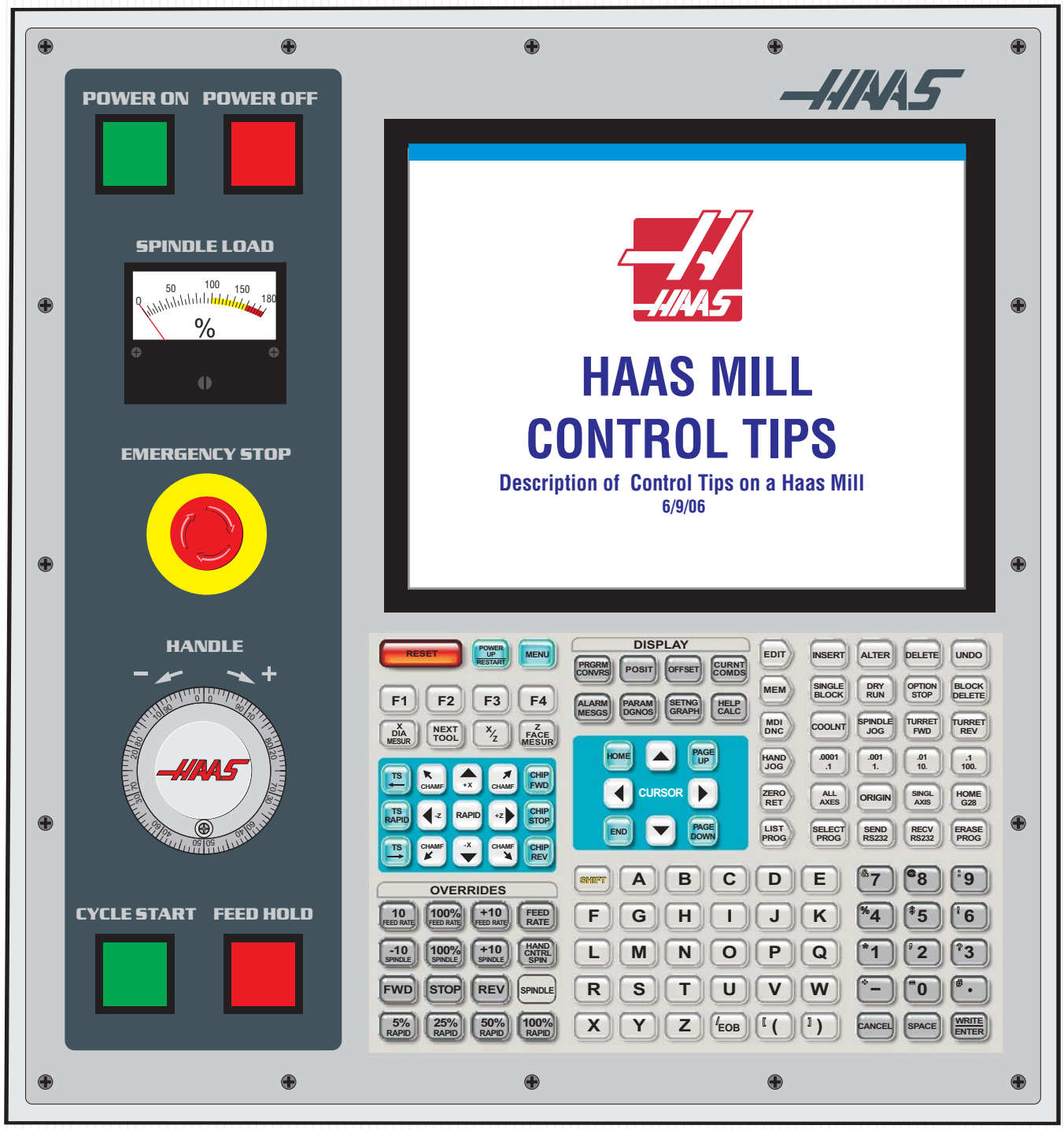
POWER ON
POWER OFF
SPINDLE LOAD
0
50
100
150
180
%
EMERGENCY STOP
HANDLE
CYCLE START
FEED HOLD
HAAS MILL
CONTROL TIPS
Description of Control Tips on a Haas Mill
6/9/06
RESET
POWER UP RESTART
MENU
F1
F2
F3
F4
DISPLAY
PRGRM CONVRS
POSIT
OFFSET
CURNT COMDS
ALARM MESGS
PARAM DGNOS
SETNG GRAPH
HELP CALC
EDIT
INSERT
ALTER
DELETE
UNDO
MEM
SINGLE BLOCK
DRY RUN
OPTION STOP
BLOCK DELETE
MDI DNC
COOLNT
SPINDLE JOG
TURRET FWD
TURRET REV
HAND JOG
ZERO RET
LIST PROG
HOME
PAGE UP
CURSOR
END
PAGE DOWN
ALL AXES
ORIGIN
SINGL AXIS
HOME G28
SELECT PROG
SEND RS232
RECV RS232
ERASE PROG
CHAMF
RAPID
CHIP FWD
CHIP STOP
CHIP REV
OVERRIDES
FEED RATE
HAND CNTRL SPIN
SPINDLE
FWD
STOP
REV
5% RAPID
25% RAPID
50% RAPID
100% RAPID
SHIFT
CANCEL
SPACE
WRITE ENTER

CONTROL TIPS

CONTROL TIPS

GENERAL TIPS

Δ Cursor Searching for a Program - When in EDIT or MEM mode, you can select and display another program quickly by simply entering the program number (Onnnnn) and pressing, cursor arrow down.

Δ Searching for a Program Command - Searching for a specific command in a program can be done in either MEM or EDIT mode. Enter the address letter code (A, B, C, etc.) or address letter code with the value (A1.23), and press the up or down cursor arrow. If you enter just the address code and no value, the search will stop at the next use of that letter, regardless of the value.

Δ Spindle Command - You can stop or start the spindle with **CW** or **CCW** any time you're at a Single Block stop or a Feed Hold. When you restart the program with **CYCLE START**, the spindle will be turned back on to the previously defined speed.

Δ Coolant Pump - The coolant pump can be turned on or off manually while a program is running, by pressing the **COOLNT** button. This will override what the program is doing until another M08 or M09 coolant command is executed. This also applies to the chip conveyor.

Δ Coolant Spigot Position - The coolant spigot position can be changed manually when a program is running, with the **CLNT UP** or **CLNT DOWN** key. This will override what the program commands until another spigot position is commanded with an Hnn or M08 command.

Δ Actual Spindle Speed - If you have the rigid tapping option, you can verify the exact spindle speed by checking on the right side of the first page on the CURNT COMDS display, where the actual spindle speed is listed next to ACT Snnnn, and in the bottom right corner**.**

Δ Memory Lock Key Switch - This is a customer machine option that prevents the operator from editing or deleting programs, and from altering settings when in the locked position. Since the Key switch (if installed) locks out the Settings, it also allows you to lock out other areas within the settings: Setting 7 locks out parameters: Parameter 57, 209, and 278 lock other features. Setting 8 locks all programs. Setting 23 locks 09xxx programs. Setting 119 locks offsets. Setting 120 locks macro variables. In order to edit or change these areas, the key switch (if installed) must be unlocked and the setting turned off .
(Any Mill Control ver. 9.25 and above; any Lathe Control ver. 2.23 and above.)

Δ Chip Conveyor - The chip conveyor can be turned on or off when a program is running, either manually using the control keys or in the program using M codes. The M code equivalent to **CHIP FWD** is M31 and M33 is **CHIP STOP**. You can set the Conveyor Cycle time (in minutes) with Setting 114, and the Conveyor On-Time (in minutes) with Setting 115.

Δ To Rapid an Axis Home - You can rapid *all* axes to machine zero by pressing the **HOME G28** key. You can also send just one axis (X, Y, Z, A or B) to machine zero in rapid motion. Enter the letter X, Y, Z, A or B, then press **HOME G28** and that axis alone will rapid home. ***CAUTION***! There is no warning to alert you of any possible collision! For example, if the Z axis is down near the part or fixture on the table, and then the X or Y axis is sent home using HOME G28, a crash may result. Care must be exercised.
(Any Mill Control ver. 9.49 and above; any Lathe Control ver. 2.24 and above.)

Δ Rotating Tool Carousel without Changing Tools (M39) - You can rotate the tool carousel without changing tools by using M39 and Tnn. This can be used to recover from some unusual conditions. However, it will also tell the control you have a different tool in the spindle, which may not be the case. You can disable this command, if you choose, by turning on Setting 86, M39 LOCKOUT.
(Any Mill Control ver. 2.3 and above.)

CONTROL TIPS

PRGRM /CONVRS

Δ Program Review - In PRGRM display, Program Review allows you to cursor through and review an active program that's on the right side of the display screen, while viewing the same program running on the left side of the screen. To bring up Program Review, press **F4** while running a program in MEM mode and in PRGRM display.

Δ Background Edit - This feature allows you to edit a program in MEM mode and PRGRM display while a program is running. Type in the program number you want to edit (**Onnnnn**) and press **F4**. You can then do simple edits (**INSERT**, **ALTER**, **DELETE** and **UNDO**) either to an existing program, a new program, or even the program that is presently running. However, edits to the program that's running will not take effect until that program cycle ends with an M30 or **RESET**.

POSIT

Δ Quick Zero on DIST-TO-GO Display - A quick way to zero out a position display, for a distance move, when manually moving the machine, use the DIST-TO-GO position display. When you're jogging in the POSIT display and in HANDLE JOG mode, press any other operation mode (EDIT, MEM, MDI, etc.) and then back to HANDLE JOG. This will zero out all axes on DIST-TO-GO display and begin showing a distance moved to the next location.

Δ To Origin the POS-OPER Display - This display is used for reference only. Each axis can be zeroed out independently, to then show its position relative to where you selected to zero that axis. To zero out a specific axis, **PAGE UP** or P**AGE DOWN** in the POSIT display to the POS-OPER large-digit display page. When you Handle Jog the X, Y or Z axis and then press **ORIGIN**, the axis that is selected will be zeroed. Or, you can press an **X**, **Y** or **Z** letter key and then **ORIGIN** to zero that axis display. You can also press the **X**, **Y** or **Z** key and enter a number (X2.125), then press **ORIGIN** to enter the number in that axis display.

OFSET

Δ Entering Offsets - Pressing **WRITE/ENTER** will *add* the number in the input buffer to the cursor-selected offset value. Pressing **F1** will *replace* the selected offset with the number in the input buffer.

Δ Entering Offsets - Pressing **OFSET** again will toggle back and forth between the Tool Length Offsets and Work Zero Offsets pages.

Δ Entering Offsets - In the OFSET display, **F1** will set the entered value into the offsets. **F2** will set the negative of the entered value into the offsets.

Δ Tool Offset Measure and Part Zero Set Measurement keys - With the touch of a key you can enter in either a tool length offset or a work offset to help save time during a part setup. The NEXT TOOL key is then used to send the machine home for a toolchange for the next tool. The cursor will then position to the next offset register and the jog handle increment will go back to 10. inches per minute.

Δ Coolant Spigot Position - The coolant spigot can be programmed to move to the location entered in the OFSET display, on the Tool Geometry Offset page under CLNT POS. You can adjust the coolant nozzle up to 10 different positions for a tool. Position 1 is the farthest up and 10 is the farthest down. The nozzle will shift to that position whenever an M08 or Hnn code is executed in the program. When you cursor onto the CLNT POS column, the coolant nozzle location is displayed in the lower left corner.

Δ Clearing All Offsets - When you're in the Tool Length Offset display, you can clear all offsets by pressing the **ORIGIN** key. The control will prompt: "ZERO ALL (Y/N)?" to make sure this is what you really want to do. If **Y** is entered, all the offsets in the area being displayed will be zeroed. If you're in the Work Zero Offsets display, pressing ORIGIN will do the same thing.
(Any Mill Control ver. 10.02 and above; any Lathe Control ver. 3.00 and above.)

CONTROL TIPS

Δ The Mill Offers up to 200 Tool Offsets - Mills now offer up to 200 tool offsets. In all previous versions, the maximum number available was 100 tool offsets.
(Any Mill Control ver. 10.22 and above.)

Δ Send and Receive Offsets, Settings, Parameters and Macro Variables to/from Disk - You can save offsets, settings, and parameters to a floppy disk. *Press* **LIST PROG** *first*, then select an OFSET, SETNG or PARAM display page. Type in a file name and then press **F2** to write that display information to disk (or F3 to read that file from a disk). You can also do this with the macro variables by pressing **LIST PROG** *first*, then selecting the macro variable display page (**PAGE DOWN** from CURNT COMDS).

CURNT COMDS

Δ Current Commands - You can see 24 lines of the current program along with the programmed feed rate and spindle speed on the first page of the CURNT COMDS display, by looking at PGM Fnnnn and PGM Snnnn in the column on the far right. You can verify the actual feed rate and commanded spindle speed in the same column, with ACT Fnnnn and CMD Snnnn. The actual speed and feed are what the spindle speed and feed rate are really operating at with any adjustments to the OVERRIDE keys. In addition, this display shows spindle load, axes load, Surface Speed, Chip Load, spindle CW, CCW or spindle STOP commands. This display also shows the position of the axes in the upper right hand corner. The coordinates that can be displayed (operator, work, machine, or distance to go) are selected using the cursor up and down keys.

Δ Tool Life Management - In the CURNT COMDS display, you can **PAGE DOWN** to the Tool Life management page. On this page the tool Usage register is added to every time that tool is called up in the spindle. You enter the number of times you want that tool to be used in the Alarm column. When the Usage number for that tool reaches the number of uses in the Alarm column, it will stop the machine with an alarm. This will help you monitor tools to prevent them from breaking and parts being scrapped.

Δ Tool Load Management - The next **PAGE DOWN** in CURNT COMDS will bring you to the Tool Load page. Spindle load condition can be defined for a particular tool, and the machine will stop if it reaches the spindle load limit defined for that tool. A tool overload condition can result in one of four actions by the control. The action is determined by Setting 84. ALARM will generate an alarm when overload occurs; FEED HOLD will stop with a Feed Hold when overload occurs; BEEP will sound an audible alarm when overload occurs; or AUTOFEED will automatically decrease the feed rate. This will also help you monitor tools.

Δ Clearing Current Commands Values - The values in the CURNT COMDS display pages for Tool Life, Tool Load and Timer registers can be cleared by cursor-selecting the one you wish to clear and pressing **ORIGIN**. To clear everything in a column, cursor to the top of that column (onto the title) and press **ORIGIN**.

Δ Clearing All Macro Variables - When you're in the Tool Length Offset display, you can clear all the macro variables by pressing the **ORIGIN** key. The control will prompt: "ZERO ALL (Y/N)?" to make sure this is what you really want to do. If **Y** is entered, all the variables will be zeroed.
(Any Mill Control ver. 10.02 and above; any Lathe Control ver. 3.00 and above.)

Δ Send and Receive Offsets, Settings, Parameters and Macro Variables to/from Disk - You can save offsets, settings, and parameters to a floppy disk. *Press* **LIST PROG** *first*, then select an OFSET, SETNG or PARAM display page. Type in a file name and then press **F2** to write that display information to disk (or F3 to read that file from a disk). You can also do this with the macro variables by pressing **LIST PROG** *first*, then selecting the macro variable display page (**PAGE DOWN** from CURNT COMDS).

CONTROL TIPS

HELP

Δ Helpful Information - The HELP display has a list of all the G and M codes available. To see them, press the **HELP** display key, and then the letter **C** for a list of all the G and M codes for the machine.

Δ Helpful Information - You can also see a listing of all the different subject area's available in the Help directory by pressing the letter **D.** Select it by pressing the letter next to the subject area you wish to see.

CALC

Δ Transferring Simple Calculations - While in the Trigonometry, Circular or Milling and Tapping calculator, the number in the simple calculator box (in the upper left corner) can be transferred down to any cursor-selected data line. Cursor to the register you wish to transfer the calculator number to and press **F3**.

Δ Transferring Calculated Values - You can transfer the highlighted value in a Trig, Circular, or Milling data register into the calculator box by pressing F4. Use the up and down arrow keys to select the data input register, including the calculator box, and the left and right arrows to select LOAD + – * /. To enter a highlighted data register value into the calculator box, LOAD must be selected when you press **F4**. If one of the operations is selected (+ – * /.), pressing **F4** will perform that operation using the number in the highlighted data register and the number in the calculator box.

Δ Transferring to EDIT or MDI - In either EDIT or MDI mode, pressing **F3** will transfer the number in the calculator box (when the cursor is on the number in the box) to either the EDIT or MDI input buffer. You will need to enter the letter (X, Y or Z) you wish to use with the number from the calculator.

Δ Circular Calculator - The Circular Calculator will list four different ways that a circular move can be programmed using the values entered for a calculated solution. Four different program lines for executing the circular move will be listed at the bottom of the display. One of the four program lines can be transferred to either EDIT or MDI.

1. In the circular calculator, cursor onto the program line you wish to use.
2. Press either **EDIT** or **MDI**, where you wish to insert the circular move.
3. Press the **F3** key, which will transfer the circular move that you highlighted into the input buffer line at the bottom of the EDIT or MDI display.
4. Press **INSERT** to add that circular command line into your program.

Δ One-Line Expressions - The CALC display will now accept and evaluate a simple expression. This is a new feature; previously it was only possible to enter a number into the input line of the calculator. Now, the calculator allows you to enter a simple, one-line expression without parentheses, such as 23*4–5.2+6/2. It will be evaluated when the **WRITE/ENTER** key is pressed and the result (89.8 in this case) displayed in the calculator box. Multiplication and division are performed before addition and subtraction.
(Any Mill Control ver. 9.49 and above; any Lathe Control ver. 2.24 and above.)

SETNG

Δ Scrolling through Settings with the Jog Handle - The jog handle can now be used to scroll through the settings. In previous versions, the jog handle could only be used to scroll through (cursor-highlight) the parameters, but not the settings. This has been corrected.
(Any Mill Control ver. 10.15 and above; any Lathe Control ver. 3.05 and above.)

CONTROL TIPS

Δ There are so many Settings which give the user powerful and helpful command over their control, that you should read the entire Settings section of the operator's manual. Here are some of the useful settings to give you an idea of what is possible.

Setting 1 - **AUTO POWER OFF TIMER** - This turns the machine off after it is idle for the number of minutes defined in this setting.

Setting 2 - **POWER OFF AT M30** - This will power off the machine when an M30 command is executed. In addition, for safety reasons, the control will turn itself off if an overvoltage or overheat condition is detected for longer than four minutes.

Setting 8 - **PROG MEMORY LOCK** - When this is Off, control program memory can be modified. When this setting is turned On, memory edits cannot be done and programs cannot be erased.

Setting 9 - **DIMENSIONING** - This changes the machine control from inch to metric, which will change all offset values and position displays accordingly. This setting *will not* change your program to either inch or metric.

Setting 15 - **H & T CODE AGREEMENT** - When this is OFF, no special functions occur. When it's ON, a check is made to ensure that the H offset code matches the tool presently in the spindle. Usually you have one offset per tool, and it's usually the same number as the tool number. If it's not the same and this setting is ON, you will get an alarm of H AND T NOT MATCHED. This check can help prevent crashes. If you need to use a different offset number or more than just one, this setting will need to be switched OFF. In program restart this check is not done until motion begins.

Setting 31 - **RESET PROGRAM POINTER** - When this is On, the **RESET** key will send the cursor (program pointer) back to the beginning of the program.

Setting 36 - **PROGRAM RESTART** - When it is OFF, starting a program from anywhere other than the beginning of a program or a tool sequence may produce inconsistent results. When it is ON, you are able to start a program from the middle of a tool sequence. You cursor onto the line you want to start on and press CYCLE START. It will cause the entire program to be scanned to ensure that the correct tools, offsets, G codes, and axes positions are set correctly before starting and continuing at the block where the cursor is positioned. Some alarm conditions are not detected prior to motion starting. You could leave this setting ON all the time if you want, but it might do some things unnecessarily, so you would probably prefer to turning it OFF when you're done using it.

Setting 51 - **DOOR HOLD OVERRIDE** - 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 turned ON, and Parameter 57 bits DOOR STOP SP and SAFETY CIRC are set to zero, this door condition is ignored. When the control is turned off and then back on again, this is one of a few settings that will switch back to its default condition, and will then be OFF when you power up, so you will need to turn it ON again if that's what's needed.

Setting 77 - **SCALE INTEGER F** - This can be used to change how the control interprets a feed rate. A feedrate that's entered in your program can be misinterpreted if you don't enter a decimal point in the Fnn.nn command. The choices for this setting are DEFAULT, which assumes a 4 place decimal, if no decimal point is entered (i.e., if you enter F10, it assumes F0.0010); INTERGER, which assumes a whole number (enter F10, it assumes 10.0); or .1 (enter F10, it assumes 1.0), .01 or .001 (you get the idea), or .0001, which is the same as Default.

CONTROL TIPS

Setting 84 - **TOOL OVERLOAD ACTION** - This is used to determine tool overload conditions as defined by the Tool Load monitor page in the CURNT COMDS display (use **PAGE DOWN** to get there). A tool overload condition can result in one of four actions by the control, depending on Setting 84. ALARM will generate an alarm when overload occurs; FEED HOLD will stop with a Feed Hold when overload occurs; BEEP will sound an audible alarm when overload occurs; or AUTOFEED will automatically decrease the feed rate.

Setting 85 - **MAX CORNER ROUNDING** - This setting is used to set the corner rounding accuracy required by the user. The accuracy defined in Setting 85 will be maintained even at maximum feed rate. The control will ONLY slow at corners *when it is needed*.

Setting 88 - **RESET RESETS OVERRIDE** - When this is On, the RESET key sets all overrides back to 100%.

Setting 101- **FEED OVERRIDE > RAPID** - When this setting is OFF, the machine will behave normally. When it is ON and HANDLE CONTROL FEED RATE is active, the jog handle will affect both the feed rate override and the rapid rate override simultaneously. That is, changing the feed rate override will cause a proportional change to the rapid rate. The maximum rapid rate will be maintained at 100% or 50% according to setting 10.
(Any Mill Control ver. 10.22 and above. Any Lathe Control ver. 4.11 and above.)

Setting 103- **CYC START / FH SAME KEY** - This is really great to use when you're carefully running through a program to test it out, when machining close to a part, fixture or chuck so that you don't accidentally bump into it. When this setting is ON, the CYCLE START button functions as the Feed Hold key as well. When CYCLE START is pressed and held in, the machine will run through the program; when it is released, the machine will stop in a Feed Hold. This gives you much better control when testing a program. When you're done using this feature, turn it Off. This setting can be changed while running a program. It cannot be ON if Setting 104 is ON. When one of them is turned ON (Setting 103 or 104), the other will automatically turn OFF.
(Any Mill Control ver. 9.06 and above. Any Lathe Control ver. 4.11 and above.)

Setting 104- **JOG HANDL TO SNGL BLK** - When running a program in MEM mode in the Program or Graphics display, you can use the SINGLE BLOCK key to cycle through your program one line at a time with each press of the CYCLE START button, when the machine is running or you're in Graphics. If you turn Setting 104 ON, and SINGLE BLOCK has been selected. You first press the CYCLE START button, then each counterclockwise click of the jog handle will step you through a program line by line. Turning the handle clockwise will cause a FEED HOLE. This setting can be changed while running a program. It cannot be ON when Setting 103 is ON. When one of them is turned ON, the other will automatically turn OFF.
(Any Mill Control ver. 9.06 and above. Any Lathe Control ver. 4.11 and above.)

Setting 114- **CONVEYOR CYCLE (MIN)** - If this is set to zero, the conveyor will operate normally. If a number is entered, it defines how long (in minutes) the cycle will be when the chip conveyor is turned on. The chip conveyor cycle is started with an M31 code or the **CHIP FWD/REV** control keys. It will stay on for the time defined in **Setting 115-CONVEYOR ON TIME (MIN)**, then turn off and not restart until the cycle time in Setting 114 has elapsed. Short programs looped with an M99 many times will not reset the chip conveyor if the intermittent feature is activated. The conveyor will continue to start and stop at the commanded times.

Setting 130- **RIG. TAP RETRACT MULT** - This feature augments one introduced in version 10.13, the quick reversal out of a G84 rigid-tapped hole. If it is set to 0 or 1, the machine behaves normally. Setting it to 2 is the equivalent of a G84 command with a J value of 2. That is, the spindle will retract twice as fast as it went in. If this setting is set to 3, the spindle will retract three times as fast. Note that specifying a J value in a G84 command for rigid tapping will override Setting 130. (Any Mill Control ver. 10.18 and above).

CONTROL TIPS

Setting 144- **FEED OVERRIDE->SPINDLE** - This is feature that is intended to keep the chip load constant when an override is applied. When this setting is OFF, the control behaves normally. When it is ON, any feed rate override that is applied will be applied to the spindle speed also, and the spindle overrides will be disabled.
(Any Mill Control ver. 11.10 and above, Any Lathe Control ver. 4.11 and above.)

Δ Send and Receive Settings to/from Disk - You can save settings to a floppy disk. *Press* **LIST PROG** *first*, then select SETNG display page. Type in a file name and then press **F2** to write that display information to disk or F3 to read a file from a disk.

GRAPH

Δ Zooming in - In Graphics display, use **F2** to zoom in on the graphic. After pressing F2, press P**AGE DOWN** to zoom in further and **PAGE UP** to expand the view. Use the cursor arrows (▲ ▼ ◀ ▶) to position the zoom window over the section of the part that you wish to view in close-up. Press **WRITE/ENTER** to save the new zoom window, and **CYCLE START** to see the close-up graphic run. Press **F2** and then **HOME** to get back to the original full table view.

PARAM

Δ Changing 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. *Modifying some of these functions will void the warranty on the machine.* If you need to change parameters, contact Haas Automation or your dealer. Parameters are protected from being changed by Setting 7. Be sure to *download and save* a copy of your machine parameters so you'll have a backup if needed (refer to the LIST PROG section to see how to save your offsets, settings and parameters to a floppy disk).

Δ 200 hour Try-Out feature - Options that normally require a magic code to activate (Rigid Tap, Macros, Floppy Enable, etc.) can now be activated and deactivated as desired by entering a 1 to turn it on, for a tryout, and a 0 to turn it off, instead of turning it on with a magic code on the parameter for that option that has a T next to it. An option activated in this manner will automatically deactivate after a total of 200 power-on hours. An option can be permanently activated by entering the magic code, (contact your dealer for that option) as before. Note that the letter T will be displayed to the right of the option in the parameter display during the 200 hour period indicating that it can be activated by entering a 1, and deactivated by entering an 0. If deactivated the T is cleared from the screen. The T is likewise cleared if the feature is enabled with a magic code. Note that the deactivation only occurs when power to the machine is turned off, not while it is running, and the following message will be displayed when power is turned on and a feature that was being tried out has been automatically deactivated.

> One or more of your tryout features have expired. They are now disabled.
> Press 'CANCEL' to continue.

(Any Mill control ver. 11.02 and above; any Lathe control ver. 4.05 and above.)

Δ Send and Receive Parameters to/from Disk or RS-232- (See the Control Tips Communications section.)

DGNOS

This display is used to show the status of the machine for diagnostics and servicing.

Δ Rigid Tapping Spindle Speed - If you have the rigid tapping option, you can verify the exact spindle speed by checking the second page of Diagnostic data. You can also verify the exact spindle speed by checking the Curnt Comds "Act" display.

CONTROL TIPS

ALARM

Δ Alarm History Display - There is an alarm history that displays the previous 500 alarms (100 alarms on older machines). Pressing the right cursor arrow (▶) in the Alarm display will list the last 500 alarms, with a date and time. You then need to cursor arrow up (▲) to see the alarms previous to the last one. Pressing either the left or right arrow again will bring you back to the normal alarm display. On New controls, alarm history will also list any parameter changes and offset changes.

Δ Alarm History saved to RS-232 and Disk - This is a new feature. From the alarms history screen, the user can now save the alarm history (the last 100 alarms) to a floppy disk file by entering a file name and pressing F2. Alternately, the alarm history can be sent to a PC using RS-232 by pressing SEND RS232. The output from either method will contain a percent sign (%) on the first and last lines.
(Any Mill control ver. 10.22 and above; any Lathe control ver. 4.02 and above)

MESGS

Δ Leaving Messages - You can enter a message in the MESGS display for the next person, or for yourself. It will be the first display shown when you power up the machine, *if there are no alarms* other than the usual 102 SERVOS OFF alarm. If the machine was powered down using **EMERGENCY STOP**, the MESGS display will not show up when you turn the machine on again. Instead, the control will display the active alarm generated by the emergency stop. In this case, you would have to press the **ALARM/MESGS** key to view a message.

EDIT

Δ The Edit Display - When you press **EDIT**, the first display you see is the Advanced Editor display. This display has menus in the upper left that allow the user to access features of the Advanced Editor. The menus are entered into by pressing **F1**. Pressing the **PRGRM/CONVRS** key while in Edit will bring up and display the program only, to edit. Pressing **PRGRM/CONVRS** a second time will bring up the Quick Code editor, and again will display Visual Quick Code. Quick Code can also be accessed from the Advanced Editor within the F1: HELP pull-down menu.

Δ To Undo a Simple Edit - Pressing **UNDO** will change back as many as the last ten simple edits (INSERT, ALTER, DELETE) that were done. Sometimes you can even edit some code, run the program, and then if you need to change it back after running, it may let you do that using the **UNDO** key. **UNDO** *does not undo program edits done in Block Edit*. When doing Block Edit, UNDO will only deselects text.

Δ Advanced Editor Menus - Pressing **F1** in the Advanced Editor will activate the menus. The menu selections are made with the jog handle, turning it either clockwise or counterclockwise, or with the cursor arrow keys (▲ ▼ ◀ ▶). Press the **WRITE/ENTER** key to activate the cursor-selected menu item.

Δ Advanced Editor On-line Help - In the Advanced Editor, after pressing **F1** to access the menus, on-line Help is displayed in the lower right corner of the screen. To scroll through the Help text, use the **PAGE UP**, **PAGE DOWN**, **HOME**, and **END** keys (remember, the cursor arrows move you through the menu items, not the Help text). In addition, if the **F1** key is pressed during the use of a menu option, the Help text is likewise displayed. Pressing **F1** again will exit the Help display.
(Any Mill Control ver. 9.32 and above; any Lathe Control ver. 2.16 and above.)

Δ Advanced Editor Searching - When the SEARCH menu FIND TEXT item is used and the text is found, the next press of **F1** to activate the menus will automatically select the FIND AGAIN option. Likewise, when the SELECT TEXT function on the EDIT menu is used, the next activation of the menus will cause the COPY SELECTED TEXT option to be highlighted.

CONTROL TIPS

Δ Advanced Editor Block Editing - In the 80 column Advanced Editor you can select a program block and copy it to another location, move it to another location, or delete it. To start the block definition, press **F1** to get into the menus, use the jog handle or the cursor arrows to select to select the EDIT menu and the SELECT TEXT menu item, then press **WRITE/ENTER**. Another way to begin text selection is to put the cursor on the program line where you want the selection to begin and press **F2**. In either case, once you've defined the beginning, you then use the cursor arrows to go to the line where the selection should end, and press **F2** or **WRITE/ENTER**. This will highlight the section you want to copy, move, or delete. Then, use the EDIT menu (or **EDIT** hot keys), to COPY SELECTED TEXT (or press **INSERT** to copy), MOVE SELECTED TEXT (or press **ALTER** to move), or DELETE SELECTED TEXT (or press **DELETE**) for the selected block.

Δ Advanced Editor Block Editing to Another Program - You can copy a line or a block of lines from one program into another. Select the program block you wish to transfer to another program using the method outlined above (in the "in the Advanced Editor Block Editing" paragraph). Then press **SELECT PROG** (or use the PROGRAM menu), scroll to the program you want to copy to, and press **WRITE/ENTER** to select it. It will open up on the right side of the screen. Cursor to where you want the selected text to be placed and press **INSERT**. Use the **EDIT** key to go back and forth between two open programs on the screen (to deselect text after it's been copied, press **UNDO**).

Δ Larger Display Editor Block Editing Display - You can copy, move, or delete a block of lines in the same program, or move a block in a program from one program to another in the larger Editor display (press **EDIT** and then **PRGRM/CONVRS** once to get to the larger editor, and you will see F1-BEGBLK and F2-ENDBLK, at the bottom of display). In this Editor, define the first block by pressing **F1**, and then cursor to the last line you want to select and press **F2**. This will highlight that section of the program to copy, move, or delete in a program. You will see the commands at the bottom INS-COPY ALT-MOVE DEL-DELETE to perform on the selected block. Or you can copy into another program using LIST PROG and copy the selection into the new program by pressing INSERT.

Δ Exiting Block Edit - You can turn off Block Edit highlighting by pressing the **UNDO** key; the cursor will remain where you're at in the program. **UNDO** *will not* change back an edit done in Block Edit. **RESET** will also turn off the block highlight, but the cursor will go back to the beginning of the program.

Δ Advanced Editor Quick Cursor Arrow - You can call up a cursor arrow with which to scroll through your program quickly, line by line, when you're in the Advanced Editor. For the quick cursor arrow, press **F2** once; then you can use the jog handle to scroll line by line through the program. To get out of this quick-cursor mode and remain where you are in the program, just press the **UNDO** key.
(Any Mill Control ver. 9.49 and above; any Lathe Control ver. 2.24 and above.)

Δ Advanced Editor - The 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 a 80 column display, and includes pull-down menus that allow the user to access the features of the editor. Additionally, context-sensitive help function is available to provide information on all of the editors features.

Δ Editing in Two Locations of the Same Program - In Edit mode, **F4** is the hot key that displays another version of the active program for editing. The same program will be displayed on both halves of the screen, and each program can be edited alternately by using the **EDIT** key to switch from one side to the other. Both programs will be updated with the edits done while you're switching back and forth. This is useful for editing a long program; you can view and edit one section of the program on one side of the screen and another section on the other side.

CONTROL TIPS

Δ Lower-Case Text - In the Editor, you can enter lowercase text if it's between parentheses (that is, for comments only). Press the **SHIFT** key first (or hold it in) and then the letter you want in lower case (remember, this works for parenthetical comments only). When lowercase text is selected (highlighted), it will appear in caps; deselected, it returns to lower case. To type the white symbol in the upper left corner of the numeric keys, press **SHIFT** and then the key. These symbols are used for parenthetical comments or for macros.

MEM

Δ Run-Stop-Jog-Continue (RSJC) - This feature allows the operator to interrupt program execution, jog away from the part to perform a desired task, and then return to the interruption point and resume program execution. Once RSJC is initiated, the operator is able to stop and start the spindle, jog the XYZ axes individually (axes other than X, Y, and Z cannot be jogged), or command a tool release. The following describes the RSJC procedure. *(Any Mill Control ver. 11.10 and above.)*

1) While a program is running, press **FEED HOLD**. This will stop all motion (after any canned cycle in process has been completed.)
2) Press **X**, **Y** or **Z** followed by the **HANDLE JOG** key. The control will store the current X, Y or Z position. Axes other than X, Y, and Z cannot be jogged.
3) At this point, the control will display the message JOG AWAY, and will tick once each second or so. The operator can use the jog handle, remote jog handle, the **HANDLE JOG** increment buttons (.0001/.1, .001/1., .01/10., .1/100.) or the JOG LOCK buttons to move the tool away from the part. Now you can use the COOLNT key to cycle the coolant, and CW, CCW, STOP to operate the spindle. You can also use the TOOL RELEASE button, and turn Through-Spindle Coolant (TSC) on and off using the AUX CLNT key. Note that using AUX CLNT requires that the spindle be rotating and that the door be closed. At this point tools can be swapped out and the associated length and diameter offsets adjusted. However, when the program is continued, the old offsets will still be used for the return position and any motion commands already in the queue. It is therefore unsafe to swap out tools and adjust offsets when the program is interrupted during a cut.
4) When you're ready to continue, jog to a position as close as possible to the stored position, or to a point where there will be an unobstructed rapid path back to the stored position.
5) Return to the previous mode by pressing **MEM**, **MDI** or **DNC**. The control will only continue normally if the mode that was in effect at the time of the interrupt is reentered.
6) Press **CYCLE START**. The control will display the message JOG RETURN and rapid X and Y at 5% to the position where FEED HOLD was pressed; then it will do the same for Z. The rapid rate override keys have no effect during JOG RETURN. Note that the control will not follow the path the operator used to jog away. Instead, it will perform simple moves without regard for obstacles. Therefore, crash is possible. If FEED HOLD is pressed during this motion, the control will go into a feed hold state and display the message JOG RETURN HOLD. Pressing **CYCLE START** will cause the control to resume the JOG RETURN motion. When the motion is completed, the control will again go into a feed hold state.

Δ A Block Delete - Takes effect four lines after that key is pressed when cutter compensation is in use, or two lines later when cutter compensation is not in use.

Δ Block Look-Ahead - This control actually does look ahead for block interpretation, up to 20 blocks. This is not needed for high-speed operation. It is instead used to ensure that DNC program input is never starved, and to allow Cutter Compensation to have non-XY moves inserted while Cutter Compensation is On.

MDI

Δ Transferring an MDI Program - You can transfer and save a program in MDI to your list of programs. When in the MDI display, *make sure that the cursor is at the beginning* of the MDI program. Enter a program number (Onnnnn) that's not being used. Then press **ALTER** and this will transfer the MDI data into your list of programs under that program number.

CONTROL TIPS

HAND JOG

Δ Jog Keys - The **JOG** keys (+X, -X, +Y, -Y, +Z, -Z, +A, -A, +B, -B) use the jog speeds of 100., 10., 1. and .1 inches per minute listed next to the **HANDLE JOG** key (jogging with the handwheel uses the .1, .01, .001 and .0001 inch increments). You can also adjust feed rate using the **FEED RATE OVERRIDE** buttons, which allow you to increase or decrease feed rate in 10% increments, up to 200% or using the **HANDLE CONTROL FEED RATE** or **HANDLE CONTROL SPINDLE** keys to adjust the programmed feed or speed 1% up or down with every increment of the Handle.

Δ Jog Keys - You can also select an axis for jogging by entering the axis letter on the input line and then pressing the **HANDLE JOG** button. This works for the X, Y, Z, and A axes as well as the B, C, U, and V auxiliary axes.

ZERO RET

Δ To Rapid an Axis Home - You can rapid *all* axes to machine zero by pressing the **HOME G28** key. You can also send just one axis (X, Y, Z, A or B) to machine zero in rapid motion. Enter the letter X, Y, Z, A or B, then press **HOME G28** and that axis alone will rapid home. ***CAUTION***! There is no warning to alert you of any possible collision! For example, if the Z axis is down near the part or fixture on the table, and then the X or Y axis is sent home using HOME G28, a crash may result. Care must be exercised.
(Any Mill Control ver. 9.49 and above; any Lathe Control ver. 2.24 and above.)

LIST PROG

Δ Duplicating a Program in LIST PROG - In the LIST PROG mode, you can duplicate an existing program by cursor-selecting the program number you wish to duplicate, typing in a new program number (**O**nnnnn), and then pressing **F1**. You can also go to the Advanced Editor menu to duplicate a program, using the PROGRAM menu and the DUPLICATE ACTIVE PROGRAM item.

PROGRAMMING

Δ Program Format at the Beginning and End - Programs written on a PC and sent to the control from a floppy disk or through the RS-232 port must start and end with a % sign, on a line by itself. The second line in a program received via floppy or RS-232 (which will be the first line the operator sees) must be **O**nnnnn, a six-character program number that starts with the letter **O** followed by five digits. When you create a program on the Haas control the percent (%) signs will be entered automatically, though you won't see them displayed.

Δ Program Format with M06 - It is not necessary to turn off the coolant (M09), stop the spindle (M05), or move the Z axis home (G28) prior to a tool change. The control will do these tasks for you during a tool change M06 command. However, you may decide to program these commands anyway for convenience and timely execution of a tool change sequence. If you're using Single Block to step through a program you will be able to see the commands when you stop on that line.

Δ M19 (Orient Spindle) with a P Value - This feature works on any vector drive mill. Previously, the M19 command would simply orient the spindle to only one position — that suitable for a tool change. Now, a P value can be added that will cause the spindle to be oriented to a particular position (in degrees). If a whole number is used for the P value, no decimal point is needed. For example, M19 P270 will orient the spindle to 270 degrees. Note that P270.001 (or any other fraction) will be truncated to P270, and P365 will be treated as P5.
(Any Mill Control ver. 9.49 and above. Any Lathe Control ver. 2.21 and above.)

Δ M19 (Orient Spindle) with a Fractional R Value - This feature works on any vector drive mill. An M19 R123.4567 command will position the spindle to the angle specified by the R fractional value; up to 4 decimal places will be recognized. This R command now *needs a decimal point*: if you program M19 R60, the spindle will orient to 0.060 of a degree. Previously, R commands were not used for this purpose and only integer P values could be used.
(Any Mill Control ver. 9.49 and above; any Lathe Control ver. 2.29 and above.)

CONTROL TIPS

Δ G150 Pocket Milling with 40 Moves - In the G150 command line, it calls up a subprogram with a P command (P1234) which is calling up a separate program (O1234) that defines in it, the geometry of a pocket. This pocket geometry must be defined in 40 moves (strokes) or less. In software ver. 11.11, the G150 Pocket Milling was increased to 40 strokes. In all previous versions, the control could only accommodate a G150 Pocket Milling subprogram with 20 moves or less.
(Any Mill Control ver. 11.11 and above)

Δ G84 or G74 Tapping - When tapping, you don't need to start the spindle with an M03 or M04 command. The control starts the spindle for you automatically with each G84 or G74 cycle, and it will in fact be faster if you don't turn on the spindle with an M03 or M04. The control stops the spindle and turns it back on again in the G84 or G74 tapping cycle to get the feed and speed in sync. The operator just needs to define the spindle speed.

Δ G84 Rigid Tapping Cycle Quick Reverse - This feature for rigid tapping has the spindle back out faster than it went into a tapped hole. The way to specify this is with a J code on the G84 command line. J2 retracts twice as fast as the entry motion; J3 retracts three times as fast, and so on, up to J9. A J code of zero will be ignored. If a J code less than 0 or greater than 9 is specified, Alarm 306 — "INVALID I, J, K or Q" — is generated. The J code is not modal and must be specified in each block where this effect is wanted. The J value should not contain a decimal point.
(Any Mill Control ver. 10.13 and above.)

Δ G84 or G74 Tapping Back into a Tapped Hole - You can return a tap back into a hole to go deeper, that has already been tapped, if you have the Rigid Tap option ON. If "you have not" moved the tool or part, you are able to return back into a part with tap, if your Parameter 57 bit REPT RIG TAP is on with a 1. If this parameter is on with 1, you edit the Z-depth to go deeper, or offset down the amount of one pitch to rerun a tapped hole. WARNING: If you move, offset, or change the starting position of the part or tap that is not equal to one pitch of the thread, you will cross-thread hole.

Δ G84 or G74 Peck Tapping - You can peck tap into a hole to go deeper for tough/hard material, if you have the Repeat Rigid Tap option on your machine. You're able to peck tap back into a hole, if Setting 133 is ON for newer machines or Parameter 57, REPT RIG TAP is on with a 1 for older machines. Then all you would need to do is repeat the tapping cycle at the same XY location and going deeper in the Z axis on each command line to peck deeper into the part.

Example 1:	Example 2:	Example 3:	Example 4:
G90 G54 X1.5 Y-.5	G90 G54 X1.5 Y-.5	G90 G54 X1.5 Y-.5	G90 G54 X1.5 Y-.5
S450	S450	S450	S450
G43 H01 Z1.0 M08	G43 H01 Z1.0 M08	G43 H01 Z1.0 M08	G43 H01 Z1.0 M08
G84 G99 Z-.1 R.1 F22.5	G84 G99 Z-.1 R.1 F22.5	G84 G99 Z-.1 R.1 F22.5	G84 G99 Z-.1 R.1 P22.5
G84 G99 Z-.15 R.1 F22.5	X1.5 Y-.5 Z-.15	X1.5 Z-.15	G91 X0. Z-.25
G84 G99 Z-.2 R.1 F22.5	X1.5 Y-.5 Z-.2	X1.5 Z-.2	X0. Z-.3
G84 G99 Z-.25 R.1 F22.5	X1.5 Y-.5 Z-.25	X1.5 Z-.25	X0. Z-.35
G84 G99 Z-.3 R.1 F22.5	X1.5 Y-.5 Z-.3	X1.5 Z-.3	X0. Z-.4
G80 G00 Z1. M09	G80 G00 Z1. M09	G80 G00 Z1. M09	G80 G90 G00 Z1. M09

Note: On Mill software 12.09 and above, REPT RIG TAP was moved from the parameters to setting 133. This is now an ON/OFF setting that is now easier to changed by the user.

Δ Tool Change Command - It is not necessary to turn off the coolant, stop the spindle, or move the Z axis prior to a tool change (M06). The control will do this for you. Though you may choose to program in these commands sooner, then when they actually happen in an M06.

CONTROL TIPS

Δ Quick Code Programming - One of the best features of Quick Code, is that you can customize it to the way you like to see your programs formatted. It combines the simplicity and flexibility of G code programming with English descriptive sentences to enable even beginning programmers to construct most two dimensional parts. Experienced programmers will enjoy the speed they can now enter programs in the way they like them to be formatted. Everybody programs a little differently and may have special preferences, such as, do you put the "T" command on the same line as the tool change "M06" command or before. With Quick Code you can edit the source program so that any English Quick Code text you customize can be matched with any G code that you desire to be with that selection, and have your own query's and default values.

COMMUNICATIONS

Δ Program Format to Receive - You can receive program files from a floppy disk or the RS-232 port on the Haas control. A program must begin and end with a % sign on a line, with nothing else on that line. The next line must begin with the letter "O" and then the program number (newer machines have five digits, older machines have four digits). If you'd like to enter a program name in text after a program number. Enter it between parenthesis (Program Name), on the same line after the program number, or in the next line. The program text name will show up with the program number in the list of programs.

Δ I/O Edit Menu Receiving Program Files on a Disk - You can load program files from a program disc using the I/O menu and the DISK DIRECTORY item of the Advanced Editor. Pressing **WRITE/ENTER** when this menu item is selected will display a list of the programs on the program disk. Use the cursor arrow keys or the handwheel to select the file you need to load, and press **WRITE/ENTER**. After loading that file, the disk directory will remain on display to allow more files to be selected and loaded into the control. **RESET** or **UNDO** will exit this display.

Δ I/O Edit Menu SEND RS232 or SEND DISK Commands - You can send programs to the RS232 port or a program disk from the Advanced Editor. After selecting the menu item you want (SEND RS232 or SEND DISK), a program list will appear. Select the program you want to save, or “ALL” (at the end of the list) if you wish to send all programs under one file name. You can also select any number of programs using the up and down cursor arrow keys or the handwheel and the **INSERT** key to mark the specific programs to send. If no programs are selected from the list using the **INSERT** key, the currently highlighted program will be sent.

Δ I/O Edit Menu Sending Multiple Programs Under One File Name - In the Advanced Editor, you can send multiple program files via the RS232 port or to a program disk, using the SEND RS232 or SEND DISK commands under the I/O menu (see the previous paragraph for how to do this). The Advanced Editor allows you to choose more then one program in the list of programs that you want to save (you select them with the cursor up or down arrow and press **INSERT**), and save them under one file name that you type in; then press **WRITE/ENTER** to save and send it. *(Any Mill Control ver. 9.49 and above; any Lathe Control ver. 3.00 and above.)*

Δ I/O Edit Menu Sending Multiple Programs Using Program Numbers - The SEND DISK item from the I/O menu of the Advanced Editor allows the operator to select one or more programs to be saved to the program disk. It will prompt you to “ENTER DISK FILENAME.” In previous versions, the control would insist on a filename. Now, however, if you do not enter a file name, but simply press **WRITE/ENTER**, the control will save each program (the ones you selected using the cursor and the **INSERT** key) to a separate file on the disk and use the five-digit program number as the file name. For example, if programs O00123 and O45678 are selected and sent, the new file names that are created will be O00123 and O45678.
(Any Mill Control ver. 9.49 and above; any Lathe Control ver. 3.00 and above.)

Δ LIST PROG Display Sending a Program File - You can send a file or files to a program disk or through the RS-232 port from the LIST PROG display. Use the cursor arrow to select the program you want, or on “ALL” if you want to send all of the programs under one file name. When you press **F2** to send the selected program(s), the control will ask for a file name, which can be up to eight characters long with a three-letter extension (8CHRCTRS.3XT). Then press **F2** again to send it. You can also use the I/O menu in Edit to send and receive program files.

CONTROL TIPS

Δ LIST PROG Display Sending Multiple Program Files with SEND RS232 key - Several programs can be sent through the serial RS-232 port from the LIST PROG operation mode, by typing in all the program numbers together on the input line without spaces, starting with the letter "O" and then the number. You can leave off any leading zero's if you choose to (e.g., O123O4545O13579) and then press SEND RS232.

Δ RS-232 Communications Using X-Modem - If you are seeing occasional errors when using RS-232 communications, X-Modem (Setting 14) is a standard communications mode which is very reliable when only a few errors occur. Our control supports this, as do almost all software communication packages for PCs.

Δ Haas Rotary Table Using the Serial Port and Macros - It is possible to regulate a Haas rotary table using the serial port and macros from the Haas control, or *any* Fanuc-compatible control. There is a set of sample macros available from the Haas applications department.

Δ Send and Receive Offsets, Settings, Parameters and Macro Variables to/from Disk - You can save offsets, settings, and parameters to a floppy disk. *Press* **LIST PROG** *first*, then select an OFSET, SETNG or PARAM display page. Type in a file name and then press **F2** to write that display information to disk (or F3 to read that file from a disk). You can also do this with the macro variables by pressing **LIST PROG** *first*, then selecting the macro variable display page (**PAGE DOWN** from CURNT COMDS).

Δ Send and Receive Offsets, Settings, Parameters and Macro Variables to/from RS232 - You can also save offsets, settings, and parameters via the RS-232 port. *Press* **LIST PROG** *first*, and then select an OFSET, SETNG, or PARAM display page. Press **SEND RS232** to send that display page to the RS-232 port under the file name that you enter. Press **RECV RS232** to read the file via RS-232. You can also do this with the macro variables by pressing **LIST PROG** *first*, then selecting the macro variable display page (**PAGE DOWN** from CURNT COMDS).

Δ Deleting a Program File from a Floppy Disk - Haas machines allow you to delete files from a floppy disk. (Note that this requires the latest floppy driver EPROM chip version FV 2.11.) Go to the **LIST PROG** display page and type "**DEL <filename>**" where <filename> is, naturally, the name of the floppy disk file you want to delete. Press **WRITE/ENTER** to delete the file. The message "FLOPPY DELETE" will appear, and the file will be deleted from the floppy disk. If you need to see the list of file names on the floppy, press F4 and then, when the "DISK DONE" message is displayed, press EDIT or MEM.
(Haas mill control software version 9.63 and above; Haas lathe control ver. 3.00 and above.)

CONTROL TIPS

ADVANCED EDITOR SHORTCUTS

In the Advanced Editor display, pressing these keys will quickly get you to these menu items without having to press the F1 key and cursor to that menu item.

HOT KEYS	DESCRIPTION OF HOT KEY
SELECT PROG	Pressing **SELECT PROG**, while in edit, will ***Bring up the Program List*** thats in memory on other side of the edit display so as to select a program from list and display it.
F2	Pressing **F2** on a line in a program, will begin ***Selecting a Block in a Program*** on the first line of a block to be edited. Scroll down to the last line to be defined in the block, and press **F2** or WRITE. The selected block of text will then be highlighted. **You cannot select a block with the "Onnnnn" program number.**
EDIT	The **EDIT** key can be used to ***Switch to the Left or Right Side*** between two programs that have been selected and are displayed to edit.
INSERT	**INSERT** can be used to ***Copy Selected Text*** in a program to the line after where you place the cursor arrow pointer.
ALTER	**ALTER** can be used to ***Move Selected Text*** in a program to the line after where you place the cursor arrow pointer.
DELETE	**DELETE** can be used to ***Delete Selected Text*** in a program.
UNDO	**UNDO** will simply ***exit a block definition***. It will NOT change back anything done in block edit.
F4	Pressing **F4** will open up ***Another Version of the Same Program*** on the other side of the Advanced Editor. The user can edit two different locations on the same program. Pressing EDIT will switch back-and-forth which will update between the two programs. If you enter a program number (Onnnnn) and then press F4, or the arrow down key, that program will be brought up on the other side of the Advanced Editor.
SEND RS232	Pressing the **SEND RS-232** key will activate that I/O menu selection.
RECV RS232	Pressing **RECV RS-232** key will activate that I/O menu selection.
ERASE PROG	Pressing the **ERASE PROG** key will activate that I/O menu selection. This will bring up program list on the inactive side of edit display for you to cursor to a program and delete it.