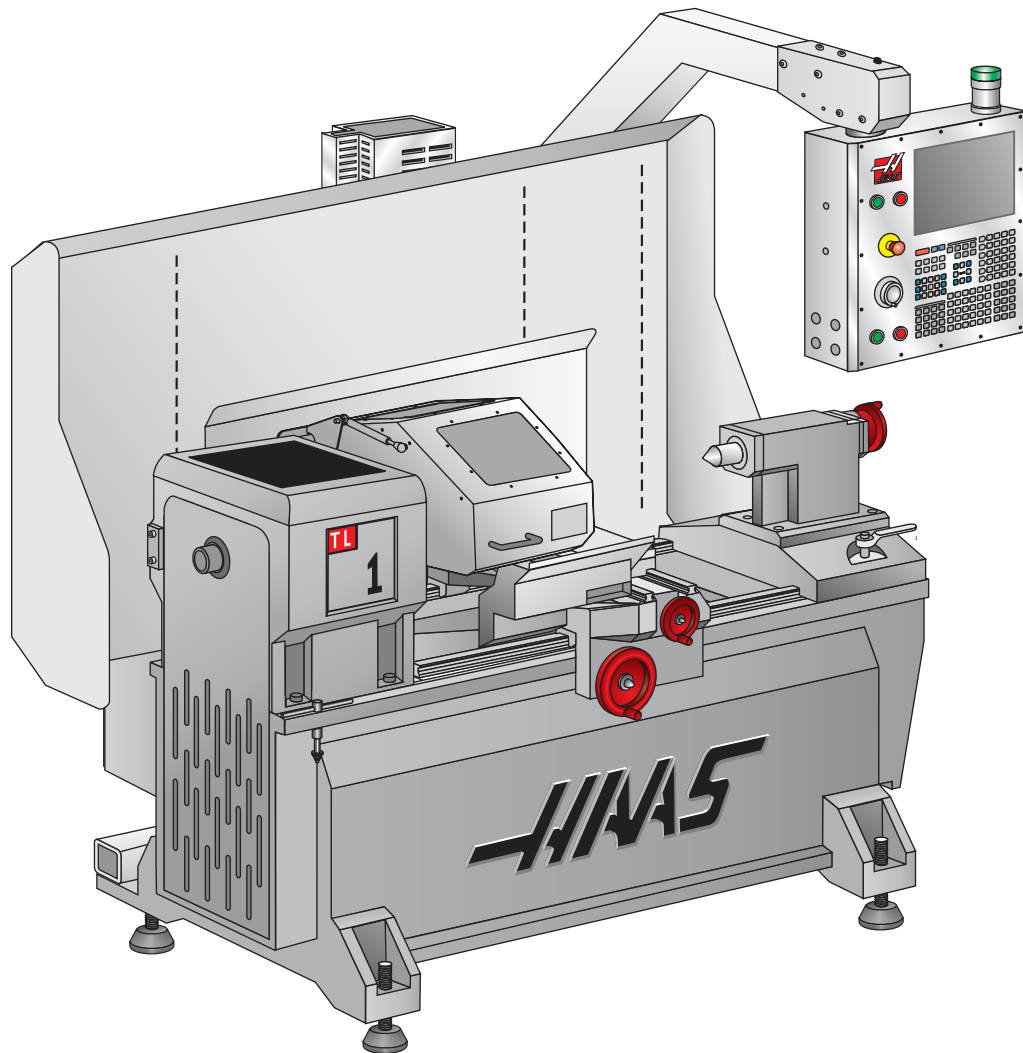


Intuitive Programming System Walk-Through For Lathes



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.

MANUAL	UP	MFER	& FACE	FER & RADIUS	L & TAP	ADING	AD RE-CUT	MING
--------	----	------	--------	--------------	---------	-------	-----------	------

X AND Z AXES
THE AXES CAN BE ELECTRONICALLY LOCKED AND UNLOCKED. THIS IS SHOWN BY XZ-MAN DISPLAYED AT THE BOTTOM OF THE SCREEN. IN THIS MODE BOTH THE X AND Z AXES ARE UNLOCKED AND CAN BE POSITIONED USING THE MANUAL HAND WHEELS. PRESSING [SHIFT] AND EITHER [+X] OR [-X], [+Z], OR [-Z] WILL ELECTRONICALLY LOCK THAT AXIS. PRESSING [SHIFT] AND THE SAME BUTTON A SECOND TIME WILL UNLOCK THE AXIS.

SPINDLE
THE SPINDLE IS COMMANDED 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.

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.

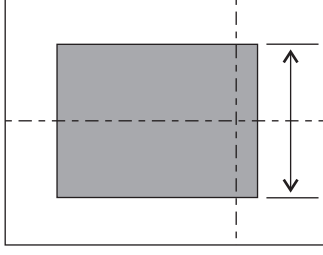
The spindle speed override keys (+/- 10%) can be used to adjust the commanded speed. This also works on most screens.

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

SETUP

Stock Setup

MANU	SETUP	FN & FACE	FER & RADIUS	L & TAP	ADING	AD RE-CUT	MING
STOCK		JAWS		BAR FEEDER			
STOCK DIA. 3.0000 in		JAW THICKNESS 2.0000 in		PUSH LENGTH N/A			
STOCK LENGTH 3.0000 in		JAW HEIGHT 2.0000 in					
STOCK FACE 0.1000 in		STEP HEIGHT 1.5000 in					
HOLE SIZE 0.0000 in		CLAMP STOCK 0.5000 in					



STOCK	TOOL	WORK	TAILSTOCK

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.

MANU	SETUP	FN & FACE	FER & RADIUS	L & TAP	ADING	AD RE-CUT	MING
TOOL <input type="text" value="1"/>		Z WEAR <input type="text" value="0.0000 in"/>		TL THICKNESS <input type="text" value="0.0000 in"/>		 <p>Selected Tool: 1 Active Tool: 1</p> <p>Press [TURRET FWD] or [TURRET REV] to change the selected tool.</p> <p>Press [NEXT TOOL] to make selected tool active.</p>	
TOOL TYPE <input type="text" value="CUT OFF"/>		RADIUS <input type="text" value="0.0000 in"/>		INSRT THCKNES <input type="text" value="0.0000 in"/>			
OFFSET NUM <input type="text" value="1"/>		TIP <input type="text" value="0"/>		TOOL NOSE <input type="text" value="0 deg"/>			
X OFFSET <input type="text" value="0.0000 in"/>		TOOL SHANK <input type="text" value="0.0000 in"/>		INSERT HEIGHT <input type="text" value="0.0000 in"/>			
X WEAR <input type="text" value="0.0000 in"/>		TOOL LENGTH <input type="text" value="0.0000 in"/>		FROM CENTER <input type="text" value="0.0000 in"/>			
Z OFFSET <input type="text" value="0.0000 in"/>		STEP HEIGHT <input type="text" value="0.0000 in"/>		DIAMETER <input type="text" value="0.0000 in"/>			
STOCK		TOOL		WORK		TAILSTOCK	

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

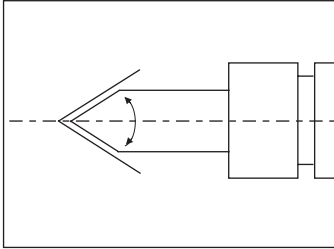
MANU	SETUP	IN & FACE	FER & RADIUS	L & TAP	ADING	AD RE-CUT	MING
<div> <div>Wrk Zero Ofst 54</div> <div> X Offset 0. </div> <div> Z Offset 0. </div> </div>							
<div> <div>STOCK</div> <div>TOOL</div> <div>WORK</div> <div>TAILSTOCK</div> </div>							

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

MANU	SETUP	IN & FACE	FER & RADIUS	L & TAP	ADING	AD RE-CUT	MING
<div> <div> <div>LIVE CTR ANG 60.000 deg</div> <div>TS POSITION 0.0000 in</div> <div>X CLEARANCE -10.0000 in</div> <div>DIAMETER 1.2500 in</div> <div>TS OFFSET 0.0000 in</div> <div>Z CLEARANCE -0.5000 in</div> <div>LENGTH 2.0000 in</div> <div>RETRACT DIST 0.0000 in</div> <div>ADVANCE DIST 0.0000 in</div> </div> <div>  <div>TS HOLD POINT 0.0000 in</div> <div> <p>NOT MODIFIABLE TS HOLD POINT is the sum of TS POSITION and TS OFFSET and is stored in setting 107.</p> </div> </div> </div>							
<div> <div>STOCK</div> <div>TOOL</div> <div>WORK</div> <div>TAILSTOCK</div> </div>							

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.

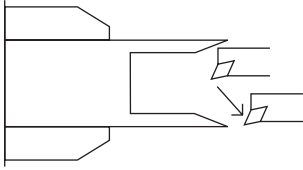
MANU	SE	TURN & FACE	FER & RADIUS	L & TAP	ADING	AD RE-CUT	MING
------	----	-------------	--------------	---------	-------	-----------	------

TOOL NUMBER
1

WORK OFFSET
54

X POSITION
0.0000 in

Z POSITION
0.0000 in



Press <CYCLE START> to run
in MDI or <F4> to record
output to a program

RAPID	FEED	OD TURN	ID TURN	FACE	PROFILE
-------	------	---------	---------	------	---------

Tool Number – Enter the tool to be used.

Work Offset – Enter the work offset to be used.

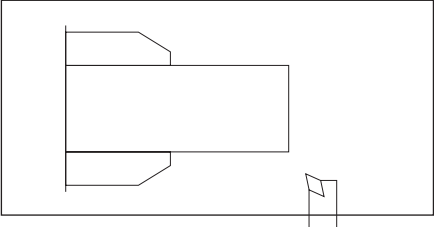
X Position – Enter end point or move tool to end point desired. Press X DIA. MEAS to record this position.

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.

MANU	SET	TURN & FACE	FER & RADIUS	L & TAP	ADING	AD RE-CUT	VING				
TOOL NUMBER 1		FEED PER REV 0.0000 in									
WORK OFFSET 54		SPINDLE RPM 0.0000 in									
DELTA X 0.0000 in											
DELTA Z 0.0000 in											
Press <CYCLE START> to run in MDI or <F4> to record output to a program											
RAPID		FEED		OD TURN		ID TURN		FACE		PROFILE	

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).

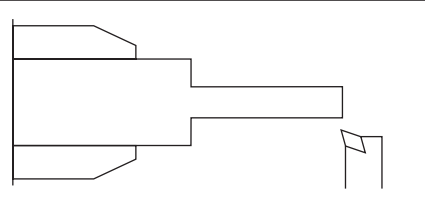
***Spindle RPM** – Enter the spindle RPM.

***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.

MANU	SET	TURN & FACE	FER & RADIUS	L & TAP	ADING	AD RE-CUT	MING
TOOL NUMBER	DIA. TO CUT	MAX RPM					
1	0.0000 in	1000					
WORK OFFSET	Z DIMENSION	SFM					
54	0.0000 in	500					
Z START PT	DEPTH OF CUT	FILLET RADII	Press <CYCLE START> to run in MDI or <F4> to record output to a program				
0.0000 in	0.0000 in	0.0000 in					
OUTSIDE DIA.	FEED PER REV	TOOL NOSE					
0.0000 in	0.0000 in	0.0000 in					
<div> <div>RAPID</div> <div>FEED</div> <div>OD TURN</div> <div>ID TURN</div> <div>FACE</div> <div>PROFILE</div> </div>							

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.

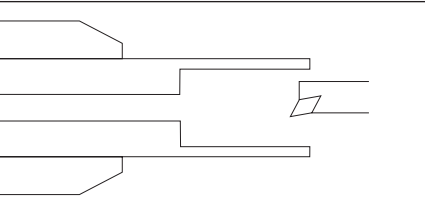
SFM – Enter the Surface Feed per Minute.

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.

MANU	SET	TURN & FACE	FER & RADIUS	L & TAP	ADING	AD RE-CUT	MING
TOOL NUMBER	DIA. TO CUT	MAX RPM					
1	0.0000 in	1000					
WORK OFFSET	Z DIMENSION	SFM					
54	0.0000 in	200					
Z START PT	DEPTH OF CUT	FILLET RADII	Press <CYCLE START> to run in MDI or <F4> to record output to a program				
0.0000 in	0.0000 in	0.0000 in					
INSIDE DIA.	FEED PER REV	TOOL NOSE					
0.0000 in	0.0000 in	0.0000 in					
<div> <div>RAPID</div> <div>FEED</div> <div>OD TURN</div> <div>ID TURN</div> <div>FACE</div> <div>PROFILE</div> </div>							

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.

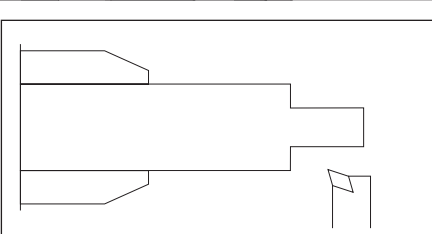
Fillet Radii – Enter the corner fillet radii or enter '0' for none.

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.

MANU	SET	TURN & FACE	FER & RADIUS	L & TAP	ADING	AD RE-CUT	VING
TOOL NUMBER 1		Z DIMENSION 0.0000 in		SFM 200			
WORK OFFSET 54		DEPTH OF CUT 0.0350 in					
OUTSIDE DIA. 0.0000 in		FEED PER REV 0.0060 in					
DIA. TO CUT 0.0000 in		MAX RPM 1000					
							
Press <CYCLE START> to run in MDI or <F4> to record output to a program							
RAPID		FEED		OD TURN		ID TURN	
		FACE		PROFILE			

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.

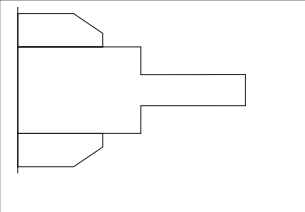
SFM – Enter the Surface Feed per Minute.

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.

MANU	SET	TURN & FACE	FER & RADIUS	L & TAP	ADING	AD RE-CUT	MING
TOOL NUMBER 1		X DISTANCE N/A		SPINDLE DIR FORWARD		 <p>Press <CYCLE START> to run in MDI or <F4> to record output to a program</p>	
WORK OFFSET 54		Z DISTANCE N/A		CUTTER COMP OFF			
CUT TYPE HRZ TYPE I		NUM OF PASSES N/A		COOLANT OFF			
X STOCK ALLOW 0.0000 in		FEED PER REV 0.0000 in		MIRROR X OFF			
Z STOCK ALLOW 0.0000 in		MAX RPM 0		GRAPHIC MODE OFF			
DEPTH OF CUT 0.0000 in		SFM 0		PROFILE NUMBR 0			
RAPID		FEED		OD TURN		ID TURN	
FACE		PROFILE					

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.

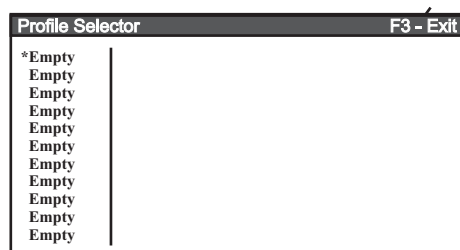
Graphic Mode – Use the left/right cursor keys to turn on/off Graphic Mode. This allows you to view the process in graphics.

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

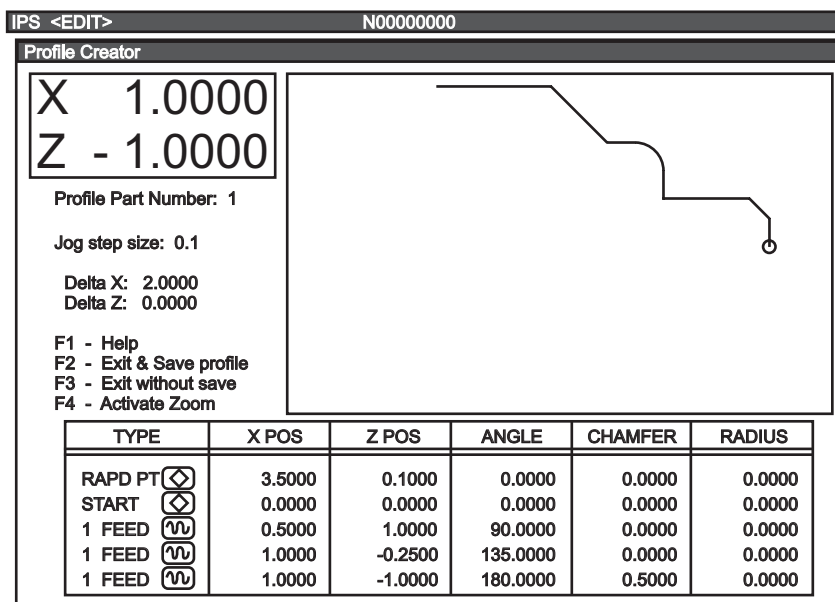
Advanced Users: In the full CNC Mode, this is a G71 command.

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.



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.



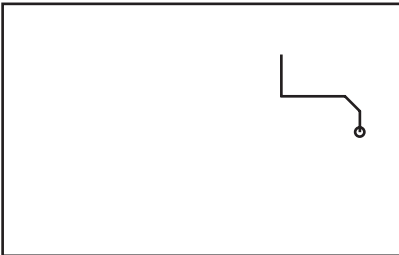
Go to the beginning of the next line. Press 1 to activate a Feed move.

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



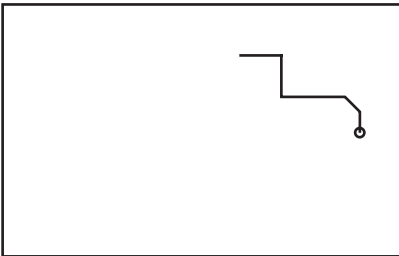
Go to the beginning of the next line. Press 1 to activate a Feed move.

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

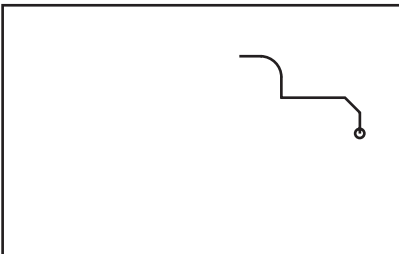


Go to the beginning of the next line. Press 1 to activate a Feed move.

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

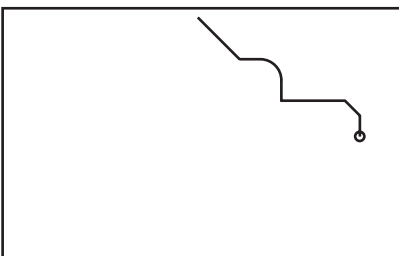


Use the cursor keys to go back to the previous line and select the Radius column. Enter 0.25, press Write/Enter and use the cursor keys to come back to this line.



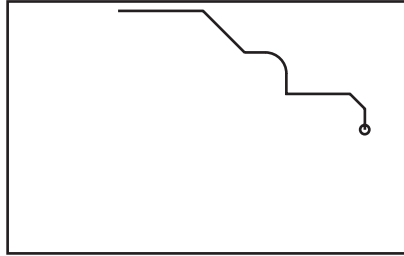
Go to the beginning of the next line. Press 1 to activate a Feed move.

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.



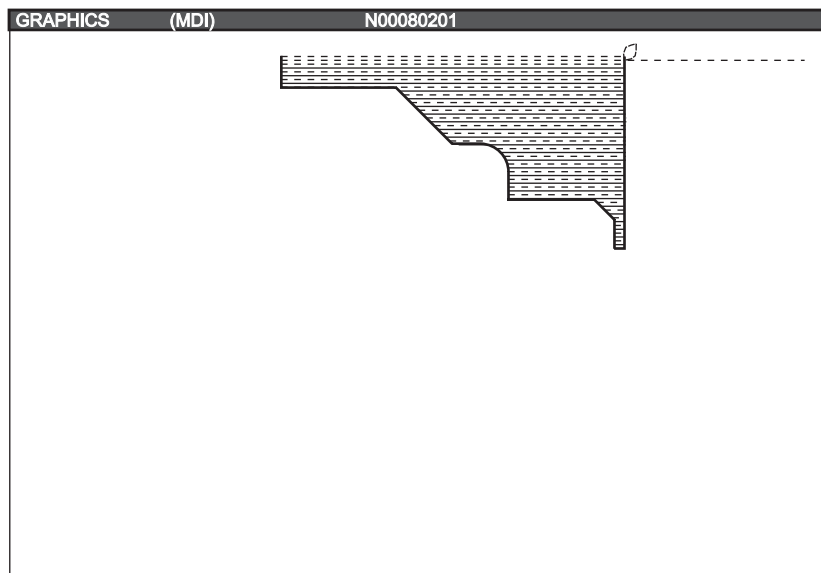
j. Press F2 to Save and exit the Profile Creator.

k. Press Cycle Start to cut the profile.

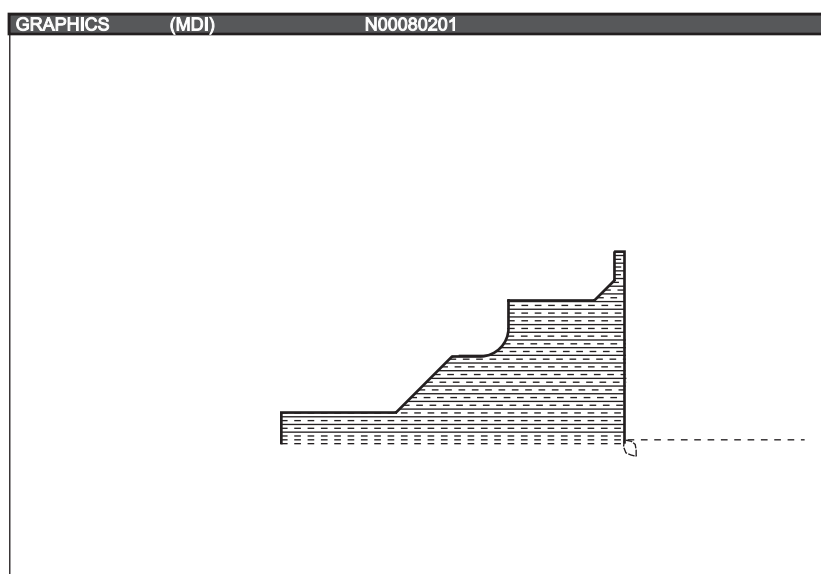
NOTE: The program may be saved to memory from MDI by typing in 0xxxxx and pushing the Alter key. This action moves the program from MDI memory.

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.

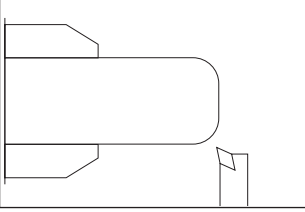
Move Down To Next Data Box - Moves down to next data box below its current location. Will not move if already at the bottom of the table.

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.

MANU	SE	TURN &	CHAMFER & RADIUS	L & TAP	ADING	AD RE-CUT	MING
TOOL NUMBER		RADIUS		TOOL NOSE			
1		0.0000 in		0.0310 in			
WORK OFFSET		DEPTH OF CUT		SFM			
54		0.0500 in		200			
Z START PT		FEED PER REV		Press <CYCLE START> to run in MDI or <F4> to record output to a program			
0.0000 in		0.0060 in					
OUTSIDE DIA.		MAX RPM					
0.0000 in		1000					
OD RADIUS ID RADIUS OD CHAMFER ID 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 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 revolution. This is distance tool will move for each revolution of the spindle.

MAX RPM – Enter the maximum spindle turning speed.

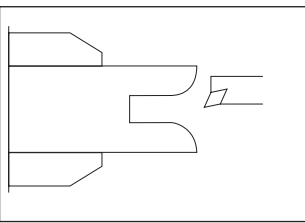
Tool Nose – Enter the tool nose radius. This is the radius of the selected tool. Normally this information is included with the tool.

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.

MANU	SE	TURN &	CHAMFER & RADIUS	L & TAP	ADING	AD RE-CUT	MING
TOOL NUMBER 1		RADIUS 0.0000 in		TOOL NOSE 0.0000 in			
WORK OFFSET 54		DEPTH OF CUT 0.0000 in		SFM 200			
Z START PT 0.0000 in		FEED PER REV 0.0000 in					
INSIDE DIA. 0.0000 in		MAX RPM 1000					
Press <CYCLE START> to run in MDI or <F4> to record output to a program							
OD RADIUS		ID RADIUS		OD CHAMFER		ID 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 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 revolution. This is the distance the tool will move for each revolution of the spindle.

MAX RPM – Enter the maximum spindle turning speed.

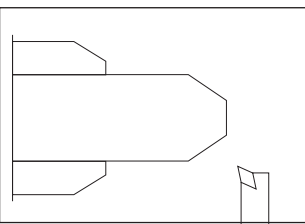
Tool Nose – Enter the tool nose radius. This is the radius of the selected tool. Normally this information is included with the tool.

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.

MANU	SE	TURN &	CHAMFER & RADIUS	L & TAP	ADING	AD RE-CUT	MING
TOOL NUMBER 1		CHAMFER 0.0000 in		MAX RPM 1000			
WORK OFFSET 54		ANGLE 0.000 deg.		TOOL NOSE 0.0315 in			
Z START PT 0.0000 in		DEPTH OF CUT 0.0500 in		SFM 200			
OUTSIDE DIA. 0.0000 in		FEED PER REV 0.0060 in					
<p>Press <CYCLE START> to run in MDI or <F4> to record output to a program</p>							
OD RADIUS		ID RADIUS		OD CHAMFER		ID 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.

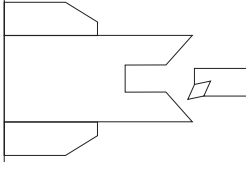
Tool Nose – Enter the tool nose radius.

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.

MANU	SE	TURN &	CHAMFER & RADIUS	L & TAP	ADING	AD RE-CUT	VING
TOOL NUMBER 1		CHAMFER 0.0000 in		MAX RPM 1000			
WORK OFFSET 54		ANGLE 0.000 deg.		TOOL NOSE 0.0315 in			
Z START PT 0.0000 in		DEPTH OF CUT 0.0400 in		SFM 200			
INSIDE DIA. 0.0000 in		FEED PER REV 0.0060 in					
Press <CYCLE START> to run in MDI or <F4> to record output to a program							
OD RADIUS		ID RADIUS		OD CHAMFER		ID 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.

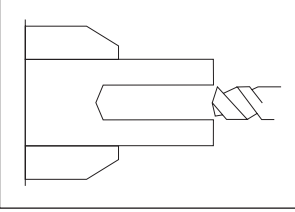
Tool Nose – Enter the tool nose radius.

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.

MANU	SE	TURN &	CHAMFER & RA	DRILL & TAP	ADING	AD RE-CUT	VING
TOOL NUMBER 1		FEED PER REV 0.0030 in		 <p>Press <CYCLE START> to run in MDI or <F4> to record output to a program</p>			
WORK OFFSET 54		MAX RPM 1000					
Z START PT 0.0000 in		DWELL 0.5000 sec					
DEPTH OF HOLE 0.0000 in							
<div>DRILL</div> <div>PECK DRILL</div> <div>TAP</div> <div>REVERSE TAP</div>							

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).

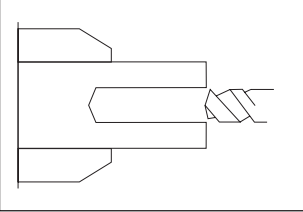
MAX RPM – Enter the spindle RPM.

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.

MANU		SE	TURN &	CHAMFER & RA	DRILL & TAP	ADING	AD RE-CUT	MING
TOOL NUMBER 1		PECK DISTANCE 0.0000 in						
WORK OFFSET 54		FEED PER REV 1000						
Z START PT 0.0000 in		SPINDLE RPM 0.0000 sec						
DEPTH OF HOLE 0.0000 in								
Press <CYCLE START> to run in MDI or <F4> to record output to a program								
DRILL		PECK DRILL		TAP		REVERSE TAP		

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.

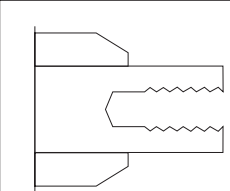
Feed Per Rev – Enter feed per revolution (distance the tool will move for each revolution of the spindle).

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.

MANU	SE	TURN &	CHAMFER & RA	DRILL & TAP	ADING	AD RE-CUT	VING
TOOL NUMBER 1		TPI 0.0000 in		 <p>Press <CYCLE START> to run in MDI or <F4> to record output to a program</p>			
WORK OFFSET 54		SPINDLE RPM 350					
Z START PT 0.0000 in							
TAP DEPTH 0.0000 in							
DRILL		PECK DRILL		TAP		REVERSE TAP	

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 – Enter 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.

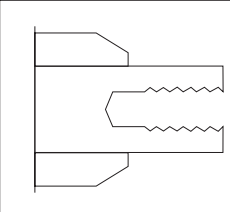
Spindle RPM – Enter spindle RPM (commanded spindle speed). Spindle speed should not exceed 500 RPM.

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.

MANU	SE	TURN &	CHAMFER & RA	DRILL & TAP	ADING	AD RE-CUT	VING
TOOL NUMBER 1		TPI 0.0000 in		 <p>Press <CYCLE START> to run in MDI or <F4> to record output to a program</p>			
WORK OFFSET 54		SPINDLE RPM 350					
Z START PT 0.0000 in							
TAP DEPTH 0.0000 in							
DRILL		PECK DRILL		TAP		REVERSE TAP	

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.

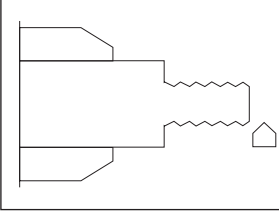
Spindle RPM – The commanded spindle speed. The spindle speed should not exceed 500 RPM.

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.

MANU	SE	TURN &	CHAMFER & RA	DRILL	THREADING	AD RE-CUT	VING
TOOL NUMBER 1		MINOR 0.0000 in		SPINDLE RPM 1000		 <p>Press <CYCLE START> to run in MDI or <F4> to record output to a program</p>	
WORK OFFSET 54		MAJOR 0.0000 in		TAPER 0.0000 in			
Z START PT 0.0000 in		TPI 0.000		THREAD DIR ◀ RIGHT ▶			
THREAD LENGTH 0.0000 in		DEPTH OF CUT 0.0150 in		CHAMFER ◀ OFF ▶			
				COOLANT ◀ OFF ▶			
OD THREAD		ID THREAD		OD THREAD REPAIR		ID THREAD REPAIR	

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.

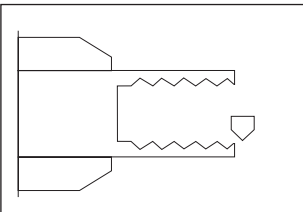
Chamfer – ‘ON’ turns on chamfer at end of threads. ‘OFF’ turns off chamfer at end of threads. (Check setting 95, 96, 86, 99).

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.

MANU	SE	TURN &	CHAMFER & RA	DRILL	THREADING	AD RE-CUT	TING
TOOL NUMBER		MINOR		SPINDLE RPM			
1		0.0000 in		1000			
WORK OFFSET		MAJOR		TAPER			
54		0.0000 in		0.0000 in			
Z START PT		TPI		THREAD DIR			
0.0000 in		0.000		RIGHT			
THREAD LENGTH		DEPTH OF CUT		CHAMFER		Press <CYCLE START> to run in MDI or <F4> to record output to a program	
0.0000 in		0.0150 in		OFF			
				COOLANT			
				OFF			

OD THREAD	ID THREAD	OD THREAD REPAIR	ID THREAD REPAIR
-----------	-----------	------------------	------------------

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.

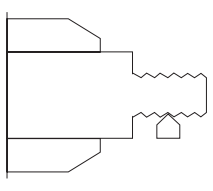
Chamfer – ‘0’ turns on chamfer at end of threads. ‘1’ turns off chamfer at end of threads. (Check setting 95, 96, 86, 99).

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.

MANU	SE	TURN &	CHAMFER & RA	DRILL	THREADING	AD RE-CUT	MING
REFERENCE NOT SET		THREAD LENGTH 0.0000 in		X OFFSET 0.0000 in			
TPI ◀ 48.0 ▶		THRDS TO CLR 3		Z OFFSET 0.0000 in			
THRD HEIGHT 0.0131 in		DEPTH OF CUT 0.0125 in		THREAD DIR ◀ RIGHT ▶			
TL CLEARANCE 0.1000 in		SPINDLE RPM 350		CHAMFER ◀ OFF ▶			
NO OF THREADS 0		TAPER 0.0000 in		COOLANT ◀ OFF ▶			
Press <CYCLE START> to run in MDI or <F4> to record output to a program							
OD THREAD		ID THREAD		OD THREAD REPAIR		ID THREAD REPAIR	

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.

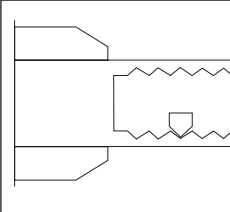
Thread Dir – Enter ‘0’ for right hand threads or enter ‘1’ for left hand threads.

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.

MANU	SE	TURN &	CHAMFER & RA	DRILL	THREADING	AD RE-CUT	VING
REFERENCE NOT SET		THREAD LENGTH 0.0000 in		X OFFSET 0.0000 in			
TPI 48.0		THRDS TO CLR 3		Z OFFSET 0.0000 in			
THRD HEIGHT 0.0131 in		DEPTH OF CUT 0.0125 in		THREAD DIR RIGHT			
TL CLEARANCE 0.1000 in		SPINDLE RPM 350		CHAMFER OFF			
NO OF THREADS 0		TAPER 0.0000 in		COOLANT OFF			
Press <CYCLE START> to run in MDI or <F4> to record output to a program							
OD THREAD		ID THREAD		OD THREAD REPAIR		ID THREAD REPAIR	

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.

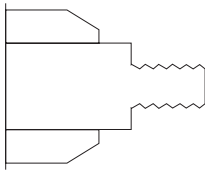
Chamfer – '0' turns on chamfer at end of threads. '1' turns off chamfer at end of threads. (Check setting 95, 96, 86, 99).

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.

MANU	SE	TURN &	CHAMFER & RA	DRILL	THREA	THREAD RE-CUT	VING
WORK OFFSET		STEP 1 OF 3					
G54							
Z OFFSET		0.0000 in					
PGM NUMBER		0					
X CLEARANCE		0.0000 in					
Z CLEARANCE		0.0000 in					
ORIENT RPM		10					
SET OFFSET & PUSH BACK		THREAD TEACH POSITION		SET REFERENCE & RE-CUT			

Work Offset –

Z Offset –

Program Number –

X Clearance –

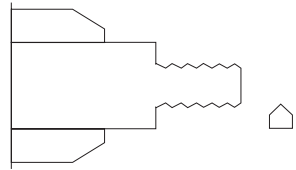
Z Clearance –

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.

MANU	SE	TURN &	CHAMFER & RA	DRILL	THREA	THREAD RE-CUT	VING
TOOL NUMBER		STEP 2 OF 3					
1							
WORK OFFSET		G54					
ORIENT SP		N/A					
DEGREES		N/A					
X POSITION		0.0000 in					
Z POSITION		0.0000 in					
SET OFFSET & PUSH BACK		THREAD TEACH POSITION		SET REFERENCE & RE-CUT			

Tool Number –

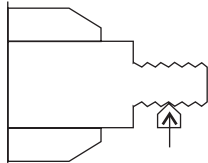
Work Offset –

Orient SP –
Degrees –
X Position –
Z Position –

Thread Re-Cut - Set Reference & Re-Cut

Step 3 of 3

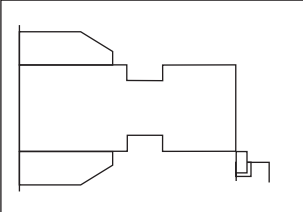
This mode is used to teach the threading tool OD or ID. Then re-cut the threads using an existing program in program memory.

MANU	SE	TURN &	CHAMFER & RA	DRILL	THREA	THREAD RE-CUT	VING
REFERENCE		STEP 3 OF 3					
NOT SET							
PGM NUMBER							
0							
X CLEARANCE							
0.0000 in							
Z CLEARANCE							
0.0000 in							
X TOOL WEAR							
0.0000 in							
Z OFFSET		ORIENT RPM					
0.0000 in		10					
SET OFFSET & PUSH BACK			THREAD TEACH POSITION			SET REFERENCE & RE-CUT	

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.

MANU	SE	TURN &	CHAMFER & RA	DRILL	THREA	THREAD R	GROOVING
TOOL NUMBER 1		DIA. TO CUT 0.0000 in		FEED PER REV 0.0000 in			
WORK OFFSET 54		Z DIMENSION 0.0000 in		MAX RPM 1000			
Z START PT 0.0000 in		GROOVE WIDTH 0.0000 in		SFM 200			
OUTSIDE DIA. 0.0000 in		TOOL WIDTH 0.0000 in					
Press <CYCLE START> to run in MDI or <F4> to record output to a program							
OD GROOVE		ID GROOVE		PART OFF		PART OFF WITH 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 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.

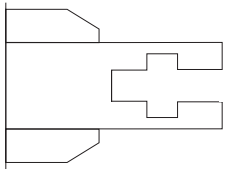
MAX RPM – Enter the maximum spindle turning speed.

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.

MANU	SE	TURN &	CHAMFER & RA	DRILL	THREA	THREAD R	GROOVING
TOOL NUMBER 1		DIA. TO CUT 0.0000 in		FEED PER REV 0.0000 in			
WORK OFFSET 54		Z DIMENSION 0.0000 in		MAX RPM 1000			
Z START PT 0.0000 in		GROOVE WIDTH 0.0000 in		SFM 200			
INSIDE DIA. 0.0000 in		TOOL WIDTH 0.0000 in					
Press <CYCLE START> to run in MDI or <F4> to record output to a program							
OD GROOVE		ID GROOVE		PART OFF		PART OFF WITH 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 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.

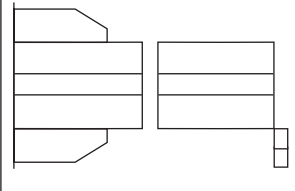
MAX RPM – Enter the maximum spindle turning speed.

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.

MANU	SE	TURN &	CHAMFER & RA	DRILL	THREA	THREAD R	GROOVING
TOOL NUMBER 1		DIA. TO CUT 0.0000 in		MAX RPM 0.0000 in			
WORK OFFSET 54		PART LENGTH 0.0000 in		SFM 1000			
Z START PT 0.0000 in		TOOL WIDTH 0.0000 in					
OUTSIDE DIA. 0.0000 in		FEED PER REV 0.0000 in					
<p>Press <CYCLE START> to run in MDI or <F4> to record output to a program</p>							
OD GROOVE		ID GROOVE		PART OFF		PART OFF WITH 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.

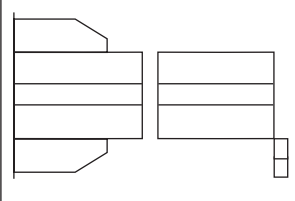
MAX RPM – Enter the maximum spindle turning speed.

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.

MANU	SE	TURN &	CHAMFER & RA	DRILL	THREA	THREAD R	GROOVING
TOOL NUMBER 1		DIA. TO CUT 0.0000 in		MAX RPM 0.0000 in			
WORK OFFSET 54		PART LENGTH 0.0000 in		PECK VALUE 1000			
Z START PT 0.0000 in		TOOL WIDTH 0.0000 in		SFM 200			
OUTSIDE DIA. 0.0000 in		FEED PER REV 0.0000 in					
Press <CYCLE START> to run in MDI or <F4> to record output to a program							
OD GROOVE		ID GROOVE		PART OFF		PART OFF WITH 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.

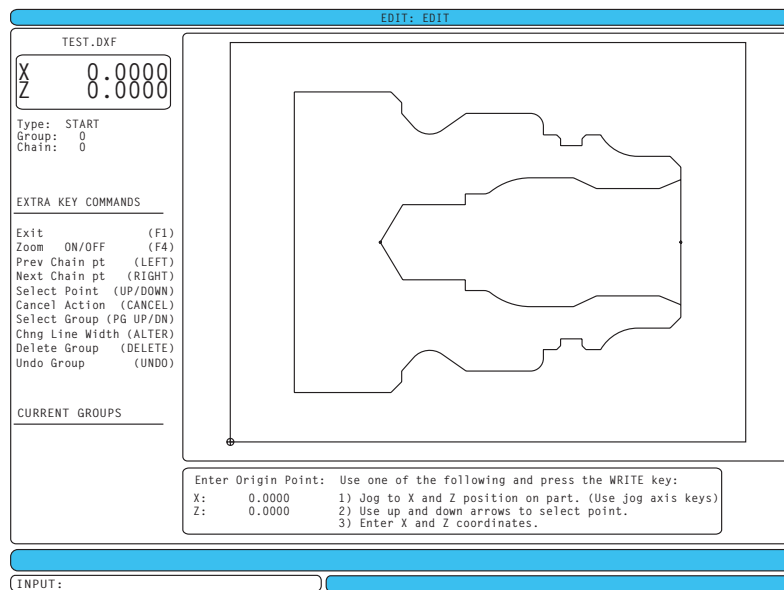
DXF FILE IMPORTER

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.

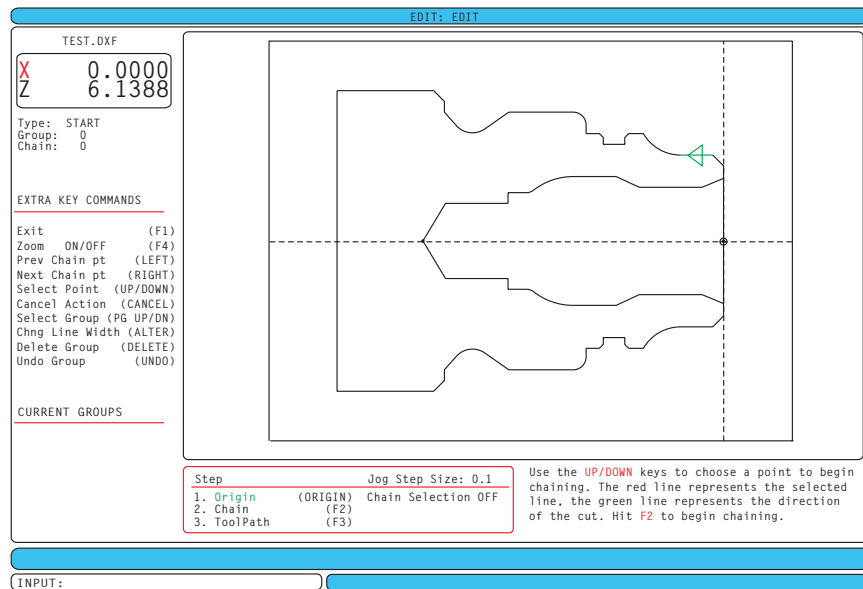
DXF Importer creates programs using simple input given in the following steps:

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:

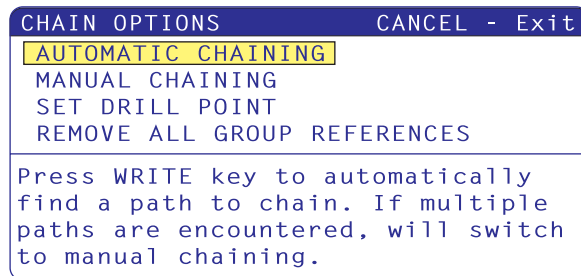
- Point Selection - Use the up and down arrow keys to select a point.
- Jogging - Jog to the X and Z position on a part (use jog axis keys).
- Enter Coordinates - Type in X coordinate and press WRITE/ENTER, then type in Z coordinate and press WRITE/ENTER.



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.

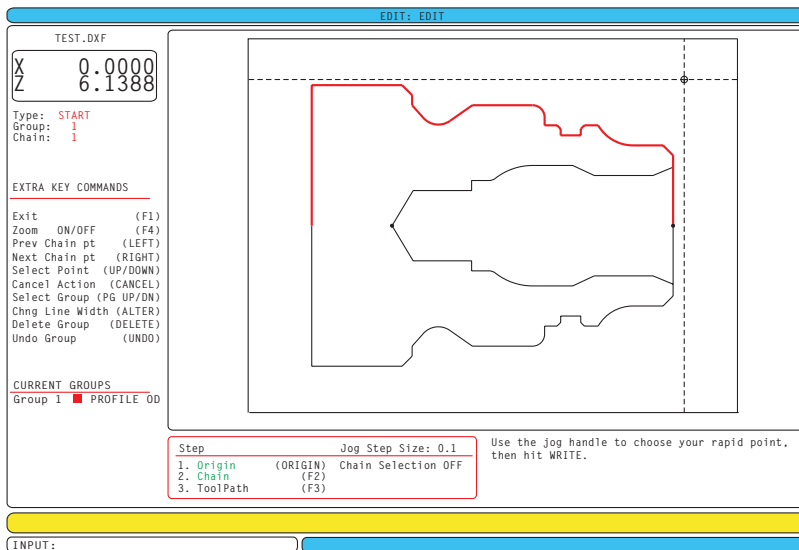
CHAIN/GROUP

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.



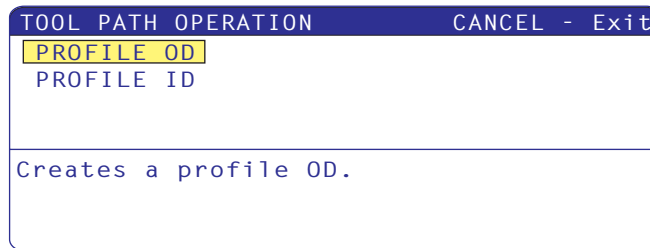
The Automatic Chaining function is typically the best choice as it will automatically plot the tool path for a part feature. Press "Enter" This will change the color of that part feature and add a group to the register under "Current group" on the left hand side of the window.

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.

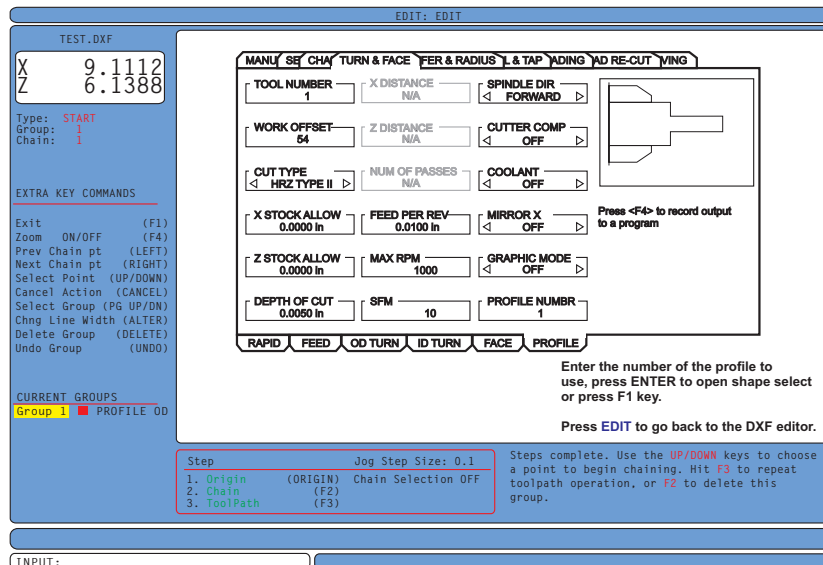


SELECT TOOL PATH

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.

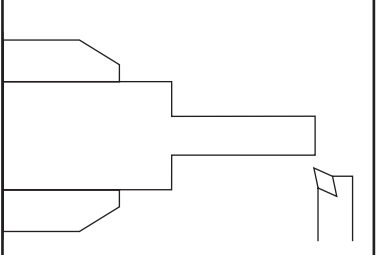
Press F4 to save the toolpath once the template is completed. Refer to the "IPS Recorder" section for details on saving the path into a new or existing program.

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 PROGRAM/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:

MANUAL	SETUP	TURN & FACE	HAMFER AND RADIUS	RILL & TAP	THREADING	ROOVING	QC				
TOOL NUMBER 1		DIA TO CUT 0.0000 in		MAX RPM 1000		 <p>Press < CYCLE START > to run in MDI or < F4 > to record output to a program.</p>					
WORK OFFSET 54		Z DIMENSION 0.0000 in		SFM 200							
Z START PT 0.0000 in		DEPTH OF CUT 0.0500 in		FILLET RADII 0.0000 in							
OUTSIDE DIA. 0.0000 in		FEED PER REV 0.0100 in		TOOL NOSE 0.0315 in							
RAPID		FEED		OD TURN		ID TURN		FACE		PROFILE	

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		F4 – CANCEL
1.) Select / Create Program 2.) Output to current program 3.) Cancel		
		program.

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.

MANUAL	SETUP	TURN & FACE	HAMFER AND RADIUS	RILL & TAP	THREADING	ROOVING	QC
--------	-------	-------------	-------------------	------------	-----------	---------	----

TOOL NUMBER 1

WORK OFFSET 54

Z START PT 0.0000 in

OUTSIDE DIA. 0.0000 in

Select / Create Program

F4—CANCEL

000000 (PROGRAM A)

000001 (PROGRAM B)

000002 (PROGRAM C)

000003 (PROGRAM D)

000004 (PROGRAM E)

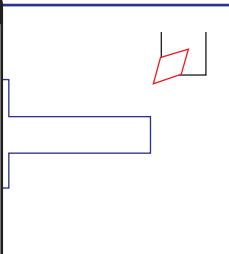
000005 (PROGRAM F)

000006 (PROGRAM G)

Choose a program by using the cursor
keys and press WRITE to select.

or

Enter a '0' followed by a new program
number and press WRITE to create.



CYCLE START >

MDI or <F4>

d output to a

RAPID	FEED	OD TURN	ID TURN	FACE	PROFILE
-------	------	---------	---------	------	---------

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.

OPERATOR KEYBOARD

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.

