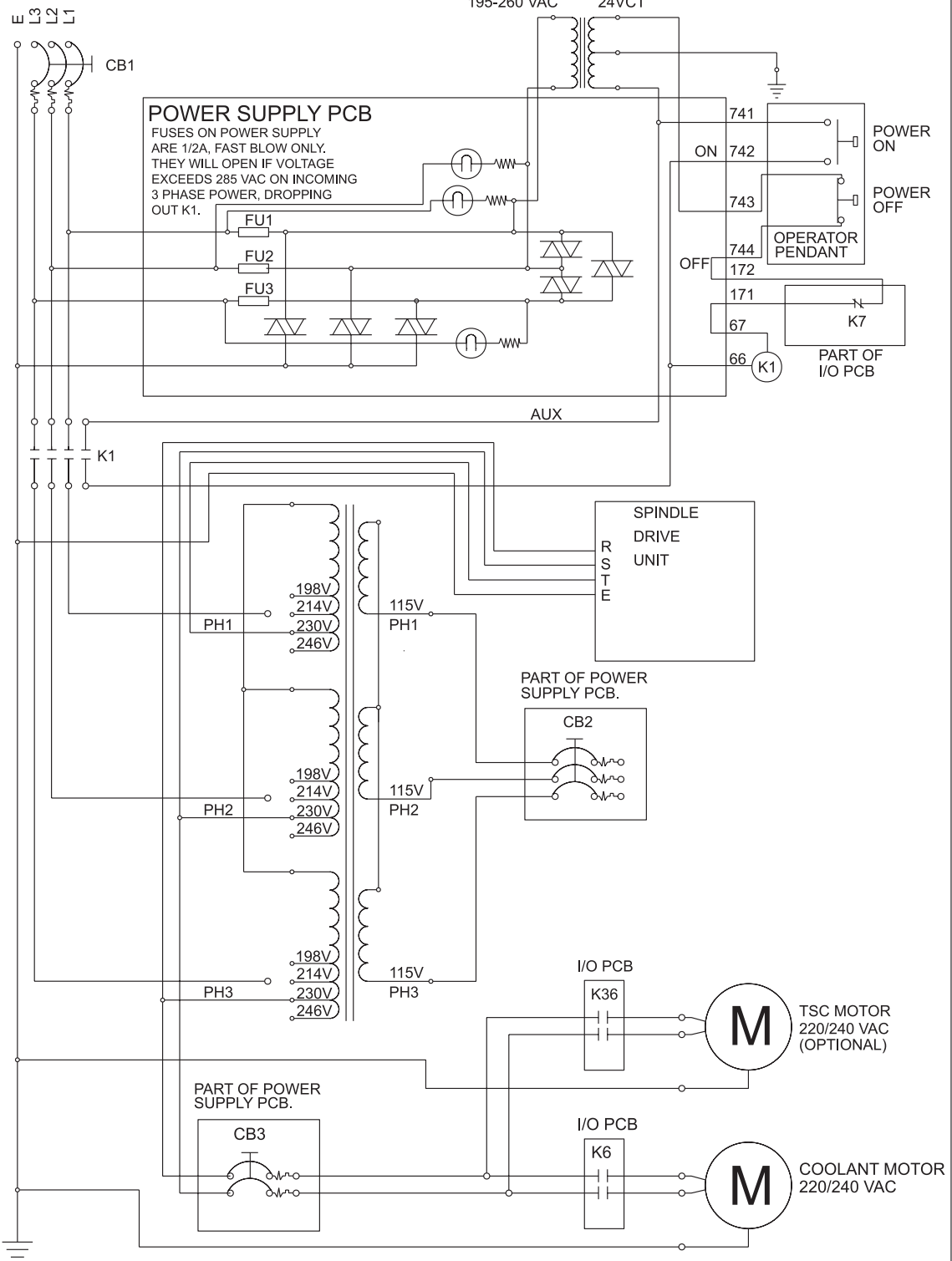


SERVO SYSTEM
6/99 HAAS AUTOMATION

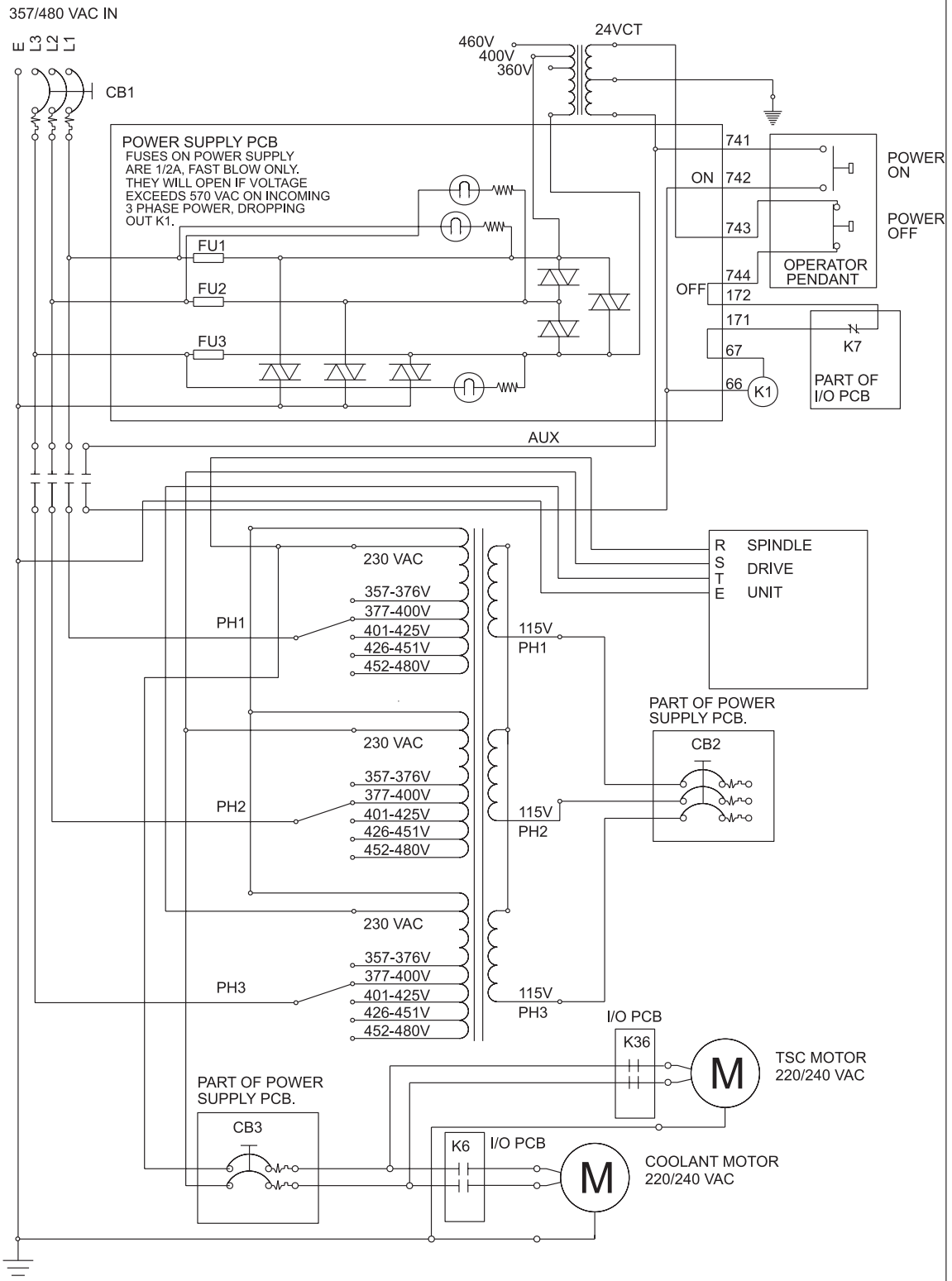
VF SERIES PAGE 1



195/260 VAC IN

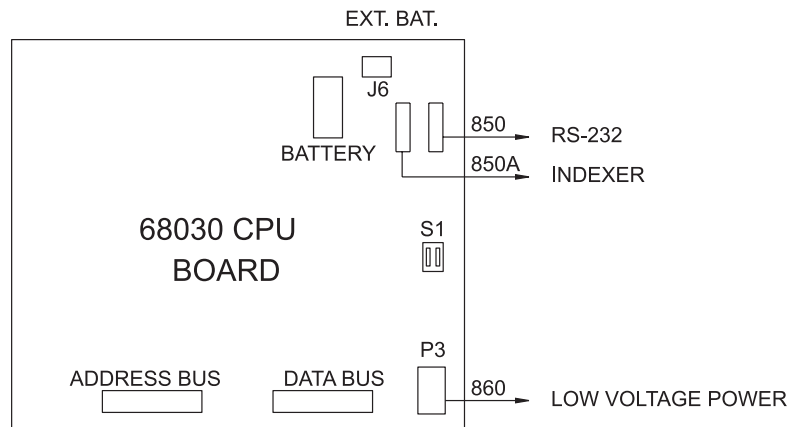
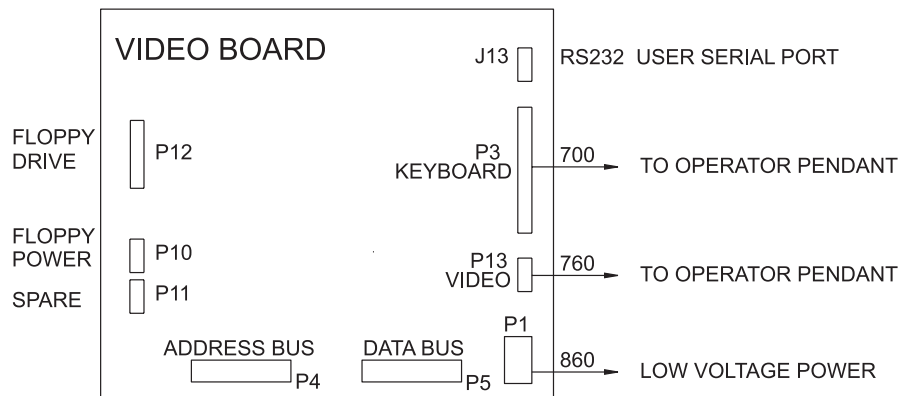
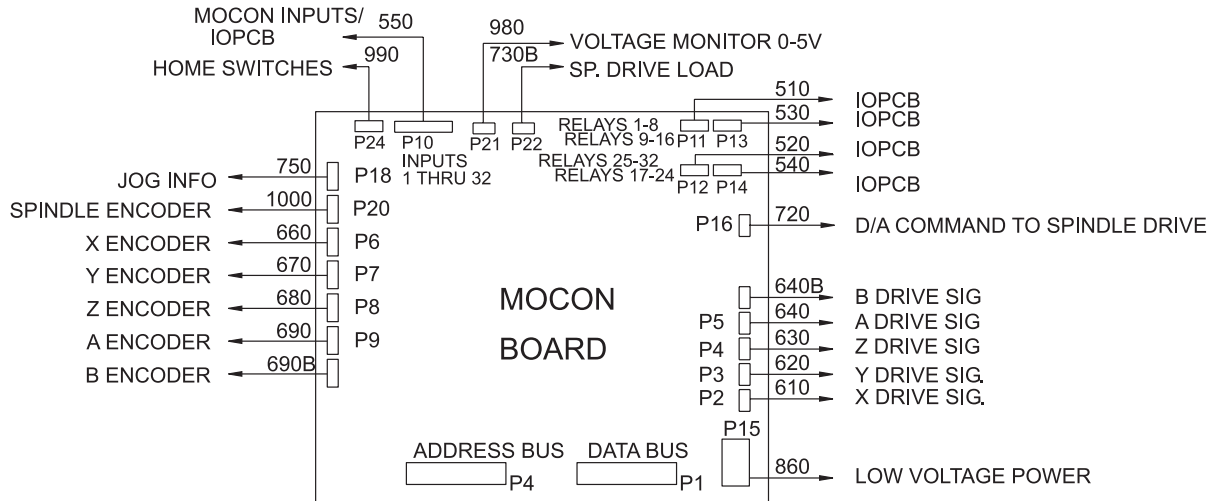


MAIN TRANSFORMER - LOW VOLTAGE
6/99 HAAS AUTOMATION VF SERIES PAGE 2



MAIN TRANSFORMER - HIGH VOLTAGE

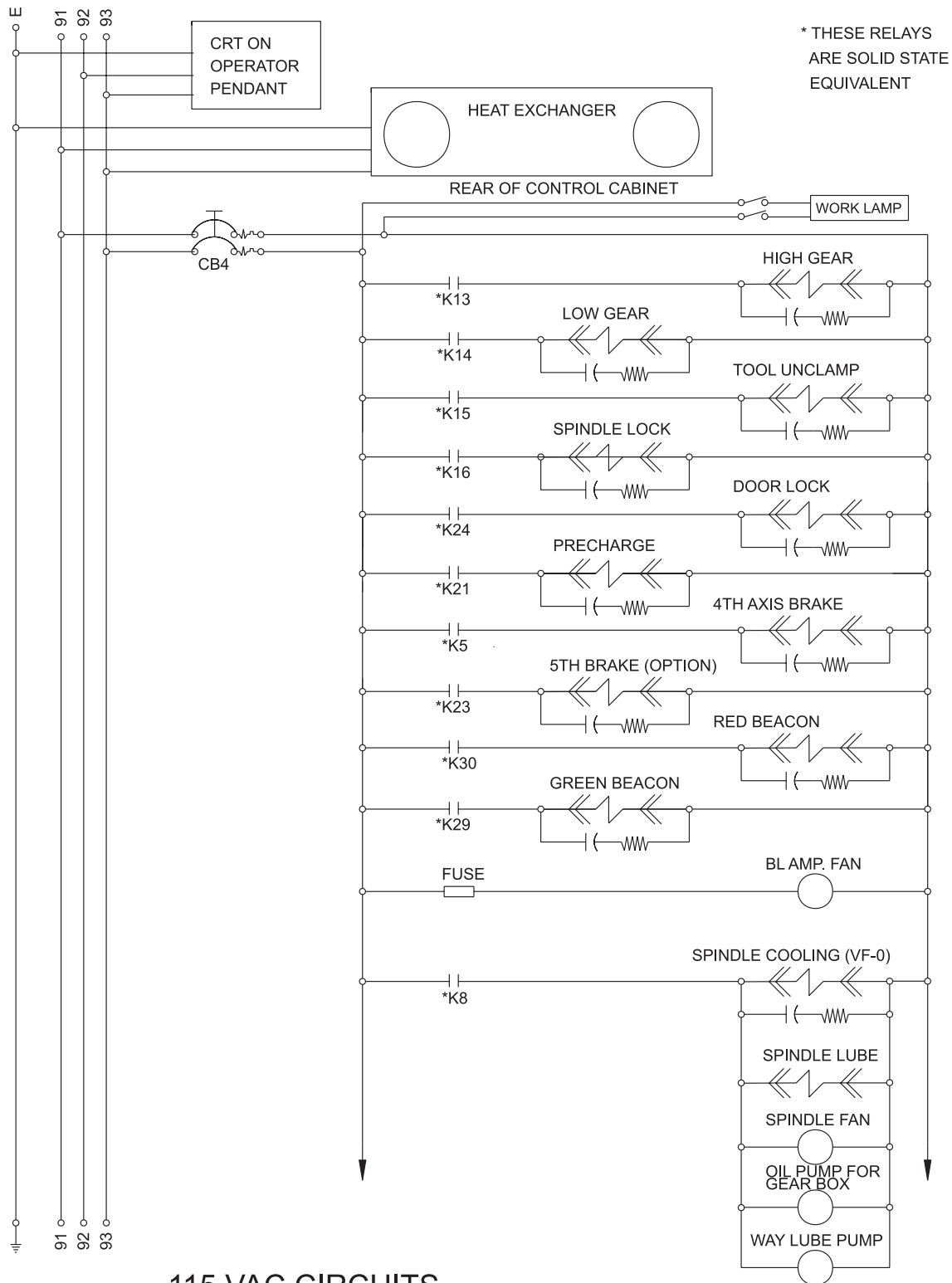
6/99 HAAS AUTOMATION VF SERIES PAGE 3



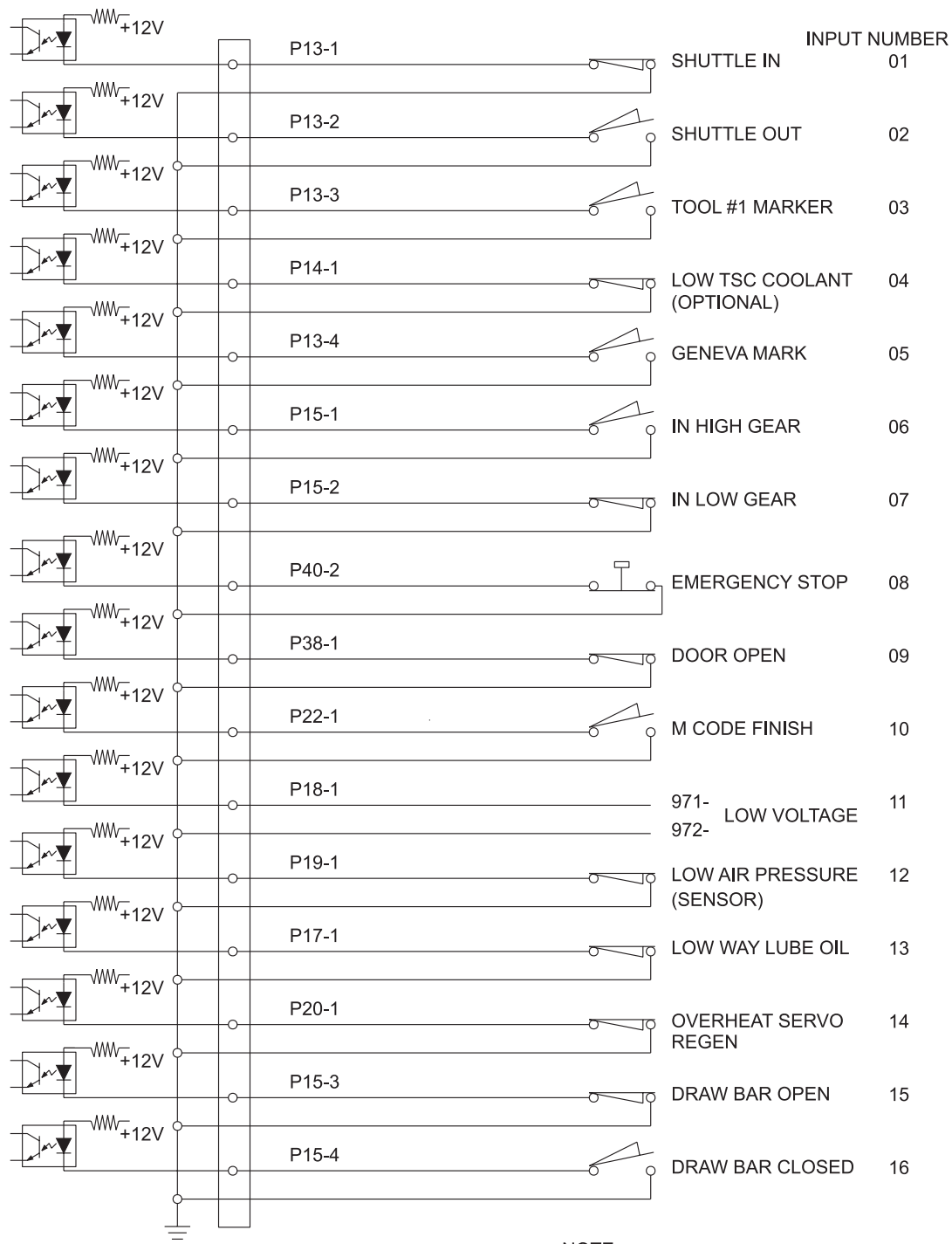
CNC UNIT
6/99 HAAS AUTOMATION VF SERIES PAGE 4



115 VAC 3 PHASE FROM T1

115 VAC CIRCUITS
6/99 HAAS AUTOMATION

VF SERIES PAGE 5

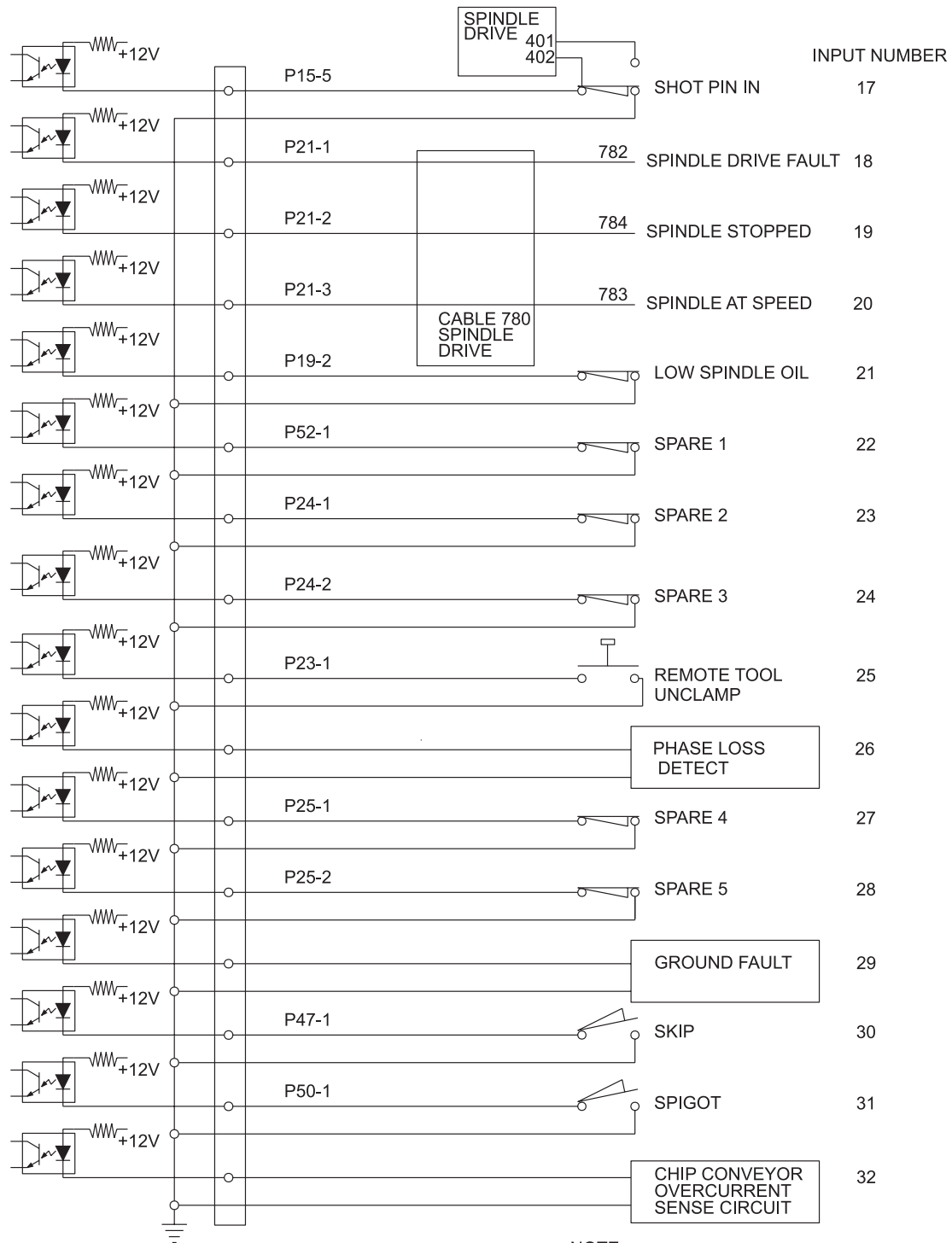


IOPCB CABLE 550

NOTE:
SWITCHES SHOWN ARE IN A
NON - ALARM STATE / HIGH GEAR /
SHUTTLE OUT / TURRET AT TOOL 1 POSIT.

DISCRETE INPUTS 1 THROUGH 16

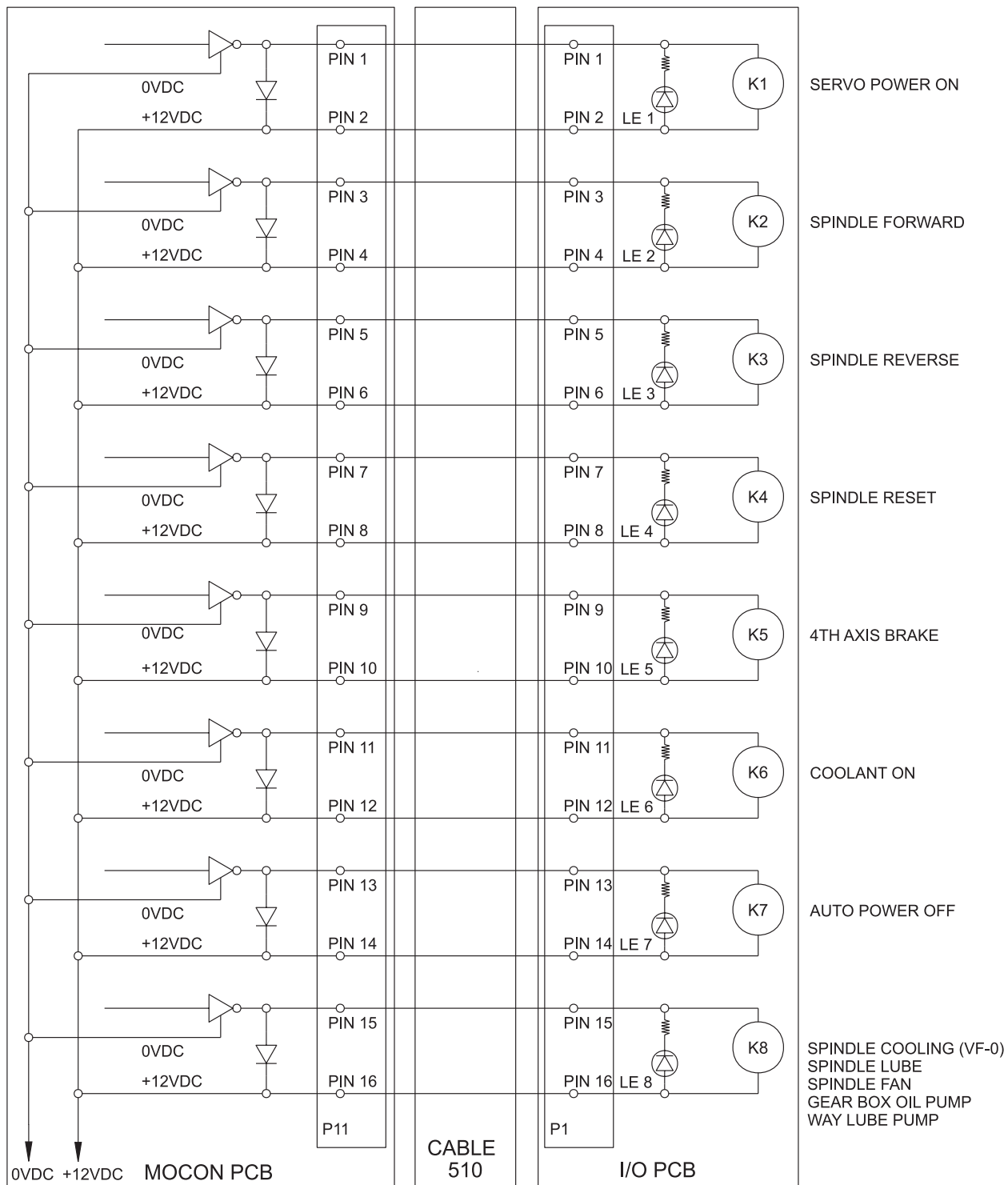
6/99 HAAS AUTOMATION VF SERIES PAGE 6



IOPCB CABLE 550 (CONT.)

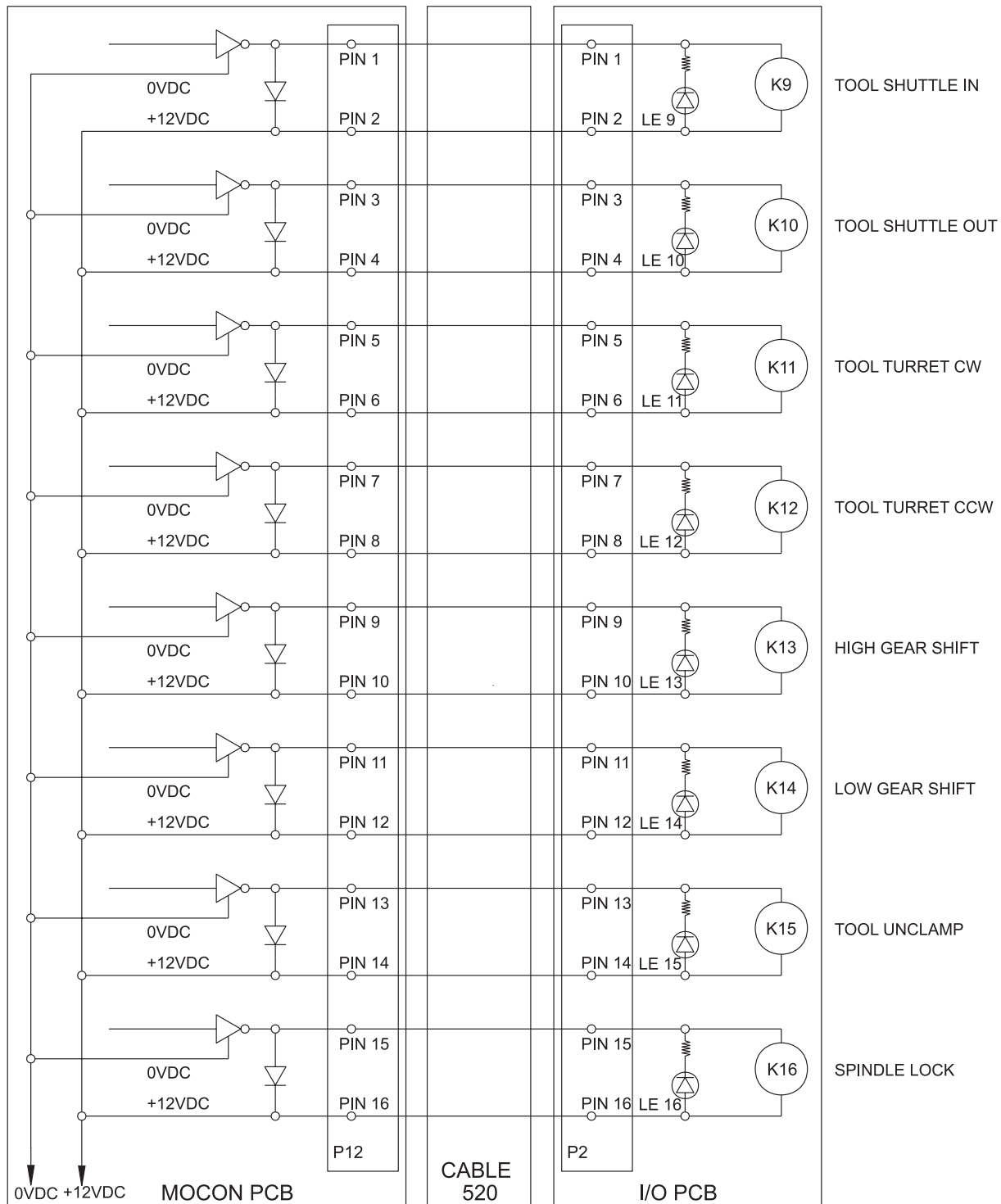
DISCRETE INPUTS 17 THROUGH 32

6/99 HAAS AUTOMATION VF SERIES PAGE 7



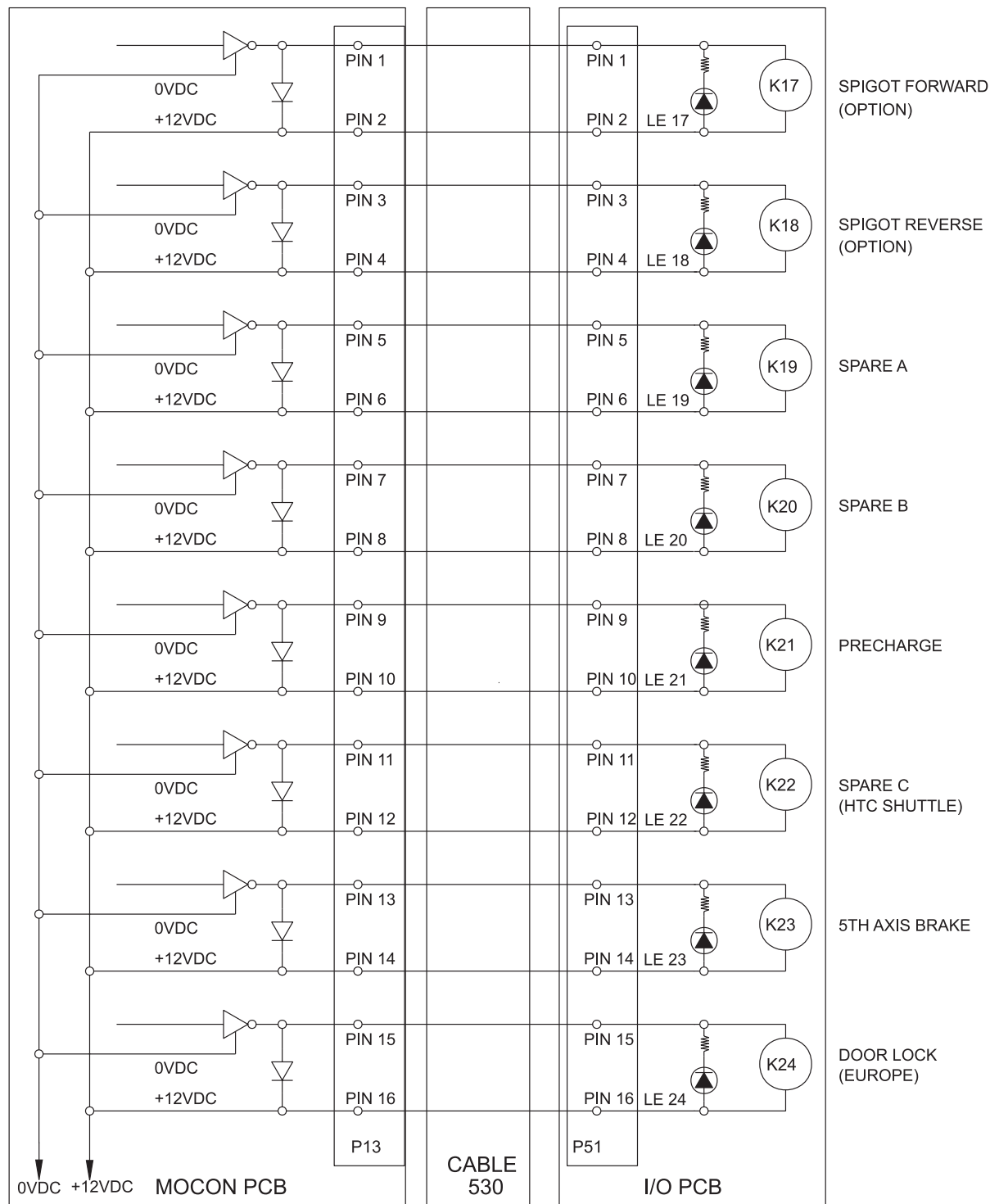
NOTE: ANY RELAY K1K35 CAN BE REPLACED
WITH A SOLID STATE EQUIVALENT

RELAY COIL DRIVERS, K1 THROUGH K8
6/99 HAAS AUTOMATION VF SERIES PAGE 8



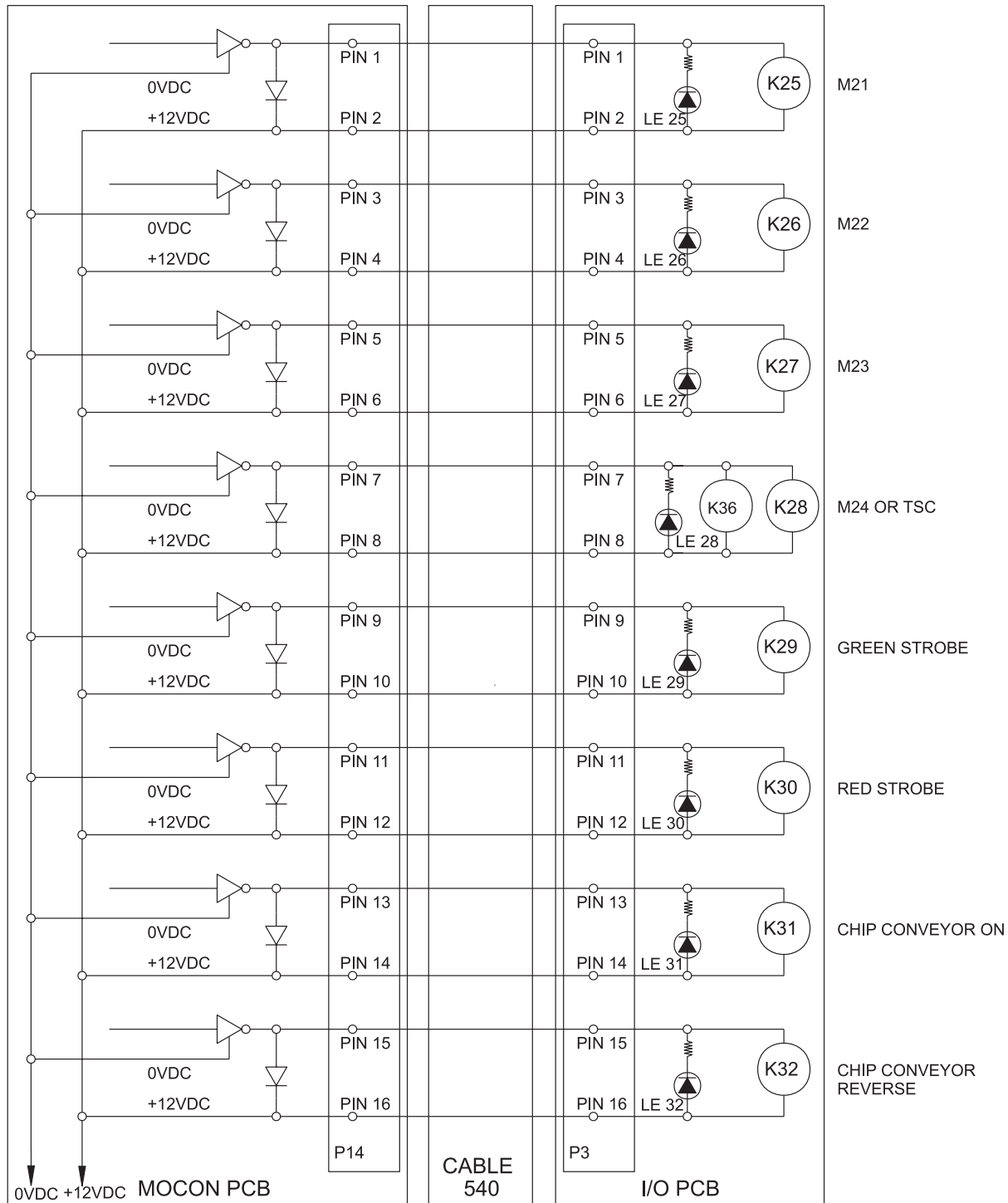
NOTE: ANY RELAY K1K35 CAN BE REPLACED
WITH A SOLID STATE EQUIVALENT

RELAY COIL DRIVERS, K9 THROUGH K16
6/99 HAAS AUTOMATION VF SERIES PAGE 9



NOTE: ANY RELAY K1K35 CAN BE REPLACED
WITH A SOLID STATE EQUIVALENT

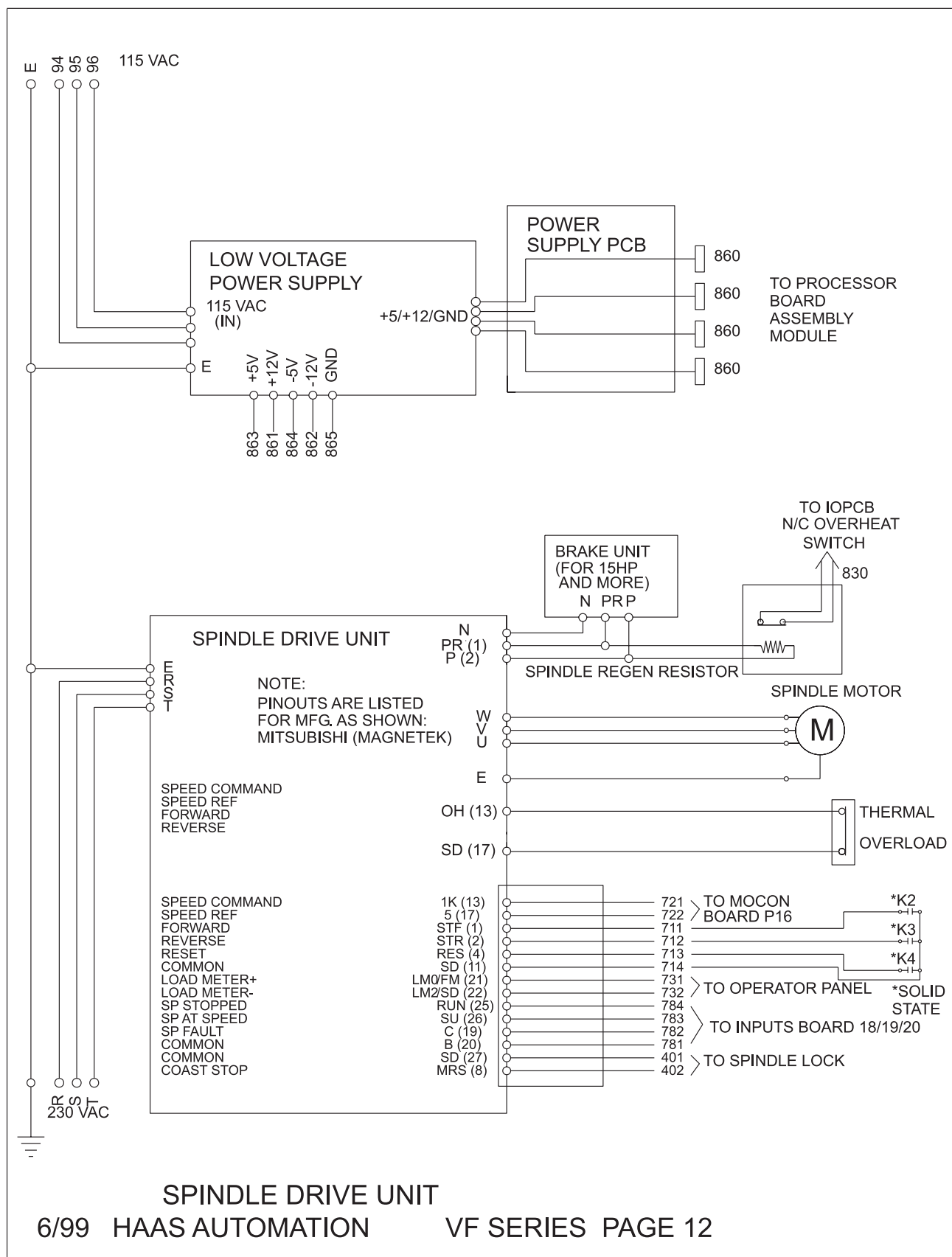
RELAY COIL DRIVERS, K17 THROUGH K24
6/99 HAAS AUTOMATION VF SERIES PAGE 10

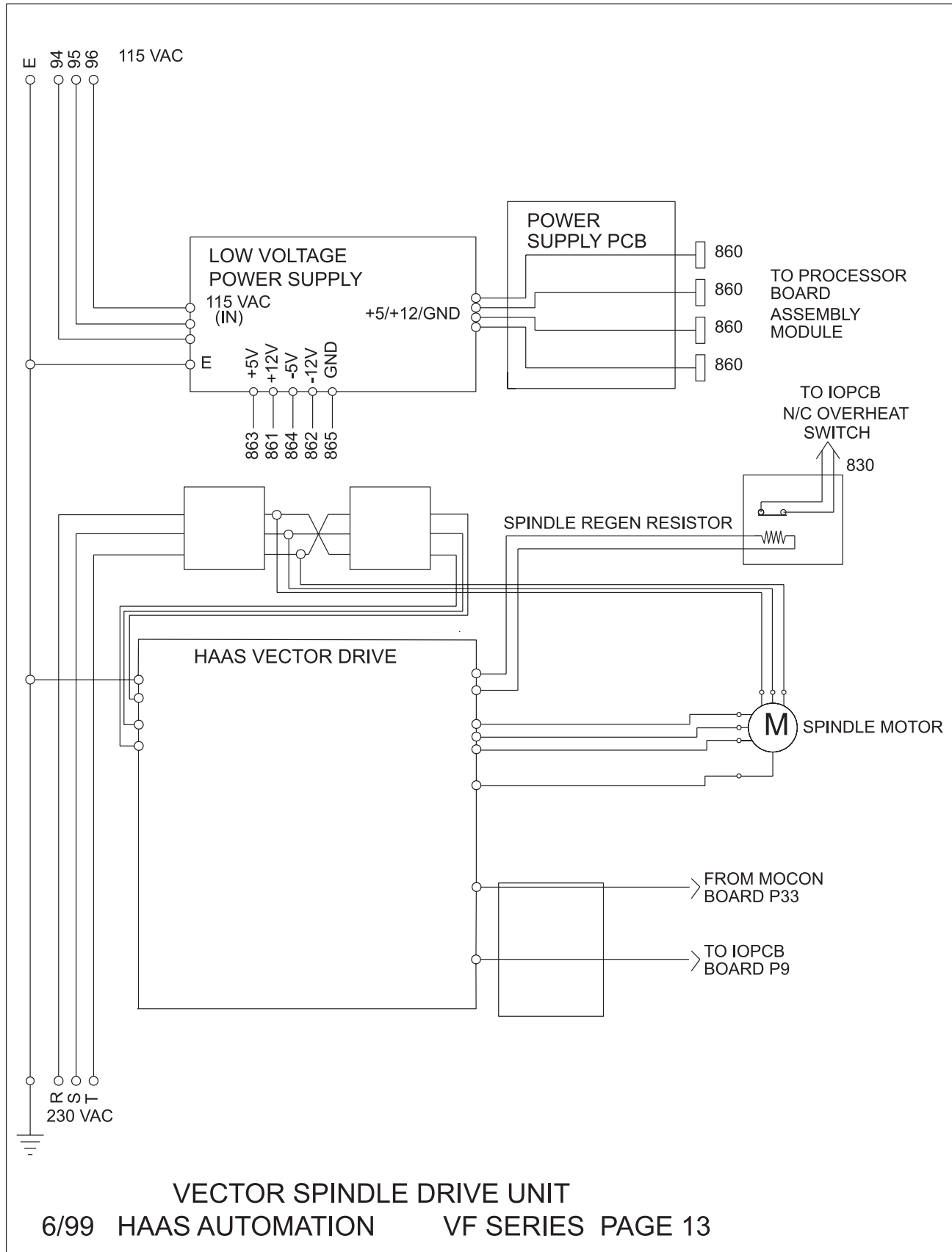


NOTE: ANY RELAY K1K35 CAN BE REPLACED
WITH A SOLID STATE EQUIVALENT

RELAY COIL DRIVERS, K25 THROUGH K32

6/99 HAAS AUTOMATION VF SERIES PAGE 11



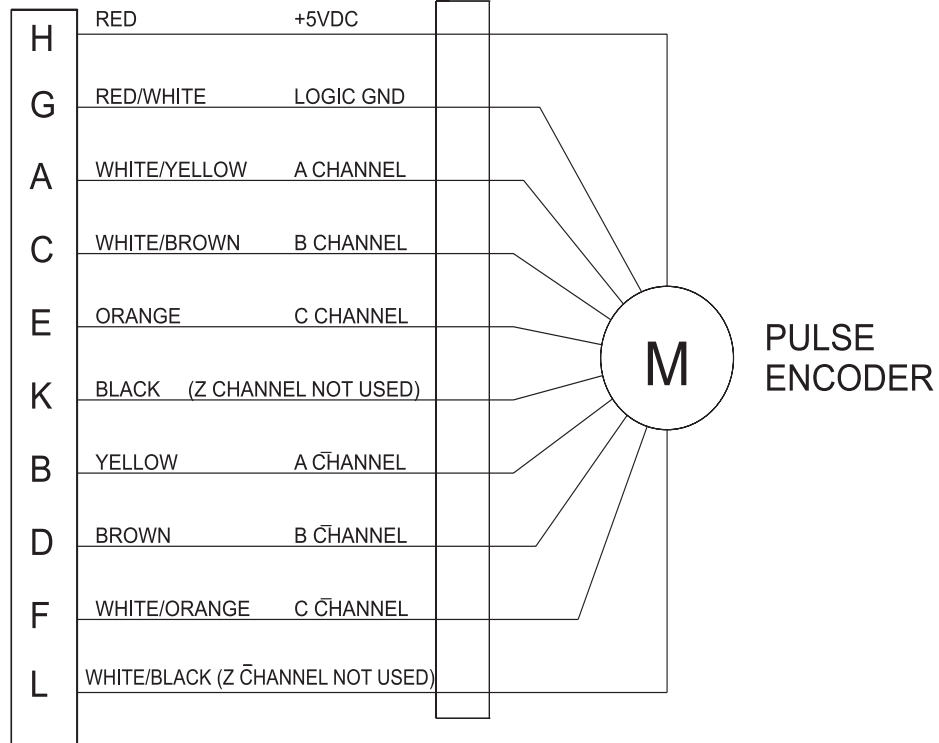




TO MOCON PCB
THROUGH CABLES:
32-1425A TO
32-1430A

X AXIS 660
Y AXIS 670
Z AXIS 680
A AXIS 690
*B AXIS 690B

17 PIN CONNECTOR



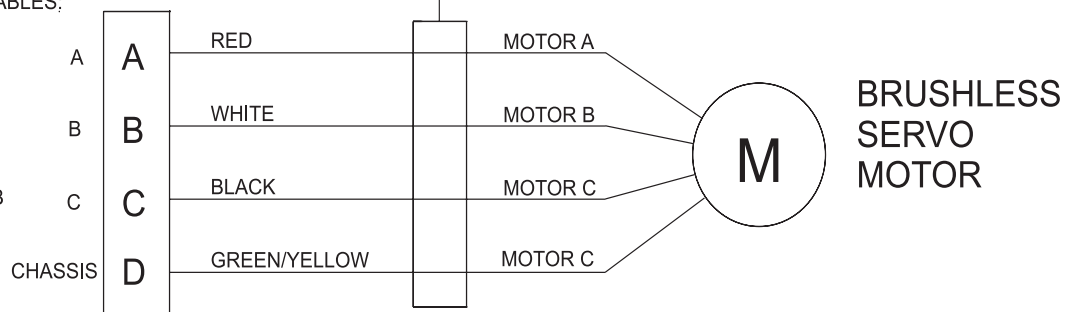
PULSE
ENCODER

TO BRUSHLESS
SERVO AMPLIFIERS
THROUGH CABLES;

32-1425A TO
32-1430A

X=491/492
Y=493/494
Z=495/496
A=497/498
*B=497B/498B

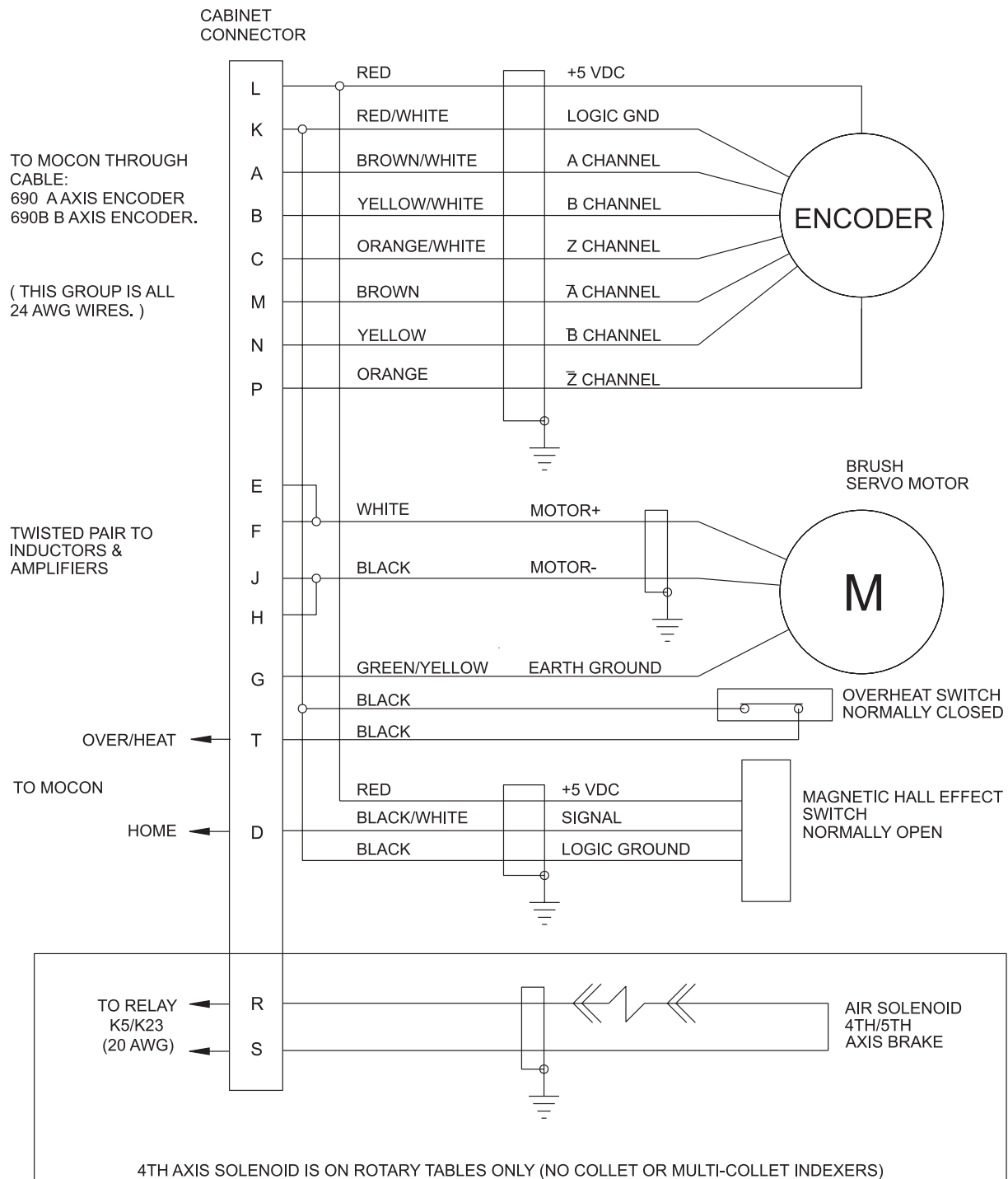
4 PIN CONNECTOR



BRUSHLESS
SERVO
MOTOR

*B AXIS OPTIONAL

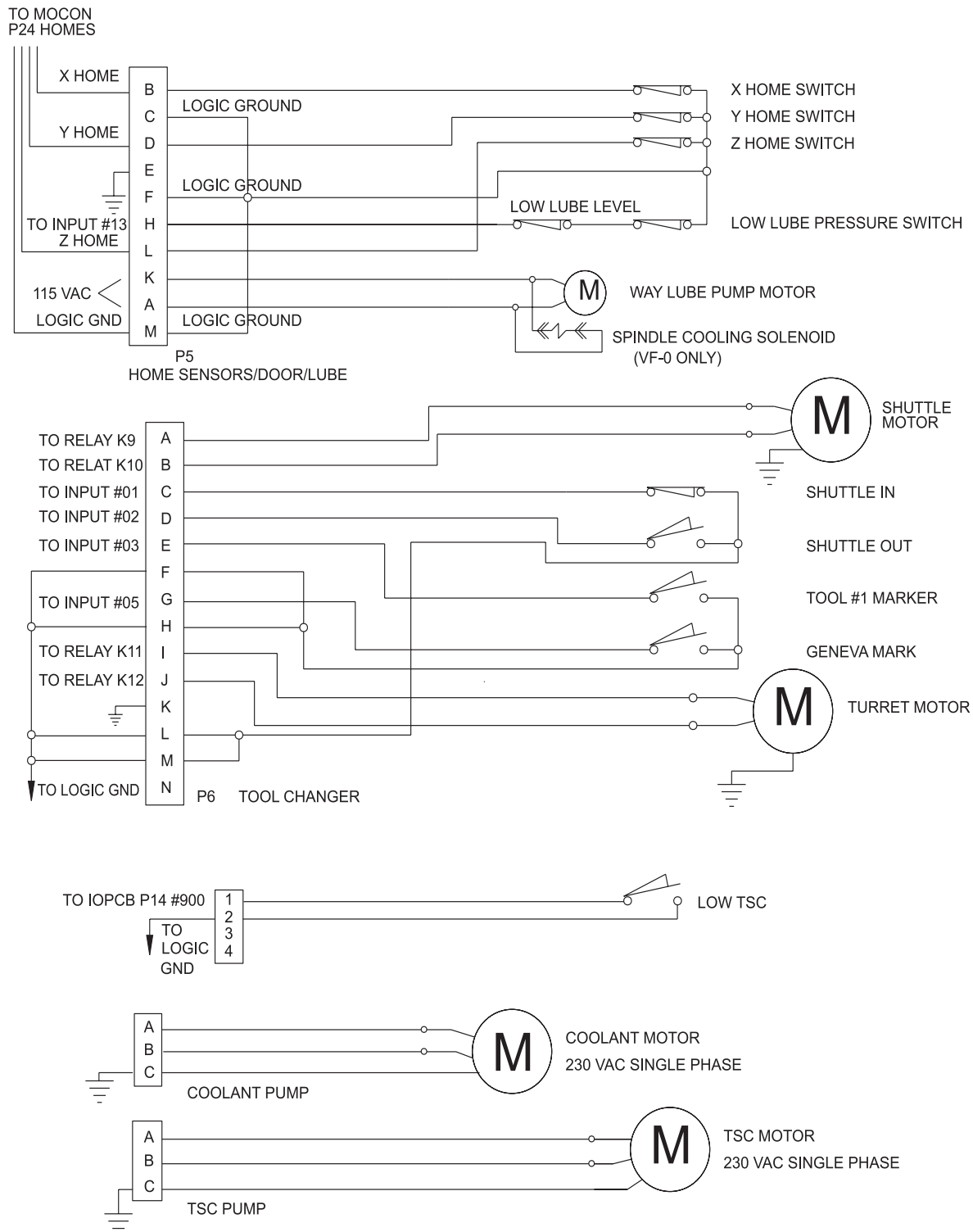
X/Y/Z AXIS MOTOR & ENCODER
6/99 HAAS AUTOMATION VF SERIES PAGE 14



A/B AXIS

6/99 HAAS AUTOMATION

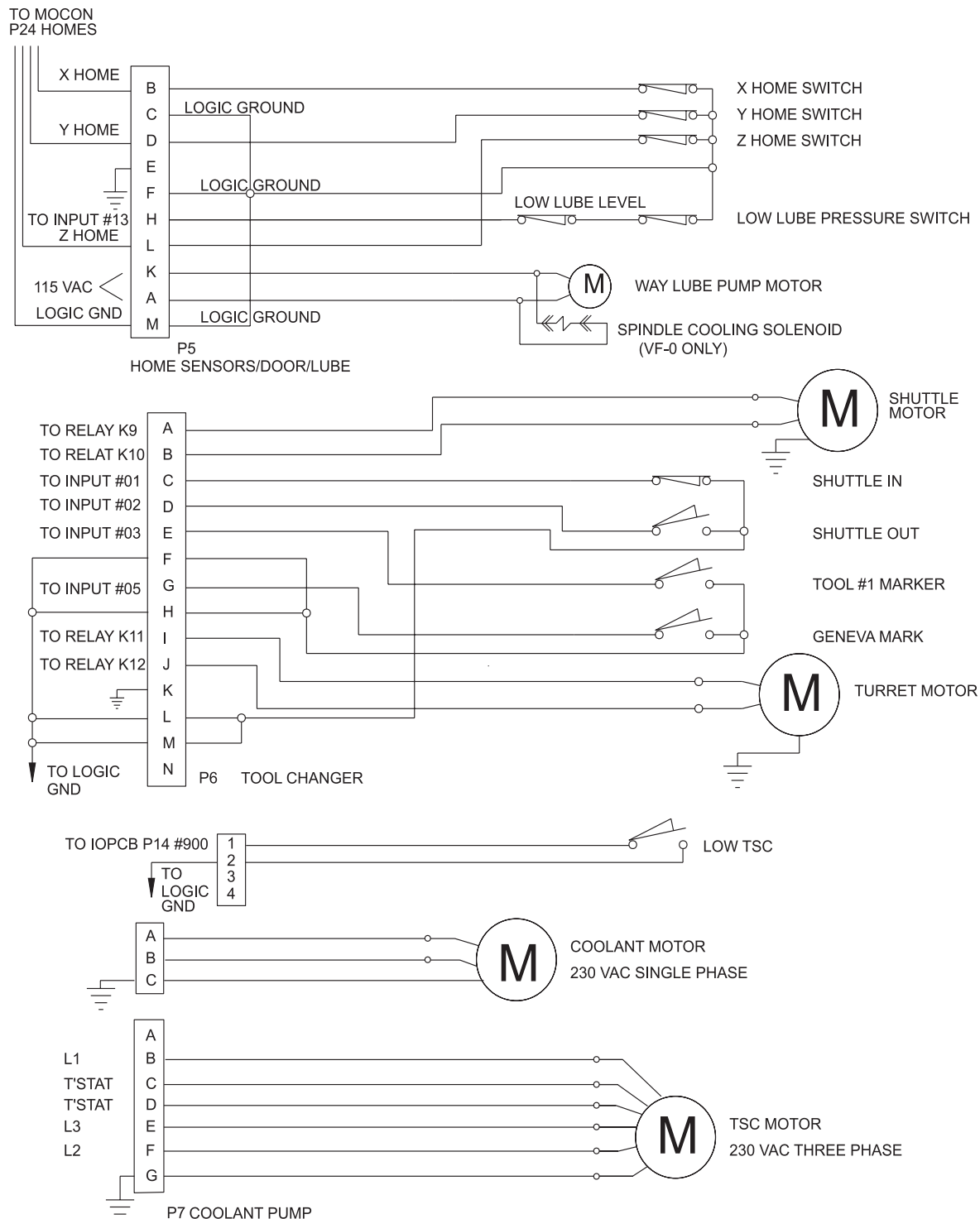
VF SERIES PAGE 15



NOTE: CONNECTORS ARE LOCATED ON SIDE OF CONTROL CABINET.

CABINET CONNECTORS

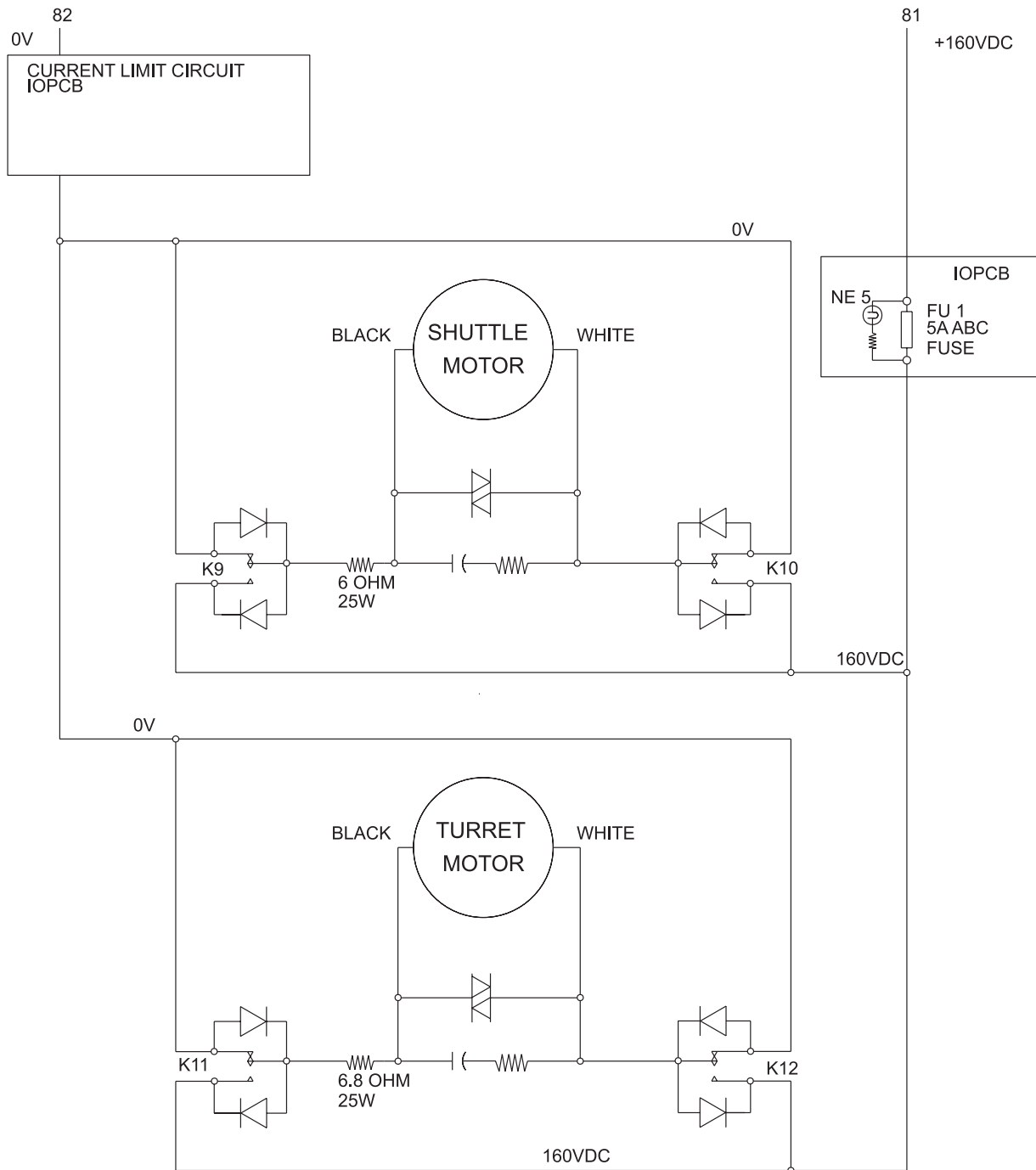
6/99 HAAS AUTOMATION VF SERIES PAGE 16



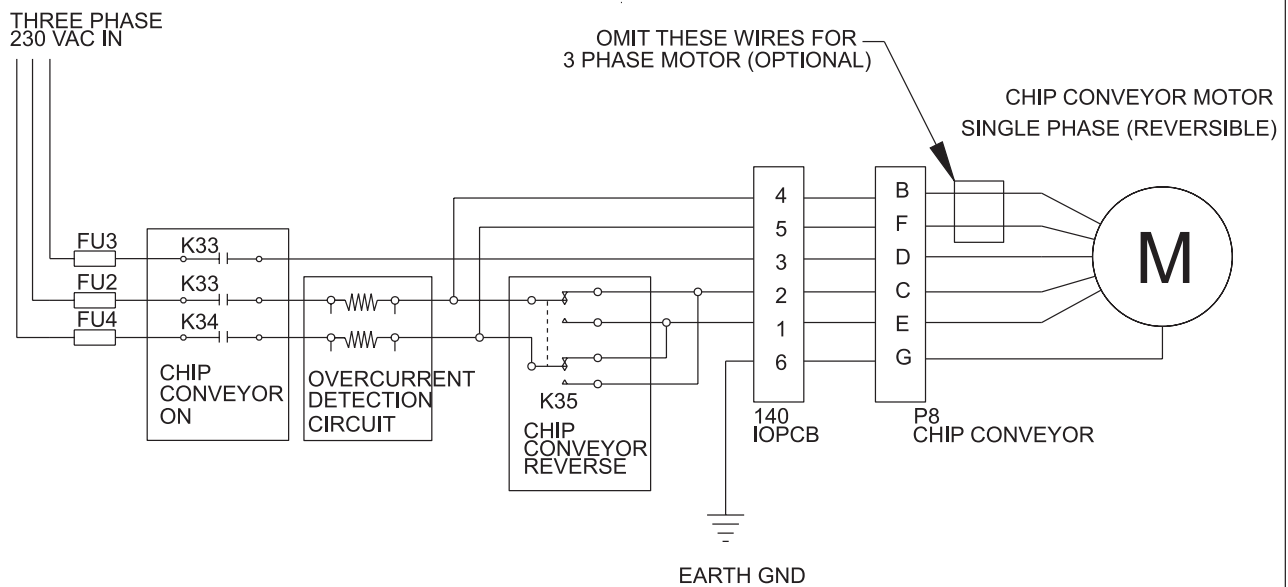
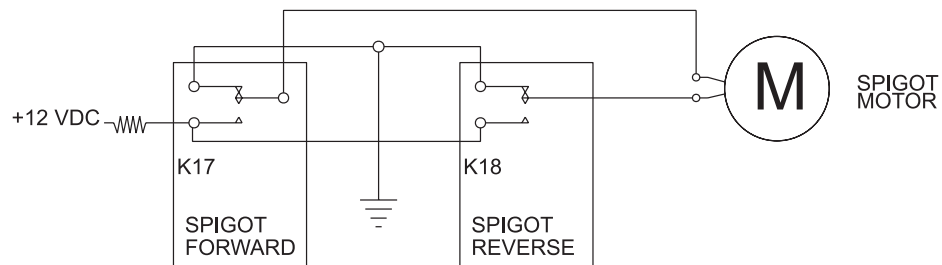
NOTE: CONNECTORS ARE LOCATED ON SIDE OF CONTROL CABINET.

CABINET CONNECTORS

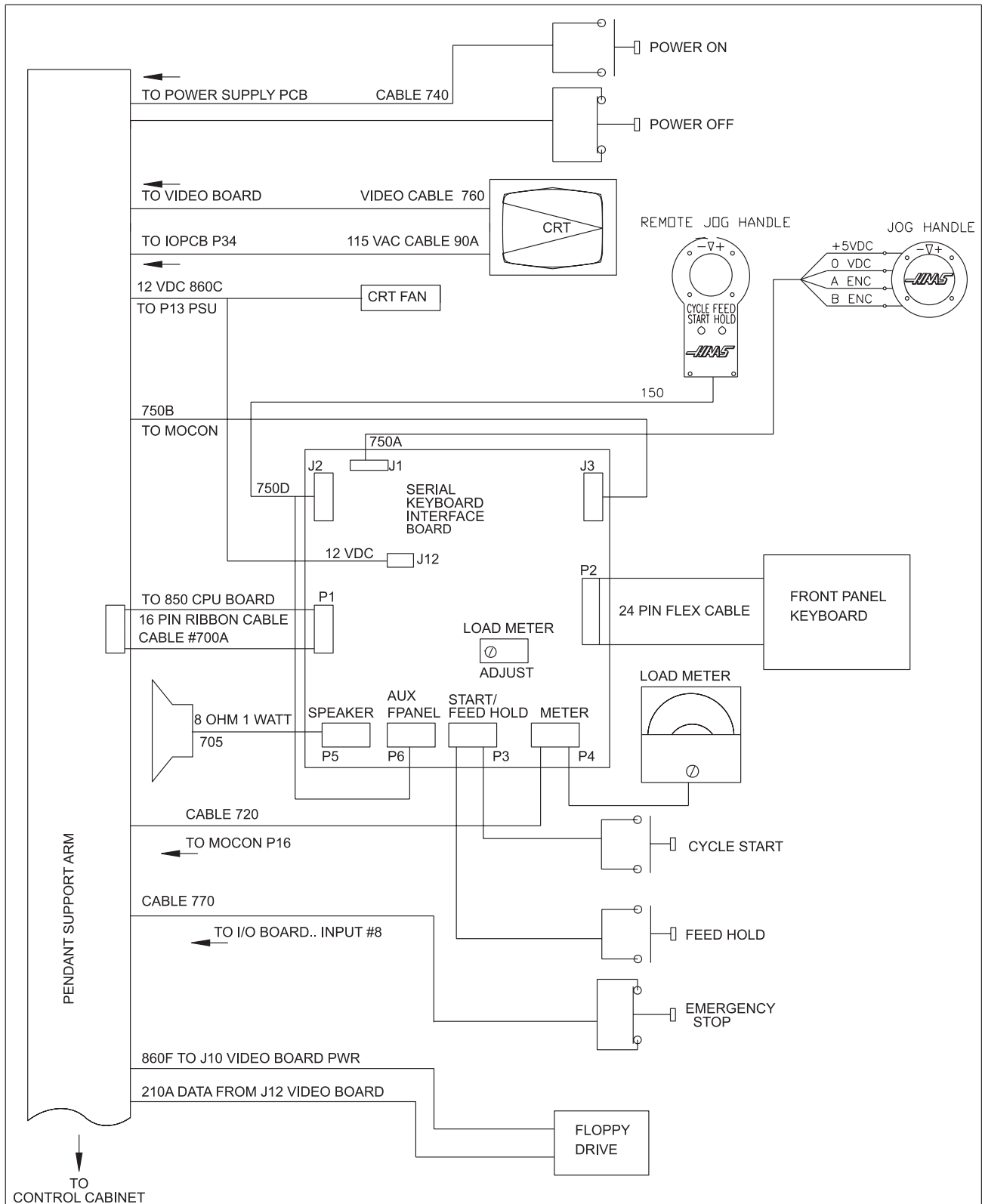
6/99 HAAS AUTOMATION VF SERIES PAGE 16A



TOOL CHANGE MOTORS
6/99 HAAS AUTOMATION VF SERIES PAGE 17



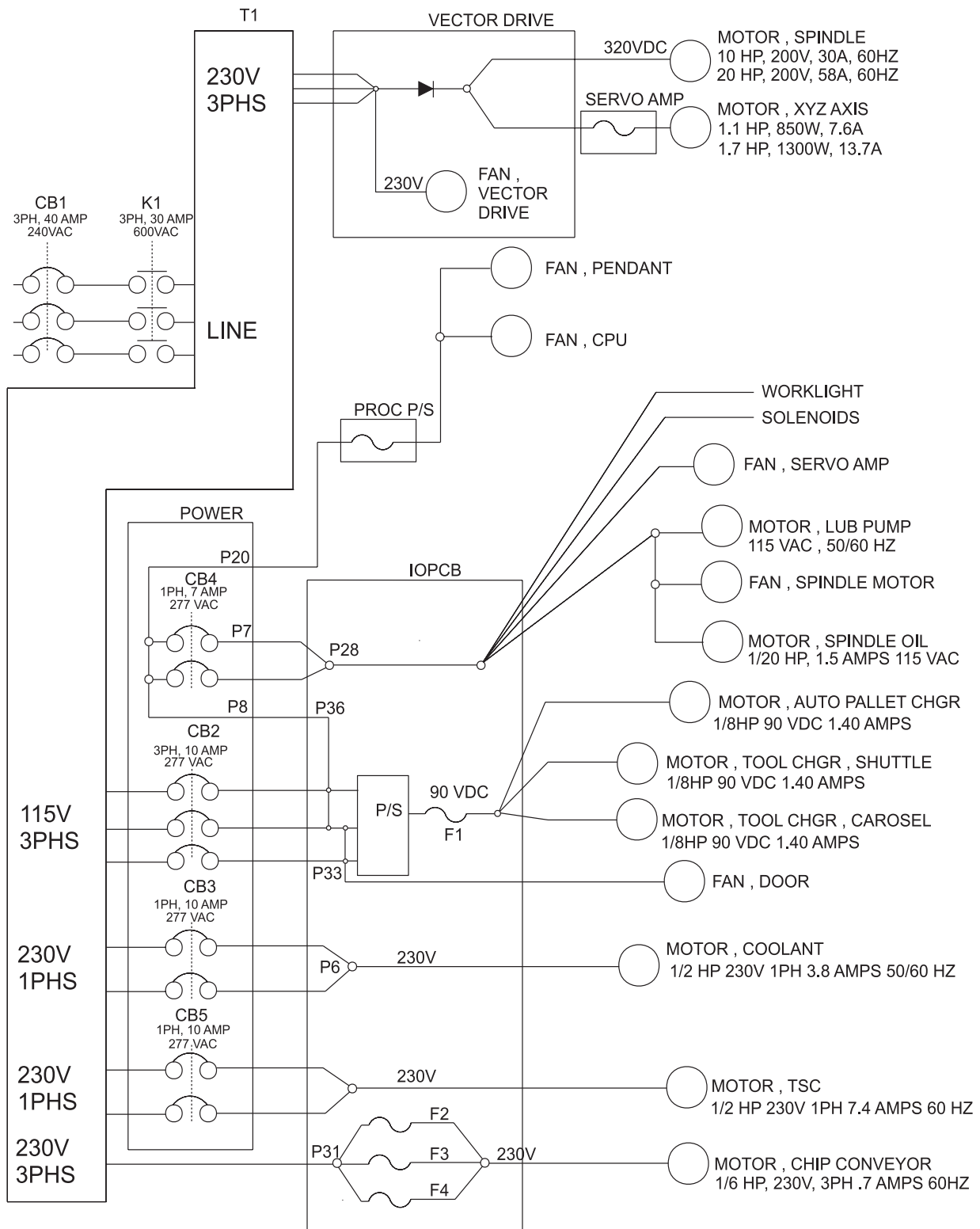
CHIP CONVEYOR AND SPIGOT MOTORS
6/99 HAAS AUTOMATION VF SERIES PAGE 18

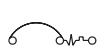
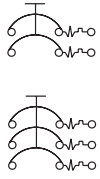


OPERATOR PENDANT

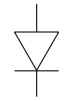
6/99 HAAS AUTOMATION

VF SERIES PAGE 19



CIRCUIT BREAKER
(SINGLE)CIRCUIT BREAKER
(MULTI)

COIL



DIODE



GROUND



LAMP

LED
(LIGHT EMITTING DIODE)

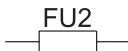
LIMIT SWITCH (CLOSED)



LIMIT SWITCH (OPEN)



MOTOR



FUSE



VARISTOR



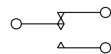
NEON BULB (W/ RESISTOR)

PUSH BUTTON SWITCH
(NORMALLY CLOSED)PUSH BUTTON SWITCH
(NORMALLY OPEN)

RELAY (CLOSED)



RELAY (OPEN)

RELAY (SINGLE POLE
DOUBLE THROW)

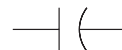
RESISTOR



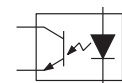
SOLENOID



TRANSFORMER



CAPACITOR



OPTO-ISOLATOR

ELECTRICAL SYMBOLS

6/99 HAAS AUTOMATION

VF SERIES PAGE 21

