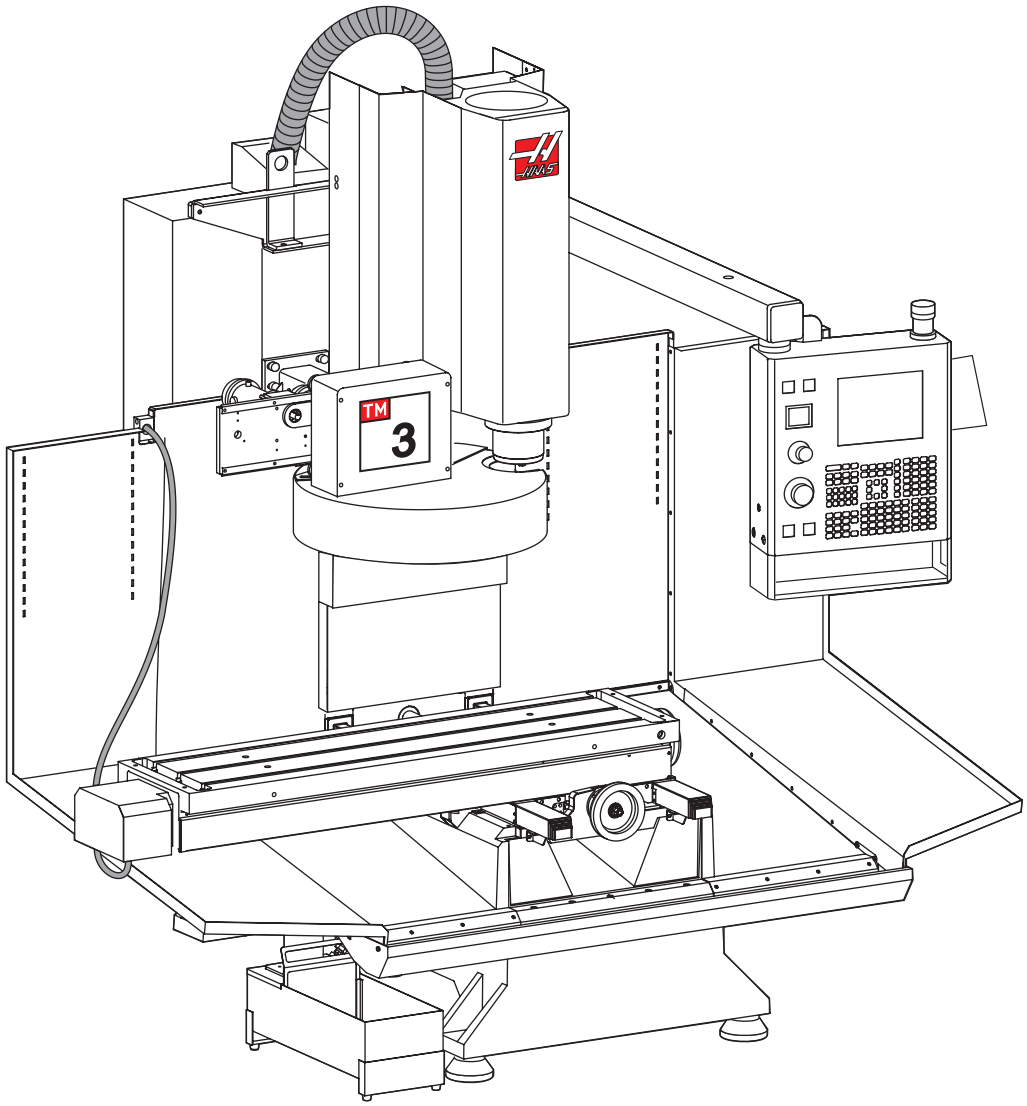
Intuitive Programming System Walk-Through For Mills



Introduction

These instructions provide an in-depth look at each of the Intuitive Programming System (IPS) menus and are to be used with the Mill Operator's manual (96-8000). A more formal description is given for each of the entries to help better define the on-screen help for new users.

A program created through IPS is also accessible in MDI mode. The program can be edited and saved to memory from the full CNC mode, or run in graphics.

Accessing IPS

Software versions prior to 16.xx: The IPS menu is displayed at power up. The IPS initial screen displays current Axis positions and a spindle status (direction / speed) indicator.

16.xx and later: To access IPS, press MDI/DNC, then PRGRM/CONVRS. The IPS tabbed menu appears in the upper-right display pane. Axis positions and spindle status are always available in their respective panes.

NOTE: Some IPS functions vary on 15" display machines, depending on the software version your lathe is equipped with. Where significant differences exist, they are indicated as seen above, or in separate sections. Please verify your software version and ensure that you are following the correct instructions. Please refer to "software version identification" at the end of this document if you are not sure how to check your software version.

Menu Navigation

Navigate tabbed menus using the left and right arrow keys. To select a menu item, press Write/Enter. Some menus have sub-tabs; in this case, use the left and right arrow keys and press Write/Enter to select.

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

Press Cancel to go back one menu level. Pressing Cancel at a top-level menu exits IPS.

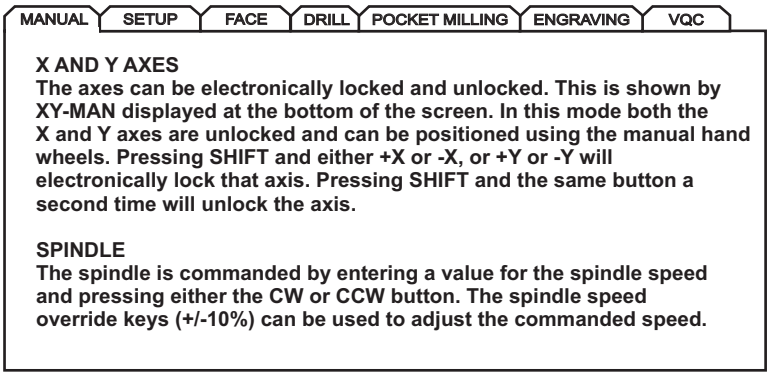
Pressing any of the buttons under the "Display" heading will also exit the IPS menus, as will any of the mode keys (i.e. Edit, Mem, MDI, etc.).

Software versions prior to 16.xx: To return to the IPS menu, press HAND JOG.

16.xx and later: To return to the IPS menu, press MDI/DNC, then PRGRM/CONVRS.

NOTE: Depending on the software version currently installed on the mill, the machine menu displays may vary slightly from those pictured in this manual. Unless indicated otherwise, these differences are simply cosmetic.

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.



15" Display Shown

X and Y Axes

Just below the on-screen text is a line of text that shows the state the mill is in. For example, "X -MAN" means the X -axis is in manual mode (i.e., you can turn the X-axis handwheel, but not the Y-axis). No text beneath the on-screen help means that both axes (X and Y) are locked. In this case, the axes can be jogged by pressing +X/-X or +Y/-Y or by using the electronic jog handle on the pendant. Select a jog speed before using the jog handle. To quickly return to the manual handwheel mode, press Write/Enter while in the Manual tab (look for XY-MAN to be displayed), or from a different tab, press the Shift key and X, then the Shift key and Y.

Spindle

The spindle is controlled using keys on the control pendant. Enter a spindle speed; for example, press 5, then 0, then Write/Enter. This will enter a speed of 50 RPM. 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.

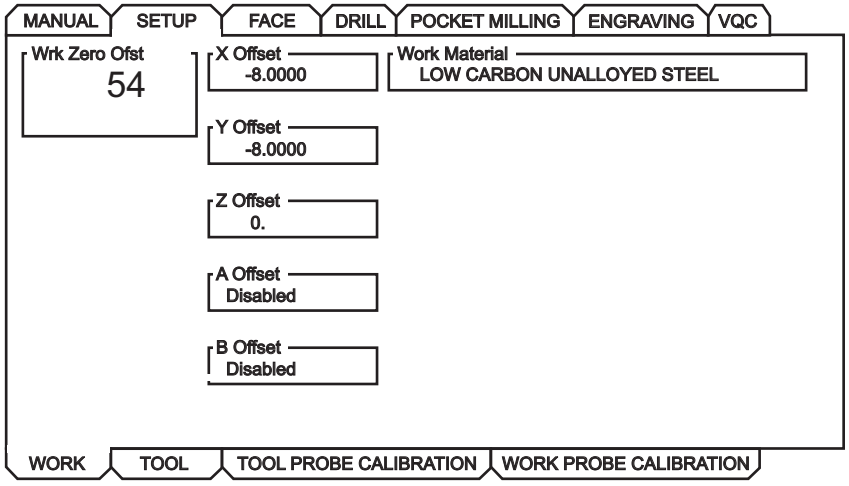
The spindle is stopped by letting go of the hold to run switch, pressing Reset, or pressing the Stop button.

Setup Mode

Select Setup Mode by moving the highlighted tab to the Setup tab and pressing Write/Enter.

Work Tab

The Work tab is displayed by moving to the Work tab on the Setup screen and pressing Write/Enter. The Work tab is used to enter Work Offsets and to select the material. In order for the mill to accurately machine a work piece, it needs to know where the part is located on the table. Jog the mill with a pointer tool in the spindle, until it reaches the top left corner of the part. This position is part zero.



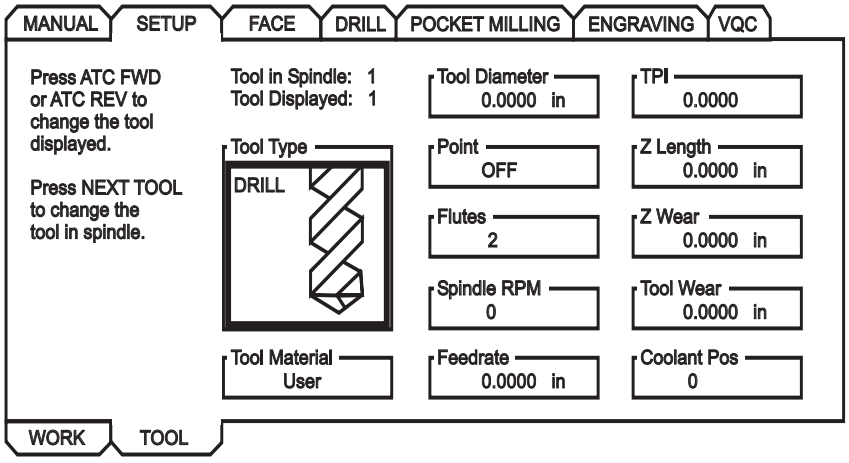
Offsets – Select the required Work Zero Offset by scrolling through the available choices and designating one. Select the X, Y and/or Z Offset setting, and enter a value. Press Write/Enter to add the value to the current value, F1 to set the value, or Part Zero Set to record current position.

Work Material – Select the Work Material setting and use the Up and Down cursor arrows to change the material type. Press Write/Enter to select the material type.

NOTE: Use “No Material Selected” to enter speeds and feeds for tools.

Tool Tab

The Tool tab is displayed by moving to the Tool tab on the Setup Screen and pressing Write/Enter. The Tool tab is used to set up the tools used in the milling operation.



Tool Type Parameters - Drill

Tool Displayed (All Tools) – Current tool number. Use ATC FWD or ATC REV to change the tool displayed.

Tool Type (All Tools) – Right/Left arrows select among 5 tools: Drill, Tap, Shell Mill, End Mill and Center Drill.

Tool Material (Drill, Shell Mill, End Mill, Center Drill) – Right/Left arrows select among 3 tool materials: Carbide, High Speed Steel and User.

Tool Diameter (All Tools) – Enter the actual diameter/radius of the tool.

Point (Drill, Center Drill) – Enter the included angle of the tool. Enter 0 or 180 to cancel.

Flutes (All Tools) – Enter the number of flutes the tool has.

Spindle RPM (All Tools) – Enter the spindle RPM for the tool, when the tool material is set to user.

Feedrate (Drill, Shell Mill, End Mill, Center Drill) – Enter feedrate for tool, when tool material is set to user.

TPI (Tap) – Enter the Threads per Inch for the tap tool.

Z Length (All Tools) – Press TOOL OFFSET MEASUR to record the current position or enter a value.

Z Wear (All Tools) – Enter the amount of wear to the tool length.

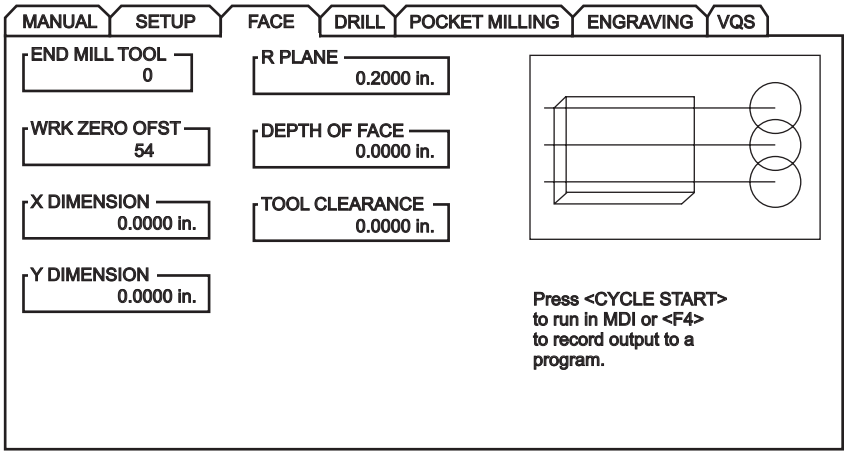
Tool Wear (All Tools) – Enter the amount of wear to the tool's diameter/radius.

Coolant Pos (All Tools) – Enter the Coolant Spigot position.

Face Mode

The Face Mode is displayed by moving to the Face tab and pressing Write/Enter. The Face tab is used to set up any tools to be used in the milling operation.

Face milling is a form of milling that produces a flat surface, generally at right angles to the rotating axis of a cutter. The tool is usually an End Mill.



Face Milling Parameters:

END MILL TOOL – Enter the End Mill tool number.

WRK ZERO OFST – Enter a work zero offset number.

X DIMENSION – Enter the X dimension in width. Must be a positive value.

Y DIMENSION – Enter the Y dimension in width. Must be a positive value.

R PLANE – Enter the location of the retract point above the part.

DEPTH OF FACE – Enter the Z dimension to be cut from the top of the part.

TOOL TOLERANCE – Enter a dimension between the edge of the part and the edge of the tool.

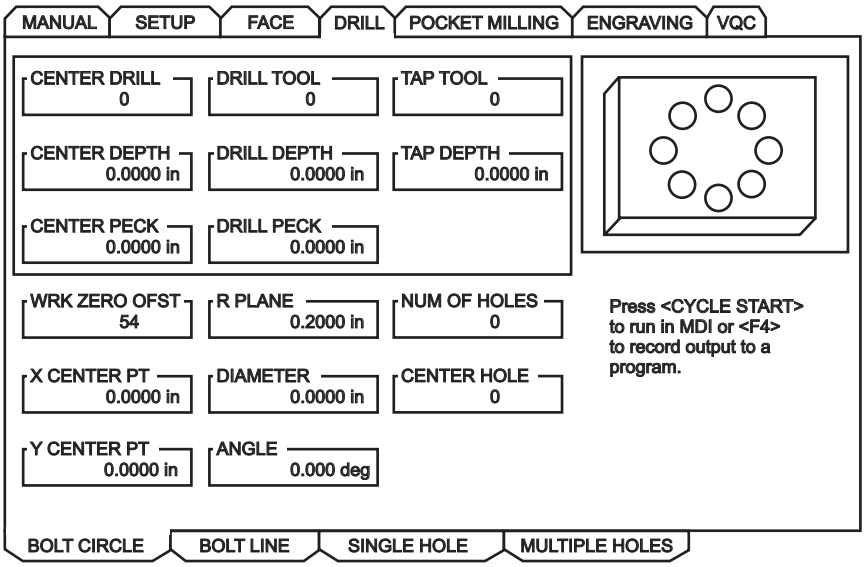
Advanced Users: In full CNC Mode, this is a G01 command.

Drill Mode

The Drill Mode is displayed by moving to the Drill tab and pressing Write/Enter. The Drill tab is used to set up the type of drilling to be done in the milling operation.

Bolt Circle Tab

The Bolt Circle tab is displayed in Drill Mode by selecting the tab and pressing Write/Enter. The Bolt Circle tab is used to set up drilling a number of holes in a circular pattern.



Bolt Circle Parameters:

CENTER DRILL – Enter the center drill tool number. Enter ‘0’ to skip center drilling cycle.

CENTER DEPTH – Enter how deep the holes are to be drilled. Calculated point value will be added if active.

CENTER PECK – Enter the distance for each peck move during center drilling.

DRILL TOOL – Enter drill tool number. Enter ‘0’ to skip drilling cycle.

DRILL DEPTH – Enter how deep the holes are to be drilled. Calculated point value will be added if active.

DRILL PECK – Enter the distance for each peck move during drilling.

TAP TOOL – Enter the tap tool number. Enter ‘0’ to skip tapping cycle.

TAP DEPTH – Enter how deep the holes are to be tapped.

WRK ZERO OFST – Enter a work zero offset number.

X CENTER PT – Enter the X axis dimension reference point from work zero offset.

Y CENTER PT – Enter the Y axis dimension reference point from work zero offset.

R PLANE – Enter the location of the retract point above the part.

DIAMETER – Enter the diameter of the bolt hole circle.

ANGLE – Enter the starting angle of holes from the three o’clock position.

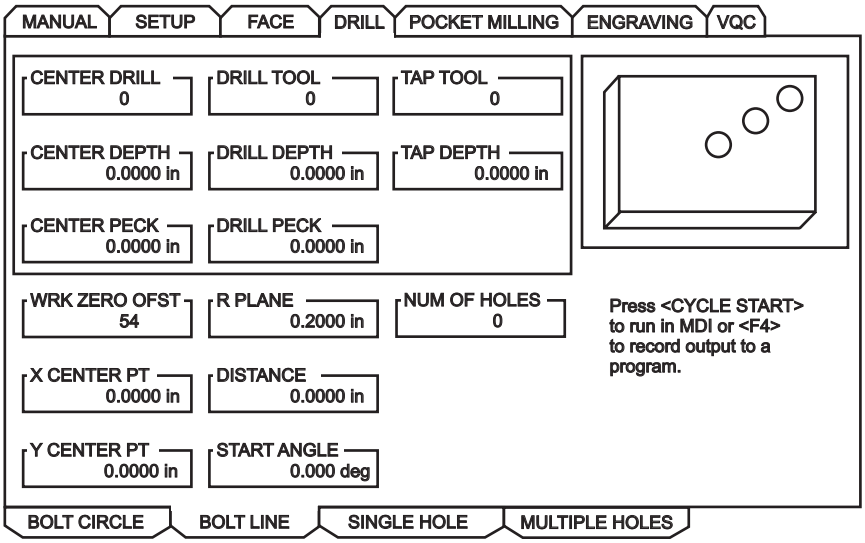
NUM OF HOLES – Enter the number of holes to be drilled in the bolt circle pattern.

CENTER HOLE – Do you want a hole in the center of the pattern? Enter ‘0’ for NO and ‘1’ for YES.

Advanced Users: In full CNC Mode, this is a G70 command.

Bolt Line Tab

The Bolt Line tab is displayed in Drill Mode by selecting the tab and pressing Write/Enter. The Bolt Line tab is used to set up drilling a number of holes in a line.



Bolt Line Parameters:

CENTER DRILL – Enter the center drill tool number. Enter '0' to skip center drilling cycle.

CENTER DEPTH – Enter how deep the holes are to be drilled. Calculated point value will be added if active.

CENTER PECK – Enter the distance for each peck move during center drilling.

DRILL TOOL – Enter drill tool number. Enter '0' to skip drilling cycle.

DRILL DEPTH – Enter how deep the holes are to be drilled. Calculated point value will be added if active.

DRILL PECK – Enter the distance for each peck move during drilling.

TAP TOOL – Enter the tap tool number. Enter '0' to skip tapping cycle.

TAP DEPTH – Enter how deep the holes are to be tapped.

WRK ZERO OFST – Enter a work zero offset number.

X CENTER PT – Enter the X axis dimension reference point from work zero offset.

Y CENTER PT – Enter the Y axis dimension reference point from work zero offset.

R PLANE – Enter the location of the retract point above the part.

DISTANCE – Enter the distance between the holes. Must be a positive value.

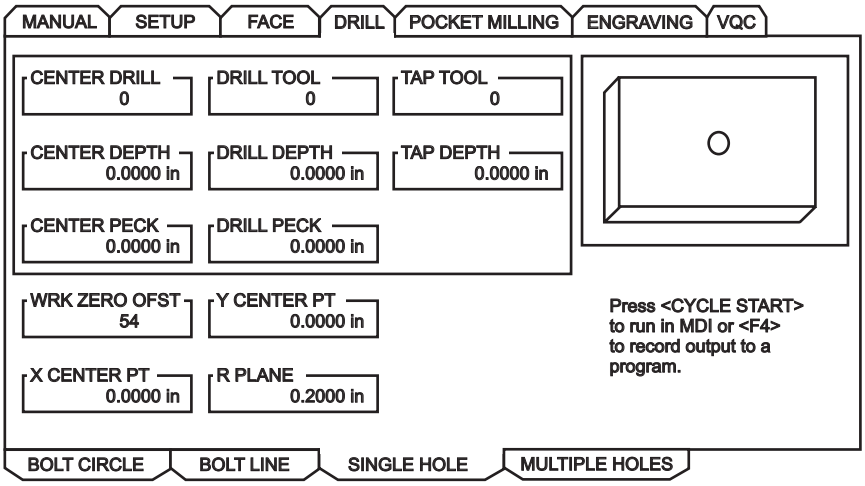
START ANGLE – Enter the starting angle of holes from the three o'clock position.

NUM OF HOLES – Enter the number of holes to be drilled in a linear path.

Advanced Users: In full CNC Mode, this is a G72 command.

Single Hole Tab

The Single Hole tab is displayed in Drill Mode by selecting the tab and pressing Write/Enter. The Single Hole tab is used to set up drilling a single hole.



Single Hole Parameters:

CENTER DRILL – Enter the center drill tool number. Enter '0' to skip center drilling cycle.

CENTER DEPTH – Enter how deep the hole is to be drilled. Calculated point value will be added if active.

CENTER PECK – Enter the distance for each peck move during center drilling.

DRILL TOOL – Enter drill tool number. Enter '0' to skip drilling cycle.

DRILL DEPTH – Enter how deep the hole is to be drilled. Calculated point value will be added if active.

DRILL PECK – Enter the distance for each peck move during drilling.

TAP TOOL – Enter the tap tool number. Enter '0' to skip tapping cycle.

TAP DEPTH – Enter how deep the hole is to be tapped.

WRK ZERO OFST – Enter a work zero offset number.

X CENTER PT – Enter the X axis dimension reference point from work zero offset.

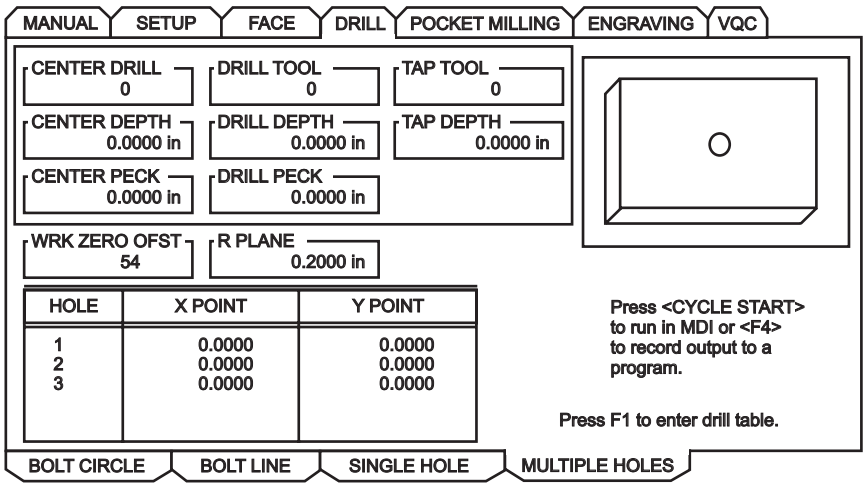
Y CENTER PT – Enter the Y axis dimension reference point from work zero offset.

R PLANE – Enter the location of the retract point above the part.

Advanced Users: In full CNC Mode, a G83 command is used for the drills, and a G84 command is used for the tap, to set the dimensions of a single hole.

Multiple Holes Tab

The Multiple Holes tab is used to set up various locations where identical holes are to be drilled.



Multiple Holes Parameters

CENTER DRILL – Enter the center drill tool number. Enter '0' to skip center drilling cycle.

CENTER DEPTH – Enter how deep the hole is to be drilled. Calculated point value will be added if active.

CENTER PECK – Enter the distance for each peck move during center drilling.

DRILL TOOL – Enter drill tool number. Enter '0' to skip drilling cycle.

DRILL DEPTH – Enter how deep the hole is to be drilled. Calculated point value will be added if active.

DRILL PECK – Enter the distance for each peck move during drilling.

TAP TOOL – Enter the tap tool number. Enter '0' to skip tapping cycle.

TAP DEPTH – Enter how deep the hole is to be tapped.

WRK ZERO OFST – Enter a work zero offset number.

R PLANE – Enter the location of the retract point above the part.

Drill Table – Press F1 to enter the drill table. Define hole locations by X and Y reference dimension points from the work zero offset. Alternately, jog to the desired hole position and press TOOL OFFSET MEASURE to record the current X and Y positions in the table. Press INSERT to add a new hole to the table.

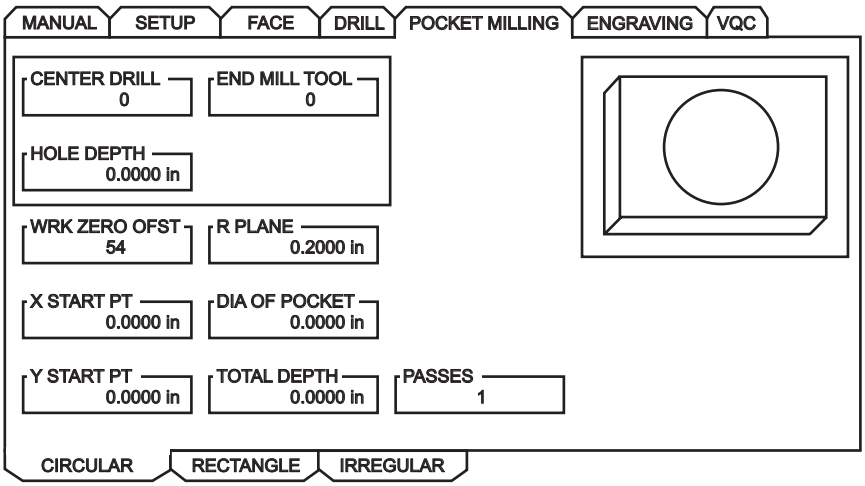
Advanced Users: In full CNC Mode, a G83 command is used for the drills, and a G84 command is used for the tap, to set the dimensions of the holes. A G00 command is used for rapid movements to each hole position defined in the drill table.

Pocket Milling Mode

The Pocket Milling Mode is displayed by moving to the Pocket Milling tab and pressing Write/Enter. The Pocket Milling tab is used to mill a cavity in a piece of material.

Circular Tab

The Circular tab is displayed in Pocket Milling Mode by selecting the tab and pressing Write/Enter. The Circular tab is used to mill a circular cavity in a piece of material.



Circular Parameters:

CENTER DRILL – Enter the center drill (or drill) tool number. Enter '0' to skip center drilling cycle.

HOLE DEPTH – Enter how deep the hole is to be drilled in the center of the pocket.

END MILL TOOL – Enter End Mill tool number. Enter '0' to skip milling cycle.

WRK ZERO OFST – Enter a work zero offset number.

X START PT – Enter the X axis dimension reference point from work zero offset.

Y START PT – Enter the Y axis dimension reference point from work zero offset.

R PLANE – Enter the location of the retract point above the part.

DIA OF POCKET – Enter the diameter of the pocket to be cut.

TOTAL DEPTH – Enter the total depth of the pocket.

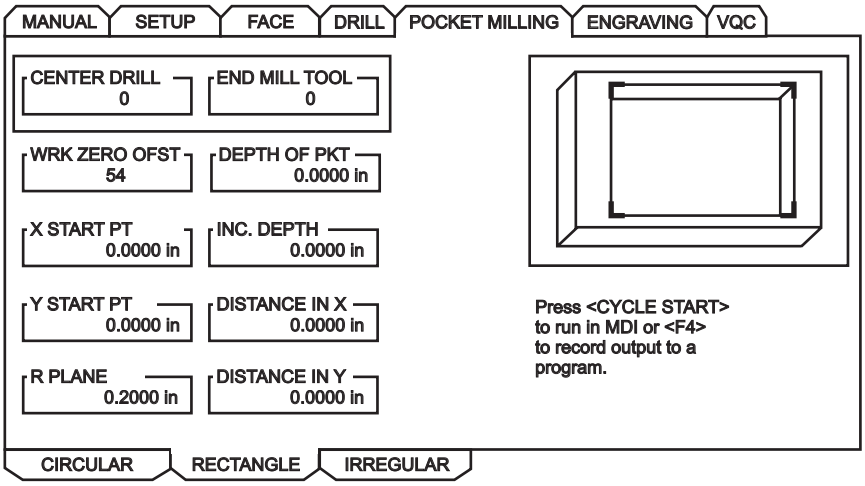
PASSES – Enter the number of passes to cut the pocket.

Advanced Users: In full CNC Mode, this is a G12 command for CW milling, or a G13 command for CCW milling.

NOTE: The initial end mill move assumes there is either a hole for the end mill, or the proper end cutting end mill to plunge straight down in the Z direction.

Rectangle Tab

The Rectangle tab is displayed in Pocket Milling Mode by selecting the tab and pressing Write/Enter. The Rectangle tab is used to mill a rectangular cavity in a piece of material.



Rectangle Parameters:

CENTER DRILL – Enter the center drill (or drill) tool number. Enter '0' to skip center drilling cycle.

END MILL TOOL – Enter End Mill tool number. Enter '0' to skip milling cycle.

WRK ZERO OFST – Enter a work zero offset number.

X START PT – Enter the dimension of the edge of the pocket for the X axis from the work zero offset.

Y START PT – Enter the dimension of the edge of the pocket for the Y axis from the work zero offset.

R PLANE – Enter the location of the retract point above the part.

DEPTH OF PKT – Enter the value for the total depth of the pocket.

INC. DEPTH – Enter the value for the incremental cut made while cutting the pocket.

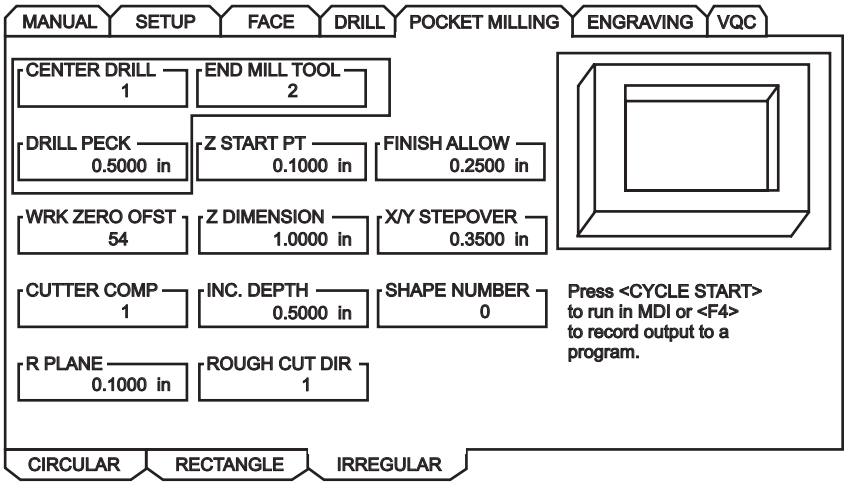
DISTANCE IN X – Enter the size of the pocket to be cut in the X direction.

DISTANCE IN Y – Enter the size of the pocket to be cut in the Y direction.

Advanced Users: In full CNC Mode, this is a G01 command.

IrregularTab

The Irregular tab is the main screen used to execute the program for the selected shape. Information on this screen includes which tool is used, cutter compensation, how deep the pocket will be, the amount of finish allowance and the shape. This tab is only available if the machine has a control pendant with a 15" screen and software version 15.02A or later.



Irregular Parameters:

CENTER DRILL – Enter drilling tool number here.

END MILL TOOL – Enter end mill tool number here.

DRILL PECK – Enter distance for drill to peck if desired.

WRK ZERO OFST – Enter the work zero offset number.

CUTTER COMP – Enter a 1 for cutter compensation left or 2 for cutter compensation right.

R PLANE – Enter the location of the retract point above the part.

Z START PT – Enter the absolute Z-axis position of the top of the part.

Z DIMENSION – Enter the distance from the Z-axis start position to the bottom of the pocket.

INC. DEPTH – Enter the incremental Z-axis step distance for the rough pocketing cycle.

ROUGH CUT DIR – Enter 1 for X-axis roughing or 2 for Y-axis roughing.

FINISH ALLOW – Enter a positive value for the finishing allowance.

X/Y STEPOVER – Enter a value for the stepover cut in X or Y axis.

SHAPE NUMBER – Enter shape program number, or press Write/Enter without a number to bring up the Shape Selector window.

Advanced Users: In full CNC Mode, this is a G01 command.

Basic Shape Creation (Example)

1. Start the machine with IPS active.

2. Clear any alarms, then press Power Up/Reset to zero the machine.

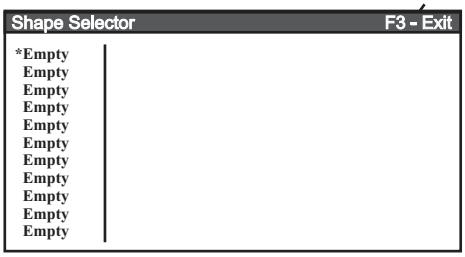
3. Select the Setup tab, then the Work tab to set up the work offsets and material.

4. Select the Tool tab (under the Setup tab) to set up the tools to be used.

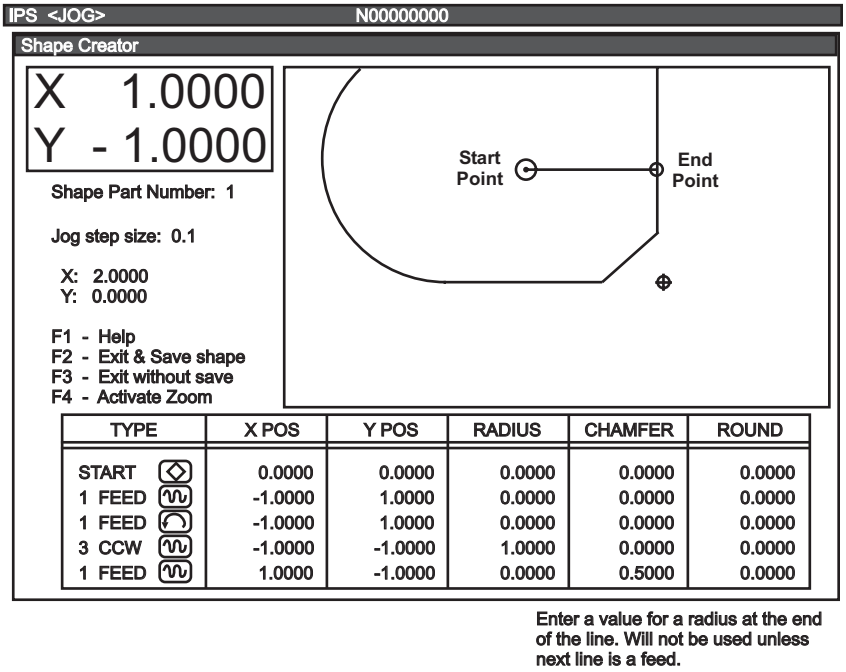
5. Press Cancel a few times to get out of the Setup tab. Select the Pocket Milling tab, then the Irregular tab.

6. Enter the tool number for the Center Drill, set Drill Peck to 0.5, enter the tool number for the End Mill Tool, set Z Start PT to 0.1, Z Dimension to 1.0, INC Depth to 0.5, Finish Allow to 0.25 and X/Y Stepover to 0.35.

7. Select Shape Number data box and press Write/Enter or press F1 when in the Irregular tab. A Shape Selector popup window is displayed. The Shape Selector popup is used to select a shape, alter an existing shape, choose a storage location for a new shape or delete a shape.



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



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

F2 – Saves shape on the screen, exits Shape Creator screen and transfers control to Irregular tab.

F3 – Exits Shape Creator screen and transfers control to Irregular tab screen. Does not save shape's data.

F4 – Activates and deactivates 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.

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

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

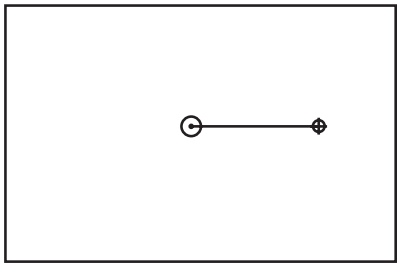
(.0001), (.001), (.01), (.1) – Changes the jog step size while drawing in the graphic window.

To Build the Shape Shown:

a. Leave the Start PT at X0 Y0. Use the arrow keys to go to the beginning of the first line in the table (Start Point). Press 1 to activate a Feed move.

b. Jog X to 1.0 by turning the handwheel clockwise, press Write/Enter.

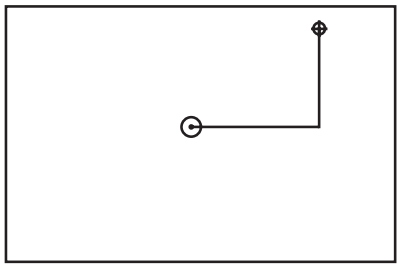
NOTE: Each handwheel click either increments or decrements the position by 0.1".



NOTE: Any contour must be closed. Last move ends at the point in the cross-hairs as shown above. The start point must be inside the contour for the G150 cycle to work.

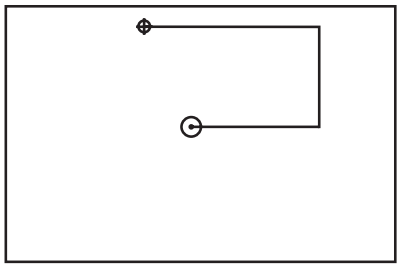
Use the arrow keys to go to the beginning of the next line in the table and press 1 to activate a Feed move.

c. Press Write/Enter until Y POS is selected. Jog Y to 1.0 and press Write/Enter.



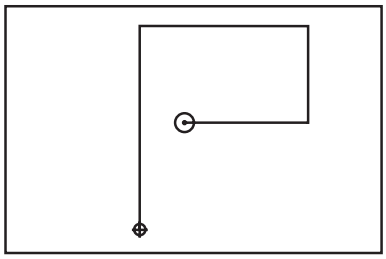
Go to the beginning of the next line. Press 1 to activate a Feed move.

d. Jog X to -1.0 and press Write/Enter.

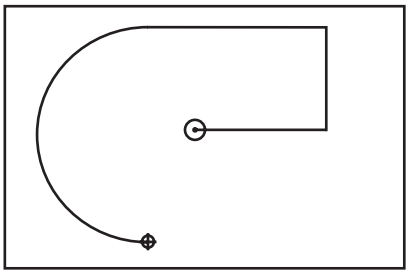


Go to the beginning of the next line. Press 3 to activate a CCW move.

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

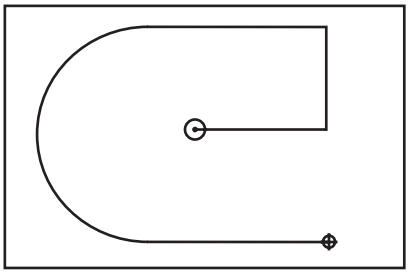


Select the Radius column, enter 1.0 and press Write/Enter.



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

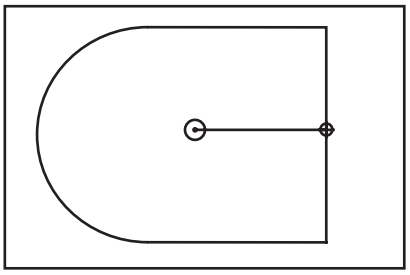
f. Jog X to 1.0, press Write/Enter.



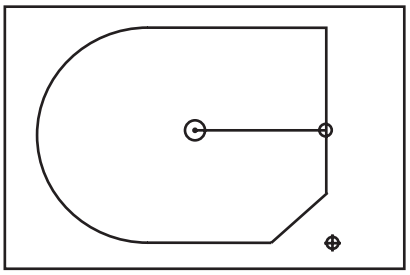
Go to the beginning of the next line. Press 1 to activate a Feed move.

g. Jog Y to 0.0 and press Write/Enter.

NOTE: The contour must be returned to the point in the cross-hairs (as shown below) for the G150 cycle to work.



h. Select Chamfer in the previous row and enter 0.5.



i. Press F2 to Save and exit the Shape Creator.

NOTE: The Start Point and the End Point are not the same position.

j. Press Cycle Start to cut the pocket or, to view before cutting, press Cycle Start followed quickly by Feed Hold. Press MDI, then SETNG/Graph. Press Cycle Start to cut the pocket. When the program has run, access MDI to see the program that was created, then press SETNG/Graph to run the program in the graphics screen.

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.

Recalling Shapes

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

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

Shape Creator Help

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

Shape Creator Help	F1 - Exit
Exit and Save Shape	(F2)
Exit without Saving Shape	(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 Y Axis Data Box	(Y 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 Shape - Exits Shape Creator screen and saves shape you were working on into memory.

Exit without Saving Shape - Exits Shape Creator screen and does not save shape 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 Y Axis Data Box - Highlights Y axis data box and changes drawing cursor to only move in Y direction.

Move Up To Next Data Box - Moves up to next data box above its current location. This 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. This 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. This 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. This will wrap if already at the far left.

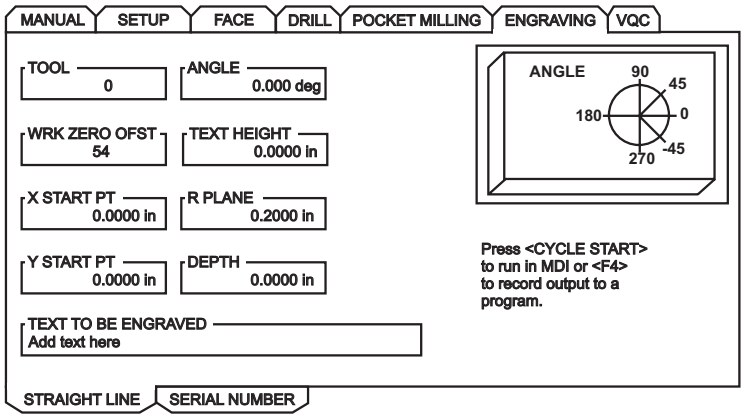
Advanced Users: In full CNC Mode, this is a G150 command.

Engraving Mode

The Engraving Mode is displayed by moving to the Engraving tab and pressing Write/Enter. The Engraving tab is used to set up the engraving of text on a piece of material.

Straight Line Tab

The Straight Line tab is displayed in Engraving Mode by selecting the tab and pressing Write/Enter. The Straight Line tab is used to set up the straight line engraving of text on a piece of material.



Straight Line Parameters:

TOOL – Enter engraving tool number.

ANGLE – Enter the angle at which the text is to be engraved along that linear path.

WRK ZERO OFST – Enter a work zero offset number.

X START PT – Enter the X axis dimension reference point from work zero offset.

Y START PT – Enter the Y axis dimension reference point from work zero offset.

TEXT HEIGHT – Enter the text height value.

R PLANE – Enter the location of the retract point above the part.

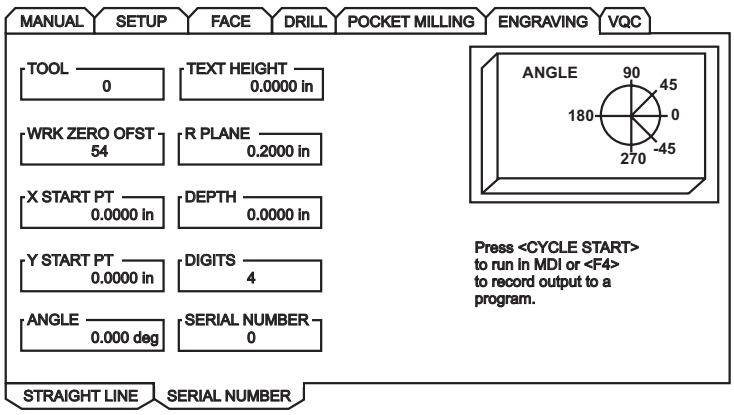
DEPTH – Enter the depth at which the text is to be engraved.

TEXT TO BE ENGRAVED – Enter text to be engraved.

Advanced Users: In full CNC Mode, this is a G47 command.

Serial Number Tab

The Serial Number tab is displayed in Engraving Mode by selecting the tab and pressing Write/Enter. The Serial Number tab is used to set up the engraving of a serial number on a piece of material.



Serial Number Parameters:

TOOL – Enter engraving tool number.

WRK ZERO OFST – Enter the work zero offset number.

X START PT – Enter the X axis dimension reference point from work zero offset.

Y START PT – Enter the Y axis dimension reference point from work zero offset.

ANGLE – Enter the angle at which the text is to be engraved along that linear path.

TEXT HEIGHT – Enter the text height value.

R PLANE – Enter the location of the retract point above the part.

DEPTH – Enter the depth at which the text is to be engraved.

DIGITS – Enter the number of digits (1 to 10) to be engraved. Not the serial number.

SERIAL NUMBER – Enter the starting serial number.

Advanced Users: In full CNC Mode, this is a G47 command.

VQC Mode

VQC Mode is described in the Mill Operator's Manual (96-8700).

DXF File Importer

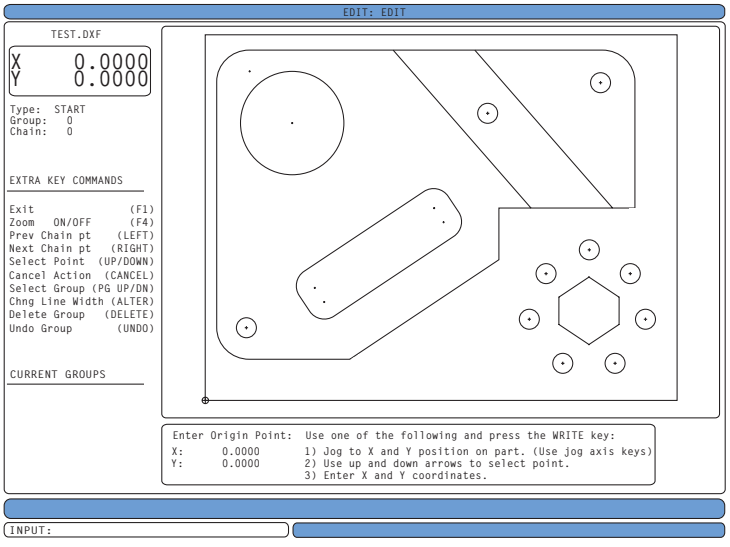
This feature can quickly build a CNC G-code program from a DXF file, a drawing file format exportable from many desktop CAD applications. Compatible DXF files are made up of arcs, lines, circles, vertices, and/or points. Refer to your CAD application's documentation for details on how to export a DXF 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.

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 identifies repetitive tasks and automatically executes them; for example, finding all the holes with the same diameter. Long contours are also automatically joined.

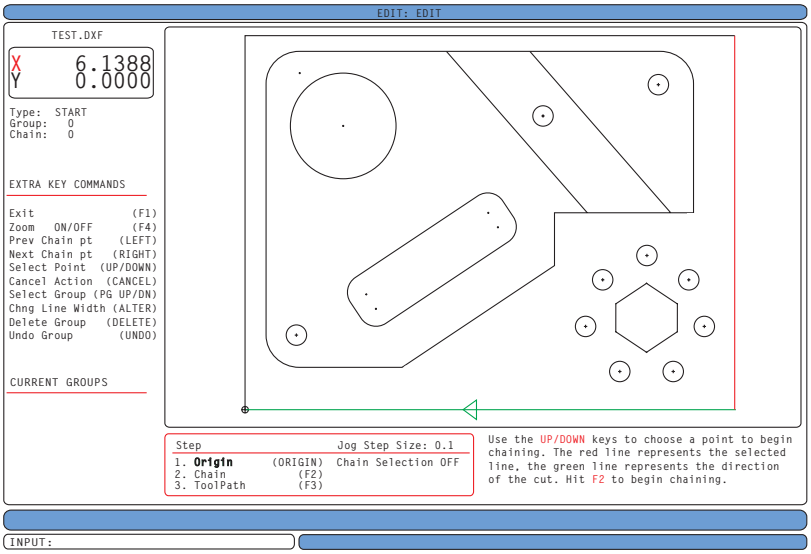
DXF Importer creates programs using simple input given in the following steps:

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 Part Origin

Use one of three methods:

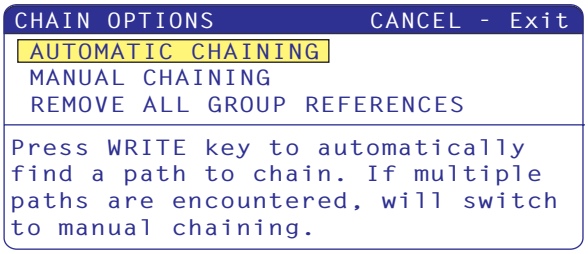
- Point Selection - Use the up and down arrow keys to select a point.
- Jogging - Jog to the X and Y position on a part (use jog axis keys).
- Enter Coordinates - Type in X coordinate and press WRITE/ENTER, then type in Y coordinate and press WRITE/ENTER.

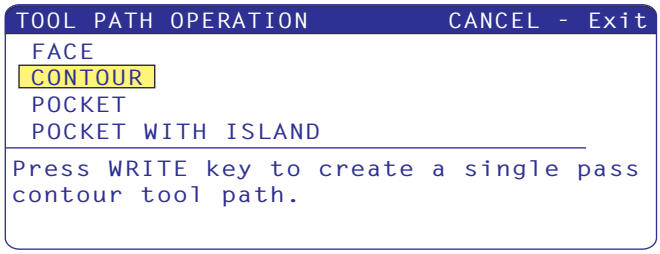


Use the jog handle or cursor arrow keys to highlight a point. Press WRITE/ENTER to accept the highlighted point as the part origin. In the Outline box, the word “Origin” turns green, indicating that this step is complete.

Chain/Group

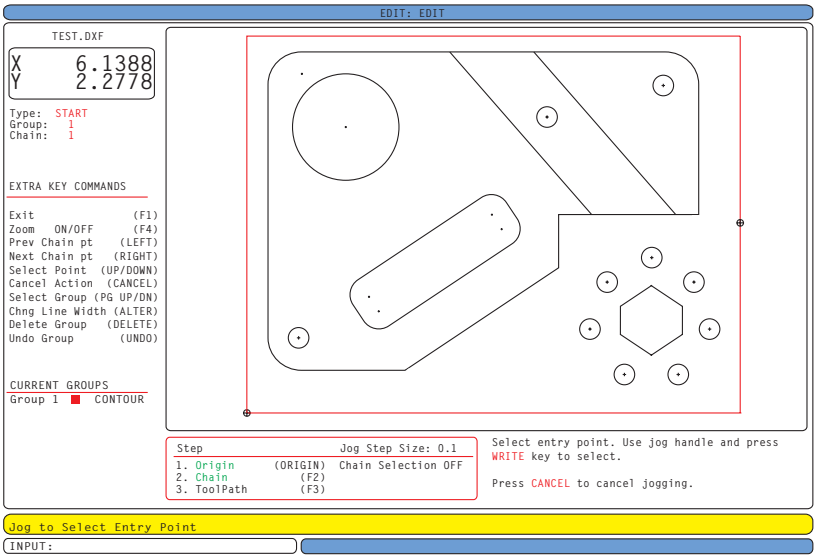
This step defines the geometry of the shape(s). The auto chaining function will find most part geometry. If the geometry branches off, a message will prompt you to select a branch and automatic chaining will continue. Similar holes are grouped together for drilling and/or tapping operations. Use the up and down cursor arrow keys, jog handle, or manually enter coordinates to select a starting point, then press F2 to access chain/group options.





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 color-coded group to the register under “Current groups” on the left hand side of the window.

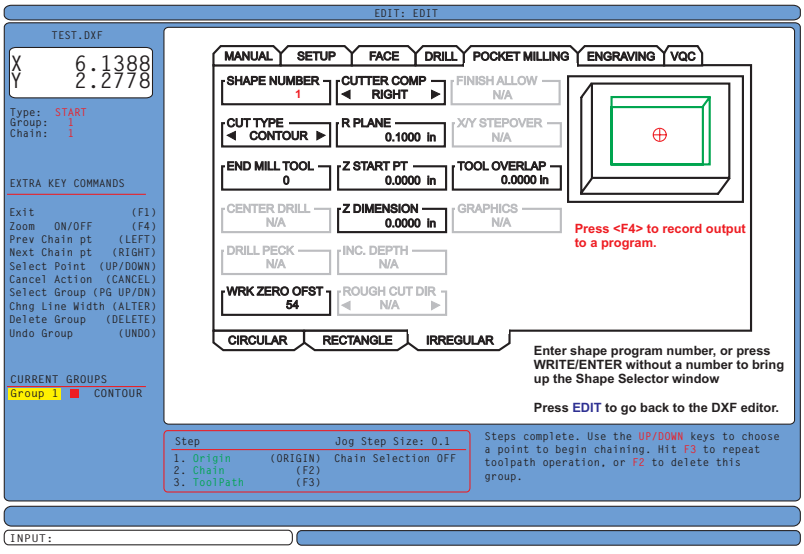
The tool path can also be manually generated. After selecting the starting point for the tool path, select “Manual Chaining” from the chain options menu. DXF Importer will begin to follow the specified line, section by section. To accept a section of the geometry, press Write/Enter. Where branches occur,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



To define a drilled hole, select the hole’s centerpoint (shown as a small cross figure) rather than the hole’s shape. DXF Importer will identify all other holes with the same diameter as the one selected and add them to the group. In the example above, defining the hole in the lower left corner of the large feature would also define the other two holes in that feature, as well as the seven holes surrounding the hexagonal feature.

Select Tool Path

This step applies a tool-path operation to a particular chained group. Select the group using Page Up/Page Down and Press F3 to choose a tool path. If the path selected represents a contour or pocket, ‘Currently Bi-secting’ is displayed at the bottom of the screen. Use the jog handle to bisect an edge of the part feature; this will be used as an entry point for the tool. Press WRITE/ENTER.



Once a tool path is selected, the IPS (Intuitive Programming System) template for that shape will display. Define Z-axis values in this template.

Most IPS template fields are filled with reasonable defaults based on the tools and materials defined in setup. Adjustments can be made as necessary

Press F4 to save the toolpath once the template is completed. Refer to the “IPS Recorder” section for details on saving.

Press Edit to return to DXF Editor.

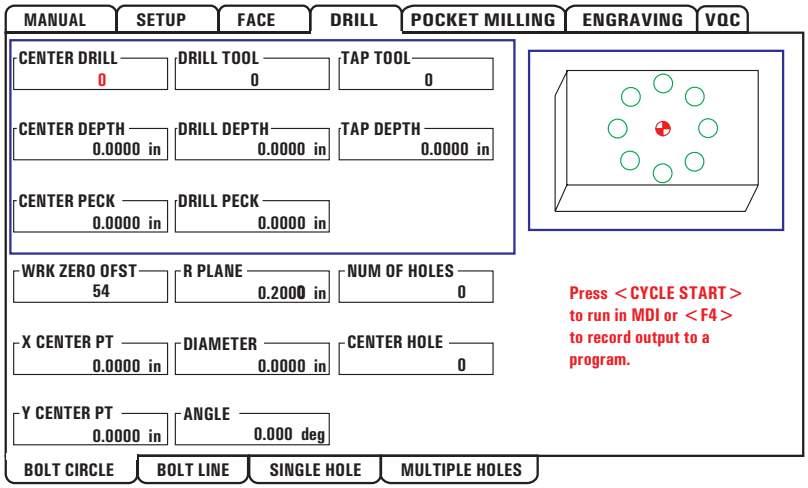
IPS Recorder

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

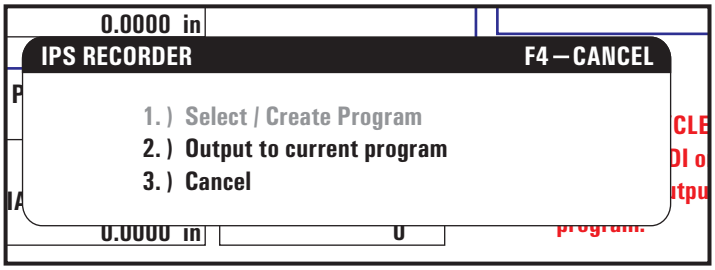
Operation

1. To access IPS, press MDI/DNC, then PROGRM/CONVRS.

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



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.



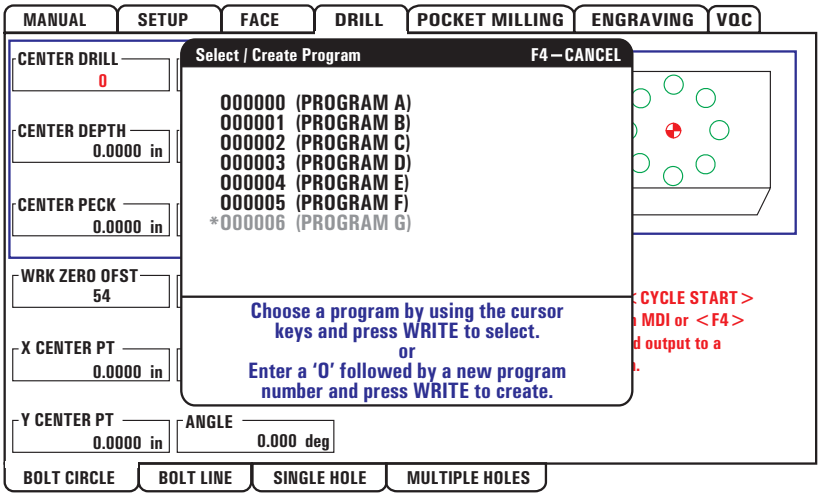
IPS Recorder Menu

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.