Haas Factory Outlet

A Division of Productivity Inc

CNC Lathe Series
Training Manual

Haas TL Series
Tool Room Lathe Operator

Revised 06-2014

This Manual is the Property of Productivity Inc

The document may not be reproduced without the express written permission of Productivity Inc.

The content must not be altered, nor may the Productivity Inc name be removed from the materials.

This material is to be used as a guide to the subject content. The Operator is responsible for following Safety Procedures as outlined by their instructor or manufacturer's specifications.

Downloading and/or other use of this manual does not indicate completion of the Training course. This manual is for reference only.

To obtain permission, please contact trainingmn@productivity.com.

Tool Room Lathe Operator Training Manual
Table of Contents

Productivity Inc®

For more information on Additional Training Opportunities or our Classroom Schedule,
Contact the Productivity Inc Applications Department in Minneapolis:
☏ 763.476.8600

Visit us on the Web: www.productivity.com

Click on the Training Registration Button

✉ trainingmn@productivity.com

Introduction to Basic TL Series Lathe Operation

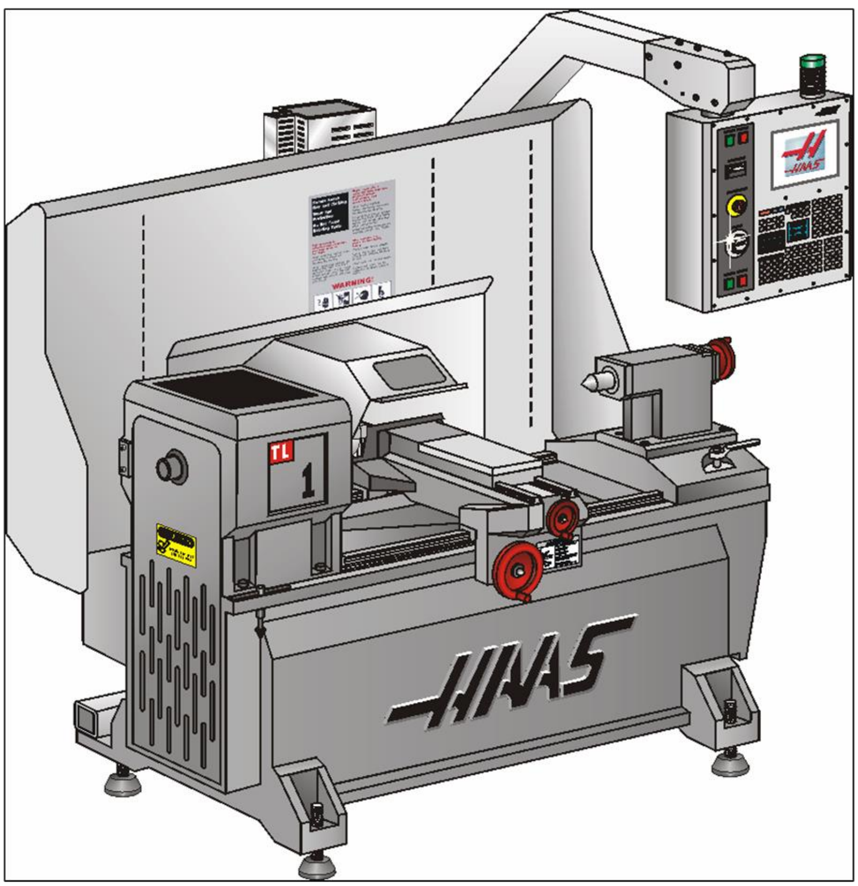
Welcome to Productivity, Inc., the Haas CNC Machine Tool Distributor for your local area. As part of your company's Haas CNC purchase, standard lifetime training is included as long as your company owns the machine.

What we plan on covering in this one-day class is the operation and programming of the unique features of the Haas TL- Series Tool Room Lathes.

These lathes are unique in their own way as they are designed for manual, semi-manual, and full CNC G&M code operation. Even though the TL series can be run from a G&M code program, Haas has equipped these unique machines with a unique control. The Haas IPS (Intuitive Programming System) allows for quick and easy setup and programming of standard tool room style parts.

Since the TL series is so unique, Productivity, Inc. had designed a specific class just for the TL series to best suit its unique features.

If you or your company would like to learn more about G&M code programming to take even more advantage of the Haas control equipped on the TL series of lathes, we would suggest the next step - Lathe Programming after completing this course.



Revised 060314-CK

The Cartesian Coordinate System

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

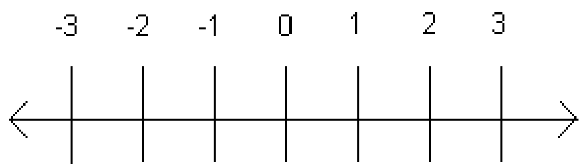


Fig-1 - 1 Horizontal number line – Z Axis

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.

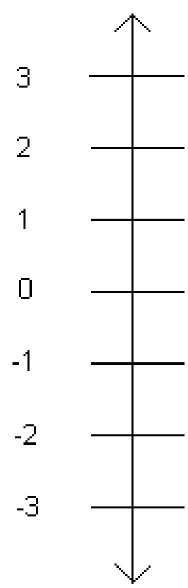


Fig.1-2 Vertical Number Line – X Axis

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

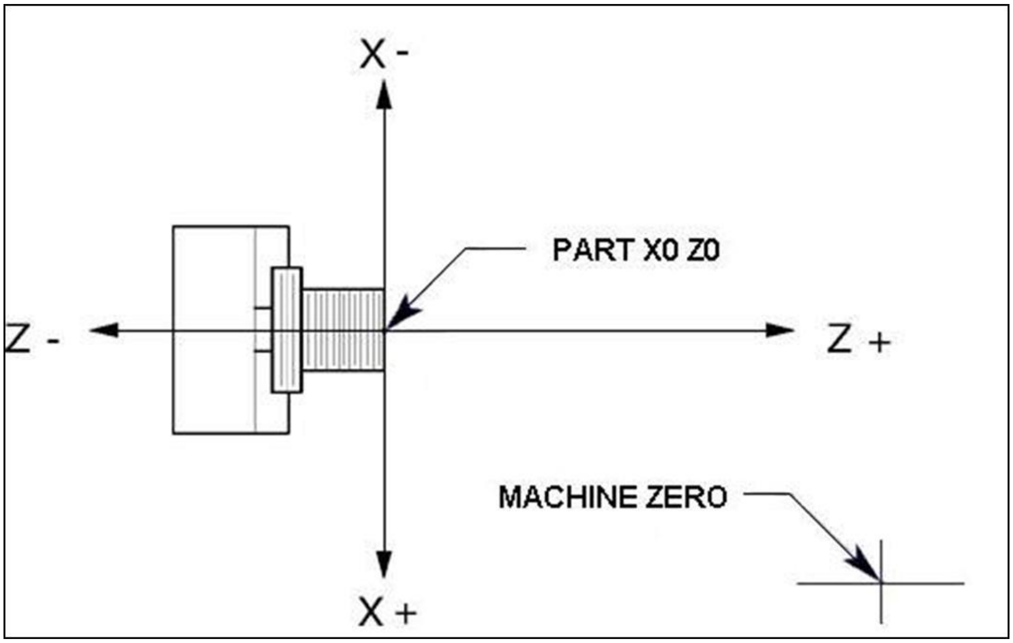


Figure 1-3

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

The zero position may be placed at any point along each of the two number lines, and in fact will probably be different for each setup of the machine. It is noteworthy to mention that the X-axis is set with the machine zero position on the center line of the spindle, while the Z axis zero is set at the finished right of the part being machined. This will place the entire X axis cutting in a positive range of travel, whereas the Z axis cutting will be in the negative range of travel.

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

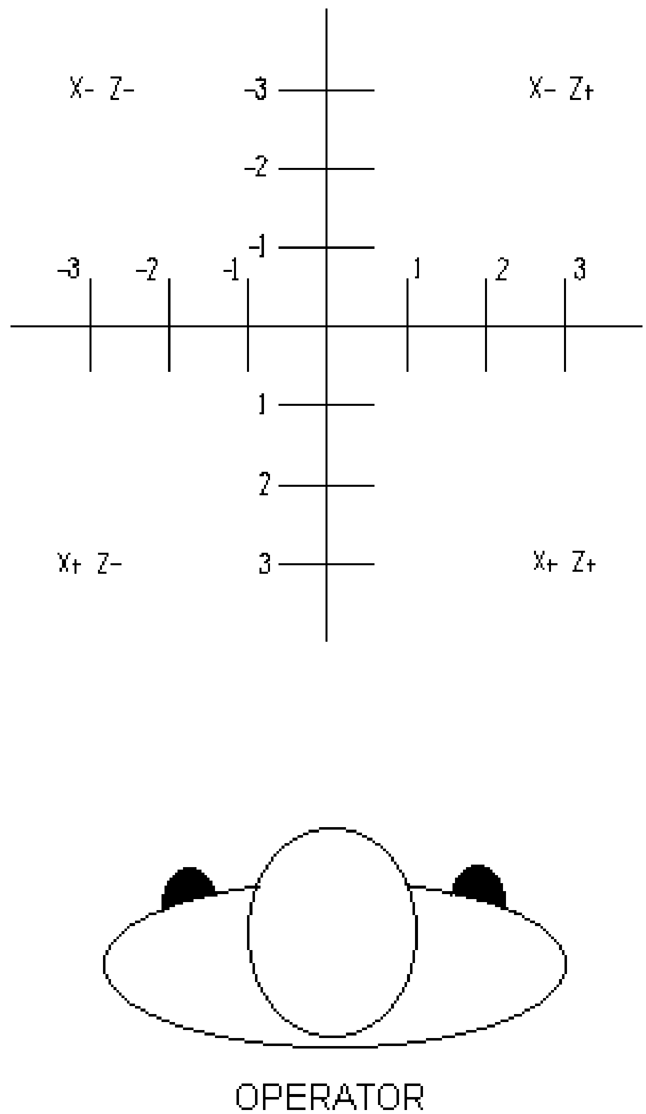


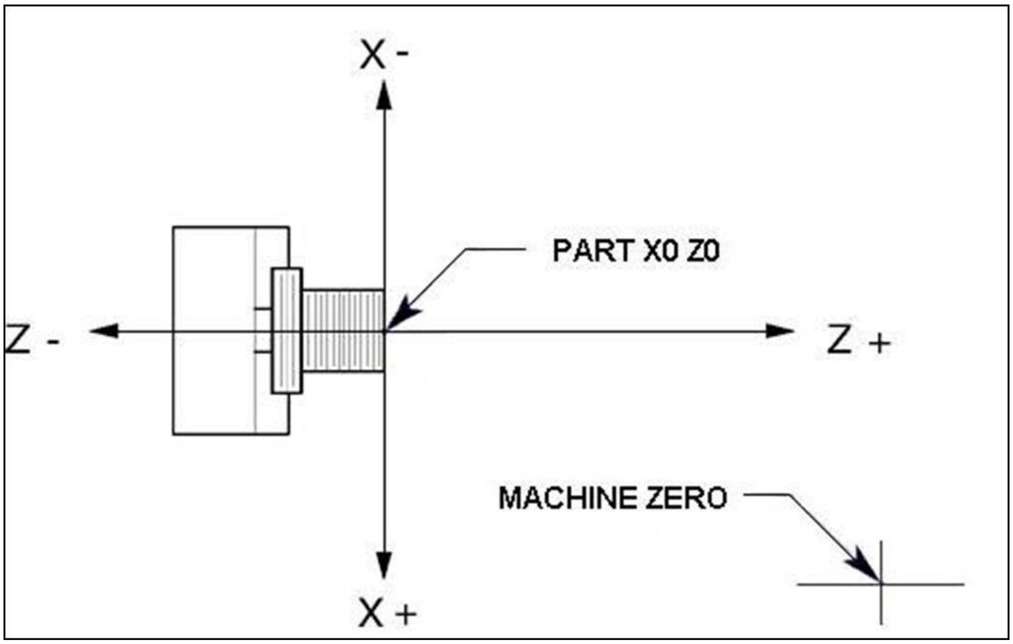
Fig. 1-4: Operator's working grid.

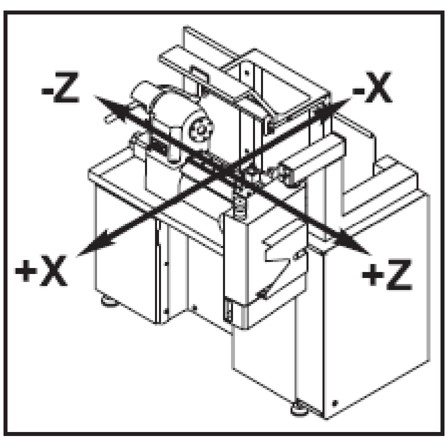
Whenever we set a zero somewhere on the X axis and somewhere on the Z axis, we have automatically caused an intersection of the two lines. This intersection where the two zeros come together will automatically have the four quadrants to its sides, above, and below it. How much of a quadrant we will be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

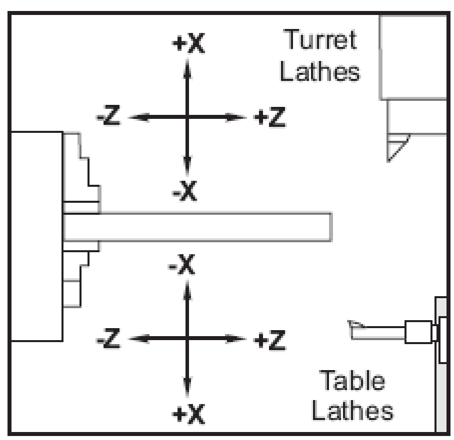
Machine Home Position

The principal of machine home 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 axes are moved to the furthest positive direction until the limit switch is reached. When this condition is satisfied, the only way to move any of the two axes is in the negative direction. This is because a new zero was set for each of the axes automatically when the machine was brought Home. Machine home is placed at the edge of each axes travel.

Sometimes this point is referred to as "Machine Zero" as pointed out below:



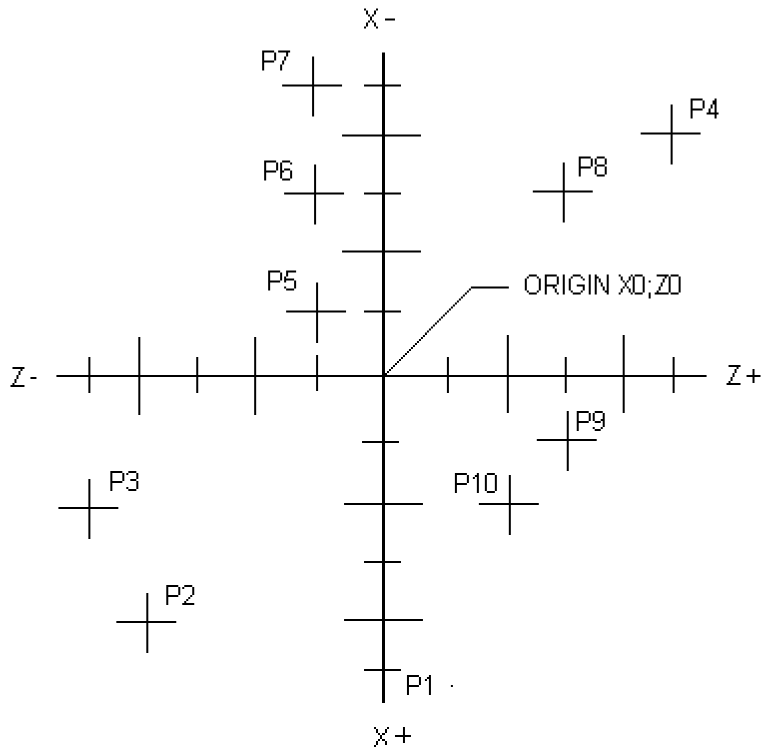




Note the difference in the x coordinate system on a turret lathe and a table lathe. Positive X is in a direction that points toward the operator on the table lathe. This is due to the fact that the tool is on the opposite side of the part compared to a turret lathe.

Cartesian Coordinate Exercise

THE CARTESIAN COORDINATE SYSTEM

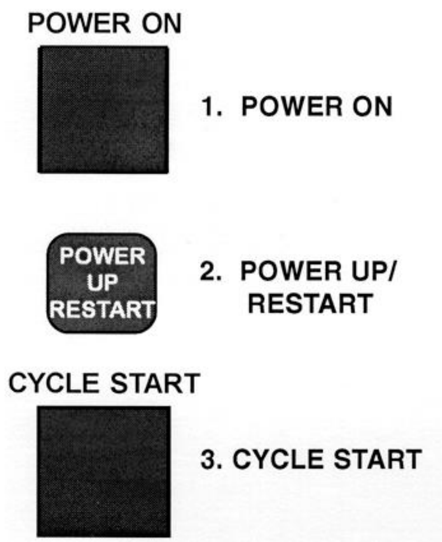


POINT #	Z Position	X Position
P1	Z 0.05.0	X -2.5
P2		
P3		
P4		
P5		
P6		
P7		
P8		
P9		
P10		

The Haas CNC Control

Powering On the Machine

To power up a Haas machine, regardless of where the machine turret was when it was turned off, press **POWER ON**. The machine must first find its fixed machine zero reference point before any operations can occur. After it's powered on, pressing **POWER UP/RESTART** will send the machine to its machine zero reference location. The machine doors must be cycled and closed to return to machine zero. Also the machine needs to see the Emergency Stop cycled. Haas provides directions on the screen on what needs to be done to start the machine up in the morning.



When powering on the machine, if there is a message in the **MESGS** display, it will be the first display seen on your control screen.

Will move all axis to machine zero and then indexes the turret to tool #! Machine will move up in X first to machine zero and then the Z move to machine zero.

If the correct program has been selected and the part program is proven to be good and it's ready to run, press cycle start.

General Machine Keys

Power On - Turns CNC machine on.

Power Off - Turns CNC machine tool off.

Emergency Stop - Stops all axis motion, stops spindle, tool changer and turns off coolant pump.

Jog Handle – Jogs axis selected, also may be used to scroll through programs, menu items while editing and also altering feeds and speeds.

Cycle Start – Starts program in run mode or graphics mode.

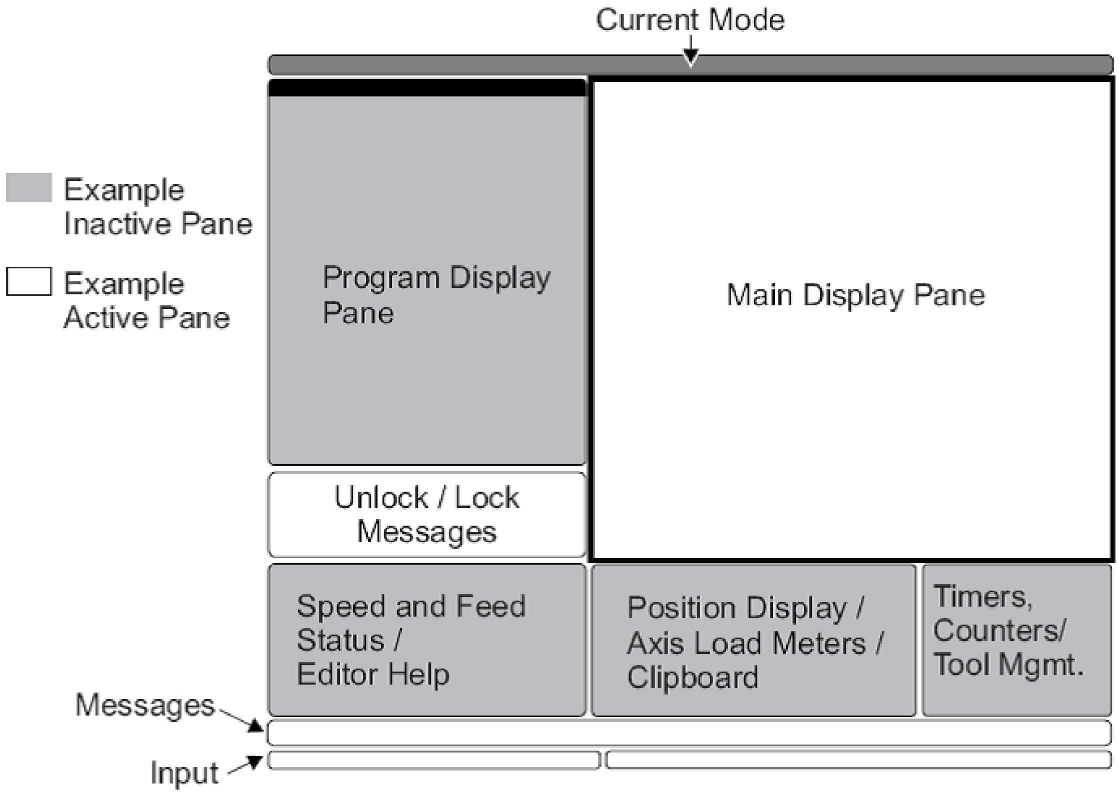
Feed Hold – Stops all axis motion. Spindle will continue to turn**.**

Reset – Stops machine, will rewind program.

Power Up/Restart – Axis will return to machine zero and tool change will occur per Setting 81

Recover – If a tool change is stopped in middle of a cycle an alarm will come up. Push the **Recover** button and follow the instructions to bring the tool change cycle to the beginning.

Control Display



The new 16 software has a larger display and more panes than older versions. Above is the basic display layout. What is displayed depends on which display keys have been used. The only pane active is the one with the white background. Only when a pane is active may changes be made to data.

Control functions in Haas machine tools are organized in three modes: **Setup, Edit** and **Operation.**

Access Modes using the mode keys as follows:

Setup: ZERO RET, HAND JOG keys. Provides all control features for machine setup.

Edit: EDIT, MDI/DNC, LIST PROG keys. Provides all program editing, management, and transfer functions.

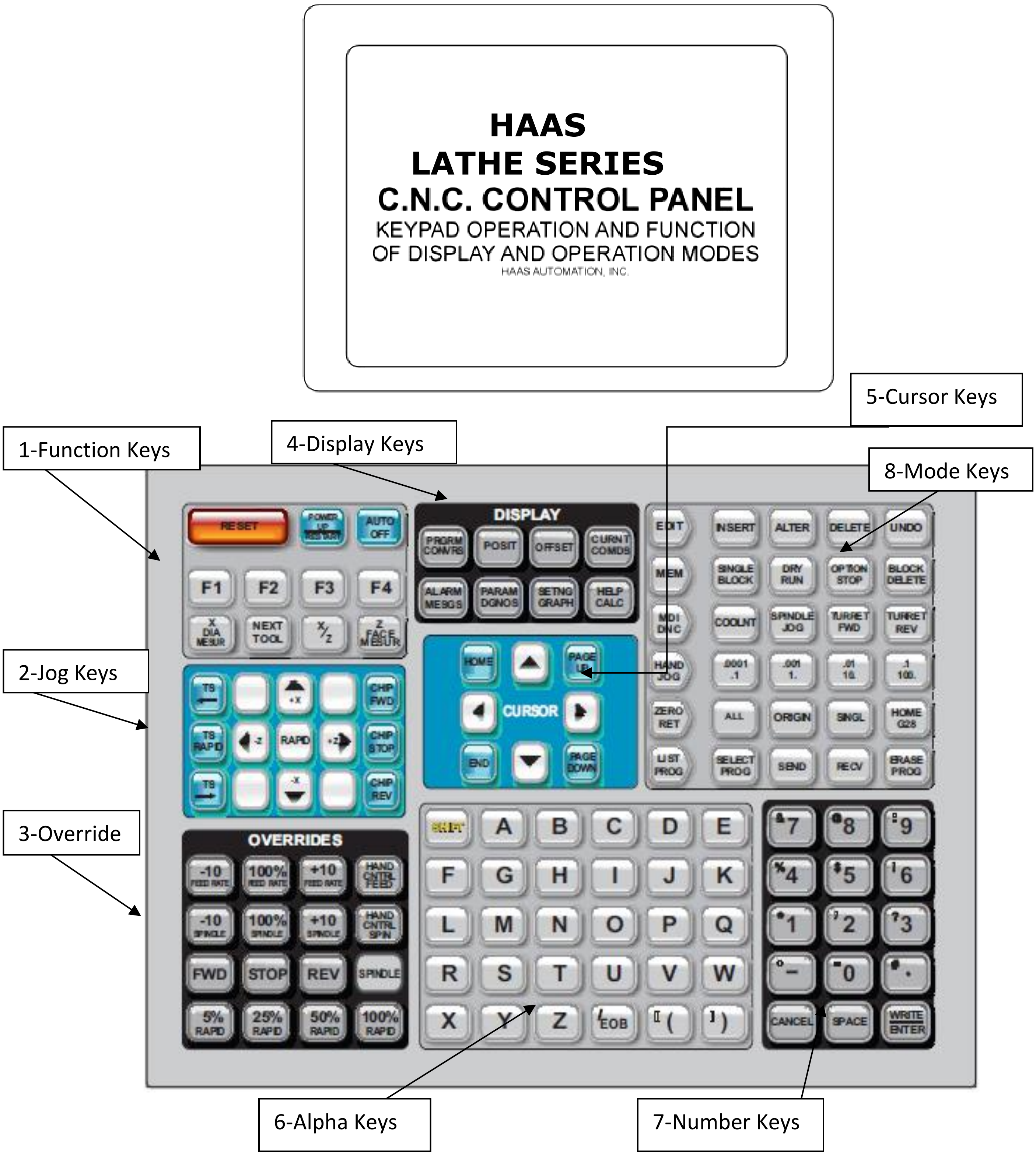
Operation: MEM key. Provides all control features necessary to make a part.

Current mode is displayed at top of display.

Functions from another mode can still be accessed within the active mode. For example, while in the Operation mode, pressing OFFSET will display the offset tables as the active pane in the Main Display Pane and offsets may be altered; press OFFSET to toggle the offset display. While running a part in operation mode another program may be edited in the Main Display Pane. Press PROGRM CONVRS in most modes to shift to the edit pane for the current active program.

Keyboard Introduction

The keyboard is divided into eight different sectors: Function Keys, Jog Keys, Override Keys, Display Keys, Cursor Keys, Alpha Keys, Number Keys and Mode Keys. In addition, there are miscellaneous keys and features located on the pendant and keyboard which are described briefly on the following pages.



1 – Function Keys

F1 – F4 – Perform different functions depending on which mode the machine is in. Example in offsets mode **F1** will directly enter value given it into offset geometry.

X DIAMETER MEASURE – Will take machine X position ask for a diameter measurement on the part which tool turned and put correct X Geometry in Tool Offsets page.

NEXT TOOL – In set up this will select the next tool and make a tool index.

X/Z - Toggles between X-axis and Z-axis jog modes during a set up.

Z FACE MEASURE – Used to record Z tool offsets and Z work offsets**.**

2 – Jog Keys

Chip FWD *(Chip Conveyer Forward)* – Turns the chip conveyer in a direction that removes chips from the work cell.

Chip STOP *(Chip Auger Stop)* – Stops chip conveyer movement.

Chip REV *(Chip Auger Reverse)* – Turns the chip conveyer in reverse.

<-TS – Moves tailstock toward the spindle.

TS Rapid – Increases speed of tailstock movement when used concurrently with the other **TS** keys.

->TS - Moves tailstock away from spindle.

+X, -X *(Axis)* Selects the X axis for continuous motion when depressed.

+Z, -Z *(Axis)* Selects the Z axis for continuous motion when depressed.

Rapid – When pressed simultaneously with X or Z keys will move at maximum jog speed.

3 – Override Keys

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

-10 FEED RATE Decreases current feed rate in increments of 10 percent.
100% FEED RATE Resets the control feed rate to the programmed feed rate.
+10 FEED RATE Increases current feed rate in increments of 10 percent.
HANDLE CONTROL FEED RATE Hand wheel will control feed rate at 1% increments.

-10 SPINDLE Decreases current spindle speed in increments of 10 percent.
100% SPINDLE Sets the control spindle speed at the programmed spindle speed
+10 SPINDLE Increases current spindle speed in increments of 10 percent.
HANDLE CONTROL FEED Hand wheel will control feed rate at 1% increments.

CW Starts the spindle in the clockwise direction.
STOP Stops the spindle.
CCW Starts the spindle in the counterclockwise direction.

5% RAPID Limits rapid moves to 5 percent of maximum.
25% RAPID Limits rapid moves to 25 percent of maximum.
50% RAPID Limits rapid moves to 50 percent of maximum.
100% RAPID Allows rapid traverse to feed at its maximum.

Override Usage

Feed rates may be varied from 0% to 999%. Feed rate override is ineffective during G74 and G84 tapping cycles. Spindle speeds may be varied from 0% to 999%. Depressing Handle Control Feed rate or Handle Control Spindle keys, the jog handle movement varies by +/-1% increments.

Setting 10 will limit rapid movement to 50%.

Settings 19, 20, 21 make it possible to disable override keys.

Coolant may be over rode by depressing **COOLNT** button.

Feed Hold - Stops rapid and feed moves. **Cycle Start** button must be depressed to resume machine feeds. Similar situation applies when Door Hold appears. Door must be closed and **Cycle Start** pressed to continue running program.

Overrides may be reset to defaults with a M06, M30 or pressing **RESET** by changing Settings 83, 87 and 88 respectively.

4 – Display Keys

PRGM/CONVRS – Selects the active program pane (highlights in white). In MDI/DNC mode pressing a second time will allow access to VQC (Visual Quick Code) and IPS (Intuitive Programming System)

POSIT *(Position)* **–** Selects the positions display window (lower middle). Repeated pressing of the **POSIT** key will toggle through relative positions in the Memory Mode. In Handle Jog mode all four are listed together.

1. **POS-OPER** digital display. This is a reference display only. Each axis can be zeroed out independently; then the display shows the axis position relative to where you decided to zero it. In the Handle Jog mode, you can press the X, Y or Z JOG keys and ORIGIN key to zero that selected axis. On this display page, you can also enter in an axis letter and number (X-1.25) and press ORIGIN to have that value entered in that axis display.

2. **POS-WORK** digital display. This position display tells how far away the tools are in X, Y and Z from the presently selected work offset zero point.

3. **POS-MACH** digital display. This is in reference to machine zero, the location that the machine moves to automatically when you press POWER UP/RESTART. This display will show the current distance from machine zero.

4. **POS-TO-GO** digital display. When you're running the machine, or when you have the machine in a Feed Hold, this incrementally displays the travel distance remaining in the active program block being run. This is useful information when you are stepping a program through during a set up.

When the position pane is active one can change which axis is displayed simply by typing X or Y or Z or any combination and pressing write. Only that particular axis or combination will be displayed.

OFFSET – Selects one of two offsets tables: Tool Geometry/Wear and Work Zero Offset. Depressing the **OFFSET** button toggles between the two tables Tool Geometry/Wear table displays 50 tool length offsets (100 tool length offsets on older machines) - labeled (LENGTH) GEOMETRY - along with wear offsets. It also displays radius and tool tip type.

The Work Zero Offset table has G54-G59 plus G154 P1 to G154 P99 offsets available.

The **WRITE/ENTER** key will add the number in the input buffer to the selected offset, and the **F1** key will replace the selected offset with the number entered into the input buffer. Offsets can also be entered using **TOOL OFSET MEASUR** and **PART ZERO SET**

CURNT COMDS – Ten different pages; use **PAGE UP** and **PAGE DOWN**

1. Operation Timers – displays Power-On Time, Cycle Start Time, Feed Cutting Time. Hitting **ORIGIN** will clear any display that is highlighted by the cursor.

2. Real time clock and date.

3. System Variables, for machines with Macro Programming.

4. All Active Codes, displays current and modal command values.

5. Position information: Machine, Distance to Go, Operator, Work Coordinate.

6. Tool life, displays the usage of each tool. An alarm can be set for the number of times you want that tool to be used, and when that condition has been met (that is, the tool has been used the set number of times), the machine will stop, with an alarm for you to check the condition of that tool. Pressing **ORIGIN** will clear the cursor-selected display, and pressing **ORIGIN** when the cursor is at the top of a column will clear the whole column.

7. Tool Load displays the Tool Load Max % of each tool being used. You can use the Limit % column to set the maximum spindle load for a particular tool. When that condition has been met (the tool has reached maximum load), the machine will stop and alarm out for you to check the condition of that tool. Pressing **ORIGIN** will clear the cursor-selected display, and pressing **ORIGIN** when the cursor is at the top of a column will clear the whole column. Setting 84 determines the Overload Action when this limit is met.
   Also vibration loads may be entered.

8. Maintenance times for various items may be loaded.

9. Bar Feeder 300 – Haas servo bar system variables displayed

ALARM/MESGS – Displays messages and current active alarms. Press right arrow key gives alarm history. Press right arrow key again goes to the Alarm Viewer Page. Enter alarm number and press write will give detailed information on a particular alarm code.

PARAM/DGNOS – Lists machine parameters that are seldom-modified values which 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. All of these are rarely changed by the user and should be protected by Setting 7, PARAMETER LOCK. A second press of **PARAM/DGNOS** will show the diagnostics display. The **PAGE UP** and **PAGE DOWN** keys are then used to select one of two different pages. This display is for service diagnostic purposes, and the user will not normally need them.

SETNG/GRAPH – Displays settings - machine parameters and control functions that the user may need to turn on and off or change to suit specific needs. A list of settings is found on page 30.

- Settings are organized into functionally similar page groups with a title.
- Settings are listed with a number and a short description, and a value or choice on the right.
- To find a particular setting, enter the setting number and then press either the up or down cursor arrow key to move to the desired setting.
- You can change a setting using the left or right cursor arrows to display the choices, or, if the setting contains a value, by typing in a new number. A message at the top of the screen will tell you how to change the selected setting. When you changed, it will flash on and off.
- A setting change is not active until it stops flashing. To activate, press **WRITE/ENTER**.

SETNG/GRAPH (2nd part) - The second press of SETNG/GRAPH will bring up the **graphics display** in the Main Display Pane. In this screen you can dry-run a program without moving the axes or risking tool damage from any programming errors. This function is far more powerful than using **DRY RUN**, because all of your offsets and travel limits can be checked before any attempt is made to move the axes. The risk of a crash during setup is greatly reduced. The **Graphics Screen** will display the programmed tool path and generate an alarm if there are any problems. Some of the features of the Graphics display are controlled by selections made in the Settings display, on the page titled GRAPHICS.

1. Press either **MEM** or **MDI** and select the program that you want to run in Graphics. Graphics will also run in the Edit Mode.
2. Press **SETNG/GRAPH** twice.
    - The top left line of the screen will list the GRAPHICS title. Above that line will list the mode you are in (MEM or MDI). The bottom lists explanations for use of function keys **F1** through **F4**.
    - The small window on the lower right side of the screen displays the whole table area during the simulation run, indicating the location of the tool and any zoom window. The center window of the display is a large window that represents a top-down perspective of the X and Y axes. This is where the tool path is displayed during graphic simulation of a CNC program.
3. Press **CYCLE START** to see all the X and Y-axis moves demonstrated.
    Note machine axis and spindle will not when graphic window is up.
4. To step through a program one block at a time in Graphics, press **SINGLE BLOCK**.
5. **F1** is a help key.
6. Press **F2** to zoom in on the Graphics view screen.
    - Use **PAGE DOWN** to zoom in further and **PAGE UP** to expand the view.
    - Use the **Cursor Keys** to position the new zoom window over the area you wish to zoom in on using the small window in the bottom right hand corner. Pressing **HOME** will display the whole table.
    - After positioning the desired zoom window, press **WRITE/ENTER** to accept the view and CYCLE START to see the new view.
    - F3 slows the execution speed of the graphic simulation
    - F4 speeds up the execution speed of simulation.

Use **SINGLE BLOCK** to step through a program in graphics to find any mistakes. During single block you can re-zoom your window to look at tool paths in tight corners etc. Also use position display to see find any discrepant values.

HELP/CALC – Will bring up a help POP UP relevant to the screen you are in. This provides information only pertaining to that screen. Pressing the **HELP/CALC** button again brings up a tabbed menu. With tabulated screens highlighting tab and pressing **WRITE/ENTER** key will open up respective tab. Pressing the **CANCEL** key will close the tab.

Help Opening up the Help tab brings you to the table of contents of the entire Mill Operators Manual. High light the topic of interest and press **WRITE/ENTER** will bring up subtopics on the area of interest. Select subtopic in similar fashion will bring up the relevant page in the manual.

Search The search tab will do a search of the manuals content for relevant information on a keyword. Type in the search term and press **F1.** Topics relevant to the keyword will appear. Highlight the topic and press **WRITE/ENTER** key to open.

Drill Table Displays a common drill sizes, decimal information and tap drill sizes.

Calculator Different calculator functions are available under this tab. The calculator gives ordinary calculations like addition, subtraction, multiplication and division in all tabs. It also will solve trig problems with information about triangles, circles, circle line tangent and circle-circle tangent. A milling and tapping tab will give you suggested cutting speeds and feeds per different materials and sized tools.

Simple Calculator It will calculate simple addition, subtraction, multiplication and division operations. Operations
are listed as: LOAD + - * /. These are selected using the left or right cursor arrow.

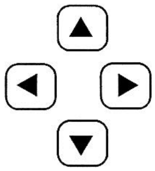
- To enter a number cursor on to LOAD; type the number you want to load and press **WRITE/ENTER**.
- To perform one of the arithmetic functions, enter the first number into the calculator window. Select the operation you want (+ - * /). Finally, enter the second number into the input buffer, press **WRITE/ENTER** to perform the calculation.

Milling and Tapping Help you solve values for feed rates SFM, RPM, and chip load under different conditions. It uses the three equations related to milling and tapping. The first one includes cutter diameter with SFM and RPM. The second one includes RPM, number of flutes, feed rate and chip load. The third one includes thread pitch, RPM and feed rate.

The Milling & Tapping Tab

MILLING:	Cutter Diameter	1.2500 IN	(entered)
	Surface Speed	210.0000 FT/MIN	(entered)
	RPM	**642**	**(calculated)**
	Flutes	4	(entered)
	Feed	**12.8343 FT/MIN**	**(calculated)**
	Chip Load	0.0005 IN	(entered)
TAPPING:	Threads	16.0/IN	(entered)
	RPM	500	(entered)
	FEED	**31.2500 IN/MIN**	**(calculated)**

5 – Cursor Keys



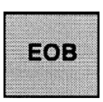
Cursor Keys The cursor keys are in the center of the control panel. They give the user the ability to move to and through various screens and fields in the control. They are used extensively for editing and searching CNC programs. They may be arrows or commands.

Up/Down – Moves up/down one item, block or field.
Page Up/Down – Used to change displays or move up/down one page when viewing a program.
HOME – Will move the cursor to the top-most item on the screen; in editing, this is the top left block of the program.
END – Will take you to the bottom-most item of the screen. In editing, this is the last block of the program.

6 and 7 – Alpha Keys and Numeric Keys

The **Alpha Keys** allow the user to enter the 26 letters of the alphabet along with some special characters. Depressing any Alphabet Key automatically puts that character in the Input Section of the control (lower left-hand corner).

SHIFT key provides access to the yellow characters shown in the upper left corner of some of the alphanumeric buttons on the keyboard. Pressing **SHIFT** and then the desired white character key will enter that character into the input buffer.



EOB key enters the end-of-block character, which is displayed as a semicolon on the screen and signifies the end of a programming block. It also moves the cursor to the next line.

Parentheses are used to separate CNC program commands from user comments. They must always be entered as a pair. Example: (T1 ½" End Mill)

Also any time an invalid line of code is received through the RS-232 port, it is added to the program between parentheses.

(–) and (**.**)	These keys are used to define negative numbers and give decimal position.
+ = # * [ ]	These symbols are accessed by first pressing the **SHIFT** key and then the key with the desired symbol. They are used in macro expressions (Haas option) and in parenthetical comments within the program.
, ? % $! & @ :	These are additional symbols, accessed by pressing the **SHIFT** key, that can be used in parenthetical comments.

6 and 7 – Alpha Keys and Numeric Keys (continued)

The **Numeric Keys** allow the user to enter numbers and a few special characters into the control. Depressing any number key automatically puts it into the Input Section of the Control.

Cancel The **Cancel** key will delete the last character put into the Input Section of the control display.

Space Is used to format comments placed into the Input Section of the control display.

Write/ Enter General purpose "Enter" key. It inserts code from the input section into a program when the program display is in EDIT mode. With offsets pages active, pressing the **WRITE/ENTER** key adds a number in the Input Section to the highlighted cell. Pressing the **F1** key will input the number into the cell.

- The (Minus Sign) is used to enter negative numbers.

. The (Decimal Point) is used to note decimal places.

8 – Mode Keys

Mode keys set the operational state of the machine tool. Once a mode is set the keys to the right may be used. The current operation mode of the machine is displayed at the top thin pane of the CRT.

<u>EDIT</u> The edit mode is used to make changes in a program stored in memory. When you press **EDIT** two panes appear at the top of the screen. In the left pane the active program appears. In the right an inactive program appears or the select program screen appears. On the bottom left a editor help pane appears and on the right a clipboard pane. Editing may be performed in either the active or inactive panes. Pressing **EDIT** toggles between the two panes, (changes background to white). To call up a program from memory and put it in one of the edit panes press **SELCT/PROG.** Highlight the program desired by using the up or down cursor buttons and press **WRITE/ENTER.**

In the edit mode you are able to use the edit keys in the **same row** as the **EDIT** key.

INSERT Enters commands keyed into the input panel in lower left pane of CRT after the cursor highlighted word in a program.

ALTER Highlighted words are replaced by text input into the input panel.

DELETE Highlighted words are deleted from a program.

UNDO Will undo up to the last 9 edit changes**.**

F1 KEY While in the edit mode pressing **F1** will bring up an edit pop up window. Using the sideways cursor buttons will toggle thru HELP, MODIFY, SEARCH, EDIT AND PROGRAM MENUS. The up and down buttons will cursor thru the different options in each of the above.

MODIFY Gives options on changing line numbers.

SEARCH Will perform a search and gives the option of replacing text.

EDIT Gives option of cutting or copying and pasting to a clipboard and to another program.

PROGRAM Gives options of creating new program, selecting a program from list to edit, duplication of programs, switching from left to right side of window panes.

Background Edit When a program is being run pushing the edit will bring up the Background Edit pane in the Main Display Pane. Simple edits may be performed on the program that is being run or another program. The edits on the running program will not take place until after the current cycle has completed.

<u>MEM</u> The memory mode is the mode used when running the machine and making a part. The active program is shown in the Program Display Pane. Keys in the memory mode line reflect different ways of running a part in memory. When the keys to the right are depressed they will show up highlighted in black on the bottom right of the CRT.

SINGLE BLOCK When depressed **SINGLE BLOCK is** highlighted in black and will appear on the bottom of the CRT. When the machine is in SINGLE BLOCK mode only one block of the program is executed every time the cycle start button is depressed. Used when first test running a program or temporarily stopping a program when it is running.

DRY RUN Used to check machine movement without cutting a part. In dry run the machine runs at one feed rate. With the availability of graphics which show visually what the machine tool path is this mode is rarely used.

OPTION STOP When **OPTION STOP** is depressed program will stop at any M01 which is in the program. Normally M01s are placed after a tool is run in a program. When a job is being set up the operator may put machine in op stop mode to check dimensions after every tool has completed cutting.

BLOCK DELETE When this button is depressed any block with a slash (/) in it is ignored of skipped.

<u>MDI</u>

DNC **(**MANUAL DATA INPUT mode) – Usually short programs are written in MDI but are not put into memory. DNC mode allows large programs to be drip fed from a computer into the control.

COOLNT Turns coolant on and off manually

ORIENT SPINDLE Rotates and locks spindle to specific angle. Used when lining up tools where spindle orientation may be a issue such as boring heads.

ATC FWD Rotates turret to next tool and performs tool change - also used to call up specific tools or pots. Enter tool number (T1) and press **ATC FWD**.

ATC REV Rotates turret to previous tool and performs tool change - also used to call up specific tools or pots. Enter tool number (T1) and press **ATC REV**.

HAND JOG Puts machine in jog mode for set ups. Top values (.0001, .001, .01, .1) represent distance traveled per click of jog handle. Bottom values (.1, 1., 10., 100) represent feed in inches/minute when jogging axis using jog buttons.

ZERO RET On pressing position display becomes highlighted in Zero Return mode.

ALL Returns all axes to machine home similar in similar fashion as a Power Up/Restart.

ORIGIN Sets selected displays to zero or other functions.

SINGL Returns a single axis to machine home. Select desired axis (X, Y, or Z) then press **Singl** axis button.

Home/G28 Rapid motion to machine home; will make a rapid move in all axes at once - may also be used for a rapid home in one-axis. Press axis to home then **G28.** ***Caution must be used that extended tools, tailstock or parts are out of the way before initiating this rapid move to home.***

LIST PROG Will bring up list of programs in a tab format. Pressing **Cancel** will return you to tab at top usually MEM or USB. Cursor to left or right for which list one wants. Pressing **Enter** will open a list of programs. Cursor UP (∧) or DOWN (∨) to program desired. Select the desired programs to be moved by pressing **WRITE/ENTER.** This will put a check mark beside it. **F2** will copy selected program or programs to be moved. A pop up menu will ask where you want the selected programs to be copied.

SELECT PROG After highlighting a program from List Program with up or down cursor pressing this button will place the program in the Active Program Pane. This is the program that will run the CNC machine in the Memory mode. Use in the Edit mode in the Main Display will enter selected program in the Main Display pane for editing.

SEND Will send a selected program or programs out thru RS-232 serial port

RECV Will get machine ready to receive program from RS-232 serial port.

ERASE PROG Will erase highlighted program or programs. A prompt will appear asking if you want to delete selected program asking for Y/N.

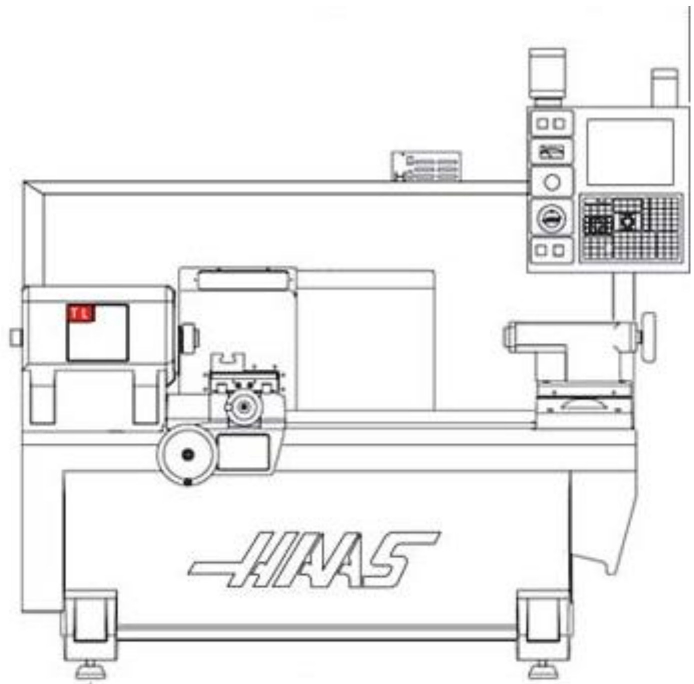
Settings

Scrolling through Settings with 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.)

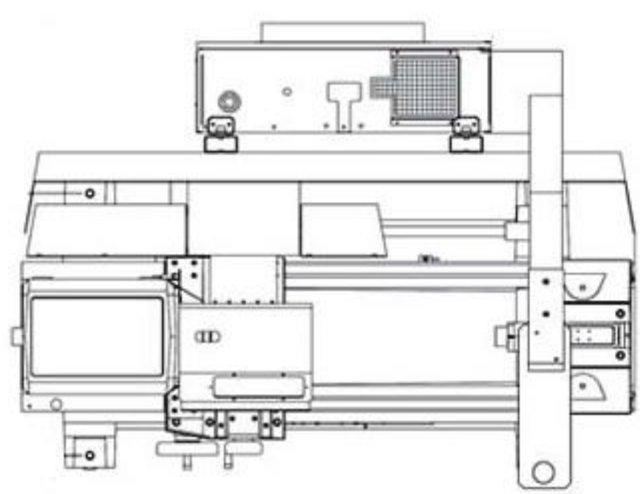
There are many settings which give the user various options over the control of their machine tool. Read the Settings section of the operator's manual for all the possible options.

Tool Room Lathe Orientation and Walk Around

The agenda for this section of the training manual is to familiarize everyone with the physical layout of the TL series machine, the functions of the mechanical features of the machine, and general maintenance of the TL series lathe.



FRONT VIEW OF TL-SERIES LATHE



TOP-DOWN VIEW OF TL-SERIES LATHE

Power-Up Procedures

The TL Series lathe follows the same standard start-up procedures as with any other Haas machine, and should be followed in the procedure demonstrated below:

Main Power

1. Locate the main power switch on the back of the machine’s control cabinet, and switch it to the “ON” position.

Power-Up Procedure

After switching the main power supply on, we need to “power-up” the Haas control. That procedure is as follows:

1. Move the tailstock (if equipped) all the way to the far right of the limits of the machine, and leave the clamps on the tailstock loose. This will provide the saddle enough room to reference itself. ***CAUTION: the control does not know where the tailstock is, and there is a chance to crash if the tailstock is not in the home position!***

2. Visually verify that all of the machine’s ways are clean and free of dirt, and any moving parts of the machine are clear of chips, coolant, etc… and verify that there is sufficient air pressure (80 psi) to the machine

3. Press the Green **POWER ON** button located on the upper left corner of the Haas control. The machine will take a couple of minutes to load its software.

4. The machine is now “Powered-On”

Machine Restart / Zero Return Procedure

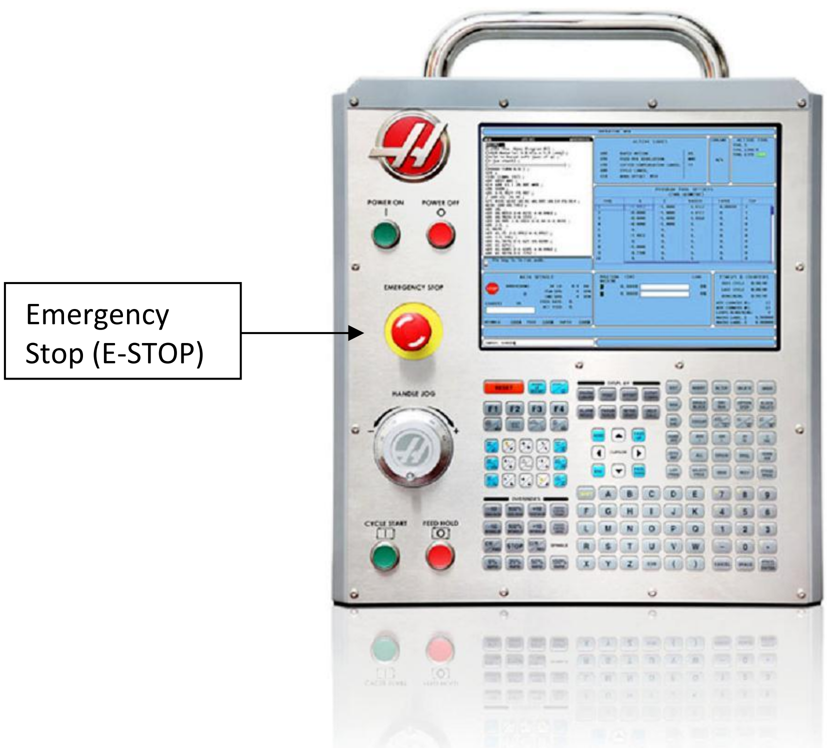
After the control is powered on, we still need to “reference” the machine. Upon start-up, the control does not know the location of the cross slide (X axis) or the saddle (Z axis), so the machine needs to “find” these 2 axis thru a procedure known as a “Zero Return”. This will travel the machine to it’s farthest most limits so that the cross slide and saddle can be detected by the limit switches.

1. Locate the orange RESET key and push it to clear any alarms that are present, and to power on the servos of the machine.
2. Next to the RESET button is the POWER UP / RESTART button. Press it once and the machine will travel the X axis first, then the Z axis to their utmost farthest limits, which is all the way towards the operator on the X axis / cross-slide, and all the way to the right for the Z axis / saddle. The machine will stop at machine home, as described in section-4.a.iii

Tool Room Lathe Safety

Emergency Stop Switch

Located on the left side of the control panel, there is a large, round, red button that is circled with a yellow ring. This is the EMERGENCY STOP or “E-STOP” button. In the event of a un-controlled machine condition, a crash of the machine, or any situation that requires an immediate shut down of all axis functions, movement, spindle functions and power to the servos of the machine, the Emergency Stop button should be used. This button is used to halt everything on the machine immediately!



Proper Use of Machine Guarding

The Toolroom Series of lathes comes equipped as an "open" machine, much like that of a manual lathe. The handwheels, chuck, tailstock and toolpost are all exposed and in an open environment. Extreme concern should be used when the machine is moving in ANY mode of operation, but especially in CNC mode. The handwheels will be turning, and it is possible for loose clothing, hair, etc… to get caught within the moving parts of the machine.

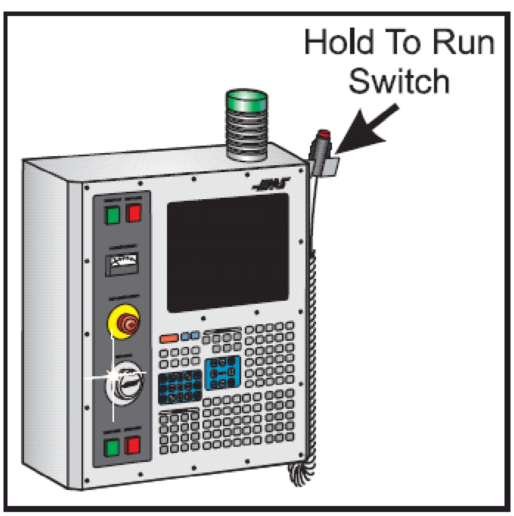
Also, extreme care should be taken when around the chuck and part as it turns. The main motor of the machine has a large amount of power, and can be dangerous. Extreme care should be taken to avoid a collision between any tooling or the tool-post and the chuck. It is possible to do exactly that.

In any event, all TL series machines come either equipped with a hinged shield that surrounds the chuck, or a complete set of sheet metal that encloses the machine and the machine should not be run with any guards open.

In the current version of the TL series of software, it is impossible to run in either manual or CNC modes with the guarding open. The setting on the control may be changed, but changing this setting rests all responsibility on the operator and or the owner's of the machine. Neither Haas, Productivity, nor any individuals of either company can be responsible for bypassing ANY safety settings on the machine.

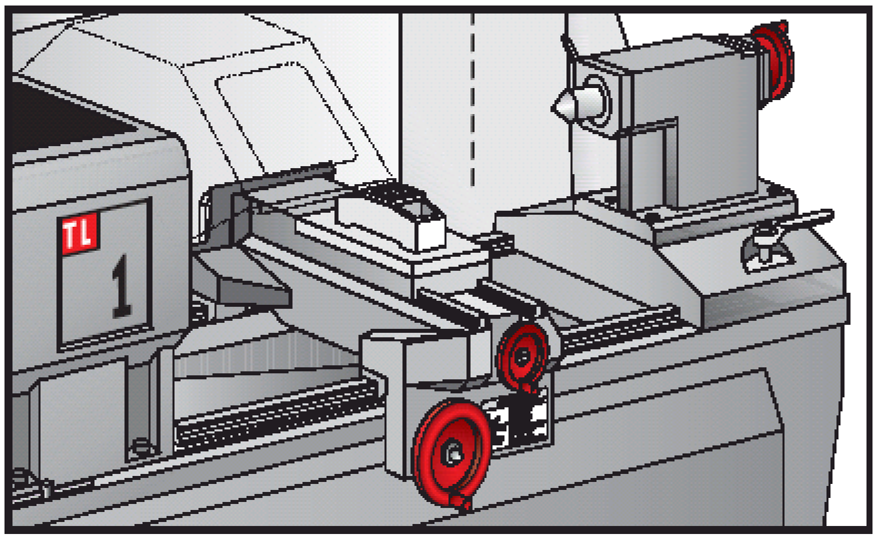
Dead Man Switch

Located on the right hand side of the Haas control, with a coiled cord, is a "Dead-Man Switch". Anytime a command is given to the control, where-as it is by MDI mode, Manual Mode, or thru full CNC Mode, and this command generates machine movement (axis, spindle, etc….) both the Dead Man Switch and the Cycle Start Button will need to be held at the same time. This is a requirement of OSHA to be complient with their standards for safe operation on any machine that has automation. Other words, a two handed start I needed to start any function of the machine.



Hand Wheel Safety

Again, the TL Series of lathes when in CNC mode will still move the manual handwheels as the machine moves. PLEASE PAY ATTENTION TO THE HANDWHEELS TO AVOID ANY INJURY!!!! Even though elaborate thought has been put into making sure the TL series is safe, extra caution needs to been given to any moving part of the machine!!!



Maintenance of the TL Series Lathe

Even though your TL Series machine has been built to the highest standards, using the highest available techniques and materials, general maintenance is required on both the machine and on the control to insure long life of its components.

Haas has gone thru extensive testing to verify the specifications on the required lubricants and filters that went into your new machine before they ever entered Haas's production, please use parts, filters, and lubricants that are as specified to insure longevity of the life of your or your boss's investment. This will maintain the highest levels of performance and accuracies for the longest possible time.

If there is any question on what exactly to use, please refer to the Operator's Manual for these specifications or contact the Productivity Service and Parts Department for the proper fluids and parts required for maintenance.

Productivity, Inc. - Service and Parts

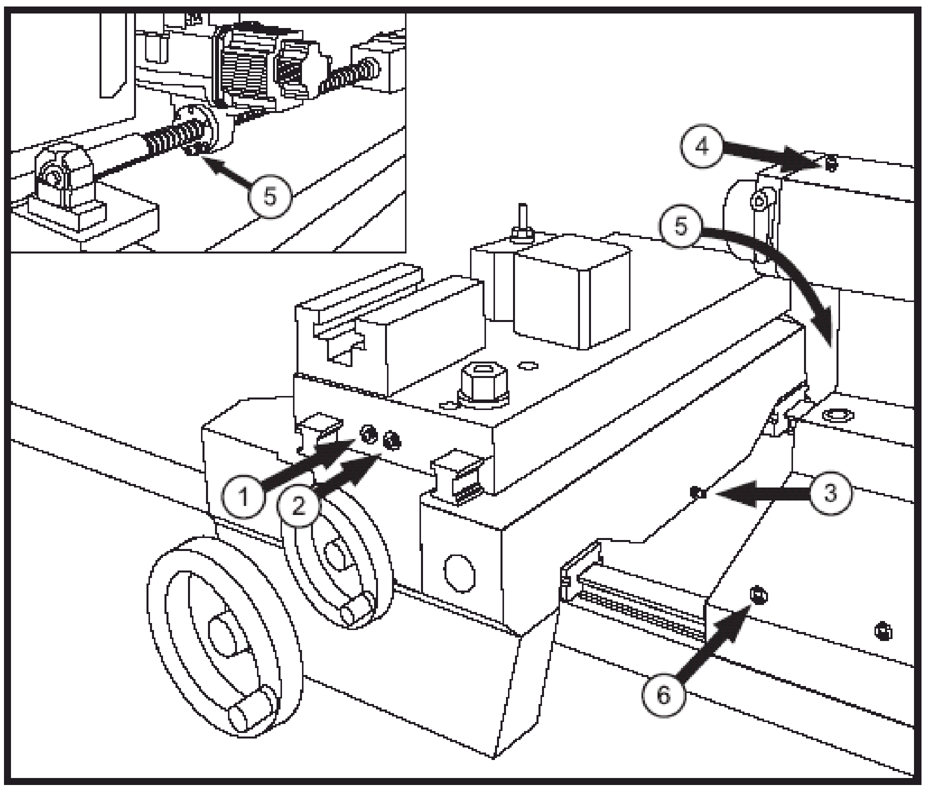
Haas Factory Outlet
A Division of Productivity, Inc.
www.productivity.com
MN – 15150 – 25th Avenue North, Plymouth, MN 55447
Ph 763-476-8600 Fax 763-476-4092
IA – 9440 Atlantic Drive SW, Cedar Rapids, IA 52404
Ph 319-734-3403 Fax 319-632-4289
NE – 8402 South 117th Street, Omaha, NE 68128
Ph 402-330-2323 Fax 402-330-6901

Headstock Lubricant

The headstock or spindle of the TL series machine is a grease-pack design, and will not require any regular maintenance.

Grease Points

The TL series lathes are equipped with grease fittings rather than an oil lube system to provide surface to surface lubrication for all contact points on the machine. The linear ways, the ballscrews, and the tailstock are all equipped with grease fittings, and they should be lubricated weekly to insure consistent performance and accuracy of your machine.



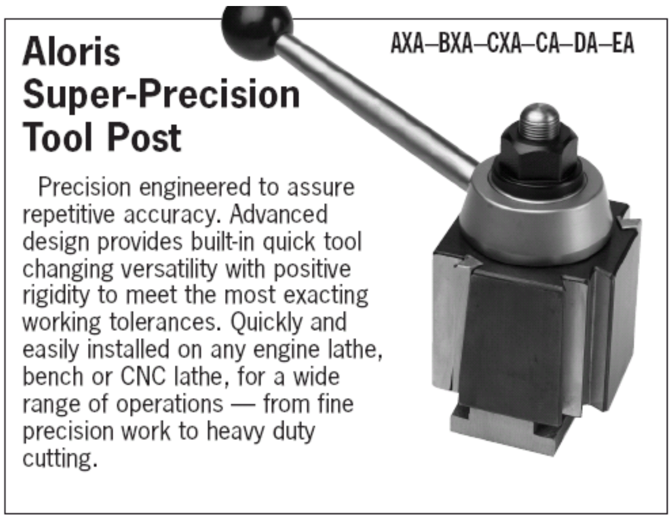
TL Series Lubrication Points

1. X-axis cross-slide trucks
2. X-axis cross-slide ball screw
3. Z-axis saddle trucks
4. Tail-stock screw
5. Z-axis saddle ball screw
6. Tailstock base; four places

For the lubrication points shown above, a general purpose lithium grease is required with enough volume to push grease out from the front and back of the linear ways, and from around the ball screw bearing housings

Aloris Tool Post Operation

The most often equipped option for the TL series lathe for holding your turning tools is the Aloris Single Station Tool Post option. This system allows for setting individual tools in their own respective holders, and allows for a repeating, quick change system, that has extensive options to allow for safe holding of a wide variety of different tooling products.



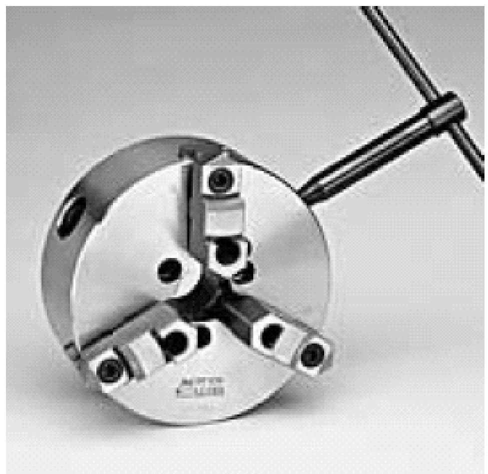
Aloris Super-Precision Tool Post

All that is required of the operator to switch from one tool to the next, is to swing the handle in a counter-clockwise direction to release the current tool, pull the tool-holder up and off of the Aloris dovetail, slide on the next tool onto the Aloris dovetail, and swing the lever till clockwise to firmly clamp the tool-holder against the toolpost.

CAUTION: Please take care to never change tools while the machine has the spindle turning or while the axis are moving!!

3 Jaw Scroll Jaw Chuck Operation

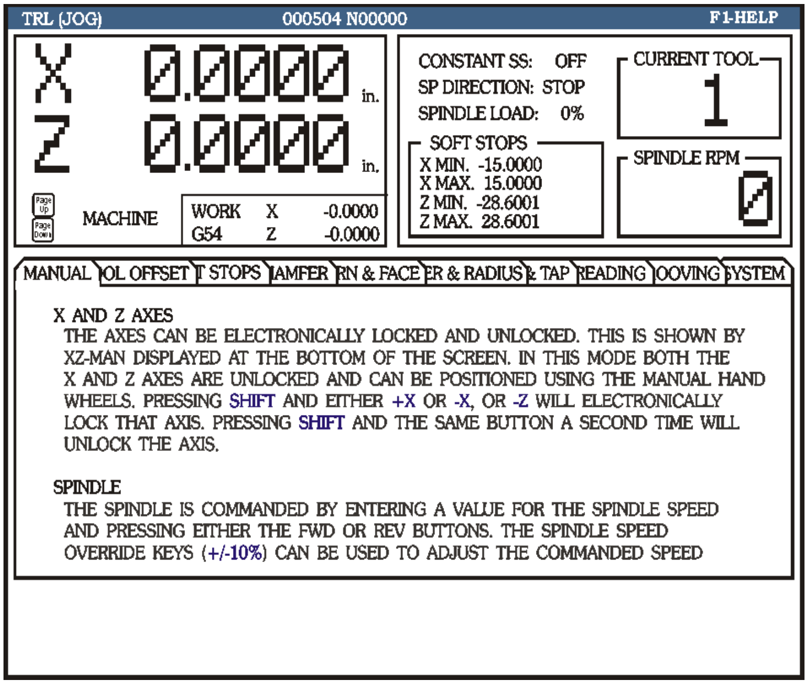
The standard Haas option for the TL series lathe for part holding is a 3-jaw scroll plate chuck, that is key operated from the outer diameter of the chuck. Simply rotate the chuck key in a clockwise direction to draw the jaws of the chuck closer to the chuck center, and counter-clockwise to move the jaws away from the chuck center. The configuration of the chuck jaws and part holding is up to the operator to decide which is best for that particular project.



3-Jaw Scroll Chuck / Key

CAUTION: The chuck key comes equipped with a spring on the end to prevent accidentally starting the spindle with the chuck key engaged to the chuck. DO NOT REMOVE THE SPRING FROM THE KEY!!!!! Starting the spindle with the chuck key engaged will lead to extensive machine damage and potential physical harm for the operator!

HAAS Intuitive Programming System (IPS)

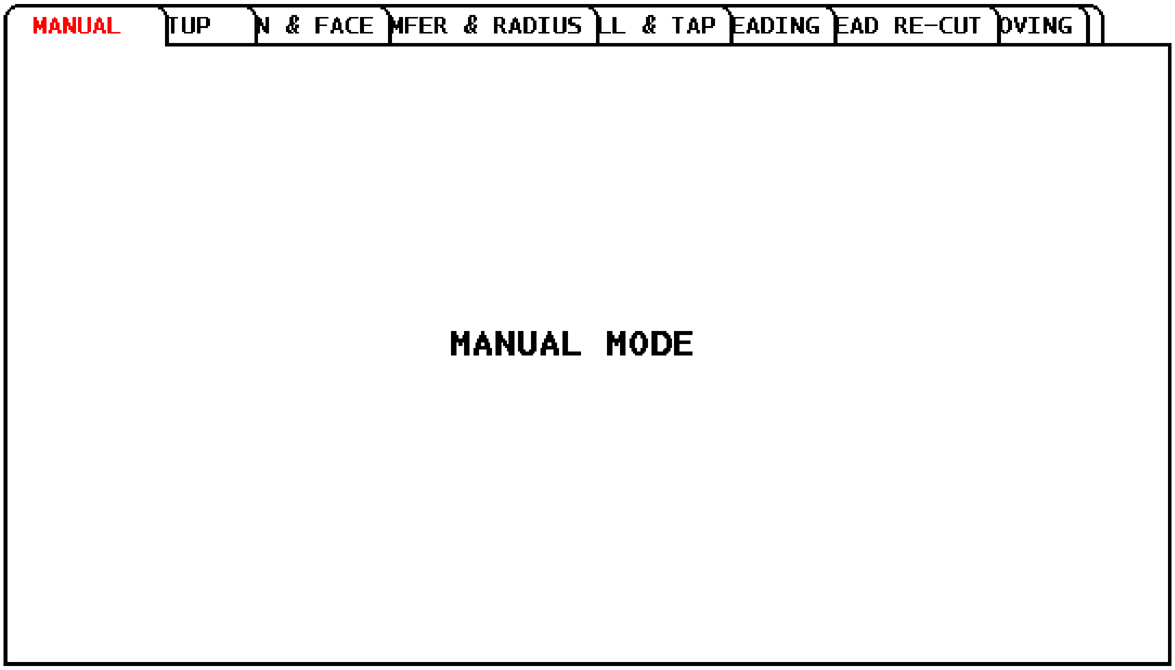


The most outstanding feature on the TL Series of Haas machines is the Intuitive Programming System, or IPS. This is a simplified programming system that allows for much faster programming and setup of the machine, and allows the operator to program using a question and answer format, rather than a G&M code format.

The objective of this section is to go thru the different IPS system “Tabs” and “Menus” and how each relates to generating fast and accurate parts from the TL Series Lathe.

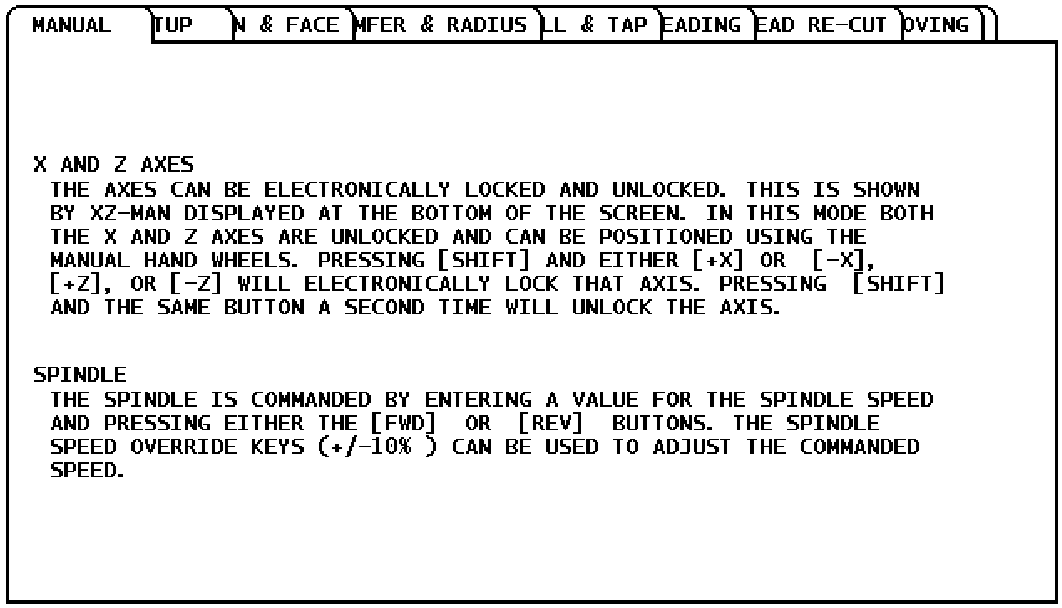
The best way to do this from a training point of view, is to start with a typical part print that would be seen by a Toolroom Lathe machinist, and go thru step by step with the IPS to show how to manufacture that part on the TL Series Lathe.

To enter the IPS system, press the **HAND/JOG** mode key on the right hand side of the control pendant.   The screen below appears:



The cursor keys at the middle of the Haas control navigate thru the different “**TABS**” from left to right, Pressing the **WRITE/ENTER** key will access the selected tab. If there are multiple choices under each tab, again use the right and left cursor control to navigate in the “**SUB-TABS**” to select the cycle you want to program. At any time, pressing the **CANCEL** button will back out of any SUB-TAB or TAB.

Using the UP/DOWN cursor keys will navigate you thru each piece of information that has to be entered into each specific cycle the IPS can program. Below the tab Manual has been opened.



One way the machine may be moved manually using the red handles is by opening this screen up.

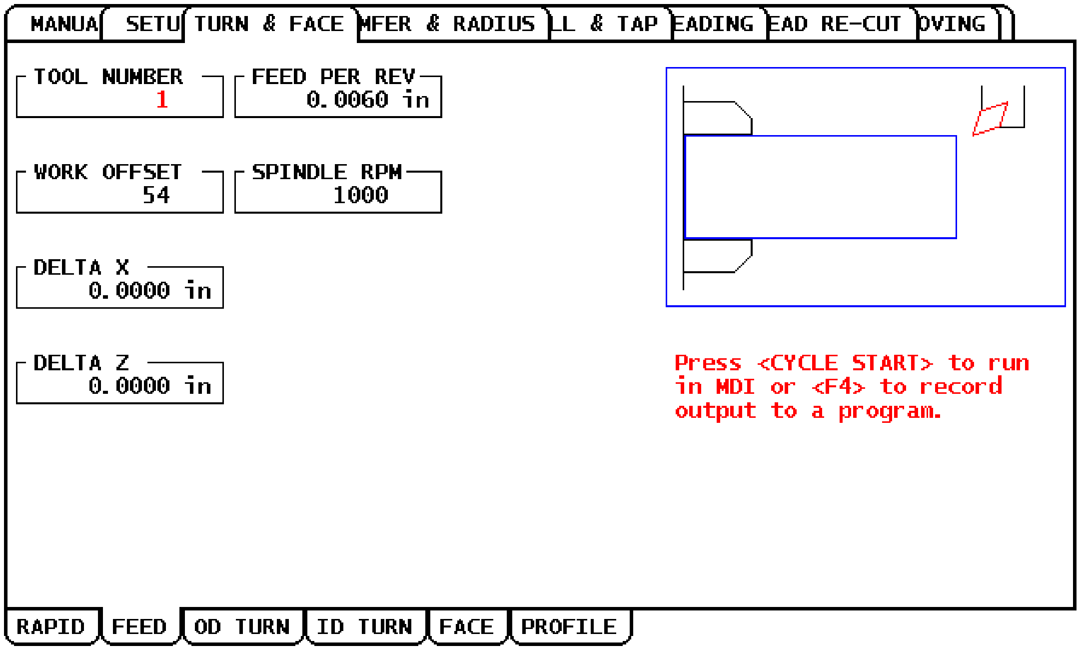
The only other way the machine may be put in handle manual mode is by pressing the **HAND JOG** key and pressing the yellow shift key in conjunction with either the +Z, -Z or +X, -X or both jog keys.

In this mode Tool Geometry may be set by establishing a part Z zero by facing the part off manually.

The **spindle** is commanded on by entering a value for the spindle speed and pressing either the FWD or REV buttons. The spindle speed override keys (+/- 10%) can be used to adjust the commanded speed from the manual mode screen.

With the spindle turned on a facing and turning cut may be made manually by turning the handles just like on an engine lathe. This will allow geometries to be set for most of the outside diameter cutting tools.

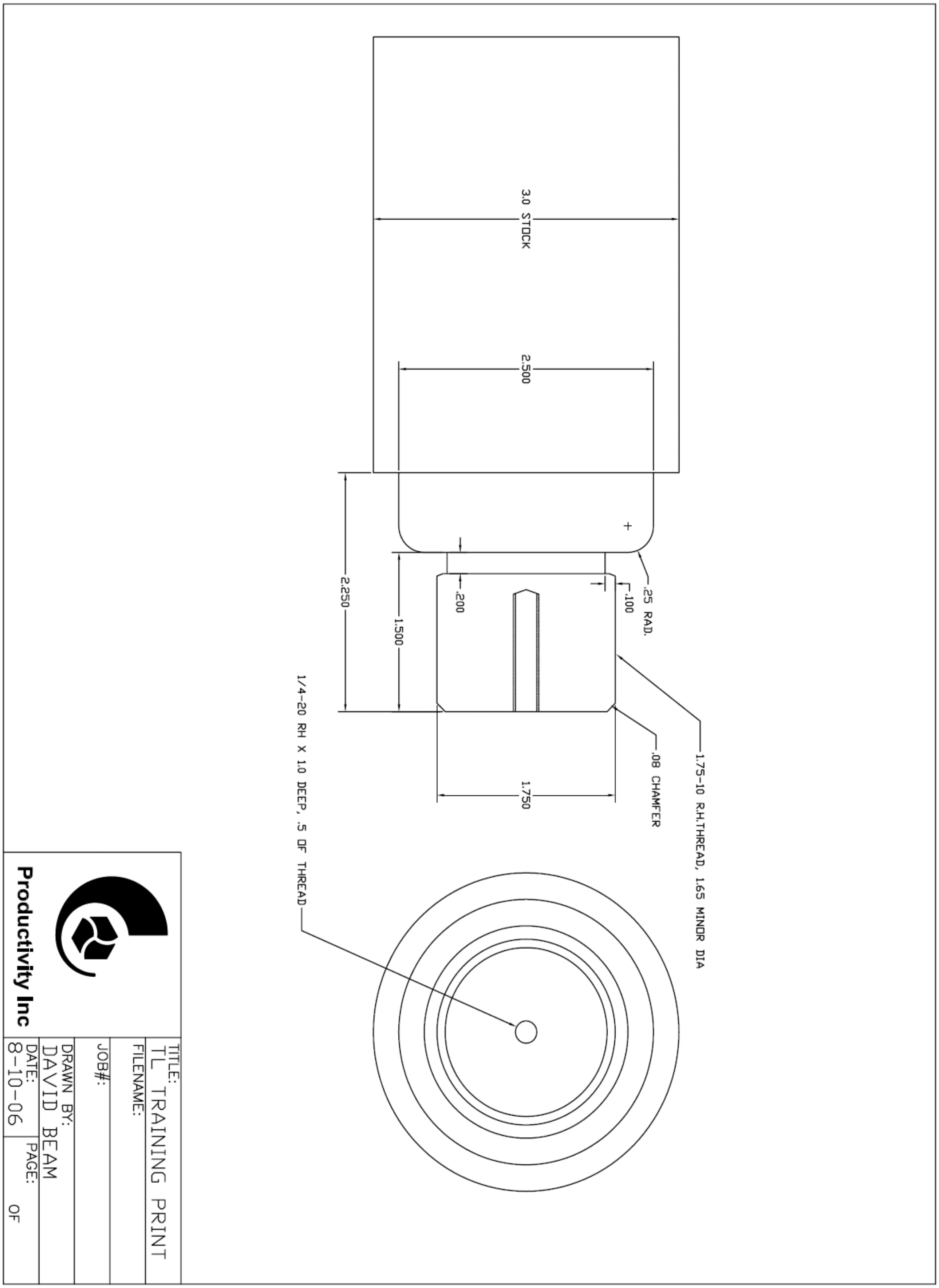
Another way to establish a face and turned surface is to go the IPS screen TURN & FACE and open the FEED tab below to open the following screen.



In this mode the tool is brought up to the face or turning position manually. Next, a DELTA X or a DELTA Z distance selected. A feed and spindle speed is entered. The delta x or delta z values are the incremental distance machine will move at the prescribed feed and speed from the present position it is at. Pressing the F4 key then selecting Output to MDI will create the program into MDI.

With a DELTA X of -3.0 entered the following short facing program may be created and run in MDI.

```
(FEED) ;
T101 ;
G54 ;
G97 S1000 M03 ;
G01 U-3. W0. F0.006 ;
M01 ;
;
```



TL Series Training Part

Tool Offsets Tab

Tool offsets must be set before an automatic operation can be run. We must define each tool to the control so that the control knows the type, length, size, shape and direction that the tool is made to cut.

Before we can cut a part, we need to determine how many operations are needed (turning, threading, drilling, tapping, grooving, etc…..) and what tools will be needed for each operation.

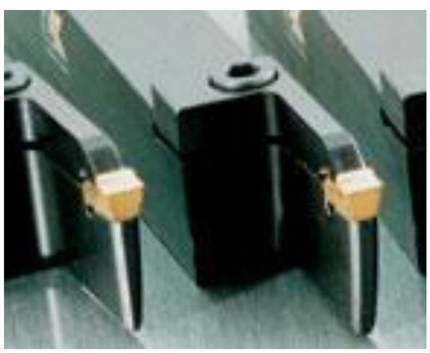
For our part that is drawn above, we need have the following processes that need to be done:

1. Turn the 2.500 Dia
2. Turn the 1.750 Dia
3. Face the end of the part
4. Turn the .08x.08 chamfer
5. Turn the .250 Radius
6. Groove the back of the thread
7. Turn the 1.75x10 thread
8. Drill the ¼-20 tapped hole
9. Tap the ¼-20 tapped hole

Processes #1-5 can be performed with any standard OD turning tool such as the ones pictured here:

We will call this TOOL # 1

Process #6 Needs a grooving tool that can plunge and cut on the sides, such as the one pictured here:



We will call this TOOL# 2

Process # 7 is our OD Thread, and needs to be performed with an OD Threading tool, like this:

We will call this Tool #3

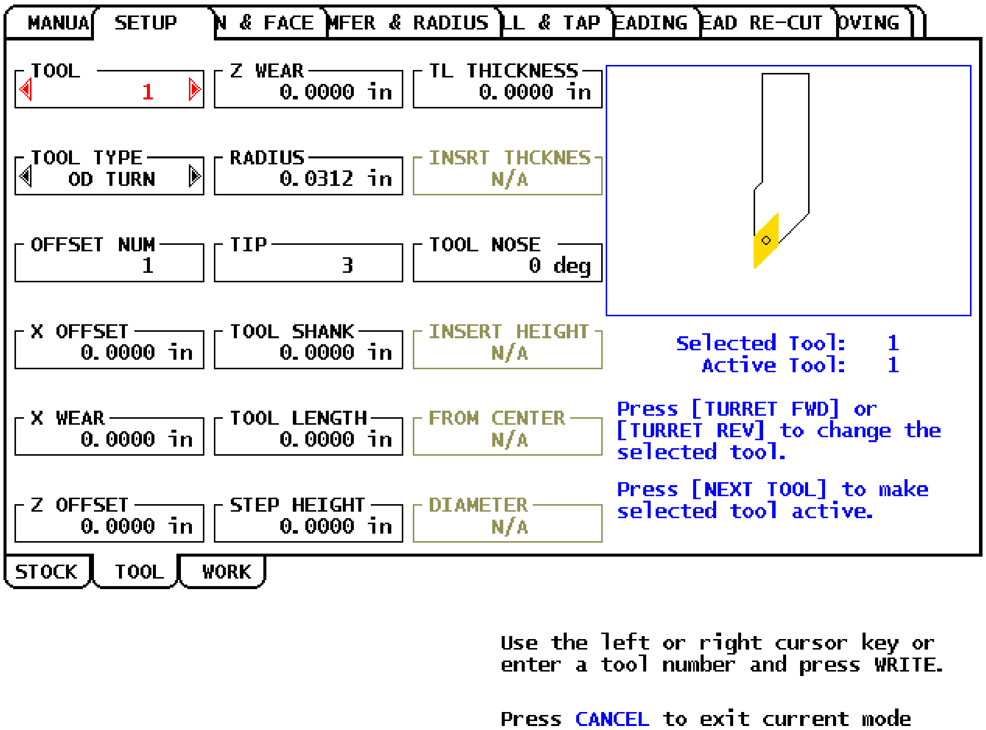
For Process # 8, we need to drill the hole for our tap, a .201 Dia drill like this:

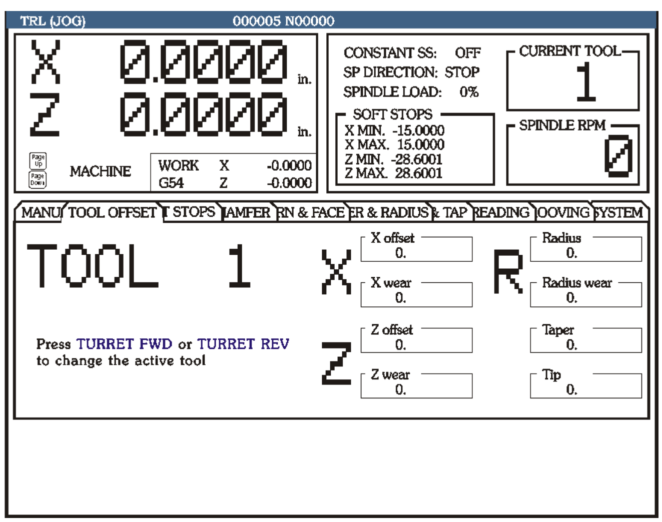
Tool #4

And for Process # 9, we will tap our hole with a ¼-20 tap like this:

Tool #5

Navigate the IPS system to the SETUP tab and open the Tool tab by pressing **WRITE/ENTER**. The screen below will be displayed.





By pressing the **TURRET FWD** or **TURRET REV** key, one can cycle between tools #1-20, depending on the version of software your TL series has. Each tool will have the same information listed. After selecting the tool it must be made **active** by pressing the **"NEXT TOOL"** key. The active tool will be displayed in the lower right hand corner of the control display. Arrow down to Tool Type and select the correct designation by using right and left cursor keys. Enter the OFFSET number, usually the same number as the tool number.

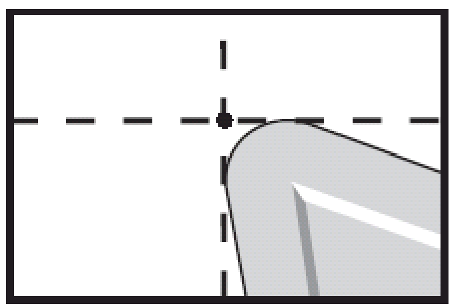
Defining Turning Tools

Load Tool #1, the OD Turning Tool on to the Aloris Toolpost and handwheel the machine over to the OD of our material, and using a feeler gauge bring the tool in till it touches.

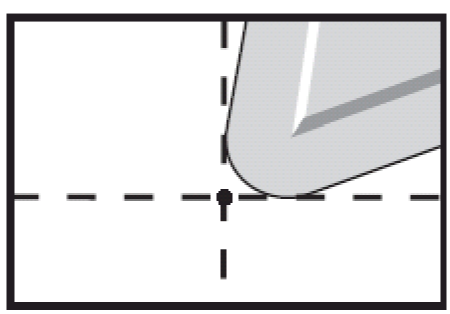
Press the **X DIA MEASURE** key, and the control will ask for the diameter at which we are holding the tool at. Enter the measured diameter example of **3.000** and press the **WRITE ENTER** key. The control will automatically write the “X Offset“ into the X offset box.

Next, handwheel the machine over and touch Tool #1 to the face of our part using the feeler gauge, and simply press the **Z FACE MEASURE** key. This will automatically store the “Z Offset” for that tool.

All that is left to do is enter the tool nose radius for that insert, and define the Tool Tip Type, or in simpler terms, which direction the tool is pointing towards the part. This is needed wherever the control has to perform a cutter compensation command. With the TL series lathe, we will use mainly one of two tool tip definitions, on the next page:



Tool Tip # 3 – For OD Turning Operations



Tool Tip #2 – For ID Turning Operations

All other tools such as drilling tools (drills, taps, reamers, etc….) parting tools and threading tools will use tool tip #0

Other information such as TOOL SHANK, TOOL LENGTH, STEP HEIGHT, TL THICKNESS and TOOL NOSE need not be entered. This information is needed only if "live imaging" is used.

Define Tool #2 and Tool #3 the same way as we did number one. The only difference is that a radius or a tool tip definition is not required on either tool. Make sure the respective tools are made active before setting offsets by using the **NEXT TOOL** key.

Defining Drills, Reamers, etc.

The process to defining a center drilling tool that is mounted to the tool post is also performed the same way. For example, mount the drilling chuck on to the Aloris Tool Post, and insert our .201 drill into the chuck, and align the drill so that the flute tips of the drill are horizontal to the cross slide of the machine. Simply handwheel the drill over to the diameter of our stock and use a feeler gauge, pick up the edge of the part. Press the **X DIAM MEASURE** key, and again the control will ask for our diameter, but since we need to compensate for our radius of our drill, **ADD .201** to the diameter of the stock, thus we enter for example **3.201** for a diameter.

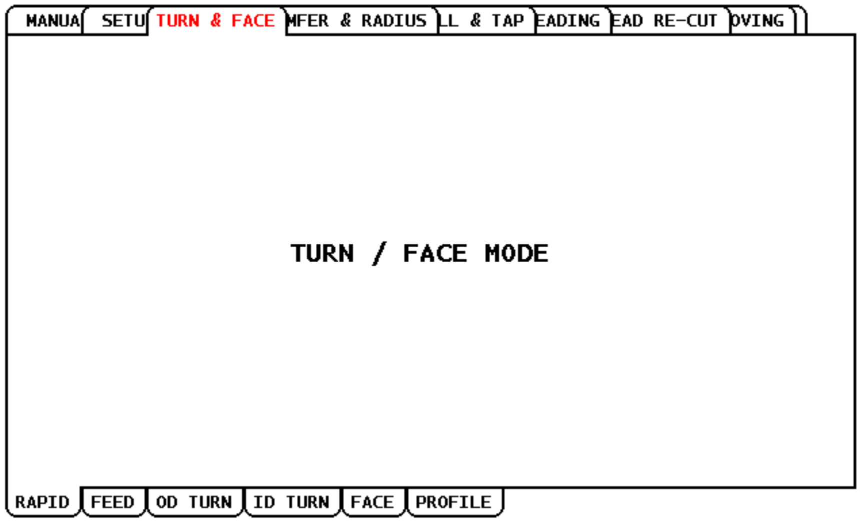
The same process is used for touching the Z, simply by bringing the tool to the face of the part, and press **Z FACE MEASURE.**

Since we have "found" the center of the drill chuck, it is the same for all tools we use in the drill chuck. We can use the same X offset for the drill and the tap. Simply index to Tool #5, insert the tap into the drill chuck, Manual enter the X offset distance from Tool #4, and use the Z FACE MEASURE feature to touch the tip of the tap to the face of the part.

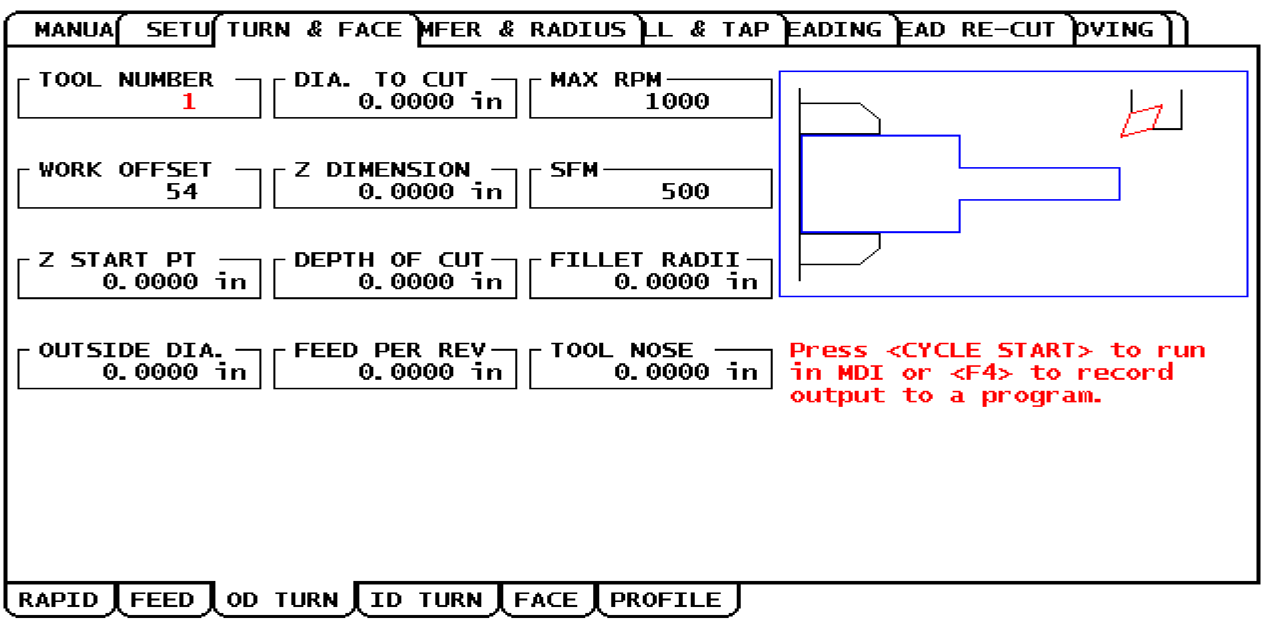
Our tooling for our part is now defined, and the program for the machine now needs to be created.

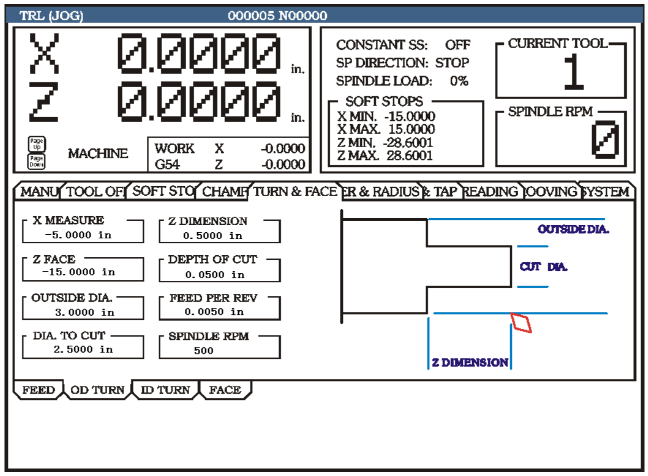
Turn & Face Tab

All of the Turning and Facing operations for the IPS are located under the **TURN & FACE** main tab, so back out to the main tab selection view, and use the right or left cursor keys to highlight the **TURN & FACE** tab.



Press WRITE ENTER to access the TURN / FACE mode, then select OD TURN to start our processes 1&2. The display below will appear:





The variables that the OD Turn Function will need to know, and are listed as follows:

IPS OD Cycle Question	Relationship To The Part
Tool Number	Tool To Be Used
Part Offset	Always G54 If IPS Used to Touch Off Tools
Start Diameter	What diameter to start cutting at
Finish Diameter	Diameter to cut to
Depth Of Cut	How Much Material Per Pass
Z Dimension	How Far From Face - Back to Turn
Feed Per Revolution	Amount of Feed Per 1 Spindle Turn
Surface Footage	Recommended Insert SFM
Max Spindle Speed	Max RPM Of Spindle For The Cycle

For the 1st process, our 2.500” Diameter, we will enter the answers below:

IPS OD Cycle Question	Relationship To The Part
Tool Number	1
Part Offset	G54
Start Diameter	3.000
Finish Diameter	2.500
Depth Of Cut	.100
Z Dimension	2.25
Feed Per Revolution	.011
Surface Footage	350
Max Spindle Speed	1500

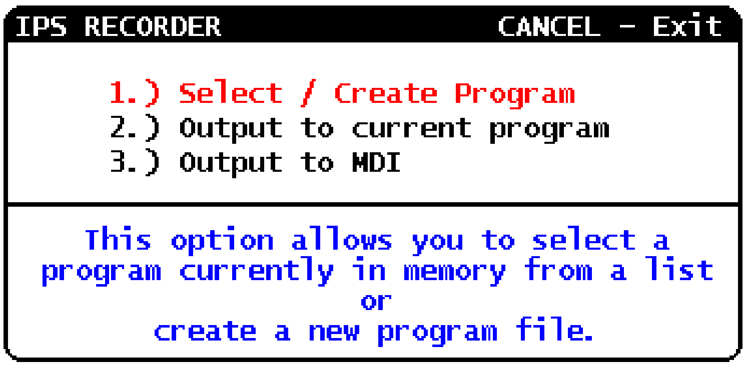
Once all the variables have been answered, make sure Tool #1 is mounted to the tool post, reduce our rapid override to 5%, and press **CYCLE START.** The turning program created will be run in the MDI mode.

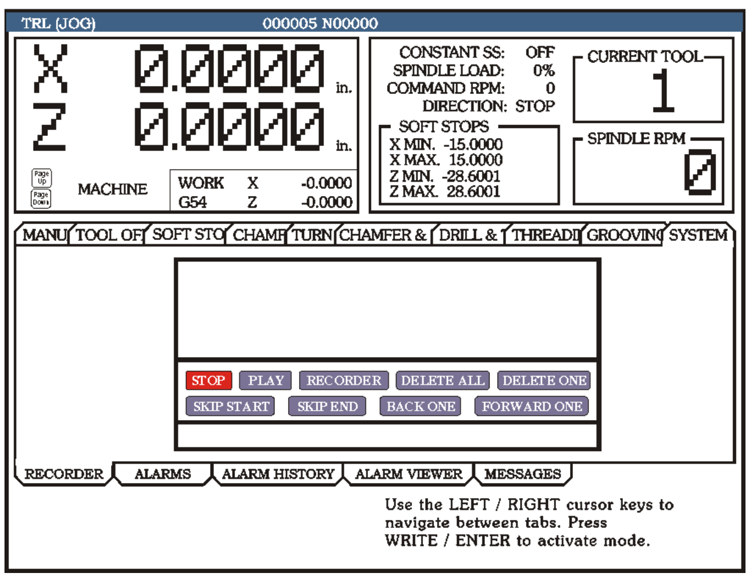
The TL Lathe will rough out from 3.000” in Diameter, to 2.500”, 2.250” back from the face of the part, then take a finish pass.

IPS Recorder Feature

Now, the next part of the IPS system that makes it unique is that the program created may be recorded into a new program or added to a program in memory.

Press the F4 key and the **IPS RECORDER** up screen will appearCoption ,press enter and , For this example program O1 was created.and **STOP** will be highlighted.





Use the right cursor key to select **RECORD** and press **F4** to escape the Recorder Screen, go back to our OD Turning Cycle, make sure the numbers are still the same, and press **CYCLE START**. The IPS will “Record” the cycle we programmed. Go back to the recorder and set the recorder to **STOP.** We have now recorded our process #1

Now we need to setup process #2, our 1.750” Diameter. Enter the OD Roughing variables into the chart below for the 1.750” OD Cycle. Now create the code with IPS. This code will be added to the bottom of program O1.

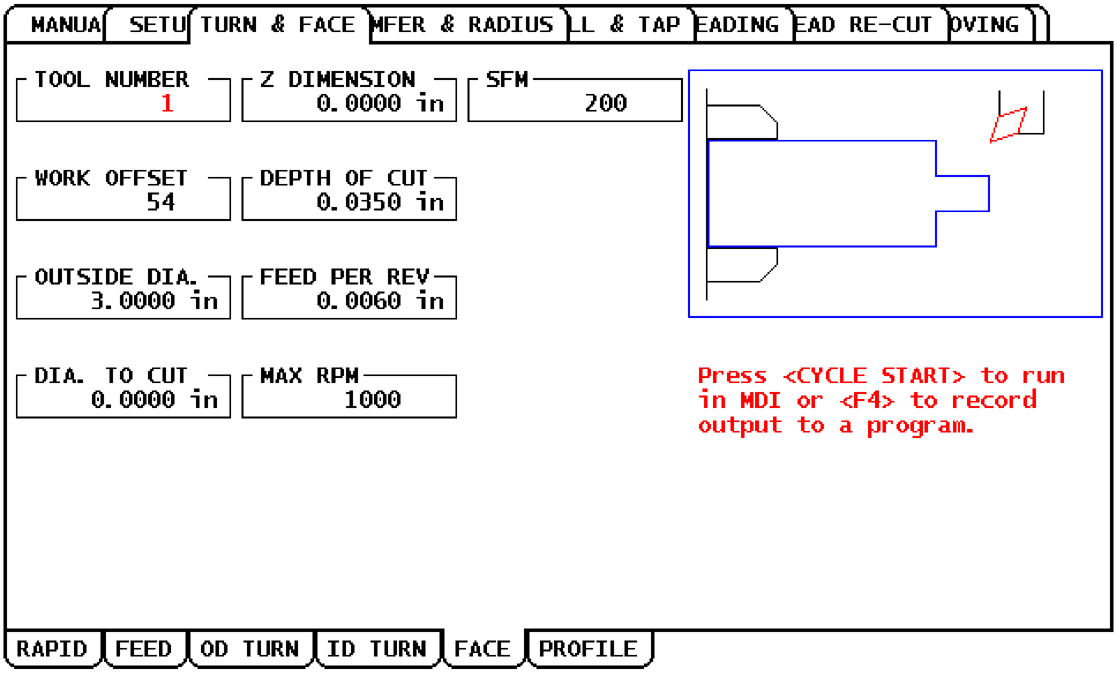
IPS OD Cycle Question	Relationship To The Part
Tool Number	
Part Offset	
Start Diameter	
Finish Diameter	
Depth Of Cut	
Z Dimension	
Feed Per Revolution	
Surface Footage	
Max Spindle Speed	

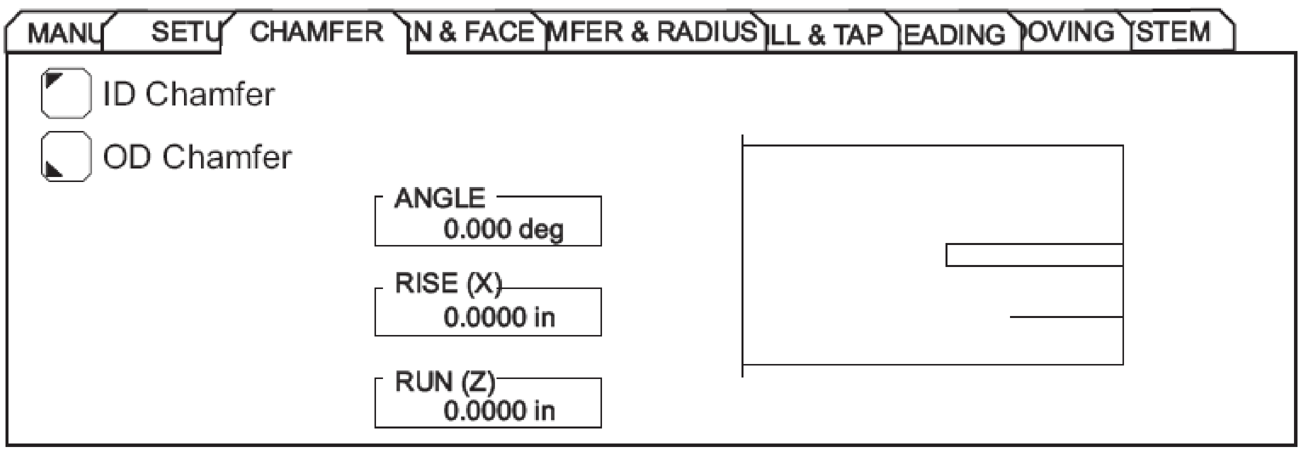
Face Cutting Cycle

The face cutting cycle is similar to the OD Cycle; it just roughs and finishes across the end of the part, then the OD. Enter the variables to face the end of our part.

IPS Face Cycle Question	Relationship To The Part
Tool Number	
Part Offset	
Start Diameter	
Finish Diameter	
Depth Of Cut	
Z Dimension	
Feed Per Revolution	
Surface Footage	
Max Spindle Speed	

Since on the first part all the tools were touched off on a cleaned up face , no material would be removed when if this cycle was run, but assuming there are multiple parts to run, one would record this feature also. This code would be the first operation in program O1.

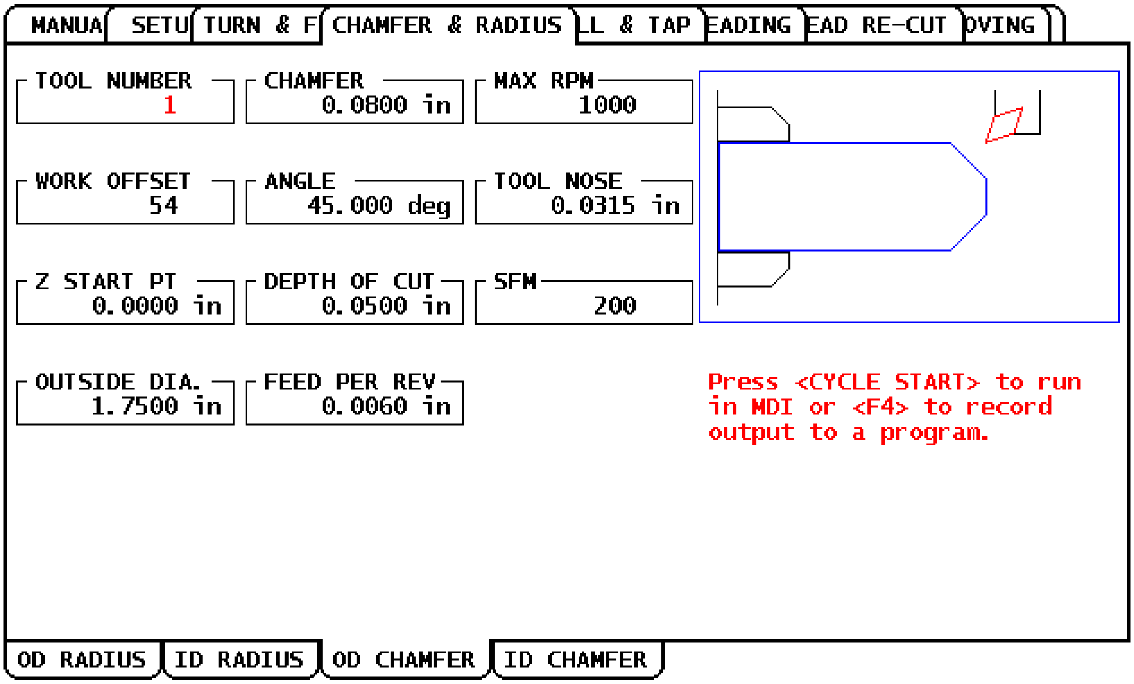




Fill out the details that are needed for the chamfer:

IPS Chamfer Cycle Question	Relationship To The Part
Tool Number	
Part Offset	
X Diameter	
Length	
Depth Of Cut	
Z Dimension	
Feed Per Revolution	
Surface Footage	
Max Spindle Speed	

The variables are entered into the IPS chamfer menu. This code will be created and recorded at the end of O1.

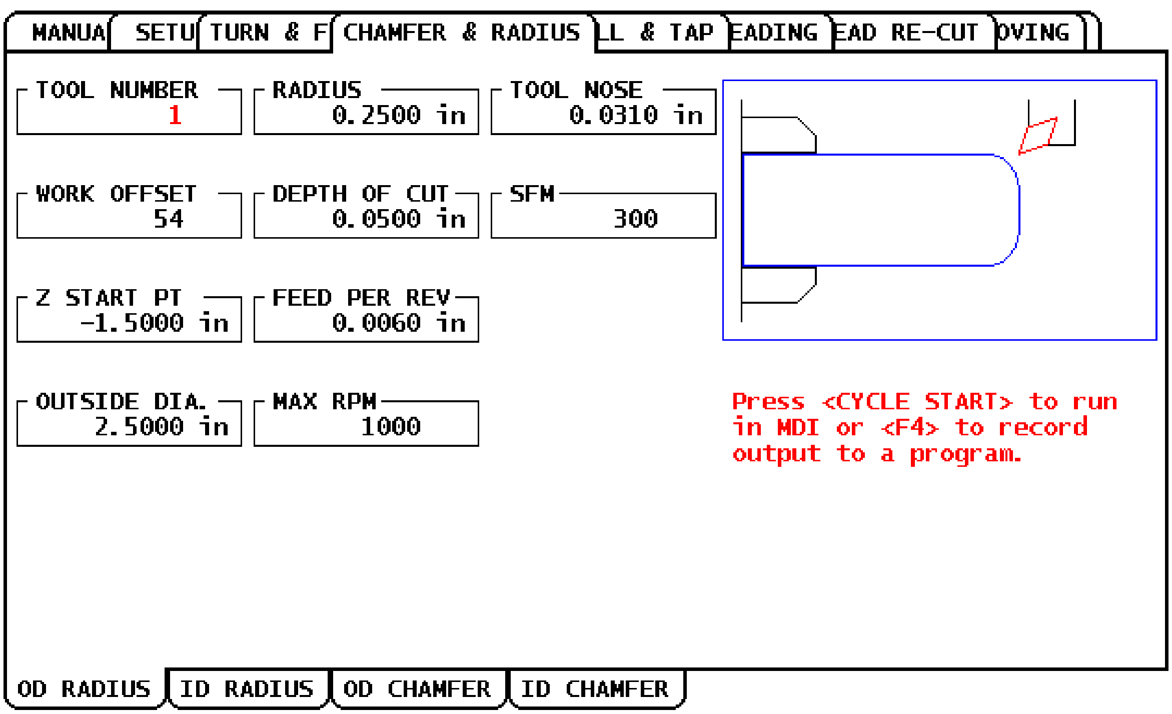


Radius Cycle Menu

Many of the IPS menus are very similar to each other; all that is need is answering questions to the control and its pictures.

Operation #5 is to turn the .250 radius on the 2.500” diameter. Navigate to the **OD RADIUS** Tab and enter the information needed below:

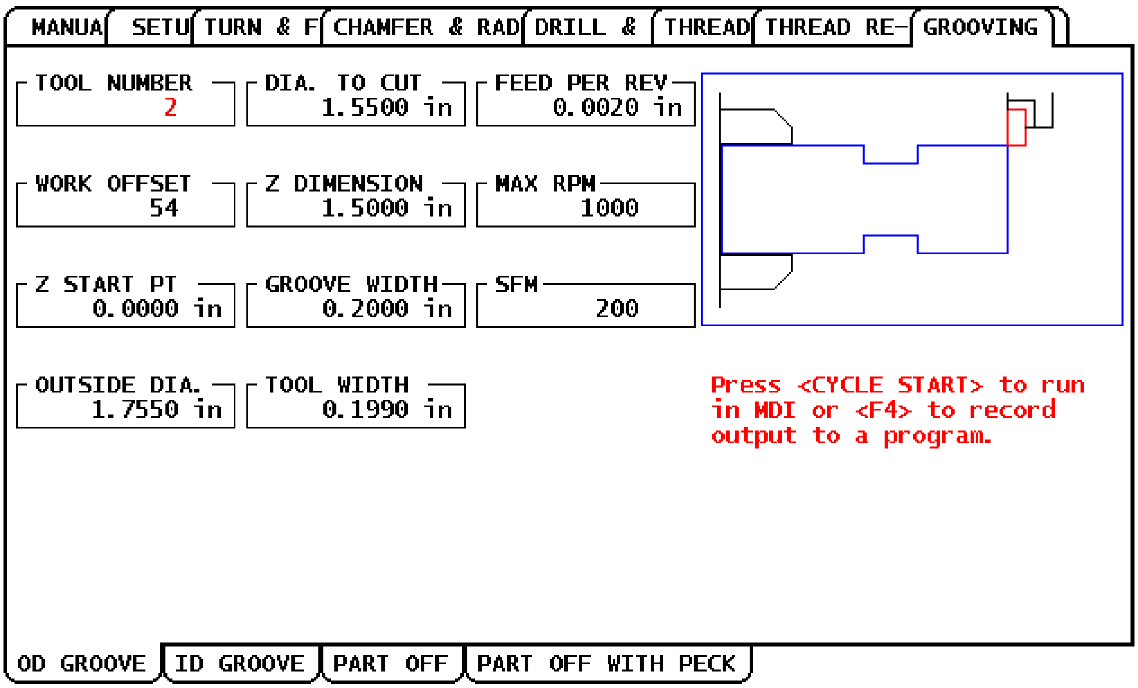
IPS OD Radius Cycle Question	Relationship To The Part
Tool Number	
Part Offset	
X Diameter	
Radius	
Depth Of Cut	
Z Start	
Feed Per Revolution	
Surface Footage	
Max Spindle Speed	



Add the code created from above to the end of Program O1.

Groove Cutting Cycle

Navigate back to the main tabs menu. Select **GROOVING**, then select **OD GROOVE** to do process #6. Input the information the IPS is asking for, verify it cuts a good part, then use the recorder again to record this feature.



IPS OD Groove Cycle Question	Relationship To The Part
Tool Number	
Part Offset	
X Start Diameter	
Z Face	
Diam. To Cut	
Groove Width	
Grooving Tool Width	
Feed Per Revolution	
Surface Footage	
Max Spindle Speed	

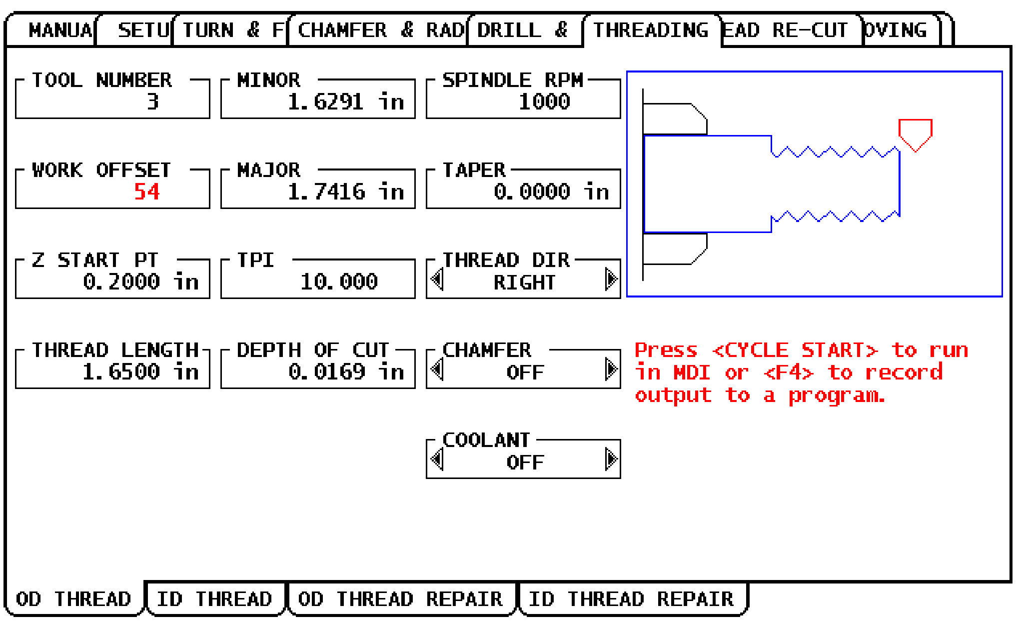
Thread Cutting Cycle

The next operation, #7, is to turn the 1.75 x 10 OD thread. Threading with IPS is simple. Navigate to the main tabs menu, and go to the **THREADING** tab, and then select **the OD THREAD** sub-tab.

The threading uses standard threading terms. Simply answer the questions the menu asks for.

IPS OD Thread Cycle Question	Relationship To The Part
Tool Number	
Part Offset	
X Start Diameter	
Z Face	
Thread Length	
Minor Diam.	
Major Diam.	
TPI	
Depth Of Cut	
Spindle RPM	

Input the answers into the threading cycle, and run the cycle to make sure we have a good thread. Once we are to size, RECORD the thread cycle with the IPS recorder.

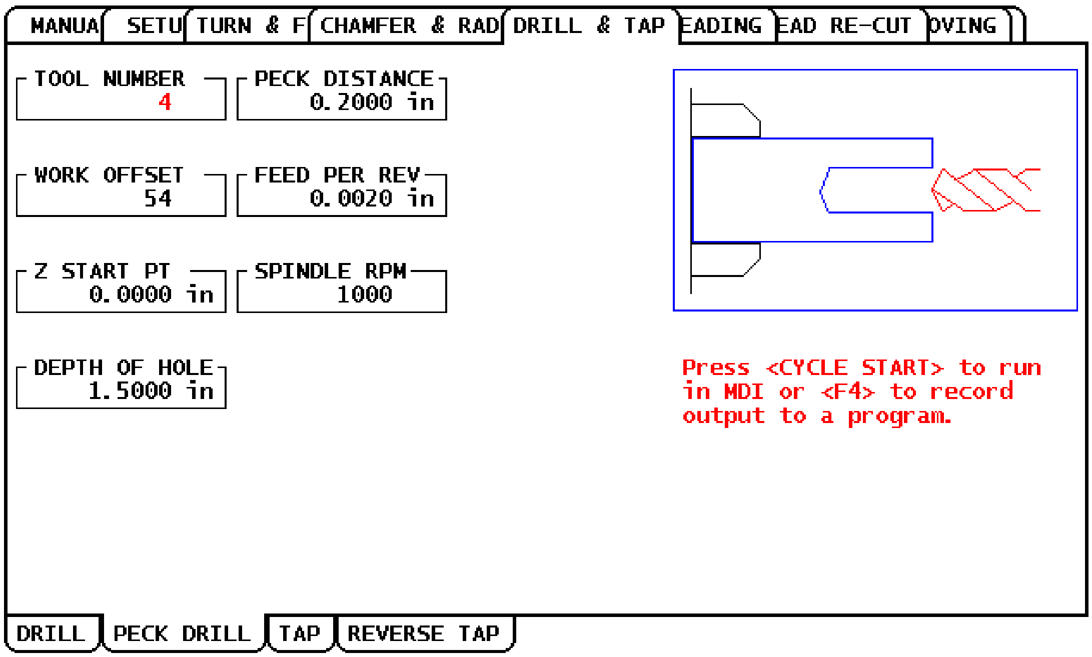


CAUTION: Once the control has started a pass of threading, FEED HOLD does not stop the machine till in-between passes. If the event of a potential crash, EMERGENCY STOP needs to be used!!

Drill Cycle

We could always use the tailstock to drill a centerline hole in the part, but with CNC capability, we can simply use the Tool Post to do so, and have an automatic drilling with pecks.

Go to the **DRILL / TAP** menu and select **PECK DRILLING** and we will get the menu below:

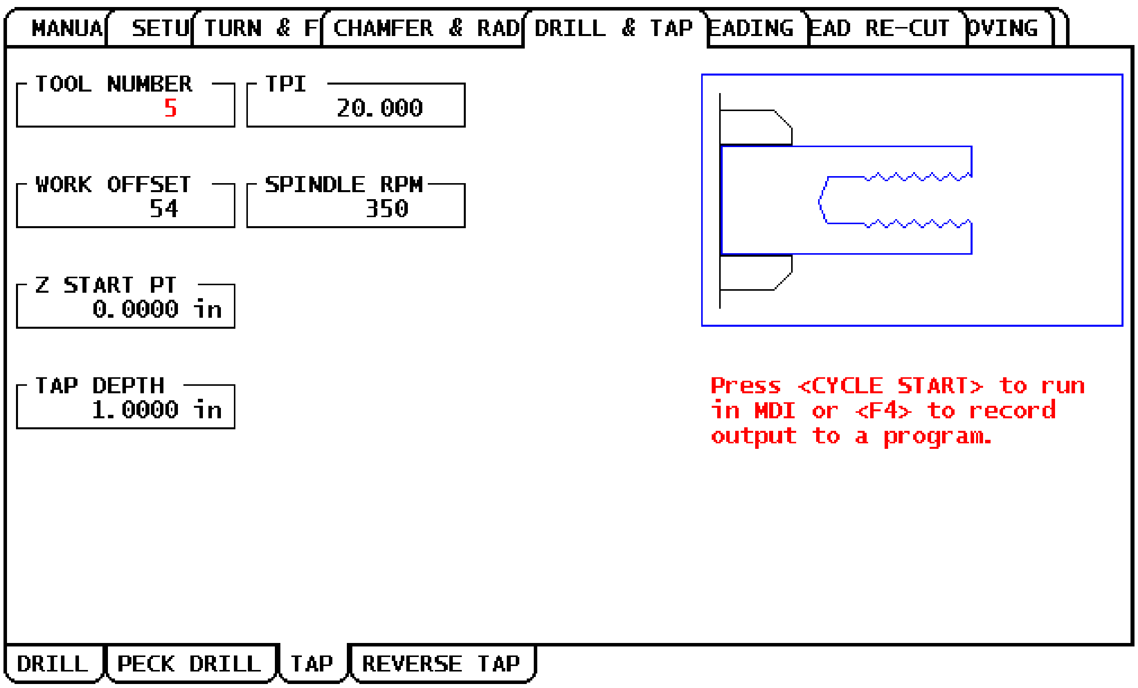


For peck drilling, just simply enter the values listed below, cycle the menu to make sure we get the correct depth, and then record it with IPS.

IPS Peck Drilling Cycle Question	Relationship To The Part
Tool Number	
Part Offset	
X Centerline	
Z Face	
Depth	
Spindle RPM	
Feed Per Rev.	
Peck Depth	

Tapped Hole Cycle

When we tap a hole with the TL Series, the **TAP CYCLE** sub tab has the same variables that the Drill Cycle uses, the only difference is that the Spindle will reverse at the bottom of the hole and back the tap out automatically.



IPS RH Tap Cycle Question	Relationship to the Part
Tool Number	
Part Offset	
X Centerline	
Z-Face	
Depth	
Spindle RPM	
TPI	

Once we have recorded all of our features of our part, we will have one complete program for the part. If we want to run another part just like it, select O1 and Cycle Start. We will have to press cycle start between every operation to change our tools when we need to.

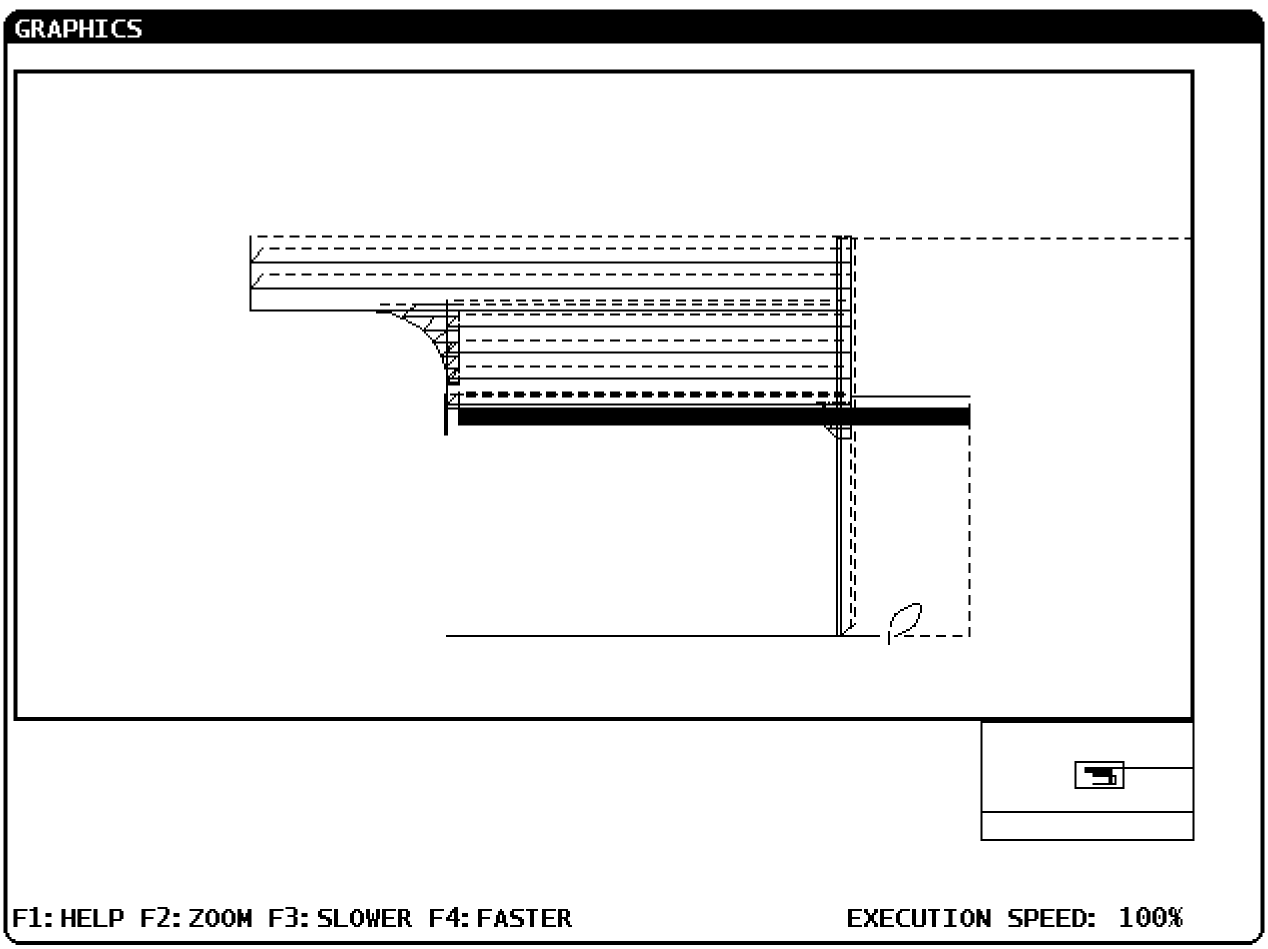
To run an old G&M program saved to memory, select the program that you want to run, load up the tools that were last used to run that program, and enter the Tool Offsets in the IPS menu just like we did before.

Go to the LIST PROG menu and select the program, press MEMORY mode on the Haas Control, and push CYCLE START.

The Haas control will accept any standard ISO G&M code program, and will understand the code, but if you are interested in learning more about G&M code programming, please contact Productivity Inc. and sign up for our Lathe Programming class.

The TL Series is built around the IPS system and is designed to take advantage of it, and thus the only reason we teach only the IPS for it's operation.

Before running a complete part run the part in the graphics mode. If there are any code structure problems, tool over travels the graphic mode will alarm out. Below is the code created for the training part run in graphics mode.



.

Below is the program created using IPS to manufacture the training part.

```
%
O00001 (TRAINING PART)

(END FACE)
T101
G54
G50 S1000
G96 S200 M03
G00 X3.05
G00 Z0.05
G72 P101 Q102 U0 W0 D0.035 F0.006
N101 G00 Z0.
G01 X0.
N102 G01 X0. Z0.05
G00 X3. Z0.
M01

(OD TURN 2.5 DIAM)
T101
G54
G50 S1500
G96 S350 M03
G00 X3.075
G00 Z0.05
G71 P101 Q102 U0 W0 D0.1 F0.011
N101 G00 X2.5
G01 X2.5 Z-2.25
N102 G01 X3.075
G00 X3.075 Z0.05
M01

(OD TURN 1.75)
T101
G54
G50 S1500
G96 S350 M03
G00 X2.575
G00 Z0.05
G71 P101 Q102 U0 W0 D0.1 F0.011
N101 G00 X1.75
G01 X1.75 Z-1.5
N102 G01 X2.575
G00 X2.575 Z0.05
M01
```

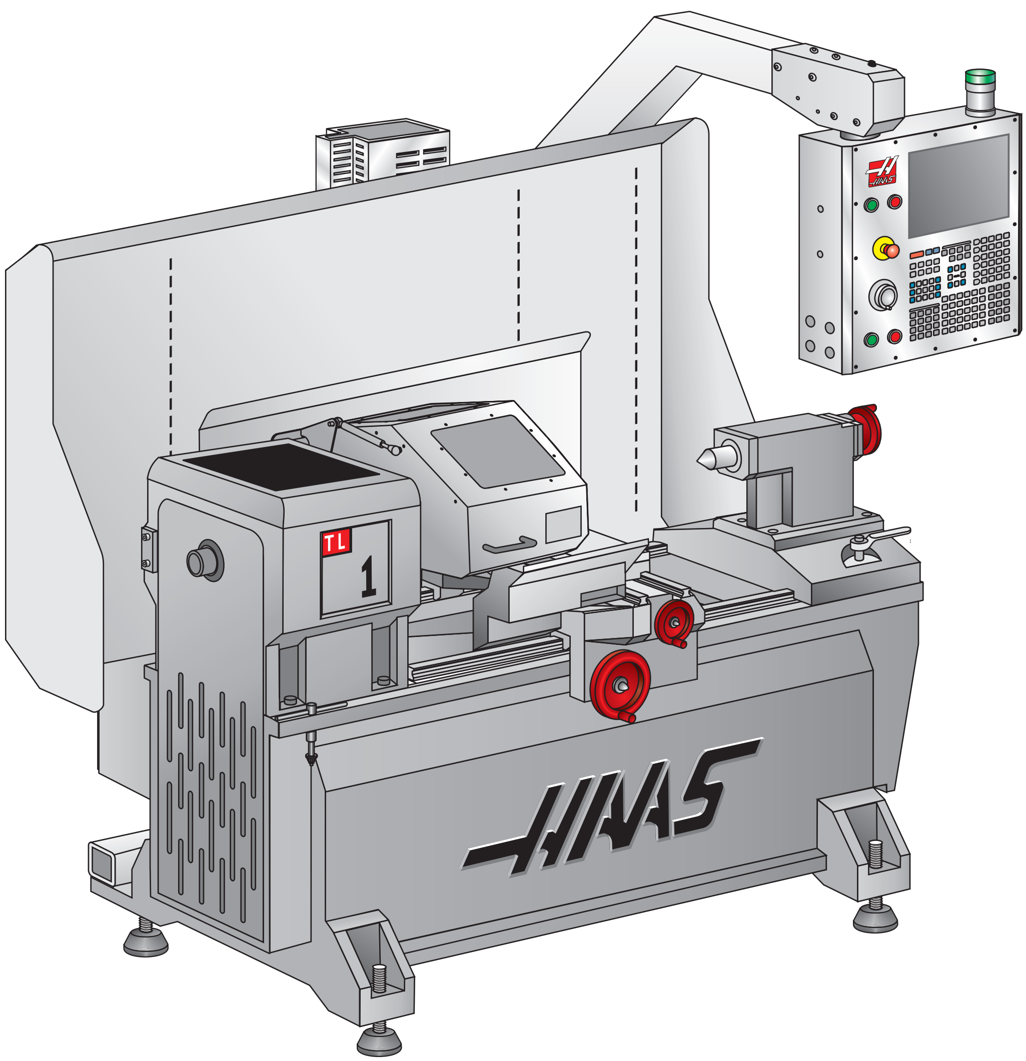
```
(OD CHAMFER)
T101
G54
G50 S1000
G96 S200 M03
G00 X1.8
G00 Z0.05
G71 P101 Q102 U0 W0 D0.05 F0.006
N101 G00 X1.527
G01 Z0.
N102 G01 X1.75 Z-0.1115
G00 X1.75 Z0.
M01

(OD RADIUS)
T101
G54
G50 S1000
G96 S300 M03
G00 X2.55
Z-1.45
G71 P101 Q102 U0 W0 D0.05 F0.006
N101 G00 X1.938
G01 Z-1.5
N102 G03 X2.5 Z-1.781 R0.281
G00 X2.55 Z-1.45
M01

(OD GROOVE)
T202
G54
G50 S1000
G96 S200 M03
G00 X1.855
G00 Z0.05
G00 X1.855 Z-1.501
G75 X1.555 Z-1.498 K0.189 F0.002
G00 X1.855
G00 Z-1.499
G01 X1.55 F0.002
G01 Z-1.5
G01 X1.855
G00 Z0.05
M01
```

```
(OD THREAD)
T303
G54
G97 S1000 M03
G00 X1.8416
Z0.5
G04 P1.
M09
M24
G76 X1.6291 Z-1.45 K0.0763 I0. D0.0169 F0.1
G00 X1.8416 Z0.5
M09
M01

(PECK DRILL)
T404
G54
G97 S1000 M03
G00 X0.
Z0.1
G83 X0. Z-1.5 Q0.2 F0.002
G80
G00 Z0.1
M01

(TAP)
T505
G54
G97 S350
G00 X0.
Z0.2
G84 X0. Z-1. R0.2 F0.05
G80
M01

M30
%
```

Section II – IPS Walk-Through for Lathes

Haas ES Doc #ES0609

Intuitive Programming System Walk-Through For Lathes



Introduction

These instructions are an in-depth look at each of the Intuitive Programming System (IPS) menus. A more formal description is given for each of the entries to help better define the on-screen help for new users.

These instructions are to be used with the Lathe Operator's Manual (96-8700) and Toolroom Lathe Operator's Addendum (96-0112)

The menus are navigated by using the left and right arrow keys. To select a menu item, press Write/Enter. Some menus have sub-menus, in which case, use the left and right arrow keys and press Write/Enter to select a sub-menu.

Use the arrow keys to navigate through the variables, enter values by using the number pad, and then press Write/Enter.

To exit, or go back to another selection, press Cancel.

Pressing any of the buttons under the "Display" heading will exit the IPS menus, as will any of the mode keys (i.e. Edit, Mem, MDI, etc.). To return to the IPS menu, press Handle Jog.

A representation of the machine keypad is included at the end of this document for reference.

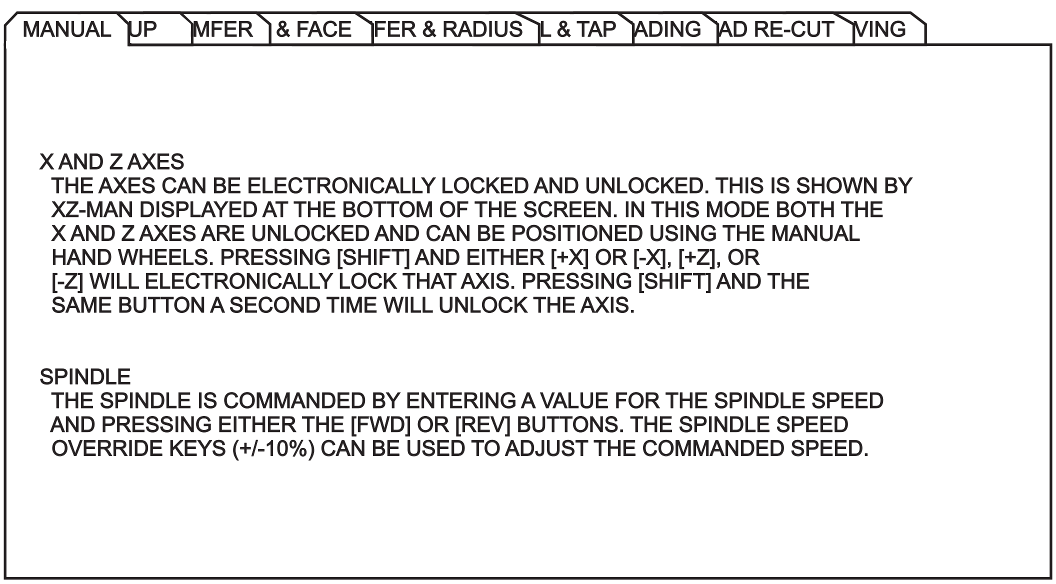
This guide will help the user develop full CNC programs by means of the IPS screen. Note that a program entered through the Toolroom Lathe screens is also accessible by going to full CNC MDI mode. The program can be edited and saved from the full CNC mode.

NOTE: The IPS menu is displayed at power up, and is available in the following configurations:

- 10" LCD and software version 7.xx and earlier - IPS
- 15" LCD and software version 8.03 and earlier - IPS (upgradable to Profile Creator)
- 15" LCD and software version 8.04A and later - IPS with Profile Creator

Manual Mode

Power on the machine and press RESET until all alarms have cleared. Press POWER ON/RESTART to zero the machine. The IPS menu can now be accessed by pressing MDI DNC, then pressing PRGRM CONVRS. Press WRITE/ENTER to display the IPS menu MANUAL tab.



X and Z Axes

Just below the on-screen text is a line of text that shows the state the lathe is in. For example, "X -MAN" means the X -axis is in manual mode. No text beneath the on-screen help means both axes (X and Z are locked. In this case the axes can be jogged, by pressing +X/-X or +Z/-Z and then using the jog handle on the pendant. Select a jog speed before using the jog handle.

Spindle

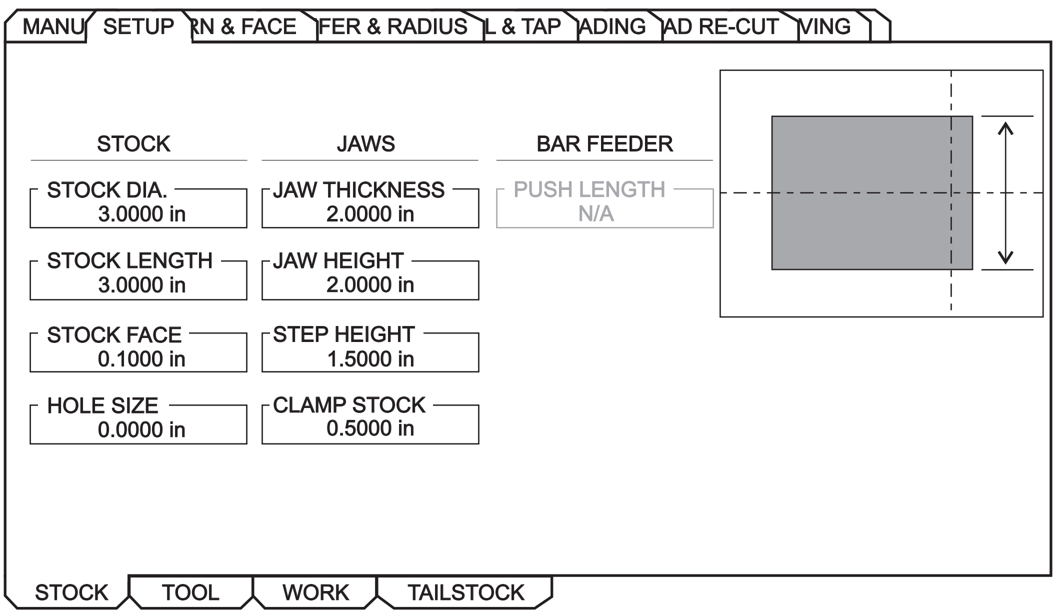
The spindle is controlled using the keys on the control pendant. Enter a spindle speed; for example, press 5, then 0, then Write/Enter. This will enter a speed of 50RPM. Ensure the area around the spindle is free of tools and workpieces, press the hold to run switch and then press either the FWD or REV button.

The spindle speed override keys (+/- 10%) can be used to adjust the commanded speed. This also works on most screens.

The spindle is stopped by letting go of the hold to run switch, pressing Reset, or the pressing the Stop button.

Setup

Stock Setup



Stock Dia. – Controls the diameter of the raw part that will be displayed in live image.

Stock Length – Controls the length of the raw part that will be displayed in live image.

Stock Face – Controls the Z stock face of the raw part that will be displayed in live image.

Hole Size – Controls the stock hole of the raw part that will be displayed in live image.

Jaw Thickness – Controls the thickness of the chuck jaws that will be displayed in live image.

Jaw Height – Controls the height of the chuck jaws that will be displayed in live image.

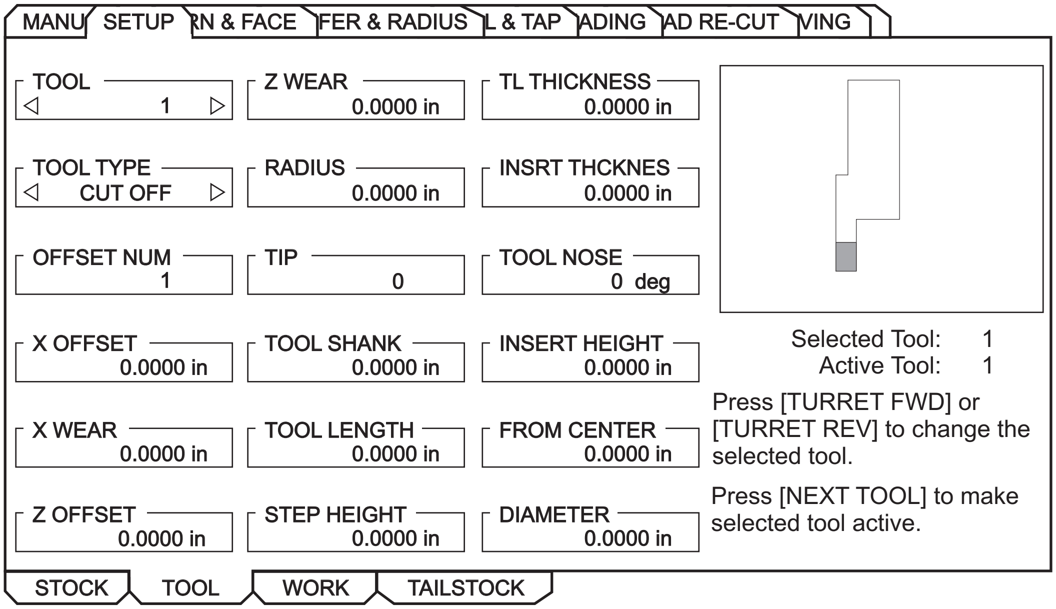
Step Height –

Clamp Stock – Controls the clamp stock size of the chuck jaws that will be displayed in live image.

Push Length –

Tool Offsets

Tool offsets are described in detail in the Operator's manual. See the "Tool Nose Compensation" section within the "Operation/Programming" Tab for specific instructions on Radius, Radius Wear, Taper, and Tip.



Tool – The current tool number. Use the turret FWD/REV or the Next Tool buttons to set-up another tool.

Tool Type – Right/Left arrows select among 16 tool types: Drill, Tap, Vert Tap, Vert Drill, End Mill, V End Mill, Ballnose, V Ballnose, OD Turn, ID Bore, OD Groove, ID Groove, Face Groove, OD Thread, ID Thread and Cut Off.

Offset Num –

X Offset – The X axis offset for the current tool. Press X Dia Meas to record this position.

X Wear – The amount of tool wear, in the X axis for the current tool.

Z Offset – The Z axis offset for the current tool. Press Z Face Meas to record this position.

Z Wear – The amount of tool wear, in the Z axis for the current tool.

Radius** – The tip radius of the current tool.

Tip** – Tool tip direction will be a value of 0-9. Must be entered to use Cutter Compensation

Tool Shank –

Tool Length –

Step Height –

TL Thickness –

Tool Nose – The nose radius of the current tool.

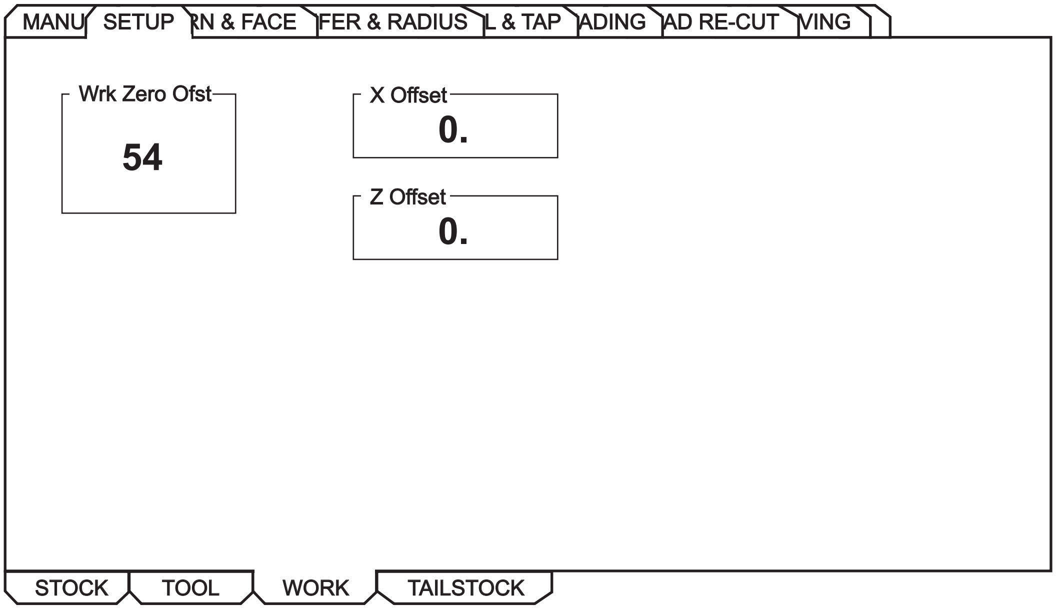
Insert Height –

From Center –

Diameter – Compensation value for part deflection.

**Must be entered to use Cutter Compensation; See the Operator's manual for information on Cutter Compensation.

Work Offsets

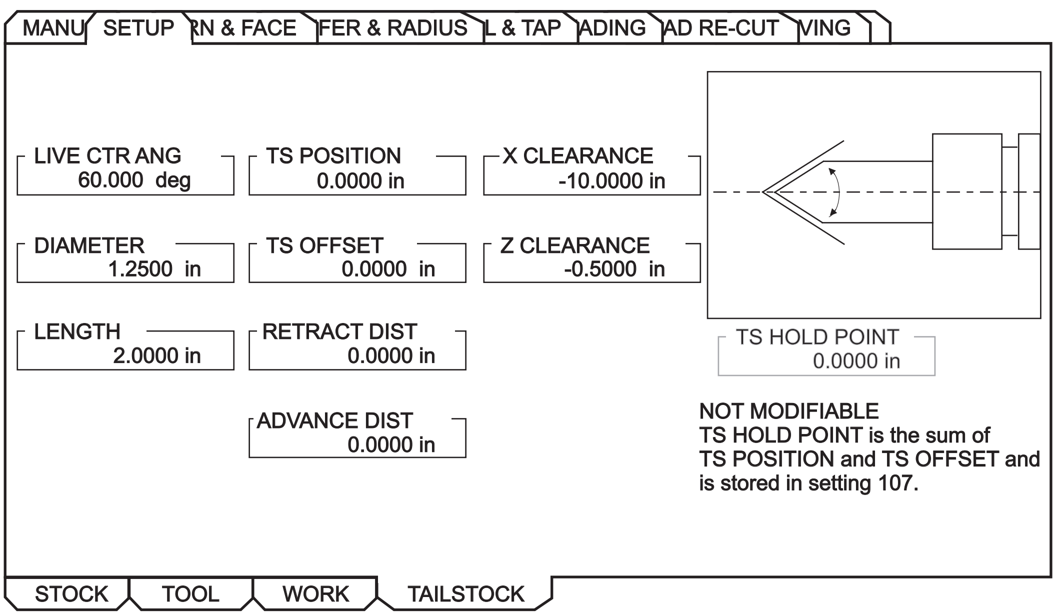


Work Zero Offset – Press the Up and Down arrows to change the displayed Work Zero Offset.

X Offset – Press Write to add or F1 to set position. Enter a value and either press Write to add the value to the current value, or F1 to replace the value with the entered value.

Z Offset – Press Write to add F1 to set or Part Zero Set to record current position. Enter a value and either press Write to add the value to the current value, or F1 to replace the value with the entered value. Press Part Zero Set to record the Z Offset current position.

Tailstock Setup



Live Center Angle – Controls center angle of tailstock.

Diameter – Controls the diameter of the tailstock.

Length – Controls the length of the tailstock.

TS Position –

TS Offset –

Retract Dist – The distance from the Hold Point (Setting 107) the tail stock will retract when commanded. This setting should be a positive value. Press Write to add F1 to set or Part Zero Set to record current position. Enter a value and either press Write to add the value to the current value, or F1 to replace the value with the entered value. Press Part Zero Set to record the Z Offset current position.

Advance Dist – When the tail stock is moving toward the Hold Point (Setting 107), this is the point where it will stop its rapid movement and begin a feed. This setting should be a positive value. Press the Up and Down arrows to change the displayed Work Zero Offset.

X Clearance – Works with Z Clearance to define a tail stock travel restriction zone that limits interaction between the tail stock and the tool turret. It determines the X-axis travel limit when the difference between the Z-axis location and the tail stock location falls below the value in Z Clearance. If this condition occurs and a program is running, an alarm is generated. When jogging, no alarm is generated, but travel is limited. Press Write to add or F1 to set position. Enter a value and either press Write to add the value to the current value, or F1 to replace the value with the entered value. When highlighting X CLEARANCE, pressing X DIA MEAS takes the value of the X axis and places it in X CLEARANCE. Pressing ORIGIN when highlighting X CLEARANCE sets clearance to max travel.

Z Clearance – Minimum allowable difference between the Z-axis and the tail stock. A value of -1.0000 means that when the X-axis is below the X clearance plane, the Z-axis must be more than 1 inch away from the tail stock position in the Z-axis negative direction. The default value for this setting is zero. Press Write to add F1 to set or Part Zero Set to record current position. Enter a value and either press Write to add the value to the current value, or F1 to replace the value with the entered value. Press Part Zero Set to record the Z Offset current position. When highlighting Z CLEARANCE, pressing Z FACE MEAS takes the value of the Z axis and places it in Z CLEARANCE. Pressing ORIGIN when highlighting Z CLEARANCE sets clearance to zero.

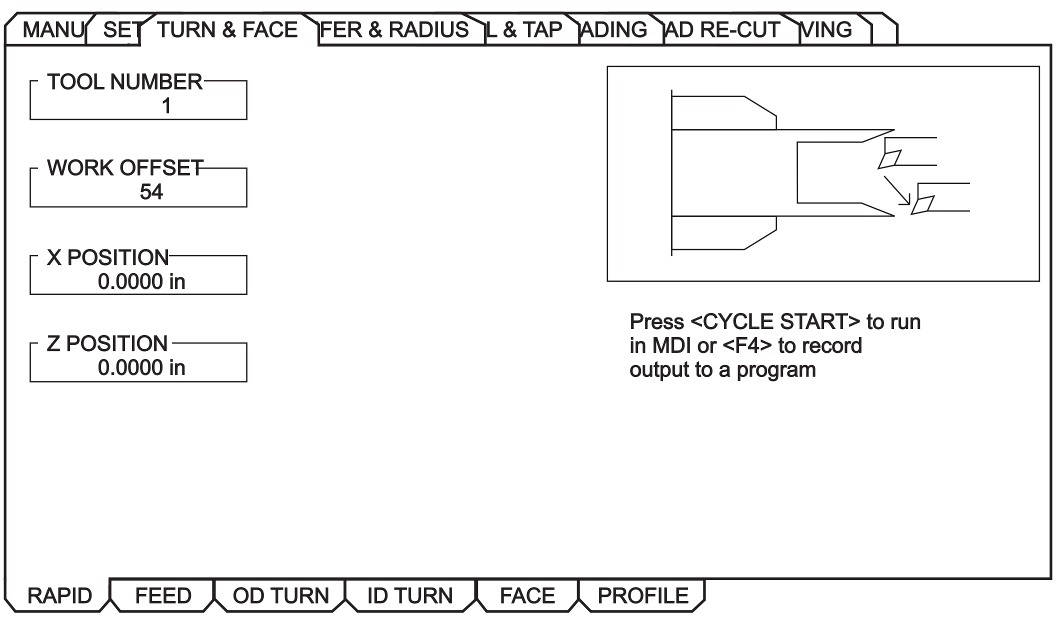
Automatic Mode

On each of the following interactive screens, the user is asked to enter data needed to complete common machining tasks. When all data has been entered, press “Cycle Start” to begin the machining process.

The following are examples of the types of the Automatic Mode screens and the definitions of the variables that will need to be entered.

Turn and Face - Rapid

This mode is for making a rapid move.



Tool Number – Enter the tool to be used.

Work Offset – Enter the work offset to be used.

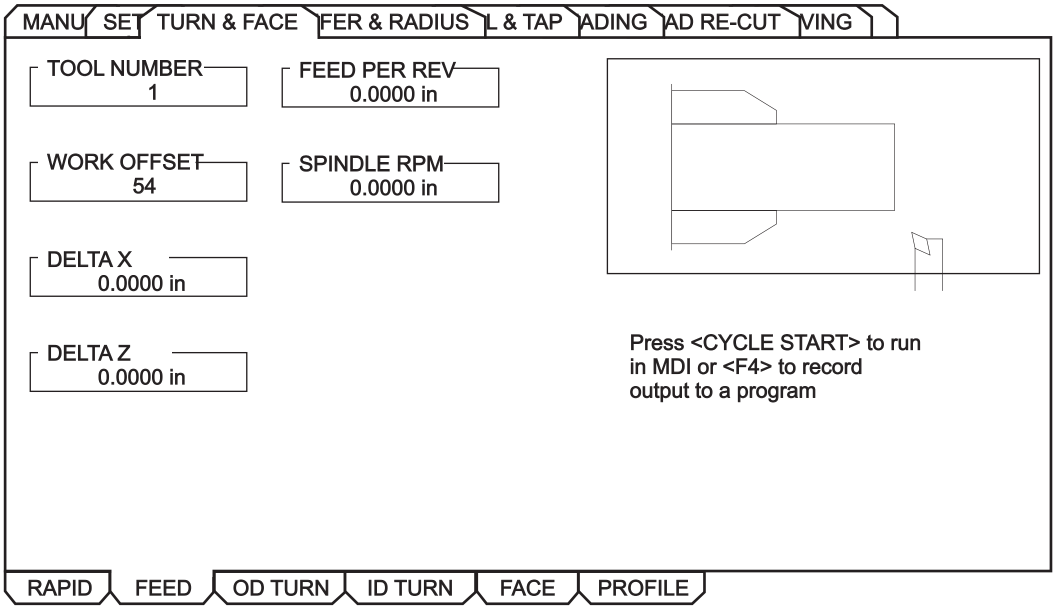
X Position – Enter end point or move tool to end point desired. Press X DIA. MEAS to record this position.

Z Position – Enter end point or move tool to end point desired. Press Z FACE MEAS to record this position.

Turn and Face - Feed

This mode provides for straight line (linear) motion from the machines current position to the specified ‘X’ and ‘Z’ end points.

NOTE: The Feed command is a single pass movement for features smaller than the maximum cut depth for the tool. For larger features use the turn and face programs.



Tool Number – Enter the tool to be used.

Work Offset – Enter the work offset to be used.

Delta X – Enter the X-coordinate of the end point of the linear motion desired.

Delta Z – Enter the Z-coordinate of the end point of the linear motion desired.

***Feed Per Rev.** – Enter the feed per revolution (in inches or millimeters).

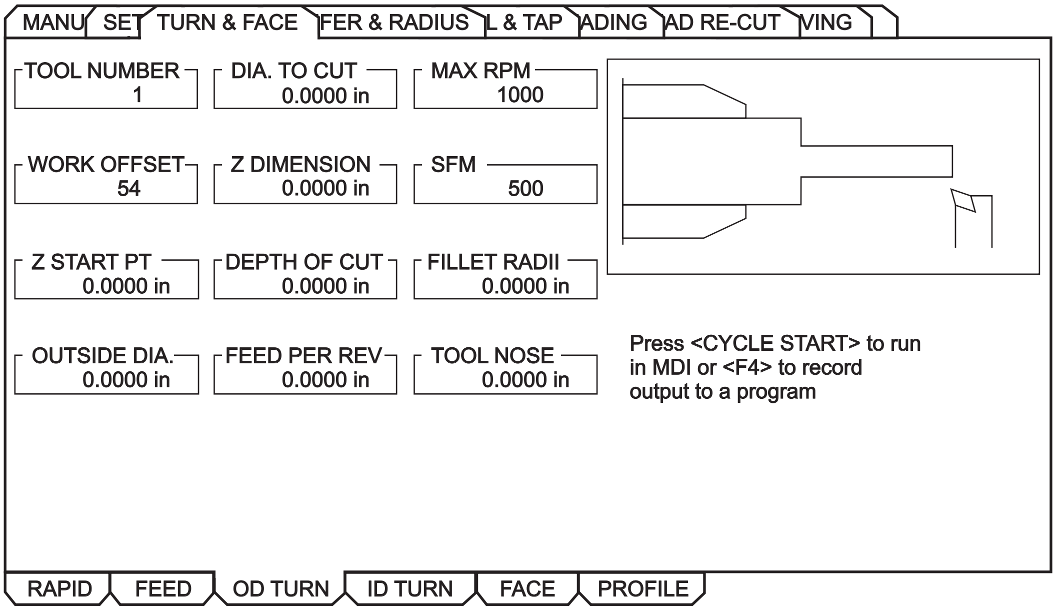
***Spindle RPM** – Enter the spindle RPM.

*Mandatory Values

Advanced Users: In the full CNC mode this is a G01 command.

Turn & Face - OD Turn

This mode is for an outside diameter cut.



Tool Number – Enter the tool to be used.

Work Offset – Enter the work offset to be used.

Z Start Pt – Enter the Z axis starting point.

Outside Dia. – Enter the current diameter of the work piece. Manually measure the diameter.

Dia. to Cut – Enter the finished diameter.

Z Dimension – Enter the Z axis dimension of the part from the Z start point.

Depth of Cut – Enter the depth of cut for each pass of the stock removal.

Feed Per Rev – Enter the feed per revolution.

MAX RPM – Enter the maximum spindle turning speed.

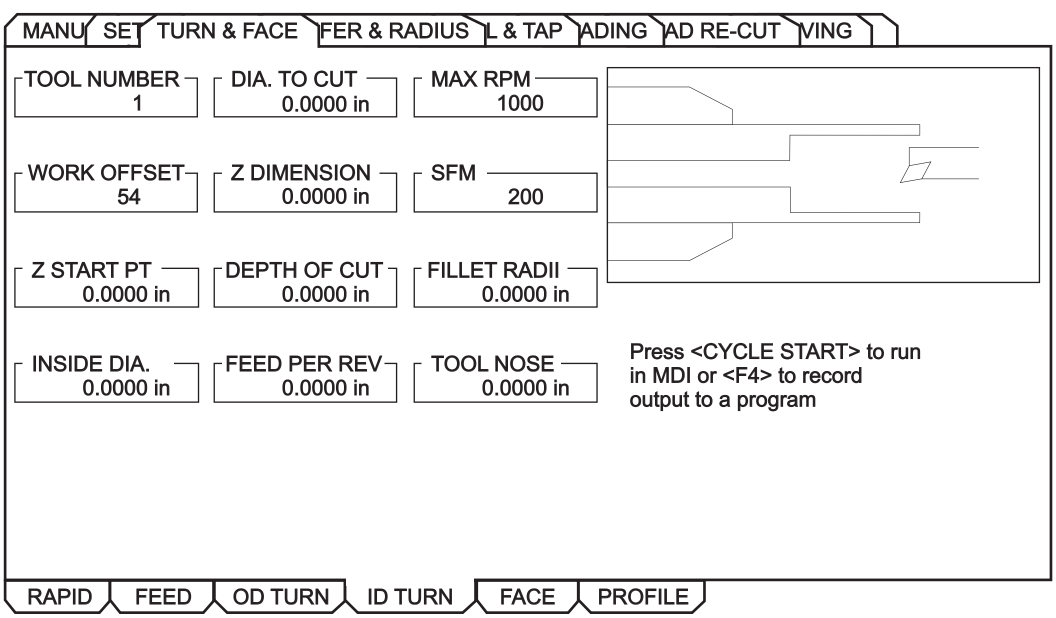
SFM – Enter the Surface Feed per Minute.

Fillet Radii – Enter the corner fillet radii or enter ‘0’ for none.

Tool Nose – Enter the tool nose radius.

Turn & Face - ID Turn

This mode is for an inside diameter cut.



Tool Number – Enter the tool to be used.

Work Offset – Enter the work offset to be used.

Z Start Pt – Enter the Z axis starting point.

Inside Dia. – Enter the current diameter of the work piece. Manually measure the diameter.

Dia. to Cut – Enter the finished diameter.

Z Dimension – Enter the Z axis dimension of the part from the Z start point.

Depth of Cut – Enter the depth of cut for each pass of the stock removal.

Feed Per Rev – Enter the feed per revolution.

MAX RPM – Enter the maximum spindle turning speed.

SFM – Enter the Surface Feed per Minute.

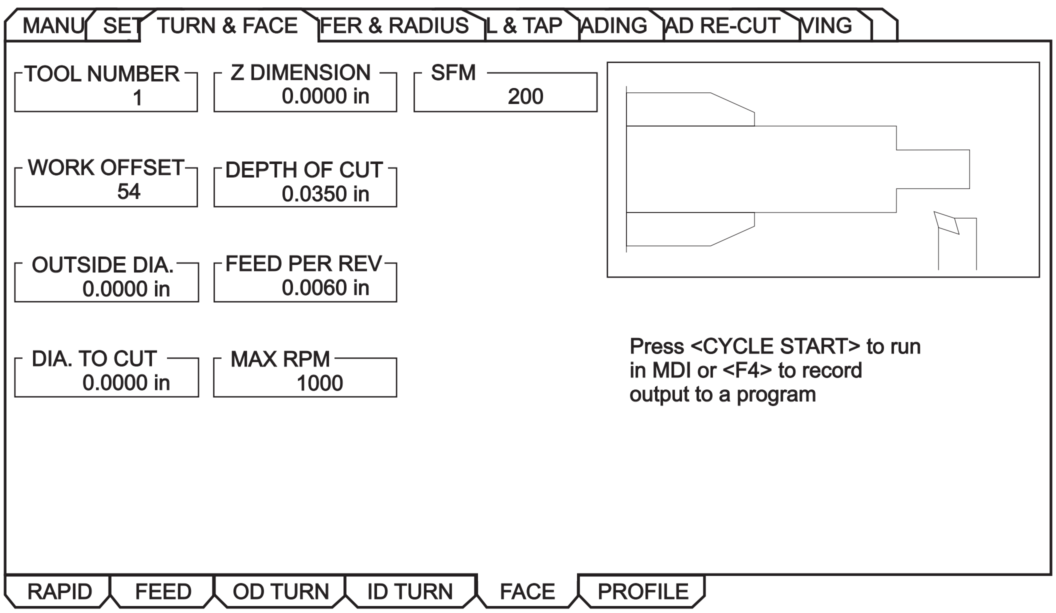
Fillet Radii – Enter the corner fillet radii or enter ‘0’ for none.

Tool Nose – Enter the tool nose radius.

Advanced Users: In the full CNC mode this is a G71 command.

Turn & Face - Face

This mode is for making an end facing cut.



Tool Number – Enter the tool to be used.

Work Offset – Enter the work offset to be used.

Outside Dia. – Enter the current diameter of the work piece. Manually measure the diameter.

Dia. to Cut – Enter the finished diameter.

Z Dimension – Enter the Z axis dimension of the part from the Z start point.

Depth of Cut – Enter the depth of cut for each pass of the stock removal.

Feed per Rev – Enter the feed per revolution. This is the distance the tool will move for each revolution of the spindle.

MAX RPM – Enter the maximum spindle turning speed.

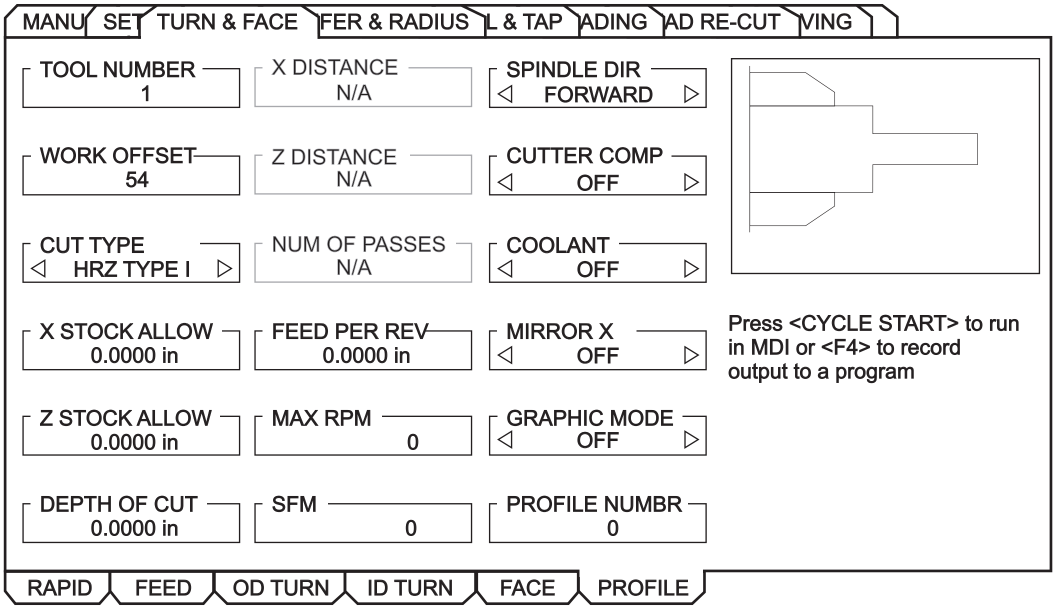
SFM – Enter the Surface Feed per Minute.

Advanced Users: In the full CNC mode this is a G72 command.

NOTE: Entering a negative value for “Dia to Cut” causes the tool to pass spindle center and machine the entire face of the part; **Do Not** enter a value larger than -.100”.

Turn & Face - Profile

This tab is only available if the machine has a control pendant with a 15” screen and lathe software version 8.04A or later.



Tool Number – Enter the tool to be used.

Work Offset – Enter the work offset to be used.

Cut Type – Use the left/right cursor keys to select the type of cut (Horizontal, Vertical, Profile, Finish Fwd, Finish Rev).

X Stock Allow – Enter the amount to leave on the diameter of the profile.

Z Stock Allow – Enter the amount to leave on the faces of the profile.

Depth of Cut – Enter the depth of cut for each pass of the stock removal.

Num of Passes – Enter the number of cutting passes. (Must be a positive number).

X Distance – Enter the X-axis distance and direction from first cut to last. (Radius value).

Z Distance – Enter the Z-axis distance and direction from first cut to last.

Feed Per Rev – Enter the feed per revolution.

MAX RPM – Enter the maximum spindle turning speed.

SFM – Enter the Surface Feed per Minute.

Spindle Dir – Use the left/right cursor keys to select spindle direction (Forward/Reverse). This depends on tool type.

Cutter Comp – Use the left/right cursor keys to select cutter compensation (Off/Left/Right).

Coolant – Use the left/right cursor keys to turn coolant on or off (On/Off).

Mirror X – Use the left/right cursor keys to mirror the X axis (On/Off). This allows you to cut on the other side of the part.

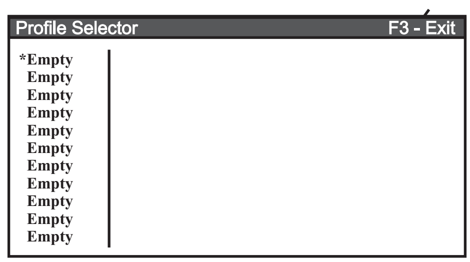
Graphic Mode – Use the left/right cursor keys to turn on/off Graphic Mode. This allows you to view the process in graphics.

Profile Number – Enter the number of the profile to use, press Enter to open Profile Select or press F1 key.

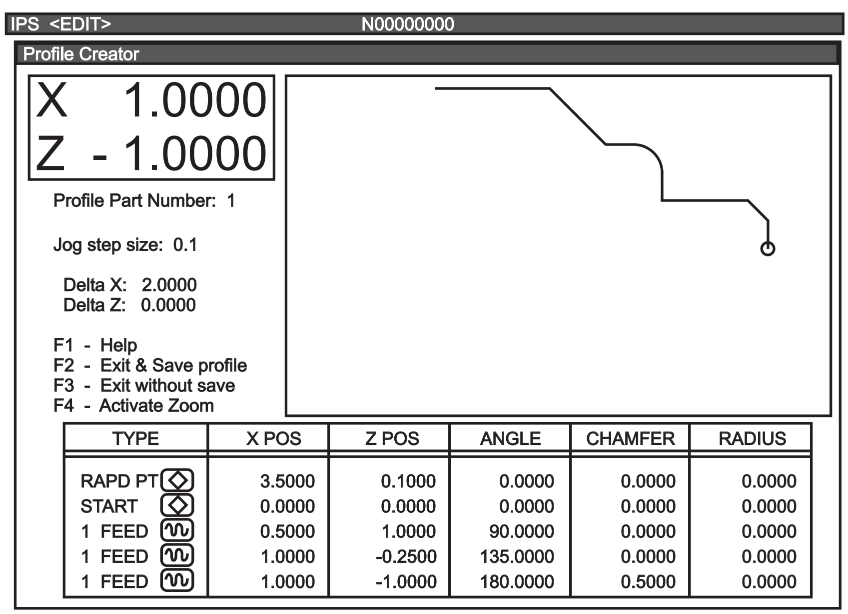
Advanced Users: In the full CNC Mode, this is a G71 command.

Basic Profile Creation (Example)

1. Start the machine with IPS active.
2. Clear any alarms, then press Power Up/Reset to zero the machine.
3. Select the Setup tab, then the Work tab to set up the work offsets.
4. Select the Tool tab (under the Setup tab) to set up the tools to be used.
5. Press Cancel a few times to get out of the Setup tab. Select the Turn & Face tab, then the Profile tab.
6. Enter the tool number (1), set Cut Type to Horizontal, set X Stock Allow to 0.02, set Z Stock Allow to 0.005, set Depth of Cut to 0.075, set Feed per Rev to 0.01, set Max RPM to 1500, set SFM to 350, set Graphic Mode to ON and Profile Number to 1.
7. Select the Profile Number data box and press Write/Enter or press F1 when in the Profile tab. A Profile Selector popup window is displayed. The Profile Selector popup is used to select a profile, alter an existing profile, choose a storage location for a new profile or delete a profile.



8. Select an ‘Empty’ slot and press Write/Enter to display the Profile Creator screen. This is used to draw a profile on the screen using either the jog handle or entering data directly into the table.



TYPE	X POS	Z POS	ANGLE	CHAMFER	RADIUS
RAPD PT	3.5000	0.1000	0.0000	0.0000	0.0000
START	0.0000	0.0000	0.0000	0.0000	0.0000
1 FEED	0.5000	1.0000	90.0000	0.0000	0.0000
1 FEED	1.0000	-0.2500	135.0000	0.0000	0.0000
1 FEED	1.0000	-1.0000	180.0000	0.5000	0.0000

Profile Creator Screen Hot keys:

F1 – Help screen popup. Lists available keys used in the Profile Creator along with a short description of each key's function.

F2 – Saves the profile on the screen, exits the Profile Creator screen and transfers control back to the Profile tab.

F3 – Exits the Profile Creator screen and transfers control back to the Profile tab screen. Does not save the profile's data.

F4 – Activates and de-activates the zoom and scrolling feature.

INSERT – Inserts a line into the table. This feature will not work if the table is full (all 30 lines used).

ORIGIN – Clears all data in the table.

X JOG KEY – Jumps to the X-axis position in the data table for the currently selected row.

Z JOG KEY – Jumps to the Z-axis position in the data table for the currently selected row.

CURSOR KEYS – Moves around in the data table. If zoom is active, the cursor keys move the part around on the screen.

(.0001), (.001), (.01), (.1) – Changes the jog step size while drawing in the graphic window.

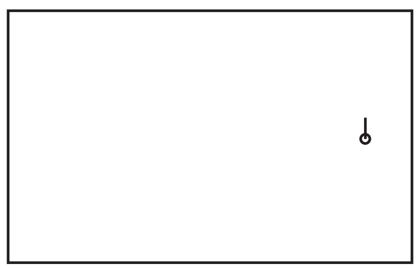
To Build the Profile Shown:

a. Select the Rapd Pt row. Use the arrow keys to select the X POS column and enter 3.5. Use the arrow keys to select the Z POS column and enter 0.1. Use the arrow keys to go to the beginning of the Start row.

b. Leave the Start PT at X0 Z0. Use the arrow keys to go to the beginning of the next line in the table (4 None). Press 1 to activate a Feed move.

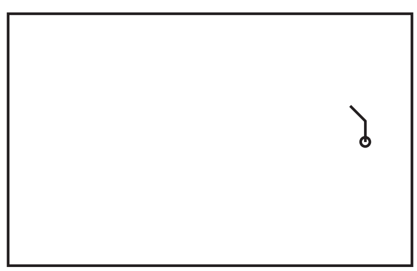
c. Jog X POS to 0.5 by turning the handwheel clockwise, and press Write/Enter.

NOTE: Each handwheel click either increments or decrements the position by 0.1".



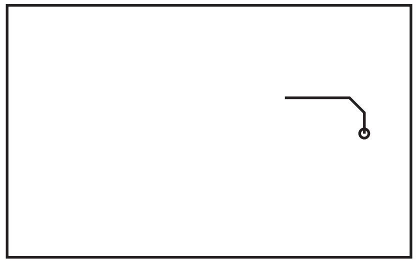
Use the arrow keys to go to the beginning of the next line in the table and press 1 to activate a Feed move.

d. Press Write/Enter until X POS is selected. Jog X to 1.0 and press Write/Enter until Z POS is selected. Jog Z to -0.25 and press Write/Enter.



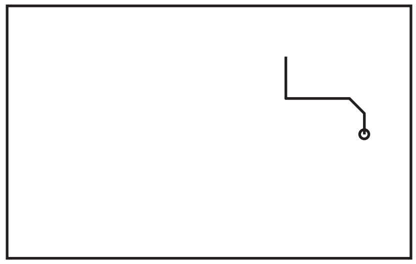
Go to the beginning of the next line. Press 1 to activate a Feed move.

e. Press Write/Enter until Z POS is selected. Jog Z to -1.0 and press Write/Enter.



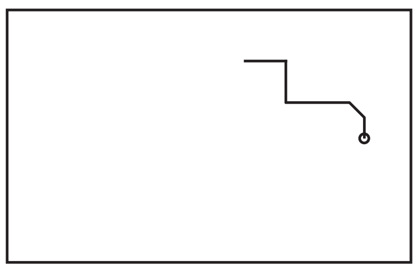
Go to the beginning of the next line. Press 1 to activate a Feed move.

f. Press Write/Enter until X POS is selected. Jog X to 2.0 and press Write/Enter.

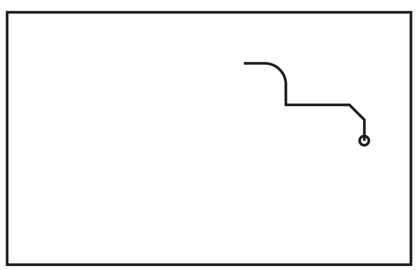


Go to the beginning of the next line. Press 1 to activate a Feed move.

g. Press Write/Enter until Z POS is selected. Jog Z to -1.5 and press Write/Enter.

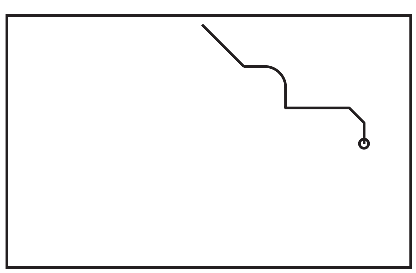


Use the cursor keys to go back to the previous line and select the Radius column. Enter 0.25, press Write/Enter and use the cursor keys to come back to this line.



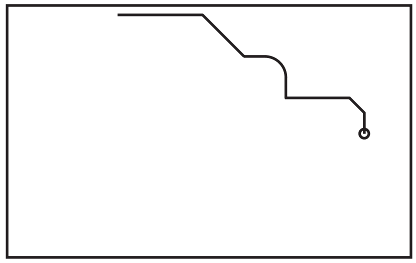
Go to the beginning of the next line. Press 1 to activate a Feed move.

h. Press Write/Enter until X POS is selected. Jog X to 3.0. Press Write/Enter until Z POS is selected. Jog Z to -2.0 and press Write/Enter.



Go to the beginning of the next line. Press 1 to activate a Feed move.

i. Press Write/Enter until Z POS is selected. Jog Z to -3.0 and press Write/Enter.



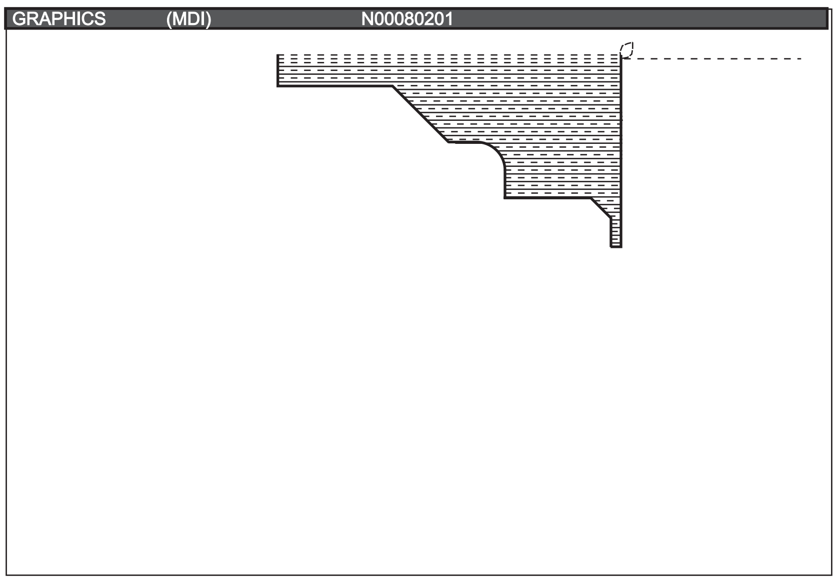
j. Press F2 to Save and exit the Profile Creator.

k. Press Cycle Start to cut the profile.

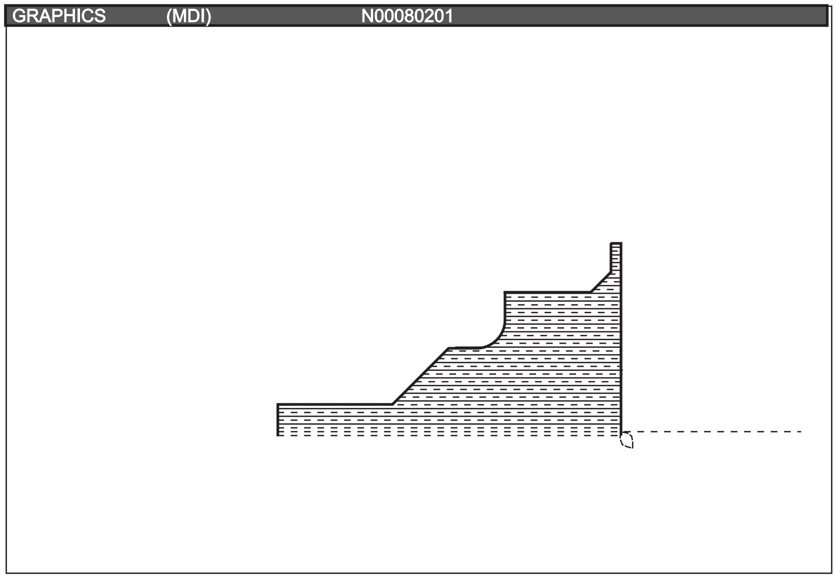
NOTE: The program may be saved to memory from MDI by typing in 0xxxxx and pushing the Alter key. This action moves the program from MDI memory.

If Graphic Mode is set to ON in the Turn & Face Profile screen, when Cycle Start is pressed to run a profile, a graphic screen is displayed showing the graphical representation of the profile.

Graphic Mode



To cut the profile on the other side of the workpiece, set Mirror X to ON in Turn & Face Profile screen (it is not necessary to change Mirror X in Settings). When Cycle Start is pressed, the opposite side of the workpiece is cut, and if Graphic Mode is set to ON, a graphical representation of the mirrored profile is displayed.



Recalling Profiles

The Profile Selector popup is used to select a profile, alter an existing profile, choose a storage location for a new profile or delete a profile, and is accessed by pressing F1 in the Profile tab or by selecting the Profile Number box and pressing Write/Enter.

Once in the Profile Selector popup screen, cursor to the number of the previously created profile and press Alter. Cursor to any data cell to change its information, then press F2 to Exit the Profile Selector popup screen and Save the new information, or F3 to Exit without Saving.

Profile Creator Help

Press F1 when in the Profile Creator screen to display a Profile Creator Help popup screen. This popup screen lists available keys used in the Profile Creator along with a short description of each key's function.

Profile Creator Help	F1 - Exit
Exit and Save Profile	(F2)
Exit without Saving Profile	(F3)
Activate Zoom	(F4)
--- ZOOM HELP ---	
Zoom In	(PAGE UP)
Zoom Out	(PAGE DOWN)
Scroll Up	(UP CURSOR KEY)
Scroll Down	(DOWN CURSOR KEY)
Scroll Right	(RIGHT CURSOR KEY)
Scroll Left	(LEFT CURSOR KEY)
Exit Zoom	(F4)
--- DATA TABLE HELP ---	
Enter Data Into Table	(WRITE/ENTER)
Insert Line Into Table	(INSERT)
Clear All Data In Table	(ORIGIN)
Go To X Axis Data Box	(X JOG KEY)
Go To Z Axis Data Box	(Z JOG KEY)
Move Up To Next Data Box	(UP CURSOR KEY)
Move Down To Next Data Box	(DOWN CURSOR KEY)
Move Right To Next Data Box	(RIGHT CURSOR KEY)
Move Left To Next Data Box	(LEFT CURSOR KEY)

Exit and Save Profile - Exit Profile Creator screen and saves the profile you were working on into program memory.

Exit without Saving Profile - Exit Profile Creator screen and does not save the profile you were working on.

Activate Zoom - Turns on the Zoom and Scrolling function.

Zoom In - Allows you to zoom into a part for a closer look.

Zoom Out - Allows you to zoom out from the part and see more in the window.

Scroll Up - Allows you to scroll the view window up.

Scroll Down - Allows you to scroll the view window down.

Scroll Right - Allows you to scroll the view window to the right.

Scroll Left - Allows you to scroll the view window to the left.

Exit Zoom - Turns off the zoom and scrolling function.

Enter Data Into Table - Transfers data from command line into selected data box or accepts value jogged.

Insert Line Into Table - Moves selected line down and inserts new line into table. Will not work if table is full!

Clear All Data In Table - Clears all the data in current table and puts the table in its home position.

Go To X Axis Data Box - Highlights X axis data box and changes drawing cursor to only move in X direction.

Go To Z Axis Data Box - Highlights Z axis data box and changes drawing cursor to only move in Z direction.

Move Up To Next Data Box - Moves up to next data box above its current location. Will not move if already at the top of the table.

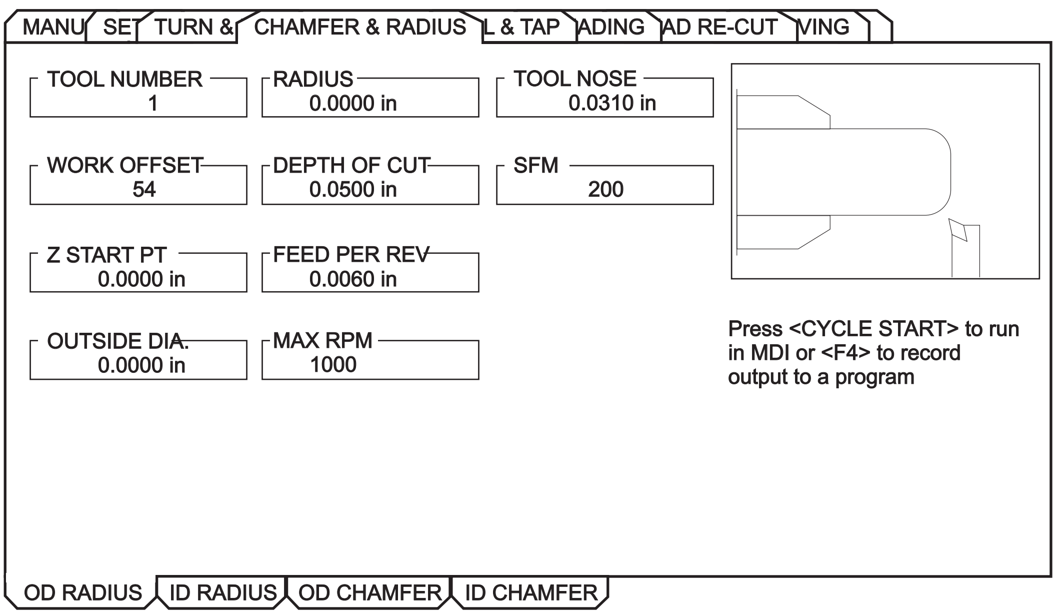
Move Down To Next Data Box - Moves down to next data box below its current location. Will not move if already at the bottom of the table.

Move Right To Next Data Box - Moves to the next data box to the right of its current location. Will wrap if already at the far right.

Move Left To Next Data Box - Moves to the next data box to the left of its current location. Will wrap if already at the far left.

Chamfer & Radius - OD Radius

This mode is used to cut an outside diameter radius.



Tool Number – Enter the tool to be used.

Work Offset – Enter the work offset to be used.

Z Start PT – Enter the Z axis starting point.

Outside Dia. – Enter the current diameter of the work piece. Manually measure the diameter.

Radius – Enter the desired radius. This is the desired corner radius. Note that the larger the radius or material to be removed, the more passes required to rough out the profile.

Depth of Cut – Enter depth of cut for each pass of stock removal. This is the amount of stock to be removed on each tool pass. A pass must be less than or equal to maximum single pass cut depth for selected tool.

Feed Per Rev – Enter feed per revoultion. This is distance tool will move for each revolution of the spindle.

MAX RPM – Enter the maximum spindle turning speed.

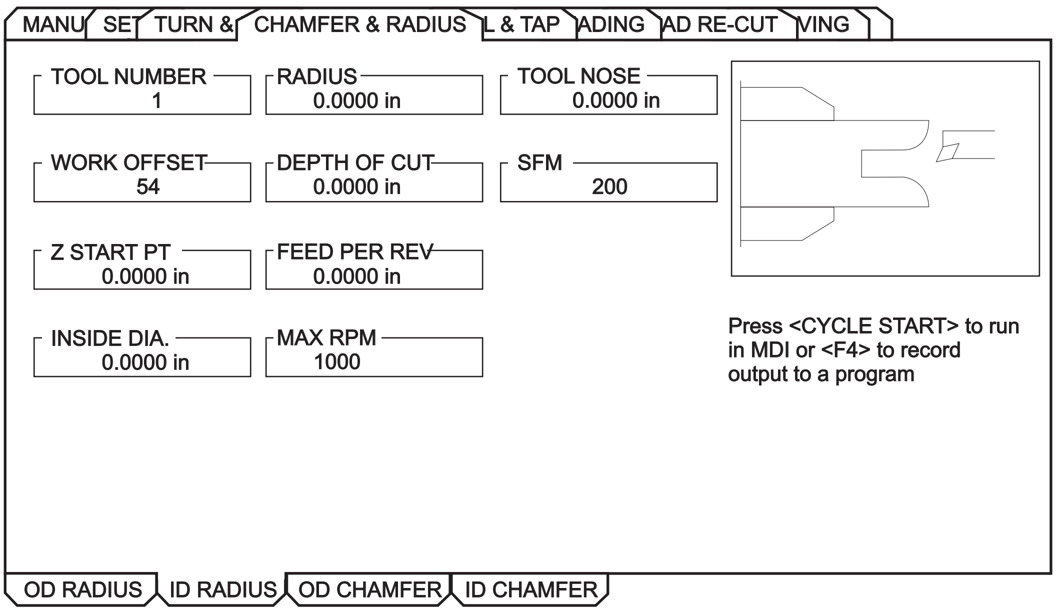
Tool Nose – Enter the tool nose radius. This is the radius of the selected tool. Normally this information is included with the tool.

SFM – Enter the Surface Feed per Minute.

Advanced Users: In the full CNC mode this is a G71 command.

Chamfer & Radius - I.D. Radius

This mode is used to cut an inside diameter radius.



Tool Number – Enter the tool to be used.

Work Offset – Enter the work offset to be used.

Z Start PT – Enter the Z axis starting point.

Inside Dia. – Enter the current diameter of the work piece. Manually measure the diameter.

Radius – Enter the desired radius. This is the desired corner radius. Note that the larger the radius or material to be removed, the more passes required to rough out the profile.

Depth of Cut – Enter the depth of cut for each pass of the stock removal. This is the amount of the stock to be removed on each tool pass. A pass must be less than or equal to the maximum single pass cut depth for the selected tool.

Feed Per Rev – Enter the feed per revoultion. This is the distance the tool will move for each revolution of the spindle.

MAX RPM – Enter the maximum spindle turning speed.

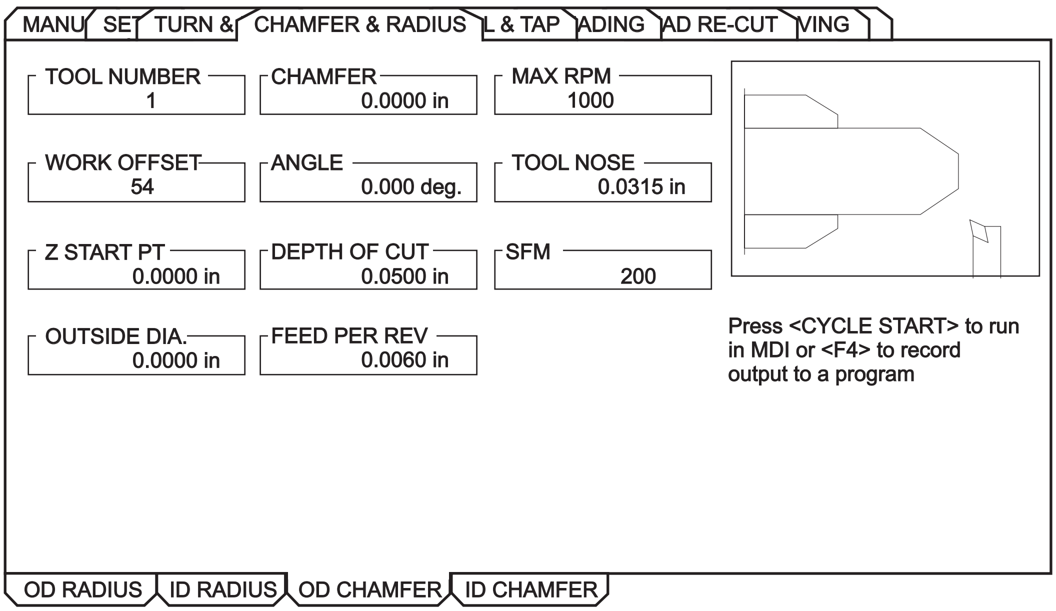
Tool Nose – Enter the tool nose radius. This is the radius of the selected tool. Normally this information is included with the tool.

SFM – Enter the Surface Feed per Minute.

Advanced Users: In the full CNC mode this is a G71 command.

Chamfer & Radius - OD Chamfer

This mode is used to cut an outside diameter chamfer.



Tool Number – Enter the tool to be used.

Work Offset – Enter the work offset to be used.

Z Start PT – Enter the Z axis starting point.

Outside Dia. – Enter the outside diameter of the part. Manually measure the work piece.

Chamfer – Enter the Z dimension of the chamfer desired. Entered value must be positive.

Angle – Enter the angle of the chamfer (0°–90°). Entered value must be positive.

Depth of Cut – Enter the depth of cut for each pass of the stock removal.

Feed Per Rev – Enter the feed per revolution.

MAX RPM – Enter the maximum spindle turning speed.

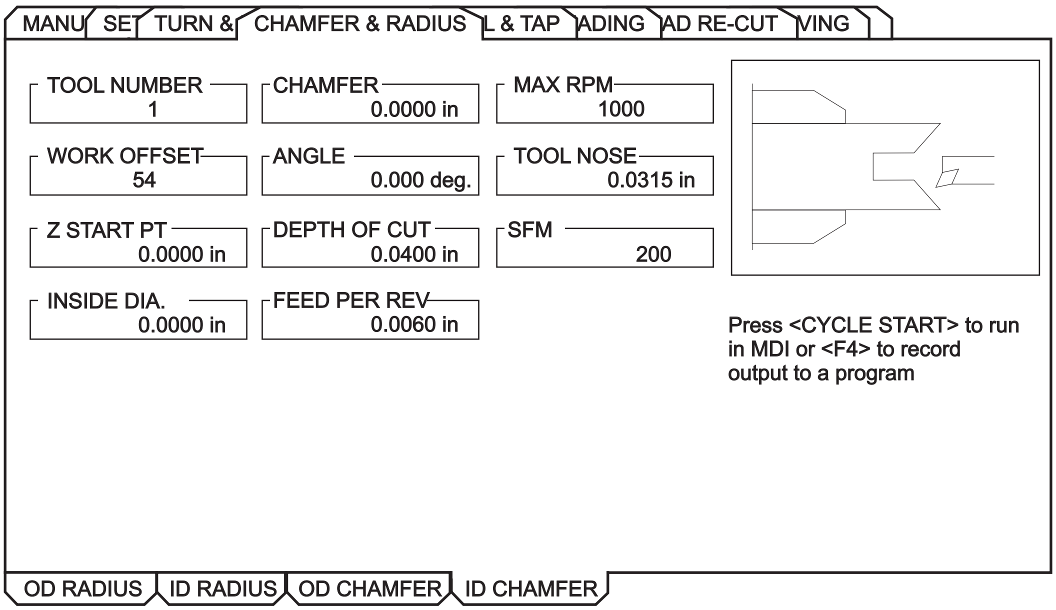
Tool Nose – Enter the tool nose radius.

SFM – Enter the Surface Feed per Minute.

Advanced Users: In the full CNC mode this is a G71 command.

Chamfer & Radius - ID Chamfer

This mode is used to cut an outside diameter chamfer.



Tool Number – Enter the tool to be used.

Work Offset – Enter the work offset to be used.

Z Start PT – Enter the Z axis starting point.

Inside Dia. – Enter current diameter of work piece (outside diameter of part). Manually measure work piece.

Chamfer – Enter the Z dimension of the chamfer desired. Entered value must be positive.

Angle – Enter the angle of the chamfer (0°-90°). Entered value must be positive.

Depth of Cut – Enter the depth of cut for each pass of the stock removal.

Feed Per Rev – Enter the feed per revolution.

MAX RPM – Enter the maximum spindle turning speed.

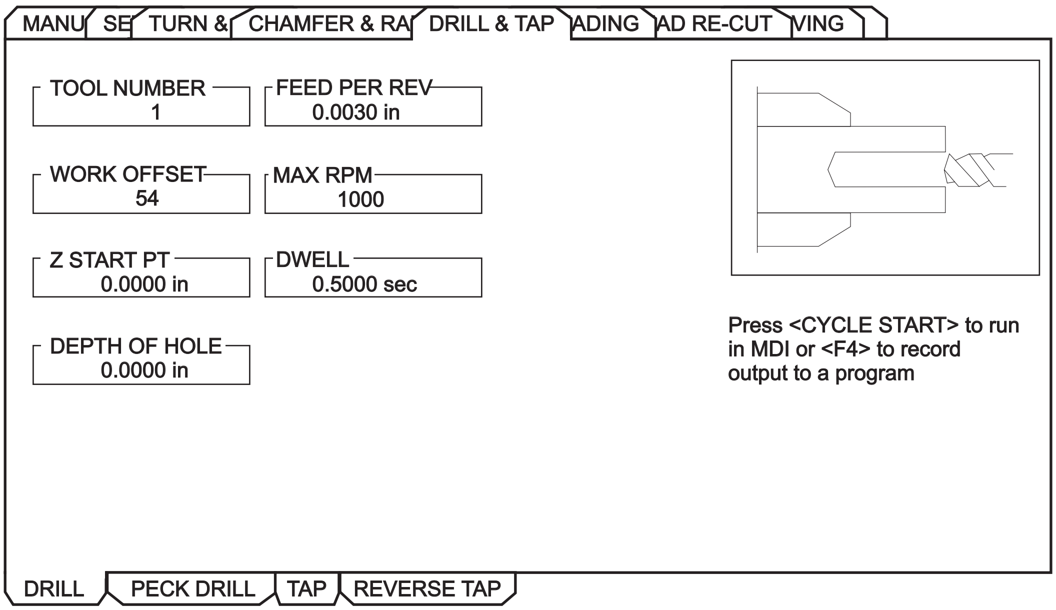
Tool Nose – Enter the tool nose radius.

SFM – Enter the Surface Feed per Minute.

Advanced Users: In the full CNC mode this is a G71 command.

Drill & Tap - Drill

This mode is a drill cycle that can pause at the bottom of the hole.



Tool Number – Enter the tool to be used.

Work Offset – Enter the work offset to be used.

Z Start PT – Enter the Z axis starting point.

Depth of Hole – Enter the depth to drill.

Feed Per Rev – Enter feed per revolution (distance the tool will move for each revolution of the spindle).

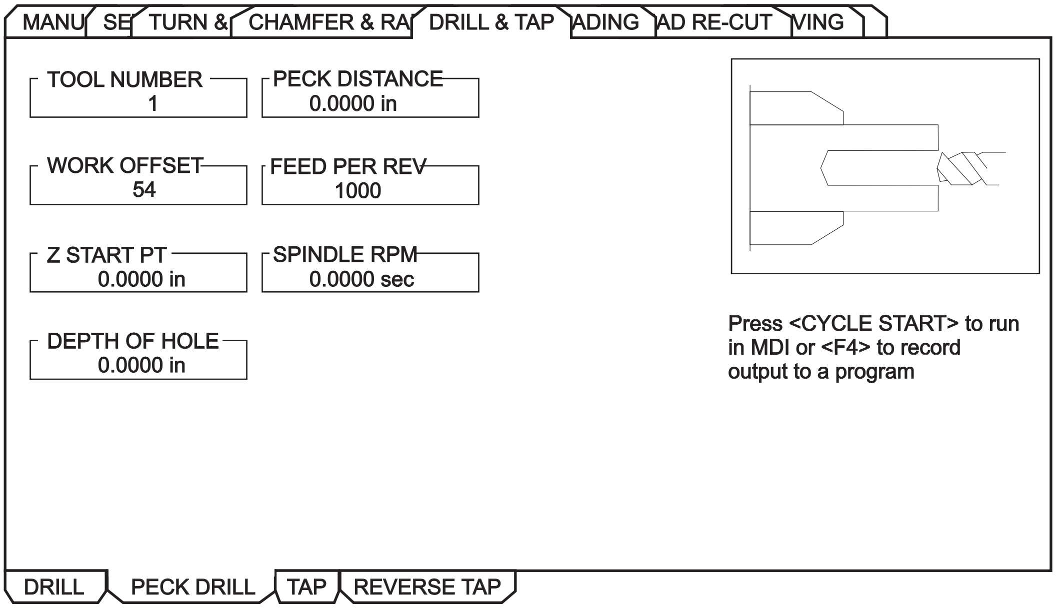
MAX RPM – Enter the spindle RPM.

Dwell – Enter dwell time (time, in seconds, that the tool pauses at the bottom of the hole to clear chips).

Advanced Users: In the full CNC mode this is a G82 command.

Drill & Tap - Peck Drill

This mode is for drilling in a pecking motion in order to remove the chip build up while drilling the hole.



Tool Number – Enter the tool to be used.

Work Offset – Enter the work offset to be used.

Z Start Pt – Enter the Z axis starting point.

Depth of Hole – Enter the depth to drill. Entered value must be positive.

Peck Distance – Enter the length of each 'peck' before retracting to clear chips. This is the distance the tool will advance at each "peck". This value cannot be negative.

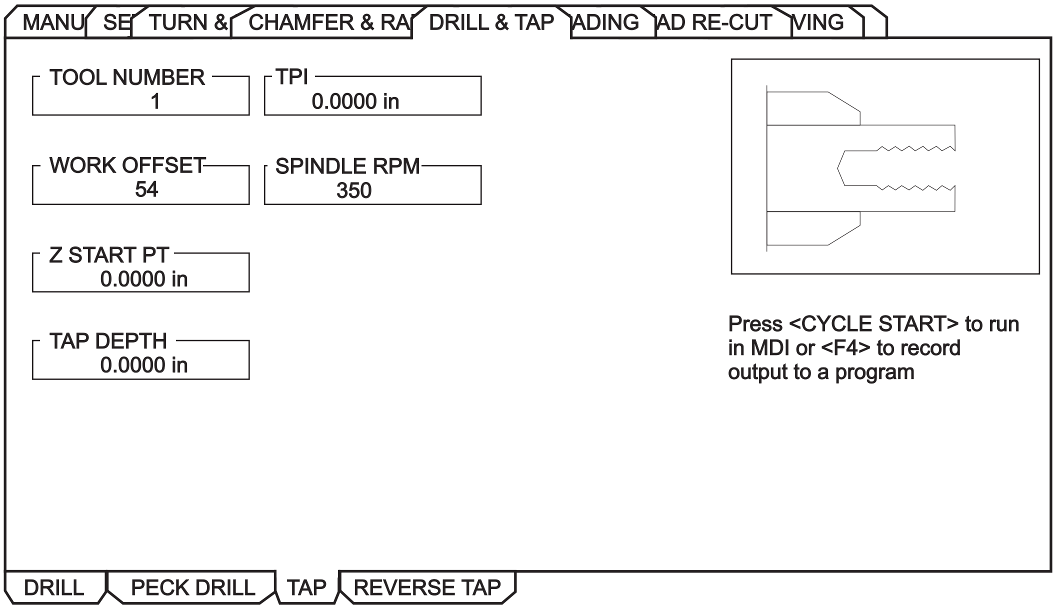
Feed Per Rev – Enter feed per revolution (distance the tool will move for each revolution of the spindle).

Spindle RPM – Enter the spindle RPM. This is the commanded spindle speed.

Advanced Users: In the full CNC mode this is a G83 command.

Drill & Tap - Tap*

This mode is for cutting right hand threads using a tapping tool.



Tool Number – Enter the tool to be used.

Work Offset – Enter the work offset to be used.

Z Start PT – Enter the Z axis starting point.

Tap Depth – Enhter the depth to tap. Entered value must be positive.

TPI – (Threads per Inch) – Enter the number of Threads per Inch. This is how many threads to cut per inch.

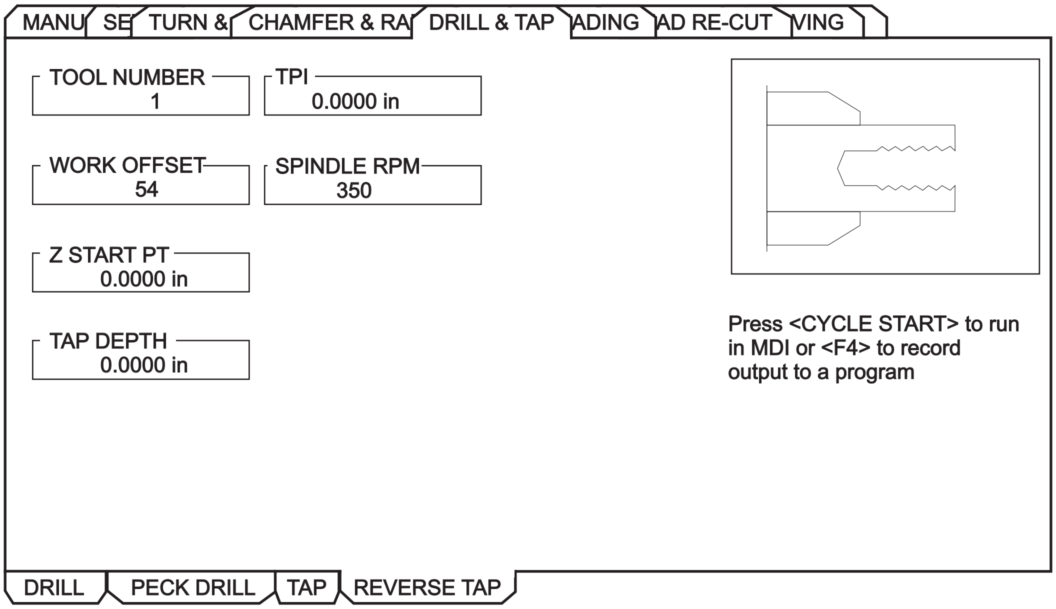
Spindle RPM – Enter spindle RPM (commanded spindle speed). Spindle speed should not exceed 500 RPM.

Advanced Users: In the full CNC mode this is a G84 command.

(*Rigid Tapping Option needed.)

Drill & Tap - Reverse Tap*

This mode is for cutting left hand threads using a tapping tool.



Tool Number – Enter the tool to be used.

Work Offset – Enter the work offset to be used.

Z Start PT – Enter the Z axis starting point.

Tap Depth – The depth to tap. Entered value must be positive.

TPI – How many threads to cut per inch.

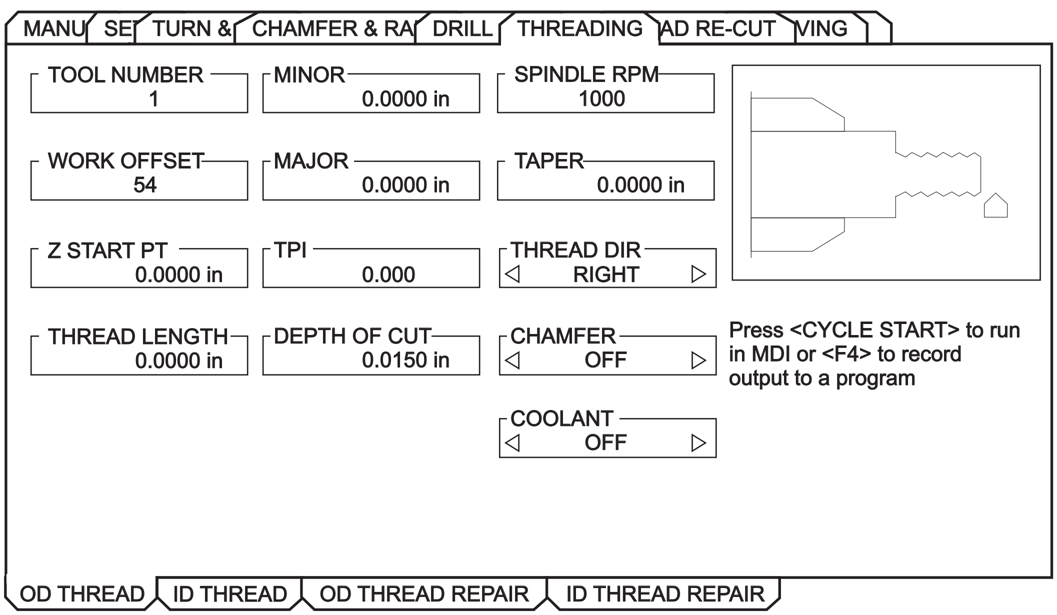
Spindle RPM – The commanded spindle speed. The spindle speed should not exceed 500 RPM.

Advanced Users: In the full CNC mode this is a G184 command.

(*Rigid Tapping Option needed.)

Threading - OD Thread

This mode is used for cutting outside diameter threads using multiple passes.



Tool Number – Enter the tool to be used.

Work Offset – Enter the work offset to be used.

Z Start Pt – Enter the Z axis starting point.

Thread Length – Enter the length of the threaded portion of the part. This value cannot be negative.

Minor – Enter minor diameter of threads (smallest part of the thread (Min Diameter). Cannot be negative.

Major – Enter major diameter of threads (largest part of the thread (Max Diameter). Cannot be negative. Manually measure the diameter of the work piece at the point where “X Dia Meas” was pressed.

TPI (Threads per Inch) – Enter number of Threads per Inch (how many threads to cut per inch of length).

Depth of Cut – Enter the amount of stock to be removed on each pass. Must be less than or equal to the maximum single pass cut depth for the selected tool.

Spindle RPM – Enter the spindle RPM. This is the commanded spindle speed.

Taper – Enter a positive value for thread taper per ft.

Thread Dir – Enter ‘0’ for right hand threads or enter ‘1’ for left hand threads.

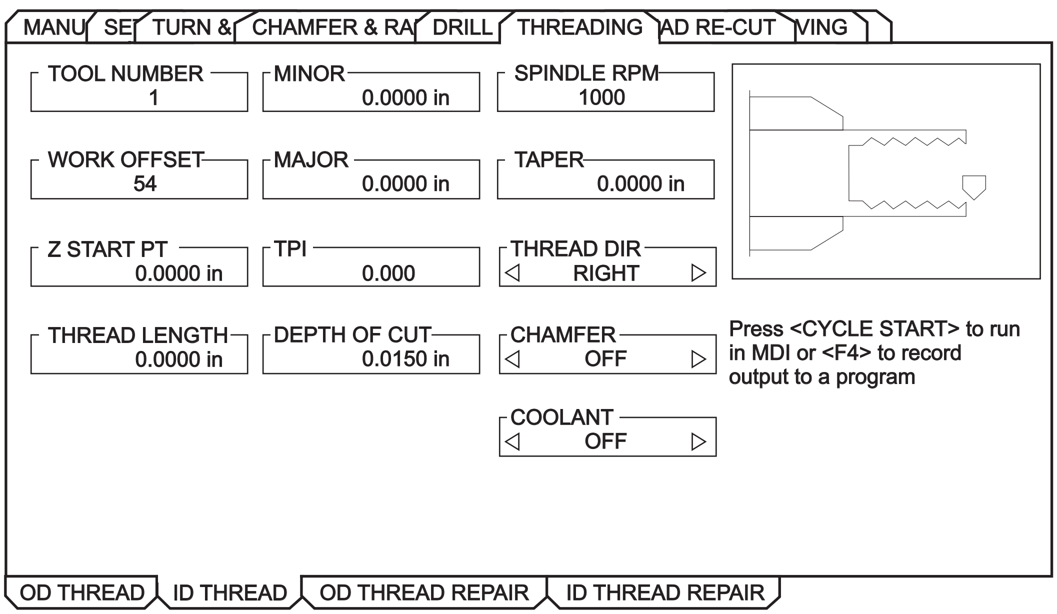
Chamfer – ‘ON’ turns on chamfer at end of threads. ‘OFF’ turns off chamfer at end of threads. (Check setting 95, 96, 86, 99).

Coolant – ‘ON’ turns on machine coolant. ‘OFF’ turns off machine coolant.

Advanced Users: Additional Settings may need to be modified to create the required groove. These setting numbers are: 22, 28, 72, 73, 86, 95, 96, 99. See the definitions of the setting in the Operator’s Manual.
In the full CNC mode this is a G76 command.

Threading - ID Thread

This mode is used for cutting inside diameter threads using multiple passes.



Tool Number – Enter the tool to be used.

Work Offset – Enter the work offset to be used.

Z Start Pt – Enter the Z axis starting point.

Thread Length – Enter the length of the threaded portion of the part. This value cannot be negative.

Minor – Enter minor diameter of threads (smallest part of the thread (Min Diameter). Cannot be negative.

Major – Enter major diameter of the threads (largest part of the thread (Max Diameter). Cannot be negative. Manually measure the diameter of the work piece at the point where “X Dia Meas” was pressed.

TPI (Threads per Inch) – Enter number of Threads per Inch (how many threads to cut per inch of length).

Depth of Cut – Enter the amount of stock to be removed on each pass. Must be less than or equal to the maximum single pass cut depth for the selected tool.

Spindle RPM – Enter the spindle RPM. This is the commanded spindle speed.

Taper – Enter a positive value for thread taper per ft.

Thread Dir – Enter ‘0’ for right hand threads or enter ‘1’ for left hand threads.

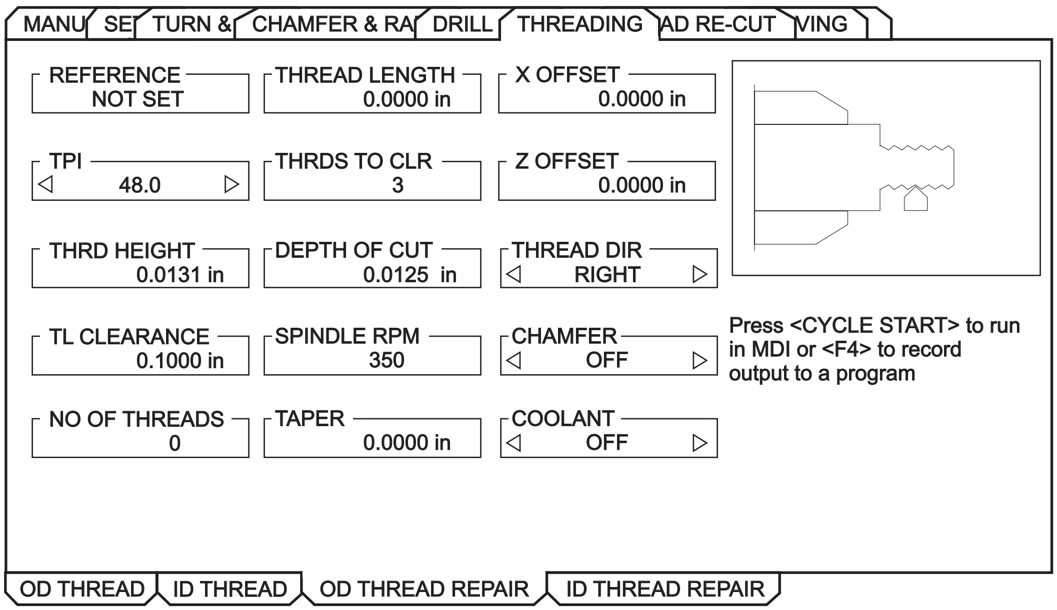
Chamfer – ‘0’ turns on chamfer at end of threads. ‘1’ turns off chamfer at end of threads. (Check setting 95, 96, 86, 99).

Coolant – ‘ON’ turns on machine coolant. ‘OFF’ turns off machine coolant.

Advanced Users: Additional Settings may need to be modified to create the required groove. These setting numbers are: 22, 28, 72, 73, 86, 95, 96, 99. See the definitions of the setting in the Operator’s Manual.
In the full CNC mode this is a G76 command.

Threading - OD Thread Repair

This mode is for repairing outside diameter threads using multiple passes.



Reference – Jog the tool into the threads, then press the X DIA MEAS key. 1 = reference point recorded.

TPI – Enter the number of Threads per Inch (or Threads per Millimeter).

Thread Height –

TL Clearance –

No Of Threads – Enter the number of threads from the tool to the end of the part.

Thread Length – Enter the length of the threaded portion of the part.

Threads to Clear –

Depth of cut – Enter the amount of stock to be removed on each pass.

Spindle RPM – Enter the spindle RPM.

Taper – Enter a positive value for thread taper per ft.

X Offset – Enter a value only if minor adjustments are needed in the X axis.

Z Offset – Enter a value only if minor adjustments are needed in the Z axis.

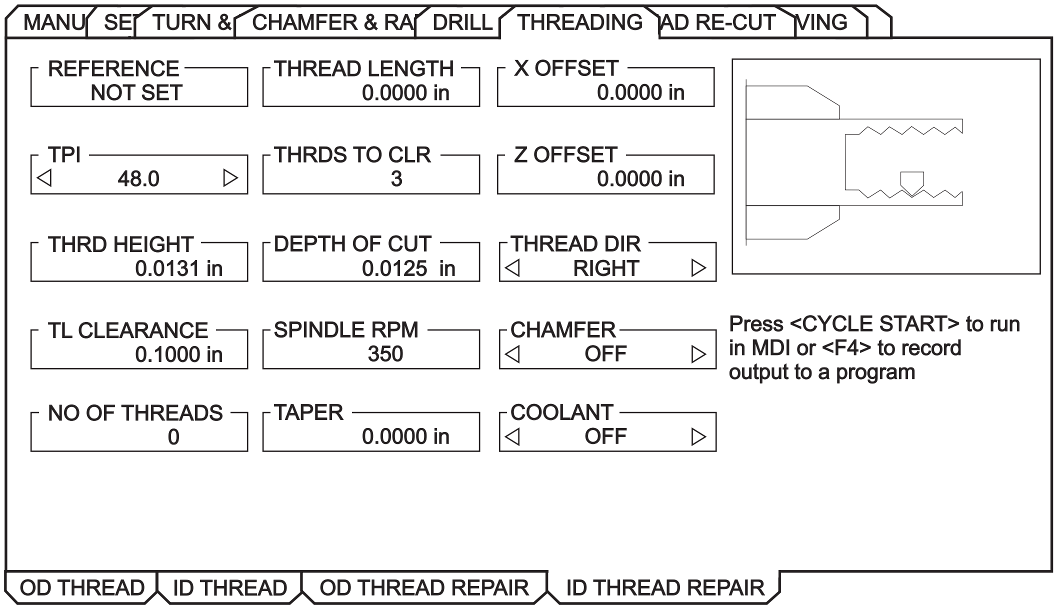
Thread Dir – Enter ‘0’ for right hand threads or enter ‘1’ for left hand threads.

Chamfer – ‘0’ turns on chamfer at end of threads. ‘1’ turns off chamfer at end of threads. (Check setting 95, 96, 86, 99).

Coolant – ‘ON’ turns on machine coolant. ‘OFF’ turns off machine coolant.

Threading - ID Thread Repair

This mode is for repairing outside diameter threads using multiple passes.



Reference – Jog the tool into the threads, then press the X DIA MEAS key. 1 = reference point recorded.

TPI – Enter the number of Threads per Inch (or Threads per Millimeter).

Thread Height –

TL Clearance –

No Of Threads – Enter the number of threads from the tool to the end of the part.

Thread Length – Enter the length of the threaded portion of the part.

Threads to Clear –

Depth of cut – Enter the amount of stock to be removed on each pass.

Spindle RPM – Enter the spindle RPM.

Taper – Enter a positive value for thread taper per ft.

X Offset – Enter a value only if minor adjustments are needed in the X axis.

Z Offset – Enter a value only if minor adjustments are needed in the Z axis.

Thread Dir – Enter ‘0’ for right hand threads or enter ‘1’ for left hand threads.

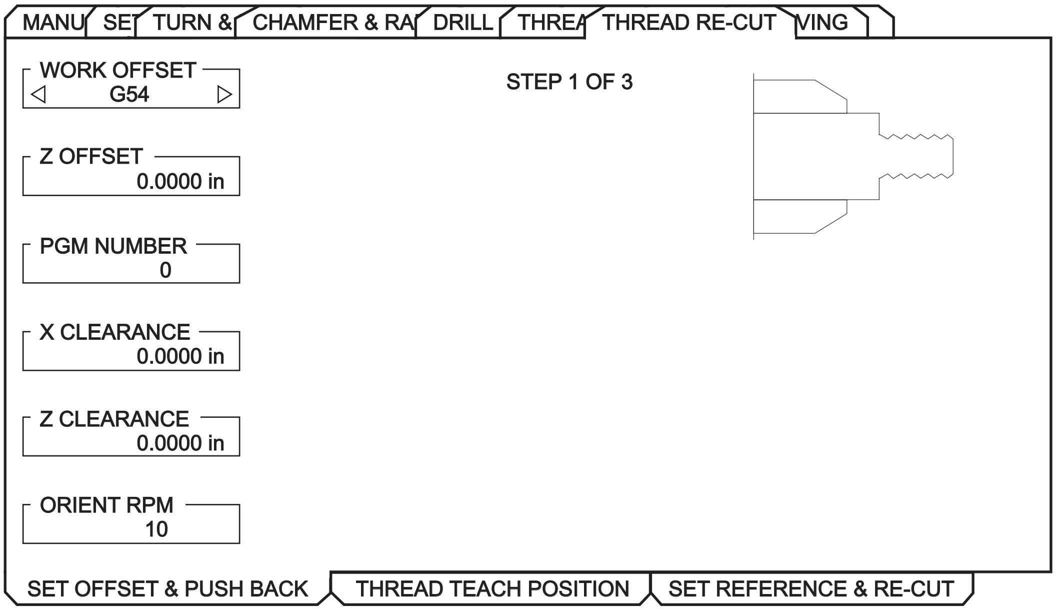
Chamfer – ‘0’ turns on chamfer at end of threads. ‘1’ turns off chamfer at end of threads. (Check setting 95, 96, 86, 99).

Coolant – ‘ON’ turns on machine coolant. ‘OFF’ turns off machine coolant.

Thread Re-Cut - Set Offset & Push Back

Step 1 of 3

This mode is used for setting work offset, then shifting work offset the amount that needs to be removed from the face of the pipe or work piece using an existing program in program memory.



Work Offset –

Z Offset –

Program Number –

X Clearance –

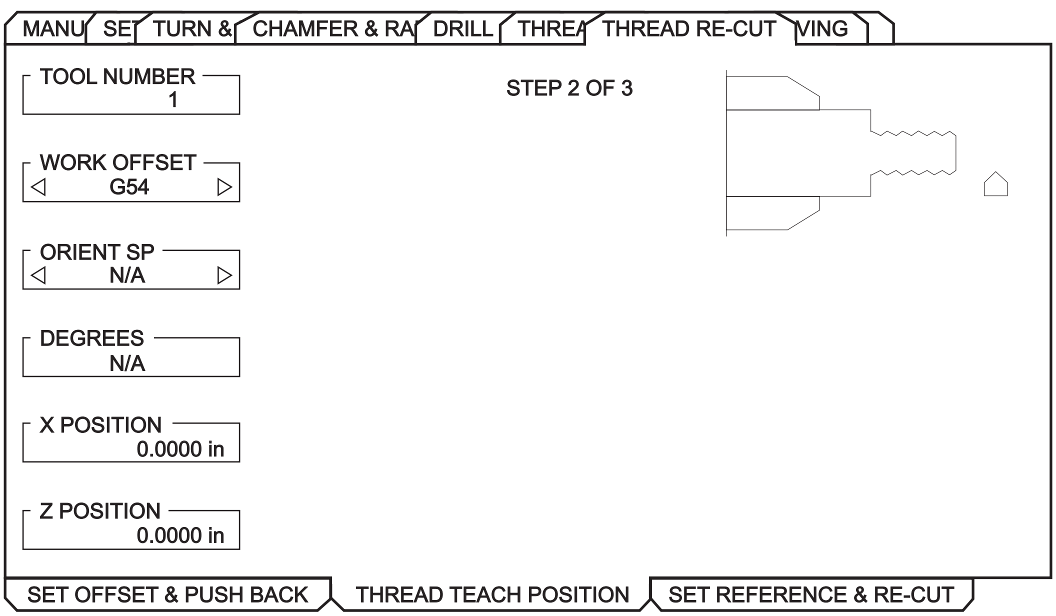
Z Clearance –

Orient RPM –

Thread Re-Cut - Thread Teach Position

Step 2 of 3

This mode is used to position the OD or ID thread tool to a specific diameter by a Z-minus position based on thread pitch.



Tool Number –

Work Offset –

Orient SP –

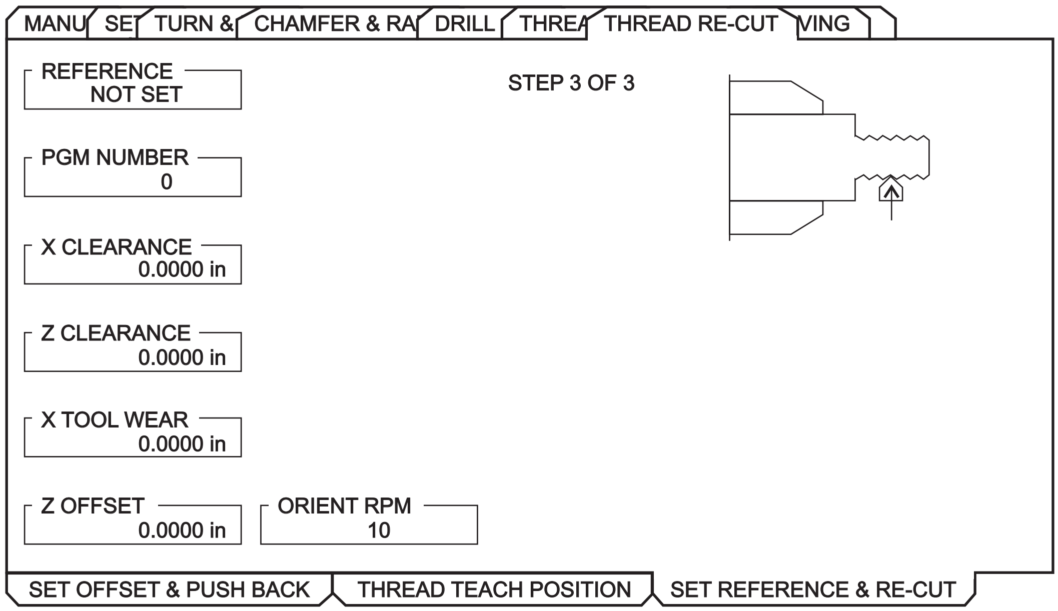
Degrees –

X Position –

Z Position –

Thread Re-Cut - Set Reference & Re-Cut

Step 3 of 3
This mode is used to teach the threading tool OD or ID. Then re-cut the threads using an existing program in program memory.



Reference –

Program Number –

X Clearance –

Z Clearance –

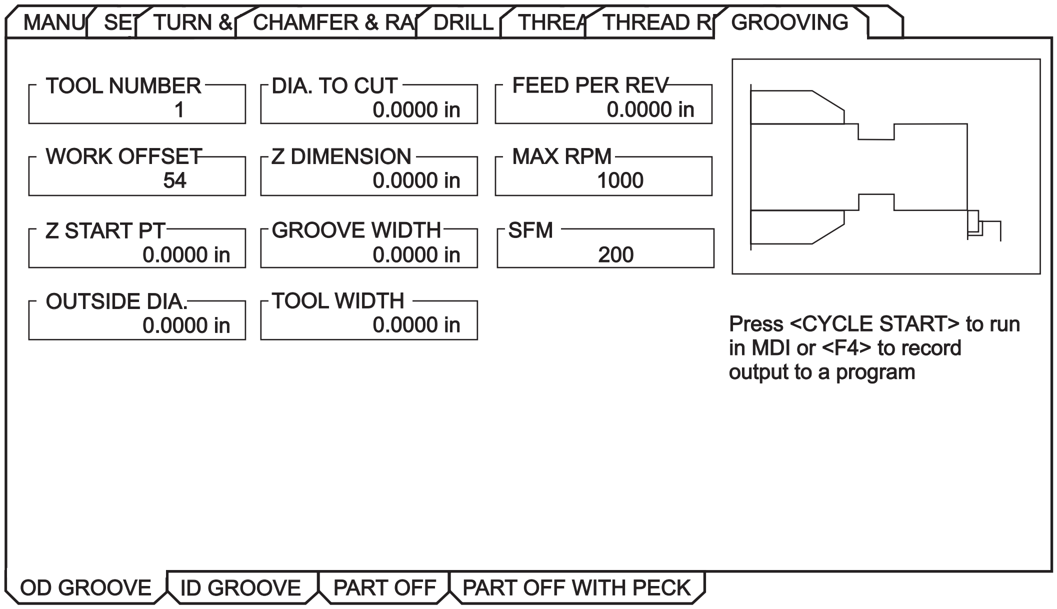
X Tool Wear –

Z Offset –

Orient RPM –

Grooving Mode - OD Groove

This mode is for an outside diameter groove.



Tool Number – Enter the tool to be used.

Work Offset – Enter the work offset to be used.

Z Start PT – Enter the Z axis starting point.

Outside Dia – Enter the current diameter of the work piece. Manually measure the diameter.

Dia to Cut (Diameter to Cut) – Enter the finished diameter.

Z Dimension – Enter the Z-axis dimension of the groove. Entered value must be positive.

Groove Width – Enter the finished width of the groove. Entered value must be positive.

Tool Width – Enter the actual width of the tool.

Feed per Rev – Enter the feed per revolution.

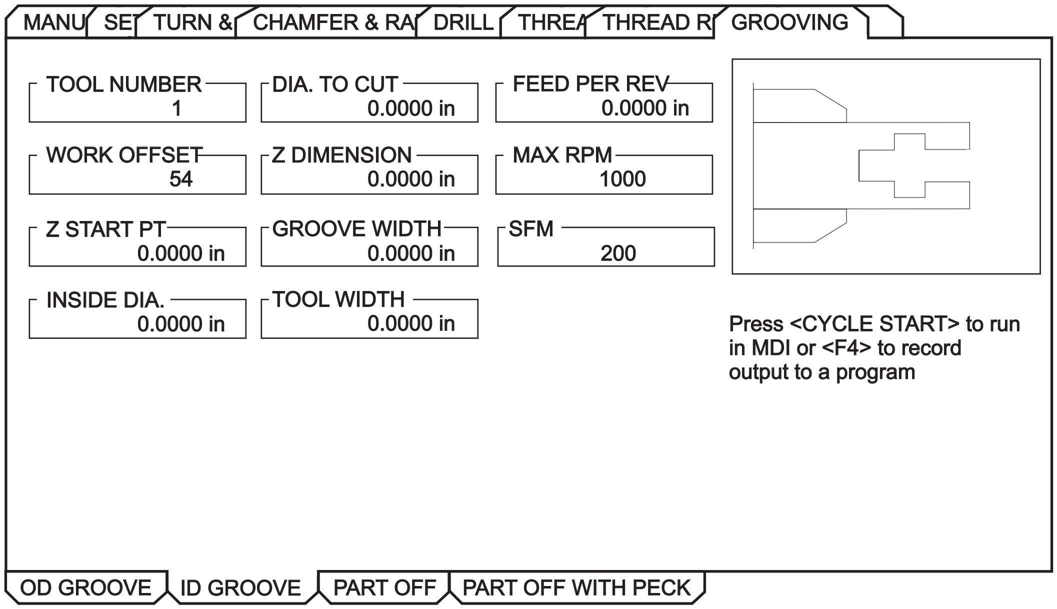
MAX RPM – Enter the maximum spindle turning speed.

SFM – Enter the Surface Feed per Minute.

Advanced Users: Additional Settings may need to be modified to create the required groove. These setting numbers are: 22, 28, 72, 73, 86. See the definitions of the setting in the Operator’s Manual.
In the full CNC mode this is a G75 command.

Grooving Mode - ID Groove

This mode is for an outside diameter groove.



Tool Number – Enter the tool to be used.

Work Offset – Enter the work offset to be used.

Z Start PT – Enter the Z axis starting point.

Outside Dia – Enter the current diameter of the work piece. Manually measure the diameter.

Dia to Cut (Diameter to Cut) – Enter the finished diameter.

Z Dimension – Enter the Z-axis dimension of the groove. Entered value must be positive.

Groove Width – Enter the finished width of the groove. Entered value must be positive.

Tool Width – Enter the actual width of the tool.

Feed per Rev – Enter the feed per revolution.

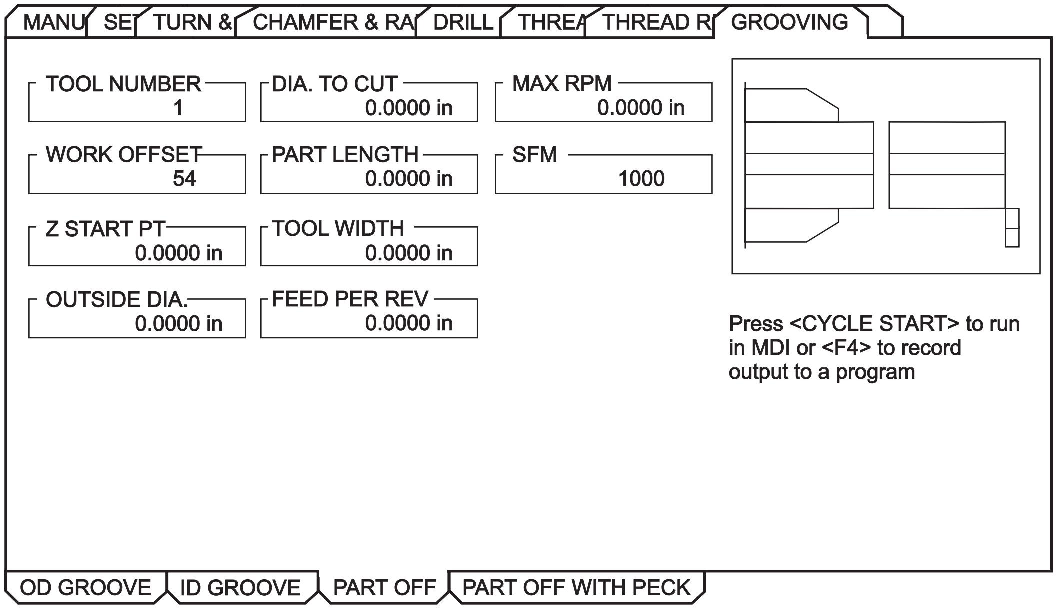
MAX RPM – Enter the maximum spindle turning speed.

SFM – Enter the Surface Feed per Minute.

Advanced Users: Additional Settings may need to be modified to create the required groove. These setting numbers are: 22, 28, 72, 73, 86. See the definitions of the setting in the Operator's Manual. In the full CNC mode this is a G75 command.

Grooving Mode - Part Off

This mode is for parting off.



Tool Number – Enter the tool to be used.

Work Offset – Enter the work offset to be used.

Z Start PT – Enter the Z axis starting point.

Outside Dia. – Enter the actual diameter of the part. This is the current diameter of the workpiece. Manually measure the diameter.

Dia. to Cut – (Diameter to Cut) – Enter the depth the tool is to cut into the part.

NOTE: Entering a negative value for "Dia to Cut" causes the tool to pass spindle center and machine the entire face of the part; **Do Not** enter a value larger than -.100".

Part Length – Enter the finished part length. Entered value must be positive.

Tool Width – Enter the parting tool width.

Feed per Rev – Enter the feed per revolution. This is the distance the tool moves for each spindle revolution.

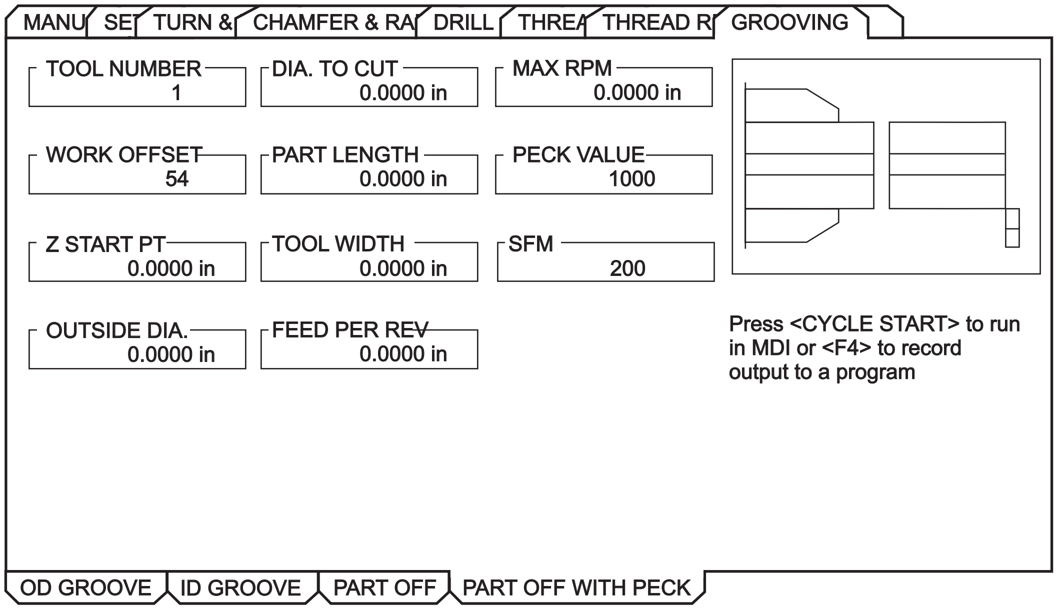
MAX RPM – Enter the maximum spindle turning speed.

SFM – Enter the Surface Feed per Minute.

Advanced Users: In the full CNC mode this is a G01 command.

Grooving Mode - Part Off with Peck

This mode is for parting off with a peck.



Tool Number – Enter the tool to be used.

Work Offset – Enter the work offset to be used.

Z Start PT – Enter the Z axis starting point.

Outside Dia. – Enter the actual diameter of the part. This is the current diameter of the workpiece. Manually measure the diameter.

Dia to Cut (Diameter to Cut) – Enter the depth the tool is to cut into the part.

NOTE: Entering a negative value for “Dia to Cut” causes the tool to pass spindle center and machine the entire face of the part; **Do Not** enter a value larger than -.100”.

Part Length – Enter the finished part length. Entered value must be positive.

Tool Width – Enter the parting tool width.

Feed per Rev – Enter the feed per revolution. This is the distance the tool moves for each spindle revolution.

MAX RPM – Enter the maximum spindle turning speed.

Peck Value – Enter the distance the tool will advance at each ‘PECK’.

SFM – Enter the Surface Feed per Minute.

Advanced Users: In the full CNC mode this is a G75 command.

DXF File Importer

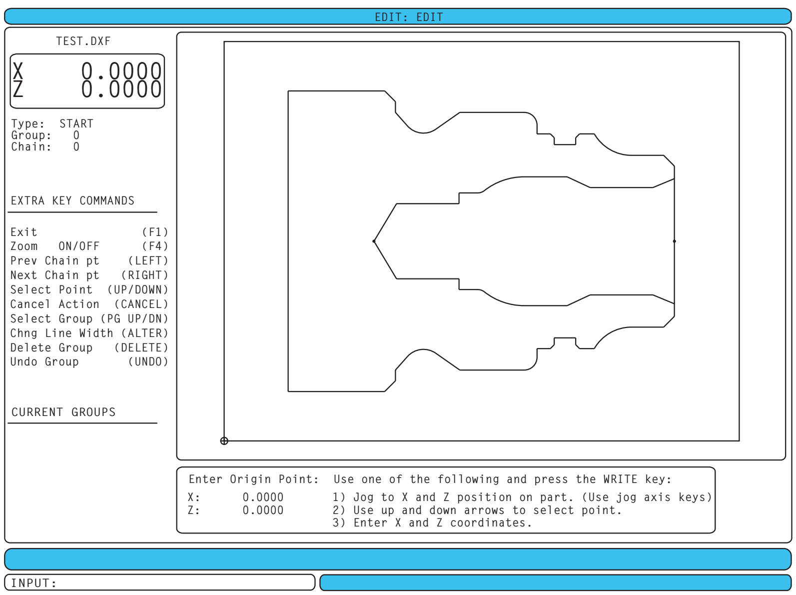
This feature can quickly build a CNC G-code program from a DXF file, a drawing file format exportable from many desktop CAD applications. Compatible DXF files are made up of arcs, lines, circles, vertices, and/or points. Refer to your CAD application's documentation for details on how to export a DXF file. When importing a DXF file, you define its features one by one as tool paths; G-code is generated for each tool path that can then be placed in any new or existing program. DXF Importer for lathes is used to create ID and OD part profiles, for other features (threads, etc.), use IPS.

Importing the DXF File

Note: Tools should be set up in IPS before starting this process.

1. Press LIST PROG, select the tab for the device (USB, Hard Drive, or Floppy) containing the DXF file and press Write/Enter. Use the cursor arrows to highlight the DXF file and press Write/Enter to select it.

2. Press F2 and select "memory". The control will recognize the DXF file and import it into the editor.



The DXF importer feature provides on-screen help in the lower right corner of the display. The keys needed are defined beside the steps. Additional keys are identified in the left hand column.

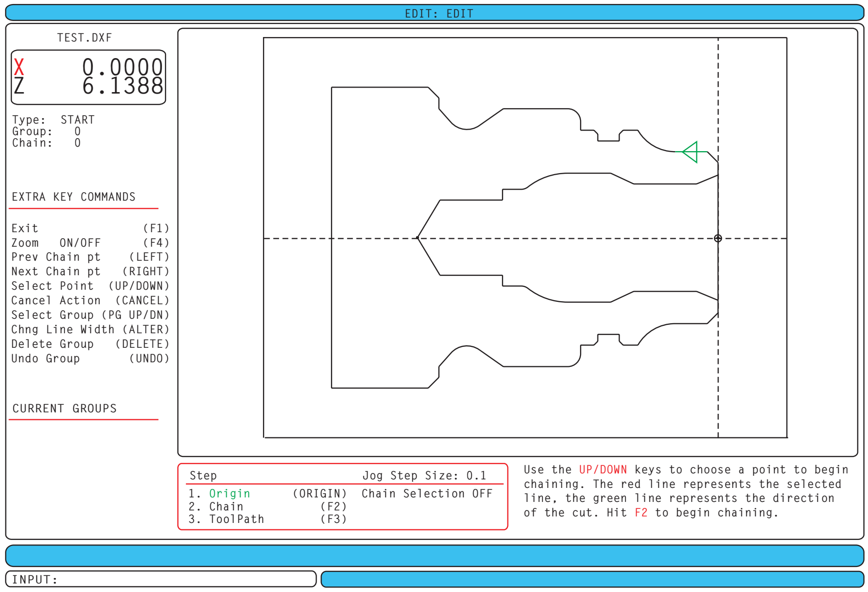
DXF Importer creates programs using simple input given in the following steps:

1. Set the part origin point
2. Chain a tool-path
3. Set the tool-path
4. Repeat steps 2 and 3 for remaining features

Set the Origin Point

Use one of three methods:

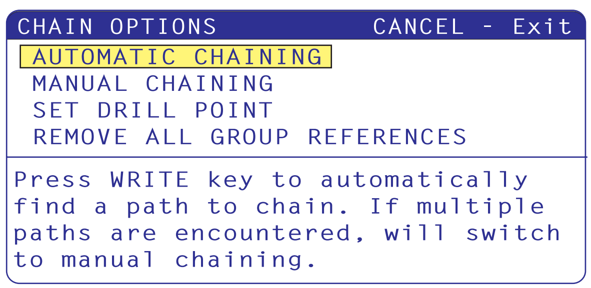
- Point Selection - Use the up and down arrow keys to select a point.
- Jogging - Jog to the X and Z position on a part (use jog axis keys).
- Enter Coordinates - Type in X coordinate and press WRITE/ENTER, then type in Z coordinate and press WRITE/ENTER.



The jog handle or arrow buttons are used to highlight a point. Press WRITE/ENTER to accept the highlighted point as the start of the tool path. Press F2 to display a CHAIN OPTIONS pop-up screen.

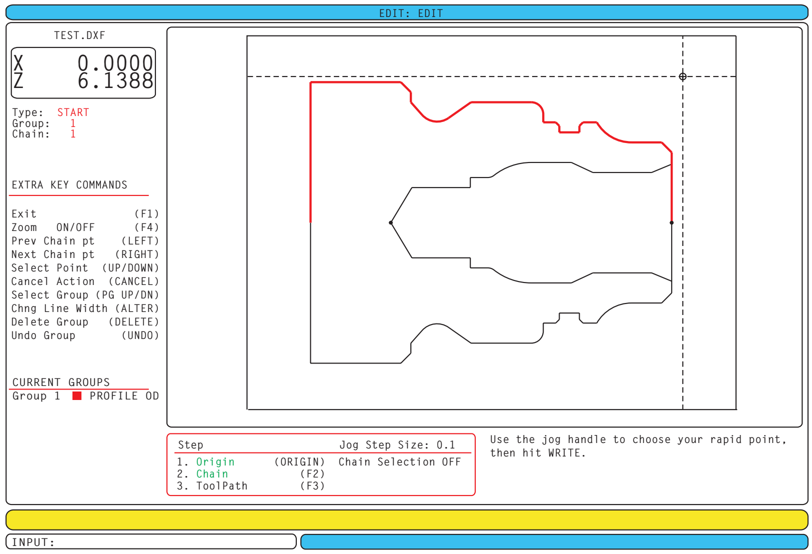
Chain/Group

This step finds the geometry of the shape(s). The auto chaining function will find most part geometry. If the geometry branches off, a pop-up will prompt you to select a branch and automatic chaining will continue.



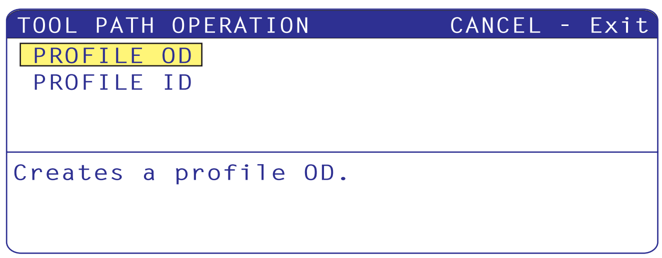
The Automatic Chaining function is typically the best choice as it will automatically plot the tool path for a part feature. Press “Enter” This will change the color of that part feature and add a group to the register under “Current group” on the left hand side of the window.

The tool path can also be manually generated. After selecting the starting point for the tool path, select “Manual Chaining” from the chain options menu. DXF Importer will begin to follow the specified line, section by section. To accept a section of geometry, press Write/Enter. Where branches occur, choose the branch to follow.

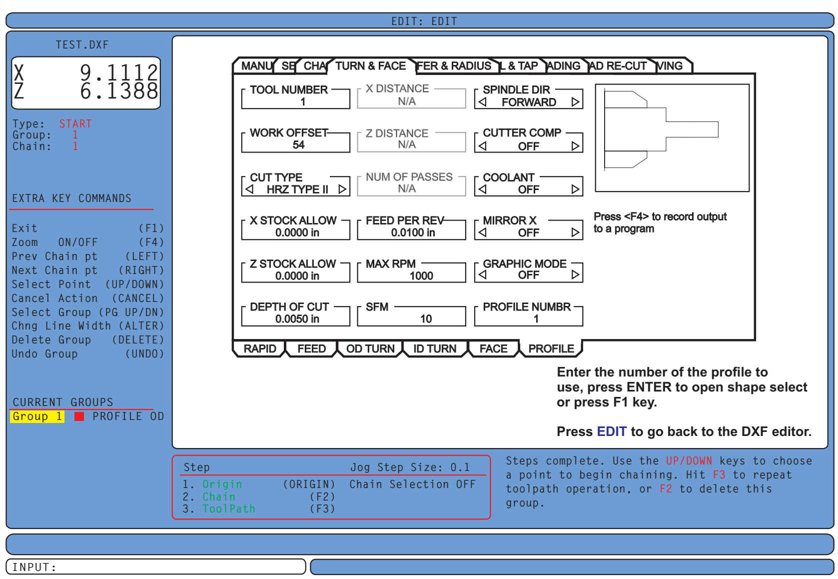


Select Tool Path

This step applies a tool-path operation to a particular chained group. Select group and Press F3 to choose a tool path.



'Select Rapid Point' is displayed at the bottom of the screen. Use the jog handle to choose the rapid point and press WRITE/ENTER.



Once a tool-path is selected, the IPS (Intuitive Programming System) template for that shape is displayed. Most IPS templates are filled with reasonable defaults derived from tools and materials that have been setup.

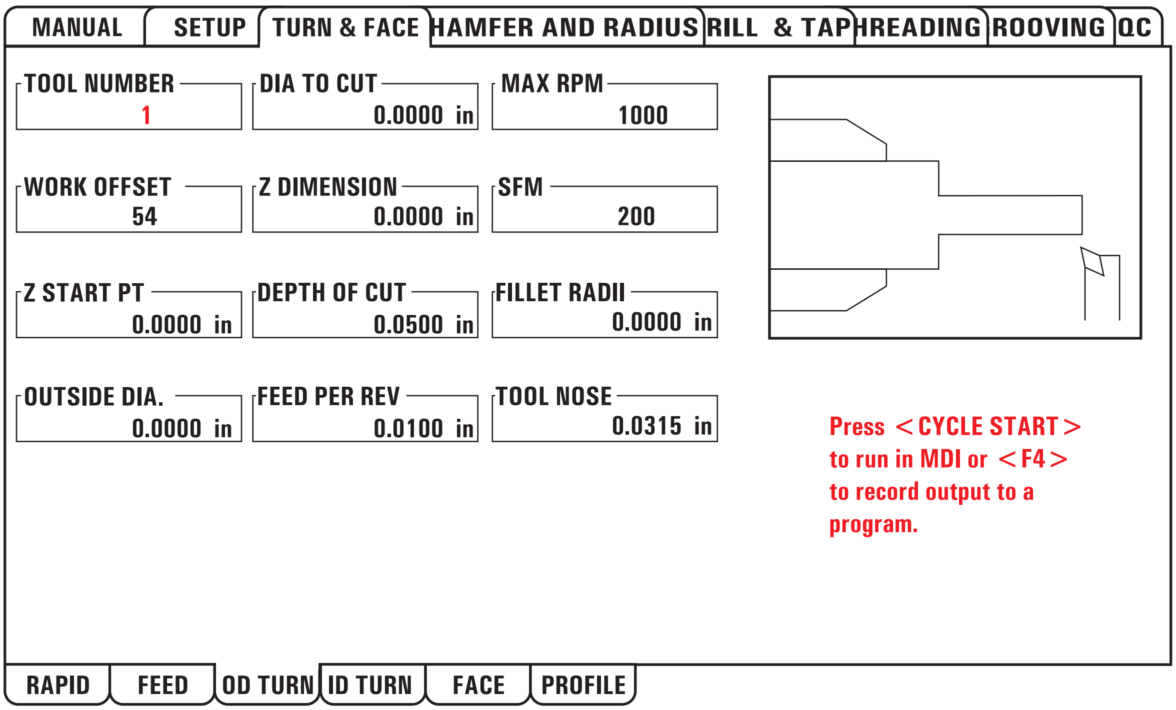
Press F4 to save the toolpath once the template is completed. Refer to the "IPS Recorder" section for details on saving the path into a new or existing program.

Press Edit to return to DXF Editor.

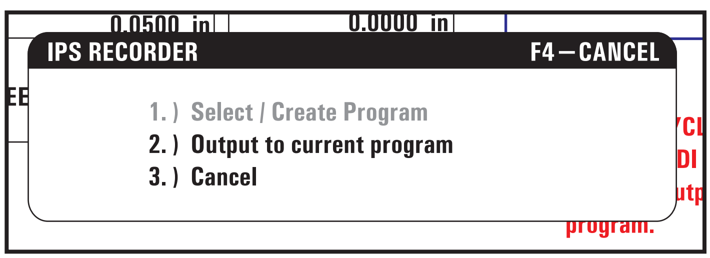
The IPS recorder provides a simple method to place G-code generated by IPS into new or existing programs.

1. To access IPS, press MDI/DNC, then PROGRM/CONVRS. Refer to your Intuitive Programming System Operator Manual (ES0609 Lathe) for more information on using IPS.

2. When the recorder is available, a message appears in red in the lower right corner of the tab:



3. Press F4 to access the IPS recorder menu. Choose menu option 1 or 2 to continue, or option 3 to cancel and return to IPS. F4 can also be used to return to IPS from any point within IPS recorder.



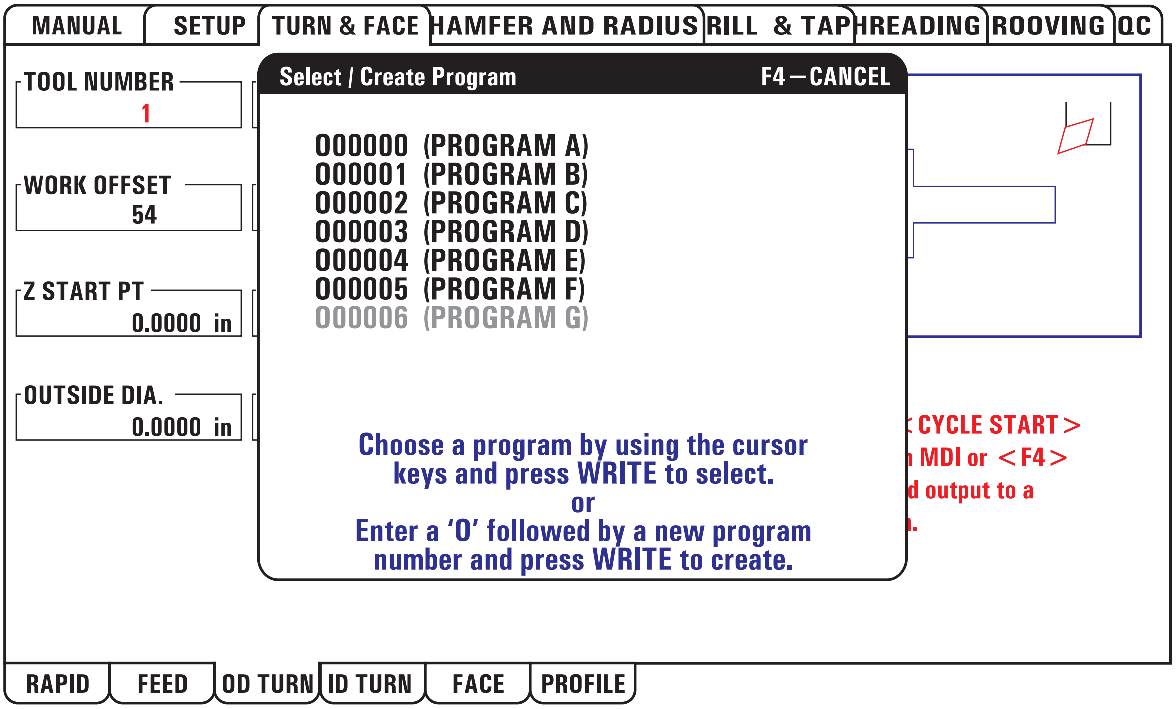
IPS Recorder Menu

Menu Option 1: Select / Create Program

Select this menu option to choose an existing program in memory or to create a new program into which the G-code will be inserted.

1. To create a new program, input the letter 'O' followed by the desired program number and press the WRITE key. The new program is created, selected, and displayed. Press the WRITE key once more to insert the IPS G-code into the new program.

2. To select an existing program, enter an existing program number using the O format (Onnnnn), then press the WRITE key to select and open the program. To choose from a list of existing programs, press the WRITE key without input. Use the cursor arrow keys to choose a program and press WRITE to open it.



3. Using the arrow keys, move the cursor to the desired insertion point for the new code. Press WRITE to insert the code.

Menu Option 2: Output to Current Program

1. Select this option to open the currently selected program in memory.

2. Use the arrow keys to move the cursor to the desired insertion point for the new code. Press WRITE to insert the code.

The keyboard is broken up into eight sections: Function Keys, Jog Keys, Override Keys, Display Keys, Cursor Keys, Alpha Keys, Mode Keys and Number Keys. In addition there are miscellaneous keys and features located on the pendant.

Section III – TL Live Images for Lathes

Haas ES Doc #ES0666

Live Image for Lathes

LIVE IMAGE FEATURE

This feature allows an operator to view a real time simulation of a part as it is cut. Live Image is standard with lathe software version 9.03 and later.

Live Imaging of a part requires that the operator setup stock and tools before running the part program.

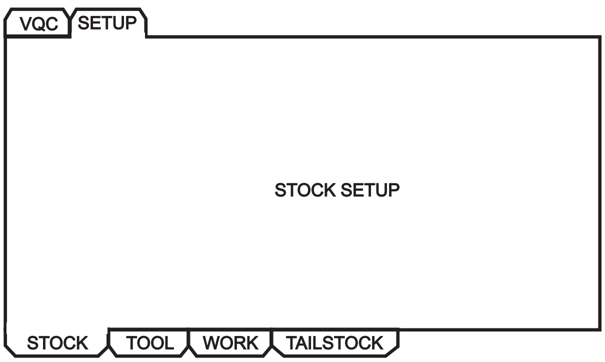
Setup

Stock Setup

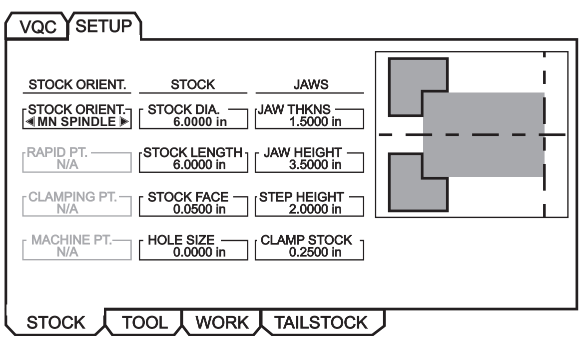
Data values for stock and jaw dimensions are stored in the Stock Setup screen. Live Image applies this stored data to each tool.

NOTE: Turn Setting 217 ON (as shown in Settings) to show the chuck jaws in the display.

1. Press MDI/DNC, then PRGRM CONVRS to enter IPS JOG mode.



2. Use the right/left arrow keys to select the SETUP tab and press Write/Enter. Use the right/left arrow keys to select the STOCK tab and press Write/Enter to display the Stock Setup screen.



Navigate screens using the left/right/up/down arrow keys to select fields. To enter the information requested by a parameter selection, use the number pad, then press Write/Enter. To exit a screen, press Cancel.

The Stock Setup screen displays stock and chuck jaw parameters that may be changed to run a particular part. Stock can be set up in main spindle, part flip, or sub spindle (if equipped) orientations.

Once the values are entered press F4 to save the stock and jaw information to the program. Select one of the choices and press enter. The control will enter the new lines of code at the cursor. Ensure the new code is entered at the line after the program number.

Program Example

```
%
O01000;
;
G20 (INCH MODE) ; (Start of Live Image information)
(STOCK);
([0.0000, 0.1000] [[6.0000, 6.0000]) ; ([Hole Size, Face] [Diameter, Length])
(JAWS);
([1.5000, 1.5000] [0.5000, 1.0000]) ; ([Height, Thickness] [Clamp, Step Height]) (End of Live Image Information)
M01 ;
;
[Part Program]
```

The advantage of entering the Stock Settings into the program is that these settings may be saved with the program, and the Stock Setup screen does not require the entry of data when the program is run in the future.

Further settings for Live Image, such as X and Z Offset, Rapid Path and Feed Path Live Image and Show Chuck Jaws are accessed by pressing SETNG GRAPH, typing in the first LIVE IMAGE setting (202) and pressing the up arrow.

GENERA | PROGRA | I/ | CONTROL PANEL | YSTEM | AINTENANCE | OWER SETTINGS | IVE IMAGE

LIVE IMAGE

202	LIVE IMAGE SCALE (HEIGHT)	1.1050
203	LIVE IMAGE X OFFSET	0.0000
205	LIVE IMAGE Z OFFSET	0.0000
206	STOCK HOLE SIZE	0.0000
207	Z STOCK FACE	0.0500
208	STOCK OD DIAMETER	6.5000
209	LENGTH OF STOCK	6.0000
210	JAW HEIGHT	3.5000
211	JAW THICKNESS	2.5000
212	CLAMP STOCK	0.2500
213	JAW STEP HEIGHT	2.0000
214	SHOW RAPID PATH LIVE IMAGE	OFF
215	SHOW FEED PATH LIVE IMAGE	OFF
217	SHOW CHUCK JAWS	ON
218	SHOW FINAL PASS	OFF
219	AUTO ZOOM TO PART	OFF
220	TS LIVE CENTER ANGLE	OFF
221	TAILSTOCK DIAMETER	OFF
222	TAILSTOCK LENGTH	OFF

Settings

202 - Live Image Scale (Height) - Specifies the height of the work area that is displayed in the live image screen. The maximum size is automatically limited to the default height. The default shows the machine's entire work area.

203 - Live Image X Offset - Locates the top of the scaling window relative to the machine X zero position. The default is zero.

205 - Live Image Z Offset - Locates the right side of the scaling window relative to the machine X zero position. The default is zero.

206 - Hole Size - Demonstrates the I.D. of the part. This setting can be adjusted by entering a value in HOLE SIZE in the STOCK SETUP tab in IPS.

207 - Z Stock Face - Controls the Z stock face of the raw part that will be displayed in live image. This setting can be adjusted by entering a value in STOCK FACE in the STOCK SETUP tab in IPS.

208 - Stock OD Diameter - Controls the diameter of the raw part that will be displayed in live image. This setting can be adjusted by entering a value in STOCK DIA. in the STOCK SETUP tab in IPS.

209 - Length of Stock - Controls the length of the raw part that will be displayed in live image. This setting can be adjusted by entering a value in STOCK LENGTH in the STOCK SETUP tab in IPS.

210 - Jaw Height - Controls the height of the chuck jaws that will be displayed in live image. This setting can be adjusted by entering a value in JAW HEIGHT in the STOCK SETUP tab in IPS.

211 - Jaw Thickness - Controls the thickness of the chuck jaws that will be displayed in live image. This setting can be adjusted by entering a value in JAW THICKNESS in the STOCK SETUP tab in IPS.

212 - Clamp Stock - Controls the clamp stock size of the chuck jaws that will be displayed in live image. This setting can be adjusted by entering a value in CLAMP STOCK in the STOCK SETUP tab in IPS.

213 - Jaw Step Height - Controls the height of the chuck jaws step that will be displayed in live image. This setting can be adjusted by entering a value in JAW STEP HEIGHT in the STOCK SETUP tab in IPS.

214 - Show Rapid Path Live Image - Controls visibility of red dashed line representing rapid path.

215 - Show Feed Path Live Image - Controls visibility of solid blue line representing feed path in live image.

217 - Show Chuck Jaws - Controls whether the green chuck jaws will be visible in live image.

218 - Show Final Pass - Controls the visibility of a solid green line that represents a final pass in live image. This is shown if the program has been previously run or simulated.

219 - Auto Zoom to Part - Controls whether or not live image will auto zoom the part to the bottom left corner. Turn on or off by pressing F4.

220 - TS Live Center Angle - Controls center angle of tailstock. Used to display the tailstock in live image.

221 - Tailstock Diameter - Controls the diameter of the tailstock. Used to display the tailstock in live image.

222 - Tailstock Length - Controls the length of the tailstock. Used to display the tailstock in live image.

224 - Flip Part Stock Diameter - Controls the new diameter location of the jaws after flipping the part

225 - Flip Part Stock Length - Controls the new length location of the jaws after flipping the part

226 - SS Stock Diameter - Controls the diameter of the part where the sub spindle clamps it.

227 - SS Stock Length - Controls the length of the sub spindle from the left of the part.

228 - SS Jaw Thickness - Controls the sub spindle jaw thickness.

229 - SS Clamp Stock - Controls the sub spindle clamp stock value.

230 - SS Jaw Height - Controls the sub spindle jaw height.

231 - SS Jaw Step Height - Controls the sub spindle jaw step height.

233 - SS Clamping Point - Controls the clamping point (the location on the part where the sub spindle clamps it) for display purposes in Live Image. This value is also used to create a G code program that will perform the desired sub spindle operation.

234 - SS Rapid Point - Controls the rapid point (the location to which the sub spindle rapids before clamping a part) for display purposes in Live Image. This value is also used to create a G code program that will perform the desired sub spindle operation.

235 - SS Machine Point - Controls the machining point (the location where the sub spindle machines a part) for display purposes in Live Image. This value is also used to create a G code program that will perform the desired sub spindle operation.

236 - FP Z Stock Face - Controls the flip part stock face for display purposes in Live Image. This value is also used to create a G code program that will perform the desired sub spindle operation.

237 - SS Z Stock Face - Controls the sub spindle stock face for display purposes in Live Image. This value is also used to create a G code program that will perform the desired sub spindle operation.

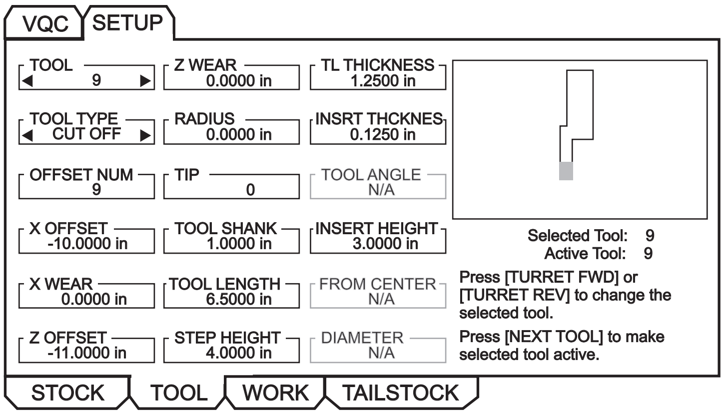
Tool Setup

Tool data is stored in offsets in the IPS tabs. Live Image uses this information to draw and simulate the tool in the cut. Required dimensions can be found in a tooling supplier's catalog or by measuring the tool.

1. From the stock setup tab, press CANCEL, select the TOOL tab and press ENTER.

2. Select the tool number, type and enter the specific parameters required for that tool (i.e., offset number, length, thickness, shank size, etc.).

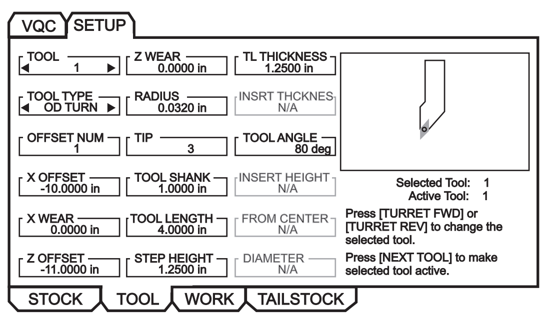
NOTE: Setup parameter entry boxes are grayed out if they do not apply to the selected tool.



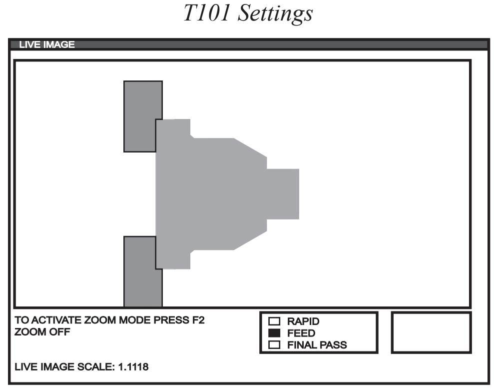
NOTE: Tool offset data may be entered for up to 50 tools.

The following section shows part of a lathe program that is cutting a piece of stock. The program is shown to the left, while the appropriate tool settings are shown to the right.

```
O01000 ;
;
;
;
T101 ;
G54 ;
G50 S4000
G96 S950 M03 ;
M08 ;
G00 X6.8 ;
Z0.15 ;
G71 P80103 Q80203 D0.25 U0.02 W0.005 F0.025 ;
N80103 ;
G00 G40 X2.
G01 X2.75 Z0. ;
G01 X3. Z-0.125 ;
G01 X3. Z-1.5 ;
G01 X4.5608 Z-2.0304 ;
G03 X5. Z-2.5606 R0.25 ;
G01 X5. Z-3.75 ;
G02 X5.5 Z-4. R0.25 ;
G01 X6.6 Z-4. ;
N80203 G01 G40 X6.8 Z-4. ;
G00 X6.8 Z0.15 ;
M09 ;
M01 ;
G28 ;
M30 ;
```

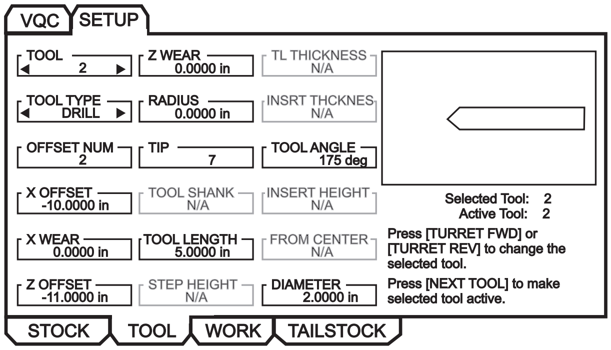


T101 Settings

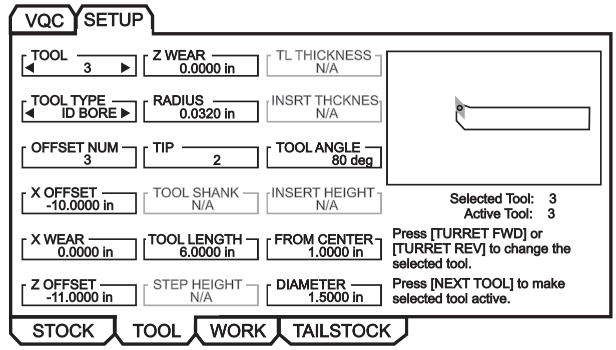


Part Worked from T101 Settings

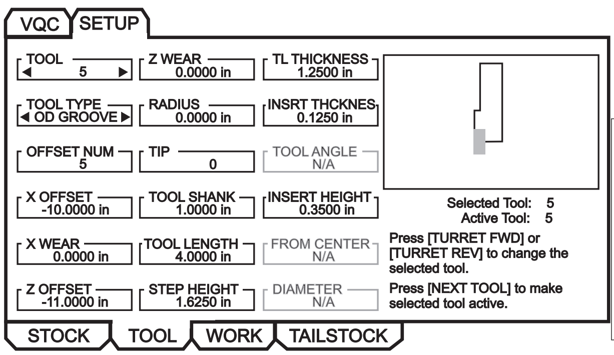
Sample Tool Setup Screens



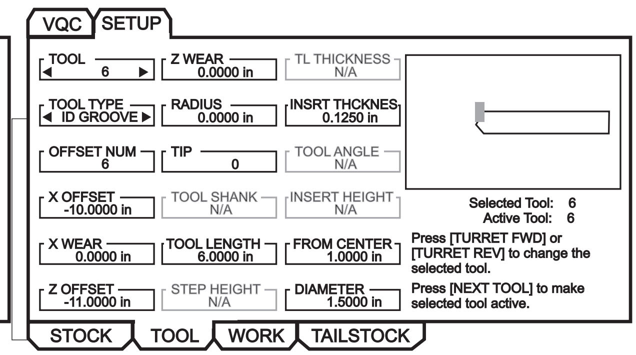
Drill



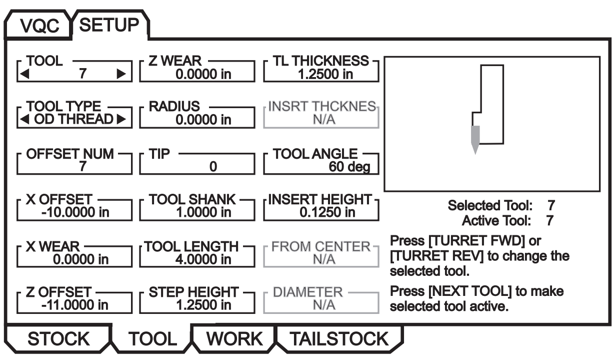
ID Bore



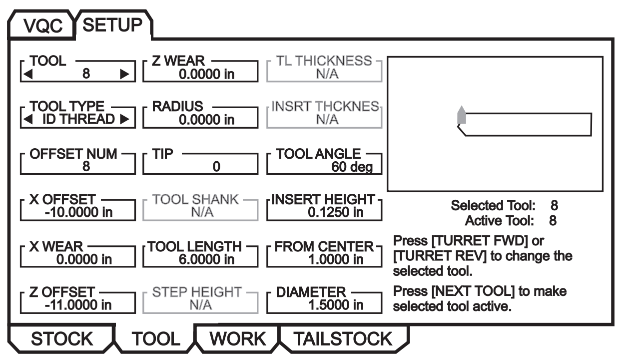
OD Groove



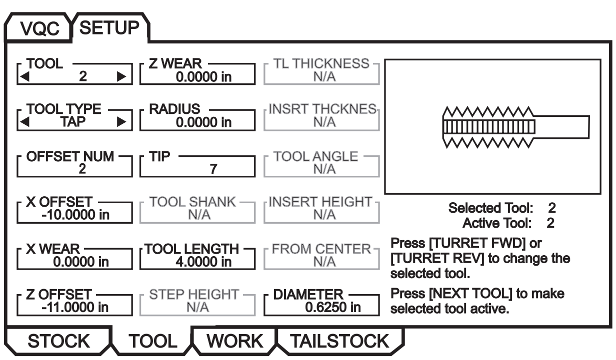
ID Groove



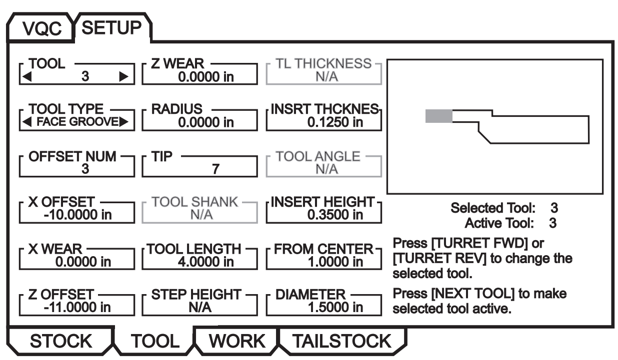
OD Thread



ID Thread



Tap



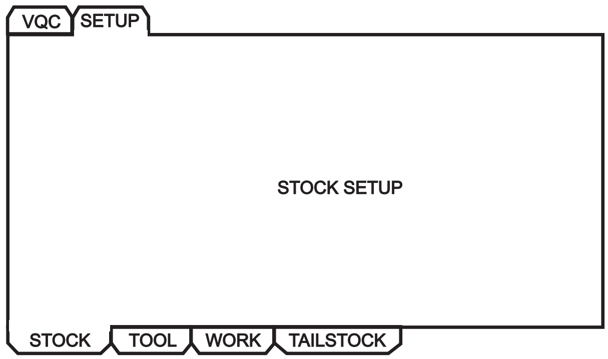
Face Groove

Tailstock Setup

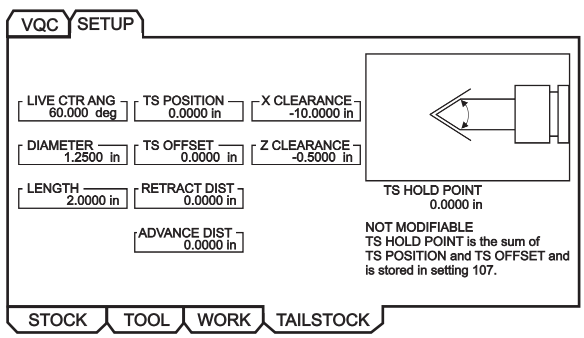
Data values for tailstock parameters are stored in offsets in the Tailstock Setup screen.

NOTE: Tailstock tab is only visible when the machine has a tailstock.

1. Press MDI/DNC, then PRGRM CONVRS to enter IPS JOG mode.



2. Use the right/left arrow keys to select the SETUP tab and press Write/Enter. Use the right/left arrow keys to select the TAILSTOCK tab and press Write/Enter to display the Tailstock Setup screen.



LIVE CTR ANG, DIAMETER and LENGTH match settings 220-222. X CLEARANCE matches setting 93. Z CLEARANCE matches setting 94. RETRACT DIST matches setting 105. ADVANCE DIST matches setting 106. TS HOLD POINT is a combination of TS POSITION and TS OFFSET and matches setting 107.

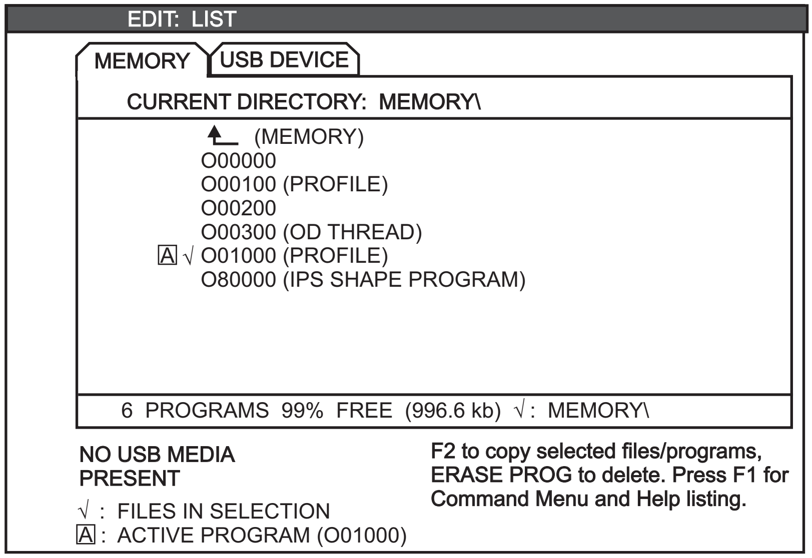
Data is incremented by entering a value on the input line and pressing WRITE, or overridden by pressing F1.

When highlighting TS POSITION, pressing Z FACE MEAS takes the value of the B axis and places it in TS POSITION. When highlighting X CLEARANCE, pressing X DIA MEAS takes the value of the X axis and places it in X CLEARANCE. When highlighting Z CLEARANCE, pressing Z FACE MEAS takes the value of the Z axis and places it in Z CLEARANCE.

Pressing ORIGIN when highlighting X CLEARANCE sets clearance to max travel. Pressing ORIGIN when highlighting Z CLEARANCE sets clearance to zero.

SELECT PROGRAM

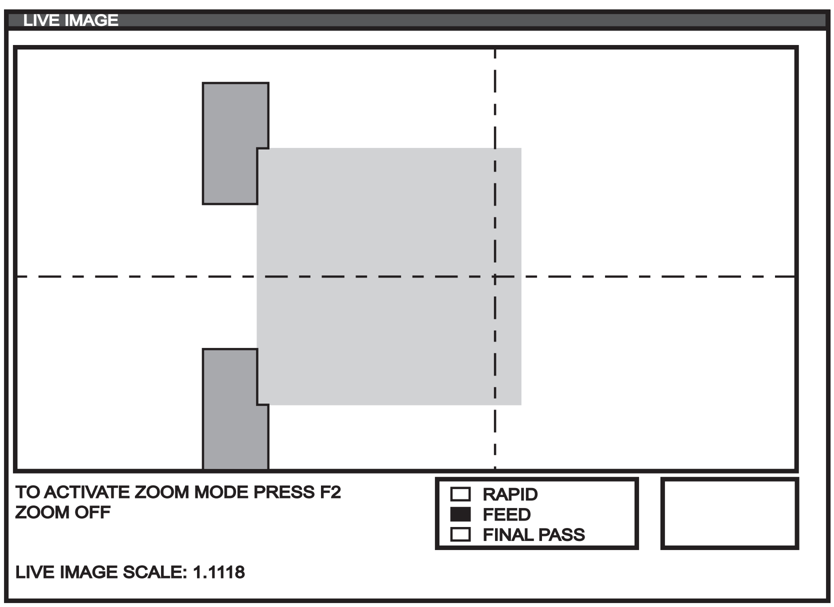
1. To select the desired program, press LIST PROG to display the EDIT: LIST screen. Select the MEMORY tab and press WRITE/ENTER to display CURRENT DIRECTORY: MEMORY\ screen.



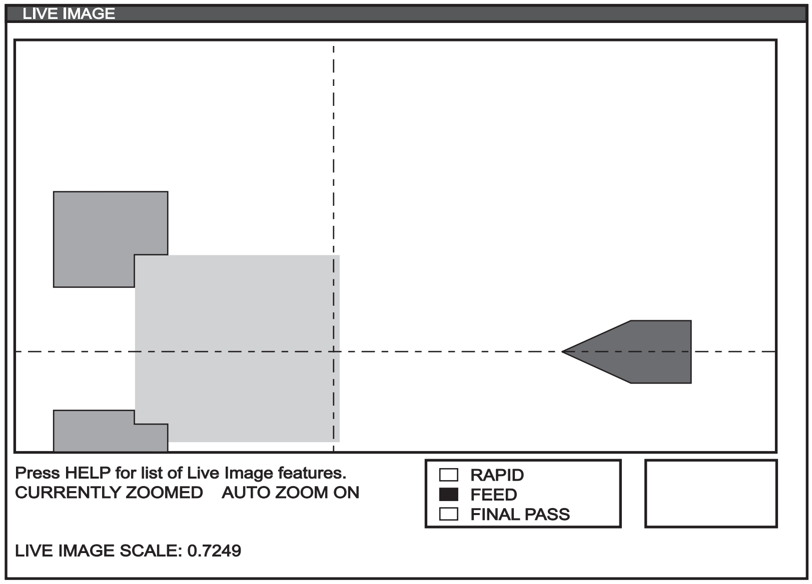
2. Select a program (i.e., O01000) and press WRITE/ENTER to choose it as the active program.

RUN PART

1. Press MEM, then CURNT COMDS, then PAGE UP. When the screen appears, press ORIGIN to display the Live Image screen with stock drawn.



NOTE: Press F2 to enter ZOOM mode. Use the PAGE UP and PAGE DOWN Buttons to zoom the display and the direction buttons to move the display. Press WRITE/ENTER when the desired zoom is achieved. Press ORIGIN to return to zero zoom.



NOTE: Press F4 to auto zoom to the part. Press F1 to save a zoom and press F3 to load a zoom setting.

Press HELP for a pop-up containing a list of Live Image features.

LIVE IMAGE HELP	CANCEL - Exit
SAVE ZOOM SETTINGS	(F1)
TOGGLE ZOOM MODE	(F2)
RESTORE ZOOM SETTINGS	(F3)
TURN ON/OFF AUTO ZOOM	(F4)
ZOOM OUT	PAGE UP)
ZOOM IN	(PAGE DOWN)
MOVE ZOOM WINDOW	(ARROW KEYS)
SELECT ZOOM SIZE	(WRITE)
CLEAR IMAGE	(HOME)
RESET LIVE IMAGE	(ORIGIN)

Stores zoom settings to be restored later
by pressing F3.

2. Press CYCLE START. The following warning will pop up on the screen.

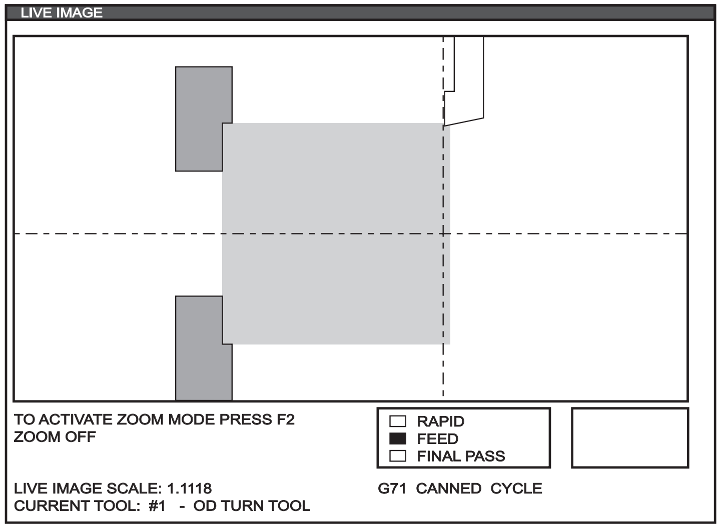
CYCLE START WARNING

LIVE IMAGE ACTIVE

Press CYCLE START to
continue or RESET to
stop.

3. Press CYCLE START again to run the program. When a program is running and tool data has been set up, the Live Image screen shows the tool working the part in real time as the program runs.

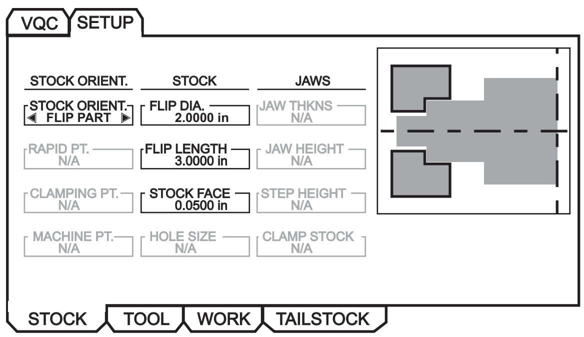
NOTE: When the barfeeder reaches G-Code 105, the part is refreshed.



NOTE: Data displayed on the screen while the program is running includes: program, main spindle, machine position and timers and counters.

Flipping a Part

A graphical representation of a part that has been flipped manually by the machinist is depicted by adding the following comments to the program following an M00. Press F4 to enter Live Image code to the program.



Live Image will redraw the part with a flipped orientation, and with the chuck jaws clamped at a position specified by x and y within the comment “(CLAMP)(x y)” if the comments “(FLIP PART)” and “(CLAMP)(x y)” follow the M00 STOP PROG instruction in the program.

O00000 ;

[Code for first operation of Live Image]

[Code for first operation of machined part]

M00 ;

G20 (INCH MODE); (Start of Live Image Information for flipped part)

(FLIP PART) ;

(CLAMP) ([2.000, 3.0000]) ; ([Diameter, Length]) (End of Live Image Information flipped part)

;

M01 ;

;

[Part Program for the second operation];