WinMax Lathe Options

Hurco Companies, Inc.
One Technology Way
P.O. Box 68180
Indianapolis, IN 46268-0180

For Hurco subsidiary contact information, go to Hurco's Web site:
www.hurco.com

TABLE OF CONTENTS

DOCUMENTATION CONVENTIONS

This documentation uses several conventions to explain the safety features and emphasize key concepts. These conventions are described in this section.

Sample Screens

Sample screens in this documentation were taken from a WinMax single-screen control. All screens are subject to change. The screens on your system may vary slightly.

Softkeys

Softkeys are located on the side of the screen. You can set the softkeys to appear on either the right or left side of the screen. Refer to the *Getting Started with WinMax* for information about making this selection. Softkeys may change upon field entries or other softkey selection. References to softkeys in the documentation appear with the softkey's corresponding F-key. For example, the Part Setup softkey from the Input screen is referenced as the PART SETUP *F1* softkey.

Screen Areas

The screens are divided into the following areas, in addition to the row of softkeys:

Data Entry

The data entry area is located on the opposite side of the screen from the softkeys. Available softkeys may change even when the text and data entry area does not.

Fields in the data entry area display or receive information. Refer to *Using the Touchscreen* for information on entering information in fields.

Prompts and Error/Status Area

The bottom portion of the screen is reserved for prompts, program status and error messages.

Prompts provide help on data entry selections based on the field with the blinking cursor.

Errors and status messages occur anytime the status or error occurs. They are not based on the field with the blinking cursor. These messages provide machine information to the operator.

Error messages may also stop and/or prevent machine operation until the cause of the error is corrected.

Status Bar

The status bar contains

- The name of the open, selected program.
- A calculator icon—select the icon to display a working, on-screen calculator.
- Units of measure (Inch or Millimeters)—select the units of measure in the status bar to toggle between Inch and Metric.
- Programming mode (R for Radius; D for Diameter)—select the programming mode in the status bar to toggle between Radius and Diameter.
- A yellow icon—indicates the feed hold is on when visible.
- A red icon—indicates the Emergency Stop button has been pressed when visible.
- A keyboard icon—select the icon to display a working on-screen keyboard.
- The current time.

When viewed on a single-screen console, all icons appear in the same status bar; when viewed on a dual-screen console, the program name and calculator icon appear on the left screen status bar, and the unit of measure, keyboard icon and time appear on the right screen status bar.

Console Buttons and Keys

References to console buttons and keys appear in bold text throughout the documentation. For example, the Start Cycle button appears as the **Start Cycle** button and the Manual key appears as the **Manual** console key in text.

Refer to the *Getting Started with WinMax* manual for information about console buttons and keys, in addition to other information about using softkeys and the pop-up text entry window.

Using the Touchscreen

The console has a touchscreen for entering programming data. To make a selection, tap the screen on a softkey, field, or drop-down list using the stylus attached to the side of the console or another suitable pointing device.

Icons

This manual may contain the following icons:

Caution/Warning

⚠ The operator may be injured and/or the machine severely damaged if the described procedure is not followed.

Important

Ensures proper operation of the machine and control.

Troubleshooting

Steps that can be taken to solve potential problems.

Hints and Tricks

Useful suggestions that show creative uses of the WinMax features.

Where can we go from here?

Lists several possible options the operator can take.

PROGRAMMING AND OPERATION INFORMATION

Hurco provides documentation for using WinMax software on a control or desktop in two formats: on-screen Help and PDF. The information contained in both formats is identical.

On-screen Help contains information about the current screen. If Help is not available for a screen, a Welcome screen appears with access to the Table of Contents, Index, or Search functions.

- To view the on-screen Help directly on a Hurco control, select the Help console key.
- To view the on-screen Help on the desktop software, select the Help icon in the menu bar.

PDF files are available on the hard drive. These files can be copied from the hard drive to a USB memory device and transferred to a PC for viewing and printing.

Using the On-screen Help

On-screen Help provides information about using WinMax. The Help is context-sensitive to the screen level. Press the console Help button to display the Help topic for the current screen. The following list describes Help functions:

- Buttons in the upper left-hand corner of the Help screen are used to move through Help topics and print screens.
  - Use the **Hide** button to hide the navigation pane.
  - Use the **Back** button to return to the previous Help screen.
  - Use the **Print** button to print the current displayed Help topic, if a printer is attached and configured. See *Printing the Programming Manuals, on page - xiii* for more information about printing.
- Use the arrow buttons to move between pages within a Help topic and to move through topics.
- Use the **Contents** tab for a list of information sorted by subject:
  1. Select the "+" to expand the topic and view sub-topics.
  2. Select the topic to display it.
- Use the **Index** tab to show the Help index:
  1. Quickly scroll to an index topic by typing the topic in the box at the top of the index.
  2. Select a topic and the Display button to view the topic.

- Use the **Search** tab to search the Help for a word or phrase:
  1. Type the search word(s) into the text box at the top of the pane.
  2. Select the List Topics button. A list of topics that contain the search word(s) is displayed.
  3. Select a topic and the Display button to view that topic.
- Use the **Favorites** tab to save Help topics for quick access:
  1. Select the Add button at the bottom of the pane to add the current topic.
  2. Select a topic from the Favorites list, and select the Display button to view it.
     - Select a topic from the Favorites list, and select the Remove button to remove it from the list.

Printing the Programming Manuals

The WinMax On-screen Help is also provided in PDF format for easy printing. The information contained in the PDF files is identical to the on-screen Help. The PDF files may be copied to a floppy disk or USB memory device to be transferred to a PC for printing. Here are the steps to access the PDF files:

1. From the Input screen, select the PROJECT MANAGER *F8* softkey.
2. Select the FILE MANAGER *F7* softkey.
3. In the left-hand pane, navigate through the folders:
   - For WinMax Lathe on a machine, the path is D:\Hurco\Hurco Lathe\hlp.
   - For WinMax Desktop on a PC, the path is C:\Program Files\Hurco\Hurco Lathe\hlp.

   The PDF files will appear in the right-hand pane.

   ⇨ The SHOW ALL FILE TYPES field in User Interface Settings must be set to YES (default is NO) in order to see the PDF files in the directory. Access the SHOW ALL FILE TYPES field in Auxiliary Mode, Utilities/ User Preferences/ User Interface Settings.
4. Highlight the PDF file(s) in the right-hand pane, and select the COPY *F2* softkey.
5. Ensure that your media is loaded (either a floppy disk in the disk drive or a USB memory device in the USB port), and navigate to the proper location in the left-hand pane of the DISK OPERATIONS screen (either the floppy drive A: or the USB port E:). Highlight the desired location.
6. Place the cursor in the right-hand pane and select the PASTE *F3* softkey to paste the PDF file(s) to the desired location.

You may now remove your media and load the PDF file(s) onto a PC for printing.

OPTIONS

The WinMax Lathe Max control contains software used to process data and display screens in much the same manner that personal computers use software programs. As with other software systems, WinMax Lathe has additional software options that can be purchased for the system. Contact Hurco or your Hurco distributor for details about purchasing software options.

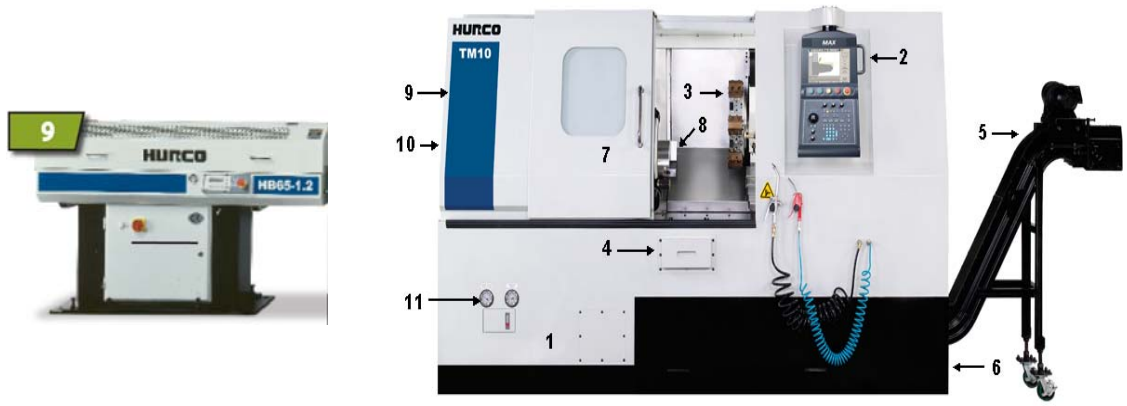
This section contains information for operating optional turning center equipment and software.

Overview

Before using the machine, you should become familiar with its components. Because of European Committee (CE) requirements, Hurco machines sold in Europe may differ from those sold elsewhere.

Hardware Options

The figure below identifies some options of a machine. The location of some components may differ on other models.



1	Machine Frame or Base	**7**	Enclosure Door
2	Console	**8**	Spindle
3	Turret	**9**	Power cabinet (rear)
4	Parts Catcher option	**10**	Communications panel (side)
5	Conveyor option	**11**	Tailstock and chuck gauge options
6	Coolant Drip Tray	**12**	Bar Feeder option

Figure 1–1. Bar Feeder next to a TM10 Turning Center with the WinMax Lathe Max Console and Options

AUTO DOOR

The optional, pneumatic Auto Door allows you to automatically open and shut the enclosure door at the beginning and end of a cycle. This option provides a clear signal that a cycle is complete by opening the door and allowing the operator to remove a part. In addition, the Auto Door allows the ability to control the door operation for automated cycles.

Auto Door Operation

The Auto Door can be operated either manually with softkeys on the Manual screen or automatically using M Codes. To program an M Code in a conversational programming data block, please refer to *Machine Function—M Code, on page 2 - 198* in *WinMax Lathe Conversational Part Programming*. For *WinMax Lathe NC Programming* information please refer to *Basic NC M Codes, on page 3 - 1* or *ISNC M Codes, on page 5 - 1*.

⚠ Machines configured with the optional Auto Door are equipped with a second pushbutton used in conjunction with the Start Cycle button to initiate cycles. This configuration requires the operator to simultaneously press the **Start Cycle** pushbutton and this **Enable** pushbutton, located on the right side of the console below the handle, to initiate any cycle.

The Auto Door can be opened by hand if air is disconnected even though the door is pneumatically powered.

⚠ An Auto Door that is opened by hand cannot be closed unless the control recognizes the door has been opened using an Open Auto Door selection. Therefore, you must use the Open Auto Door *F1* softkey then simultaneously press the **Start Cycle** and **Enable** pushbuttons before proceeding.

If an object obstructs the path of the Auto Door when closing, upon contact the door will automatically retract and return to the Open state.

Manual Operation

To operate the Auto Door in Manual mode,

1. Press the **Manual** console key to access the Manual screen.
2. Select the ACCESSORY OPERATIONS *F4* softkey.
3. Select the AUTO DOOR *F5* softkey. The softkey menu changes to these softkey selections:
   - **OPEN AUTO DOOR** *F1*—causes the **Start Cycle** button to flash. Simultaneously press the **Start Cycle** and **Enable** pushbuttons to open the Auto Door.
   - **CLOSE AUTO DOOR** *F2*—causes the **Start Cycle** button to flash. Simultaneously press and hold the **Start Cycle** and **Enable** pushbuttons until the door reaches the safety switch located on the right hand side of the enclosure opening.

   ⚠ If the pushbuttons are released prior to reaching the safety switch, the door will return to the Open state immediately upon releasing the pushbuttons. A message appears on the screen.

Automatic Operation

To operate the Auto Door automatically, program a Machine Function data block within a conversational part program. From the New Block screen or Program Review screen,

1. Select the Insert Block Before *F7* softkey.
2. Select the Miscellaneous *F4* softkey.
3. Select the M Code *F3* softkey.
   - Select M85 to Open the Auto Door and enter the number of seconds to dwell in the Dwell field before moving to the next block.
   - Select M86 to Close the Auto Door and enter the number of seconds to dwell in the Dwell field before moving to the next block.

   ⇨ If using NC programming, the M85 and M86 codes apply.

BAR FEEDER

The optional bar feeder places stock into the spindle, through the hole in the left side of the machine. As the stock is cut, the feeder replaces it with a new piece as necessary. Please refer to the bar feeder's Operator's Manual for instructions about maintenance, turning it on, and making selections on the control panel on the front of the bar feeder.

The bar feeder can be operated either with a Bar Feed data block or with M Codes. To program an M Code in a conversational programming data block, please refer to WinMax Lathe Conversational Part Programming, *Machine Function—M Code, on page 2 - 198*, WinMax Lathe NC Programming, *Basic NC M Codes, on page 3 - 1*. or *ISNC M Codes, on page 5 - 1*.

Bar Feed Block

The Bar Feed Block automates advancing the bar between parts. This data block supports both bar pushing and bar pulling cycles.

- Bar pushing requires the optional bar feeder or a servo driven bar feeder. When a piece of stock is depleted, the optional bar feeder replaces it with a new piece as necessary.
  - Select the Tool as Bump Stop or Tool Guides Stock Strategy if the optional Bar Feeder is used. The optional bar feeder places stock into the spindle, through the hole in the left side of the machine.
  - Select the Tool not Used Strategy if a servo driven bar feeder is used.
- Bar pulling requires a special pulling tool loaded in the turret. You can program a tool in Tool Setup to use with a Bar Feed Block to pull the stock away from the chuck.
  - Select the Fingers Pull Stock Strategy if a finger style pulling tool is programmed in Tool Setup and the optional Bar Feeder is not being used.
  - Select the Collet Pulls Stock Strategy if a collet style pulling tool is programmed in Tool Setup and the optional Bar Feeder is not being used.

To access the Bar Feed Block screen from the Program Review screen or from the New Block screen, select the MISCELLANEOUS ➔ F4 softkey followed by the BAR FEED *F4* softkey.

These fields are available for programming a Bar Feed block:

- **STRATEGY**—identifies the Bar Feed strategy. The fields change depending on the strategy. Select from these choices:
  - **Tool not Used**—for use with servo-driven bar feeders where a bump stop or stock guiding is not needed. If Tool not Used is selected, the servo-driven bar feeder moves the stock forward for the next cut.
  - **Tool as Bump Stop**—for use when the automatic bar feed should be stopped when it touches the tool. This selection is for programming a Bar Feed block with the optional Bar Feeder. If Tool as Bump Stop is selected, after the stock is cut, the feeder moves the stock forward for the next cut.
  - **Fingers Pull Stock**—for use when a finger style bar puller is in the turret and programmed in Tool Setup. The tool fingers grip the workpiece and pull the material out of the chuck. This selection is for programming a Bar Feed block without the optional Bar Feeder.
  - **Tool Guides Stock**—causes the turret to meet the bar near the chuck and then feed away with the bar. This is the default Strategy. If Tool Guides Stock is selected, the turret moves into position, meeting the bar near the chuck. The turret then feeds away with the bar, guiding the stock as pressure from the feeder moves the stock forward.
  - **Collet Pulls Stock**—for use when a collet style bar puller is in the turret and programmed in Tool Setup. The collet tool grips the workpiece and pulls the material out of the chuck. This selection is for programming a Bar Feed block without the optional Bar Feeder.
- **TOOL**—identifies the tool number that will be used in the part program. This field does not appear when TOOL NOT USED is selected in the STRATEGY field.

  Use a square turning tool programmed in Tool Setup to represent a pulling tool.

- **TOOL OFFSETS**—identifies the tool offset. This field does not appear when TOOL NOT USED is selected in the STRATEGY field. Refer to the “Tool Setup” section in *Getting Started with WinMax Lathe*.
- **IGNORE END OF BAR**—provides selections to stop operations when the bar feeder reaches the end of the bar.
- **USE PART CATCHER**—activates the optional Parts Catcher when YES is selected and catches the unused stock remnant. Otherwise, the stock remnant falls into the chip tray or conveyor. The control holds off all motion until the Bar Feed Complete signal is received. Set up the automatic Bar Feeder according to the manufacturer’s instructions. This field is not available with the Fingers Pull Stock or Collet Pulls Stock strategies.

  Program the data block previous to the Bar Feed Block so that the turret is in a safe position before the program arrives at the Bar Feed Block.

- **TOOL CHANGE OVERRIDE**—refer to *Tool Change Override, on page 3 - 7* for information about these fields.

Tool not Used

This selection is for programming a Bar Feed block with an automatic Bar Feeder. The **Tool not Used** strategy does not require any data fields as the Bar Feeder begins its operation and sends a Bar Feed Complete signal to the control.

The **Dwell (Secs)** field is available for setting the amount of time for the bar feeder to dwell while the chuck finishes opening or closing before continuing.

Tool as Bump Stop

This selection is for programming a Bar Feed block with the optional Bar Feeder. The automatic bar feed will stop when it touches the tool. Select **TOOL AS BUMP STOP** in the **STRATEGY** field. The screen provides this graphic representation of the tool movement.

These fields are available for programming a Bar Feed block with the optional Bar Feeder. When the cursor is in an X or Z field, the STORE MACHINE POSITION *F7* softkey appears. Select this softkey to store the current machine position into the field.

- **X1 (RAD)** or **(DIA)** and **Z1**—identifies the stop position in X (radius or diameter) and Z for the tool when the bar is pushed by the bar feed mechanism.
- **SPINDLE ON**—enables the spindle and allows it to run while bar feeding when YES is selected. This field is only active when the Rotating Bar Feed option has been installed and enabled on the machine.
- **DIRECTION**—identifies CW or CCW as the direction the spindle should turn while bar feeding. This field (not shown above) is only active when the Rotating Bar Feed option has been installed and enabled on the machine and SPINDLE ON is YES.
- **RPM**—identifies the revolutions per minute the spindle should turn while bar feeding. This field (not shown above) is only active when the Rotating Bar Feed option has been installed and enabled on the machine and SPINDLE ON is YES.
- **Dwell (Secs)**—sets the amount of time for the bar feeder to dwell while the chuck finishes opening or closing before continuing.
- **USE PART CATCHER**—activates the optional Parts Catcher when YES is selected and catches the unused stock remnant. Otherwise, the stock remnant falls into the chip tray or conveyor.
- **TOOL CHANGE OVERRIDE**—refer to *Tool Change Override, on page 3 - 7* for information about these fields.

The machine movement is as follows:

1. The tool rapids to position in X1 and Z1.
2. The chuck opens.
3. The bar feeds until it strikes the tool.
4. The chuck closes.

Fingers Pull Stock

This selection is for programming a Bar Feed block that uses a finger style tool that pulls the bar stock and does not require the optional Bar Feeder. Select **FINGERS PULL STOCK** in the **STRATEGY** field. The screen provides this graphic representation of the tool movement.

The following fields appear on the screen, in addition to the fields described above for programming a Bar Feed block. When the cursor is in an X or Z field, the STORE MACHINE POSITION *F7* softkey appears. Select this softkey to store the current machine position into the field.

- **TOOL REQUIRES COOLANT**—identifies whether coolant is required to close the puller tool. Select YES if coolant is required. NO is the default.
- **X1 (RAD)** or **(DIA)** and **Z1**—identifies the stop position in X (radius or diameter) and Z. The tool rapids in XZ. This location is the first position of the cycle.
- **X2** and **Z2**—identifies the position where the pulling tool engages the work piece. The tool feeds in Z first, then X.
- **X3** and **Z3**—identifies the intermediate position where the bar pulling tool feeds. The tool feeds in Z first, then rapids in X after the chuck is closed.
- **DWELL**—identifies the time in seconds that the tool pauses to allow the chuck to open at **X2**, **Z2** and close at **Z3**.
- **X2/Z2 FEED**—identifies the per minute velocity used when engaging the stock.
- **PULL FEED**—identifies the per minute velocity used to pull the stock.
- **TOOL CHANGE OVERRIDE**—refer to *Tool Change Override, on page 3 - 7* for information about these fields.

The machine movement is as follows:

1. The tool rapids to position in X1 and Z1.
2. The tool then feeds at the feedrate programmed in X2/Z2 Feed to Z2 then X2 and dwells for the programmed time to allow the chuck to open and the tool to grab the bar stock.
3. The tool feeds in Z first at the feed programmed in Pull Feed to the intermediate position Z3, dwells for the programmed time to allow the chuck to close, then rapids to position X3.
4. The tool rapids in Z first, then X back to position at X1 and Z1.

Tool Guides Stock

This Strategy selection causes the turret to meet the bar near the chuck and then feed away at the programmed Feed with the bar. Select **TOOL GUIDES STOCK** in the **STRATEGY** field. The screen provides this graphic representation of the tool movement.

The following fields appear on the screen, in addition to the fields described above for programming a Bar Feed block. When the cursor is in an X or Z field, the STORE MACHINE POSITION *F7* softkey appears. Select this softkey to store the current machine position into the field.

- **X1 (RAD)** or **(DIA)** and **Z1**—identifies the stop position in X (radius or diameter) and Z. This location is the first position of the cycle.
- **Z2**—identifies the position where the tool engages the work piece. The tool feeds in Z first, then X.
- **SPINDLE ON**—enables the spindle and allows it to run while bar feeding when YES is selected. This field (not shown above) is only active when the Rotating Bar Feed option has been installed and enabled on the machine.
- **DIRECTION**—identifies CW or CCW as the direction the spindle should turn while bar feeding. This field (not shown above) is only active when the Rotating Bar Feed option has been installed and enabled on the machine and SPINDLE ON is YES.
- **RPM**—identifies the revolutions per minute the spindle should turn while bar feeding. This field (not shown above) is only active when the Rotating Bar Feed option has been installed and enabled on the machine and SPINDLE ON is YES.
- **Z2 FEED**—identifies the per minute velocity used for movement from **Z1** to **Z2**.
- **DWELL**—identifies the time in seconds that the tool pauses at **X1 Z1** to allow the chuck to open.
- **USE PART CATCHER**—activates the optional Parts Catcher when YES is selected and catches the unused stock remnant. Otherwise, the stock remnant falls into the chip tray or conveyor.
- **TOOL CHANGE OVERRIDE**—refer to *Tool Change Override, on page 3 - 7* for information about these fields.

The machine movement is as follows:

1. The tool rapids to position in X1 and Z1, close to the workpiece, and dwells for the programmed time so the chuck can open.
2. The tool feeds at the specified feedrate to Z2, guiding the bar as the bar feeder provides pressure.
3. The chuck closes.

Collet Pulls Stock

This selection is for programming a Bar Feed block that uses a collet style tool puller that pulls the bar stock and does not require the optional Bar Feeder. Select **COLLET PULLS STOCK** in the **STRATEGY** field. The screen provides this graphic representation of the tool movement.

The following fields appear on the screen, in addition to the fields described above for programming a Bar Feed block. When the cursor is in an X or Z field, the STORE MACHINE POSITION *F7* softkey appears. Select this softkey to store the current machine position into the field.

- **TOOL REQUIRES COOLANT**—identifies whether coolant is required to close the puller tool. Select YES if coolant is required. NO is the default.
- **X1 (RAD)** or **(DIA)** and **Z1**—identifies the stop position in X (radius or diameter) and Z. The tool rapids in XZ. This location is the first position of the cycle.
- **X2 (RAD)** or **(DIA)** and **Z2**—identifies the position where the pulling tool engages the work piece. The tool rapids in X first, then feeds in Z.
- **Z3**—identifies the intermediate position where the bar pulling tool feeds. The tool feeds in Z.
- **X4 (RAD)** or **(DIA) and Z4**—identifies the position where the pulling tool rapids away from the work piece. The tool rapids in Z first, then X.
- **DWELL**—identifies the time in seconds that the tool pauses to allow the chuck to open at **X2**, **Z2** and close at **Z3**.
- **Z2 FEED**—identifies the per minute velocity used when engaging the stock.
- **PULL FEED**—identifies the per minute velocity used to pull the stock.
- **TOOL CHANGE OVERRIDE**—refer to *Tool Change Override, on page 3 - 7* for information about these fields.

The machine movement is as follows:

1. The tool rapids to position in X1 and Z1.
2. The tool rapids to X2 first, then feeds at the feedrate programmed in Z2 FEED to Z2 and dwells for the programmed time to allow the chuck to open and the tool to grab the bar stock.
3. The tool feeds in Z at the feedrate programmed in PULL FEED to the intermediate position Z3 and dwells for the programmed time to allow the chuck to close.
4. The tool rapids in Z first, then X to position at X4 and Z4.

Tool Change Override

The Tool Change Override fields appear when either Tool as Bump Stop, Fingers Pull Stock, Tool Guides Stock, or Collet Pulls Stock is selected. These fields are defined as follows:

- **TOOL CHANGE OVERRIDE**—when selected, contains settings to be used if the tool specified in this data block is different than the tool in the previous block, or if the current tool before the Bar Feed block is run, and necessitates a tool change.

  If the tool is the same as the previous block or the current tool, then this area is ignored. If the tool is different, the tool change will occur after the movement to the X and Z axis positions.

  When Tool Change Override is not selected, "Disabled" appears instead of the following fields as shown in *Figure 3–1. Bar Feed Block screen, on page 3 - 2*.

  The turret first moves to the X and Z positions defined for the Tool Change and performs the tool change, then the turret moves to the X1 and Z1 position defined in the data block.

  - Following that move, if the Finger Pulls Stock or the Collet Pulls Stock strategy is selected, the turret moves to the X2Z2 and X3Z3 position.
  - If the Tool Guides Stock strategy is selected, the turret moves to the X1Z1 and Z2 position.
  - **WHERE**—moves turret to the selected position:
    - **DON'T MOVE**—perform the tool change at current machine position.
    - **MOVE TO HOME**—the turret should move to the home position for the tool change.
    - **MOVE TO XZ**—the turret should move to the defined X, Z position. When MOVE TO XZ is selected, the Reference and X (Dia) or X (Rad) fields are available for entering data.
  - **REFERENCE**—determines whether the X and Z locations are relative to Part Zero or Machine Zero. This field is active when MOVE TO XZ is selected above.
  - **X (DIA)** OR **(RAD)**—contains the X diameter or radius coordinate where the turret should move. When the cursor is in this field, the STORE MACHINE POSITION F7 softkey appears. Select this softkey to store the current machine position into the field. This field is active when MOVE TO XZ is selected above.
  - **Z**—contains the Z coordinate where the turret should move. When the cursor is in this field, the STORE MACHINE POSITION F7 softkey appears. Select this softkey to store the current machine position into the field. This field is active when MOVE TO XZ is selected above.

- **AXIS ORDER**—contains the selection for the order the axes will move. This field is active when either MOVE TO HOME or MOVE TO XZ is selected above.
  - **MOVE X THEN Z** *F1*—from the current position before the Position block, the turret moves the X axis first to the X position then the Z axis to the Z position.
  - **MOVE Z THEN X** *F2*—from the current position before the Position block, the turret moves the Z axis first to the Z position then the X axis to the X position.
  - **SIMULTANEOUS** *F3*—from the current position before the Position block, the turret moves both axes together to the X and Z positions.

When the cursor is in either the X (DIA) or (RAD) (diameter or radius) or Z Tool Change Override field, the STORE MACHINE POSITION F7 softkey appears. Select this softkey to store the current machine position into the field.

Automatic Operation

To operate the Bar Feeder automatically, program a Machine Function data block within a conversational part program. From the New Block screen or Program Review screen,

1. Select the Insert Block Before *F7* softkey.
2. Select the Miscellaneous *F4* softkey.
3. Select the M Code *F3* softkey. These M Codes are available for programming:
   - M20 Chuck Open for Bar Feeder Start. Enter the number of seconds to dwell in the Dwell field before moving to the next block.
   - M21 Bar Feeder Finished Close Chuck. Enter the number of seconds to dwell in the Dwell field before moving to the next block.
   - M22 Start Bar Feeder (Bar Load). Enter the number of seconds to dwell in the Dwell field before moving to the next block.
   - M23 Start Bar Feeder for Z Guided Feed. Enter the number of seconds to dwell in the Dwell field before moving to the next block.

⇨ If using NC programming, the M20, M21, M22, and M23 codes apply.

CHIP CONVEYOR

The optional chip conveyor runs horizontal to the machine base, near the floor. Chips are flushed out of the turning center into the chip conveyor tank. When in Forward, the chip conveyor moves the chips from the chip conveyor tank and dispenses them into a chip cart or other receptacle on the right-hand side of the machine. The chip conveyor can also be operated in Reverse to dislodge jams.

⚠ Do not run in Reverse for an extended period of time as chips may collect on the left-hand side of chip conveyor tank and compact.

Chip Conveyor Operation

The chip conveyor can operate in Manual mode, automatically with M Codes, or in Auto Mode. To program an M Code in a conversational programming data block, please refer to WinMax Lathe Conversational Part Programming *Machine Function—M Code, on page 2 - 198*, or for NC programming, see WinMax Lathe NC Programming, *Basic NC M Codes, on page 3 - 1*, or *ISNC M Codes, on page 5 - 1*.

Manual Mode

To operate the chip conveyor in Manual mode,

1. Press the **Manual** console key to access the Manual screen.
2. Select the ACCESSORY OPERATIONS *F4* softkey.
3. Select the CHIP CONVEYOR *F1* softkey.
4. The softkey menu changes to display these Chip Conveyor functions:
   - **CHIP CONVEYOR FORWARD** *F1*—sets the chip conveyor motion to forward to expel chips.
   - **CHIP CONVEYOR REVERSE** *F2*—sets the chip conveyor motion to reverse. The chip conveyor runs in reverse for a preset period of time (approximately 10 seconds) to help prevent pushing chips back into the machine and damaging the motor. This time period is set at the factory.
   - **CHIP CONVEYOR STOP** *F3*—stops the chip conveyor motion.

The CONVEYOR field changes to reflect the current selection: FORWARD, REVERSE, or STOPPED.

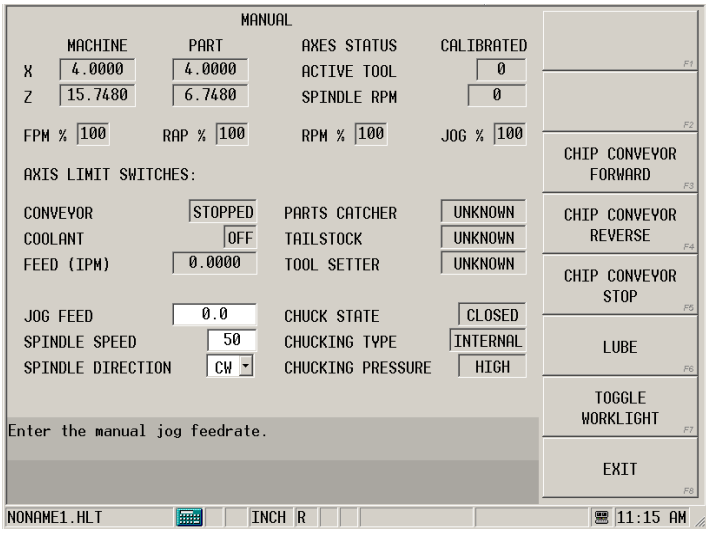


Figure 4–1. Manual screen with Chip Conveyor softkeys

Automatic Operation

To operate the Chip Conveyor automatically, program a Machine Function data block within a conversational part program. From the New Block screen or Program Review screen,

1. Select the Insert Block Before *F7* softkey.
2. Select the Miscellaneous softkey.
3. Select the M Code *F3* softkey. These M Codes are available for programming:
   - M50 Chip Conveyor On. Enter the number of seconds to dwell in the Dwell field before moving to the next block.
   - M51 Chip Conveyor Off. Enter the number of seconds to dwell in the Dwell field before moving to the next block.

When CE/ANSI interlocks are active, the conveyor stops running if the enclosure doors are opened. The conveyor restarts when the doors are shut after the flashing Start Cycle button is pressed.

Auto Mode

To operate the chip conveyor in Auto mode,

1. Press the **Auto** console key to access the Auto screen.
2. Select the RUN PROGRAM *F8* softkey. The Auto Run screen appears with the Chip Conveyor softkeys. Refer to *Manual Mode, on page 4 - 1* for their definitions.

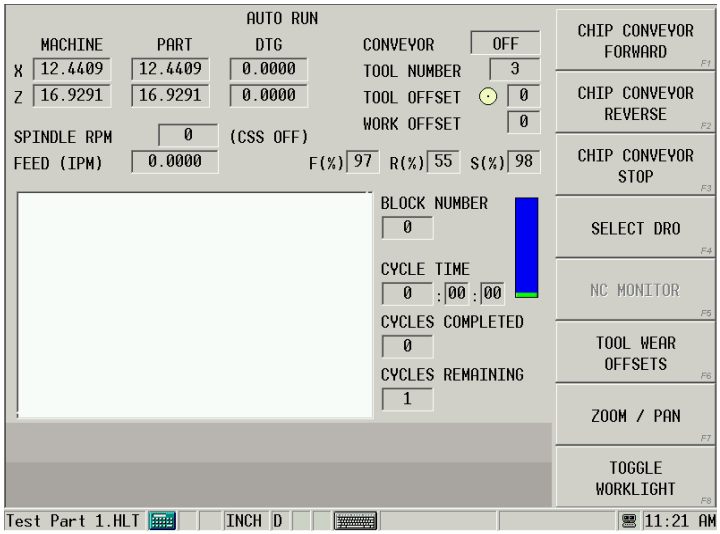


Figure 4–2. Auto Run screen with Chip Conveyor softkeys

COLLET CHUCK

Three chuck types are available: standard external, standard internal, and optional collet. This section describes accessing the optional collet chuck and operating the optional Chuck Footswitch. The chuck is located inside the enclosure on the left side of each turning center.

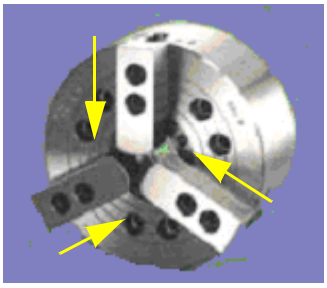
The chuck can operate in Manual mode or with M Codes. To program an M Code in a conversational programming data block, please refer to WinMax Lathe Conversational Part Programming, *Machine Function—M Code, on page 2 - 198*, WinMax Lathe NC Programming, *Basic NC M Codes, on page 3 - 1*, or *ISNC M Codes, on page 5 - 1*.

⚠ Refer to the *Maintenance and Safety Manual for Turning Centers* and the chuck's operator's manual for information about maintaining the chuck.

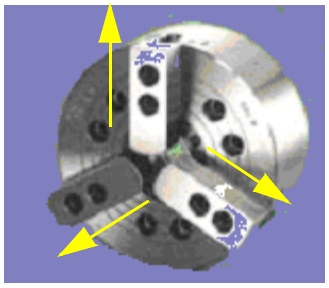
Chuck Types

Select the type of chuck to use from the Manual screen, CHUCK OPERATIONS *F5* softkey menu.

- **External Chuck**—the external type chuck is for solid stock, manually loaded into the chuck from inside the enclosure. The jaws are tightened inward to secure the stock.
- **Internal Chuck**—the internal type chuck is for hollow stock, manually loaded inside the enclosure. The stock is placed around the outside of the chuck and the jaws are tightened outward to secure the stock.
- **Collet Chuck Holder**—the optional collet chuck holder holds the material inside the collet. It is for solid stock, loaded through the chuck from the hole in the outside of the enclosure.



External Closed



Internal Closed

Collet

Figure 5–1. External Closed, Internal Closed, and Collet chucks

Optional Chuck Footswitch

A footswitch is available as an option for the Turning Centers. With each press of the pedal, the chuck opens or closes. The chuck remains in this state until the pedal is pressed again, or a Chuck Operation softkey is pressed to change the state.

The footswitch does not function while the spindle is running.

Jaws

Hurco offers two options for jaws: hard jaws and soft jaws. Hard jaws are typically used for holding raw material with the pressure set very high. Soft jaws are for machining to fit a part, typically for a second operation or a part that has been previously turned.

DXF Option

The Data Exchange Format option (DXF option) lets you convert an AutoCAD™ DXF file into a conversational data block. Add part and tool setup information to create a complete conversational part program.

⇨ You must have the 3D graphics option installed for the DXF option to function. Contact your Hurco Representative for more information.

These topics are discussed in this section:

DXF Overview

DXF files generated with AutoCAD™ Release 9 and later can be used with the DXF option. 2-D Arcs, Lines, Polylines, Points, and block inserts are also supported by the DXF option.

⇨ 3-D Entities, Polyline Mesh or Ruled surfaces, Extended Data and Text Entities cannot be converted by the DXF option.

The DXF option uses the standard features of WinMax Lathe. The following types of data blocks can be created using the DXF option:

- Turning: Profile, Hole, Thread, Groove, and Cutoff.
- Axial Milling: Axial Lines and Arcs with Segments, Axial Circle, Axial Frame, Axial Slot, and Axial Flats
- Axial Holes: Holes Locations

Enter data in the Tool Setup, Part Setup, and Program Parameters screens to create a complete conversational part program.

- If you are programming a part in inches and the DXF file was drawn in metric units, convert the metric units to inches using the DXF Units are MM feature.
- Draw line segments in the middle of the tolerance values in AutoCAD™ to improve data translation accuracy when the file is loaded into WinMax Lathe.
- Group files into subdirectories as described in Getting Started with WinMax Lathe, *Use File Manager, on page 3 - 7*.
- New data blocks added to a part program are inserted after the current block.
- To view new data blocks created from DXF drawings, press the Input console key and select the Part Programming *F3* softkey.
- Press the Draw console key to view the part created.
- If the DXF drawing was scaled, use the scaling feature in the DXF Parameters screen to revise the dimension to a 1:1 ratio.

The DXF screen softkeys provide access to all of the DXF features.

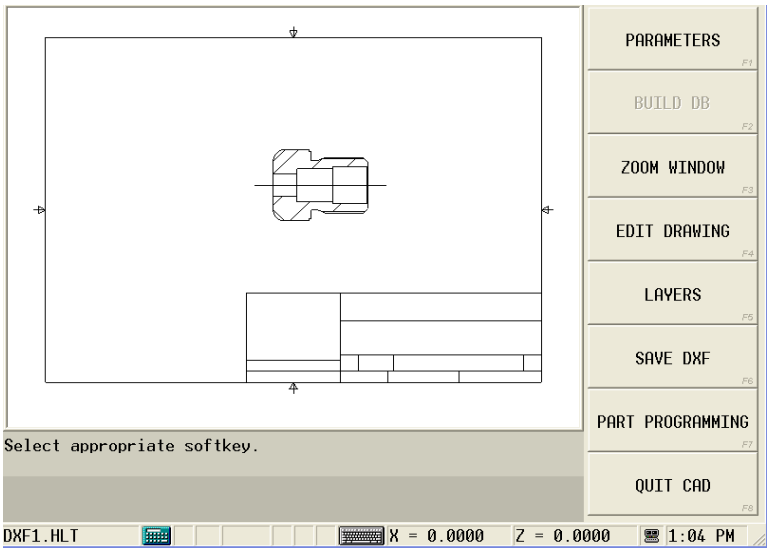


Figure 6–1. DXF Screen

- **PARAMETERS** *F1*—accesses Center Line, Part Zero, and Axial Coordinate System settings.
- **BUILD DB** *F2*—accesses the automatic data block building features described in *Build Data Blocks, on page 6 - 11*.
- **ZOOM WINDOW** *F3*—allows you to enlarge an area in the drawing, or zoom out to see a full view. To identify the zoom area, select a corner, select the opposite corner, and press the Enter console key. These softkey selections also appear:
  - **ZOOM OUT** *F1*—pull back incrementally from the drawing without re-centering the part in the drawing.
  - **FIT TO VIEW** *F2*—gives a full scale of the drawing, auto-centering the part in the drawing.
  - **PAN** *F3*—relocates the center of the drawing on the screen. Select the desired place for the drawing center and press the Enter console key. Turn off the Pan view by selecting the Enter console key, the Fit to View, or Zoom Out softkeys.
- **EDIT DRAWING** *F4*—allows you to Extend, Join, Modify, Split, Delete, or Trim line segments for a part program, and it allows you to Explode PCurves. Refer to *Edit DXF Drawing, on page 6 - 25*.

- **LAYERS** *F5*—allows you to select and display any or all of the layers in the DXF drawing. All layers are turned on when a DXF file is first loaded into WinMax Lathe. The LAYERS *F5* softkey provides a preview pane and accesses this softkey menu:
  - **SELECT LAYER** *F1*—toggles the selected layer on and off.
  - **ALL ON** *F2*—turns on all of the layers.
  - **ALL OFF** *F3*—turns off all of the layers.
  - **EXIT** *F8*—returns to the DXF screen.

  To see only the part geometry, turn off all layers except the drawing (Part) layer.

- **SAVE DXF** *F6*—displays the Program Review screen, allowing you to save the DXF file.
- **PART PROGRAMMING** *F7*—accesses part programming and the data blocks for the current program.
- **QUIT CAD** *F8*—ends the DXF session and exits the DXF file.

Creating a New DXF Part Program

The following steps are required to create a conversational part program from a DXF file:

Step 1: Create a conversational part program to receive the DXF file.

See Getting Started with WinMax Lathe, *Project Manager, on page 3 - 1* for more information.

⇨ You must create a conversational part program before loading the DXF file. If a part program is not created first, the DXF file will not be saved as a part program.

Step 2: Load a DXF file into WinMax Lathe. Refer to Load DXF File, on page 6 - 5.

Step 3: Set the Parameters. Refer to Parameters, on page 6 - 5.

Step 4: Build Data Blocks. Refer to Build Data Blocks, on page 6 - 11.

Step 5: Complete the Program. Refer to Complete the Program, on page 6 - 25.

Step 6: Edit DXF Drawing. Refer to Edit DXF Drawing, on page 6 - 25.

Load DXF File

Load the DXF file from a diskette, a USB device, or a network drive onto one of the hard drive directories on the control.

If the DXF file is large, copy the file onto the hard drive and then open it in WinMax Lathe.

1. To access a DXF file, press the Menu console key, then select the DXF Editor softkey. The Program Review screen displays.
2. Locate and select the DXF file in the Program Review screen.
3. Select the Load Selected File(s) *F1* softkey. The DXF file is loaded into WinMax Lathe and the DXF drawing appears on the screen.

Parameters

After the DXF file has been loaded into WinMax Lathe, the program parameters must be set before building data blocks from the DXF drawing.

1. Select the PARAMETERS *F1* softkey to access the DXF Parameters screen with Center Line, Part Zero, and Axial Coordinate System softkeys.

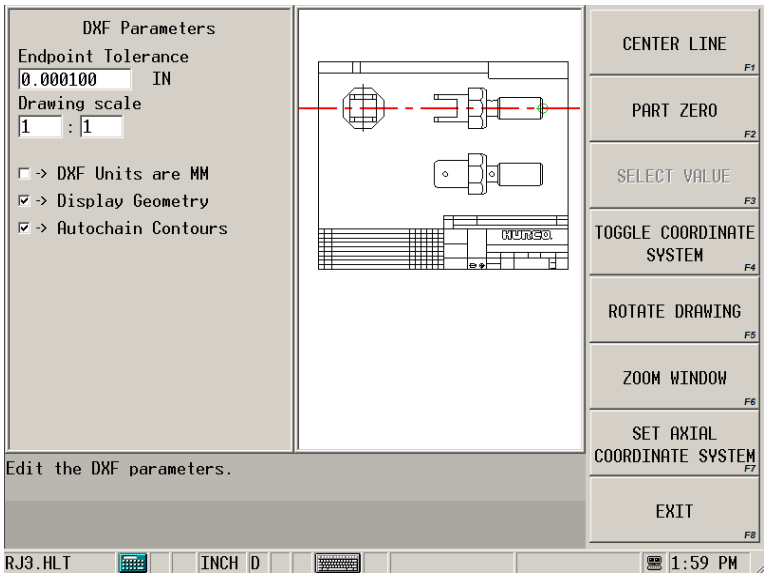


Figure 6–2. DXF Parameters screen

The fields on the left side of the DXF Parameters screen are defined as follows:

- **End Point Tolerance**—determines when the end points of segments are close enough to be considered equal. Setting this parameter to a number within the cutting tolerance links the contour segments. If the parameter exceeds the cutting tolerance, you must modify the drawing to complete contour segment linking. The default value is 0.0001 inches. The parameter range is 0 inches to 1000.0000 inches.
- **Hole Diameter**—determines the default diameter for a hole.
- **Drawing scale**—indicates the scaling ratio for the drawing to the part.
- **DXF Units are MM**—converts selected line segment to inches to match part program units. To convert the metric values of the DXF file to inches, clear the field.
- **Display Geometry**—if selected, converted lines are displayed in a different color and line style. The default value is selected (On).
- **Autochain contours**—line segments are automatically chained to create contours when a DXF file is loaded into WinMax Lathe. If this selection is cleared (Off), you can individually select line segments to create a contour, which is useful for creating open-ended contours for 3-D part programming. The default value is selected (On). Autochain paths are highlighted with dashed lines.
- **Select Holes by Diameter**—selects holes with the diameter specified in the Hole Diameter field (defined above) when the WINDOW SELECT softkey is used. This selection allows you to order the hole selection by size, which optimizes tool changes.

The DXF Parameters menu also includes these softkey choices:

- **SELECT VALUE** *F3*—
- **TOGGLE COORDINATE SYSTEM** *F4*—flips the drawing in the Preview Pane.
- **ROTATE DRAWING** *F5*—rotates the drawing 45 degrees.
- **ZOOM WINDOW** *F6*—allows you to enlarge an area in the drawing, or zoom out to see a full view.
- **EXIT** *F8*—returns to the DXF screen.

Center Line Selection

Select the Center Line F1 softkey. These softkey choices appear for selecting the Center Line:

- **LINE** *F1*—select a line in the drawing for the centerline. The selection can be changed by selecting a different point. Select the ACCEPT *F1* softkey. A dotted red line appears, identifying the programmed centerline. The softkey menu returns to the DXF Parameters menu.

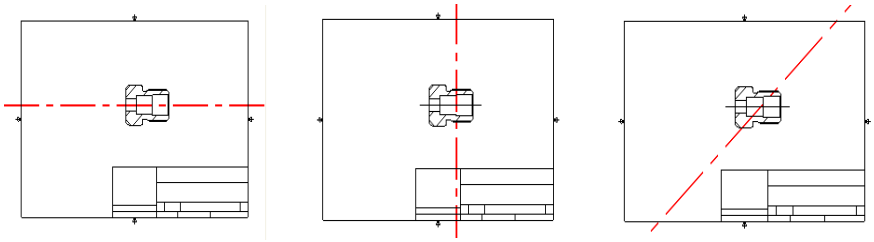


Figure 6–3. Examples of Centerline selections using the Line F1 softkey

- **POINT - POINT** *F2*—accesses selections that function like Object Snaps (Osnaps) in AutoCAD for choosing two points in the drawing to determine the centerline. Select one of the following softkeys followed by the Accept F1 softkey for the first selection. Repeat for the second selection.
  - **INTERSECTION** *F1*—snaps to the intersection where any two drawing objects cross each other.
  - **MIDDLE POINT** *F2*—snaps to the mid points of lines, arcs, and polyine segments.
  - **END POINT** *F3*—snaps to the end points of lines, arcs, and polyline vertices.

  A dotted red line appears, identifying the programmed centerline. System coordinates are also displayed in green at the bottom of the drawing. The softkey menu returns to the DXF Parameters menu.

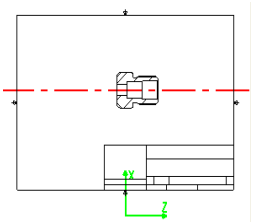


Figure 6–4. Example of Centerline selection using the Point - Point F2 softkey menu

- **POINT - ANGLE** *F3*—accesses selections that function like Object Snaps (Osnaps) in AutoCAD for choosing one point and an angle in the drawing to determine the centerline.
  - **INTERSECTION** *F1*—snaps to the intersection where any two drawing objects cross each other.
  - **MIDDLE POINT** *F2*—snaps to the mid points of lines, arcs, and polyine segments.
  - **END POINT** *F3*—snaps to the end points of lines, arcs, and polyline vertices.

  A dotted red line appears to identify the current centerline. System coordinates are also displayed in green at the bottom of the drawing.

  Softkeys appear for selecting the degree of the angle to determine the centerline: 0 DEGREES *F2*, 45 DEGREES *F3*, 90 DEGREES *F4*, 135 DEGREES *F5*, and 180 DEGREES *F6*.

  Select the appropriate degree softkey followed by the ACCEPT *F1* softkey. A dotted red line appears, identifying the programmed centerline. System coordinates are also displayed in green at the bottom of the drawing. The softkey menu returns to the DXF Parameters menu.

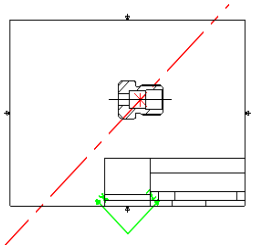


Figure 6–5. Example of Centerline selection using the Point - Angle F3 softkey menu

Part Zero Selection

1. From the DXF Parameters menu, select the PART ZERO *F2* softkey.
2. Select the Part Zero location on the drawing followed by the ACCEPT *F1* softkey. A green circle with a crosshair cursor appears at the selected location. The softkey menu returns to the DXF Parameters menu.

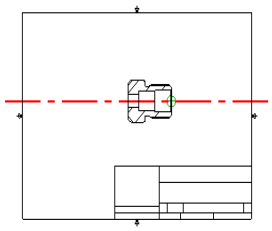


Figure 6–6. Example of Part Zero selection

Axial Coordinate System

- **SET AXIAL COORDINATE SYSTEM** *F7*—accesses the SET AXIAL PROGRAM ZERO *F1* and ROTATE COORDINATE SYSTEM *F2* softkeys for setting axial coordinates.
  - **SET AXIAL PROGRAM ZERO** *F1*—identifies Axial Part Zero. Select this softkey then select the method to select axial part zero using one of the following softkey choices:
    - **INTERSECTION** *F1*—identifies the intersection point of two lines.
    - **CENTER** *F2*—identifies an arc on the drawing.
    - **MIDDLE POINT** *F3*—identifies a line on the drawing.

    After identifying Axial Part Zero, select the ACCEPT F1 softkey. A pink circle represents the location on the Parameters screen.
  - **ROTATE COORDINATE SYSTEM** *F2*—rotates the Axial Part Zero to the opposite side of the line.

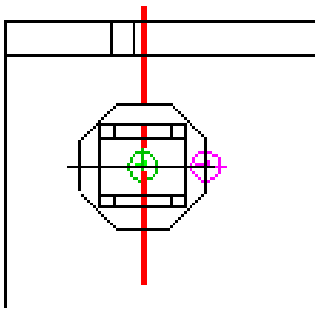


Figure 6–7. Example of Axial Part Zero

Following is a sample DXF screen with the Centerline and Part Zero programmed:

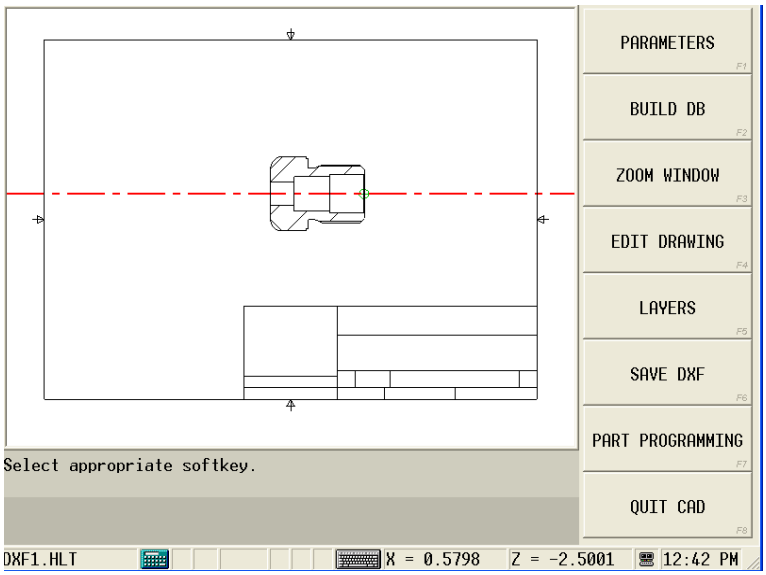


Figure 6–8. DXF Screen with Centerline and Part Zero programmed

Build Data Blocks

The Build Data Blocks feature allows you to select locations on the DXF drawing and automatically create data blocks and elements or segments.

Review and revise DXF parameters *before* building data blocks.

Select the BUILD DB *F2* softkey from the DXF screen.

The Centerline and Part Zero must be defined in the DXF Parameters screen before building data blocks.

Turning Data Blocks

The following types of Turning data blocks are available to build using a DXF file:

Profile

Select the **TURNING** *F1* softkey from the Build DB menu followed by the **PROFILE** *F1* softkey to build a Profile data block, complete with Elements.

These softkey choices are available for Profiles:

- **ACCEPT** *F1*—accepts the selection and loads the information for it into a data block.
- **UNDO LAST** *F2*—clears the previous selection and does not load the information for it into a data block.
- **ENABLE AUTO CHAINING** *F3*—forms contours by connecting individual line and arc segments together. Toggles with Disable Auto Chaining *F4*.
- **DISABLE AUTO CHAINING** *F4*—does not connect individual line and ar segments together to form contours. Toggles with Enable Auto Chaining *F3*.
- **PICK RAPID POINT** *F5*—identifies the selected location as the point where rapid traverse begins at the beginning of the cycle and ends when the cycle is complete.
- **ZOOM WINDOW** *F6*—allows you to enlarge an area in the drawing, or zoom out to see a full view. To identify the zoom area, select a corner, select the opposite corner, and press the Enter console key. These softkey selections also appear:
  - **ZOOM OUT** *F1*—pull back incrementally from the drawing without re-centering the part in the drawing.
  - **FIT TO VIEW** *F2*—gives a full scale of the drawing, auto-centering the part in the drawing.
  - **PAN** *F3*—relocates the center of the drawing on the screen. Select the desired place for the drawing center and press the Enter console key. Turn off the Pan view by selecting the Enter console key, the Fit to View, or Zoom Out softkeys.
- **EXIT** *F8*—returns to the Build DB screen.

To build a Profile data block:

1. Enter the Centerline and Part Zero information for the part program using the DXF drawing.
   a. Select the Parameters *F1* softkey.
   b. Define the Centerline and Part Zero.
2. Select the PICK RAPID POINT *F5* softkey to define the XZ Rapid Position.
3. Select the first element of the contour, either a face or turn.
4. Continue selecting Turn and Face elements to define the entire Profile.
5. Select the ACCEPT *F1* softkey to create the data block.

After building a Profile data block from the DXF drawing, it may be necessary to define a Turn element as the first element of the Profile and a Face element as the last element of the Profile to complete the required Profile geometry.

Hole

Select the **TURNING** *F1* softkey from the Build DB menu followed by the **HOLE** *F2* softkey to build an Axial Hole data block by making selections on the DXF drawing. These softkey choices are available for Holes:

- **ACCEPT** *F1*—accepts the selection and loads the information for it into a data block.
- **EXTEND INTERSECTION** *F2*—moves the end of the intersection to the selected location within the stock.
- **EXTEND DYNAMICALLY** *F3*—moves the end of the intersection to the selected location dynamically, beyond the boundaries of the stock.
- **ZOOM WINDOW** *F6*—allows you to enlarge an area in the drawing, or zoom out to see a full view. To identify the zoom area, select a corner, select the opposite corner, and press the Enter console key. These softkey selections also appear:
  - **ZOOM OUT** *F1*—pull back incrementally from the drawing without re-centering the part in the drawing.
  - **FIT TO VIEW** *F2*—gives a full scale of the drawing, auto-centering the part in the drawing.
  - **PAN** *F3*—relocates the center of the drawing on the screen. Select the desired place for the drawing center and press the Enter console key. Turn off the Pan view by selecting the Enter console key, the Fit to View, or Zoom Out softkeys.
- **EXIT** *F8*—returns to the Build DB screen.

To build an Axial Hole data block:

1. Enter the Centerline and Part Zero information for the part program using the DXF drawing.
   a. Select the Parameters *F1* softkey.
   b. Define the Centerline and Part Zero.
2. Select a line that defines the wall of the hole.
3. Use either the EXTEND INTERSECTION *F2* or EXTEND DYNAMICALLY *F3* softkey to define the Z End and Z Start of the hole.
4. Select the ACCEPT *F1* softkey to create the data block.

Thread

Select the **TURNING** *F1* softkey from the Build DB menu followed by the **THREAD** *F3* softkey to build a Thread data block, complete with choices for types of threading, by making selections on the DXF drawing. These softkey choices are available for Threading:

- **OD STRAIGHT** *F1*—loads information for a Threading data block with an outside diameter straight thread.
  - **ACCEPT** *F1*—accepts the selection and loads the information for it into a data block.
  - **EXTEND INTERSECTION** *F2*—moves the end of the intersection to the selected location within the stock.
  - **EXTEND DYNAMICALLY** *F3*—moves the end of the intersection to the selected location dynamically, beyond the boundaries of the stock.
  - **ZOOM WINDOW** *F6*—allows you to enlarge an area in the drawing, or zoom out to see a full view. To identify the zoom area, select a corner, select the opposite corner, and press the Enter console key. These softkey selections also appear:
    - **ZOOM OUT** *F1*—pull back incrementally from the drawing without re-centering the part in the drawing.
    - **FIT TO VIEW** *F2*—gives a full scale of the drawing, auto-centering the part in the drawing.
    - **PAN** *F3*—relocates the center of the drawing on the screen. Select the desired place for the drawing center and press the Enter console key. Turn off the Pan view by selecting the Enter console key, the Fit to View, or Zoom Out softkeys.
  - **EXIT** *F8*—returns to the Build DB screen.
- **OD TAPER** *F2*—loads information for a Threading data block with an outside diameter taper thread.
  - **ACCEPT** *F1*—accepts the selection and loads the information for it into a data block.
  - **EXTEND INTERSECTION** *F2*—moves the end of the intersection to the selected location within the stock.
  - **EXTEND DYNAMICALLY** *F3*—moves the end of the intersection to the selected location dynamically, beyond the boundaries of the stock.
  - **ZOOM WINDOW** *F6*—allows you to enlarge an area in the drawing, or zoom out to see a full view. To identify the zoom area, select a corner, select the opposite corner, and press the Enter console key. These softkey selections also appear:
    - **ZOOM OUT** *F1*—pull back incrementally from the drawing without re-centering the part in the drawing.
    - **FIT TO VIEW** *F2*—gives a full scale of the drawing, auto-centering the part in the drawing.
    - **PAN** *F3*—relocates the center of the drawing on the screen. Select the desired place for the drawing center and press the Enter console key. Turn off the Pan view by selecting the Enter console key, the Fit to View, or Zoom Out softkeys.

- **EXIT** *F8*—returns to the Build DB screen.

- **ID STRAIGHT** *F3*—loads information for a Threading data block with an inside diameter straight thread.
  - **ACCEPT** *F1*—accepts the selection and loads the information for it into a data block.
  - **EXTEND INTERSECTION** *F2*—moves the end of the intersection to the selected location within the stock.
  - **EXTEND DYNAMICALLY** *F3*—moves the end of the intersection to the selected location dynamically, beyond the boundaries of the stock.
  - **ZOOM WINDOW** *F6*—allows you to enlarge an area in the drawing, or zoom out to see a full view. To identify the zoom area, select a corner, select the opposite corner, and press the Enter console key. These softkey selections also appear:
    - **ZOOM OUT** *F1*—pull back incrementally from the drawing without re-centering the part in the drawing.
    - **FIT TO VIEW** *F2*—gives a full scale of the drawing, auto-centering the part in the drawing.
    - **PAN** *F3*—relocates the center of the drawing on the screen. Select the desired place for the drawing center and press the Enter console key. Turn off the Pan view by selecting the Enter console key, the Fit to View, or Zoom Out softkeys.
  - **EXIT** *F8*—returns to the Build DB screen.

- **ID TAPER** *F4*—loads information for a Threading data block with an inside diameter taper thread.
  - **ACCEPT** *F1*—accepts the selection and loads the information for it into a data block.
  - **EXTEND INTERSECTION** *F2*—moves the end of the intersection to the selected location within the stock.
  - **EXTEND DYNAMICALLY** *F3*—moves the end of the intersection to the selected location dynamically, beyond the boundaries of the stock.
  - **ZOOM WINDOW** *F6*—allows you to enlarge an area in the drawing, or zoom out to see a full view. To identify the zoom area, select a corner, select the opposite corner, and press the Enter console key. These softkey selections also appear:
    - **ZOOM OUT** *F1*—pull back incrementally from the drawing without re-centering the part in the drawing.
    - **FIT TO VIEW** *F2*—gives a full scale of the drawing, auto-centering the part in the drawing.
    - **PAN** *F3*—relocates the center of the drawing on the screen. Select the desired place for the drawing center and press the Enter console key. Turn off the Pan view by selecting the Enter console key, the Fit to View, or Zoom Out softkeys.
  - **EXIT** *F8*—returns to the Build DB screen.

- **FACE** *F5*—loads information for a Threading data block with a face thread.
  - **ACCEPT** *F1*—accepts the selection and loads the information for it into a data block.
  - **EXTEND INTERSECTION** *F2*—moves the end of the intersection to the selected location within the stock.
  - **EXTEND DYNAMICALLY** *F3*—moves the end of the intersection to the selected location dynamically, beyond the boundaries of the stock.
  - **ZOOM WINDOW** *F6*—allows you to enlarge an area in the drawing, or zoom out to see a full view. To identify the zoom area, select a corner, select the opposite corner, and press the Enter console key. These softkey selections also appear:
    - **ZOOM OUT** *F1*—pull back incrementally from the drawing without re-centering the part in the drawing.
    - **FIT TO VIEW** *F2*—gives a full scale of the drawing, auto-centering the part in the drawing.
    - **PAN** *F3*—relocates the center of the drawing on the screen. Select the desired place for the drawing center and press the Enter console key. Turn off the Pan view by selecting the Enter console key, the Fit to View, or Zoom Out softkeys.
  - **EXIT** *F8*—returns to the Build DB screen.
- **EXIT** *F8*—returns to the DXF screen.

To build a Threading data block:

1. Enter the Centerline and Part Zero information for the part program using the DXF drawing.
   a. Select the Parameters *F1* softkey.
   b. Define the Centerline and Part Zero.
2. Select the type of Threading cycle from the menu.
3. Select the geometry that defines the thread location.
4. Use either the EXTEND INTERSECTION *F2* or EXTEND DYNAMICALLY *F3* softkey to define the Z End and Z Start of the thread.
5. Select the ACCEPT *F1* softkey to create the data block.

Groove

Select the **TURNING** *F1* softkey from the Build DB menu followed by the **GROOVE** *F4* softkey to build a Groove data block by making selections on the DXF drawing. These softkey choices are available for Grooves:

- **ACCEPT** *F1*—accepts the selection and loads the information for it into a data block.
- **UNDO LAST** *F2*—clears the previous selection and does not load the information for it into a data block.
- **SKIP CORNER** *F3*—loads information for corners inside the groove into a data block without selecting them in the DXF file.
- **ZOOM WINDOW** *F6*—allows you to enlarge an area in the drawing, or zoom out to see a full view. To identify the zoom area, select a corner, select the opposite corner, and press the Enter console key. These softkey selections also appear:
  - **ZOOM OUT** *F1*—pull back incrementally from the drawing without re-centering the part in the drawing.
  - **FIT TO VIEW** *F2*—gives a full scale of the drawing, auto-centering the part in the drawing.
  - **PAN** *F3*—relocates the center of the drawing on the screen. Select the desired place for the drawing center and press the Enter console key. Turn off the Pan view by selecting the Enter console key, the Fit to View, or Zoom Out softkeys.
- **EXIT** *F8*—returns to the Build DB screen.

To build a Groove data block:

1. Enter the Centerline and Part Zero information for the part program using the DXF drawing.
   a. Select the Parameters *F1* softkey.
   b. Define the Centerline and Part Zero.
2. Select the line that defines the OD or ID X Location of the Groove.
3. Select the line that defines the right wall of the Groove.
4. Select the line that defines the bottom of the Groove.

   ⇨ The location of the bottom segment relative to the first segment determines if this is an OD or ID Groove.

5. Select the line that defines the left wall of the Groove.
6. Either select the ACCEPT *F1* softkey to create a Groove with square corners or continue to define the Groove.
7. To continue to define the Groove, select the feature, either a Chamfer or Radius, to define the upper right corner.

   ⇨ You may select the SKIP CORNER *F3* softkey to skip any corner and define the subsequent corner.

8. Select all applicable features to define each corner and select the ACCEPT *F1* softkey to create the data block.

Cutoff

Select the **TURNING** *F1* softkey from the Build DB menu followed by the **CUTOFF** *F5* softkey to build a Cutoff data block by making selections on the DXF drawing. These softkey choices are available for Cutoff:

- **ACCEPT** *F1*—accepts the selection and loads the information for it into a data block.
- **EXTEND INTERSECTION** *F2*—moves the end of the intersection to the selected location within the stock.
- **EXTEND FREEHAND** *F3*—moves the end of the intersection to the selected location dynamically, beyond the boundaries of the stock.
- **ZOOM WINDOW** *F6*—allows you to enlarge an area in the drawing, or zoom out to see a full view. To identify the zoom area, select a corner, select the opposite corner, and press the Enter console key. These softkey selections also appear:
  - **ZOOM OUT** *F1*—pull back incrementally from the drawing without re-centering the part in the drawing.
  - **FIT TO VIEW** *F2*—gives a full scale of the drawing, auto-centering the part in the drawing.
  - **PAN** *F3*—relocates the center of the drawing on the screen. Select the desired place for the drawing center and press the Enter console key. Turn off the Pan view by selecting the Enter console key, the Fit to View, or Zoom Out softkeys.
- **EXIT** *F8*—returns to the Build DB screen.

To build a Cutoff data block:

1. Enter the Centerline and Part Zero information for the part program using the DXF drawing.
   a. Select the Parameters *F1* softkey.
   b. Define the Centerline and Part Zero.
2. Select a segment to define the Cutoff wall.
3. Use either the EXTEND INTERSECTION *F2* or EXTEND FREEHAND F3 softkey to define the X End and X Start point of the Cutoff data block.
4. Select the ACCEPT *F1* softkey to create the Cutoff data block.

Axial Milling Data Blocks

The following types of Axial Milling data blocks are available to build using a DXF file:

Axial Lines and Arcs

Select the **AXIAL MILLING** *F2* softkey from the Build DB menu followed by the **AXIAL LINES AND ARCS** *F1* softkey to build an Axial Lines and Arcs data block, complete with Segments, by making selections on the DXF drawing. These softkey choices are available for Axial Lines and Arcs:

- **ACCEPT** *F1*—accepts the selection and loads the information for it into a data block.
- **ENABLE AUTO CHAINING** *F3*—forms contours by connecting individual line and arc segments together. Toggles with Disable Auto Chaining *F4*.
- **DISABLE AUTO CHAINING** *F4*—does not connect individual line and arc segments together to form contours. Toggles with Enable Auto Chaining *F3*.
- **ZOOM WINDOW** *F6*—allows you to enlarge an area in the drawing, or zoom out to see a full view. To identify the zoom area, select a corner, select the opposite corner, and press the Enter console key. These softkey selections also appear:
  - **ZOOM OUT *F1***—pull back incrementally from the drawing without re-centering the part in the drawing.
  - **FIT TO VIEW *F2***—gives a full scale of the drawing, auto-centering the part in the drawing.
  - **PAN *F3***—relocates the center of the drawing on the screen. Select the desired place for the drawing center and press the Enter console key. Turn off the Pan view by selecting the Enter console key, the Fit to View, or Zoom Out softkeys.
- **EXIT** *F8*—returns to the Build DB screen.

To build an Axial Lines and Arcs data block:

1. Enter the Part Zero information for the part program using the DXF drawing.
   a. Select the Parameters *F1* softkey.
   b. Select the Set Axial Coordinate System *F1* softkey.
   c. Select the Set Axial Program Zero *F1* softkey.
   d. Select either Intersection *F1*, Center *F2*, or Middle Point *F3* as the method of selecting Axial Part Zero.
   e. Select the Accept *F1* softkey.
2. Select the Build DB *F2* softkey.
3. Select the Axial Milling *F2* softkey.
4. Select the Axial Lines and Arcs *F1* softkey.
5. Select the start of the contour on the DXF drawing.
6. Select segments to chain the contour on the DXF drawing.
7. Select the Accept *F1* softkey to create the Axial Lines and Arcs contour data block with Segments.

Axial Circle

Select the **AXIAL MILLING** *F2* softkey from the Build DB menu followed by the **AXIAL CIRCLE** *F2* softkey to build an Axial Circle data block by making selections on the DXF drawing. Establish the center point of the circle by selecting from these softkeys:

- **INTERSECTION** *F1*
- **CENTER** *F2*
- **MIDDLE POINT** *F3*

To build an Axial Circle data block:

1. Enter the Centerline and Part Zero information for the part program using the DXF drawing.
   a. Select the Parameters *F1* softkey.
   b. Select the Set Axial Program Zero *F1* softkey.
   c. Select either Intersection *F1*, Center *F2*, or Middle Point *F3* as the method of selecting Axial Part Zero.
   d. Select the Accept *F1* softkey.
2. Select the Build DB *F2* softkey.
3. Select the Axial Milling *F2* softkey.
4. Select the Axial Circle *F2* softkey.
5. Establish the center point of the circle by selecting from these softkeys and making selections on the drawing:
   - **INTERSECTION** *F1*
   - **CENTER** *F2*
   - **MIDDLE POINT** *F3*
6. Select the Accept *F1* softkey to create the Axial Circle data block.

Axial Frame

Select the **AXIAL MILLING** *F2* softkey from the Build DB menu followed by the **AXIAL FRAME** *F3* softkey to build an Axial Frame data block by making selections on the DXF drawing. These softkey choices are available for Axial Frame:

- **ACCEPT** *F1*—accepts the selection and loads the information for it into a data block.
- **ZOOM WINDOW** *F2*——allows you to enlarge an area in the drawing, or zoom out to see a full view. To identify the zoom area, select a corner, select the opposite corner, and press the Enter console key. These softkey selections also appear:
  - **ZOOM OUT** *F1*—pull back incrementally from the drawing without re-centering the part in the drawing.
  - **FIT TO VIEW** *F2*—gives a full scale of the drawing, auto-centering the part in the drawing.
  - **PAN** *F3*—relocates the center of the drawing on the screen. Select the desired place for the drawing center and press the Enter console key. Turn off the Pan view by selecting the Enter console key, the Fit to View, or Zoom Out softkeys.
- **EDIT DRAWING** *F3*—refer to *Edit DXF Drawing, on page 6 - 25*.
- **REVERSE** *F4*—reverses the contour direction.
- **AUTOCHAIN** *F5*—defines contours by autochaining individual segments together. Autochain paths are highlighted with dashed lines.
- **DEFAULT RADIUS** *F6*—inserts the value of the default radius set in the Frame screen's Corner Radius field.
- **EXIT/CANCEL** *F8*—returns to the Build DB screen.

To build an Axial Frame data block:

1. Enter the Centerline and Part Zero information for the part program using the DXF drawing.
   a. Select the Parameters *F1* softkey.
   b. Select the Set Axial Program Zero *F1* softkey.
   c. Select either Intersection *F1*, Center *F2*, or Middle Point *F3* as the method of selecting Axial Part Zero.
   d. Select the Accept *F1* softkey.
2. Select the Build DB *F2* softkey.
3. Select the Axial Milling *F2* softkey.
4. Select the Axial Frame *F3* softkey.
5. Select the geometry on the drawing for the Frame.
6. Select the Accept *F1* softkey to create the Axial Frame data block.

Axial Slot

Select the **AXIAL MILLING** *F2* softkey from the Build DB menu followed by the **AXIAL SLOT** *F4* softkey to build an Axial Slot data block by making selections on the DXF drawing. These softkey choices are available for Axial Slot:

- **ACCEPT** *F1*—accepts the selection and loads the information for it into a data block.
- **ZOOM WINDOW** *F6*——allows you to enlarge an area in the drawing, or zoom out to see a full view. To identify the zoom area, select a corner, select the opposite corner, and press the Enter console key. These softkey selections also appear:
  - **ZOOM OUT** *F1*—pull back incrementally from the drawing without re-centering the part in the drawing.
  - **FIT TO VIEW** *F2*—gives a full scale of the drawing, auto-centering the part in the drawing.
  - **PAN** *F3*—relocates the center of the drawing on the screen. Select the desired place for the drawing center and press the Enter console key. Turn off the Pan view by selecting the Enter console key, the Fit to View, or Zoom Out softkeys.
- **EXIT** *F8*—returns to the Build DB screen.

To build an Axial Slot data block:

1. Enter the Part Zero information for the part program using the DXF drawing.
   a. Select the Parameters *F1* softkey.
   b. Select the Set Axial Program Zero *F1* softkey.
   c. Select either Intersection *F1*, Center *F2*, or Middle Point *F3* as the method of selecting Axial Part Zero.
   d. Select the Accept *F1* softkey.
2. Select the Build DB *F2* softkey.
3. Select the Axial Milling *F2* softkey.
4. Select the Axial Slot *F4* softkey.
5. Select the end point of a slot edge of the slot on the DXF drawing. An X appears to mark the location.
6. Select the Accept *F1* softkey.
7. Select the other end point of the slot edge on the DXF drawing. An X appears to mark the location.
8. Select the Accept *F1* softkey. A dotted line appears indicating the location of the first side of the slot.
9. Select the opposite edge of the slot, parallel to the first side of the slot. A dotted line appears showing the side of the slot.
10. Select the Accept *F1* softkey. Softkeys appear for identifying the Start Cap type.
11. Select Line *F4*, Included Arc *F5*, or Appended Arc *F6* for the type of Start Cap.

12. Select the Accept *F1* softkey. A dotted line appears showing the Start Cap. Softkeys appear for identifying the End Cap type.
13. Select Line F4, Included Arc *F5*, or Appended Arc *F6* for the type of End Cap.
14. Select the Accept *F1* softkey. A dotted line appears showing the End Cap.
15. Select the Accept *F1* softkey to create the Axial Slot data block.

Axial Flats

Select the **AXIAL MILLING** *F2* softkey from the Build DB menu followed by the **AXIAL FLATS** *F5* softkey to build an Axial Flats data block by making selections on the DXF drawing. These softkey choices are available for Axial Flats:

- **ACCEPT** *F1*—accepts the selection and loads the information for it into a data block.
- **ZOOM WINDOW** *F6*—allows you to enlarge an area in the drawing, or zoom out to see a full view. To identify the zoom area, select a corner, select the opposite corner, and press the Enter console key. These softkey selections also appear:
  - **ZOOM OUT *F1***—pull back incrementally from the drawing without re-centering the part in the drawing.
  - **FIT TO VIEW *F2***—gives a full scale of the drawing, auto-centering the part in the drawing.
  - **PAN *F3***—relocates the center of the drawing on the screen. Select the desired place for the drawing center and press the Enter console key. Turn off the Pan view by selecting the Enter console key, the Fit to View, or Zoom Out softkeys.
- **EXIT** *F8*—returns to the Build DB screen.

To build an Axial Flats data block:

1. Enter the Part Zero information for the part program using the DXF drawing.
   a. Select the Parameters *F1* softkey.
   b. Select the Set Axial Program Zero *F1* softkey.
   c. Select either Intersection *F1*, Center *F2*, or Middle Point *F3* as the method of selecting Axial Part Zero.
   d. Select the Accept *F1* softkey.
2. Select the Build DB *F2* softkey.
3. Select the Axial Milling *F2* softkey.
4. Select the Axial Flats *F5* softkey.
5. Select the start point of a flats geometry on the DXF drawing. An X appears to mark the location.
6. Select the Accept *F1* softkey. A dotted line appears in the shape of a square with the default of four sides.
7. Select the Increase Number of Sides *F2* or Reduce Number of Sides *F3* softkey to increase or reduce the number of flats.

8. Select the Accept *F1* softkey. A dotted line appears showing the Flats.
9. Select the Accept *F1* softkey to create the Axial Flats data block.

Axial Holes Data Blocks

The following types of Axial Holes data blocks are available to build using a DXF file:

Holes Locations

Select the **AXIAL MILLING** *F2* softkey from the Build DB menu followed by the **AXIAL HOLES** *F3* softkey to build an Axial Holes data block, complete with Holes Locations, by making selections on the DXF drawing. These softkey choices are available for Holes Locations:

- **HOLES LCOATIONS** *F1*—builds Holes Locations blocks from selected points on the drawing.
- **ACCEPT** *F1*—accepts the selection and loads the information for it into a data block.
- **CLEAR LAST SELECTED** *F2*—
- **CLEAR ALL** *F3*—
- **WINDOW SELECT** *F4*—selects a group of holes on the drawing.
- **ZOOM WINDOW** *F6*—allows you to enlarge an area in the drawing, or zoom out to see a full view. To identify the zoom area, select a corner, select the opposite corner, and press the Enter console key. These softkey selections also appear:
  - **ZOOM OUT** *F1*—pull back incrementally from the drawing without re-centering the part in the drawing.
  - **FIT TO VIEW** *F2*—gives a full scale of the drawing, auto-centering the part in the drawing.
  - **PAN** *F3*—relocates the center of the drawing on the screen. Select the desired place for the drawing center and press the Enter console key. Turn off the Pan view by selecting the Enter console key, the Fit to View, or Zoom Out softkeys.
- **EXIT** *F8*—returns to the Build DB screen.

To build an Axial Holes data block:

1. Enter the Centerline and Part Zero information for the part program using the DXF drawing.
   a. Select the Parameters *F1* softkey.
   b. Select the Set Axial Program Zero *F1* softkey.
   c. Select either Intersection *F1*, Center *F2*, or Middle Point *F3* as the method of selecting Axial Part Zero.
   d. Select the Accept *F1* softkey.
2. Select the Build DB *F2* softkey.
3. Select the Axial Holes *F3* softkey.

4. Select locations on the drawing for each hole location.
5. Select the Accept *F1* softkey to create the Axial Holes Locations data block.

Complete the Program

Before you can run a part program created from a DXF drawing file, the Tool Setup, Part Setup, and Program Parameter data must be defined. Refer to Getting Started with WinMax Lathe, *Part Setup—Work Offsets, on page 4 - 11*, *Tool Setup—Geometry Offsets, on page 4 - 17*, and *Program Parameters, on page 4 - 82* for more information.

You may display and enter tool, part, and parameter information while the data blocks are being created from the DXF file, or after all the DXF information has been loaded into data blocks.

⇨ When you save the part program any changes made to the DXF drawing will not be saved.

⚠ You must examine the tool path for the part program and use the graphics to draw the part before attempting to run a program created from a DXF file. For example, on roughing data blocks the first and last moves onto and off of the part will need to be added to the data block.

Edit DXF Drawing

To edit a DXF drawing, from the DXF screen select the EDIT DRAWING *F4* softkey. For dual-screen consoles, this softkey is located on the right-hand Graphics screen. You can extend, join, modify, split, delete, trim, and explode a line segment using the Edit Drawing function. These softkeys appear:

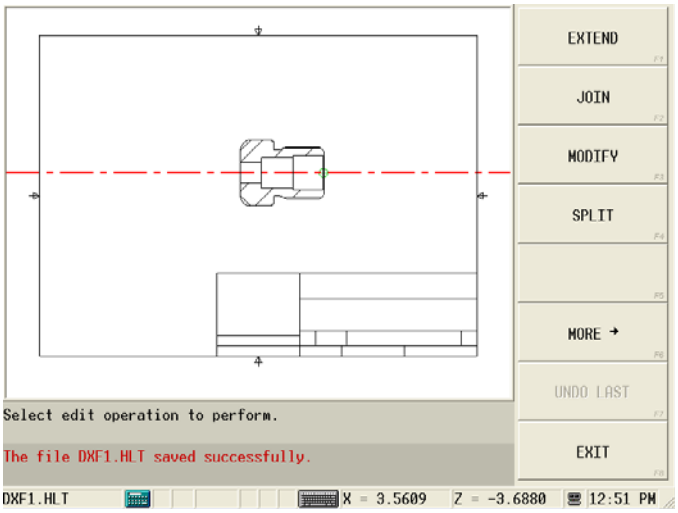


Figure 6–9. DXF Edit Drawing screen

The Edit Drawing softkeys are described below. Some examples follow.

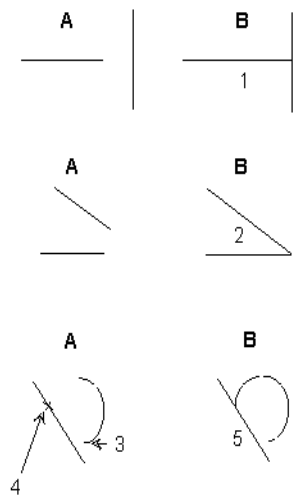
- **EXTEND** *F1*—locate the intersection of two lines and extend one or both of the lines to the intersection point.
- **JOIN** *F2*—move a selected line endpoint to the endpoint of a line or arc segment.
- **MODIFY** *F3*—view or modify the geometry of up to three line segments.
- **SPLIT** *F4*—divide segments for selection, de-selection, chaining, and 3-D operations. Line segments may split at a midpoint or any point of intersection with another segment.
- **MORE ➔** *F6*—accesses a second softkey menu or returns to the first softkey menu.
- **DELETE** *F1*—deletes a selected segment.
- **TRIM** *F2*—trims a selected segment.
- **EXPLODE PCURVE** *F3*—shows an exploded view of a selected PolyCurve.
- **UNDO LAST** *F7*—clears the previous selection and does not load the information for it into a data block.
- **EXIT** *F8*—exit the Edit Drawing screen without saving edits.

Extend

To extend lines:

1. Select the EDIT DRAWING *F4* softkey.
2. Select the EXTEND *F1* softkey.
3. Select the two lines that need to be extended. Both lines are highlighted when selected and extended to their points of intersection.
4. Select the ACCEPT *F2* softkey to accept the line extension or the EXIT/CANCEL *F8* softkey to cancel.

Following are three examples of the extend function:



A	Original drawing
B	Edited drawing
1	Extended line
2	Both lines extended
3	Start of arc
4	Selected intersection point
5	Extended arc

Figure 6–10. Extended Lines and an Extended Arc

Join

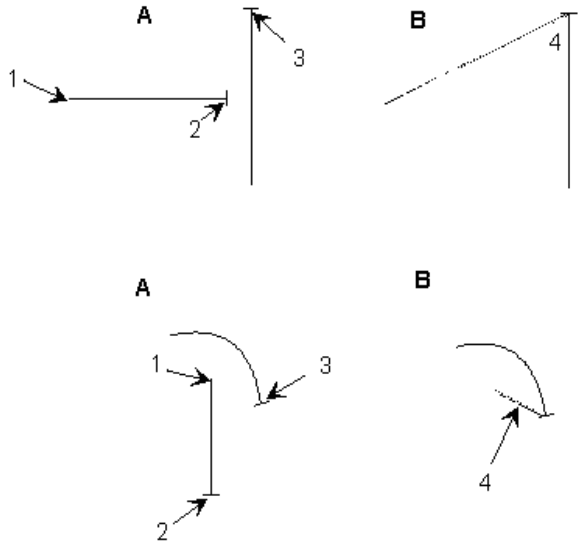
To join line segments:

1. Select the EDIT DRAWING *F4* softkey.
2. Select the JOIN *F2* softkey. The Join option moves a selected line endpoint to the endpoint of a line or arc segment. Always select as the first endpoint the point that will be joined to the other endpoint.

   The endpoint opposite the first selected line segment remains stationary and becomes a pivoting point, as shown in the examples below.

   ⇨ The first endpoint cannot be an arc endpoint.

3. Select the ACCEPT *F2* softkey to accept the join, or the EXIT/CANCEL *F8* softkey to cancel.



A	Original drawing
B	Edited drawing
1	Stationary endpoint
2	Selected segment endpoint
3	Joining endpoint
4	Newly joined segment

Figure 6–11. Joined Lines and Arcs

Modify

To modify line segments:

1. Select the EDIT DRAWING *F4* softkey.
2. Select the MODIFY *F3* softkey.
   - If you select an arc, the DXF Edit Modify - ARC screen appears with these fields and a preview pane:
     - **Start Angle**—defines the starting point of the angle.
     - **Sweep Angle**—defines the total number of degrees in the arc to be cut. This number can be greater than 350.
     - **Direction**—identifies the direction of the arc from the start point.
     - **Center Z**—identifies the Z coordinate for the center point of the arc.
     - **Center X**—identifies the X coordinate for the center point of the arc.
   - If you select a line, the DXF Edit Modify - LINE screen appears with these fields and a preview pane:
     - **Endpoint1 Z**—defines the first endpoint for the Z coordinate.
     - **Endpoint1 X**—defines the first endpoint for the X coordinate.
     - **Endpoint2 Z**—defines the second endpoint for the Z coordinate.
     - **Endpoint2 X**—defines the second endpoint for the X coordinate.
     - **Length**—identifies the line length.
     - **ZX Angle**—identifies the angle of the ZX coordinate.
3. Select the ACCEPT *F2* softkey to accept the modifications, or the EXIT/CANCEL *F8* softkey to cancel.

Split

To split a line segment:

1. Select the EDIT DRAWING *F4* softkey.
2. Select the SPLIT *F4* softkey.
3. Select the line segment.
4. Select the point where the segment will be divided. When a segment is selected for splitting, the midpoint and all intersection points with the other segments are indicated with crosshair markers.
5. Select the ACCEPT *F2* softkey to accept the split, or the EXIT/CANCEL *F8* softkey to cancel.

Delete

To delete a line segment:

1. Select the EDIT DRAWING *F4* softkey.
2. Select the MORE *F6* softkey.
3. Select the DELETE *F1* softkey.
4. Select the line segment to be deleted.
5. Select the ACCEPT *F2* softkey to accept the line deletion or the EXIT/CANCEL *F8* softkey to cancel.

Trim

To trim a line segment:

1. Select the EDIT DRAWING *F4* softkey.
2. Select the MORE *F6* softkey.
3. Select the TRIM F2 softkey.
4. Select the section of the segment to retain
5. Select one of the intersection points where the segment will be trimmed.
6. Select the ACCEPT *F2* softkey to accept the trim or the EXIT/CANCEL *F8* softkey to cancel.

Explode PCurve

To show an exploded view of a PolyCurve:

1. Select the EDIT DRAWING *F4* softkey.
2. Select the MORE *F6* softkey.
3. Select the EXPLODE PCURVE *F3* softkey.
4. Select the PolyCurve.
5. Select the ACCEPT *F2* softkey to accept the explode or the EXIT/CANCEL *F8* softkey to cancel.

NC Productivity Package Option

The NC Productivity Package (NCPP) option provides features that enhance productivity and aid in producing smaller, more powerful, and easier to maintain NC programs. NCPP features include variables, subprogram calls, macros, mathematical equations and address expressions.

⇨ NC files that are larger than dynamic RAM memory can be serially loaded to the hard disk. The CNC can run NC files that do not entirely fit into dynamic RAM memory.

Macro Modes

The CNC software provides compatibility between different NC dialects from various machine tool control manufacturers. The software calls NC macros to be compatible with existing NC macros.

Functions	Subprogram Variables
Local Variables	#1-33
Indirect Variable Referencing	#[#100]
Pass Subprogram Parameters	#1-#33
G Code Status	

Table 7–1. Subprogram Variables

Variables

Variables are used to create programs that can be easily modified. Programs with variables can be reused for various applications. All variables must begin with the "#" character followed by a valid, "writeable" register number and an equal sign.

The example that follows sets the variable value (#500) to 110:

```
# 500 = 110.
```

There are four types of variables that can be used in NC programming: global, system, local, and arguments. Some variables are read only and an error is generated when an attempt is made to write to the variable.

Global Variables

Global variables are general purpose variables that can be used by all programs. Assign a value to the global variable before it is used in an equation or expression, or the variable will be considered vacant. An error message is generated when the system attempts to read a vacant variable.

If the value of a global variable is changed in a program, all other programs can reference that variable with the new value.

Global variables range between #100 to #199 and #500 to #999.

System Variables

System variables are predefined variables that provide information about the state of the system such as X, Y, Z, external work compensation, miscellaneous system parameters, modal information, position information, and G code group status.

Read/Write Restrictions

Read only variables are fixed values. You can change write only variables. Some variables within NCPP are read only (R), some are write only (W), and others are read/write (R/W). Most variables can be used to store either real variables or 32 bit binary values, and the software performs the appropriate conversions when the variables are used within equations. The types of variables are identified as follows: Argument (A), Global (G), Local (L), and System (S).

This table lists the NCPP variable types and read/write restrictions.

Variable Number	Type	Restriction	Variable Number	Type	Restriction
#1 to #33	L	R/W	#4120	S	R
#100 to #199	G	R/W	#4201 to #4221	S	R
#3000	S	R/W	#4222	S	R
#4001 to #4021	S	R	#4302	S	R
#4022	S	R	#4307	S	R
#4102	S	R	#4309	S	R
#4107	S	R	#4311	S	R
#4109	S	R	#4313	S	R
#4111	S	R	#4314	S	R
#4113	S	R	#4315	S	R
#4114	S	R	#4319	S	R
#4115	S	R	#4320	S	R
#4119	S	R			

Table 7–1. NCPP Variable Types and Read Write Restrictions

Addresses with Variables

NC blocks contain addresses with specific numbers. Variables can be used in place of numbers for addresses in the NC blocks, making the program generic. The example below uses variables in the block's address instead of the numbers they represent:

Number	Variable
0.00	#110
-10.00	#115
1.00	#120
0.25	#121
12.00	#122

Address with Variables

G#110 X[#122+.3] Y-[#115/5.] Z[#120 + #121]

Address with Numbers

The same address would be written as follows if numbers were used instead of variables:

G0.00 X[12.00 + 0.3] Y-[10.00/5] Z[1.00 + 0.25]

–Or–

G0.00 X12.3 Y2 Z1.25

Alarm 3000 Messages

Variable #3000 writes an Alarm 3000 error message to the screen. The following is an example of this type of error message:

#3000 = 140 (ARGUMENT MISSING)

The right-hand side of the equation must begin with a number in the range of 0 to 200 followed by a left parenthesis, a string which is limited to 26 characters, and a right parenthesis. This number is added to 500 and stored to variable #3000. The message "ARGUMENT MISSING" is displayed on the screen.

Vacant Variables

A variable is considered vacant if a local or global variable has not been assigned a value before it is used in an equation or expression. An error message occurs with vacant variables.

A variable can be tested to determine if it is vacant by comparing it with the null variable #0. The variable #0 is called the “null variable” because it cannot be used to store a value and is only used to perform vacant variable tests.

For example, the following IF conditional statement is true if variable #510 is vacant and false if the variable is not vacant. (Refer to the “IF Statements” section of this chapter for information about IF statements.)

```
IF[#510 EQ #0] GOTO 100
```

⇨ The function NE (not equal) can also be used with vacant variables.

It is best to avoid using vacant variables in equations. However, when it is necessary to use them to maintain compatibility with existing programs, vacant variables can be used in some circumstances without receiving an error message.

⇨ If a variable contains a value of 0.0000 or any number, it is not vacant.

The following table shows what happens when vacant variables are used in equations versus setting variables to zero. This table shows the difference between using vacant variables and setting variables to 0 in equations:

Comparison of Vacant Variables and Setting Variables to Zero (0)			
Function	**Examples**	**Null/Vacant Variable (#10 = <vacant>)**	**Variable Set to 0 (#10 = 0)**
Assignment	#20 = #10	Error Message	#20 = 0
Multiplication	#20 = #10 * 3	#20 = 0	#20 = 0
	#20 = #10 * #10	Error Message	#20 = 0
	#20 = #10 * #0	Error Message	Error Message
	#20 = #0 * 3	Error Message	-
	#20 = #0 * #0	Error Message	-
Addition	#20 = #10 + 3	#20 = 0	#20 = 0
	#20 = #10 + #10	Error Message	#20 = 0
	#20 = #10 + #0	Error Message	Error Message
	#20 = #0 + 3	Error Message	-
	#20 = #0 + #0	Error Message	-
EQ (equal)	#10 EQ #0	True	False
	#10 EQ 0	Error Message	True
NE (not equal)	#10 NE #0	False	True
	#10 NE 0	Error Message	False
GE (greater than or equal to)	#10 GE #0	True	False
	#10 GE 0	Error Message	True
GT (greater than)	#10 GT #0	Error Message	Error Message
	#10 GT 0	Error Message	False
Other Functions	-	Error Message	Depends on Function

Table 7–2. Comparison of Vacant Variables and Setting Variables to Zero (0)

Variable Expressions

Instead of using a number after an NC parameter, a variable expression (or math expression) can be used.

- The “[”and the “]” characters serve as delimiters in the expressions.
- A negative sign entered before the left bracket ([) indicates that the expression is negative (i.e. X-[[#110+3.4] + 4.5]).

Expression Symbols and Keywords

Various keywords and symbols can be used in the expressions. At least two letters of the keyword are required: RO, ROU, ROUN, and ROUND perform the same function. The software checks spelling. RUON is not a valid abbreviation for ROUND, but ROUN is acceptable.

Symbol	Description	Example
+	Addition	#500 = #600 + 2.3
-	Subtraction	#500 = #600 - 2.3
/	Division	#500 = #600 / 2.3
*	Multiplication	#500 = #600 * 2.3
^	Power (i.e. 2^2, 2 to the 2nd power, or 4)	#500 = 4.5 ^ 2 #500 will be set to 20.25.

Table 7–3. NC Expression Symbols

The keywords are described and examples are provided in the following table:

Operation Keyword	Description	Example
ABS	Absolute Value	#500 = ABS [-#550]
ACOS	Arc or Inverse Cosine function	#500 = ACOS [#540]
AND	Logical AND	#500 = #600 AND 48
ASIN	Arc or Inverse Sine function	#500 = ASIN [#540]
ATAN	Arc Tangent (degrees)	#500 = ATAN [.34]
BCD	Convert Binary to BCD Format	#500 = BCD [#600]
BIN	Convert BCD to Binary Format	#500 = BIN [#600]
COS	Cosine (degrees)	#500 = COS [45.3]
DEGREES	Converts radians to degrees	#500 = DEGREES [5.437] #500 will be set to 311.52 degrees.
EQ	Equal	#500 = #510 EQ 3.4 #500 will be set to 0 if false; 1 if true.
EXP	Exponential function	#500 = EXP [3.67] #500 will be set to 39.252.
FIX	Discards fractions less than 1	#500 = FIX [45.2375] #500 will be set to 45
FUP	Adds 1 for fractions less than 1	#500 = FUP [45.2375] #500 will be set to 46
GE	Greater Than Or Equal To	#500 = #510 GE 3.4 #500 will be set to 0 if false; 1 if true.
GT	Greater Than	#500 = #510 GT 3.4 #500 will be set to 0 if false; 1 if true.
HSIN	Hyperbolic Sine function	#500 = HSIN[#540]
HCOS	Hyperbolic Cosine function	#500 = HCOS [#540]
INVERSE	Binary Inverse function	#500 = [7 AND [INV[3]]] #500 will be set to 4.0.
LE	Less Than or Equal To	#500 = #510 LE 3.4 #500 will be set to 0 if false; 1 if true.
LN	Natural Logarithmic function	#500 = LN [24.89] #500 will be set to 3.2144.
LOG	Logarithmic function	#500 = LOG [345.89] #500 will be set to 2.5389.
LT	Less Than	#500 = #510 LT 3.4 #500 will be set to 0 if false; 1 if true.
MOD	Modulus operator	#500 = 19 MOD 6.7 Will return a value of 5.6
NE	Not Equal	#500 = #510 NE 3.4 #500 will be set to 0 if false; 1 if true.

Operation Keyword	Description	Example
OR	Logical OR	#500 = 41 OR 4
RADIANS	Converts degrees to radians	#500 = RADIANS [270.34] #500 will be set to 4.718 radians.
ROUND	Rounds off	#500 = ROUND [34.56 result is 35]
SIN	Sine (degrees)	#500 = SIN [#610]
SQRT	Square Root	#500 = SQRT [#540]
TAN	Tangent (degrees)	#500 = TAN [32.4]
XOR	Logical Exclusive OR	#500 = #560 XOR 34

Table 7–4. NC Expression Keywords

The software automatically converts real numbers to hexadecimal format before performing logical operations. The Operation Keyword “AND” does not function on real numbers. As shown below, the #500 value is truncated to 32 and the #550 value is truncated to 48. When the “AND” function is performed, the truncated numbers are stored in variable #560.

- #500 = 32.456
- #550 = 48.98
- #560 = [#500 AND #550]

These examples are valid variable expressions:

- G01 X#140 Y [#500 + 2.] Z[#550 * [SIN [#130 + 23.5]]]
- G02 Z [2.3 / [SIN 43]] Y[2 ^ 3] G20 M25
- X [ROUN[3.45 * COS[#520]]]
- R [SQRT[[#510 ^ 2] + [#511 ^ 2]]]
- G01 X-#510 Y-[#520 + 4.5] Z4

Operation Priorities

The interpreter gives operations within the expression a certain priority in order to determine how the expression is evaluated. This is a listing of the priorities:

Priority	Operation	
Highest	Functions	
Second	Symbols	Power (^)
Third	Multiplication (*) Division (/)	
Lowest	Addition (+) Subtraction (-)	

Table 7–5. Numerical Operations Priorities

Even though the interpreter assumes this priority, in order to make the NC program more understandable and more maintainable, use brackets to divide the expressions. For example, G01 X[34.5+23.4 / 32] should be rewritten as G01 X[34.5 + [23.4/32]]. Using spacing within an expression can also make the expression more readable. Decimal points and leading or trailing zeros are not required with the numbers.

Indirect Variables

Variables can be referenced indirectly by using multiple levels of pound signs (#) and brackets ([and]).

#100 = 600 ⇐ #100 is equal to 600.

#600 = 4.5 ⇐ #600 is equal to 4.5.

#[#100] = 4.5 ⇐ #[#100] is equal to #600; #600 equals 4.5.

Saving Variable Values To a File on the Control

When running the program on the CNC, if an error occurs during the program run, the variable values are not saved. The variable values are saved if the program runs successfully.

Variable Example

This program illustrates the use of #0 in an IF statement to determine if an argument is passed to subprogram 3100. There are two IF statements in sequence numbers 100 and 200 in the subprogram which test to verify that the calling program (0100) had passed parameters I and J which correspond to #4 and #7 in subprogram 3100, respectively. If either variable #4 or #7 is vacant, an Alarm 3000 error message is written to the screen. (Refer to the “Program Control Statements” section for more information about IF statements.) View the part using the **Draw** console key to verify that the part is programmed correctly.

```
ISNC Part Program                    1                                Inch
TRU_CRC.FNC
 %
 O0100                                  ⇐ Calling Program—0100—Start
 T0101
 S1500 M03
 G00 G90 X5.0 Y5.0
 G43 Z.1 H01
 M08
 G01 Z-.5 F5.0
 G65 P3100 I.5 D2 F15.0
 G00 Z.1 M09
 G91 G28 Z0 M05
 M30                                    ⇐ End of Program—0100
 :3100(True CIRCLE TYPE 1)              ⇐ Subprogram—3100—Start
 #27 = #4001
 #28 = #4003
 #29 = #4107
 N100 IF[#4EQ#0] GOTO 1000              ⇐ Vacant Variable Check
 N200 IF[#7EQ#0] GOTO 1000
 #1 = ABS [#4]-ABS [#[2000+#7]]
 IF [#1LE0] GOTO 2
 #20 = #1/2
 #21 = ROUND [#20*1000]
 #22 = #21/1000
 #2 = #1-#22
 #3 = #1-#2
 IF [#23EQ#0] GOTO 10
 G01 G91 X-#2 Y-#3 F#9
 G17 G02 X-#3 Y#3 J#3
 I#1
 X#3 Y#3 I#3
 G01 X#2 Y-#3 F[#9*3]
 GOTO 5
 N10 G01 G91 X-#2 Y#3 F#9
 G17 G03 X-#3 Y-#3 J-#3
 I#1 J0
 X#3 Y-#3 I#3
 G01 X#2 Y#3 F[#9*3]
 GOTO 5
 N1000 #3000 = 100(ARGUMENT MISSING)    ⇐ Alarm Message
 N5 G#27 G#28 D#29
 M99                                    ⇐ M99 is end of Subprogram 3100
```

Program Control Statements

Program control statements are NC blocks which direct the flow of the NC program or subprogram. The following section describes using the different NCPP option's program control statements.

Program control statements use keywords: GOTO, IF, WHILE, and DO. At least two letters of the keyword are required. For example, WH, WHI, WHIL, and WHILE all perform the same function. Some program control statements are only effective within the current program or subprogram, and other program control statements cause program execution to go to subprograms. The software can only locate sequence numbers that are in memory.

The following program control statements are effective only within the *current program* being executed:

- WHILE [conditional expression] DO#
- DO#
- IF [conditional expression] GOTO [expression or #]
- GOTO [expression or #]
- END#
- M99 or M99 P____

These program control statements cause program execution to *call subprograms*:

- M98 P____
- G65 P____ L____ [Optional Argument List]
- G66 P____ L____ [Optional Argument List]

Variables can be *referenced indirectly* to initialize a large block of variables, for example:

- #100 = 500
- WHILE [#100 LT 1000] DO 250
- #[#100] = 1.5
- #100 = #100+1
- END 250

The alternative to indirectly referencing variables is to have a program line for each variable as shown below:

- #500 = 1.5
- #501 = 1.5
- …
- #999 = 1.5

In this case, 500 program lines would be required to perform what five program lines accomplished in the first example.

GOTO Statements

GOTO statements jump the program to a specific number in the program. Any valid address expression can be used in place of a sequence number after the GOTO. Fractions are truncated. For example, GOTO 3.45 and GOTO 3 work the same. The program cannot locate sequence numbers that are not in memory. If the search reaches the end of the program without finding the sequence number, the software generates an error message.

Positive GOTO Statement

If the resultant value is positive, the software searches for the sequence number from the point of the GOTO to the end of the program. Then it proceeds to the beginning of the program and searches for the sequence number until reaching the starting point (GOTO statement).

Negative GOTO Statement

If the resultant value of the expression is negative, the search begins at the beginning of the program.

IF Statements

IF statements contain a conditional expression and a GOTO statement. The expression which follows the GOTO must result in a valid sequence number; otherwise, an error message is generated. The program cannot locate sequence numbers that are not in memory. The following line illustrates an IF statement's components:

- IF [conditional expression] GOTO [expression or #]
- If the conditional expression has a value of 1, it is true, and the GOTO is performed.
- If the conditional expression has a value of 0, it is false, and the next NC block is executed.
- If the conditional expression has a value other than 0 or 1, it is invalid.

These are examples of conditional expressions used in IF statements:

```
IF[[[#100 LT 2.3] OR [#320LE7.34]] AND [#400LT3.4]] GOTO#340
IF[#150 EQ 2] GOTO 10
IF[#750 GT 2.34] GOTO [[#550+23]/40]
```

WHILE Loops

WHILE loops contain a conditional expression and a DO statement. This is a sample WHILE loop:

- WHILE [conditional expression] DO number
- NC block
- NC block
- NC block
- END number

The blocks between the WHILE statement and the END statement are repeated as long as the conditional expression is true. The following are other details about WHILE loops:

- A WHILE loop must have a matching END statement within the same program.
- The DO must match the number following END and must be an integer in the range of 1 to 255.
- The program cannot locate sequence numbers that are not in memory.
- No other NC commands can be contained on the same lines as the WHILE or END statements.
- If the WHILE conditional expression is false, the program continues execution with the NC block which follows the END statement.
- DO loops operate the same as WHILE loops with a conditional expression which is always true.
- The DO statement can also be used by itself without the WHILE conditional statement.

⇨ To exit an infinite WHILE loop while the program is being drawn, press the console Draw key.

DO Loops

DO loops operate the same as WHILE loops with a conditional expression which is always true. The DO statement can also be used by itself without the WHILE conditional statement. The following are some additional details about DO loops:

- DO loops must contain a matching END statement within the same program.
- The numbers following DO and END must match and must be an integer in the range of 1 to 255.
- The program cannot locate sequence numbers not in memory.
- No other NC commands can be contained on the same lines as the DO or END statements.

The following is a sample DO loop:

- DO number
- NC block
- NC block
- NC block
- END number

The blocks between the DO statement and the END statement are repeated continuously in an infinite loop unless one of the following events occurs:

- The program exits the loop with a GOTO or M99 P ____ jump statement.
- The program execution is terminated with an M02 or M30.
- The right mouse button is pressed. The right mouse button acts as a graphics reset.

⇨ To exit an infinite DO loop while the program is being drawn, press the console Draw key.

Stop Program Execution

The M02 (End of Program) and M30 (End Program) program control statements stop program execution. The following examples of program control statements are used correctly:

Nested WHILE Loops	Branch Outside WHILE Loop	Subprogram Call from Inside WHILE Loop	Reuse of DO-END Pairing Number
WHILE[...] DO 100 NC blocks **WHILE[...] DO 200** NC blocks **WHILE[...] DO 250** NC blocks **END 250** NC blocks **END 200** NC blocks **END 100**	**WHILE[...] DO 200** NC blocks **GOTO 3535** NC blocks **END 200** NC blocks **N3535**	**WHILE[...] DO 150** NC blocks **M98 P3000** NC blocks **END 150** NC blocks **WHILE[...] DO 200** NC blocks **G65 P3000** NC blocks **END 200** NC blocks	**WHILE[...] DO 100** NC blocks **END 100** NC blocks **WHILE[...] DO 100** NC blocks **END 100** NC blocks **WHILE[...] DO 100** NC blocks **END 100**

Table 7–6. Correct Program Control Statement Examples

These examples show **incorrect** use of program control statements:

Incorrectly Nested WHILE Loops	Branch Into a WHILE Loop	Improper Reuse of DO-END Pairing Number
WHILE[...] DO 100 NC blocks **WHILE[...] DO 200** NC blocks **WHILE[...] DO 250** NC blocks **END 100** NC blocks **END 200** NC blocks **END 250**	**GOTO 3535** *NC blocks* **WHILE[...] DO 200** NC blocks **N3535** NC blocks **END 200**	**WHILE[...] DO 100** NC blocks **WHILE[...] DO 100** NC blocks **END 100** NC blocks **END 100** NC blocks

Table 7–7. Incorrect Program Control Statement Examples

Subprograms

Subprograms are stand-alone NC programs that can be called from another NC program. Subprograms begin with the letter “O” or the “:” character followed by a four-digit number that identifies the subprogram. Each subprogram ends with an *M99* statement. The only limitation for the number of NC files and subprograms the software can load is the amount of available dynamic RAM memory.

The following is a sample subprogram:

```
N10     O7162        ⇐ begins with “O” followed by 4-digit number
N20     G00 G90
N40     X0 Y0
N50     T1
N60     Z5.
N70     S2000 M03
N80     Z0.05
N90     M99          ⇐ ends with M99
```

Subprograms can be nested 50 levels deep. In general, different types of subprogram calls can be used in various combinations. There are some restrictions in the use of modal subprograms, however, which will be described in more detail later.

Programs cannot call themselves as subprograms because the repetition exhausts the 50 levels of subprogram nesting. For the same reason, a user defined code cannot be used in a program which is associated with the same user defined code. For example, a G65 P5000 command is illegal within the program 5000.

G65 Subprogram Call

The G65 subprogram command has the following form:

G65 P____ L_____ [followed by optional arguments]

The P represents the subprogram number and the L represents the number of iterations that the subprogram must perform. These two methods of argument passing can be used together:

Arguments

In a G65 subprogram call, the local variables in the calling program are not copied to the local variables in the called subprogram. Arguments which follow the G65 command are copied to the local variables in the subprogram as illustrated in the following command:

G65 P5080 A0.0 B8 C2.3 S6 T2 H81 I9 J3.5 K0 Z-1 R.1

The value which follows A is copied to the local variable #1 in the subprogram. The table on the following page shows the relationships between the subprogram arguments and the local variables in the subprograms.

Multiple Arguments

Multiple I, J, and K arguments can also be used as subprogram arguments. For example, if three I arguments are used in the subprogram call, the first I maps to the #4 variable, the second I maps to the #7 variable, and the third I maps to the #10 variable. The following subprogram call is legitimate:

G65 P2000 A2.3 B3.2 I2.0 J3. K5.4 I3. I5. J2. I6. W3. U3

Only numbers may be used as arguments in a G65 subprogram call; no variables or expressions can be used. If multiple iterations of the subprogram are to be performed, the local variables will be initialized to the same argument values.

Passing Argument Lists to Subprograms

There are several methods for passing arguments and parameters to subprograms. The G65 and G66 subprogram calls allow an argument list to be provided after the G65 and G66, respectively. The argument list consists of various letters followed by values. The values are then stored as local variables within the subprogram.

The table below lists the correspondence between the arguments and the local variables. The argument list is optional. Any arguments which are not included in the list are given vacant status.

Local Variables	Subprogram Arguments	Local Variable	Subprogram Arguments
#1	Argument A	#18	Argument R or K5
#2	Argument B	#19	Argument S or I6
#3	Argument C	#20	Argument T or J6
#4	Argument I or I1	#21	Argument U or K6
#5	Argument J or J1	#22	Argument V or I7
#6	Argument K or K1	#23	Argument W or J7
#7	Argument D or I2	#24	Argument X or K7
#8	Argument E or J2	#25	Argument Y or I8
#9	Argument F or K2	#26	Argument Z or J8
#10	Argument I3	#27	Argument K8
#11	Argument H or J3	#28	Argument I9
#12	Argument K3	#29	Argument J9
#13	Argument M or I4	#30	Argument K9
#14	Argument J4	#31	Argument I10
#15	Argument K4	#32	Argument J10
#16	Argument I5	#33	Argument K10
#17	Argument Q or J5		

Table 7–8. Local Variables and Subprogram Arguments

Layering of Local Variables within Subprogram Calls

M98 subprogram calls use local variables differently from other subprogram calls since the called subprogram does not get a new set of local variables. Changes made to the local variables within the current subprogram will be retained when the calling program is re-instated.

Changes can be made to the local variables within the current subprogram, but when program execution returns to the calling program, the values of the local variables of the calling program are reinstated. The local variables in the subprogram can be changed, however, without affecting the local variables in the calling program. With other subprogram calls, unless an argument list is passed to the subprogram, the local variables are given vacant status.

Specifying Subprogram Iterations

The number of iterations for a subprogram to perform are specified with G65, G66, and M98 subprogram calls.

Using G65 and G66

When making G65 and G66 subprogram calls, the L parameter is used to specify iterations. The maximum number of iterations which can be specified with the G65 and G66 subprogram calls is 999.

Using M98

When making M98 subprogram calls, the P parameter is used to specify iterations as well as the subprogram number. Up to four digits can be used to specify iterations for a maximum of 9999 iterations. Leading zeros are not required when specifying iterations; however, leading zeros are required with a subprogram number that is less than 1000.

G65 Subprogram Example

In the following example a line of holes will be drilled along a line. The type of canned cycle can be determined along with the distance between the holes in both the X and Y axes. The type of canned cycle and various canned cycle parameters can also be set. View the part using the Draw console key to verify that the part is programmed correctly.

ISNC Part Program 1 Inch
BOLT_LN.FNC

```
O4000
T1
M03 G00 G90 X0 Y0 Z0 S1800
/
(B REPRESENTS THE NUMBER OF BOLT HOLES)
(H REPRESENTS THE DESIRED CANNED CYCLE)
(X,Y REPRESENT THE INCREMENTAL DISTANCE BETWEEN HOLES)
(Z REPRESENTS THE HOLE DEPTH)
(R REPRESENTS THE R PLANE LEVEL)
/
#500=99 (RETURN TO R LEVEL)
G65 P5070 X.5 Y.75 B10 H81 Z-1 Q0. R.1 F20.
M30
(************************************************)
(    BOLT HOLE LINE PATTERN  - SUBPROGRAM 5070    )
(************************************************)
O5070
(#2 IS THE NUMBER OF HOLES)
(#11 IS THE CANNED CYCLE NUMBER)
(#26 IS THE HOLE DEPTH)
(#500 IS 99 FOR RETURN TO R LEVEL MODE OR 98 FOR RETURN TO INITIAL POINT)
(#5003 IS THE Z COORDINATE BEFORE THE CANNED CYCLE IS PERFORMED)
/
WHILE [#2GT0] DO250
G91 G#500 G#11 Z#26 Q#17 R[#5003-#18] F#9
G00 X#24 Y#25
#2 = #2-1
END250
M99
```

Macro Instruction (G65)

G65 Macro instructions are G65 commands which are used to perform mathematical, trigonometric, or program control functions instead of subprogram calls. These commands are intended to support existing programs which use this program format.

The value in the H parameter defines the operation being performed. In all instructions except the GOTO commands H80 through H86, a variable number follows the P parameter. The operation's result is stored in that variable number. In the following command the value stored in variable #100 is added to the number 1 and the resultant value is stored in variable #115.

G65 H02 P#115 Q#100 R1

For the GOTO commands, the value which follows the P is a positive or negative integer. If the number is negative, the software begins searching for the sequence number at the beginning of the file and continues to search for the sequence number until reaching the end of the file. If the number is positive, the search for the sequence numbers begins with the block after the GOTO command and continues until reaching the end of the file. The software then searches from the beginning of the file until reaching the GOTO command block.

The values which follow Q and R are general purpose parameters which are used in mathematical, logical, or GOTO operations. The specific operations are listed in the following table.

Format

The following is the G65 Macro Instruction format:

G65 H ____, P #a__, Q #b__, R #c__,.

The table below lists the Descriptions and Instruction Functions for the H codes used in the G65 macro instructions:

H Code	Description	Instruction Function
H01	Definition, Substitution	#a = #b
H02	Addition	#a = #b + #c
H03	Subtraction	#a = #b - #c
H04	Product	#a = #b * #c
H05	Division	#a = #b / #c
H11	Logical Sum	#a = #b .OR. #c
H12	Logical Product	#a = #b .AND. #c
H13	Exclusive OR	#a = #b .XOR. #c
H21	Square Root	#a = √#b
H22	Absolute Value	#a = |#b|
H23	Remainder	#a = #b - trunc (#b/#c) * #c trunc: discard fractions less than 1.
H24	Conversion from BCD to Binary	#a = BIN(#b)
H25	Conversion from binary to BCD	#a = BCD(#b)
H26	Combined Multiplication/Division	#a = (#a * #b) / #c
H27	Combined Square Root 1	#a = √(#b2 + #c2)
H28	Combined Square Root 2	#a = √(#b2 - #c2)
H31	Sine	#a = #b * SIN(#c)
H32	Cosine	#a = #b * COS (#c)
H33	Tangent	#a = #b * TAN(#c)
H34	Arc tangent	#a = TAN(#b/#c)
H80	Unconditional Divergence (GOTO)	GOTO a
H81	If Statement, Equals	IF #b = #c, GOTO a
H82	If Statement, Not Equal	IF #b ≠ #c, GOTO a
H83	If Statement, Greater Than	IF #b > #c, GOTO a
H84	If Statement, Less Than	IF #b < #c, GOTO a
H85	If Statement, Greater Than/Equals	IF #b >= #c, GOTO a
H86	If Statement, Less Than/Equals	IF #b <= #c, GO TO a

Table 7–9. H Code Descriptions and Instruction Functions for G65 Macro Instructions

For H80 through H86, if “a” has a negative value, the software performs a GOTO but begins looking for the sequence number at the beginning of the program. No variables can be used for the P parameter for H80 through H86.

For example use #100 = 4.56 OR #110 instead of G65 #11 P#100 Q4.56 R#110.

⇨ And use IF [#150 EQ #160] GOTO 100 instead of G65 H81 P100 Q#150 R#160.

These commands can be used in either Macro A or B mode.

Example

The following example shows how to use G65 macro instructions in a Bolt Hole Circle subprogram. View the part using the Draw console key to verify that the part is programmed correctly.

ISNC Part Program 1 Inch
G65INST.FNC

```
%
(#600 IS BOLT HOLE CIRCLE X COORD)
(#601 IS BOLT HOLE CIRCLE Y COORD)
(#602 IS BOLT HOLE CIRCLE RADIUS)
(#603 IS STARTING ANGLE)
(#604 IS NUMBER OF BOLT HOLES)
#600 = 0
#601 = 0
#602 = 2.
#603 = 30.
#604 = 12.
T0101
M3 S300
G17
G00 X0 Y0 Z0.05
M98 P3030
G00 X0 Y0 Z0.05
M02
O3030
(#110 IS BOLT HOLE COUNTER)
(#112 IS ANGLE OF CURRENT HOLE)
(#113 IS X COORD OF CURRENT HOLE)
(#114 IS Y COORD OF CURRENT HOLE)
N10 G65 H01 P#110 Q0
G65 H22 P#111 Q#604
N20 G65 H04 P#112 Q#110 R360
G65 H05 P#112 Q#112 R#604
```

ISNC Part Program 2 Inch
G65INST.FNC

```
G65 H02 P#112 Q#603 R#112
G65 H32 P#113 Q#602 R#112
G65 H02 P#113 Q#600 R#113
G65 H31 P#114 Q#602 R#112
G65 H02 X#114 Q#601 R#114
G90 H00 X#113 Y#114
G81 Z-1. F20.
G80
G65 H02 P#110 Q#110 R1
G65 H84 P-20 Q#110 R#111
M99
```

Modal Subprograms

Modal subprograms are executed every time a motion is performed (i.e., after a Move command). Use them for performing repetitive tasks at different locations. The repetitive tasks can be put inside a modal subprogram. A subprogram call can be made to a program which contains X, Y, and Z locations and will be executed at each of these locations.

A modal subprogram will not be modal within another modal subprogram. If the modal subprogram contains Move commands, the modal subprogram will not be performed after Move commands within the modal subprogram. This allows Move commands to be contained within modal subprograms.

These methods allow the subprogram call to be modal:

- A Modal Subprogram Call (G66) Command
- A Modal user defined G code

Modal Subprogram Call (G66)

In a G66 Modal subprogram call, the subprogram is repeatedly executed after each Move command until the Modal Subprogram Call Cancel (G67) command is performed.

Modal Subprogram Cancel (G67)

The G67 command is used to cancel modal subprograms initiated with either the G66 or with a modal user defined G code.

Modal Subprogram Call (G66) Example

The following program draws a series of squares and rectangles. View the part using the Draw console key to verify that the part is programmed correctly.

NC Part Program 1 Inch
G66.FNC

```
%
(EXAMPLE OF MODAL SUBPROGRAM CALL G66)
(P6010 IS USED AS MODAL SUBPROGRAM)
(THE VALUES AFTER I AND J ARE PASSED TO)
(THE SUBPROGRAM. THE SUBPROGRAM IS ONLY)
(EXECUTED AFTER BLOCKS WITH MOVE COMMANDS.)
X0 Y0 Z0
X5 G66 P6010 I1. J1.5
Y-3
X-5
Y0
(MODAL SUBPROGRAM IS NOW CANCELED WITH G67)
G67
Y3
(THE MODAL SUBPROGRAM IS STARTED AGAIN WITH)
(NEW PARAMETERS.)
X0 G66 P6010 I3.J1.
Y0
Y-2
M02
:6010
(THIS SUBPROGRAM CREATES A SIMPLE BOX SHAPE.)
G91
X#4
Y#5
X-#4
Y-#5
G90
M99
```

NCPP Variable Summary

In the tables below, the Type column indicates the type of variable: Argument (A), System (S), Common (C), and Local (L). The R/W column indicates whether the variable is Read or Read/Write.

Variable Number	Type	R/W	Local Variables (Note 3)
#1	A	R/W	Address A (Note 4)
#2	A	R/W	Address B (Note 4)
#3	A	R/W	Address C (Note 4)
#4	A	R/W	Address I (Note 1)or I1 (Note 2)
#5	A	R/W	Address J (Note 1) or J1 (Note 2)
#6	A	R/W	Address K (Note 1) or K1(Note 2)
#7	A	R/W	Address D (Note 1) or I2 (Note 2)
#8	A	R/W	Address E(Note 1) or J2 (Note 2)
#9	A	R/W	Address F (Note 1) or K2 (Note 2)
#10	A	R/W	Address I3 (Note 2)
#11	A	R/W	Address H (Note 1) or J3 (Note 2)
#12	A	R/W	Address K3 (Note 2)
#13	A	R/W	Address M (Note 1)or I4 (Note 2)
#14	A	R/W	Address J4 (Note 2)
#15	A	R/W	Address K4 (Note 2)
#16	A	R/W	Address I5 (Note 2)
#17	A	R/W	Address Q (Note 1) or J5 (Note 2)
#18	A	R/W	Address R (Note 1) or K5 (Note 2)
#19	A	R/W	Address S (Note 1) or I6 (Note 2)
#20	A	R/W	Address T (Note 1) or J6 (Note 2)
#21	A	R/W	Address U (Note 1) or K6 (Note 2)
#22	A	R/W	Address V (Note 1) or I7 (Note 2)
#23	A	R/W	Address W (Note 1) or J7 (Note 2)
#24	A	R/W	Address X (Note 1) or K7 (Note 2)
#25	A	R/W	Address Y (Note 1) or I8 (Note 2)
#26	A	R/W	Address Z (Note 1) or J8 (Note 2)
#27	A	R/W	Address K8(Note 2)
#28	A	R/W	Address I9 (Note 2)
#29	A	R/W	Address J9 (Note 2)
#30	A	R/W	Address K9(Note 2)
#31	A	R/W	Address I10 (Note 2)
#32	A	R/W	Address J10 (Note 2)
#33	A	R/W	Address K2 (Note 2)

Table 7–10. NCPP Local Argument Variables (#1 - #33

1. Valid for argument assignment method 1 where multiple sets of (I,J,K) are not used.
2. Valid for argument assignment method 2 where multiple sets of (I,J,K) are used.
3. Local variables are used to pass arguments to a macro. If a local variable without a transferred argument is vacant in its initial status, it can be used freely in the macro.
4. Valid for argument assignment method 1 and 2

Variable Number #100 to #199 #500 to #999	Type	R/W	Common Variables
#100 to #199	C	R/W	Use these variables to store binary numbers as well as real numbers. All programs and subprograms can read and write to them. Variables #146, #147, and #149 also store the values which follow the B, S, and T code when B, S, and T subprogram calls are performed.
#500 to #999	C	R/W	Use these variables to store binary numbers as well as real numbers. All programs and subprograms can read and write to them.

Table 7–11. NCPP Common Variables (#100 - #199 and #500 - #599)

There are 100 variables each reserved for tool type, tool diameter, tool calibration, probe calibration, tool probe offset X, and tool probe offset Y regardless of whether or not the machine can handle that many tools. If the program tries to access a variable for a tool that does not exist, an error is generated. The variables for tool type (#3201 – #3300) have these values:

Variable Number #4001 to #4120	Type	R/W	Modal Information from Previous Block
#4001 to #4022	S	R	G Code Groups 1 to 22
#4102	S	R	B Code
#4107	S	R	D Code
#4109	S	R	F Code
#4111	S	R	H Code
#4113	S	R	M Code
#4114	S	R	Sequence Number of previous block
#4115	S	R	Program Number of previous block
#4119	S	R	S Code
#4120	S	R	T Code

Table 7–12. Modal Information from Previous Block Variables (#4001 - #4120)

Variable Number #4201 to #4210	Type	R/W	Modal Information for Current Block
#4201 to #4222	S	R	G Code Groups 1 to 22
#4302	S	R	B Code
#4307	S	R	D Code
#4309	S	R	F Code
#4311	S	R	H Code
#4313	S	R	M Code
#4314	S	R	Sequence Number of current block
#4315	S	R	Program Number of current block
#4319	S	R	S Code
#4320	S	R	T Code

Table 7–13. Modal Information for Current Block Variables (#4201 - #4320)

OIL SKIMMER

The optional Oil Skimmer removes oil from the coolant, prolonging the coolant usage.

Oil Skimmer Operation

To operate the oil skimmer in Manual mode,

1. Press the **Manual** console key to access the Manual screen.
2. Select the ACCESSORY OPERATIONS *F4* softkey.
3. Select the MORE F7 softkey.
4. Select the OIL SKIMMER *F1* softkey. The softkey menu contains these choices:

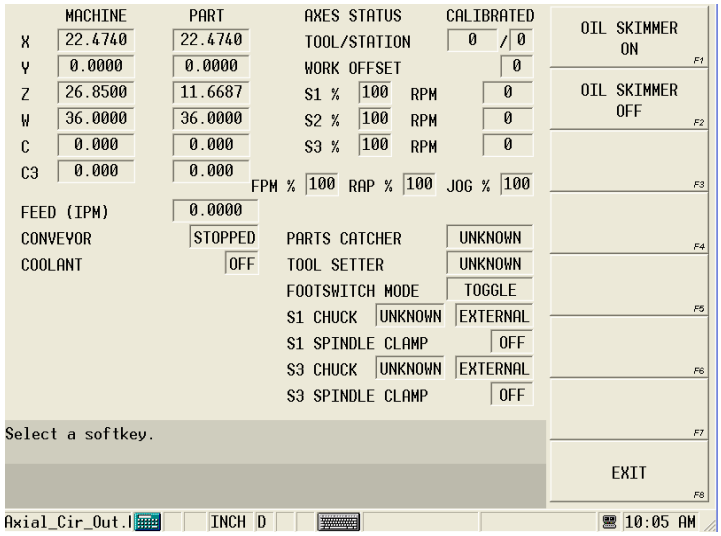


Figure 8–1. Manual screen Oil Skimmer softkey menu

The Oil Skimmer softkeys perform these functions:

- **OIL SKIMMER ON** *F1*—turns on the Oil Skimmer.
- **OIL SKIMMER OFF** *F2*—turns off the Oil Skimmer.

Automatic Operation

The oil skimmer operates automatically when the machine power is on and the coolant pump is turned off.

You can turn the oil skimmer off and back on while machine power is on using the Manual screen, Accessory Operations, Oil Skimmer menu (Oil Skimmer Off F2 and Oil Skimmer On F1).

PARTS CATCHER

The optional parts catcher is an automated pneumatic system that rotates an arm upward to catch the part that has been cut off and dispenses the part into a receptacle. The receptacle can be accessed so you can take the part out while the next part is being cut.

The Parts Catcher can be operated either manually with softkeys on the Manual screen, automatically using M Codes, or with a Cutoff Block. To program an M Code in a conversational programming data block, please refer to WinMax Lathe Conversational Part Programming *Machine Function—M Code, on page 2 - 198* and Winmax Lathe NC Programming, *Basic NC M Codes, on page 3 - 1* or *ISNC M Codes, on page 5 - 1*.

⚠ Refer to the parts catcher's operator's manual for more information about operating and maintaining the parts catcher.

Parts Catcher Operation

The parts catcher can be operated manually or programed to operate within a Cutoff Block.

Manual Operation

Advance and retract the parts catcher using the softkey selections from the Manual screen.

1. Press the **Manual** console key to access the Manual screen.
2. Select ACCESSORY OPERATIONS *F4*.
3. Select PARTS CATCHER *F3*. The softkey menu contains these choices:
   - Select ADVANCE PARTS CATCHER *F3* to advance the parts catcher. ADVANCED appears in the Part Catcher field.
   - Select RETRACT PARTS CATCHER *F4* to retract the parts catcher. RETRACTED appears in the Parts Catcher field.

Automatic Operation

To operate the Parts Catcher automatically, program a Machine Function data block within a conversational part program. From the New Block screen or Program Review screen,

1. Select the Insert Block Before *F7* softkey.
2. Select the Miscellaneous softkey.

3. Select the M Code *F3* softkey.
   - Select M58 to turn on the Parts Conveyor and enter the number of seconds to dwell in the Dwell field before moving to the next block.
   - Select M59 to turn off the Parts Conveyor and enter the number of seconds to dwell in the Dwell field before moving to the next block.

⇨ If using NC programming, the M58 and M59 codes apply.

Automatic Operation with Cutoff Block

You can also program the parts catcher to automatically advance during a Cutoff Block.

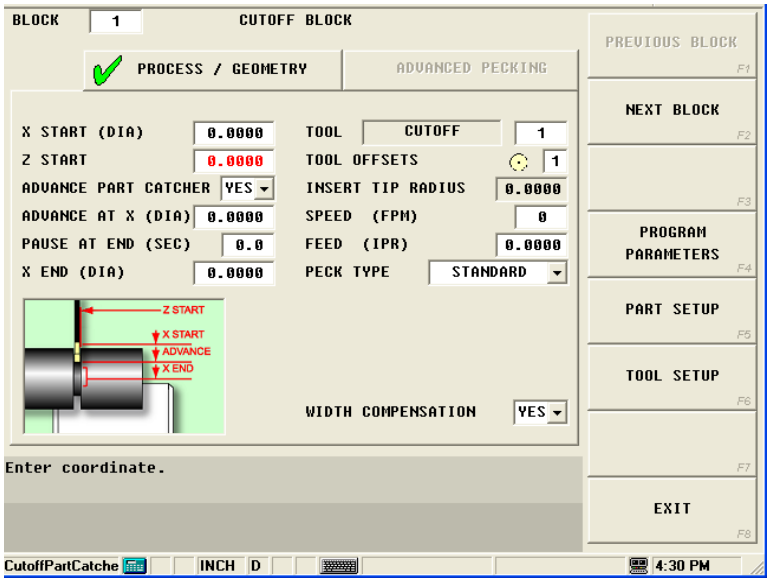


Figure 9–1. Cutoff Block with Parts Catcher fields

To program the parts catcher,

1. Access the Cutoff Block screen in your Conversational Part Program.
2. Select YES for the ADVANCE PART CATHCER field in the Cutoff block. The ADVANCE AT X field is active.
3. Enter the X diameter or radius location for the parts catcher to begin moving. This location is illustrated on the sample screen above, labeled “Advance.”
4. Enter the time in seconds to pause while the parts catcher completes its retract in the PAUSE AT END (SEC) field. After the delay, the turret moves to X End.
5. Enter the X axis end location for the Cutoff block in the X End field. The tool stops cutting at this location.

The parts catcher stays in its elevated position until the cutoff tool retracts and moves to Home.

PARTS CONVEYOR

The optional Parts Conveyor moves finished parts from the parts catcher exit chute to a bin while the machine continues to operate.

The Parts Conveyor can be operated either manually with softkeys on the Manual screen or automatically using M Codes. To program an M Code in a conversational programming data block, please refer to WinMax Lathe Conversational Part Programming, *Machine Function—M Code, on page 2 - 198* and WinMax Lathe NC Programming *Basic NC M Codes, on page 3 - 1* or *ISNC M Codes, on page 5 - 1*.

Parts Conveyor Operation

Access the parts conveyor using the softkey selections from the Manual screen.

1. Press the **Manual** console key to access the Manual screen.
2. Select ACCESSORY OPERATIONS *F4*.
3. Select PARTS CONVEYOR *F6*. The Parts Conveyor softkey menu contains these choices:
   - **PART CONVEYOR FORWARD** *F1*—moves the parts conveyor forward.
   - **PART CONVEYOR REVERSE** *F2*—moves the parts conveyor in reverse.
   - **PART CONVEYOR STOP** *F3*—stops the parts conveyor.

Automatic Operation

To operate the Parts Conveyor automatically, program a Machine Function data block within a conversational part program. From the New Block screen or Program Review screen,

1. Select the **Insert Block Before** *F7* softkey.
2. Select the **Miscellaneous** softkey.
3. Select the **M Code *F3*** softkey.
   - Select M24 to turn on the Parts Conveyor. You can enter the number of seconds to dwell in the Dwell field before moving to the next block.
   - Select M25 to turn off the Parts Conveyor. You can enter the number of seconds to dwell in the Dwell field before moving to the next block.

⇨ If using NC programming, the M24 and M25 codes apply. You can also put a G04 (Dwell) line after the M24 or M25. For example,

```
M24
G04 F0.1
```

PART EJECTOR

The optional Part Ejector ejects finished parts from the Sub-spindle chuck. This option is only available for TMX MYS machines. The Part Ejector consists of a cylinder located at the rear of the spindle and a rod that is attached to the rod of the cylinder that goes through the drawtube of the Sub-spindle. This rod also has a removable tip on the end so the user can construct any type of tip that may be needed to push a part from the Sub-spindle chuck jaws. Upon activation, the ejector pushes the stock forward, away from the chuck, and then the ejector retracts to its original position.

The Part Ejector can be operated either manually with softkeys on the Manual screen or automatically using an M Code to cycle the ejector. To program an M Code in a conversational programming data block, please refer to WinMax Lathe Conversational Part Programming, *Machine Function—M Code, on page 2 - 198* and WinMax Lathe NC Programming *Basic NC M Codes, on page 3 - 1* or *ISNC M Codes, on page 5 - 1*.

Part Ejector Operation

Access the Part Ejector using the softkey selections from the Manual screen.

1. Press the **Manual** console key to access the Manual screen.
2. Select ACCESSORY OPERATIONS *F4* softkey.
3. Select the MORE F7 softkey.
4. Select PART EJECTOR *F4*. The Part Ejector softkey menu contains these choices:
   - **PART EJECTOR ADVANCE** *F1*—moves the ejector cylinder to the forward position as it would be when pushing the stock forwared, out of the Sub-spindle chuck.
   - **PART EJECTOR RETRACT** *F2*—returns the cylinder to the retracted (normal Default) state inside the chuck.

⚠ The Sub-spindle chuck must be Open in order for the Part Ejector to advance.

The Sub-spindle chuck will not close until the Part Ejector is returned to its original position.

If left in its advanced location, the Part Ejector automatically retracts with the first cycle start after entering Auto mode.

Automatic Operation

To operate the Part Ejector automatically, program a Machine Function data block within a conversational part program. From the New Block screen or Program Review screen,

1. Select the **Insert Block Before** *F7* softkey.
2. Select the **Miscellaneous** softkey.
3. Select the **M Code *F3*** softkey.
4. Select M116 to cycle the Part Ejector as follows:
   a. Advance Part Ejector.
   b. Return Part Ejector to original position.

   ⇨ The M116 must be preceded by an M114 (Sub-Spindle Chuck Open) or an error will occur. The error message will read “ERROR: Sub-Spindle Chuck must be OPEN to cycle the Part Ejector.” and program execution will be halted.

The Part Ejector cycle sequence is as follows:

1. A check for errors is performed, e.g. Is Sub-Chuck Open?
2. If no errors were detected, the Part Ejector will then advance.
   a. NOTE: The speed at which the Ejector advances or retracts can be adjusted via a flow control valve on the Part Ejector directional Valve.
   b. The cylinder must advance in less than approximately 1.25 seconds
3. If the cylinder advances completely, which the control determines by reading the 2 proximity switches on the cylinder, then the sequence would go to step 4.
   a. If the switches did not change to the proper state within 1.5 seconds upon the first attempt, then the cylinder will automatically retract. It will then attempt to advance the cylinder again. The cylinder will attempt 4 times to eject the part.
   b. Error messages for failing to advance:
      - ERROR: Part failed to Eject – Cycle Stop. The system still detects the Ejector is at the retracted position. Check that the input DI_PrtEjctrRetrd goes to a Low State when the ejector is advanced.
        - Indicates possible failure of the retracted proximity switch or that the valve did not shift to apply air pressure to advance the ejector.
        - The flow control valve may be closed too far or may possibly be plugged.
        - Check the Air Pressure setting to the Ejector.

- ERROR: Part failed to Eject – Cycle Stop. Unable to detect the Ejector is at the advanced position. Check that a part is not stuck in the Chuck and that the input DI_PrtEjctrAdvcd goes to a High State when advanced.
    - If the ejector is advancing, then the Advanced proximity switch on the cylinder either requires adjustment or has failed.

4. Once the ejector is detected to have successfully advanced, then it will retract.
5. Once the ejector is detected to have successfully retracted, then the program will continue.
    a. If the switches did not change to the proper state within 1.5 seconds upon the first attempt, the cylinder will automatically advance. It will then attempt to retract the cylinder again. The cylinder will attempt 2 times to retract the ejector before it fails.
    b. Error messages for failing to retract
    - ERROR: Part failed to Eject – Cycle Stop. The system still detects the Ejector is at the advanced position. Check that the input DI_PrtEjctrAdvcd goes to a Low State when the ejector is retracted.
        - Indicates possible failure of the advanced proximity switch or that the valve did not shift to apply air pressure to retract the ejector.
        - The flow control valve may be closed too far or may possibly be plugged.
        - Check Air pressure setting to the Ejector as well.
    - ERROR: Part failed to Eject – Cycle Stop. Part Ejector failed to retract after ejecting the part. Check the input DI_PrtEjectrRetrd. The signal should be in a High State when the Ejector is retracted.
        - If the ejector is fully retracting, then the Retracted proximity switch on the cylinder either requires adjustment or has failed.

⇨ An error in the Part Ejector Cycle puts the machine in Cycle Stop as it may be running unattended and this leaves the machine in an idle condition.

STEADY REST

Machines may be equipped with an optional SMW-AUTOBLOK Steady Rest device. The device supports long thin stock at its mid-point. Please refer to *Maintenance and Safety Manual for Turning Centers* for some maintenance information.

> ⚠ Please refer to the manufacturer's *Instruction Manual*, located in the machine electrical cabinet, for complete details regarding installation and maintenance.

From the Manual screen,

1. Select the Accessory Options *F4* softkey. The softkey menu changes.
2. Select the Steady Rest *F6* softkey. This screen appears:

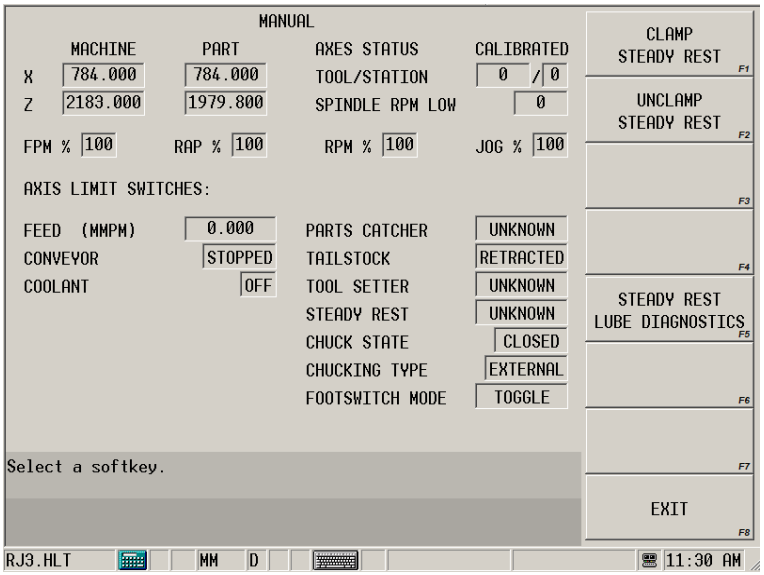


Figure 12–1. Manual screen with Steady Rest Accessory Operations selected

These softkeys are available:

- **CLAMP STEADY REST** *F1*—refer to *Clamp, on page 12 - 2* for details about clamping the Steady Rest device.
- **UNCLAMP STEADY REST** *F2*—refer to *Unclamp, on page 12 - 2* for details about unclamping the Steady Rest device.
- **STEADY REST LUBE DIAGNOSTICS** *F5*—refer to *Steady Rest Lubrication Diagnostics, on page 5 - 47* for details about performing Steady Rest lubrication diagnostics.

Clamp

Select the Clamp Steady Rest *F1* softkey to clamp the Steady Rest device. The Steady Rest device can be manually hitched to the Z-Axis to assist with movement of the Steady Rest to the proper location. The position of the Steady Rest device is tracked in order to avoid colliding with the tailstock.

> ⚠ The position of the Steady Rest can only be tracked when moving the Steady Rest with the Z-Axis. If the Steady Rest is moved manually without the use of the Z-Axis, then the control will not be aware of this new position for the Steady Rest which will either allow the tailstock to possibly collide with it or improperly limit the motion of the Z-Axis when hitched to the Tailstock.

A safety switch located on a bracket under the way cover also protects against the Steady Rest and Tailstock colliding. If the safety switch is tripped, the machine will enter the Emergency Stop mode.

This optional device is used with turning centers to hold long pieces of stock in the center while cutting.

Unclamp

Select the Unclamp Steady Rest *F2* softkey to unclamp the Steady Rest device. After the spindle is stopped, M39 unclamps the optional Steady Rest device during automatic cycle operation. The spindle MUST BE stopped when this M-Code executes or an error will appear and the cycle will be halted.

> ⚠ The spindle must be stopped prior to executing the M39 Steady Rest Unclamp.

The Steady Rest device can be manually hitched to the Z-Axis to assist with movement of the Steady Rest to the proper location. The position of the Steady Rest device is tracked in order to avoid colliding with the tailstock.

> ⚠ The position of the Steady Rest can only be tracked when moving the Steady Rest with the Z-Axis. If the Steady Rest is moved manually without the use of the Z-Axis, then the control will not be aware of this new position for the Steady Rest which will either allow the tailstock to possibly collide with it or improperly limit the motion of the Z-Axis when hitched to the Tailstock.

A safety switch located on a bracket under the way cover also protects against the Steady Rest and Tailstock colliding. If the safety switch is tripped, the machine will enter the Emergency Stop mode.

This optional device is used with turning centers to hold long pieces of stock in the center while cutting.

TAILSTOCK ASSEMBLY

The optional tailstock supports long pieces of stock while the spindle is turning and the part is being cut. The tailstock quill touches the loose end of the stock, on the right-hand side.

The Tailstock Quill can be operated either manually with softkeys on the Manual screen or automatically using M Codes. To program an M Code in a conversational programming data block, please refer to WinMax Lathe Conversational Part Programming, *Machine Function—M Code, on page 2 - 198* and WinMax Lathe NC Programming, *Basic NC M Codes, on page 3 - 1* or *ISNC M Codes, on page 5 - 1*.

⚠ Refer to the *Maintenance and Safety for Turning Centers* manual for information about maintaining the tailstock.

Tailstock Quill Operation

To access tailstock operations,

1. Press the **Manual** console key to access the Manual screen.
2. Select the ACCESSORY OPERATIONS *F4* softkey to access the Accessory softkey menu.
3. Select TAILSTOCK *F2* to access the Tailstock softkeys:

⇨ The Accessory Operations are modal. Once a selection is made with a softkey or with the optional footswitch, the tailstock output changes to that state and stays on until another selection is made.

- **ADVANCE TAILSTOCK** *F1*—advances the tailstock quill when the enclosure doors are closed. ADVANCED appears in the Tailstock field.

⇨ The Advance Tailstock *F5* softkey functions only when the enclosure doors are closed.

⇨ Keep these important points in mind when using the Advance Tailstock *F5* softkey:

  - The spindle must be stopped.
  - The quill pressure switch, if equipped, is not monitored.
  - The tailstock quill will advance either until it is fully extended, or, if it is equipped with a quill pressure switch, when the quill pressure switch is activated.

- **RETRACT TAILSTOCK** *F2*—retracts the tailstock quill when the enclosure doors are closed. RETRACTED appears in the Tailstock field.

⇨ The tailstock quill will only retract when the Retract Tailstock *F6* softkey is selected or when the footswitch is depressed.

Automatic Operation

To operate the tailstock quill automatically, program a Machine Function data block within a conversational part program. From the New Block screen or Program Review screen,

1. Select the Insert Block Before *F7* softkey.
2. Select the Miscellaneous softkey.
3. Select the M Code *F3* softkey.
   - Select M28 to advance the tailstock quill and enter the number of seconds to dwell in the Dwell field before moving to the next block.
   - Select M29 to retract the tailstock quill and enter the number of seconds to dwell in the Dwell field before moving to the next block.

If either the **Interrupt** or **Stop Cycle** console key is pressed before the quill is fully advanced, the quill will automatically retract. If in Interrupt mode and you return to Auto Mode and press **Start Cycle**, the Tailstock will advance before the program continues.

⇨ If using NC programming, the M28 and M29 codes apply.

Optional Tailstock Quill Footswitch

A tailstock footswitch is available as an option for the turning centers. The footswitch behaves differently when the enclosure doors are open or closed. Descriptions for each scenario follow in this section.

Machines may be equipped with a quill pressure switch used in conjunction with the footswitch. Below are descriptions for machines equipped with the switch and machines not equipped with the switch:

Quill Pressure Switch

When machines are equipped with the quill pressure switch,

- **Advance with Doors Closed**—When the enclosure doors are closed and the footswitch is pressed, the tailstock quill advances and reaches full travel when the quill pressure switch reports the tailstock is extended. The footswitch does not need to be held to advance.
- **Advance with Doors Open**—When the enclosure doors are open and the footswitch is pressed, the tailstock quill will advance until the pressure switch changes to High. If the footswitch is released before the switch changes to High, then the tailstock quill automatically retracts.
- **Retract with Doors Closed or Open**—When the enclosure doors are closed or open and the footswitch is pressed, the tailstock quill will retract. The footswitch does not need to be held to advance.

No Quill Pressure Switch

When machines are not equipped with the quill pressure switch,

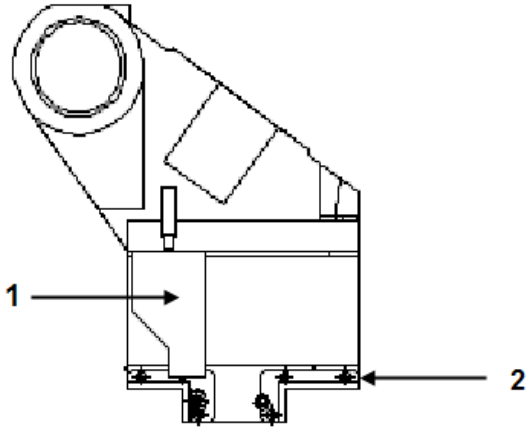
- **Advance with Doors Closed**—When the enclosure doors are closed, the tailstock quill will advance when the footswitch is pressed. The footswitch does not need to be held to advance.
- **Advance with Doors Open**—When the enclosure doors are open, the footswitch must be pressed and held down for 4 seconds before the tailstock quill will advance and hold in the advanced position. If the footswitch is released before the specified amount of time has elapsed, then the tailstock quill automatically retracts.
- **Retract with Doors Closed or Open**—When the enclosure doors are closed or open and the footswitch is pressed, the tailstock quill will retract. The footswitch does not need to be held to advance.

TM10 Tailstock

TM10 turning centers equipped with a tailstock include an engagement block that can be inserted into an opening underneath the X-axis way cover. This setup allows the tailstock to be moved with the Z axis.

A message will appear when the tailstock engagement block is engaged, stating that the Z-axis feed is limited.

When the tailstock engagement block is engaged, both the HOME ➔ *F6* softkey (accessed from the Manual screen, MANUAL FUNCTION SETUP *F2* softkey) and the CALIBRATE MACHINE *F8* softkey are grayed out (inactive).



1	Engagement block with handle
2	Bolts to be loosened

Figure 13–1. TM10 Tailstock with Engagement Block and Handle

To move the tailstock with the Z axis and engagement block,

1. Locate the engagement block handle on top of the engagement block. The assembly is on the left end of the tailstock, attached with a tailstock spacer.
2. Position the Z axis so that the opening at the bottom of the X-axis way cover is visible.
3. Pull the engagement block handle toward you until the bottom of the block fits into the opening at the bottom of the X-axis way cover.
4. Loosen the two bolts located at the bottom front of the tailstock.
5. The tailstock can now be moved using the Z axis. You cannot Calibrate or Home the machine with the engagement block engaged.
6. Tighten the two bolts located at the bottom front of the tailstock.
7. Return engagement block handle to upright position.

THREAD REPAIR CYCLE

A Thread Repair cycle allows you to re-cut a damaged thread.

- This cycle uses the program's Part Setup.
- A threading tool defined in Tool Setup, including orientation, is required for the Active Tool.
- For TM6, TM8, TM10, TM12, and TM18 machines, the spindle must be run for 2 revolutions prior to running this cycle if the machine has not been run after power up.
- This cycle may be used with an active Sub-spindle Work Offset.

To access the Thread Repair Cycle screen from the Program Review screen, select these softkeys: Insert Block Before *F7*, Turning ➔ *F2*, Thread Repair *F6* softkey.

The Thread Repair Cycle screen opens with three tabs: Process, Geometry, and Repair. The Thread Repair Cycle screen and the Threading Cycle screen are the same, except this screen has the Repair tab. Enter information for this cycle the same as you would for a Threading cycle. Refer to *Threading Cycle, on page 2 - 38* for those details.

The **Z Start** value on the Geometry tab screen is part-relative. Program the Z Start value the same as you would program a Threading cycle Z Start value.

⚠ If you jog the threading tool to the Z Start position and press the **Store Position** *F5* softkey, the location includes the offset from Part Setup. Do not change Part Offset after storing this position.

The **Z End** value on the Geometry tab screen is part-relative. The Z End value should be approximately at the last existing thread on the workpiece.

⚠ If you jog the tool to the Z End position and press the **Store Position** *F5* softkey, the location includes the offset from Part Setup. Do not change Part Offset after storing this position.

On the Repair tab screen the **X (DIA)** or **X (RAD)**, **Z**, and **Spindle** fields appear as inactive (grayed out). These are coordinates for the axes and spindle. When the threading tool is aligned with the thread and the Store Reference Positions softkey is selected, these fields will be populated.

⇨ The Reference Positions are not saved with the part program. They must be reset for each specific thread.

To set the Reference Positions,

1. Insert the workpiece containing the damaged thread into the spindle.
2. Jog the active Threading tool into position in an undamaged thread. The ideal position is as close as possible to the center of the thread and the minor diameter as shown in the screen graphic.
3. Select the **Store Reference Positions** *F7* softkey. The machine-relative values are entered into the X (DIA) or (RAD), Z, and Spindle fields.
4. Jog the tool to a safe position away from the workpiece.

The Thread Repair Cycle will run when the part program runs in Auto Mode. The machine movement for this data block is:

1. The turret moves to the Z Start location.
2. The tool cuts the thread as programmed.
3. The tool stops cutting and the turret retracts and moves the tool to the Z Start position.

TOOL SETTER

The tool setter calibrates the tools with respect to the axes. The tool setter consists of the tool setter arm and stylus. This section describes how to calibrate the tool setter, align the stylus head, operate the tool setter, and set tools.

⚠ The tool setter needs to be calibrated before you use it for the first time.

The tool setter requires re-calibration whenever

- The stylus is removed or replaced.
- Whenever an axis motor is replaced.
- If the axis home switch or absolute positions are reset or changed.
- If the turret is removed or loosened on the machine.
- If the setter head or arm is replaced.

⚠ For TMX MY and TMX MYS machines, the Y-axis must be set to 0.000 on the Tool Setter Calibration screen for all of the following procedures.

For TMX MYS machines, a sub-spindle tool setter option is available. Please refer to *Sub-Spindle Tool Setter, on page 15 - 42* for details.

Tool Setter Stylus Alignment

For all turning center models, the stylus head is located at the end of the tool setter arm, on the left side of the frame, next to the chuck. The tool setter arm must be attached squarely to the frame, and the tool setter stylus needs to be aligned correctly prior to tool calibration. To align the stylus,

1. Press the **Manual** console key to access the Manual screen.
2. Select the ACCESSORY OPERATIONS *F4* softkey to access the Accessory softkey menu.
3. Select the TOOL SETTER F4 softkey to access the Tool Setter softkey menu.

   ⇨ The Accessory Operations are modal. Once a selection is made with a softkey, the tool setter output changes to that state and stays on until another selection is made.

4. Select the ADVANCE TOOL SETTER F5 softkey. The tool setter moves into position for calibration.
5. Use the center point of a turning tool, positioned near the mid-point of the stylus head in the X-axis.
   a. Loosen the screw at the base of the stylus head to adjust the height so the center point of the tool is at the mid-point vertically of the stylus head.

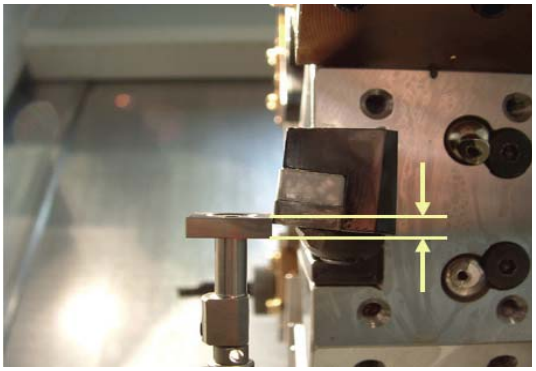


Figure 15–1. Tool Setter Height Adjustment

   b. Tighten the screw when the stylus is positioned properly.
   c. Move the tool away from the stylus head.

6. Use a dial indicator mounted to the turret to position the stylus head so it is parallel to the Z-axis travel.
   a. Jog the axis so the indicator tip runs along the top face of the stylus head to show the stylus is square to 0.002mm (0.0001").

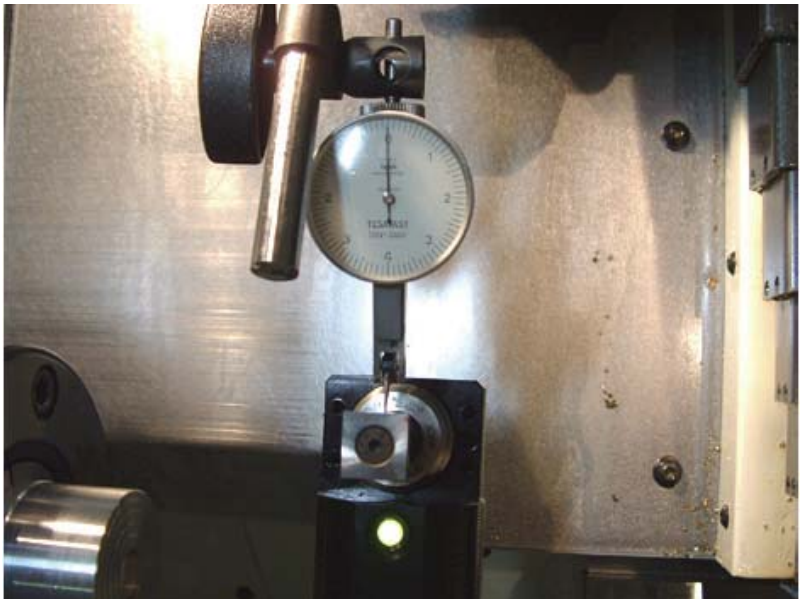


Figure 15–2. Indicating the Stylus Head

   b. Adjust the position of the stylus head with the two set screws located on the right-hand side of the tool setter arm, below the stylus head.

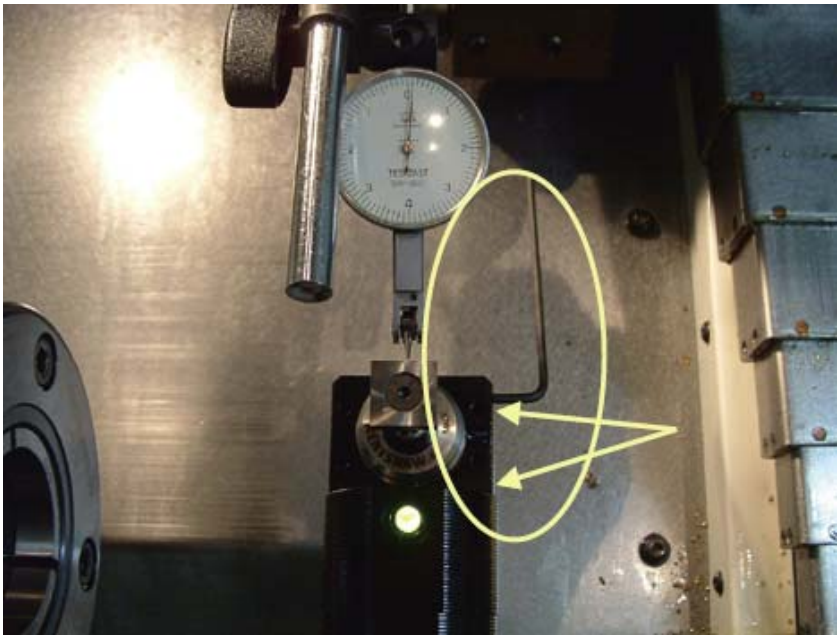


Figure 15–3. Adjusting the Stylus Head

Tool Setter Calibration

For all turning center models, the Tool Setter can be calibrated in any axis order (X first, then Z; or Z first, then X).

For machines equipped with Y-axis motion, the Machine Y field must be set to 0.000 during Tool Setter Calibration.

- Use a test bar (for example, a dowel pin or precision tool shank) and a boring block for calibrating the X axis.
- Use a precision test plate for calibrating the Z axis.

Access the Tool Setter Calibration screen to perform the calibration.

1. Press the **Menu** console button.
2. Select Utilities Screens.
3. Select the SYSTEM CONFIGURATION *F1* softkey.
4. Select the HARDWARE CONFIGURATIONS *F3* softkey.

   The Hardware Configurations screen appears with a Caution message. The message is there to alert users to be familiar with the operations available on the screen selections that follow.

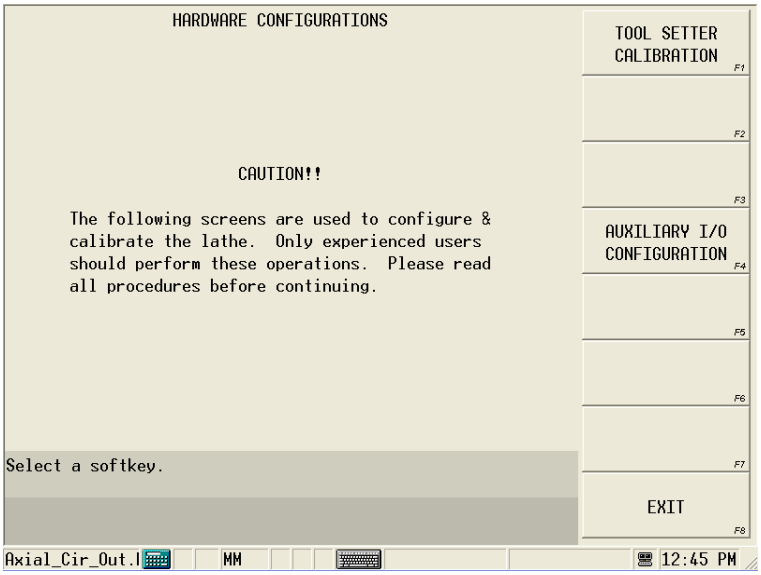


Figure 15–4. Hardware Configurations screen

5. Select the TOOL SETTER CALIBRATION *F1* softkey. The Tool Setter Calibration screen appears:

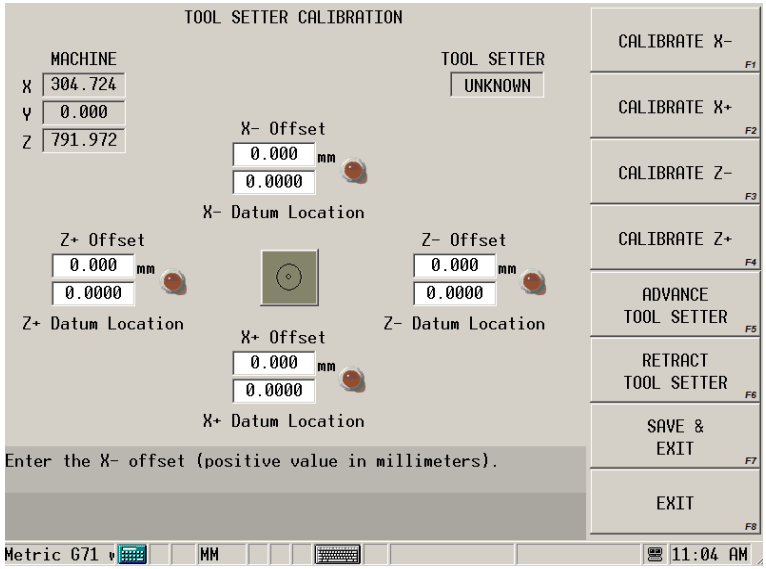


Figure 15–5. Tool Setter Calibration screen

The Tool Setter Calibration screen fields are defined as follows:

- **Machine X Y** and **Z**—displays absolute machine position of X and Z axes. For machines equipped with Y-axis motion, the Y field is also available.

  For machines equipped with Y-axis motion, the Machine Y field must be set to 0.000 during Tool Setter Calibration.

- **Tool Setter**—displays current status of tool setter, either Retracted or Advanced. The tool setter must be Advanced to complete calibration.
- **Gray Square**—(center of screen) represents the face of the tool setter stylus. Use this icon as a reference for the Offsets and Datum Locations.
- **X- Offset**—value represents the distance from 0 (the center of the tool holder), equal to the radius of the test bar. The offset must be a positive number in millimeters.
- **X- Datum Location**—displays the machine position of the X-axis face of the top of the stylus head. The default is zero if uncalibrated. If calibrated, the X-Offset is subtracted from the machine position to obtain X- Datum.
- **X+ Offset**—value represents the distance from 0 (the center of the tool holder), equal to the radius of the test bar. The offset must be a positive number in millimeters. The X-Offset should be equal to the X+ Offset.
- **X+ Datum Location**—displays the machine position of the X-axis face of the bottom of the stylus head. The default is zero if uncalibrated. If calibrated, the X+ Offset is subtracted from the machine position to obtain X+ Datum.
- **Z+ Offset**—value represents the distance from the face of the turret to the surface of the plate. The left side of the precision plate must be aligned with the turret face. The offset must be a positive number in millimeters.

- **Z+ Datum Location**—displays the machine position of the left Z-axis face of the stylus head. The default is zero if uncalibrated. If calibrated, the Z+ Offset is subtracted from the machine position to obtain Z+ Datum.
- **Z- Offset**—value represents the distance from 0, equal to the distance from the face of the turret to the surface of the plate. Set to zero if the left side of the test plate is mounted flush with the turret.
- **Z- Datum Location**—displays the machine position of the Z-axis face of the stylus head on the right side. The default is zero if uncalibrated. If calibrated, the Z- Offset is subtracted from the machine position to obtain Z- Datum.
- One LED icon appears next to each grouping of Offset fields and Datum Location fields:
  - A green icon means that the calibration process is successful.
  - A red icon means that the calibration process is not successful. If this occurs, repeat the calibration process. If the calibration process continues to be unsuccessful, move the start point closer or further from the stylus head.

Follow the Tool Setter Calibration steps for the appropriate machine model:

TM and TMX Models—X-Axis Tool Setter Calibration

Please refer to *TMX MY and TMX MYS Models—X-Axis Tool Setter Calibration, on page 15 - 26* for TMX MY and TMX MYS information.

For TM and TMX turning center models, calibrate the X- and X+ offset as described in the appropriate section below:

X- Axis Tool Setter Calibration

1. Position the axes at Home Position.
2. Use the **Turret -** or **+** key to rotate the turret to the desired location for installing the test bar in a boring block.
3. Attach the boring block to the turret and insert and securely attach the test bar into the holder.

   Measure and record the diameter of the test bar.

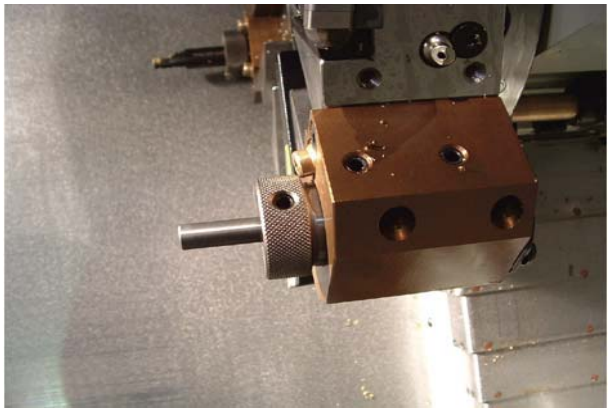


Figure 15–6. Dowel Pin Mounted in Boring Block with a Sleeve

4. Check the test bar for parallelism to the machine. Run an indicator back and forth across the test bar, verifying the bar is parallel to the Z-axis.

   The bar should not be out of alignment by any more than a couple of tenths. Make any necessary adjustments before proceeding to the next step.

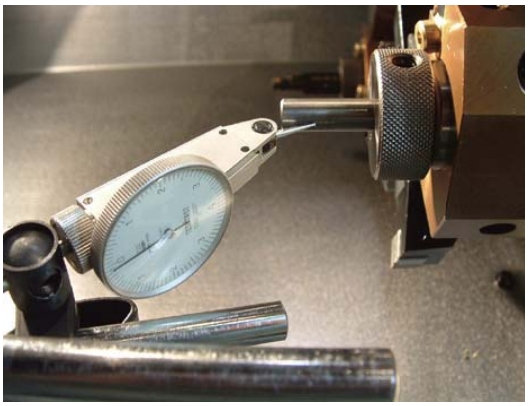


Figure 15–7. Indicating a Dowel Pin for Parallelism to Z-Axis Motion

5. Move the axes back to the Home Position (upper right-hand corner of the machine).
6. Enter the offset into the **X- Offset** field. The offset is equal to the radius of the test bar in mm.

   The X- Offset value is always a positive number in millimeters.

7. Select the ADVANCE TOOL SETTER *F5* softkey to move the tool setter into position.
8. Manually jog the axes toward the tool setter, positioning the side of the test bar to within 10 mm (0.4") of the **top** of the tool setter head. This is the start position required for performing the X- Calibration process.

   If the test bar is positioned more than 10 mm (0.4") from the tool setter head, the calibration move may time-out.

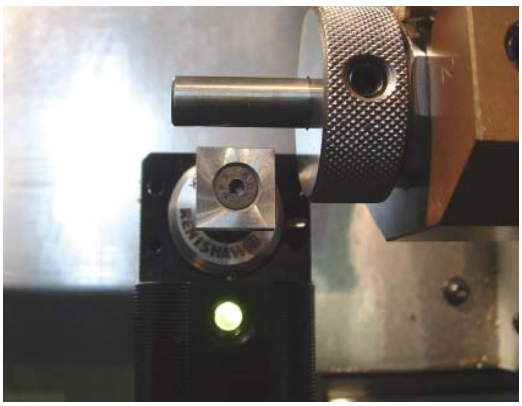


Figure 15–8. Positioning a Dowel Pin at the Tool Setter Head

9. Select the CALIBRATE X- *F1* softkey. The **Start Cycle** button flashes.

   To achieve accurate results, the axis must touch the stylus at a slow feedrate.

   - Set the **Axis Feed Rate** percentage between 10% and 20%.
   - Set the **Rapid Override** percentage at 100%.

10. Press the flashing **Start Cycle** button. The X axis moves slowly toward the tool setter head at 100 mmpm (4 ipm).

    a. When the test bar touches the tool setter head, the axis moves at rapid speed at approximately 10 mm in the opposite direction, then touches the tool setter head two more times.

    b. After the third touch, the axis moves at rapid speed to the X- Offset Calibration start position.

    - If the X- Offset Calibration is successful, the green LED icon is displayed and the machine position minus the X- Offset is displayed in the X-Datum field.
    - If the X- Offset Calibration is not successful, the red LED icon is displayed
    - If the test bar was positioned more than 10 mm from the tool setter head, the system will generate a fault. Reposition the test bar to within 10 mm of the tool setter head and repeat the steps above.

X+ Tool Setter Calibration

1. Position the axes at Home Position.
2. Use the **Turret -** or **+** key to rotate the turret to the desired location for installing the test bar in a boring block.
3. Attach the boring block to the turret and insert and securely attach the test bar into the holder.

   Measure and record the diameter of the test bar.

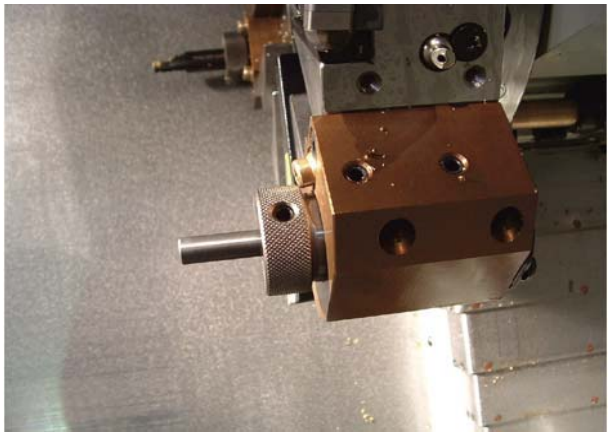


Figure 15–9. Dowel Pin Mounted in Boring Block with a Sleeve

4. Check the test bar for parallelism to the machine. Run an indicator back and forth across the test bar, verifying the bar is parallel to the Z-axis.

   The bar should not be out of alignment by any more than a couple of tenths. Make any necessary adjustments before proceeding to the next step.

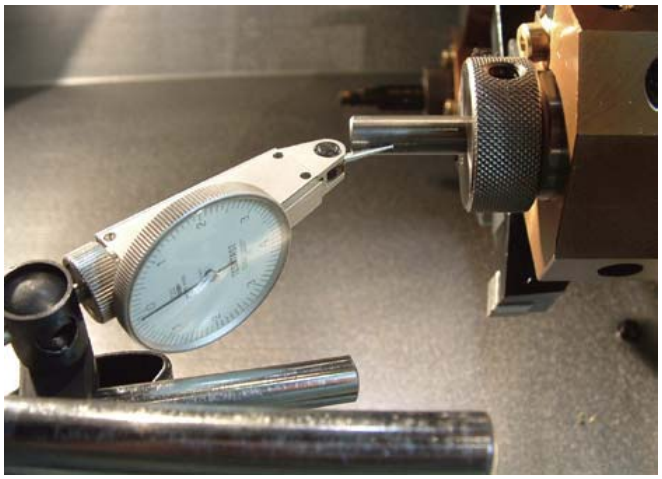


Figure 15–10. Indicating a Dowel Pin for Parallelism to Z-Axis Motion

5. Move the axes back to the Home Position (upper right-hand corner of the machine).
6. Enter the offset into the **X+ Offset** field. The offset is equal to the radius of the test bar in mm.

   The X+ Offset value is always a positive number in millimeters.

7. Select the ADVANCE TOOL SETTER *F5* softkey to move the tool setter into position.
8. Manually jog the axes toward the tool setter, positioning the side of the test bar to within 10 mm (0.4") of the **bottom** of the tool setter head. This is the start position required for performing the X+ Calibration process.

   If the test bar is positioned more than 10 mm (0.4") from the tool setter head, the calibration move may time-out.

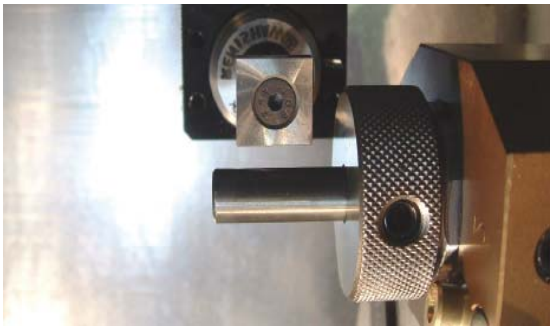


Figure 15–11. Positioning a Dowel Pin at the Tool Setter Head

9. Select the CALIBRATE X+ *F2* softkey. The **Start Cycle** button flashes.

   To achieve accurate results, the axis must touch the stylus at a slow feedrate.

   - Set the **Axis Feed Rate** percentage at 100%.
   - Set the **Rapid Override** percentage at 100%.

10. Press the flashing **Start Cycle** button. The X axis moves slowly toward the tool setter head at 60 mmpm (2.362 ipm).

    a. When the test bar touches the tool setter head, the axis moves at rapid speed at approximately 10 mm in the opposite direction, then touches the tool setter head two more times.

    b. After the third touch, the axis moves at rapid speed to the X+ Offset Calibration start position.

    - If the X+ Offset Calibration is successful, the green LED icon is displayed and the machine position minus the X+ Offset is displayed in the X- Datum field.
    - If the X+ Offset Calibration is not successful, the red LED icon is displayed
    - If the test bar was positioned more than 10 mm from the tool setter head, the system will generate a fault. Reposition the test bar to within 10 mm of the tool setter head and repeat the steps above.

X Calibration Adjustment

The difference between X+ Datum Location and X- Datum Location should be approximately the length of the stylus head in X. To validate the datum,

1. Cut a test part using a Turning operation with a tool that has been calibrated with the tool setter.
2. Measure the turned part and compare to the programmed diameter.
   - If the test part is under-size, increase the X+ or X- Datum values by half the difference of the measured diameter and the programmed diameter.
   - If the test part is over-size, decrease the X+ or X- Datum values by half the difference of the measured diameter and the programmed diameter.

TM and TMX Models—Z-Axis Tool Setter Calibration

Please refer to *TMX MY and TMX MYS Models—Z-Axis Tool Setter Calibration, on page 15 - 30* for TMX MY and TMX MYS information.

For TM and TMX models, calibrate the Z- and Z+ offset as described in the appropriate section below:

Z- Tool Setter Calibration

1. Position the axes at Home Position.
2. Use the **Turret -** or **+** key to rotate the turret to the desired location for installing the test plate in a slot in the turret.
3. Use a magnet to securely attach the test plate to the turret. The left edge of the test plate must be flush with the turret face.

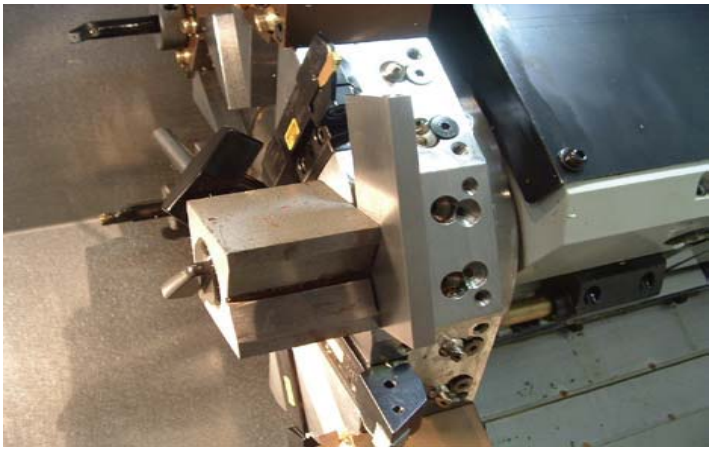


Figure 15–12. Test Plate Secured with a Magnet for Z-axis Calibration

4. Move the axes back to the Home Position (upper right-hand corner of the machine).

5. Enter the offset into the **Z- Offset** field. The offset is the distance from the face of the turret to the surface of the plate that will contact the Stylus. The value may be 0 if the plate is secured flush to the turret face. Otherwise, the value is the thickness of the plate.

   The Z- Offset value is always a positive number in millimeters.

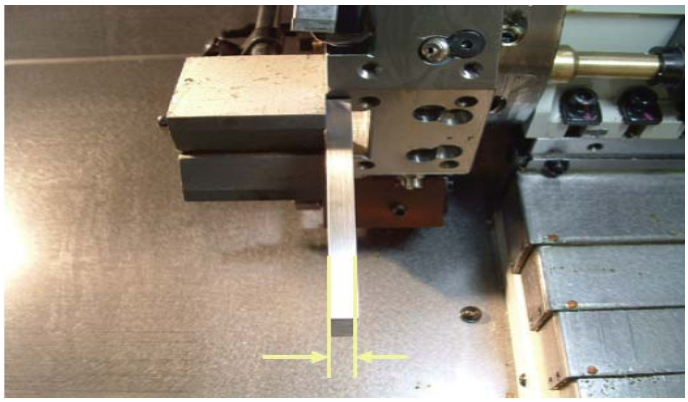


Figure 15–13. Determining the Offset Required for Z-axis Calibration

6. Select the ADVANCE TOOL SETTER *F5* softkey to move the tool setter into position.
7. Manually jog the axes toward the tool setter, positioning the side of the test bar to within 10 mm (0.4") of the **right-hand side** of the tool setter head. This is the start position required for performing the Z- Calibration process.

   If the test bar is positioned more than 10 mm (0.4") from the tool setter head, the calibration move may time-out.

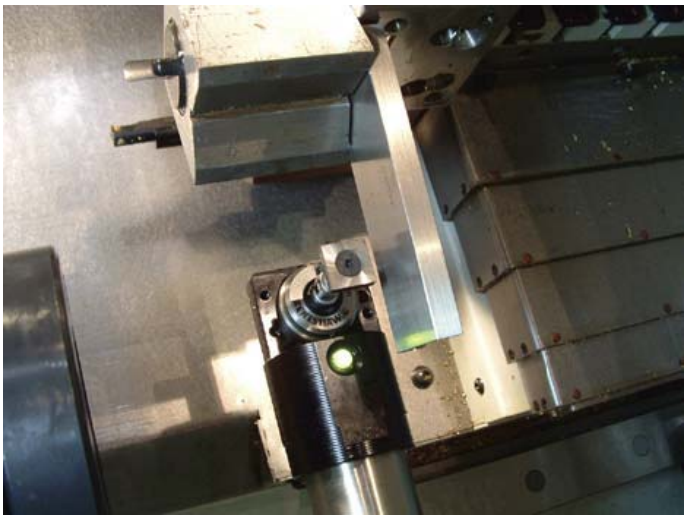


Figure 15–14. Positioning a Test Plate at the Tool Setter Head

8. Select the CALIBRATE Z- *F3* softkey. The **Start Cycle** button flashes.

   To achieve accurate results, the axis must touch the stylus at a slow feedrate.

   - Set the **Axis Feed Rate** percentage between 10% and 20%.
   - Set the **Rapid Override** percentage at 100%.

9. Press the flashing **Start Cycle** button. The Z axis moves slowly toward the tool setter head at 100 mmpm (4 ipm).

   a. When the test plate touches the tool setter head, the axis moves at rapid speed at approximately 10 mm in the opposite direction, then touches the tool setter head two more times.

   b. After the third touch, the axis moves at rapid speed to the Z- Offset Calibration start position.

   - If the Z- Offset Calibration is successful, the green LED icon is displayed and the machine position minus the Z- Offset is displayed in the Z- Datum field.
   - If the Z- Offset Calibration is not successful, the red LED icon is displayed
   - If the test bar was positioned more than 10 mm from the tool setter head, the system will generate a fault. Reposition the test bar to within 10 mm of the tool setter head and repeat the steps above.

Z+ Offset Calibration

1. Position the axes at Home Position.
2. Use the **Turret -** or **+** key to rotate the turret to the desired location for installing the test plate in a slot in the turret.
3. Use a magnet to securely attach the test plate to the turret. The left edge of the test plate must be flush with the turret face.

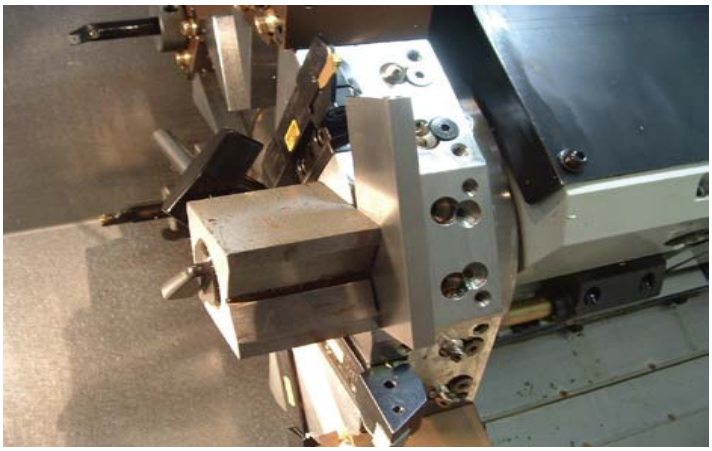


Figure 15–15. Test Plate Secured with a Magnet for Z-axis Calibration

4. Move the axes back to the Home Position (upper right-hand corner of the machine).
5. Enter the offset into the **Z+ Offset** field. The offset is the distance from the face of the turret to the surface of the plate that will contact the Stylus. The value may be 0 if the plate is secured flush to the turret face. Otherwise, the value is the thickness of the plate.

   The Z+ Offset value is always a positive number in millimeters.

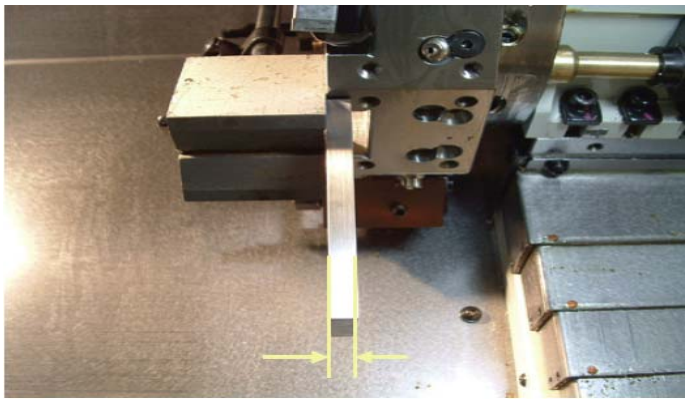


Figure 15–16. Determining the Offset Required for Z-axis Calibration

6. Select the ADVANCE TOOL SETTER *F5* softkey to move the tool setter into position.
7. Manually jog the axes toward the tool setter, positioning the side of the test bar to within 10 mm (0.4") of the **left-hand side** of the tool setter head. This is the start position required for performing the Z- Calibration process.

   If the test bar is positioned more than 10 mm (0.4") from the tool setter head, the calibration move may time-out.

Figure 15–17. Positioning a Test Plate at the Tool Setter Head

8. Select the CALIBRATE Z+ *F4* softkey. The **Start Cycle** button flashes.

   To achieve accurate results, the axis must touch the stylus at a slow feedrate.

   - Set the **Axis Feed Rate** percentage between 10% and 20%.
   - Set the **Rapid Override** percentage at 100%.

9. Press the flashing **Start Cycle** button. The Z axis moves slowly toward the tool setter head at 100 mmpm (4 ipm).

   a. When the test plate touches the tool setter head, the axis moves at rapid speed at approximately 10 mm in the opposite direction, then touches the tool setter head two more times.

   b. After the third touch, the axis moves at rapid speed to the Z+ Offset Calibration start position.

   - If the Z+ Offset Calibration is successful, the green LED icon is displayed and the machine position minus the Z+ Offset is displayed in the Z+ Datum field.
   - If the Z+ Offset Calibration is not successful, the red LED icon is displayed
   - If the test bar was positioned more than 10 mm from the tool setter head, the system will generate a fault. Reposition the test bar to within 10 mm of the tool setter head and repeat the steps above.

Z Calibration Adjustment

The difference between Z+ Datum Location and Z- Datum Location should be approximately the width of the stylus head in Z. To validate the datum,

1. Face and cut-off or part-off a test part using a tool that has been calibrated with the tool setter.
2. Measure the length of the part and compare to the programmed length.
   - If the test part is under-sized, increase the Z+ Datum value by the difference of the measured length and the programmed length.
   - If the test part is over-sized, decrease the Z+ Datum value by the difference of the measured length and the programmed length.

TMM Models—X-Axis Tool Setter Calibration

For TMM turning center models, calibrate the X- and X+ offset as described in the appropriate section below:

X- Axis Tool Setter Calibration

1. Position the axes at Home Position.
2. Use the **Turret -** or **+** key to rotate the turret to the desired location for installing the test bar in a boring block.
3. Attach the boring block to the turret and insert and securely attach the test bar into the holder.

   Measure and record the diameter of the test bar.

4. Move the axes back to the Home Position (upper right-hand corner of the machine).
5. Enter the offset into the **X- Offset** field. The offset is equal to the radius of the test bar in mm.

   The X- Offset value is always a positive number in millimeters.

6. Select the ADVANCE TOOL SETTER *F5* softkey to move the tool setter into position.
7. Manually jog the axes toward the tool setter, positioning the side of the test bar to within 10 mm (0.4") of the **top** of the tool setter head. This is the start position required for performing the X- Calibration process.

   If the test bar is positioned more than 10 mm (0.4") from the tool setter head, the calibration move may time-out.

Figure 15–18. X- Axis Tool Setter Calibration—TMM

8. Select the CALIBRATE X- *F1* softkey. The **Start Cycle** button flashes.

   To achieve accurate results, the axis must touch the stylus at a slow feedrate.

   - Set the **Axis Feed Rate** percentage between 10% and 20%.
   - Set the **Rapid Override** percentage at 100%.

9. Press the flashing **Start Cycle** button. The X axis moves slowly toward the tool setter head at 100 mmpm (4 ipm).
   a. When the test bar touches the tool setter head, the axis moves at rapid speed at approximately 10 mm in the opposite direction, then touches the tool setter head two more times.
   b. After the third touch, the axis moves at rapid speed to the X- Offset Calibration start position.

   - If the X- Offset Calibration is successful, the green LED icon is displayed and the machine position minus the X- Offset is displayed in the X-Datum field.
   - If the X- Offset Calibration is not successful, the red LED icon is displayed
   - If the test bar was positioned more than 10 mm from the tool setter head, the system will generate a fault. Reposition the test bar to within 10 mm of the tool setter head and repeat the steps above.

X+ Tool Setter Calibration

1. Position the axes at Home Position.
2. Use the **Turret -** or **+** key to rotate the turret to the desired location for installing the test bar in a boring block.
3. Attach the boring block to the turret and insert and securely attach the test bar into the holder.

   Measure and record the diameter of the test bar.

4. Move the axes back to the Home Position (upper right-hand corner of the machine).
5. Enter the offset into the **X+ Offset** field. The offset is equal to the radius of the test bar in mm.

   The X+ Offset value is always a positive number in millimeters.

6. Select the ADVANCE TOOL SETTER *F5* softkey to move the tool setter into position.

7. Manually jog the axes toward the tool setter, positioning the side of the test bar to within 10 mm (0.4") of the **bottom** of the tool setter head. This is the start position required for performing the X+ Calibration process.

   If the test bar is positioned more than 10 mm (0.4") from the tool setter head, the calibration move may time-out.

Figure 15–19. X+ Axis Tool Setter Calibration—TMM

8. Select the CALIBRATE X+ *F2* softkey. The **Start Cycle** button flashes.

   To achieve accurate results, the axis must touch the stylus at a slow feedrate.

   - Set the **Axis Feed Rate** percentage at 100%.
   - Set the **Rapid Override** percentage at 100%.

9. Press the flashing **Start Cycle** button. The X axis moves slowly toward the tool setter head at 60 mmpm (2.362 ipm).

   a. When the test bar touches the tool setter head, the axis moves at rapid speed at approximately 10 mm in the opposite direction, then touches the tool setter head two more times.

   b. After the third touch, the axis moves at rapid speed to the X+ Offset Calibration start position.

   - If the X+ Offset Calibration is successful, the green LED icon is displayed and the machine position minus the X+ Offset is displayed in the X- Datum field.
   - If the X+ Offset Calibration is not successful, the red LED icon is displayed
   - If the test bar was positioned more than 10 mm from the tool setter head, the system will generate a fault. Reposition the test bar to within 10 mm of the tool setter head and repeat the steps above.

X Calibration Adjustment

The difference between X+ Datum Location and X- Datum Location should be approximately the length of the stylus head in X. To validate the datum,

1. Cut a test part using a Turning operation with a tool that has been calibrated with the tool setter.
2. Measure the turned part and compare to the programmed diameter.
   - If the test part is under-size, increase the X+ or X- Datum values by half the difference of the measured diameter and the programmed diameter.
   - If the test part is over-size, decrease the X+ or X- Datum values by half the difference of the measured diameter and the programmed diameter.

TMM Models—Z-Axis Tool Setter Calibration

Calibrate the Z- and Z+ offset as described in the appropriate section below:

Z- Tool Setter Calibration

1. Position the axes at Home Position.
2. Use the **Turret -** or **+** key to rotate the turret to the desired location for installing the test plate in a slot in the turret.
3. Use a magnet to securely attach the test plate to the turret.
4. Move the axes back to the Home Position (upper right-hand corner of the machine).
5. Enter the offset into the **Z- Offset** field. The offset is the distance from the face of the turret to the surface of the plate that will contact the Stylus. The value may be 0 if the plate is secured flush to the turret face. Otherwise, the value is the thickness of the plate.

   ⚠ The Z- Offset value is always a positive number in millimeters.

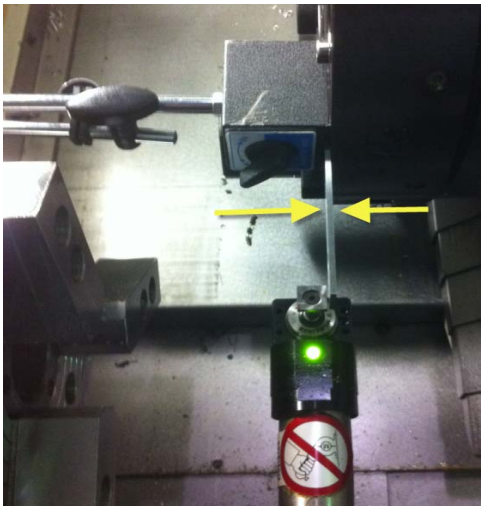


Figure 15–20. Z- Offset—TMM

6. Select the ADVANCE TOOL SETTER *F5* softkey to move the tool setter into position.

7. Manually jog the axes toward the tool setter, positioning the side of the test bar to within 10 mm (0.4") of the **right-hand side** of the tool setter head. This is the start position required for performing the Z- Calibration process.

   ⚠ If the test bar is positioned more than 10 mm (0.4") from the tool setter head, the calibration move may time-out.

Figure 15–21. Z- Axis Tool Setter Calibration—TMM

8. Select the CALIBRATE Z- *F3* softkey. The **Start Cycle** button flashes.

   ⚠ To achieve accurate results, the axis must touch the stylus at a slow feedrate.

   - Set the **Axis Feed Rate** percentage between 10% and 20%.
   - Set the **Rapid Override** percentage at 100%.

9. Press the flashing **Start Cycle** button. The Z axis moves slowly toward the tool setter head at 100 mmpm (4 ipm).

   a. When the test plate touches the tool setter head, the axis moves at rapid speed at approximately 10 mm in the opposite direction, then touches the tool setter head two more times.

   b. After the third touch, the axis moves at rapid speed to the Z- Offset Calibration start position.

   ⚠
   - If the Z- Offset Calibration is successful, the green LED icon is displayed and the machine position minus the Z- Offset is displayed in the Z- Datum field.
   - If the Z- Offset Calibration is not successful, the red LED icon is displayed
   - If the test bar was positioned more than 10 mm from the tool setter head, the system will generate a fault. Reposition the test bar to within 10 mm of the tool setter head and repeat the steps above.

Z+ Offset Calibration

1. Position the axes at Home Position.
2. Use the **Turret -** or **+** key to rotate the turret to the desired location for installing the test plate in a slot in the turret.
3. Use a magnet to securely attach the test plate to the turret.
4. Move the axes back to the Home Position (upper right-hand corner of the machine).
5. Enter the offset into the **Z+ Offset** field. The offset is the distance from the face of the turret to the surface of the plate that will contact the Stylus. The value may be 0 if the plate is secured flush to the turret face. Otherwise, the value is the thickness of the plate.

   The Z+ Offset value is always a positive number in millimeters.

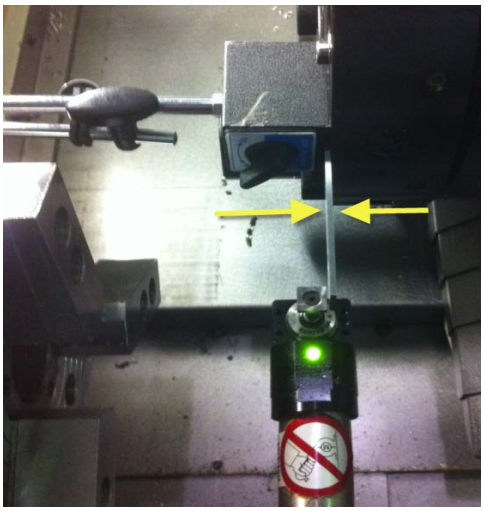


Figure 15–22. Z- Offset—TMM

6. Select the ADVANCE TOOL SETTER *F5* softkey to move the tool setter into position.

7. Manually jog the axes toward the tool setter, positioning the side of the test bar to within 10 mm (0.4") of the **left-hand side** of the tool setter head. This is the start position required for performing the Z- Calibration process.

   ⚠ If the test bar is positioned more than 10 mm (0.4") from the tool setter head, the calibration move may time-out.

Figure 15–23. Z+ Axis Tool Setter Calibration—TMM

8. Select the CALIBRATE Z+ *F4* softkey. The **Start Cycle** button flashes.

   ⚠ To achieve accurate results, the axis must touch the stylus at a slow feedrate.

   - Set the **Axis Feed Rate** percentage between 10% and 20%.
   - Set the **Rapid Override** percentage at 100%.

9. Press the flashing **Start Cycle** button. The Z axis moves slowly toward the tool setter head at 100 mmpm (4 ipm).

   a. When the test plate touches the tool setter head, the axis moves at rapid speed at approximately 10 mm in the opposite direction, then touches the tool setter head two more times.

   b. After the third touch, the axis moves at rapid speed to the Z+ Offset Calibration start position.

   ⚠
   - If the Z+ Offset Calibration is successful, the green LED icon is displayed and the machine position minus the Z+ Offset is displayed in the Z+ Datum field.
   - If the Z+ Offset Calibration is not successful, the red LED icon is displayed
   - If the test bar was positioned more than 10 mm from the tool setter head, the system will generate a fault. Reposition the test bar to within 10 mm of the tool setter head and repeat the steps above.

Z Calibration Adjustment

The difference between Z+ Datum Location and Z- Datum Location should be approximately the width of the stylus head in Z. To validate the datum,

1. Face and cut-off or part-off a test part using a tool that has been calibrated with the tool setter.
2. Measure the length of the part and compare to the programmed length.
   - If the test part is under-sized, increase the Z+ Datum value by the difference of the measured length and the programmed length.
   - If the test part is over-sized, decrease the Z+ Datum value by the difference of the measured length and the programmed length.

TMX MY and TMX MYS Models—X-Axis Tool Setter Calibration

For TMX MY and TMX MYS turning center models, calibrate the X- and X+ offset as described in the appropriate section below.

For machines equipped with Y-axis motion, the Machine Y field must be set to 0.000 during Tool Setter Calibration.

A Sub-spindle Tool Setter option is available for TMX MYS machines. Please refer to *Sub-Spindle Tool Setter, on page 15 - 42* for details.

X- Axis Tool Setter Calibration

1. Position the axes at Home Position.
2. Use the **Turret -** or **+** key to rotate the turret to the desired location for installing the gauge pin in a boring block.
3. Attach the boring block to the turret and insert and securely attach the gauge pin into the holder.

   Measure and record the diameter of the gauge pin. In the example that follows, the diameter of the gauge pin is 12.7 mm.

4. Check the gauge pin for parallelism to the machine. Run an indicator back and forth across the gauge pin, verifying the gauge pin is parallel to the Z-axis.

   The gauge pin should not be out of alignment by any more than a couple of tenths. Make any necessary adjustments before proceeding to the next step.

5. Move the axes back to the Home Position (upper right-hand corner of the machine).

6. Enter the offset into the **X- Offset** field. The offset is the distance from X Zero (the face of the turret with the tool holder mounted) to the center of the tool holder plus the pin radius. In the example below, the X- Offset is: (100mm + Gauge Pin Radius 6.35mm) = 106.35mm.

   The X- Offset value is always a positive number in millimeters.

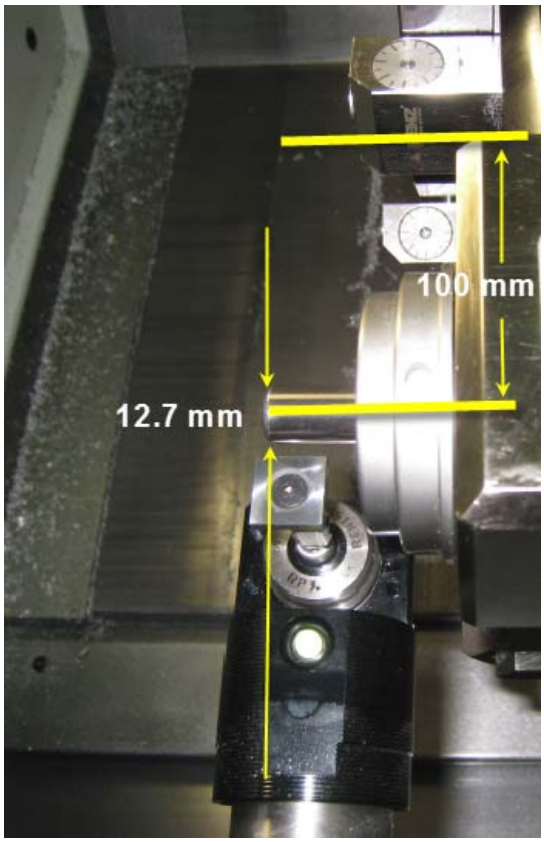


Figure 15–24. X-axis Position for X- Calibration for TMX MY and TMX MYS machines

7. Select the ADVANCE TOOL SETTER *F5* softkey to move the tool setter into position.
8. Manually jog the axes toward the tool setter, positioning the side of the gauge pin to within 10 mm (0.4") of the **top** of the tool setter head. This is the start position required for performing the X- Calibration process.

   If the gauge pin is positioned more than 10 mm (0.4") from the tool setter head, the calibration move may time-out.

9. Select the CALIBRATE X- *F1* softkey. The **Start Cycle** button flashes.

   To achieve accurate results, the axis must touch the stylus at a slow feedrate.

   - Set the **Axis Feed Rate** percentage between 10% and 20%.
   - Set the **Rapid Override** percentage at 100%.

10. Press the flashing **Start Cycle** button. The X axis moves slowly toward the tool setter head at 100 mmpm (4 ipm).
    a. When the gauge pin touches the tool setter head, the axis moves at rapid speed at approximately 10 mm in the opposite direction, then touches the tool setter head two more times.
    b. After the third touch, the axis moves at rapid speed to the X- Offset Calibration start position.

    - If the X- Offset Calibration is successful, the green LED icon is displayed and the machine position minus the X- Offset is displayed in the X-Datum field.
    - If the X- Offset Calibration is not successful, the red LED icon is displayed
    - If the gauge pin was positioned more than 10 mm from the tool setter head, the system will generate a fault. Reposition the gauge pinto within 10 mm of the tool setter head and repeat the steps above.

X+ Tool Setter Calibration

1. Position the axes at Home Position.
2. Use the **Turret -** or **+** key to rotate the turret to the desired location for installing the test bar in a boring block.
3. Attach the boring block to the turret and insert and securely attach the gauge pin into the holder.

   Measure and record the diameter of the gauge pin.

4. Check the gauge pin for parallelism to the machine. Run an indicator back and forth across the test bar, verifying the gauge pin is parallel to the Z-axis.

   The gauge pin should not be out of alignment by any more than a couple of tenths. Make any necessary adjustments before proceeding to the next step.

5. Move the axes back to the Home Position (upper right-hand corner of the machine).
6. Enter the offset into the **X+ Offset** field. The offset is the distance from X Zero (the face of the turret with the tool holder mounted) to the center of the tool holder minus the pin radius. In the example below, the X+ Offset is: (100mm - Gauge Pin Radius 6.35mm) = 93.65mm.

   The X+ Offset value is always a positive number in millimeters.

7. Select the ADVANCE TOOL SETTER *F5* softkey to move the tool setter into position.
8. Manually jog the axes toward the tool setter, positioning the side of the gauge pin to within 10 mm (0.4") of the **bottom** of the tool setter head. This is the start position required for performing the X+ Calibration process.

   If the gauge pin is positioned more than 10 mm (0.4") from the tool setter head, the calibration move may time-out.

9. Select the CALIBRATE X+ *F2* softkey. The **Start Cycle** button flashes.

   To achieve accurate results, the axis must touch the stylus at a slow feedrate.

   - Set the **Axis Feed Rate** percentage at 100%.
   - Set the **Rapid Override** percentage at 100%.

10. Press the flashing **Start Cycle** button. The X axis moves slowly toward the tool setter head at 60 mmpm (2.362 ipm).

    a. When the gauge pin touches the tool setter head, the axis moves at rapid speed at approximately 10 mm in the opposite direction, then touches the tool setter head two more times.

    b. After the third touch, the axis moves at rapid speed to the X+ Offset Calibration start position.

    - If the X+ Offset Calibration is successful, the green LED icon is displayed and the machine position minus the X+ Offset is displayed in the X- Datum field.
    - If the X+ Offset Calibration is not successful, the red LED icon is displayed
    - If the gauge pin was positioned more than 10 mm from the tool setter head, the system will generate a fault. Reposition the test bar to within 10 mm of the tool setter head and repeat the steps above.

X Calibration Adjustment

The difference between X+ Datum Location and X- Datum Location should be approximately the length of the stylus head in X. To validate the datum,

1. Cut a test part using a Turning operation with a tool that has been calibrated with the tool setter.
2. Measure the turned part and compare to the programmed diameter.
   - If the test part is under-size, increase the X+ or X- Datum values by half the difference of the measured diameter and the programmed diameter.
   - If the test part is over-size, decrease the X+ or X- Datum values by half the difference of the measured diameter and the programmed diameter.

TMX MY and TMX MYS Models—Z-Axis Tool Setter Calibration

Calibrate the Z- and Z+ offset as described in the appropriate section below:

For machines equipped with Y-axis motion, the Machine Y field must be set to 0.000 during Tool Setter Calibration.

A Sub-spindle Tool Setter option is available for TMX MYS machines. Please refer to *Sub-Spindle Tool Setter, on page 15 - 42* for details.

Z- Tool Setter Calibration

1. Position the axes at Home Position.
2. Use the **Turret -** or **+** key to rotate the turret to the desired location for installing the test plate in a slot in the turret.
3. Check the gauge pin for parallelism to the machine. Run an indicator back and forth across the test bar, verifying the gauge pin is parallel to the X-axis.

   The gauge pin should not be out of alignment by any more than a couple of tenths. Make any necessary adjustments before proceeding to the next step.

4. Move the axes back to the Home Position (upper right-hand corner of the machine).
5. Enter the offset into the **Z- Offset** field. The offset is the centerline of the tool holder, which is equal to the radius of the gauge pin. In this example, the Z- and Z+ Offset is 6.35mm.

   The Z- Offset value is always a positive number in millimeters.

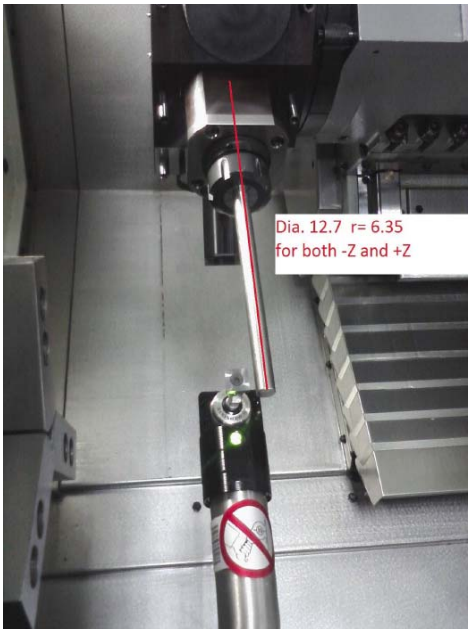


Figure 15–25. Gauge Pin measurement

6. Select the ADVANCE TOOL SETTER *F5* softkey to move the tool setter into position.
7. Manually jog the axes toward the tool setter, positioning the side of the gauge pin to within 10 mm (0.4") of the right-hand side of the tool setter head. This is the start position required for performing the Z- Calibration process.

   If the gauge bar is positioned more than 10 mm (0.4") from the tool setter head, the calibration move may time-out.

8. Select the CALIBRATE Z- *F3* softkey. The **Start Cycle** button flashes.

   To achieve accurate results, the axis must touch the stylus at a slow feedrate.

   - Set the **Axis Feed Rate** percentage between 10% and 20%.
   - Set the **Rapid Override** percentage at 100%.

9. Press the flashing **Start Cycle** button. The Z axis moves slowly toward the tool setter head at 100 mmpm (4 ipm).
   a. When the gauge bar touches the tool setter head, the axis moves at rapid speed at approximately 10 mm in the opposite direction, then touches the tool setter head two more times.
   b. After the third touch, the axis moves at rapid speed to the Z- Offset Calibration start position.

   - If the Z- Offset Calibration is successful, the green LED icon is displayed and the machine position minus the Z- Offset is displayed in the Z- Datum field.
   - If the Z- Offset Calibration is not successful, the red LED icon is displayed
   - If the gauge pin was positioned more than 10 mm from the tool setter head, the system will generate a fault. Reposition the test bar to within 10 mm of the tool setter head and repeat the steps above.

Z+ Offset Calibration

1. Position the axes at Home Position.
2. Use the **Turret -** or **+** key to rotate the turret to the desired location for installing the test plate in a slot in the turret.
3. Check the gauge pin for parallelism to the machine. Run an indicator back and forth across the test bar, verifying the gauge pin is parallel to the X-axis.

   The gauge pin should not be out of alignment by any more than a couple of tenths. Make any necessary adjustments before proceeding to the next step.

4. Move the axes back to the Home Position (upper right-hand corner of the machine).
5. Enter the offset into the **Z+ Offset** field. The offset is the centerline of the tool holder, which is equal to the radius of the gauge pin. In this example, the Z- and Z+ Offset is 6.35mm.

   The Z+ Offset value is always a positive number in millimeters.

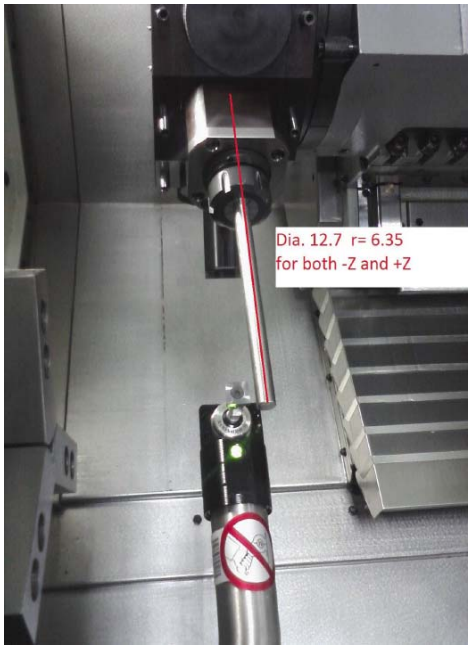


Figure 15–26. Gauge Pin measurement

6. Select the ADVANCE TOOL SETTER *F5* softkey to move the tool setter into position.
7. Manually jog the axes toward the tool setter, positioning the side of the gauge pin to within 10 mm (0.4") of the left-hand side of the tool setter head. This is the start position required for performing the Z- Calibration process.

   If the gauge pin is positioned more than 10 mm (0.4") from the tool setter head, the calibration move may time-out.

8. Select the CALIBRATE Z+ *F4* softkey. The **Start Cycle** button flashes.

   To achieve accurate results, the axis must touch the stylus at a slow feedrate.

   - Set the **Axis Feed Rate** percentage between 10% and 20%.
   - Set the **Rapid Override** percentage at 100%.

9. Press the flashing **Start Cycle** button. The Z axis moves slowly toward the tool setter head at 100 mmpm (4 ipm).

   a. When the gauge pin touches the tool setter head, the axis moves at rapid speed at approximately 10 mm in the opposite direction, then touches the tool setter head two more times.

   b. After the third touch, the axis moves at rapid speed to the Z+ Offset Calibration start position.

   - If the Z+ Offset Calibration is successful, the green LED icon is displayed and the machine position minus the Z+ Offset is displayed in the Z+ Datum field.
   - If the Z+ Offset Calibration is not successful, the red LED icon is displayed
   - If the test bar was positioned more than 10 mm from the tool setter head, the system will generate a fault. Reposition the test bar to within 10 mm of the tool setter head and repeat the steps above.

Z Calibration Adjustment

The difference between Z+ Datum Location and Z- Datum Location should be approximately the width of the stylus head in Z. To validate the datum,

1. Face and cut-off or part-off a test part using a tool that has been calibrated with the tool setter.
2. Measure the length of the part and compare to the programmed length.
   - If the test part is under-sized, increase the Z+ Datum value by the difference of the measured length and the programmed length.
   - If the test part is over-sized, decrease the Z+ Datum value by the difference of the measured length and the programmed length.

Tool Setter Operation

The Tool Setter can be operated either manually with softkeys on the Manual screen or automatically using M Codes. To program an M Code in a conversational programming data block, please refer to WinMax Lathe Conversational Part Programming, *Machine Function—M Code, on page 2 - 198* and WinMax Lathe NC *Basic NC M Codes, on page 3 - 1* or *ISNC M Codes, on page 5 - 1*.

To operate the tool setter and calibrate tools, access the tool setter softkeys from the Manual screen.

1. Press the **Manual** console key to access the Manual screen.
2. Select the ACCESSORY OPERATIONS *F4* softkey to access the Accessory softkey menu.
3. Select the TOOL SETTER F4 softkey to access the Tool Setter softkey menu.

   ⇨ The Accessory Operations are modal. Once a selection is made with a softkey, the tool setter output changes to that state and stays on until another selection is made.

- **ADVANCE TOOL SETTER** *F1*—advances the tool setter. ADVANCED appears in the Tool Setter field. The tool setter is ready to calibrate tools when ADVANCED is selected.

  ⇨ The tool setter arm cannot be advanced if the machine is not calibrated or if the turret is within 250mm (default factory setting) of the spindle.

- **RETRACT TOOL SETTER** *F2*—retracts the tool setter. RETRACTED appears in the Tool Setter field.

Automatic Operation

To operate the tool setter automatically, program a Machine Function data block within a conversational part program. From the New Block screen or Program Review screen,

1. Select the Insert Block Before *F7* softkey.
2. Select the Miscellaneous softkey.
3. Select the M Code *F3* softkey.
   - Select M71 to retract the tool setter and enter the number of seconds to dwell in the Dwell field.
   - Select M72 to advance the tool setter and enter the number of seconds to dwell in the Dwell field.

   ⇨ If using NC programming, the M71 and M72 codes apply.

Setting Tools

With the Tool Setter option active, from the Tool Setup screen, select the Geometry Offsets *F3* softkey followed by the Tool Setter *F6* softkey. This screen appears:

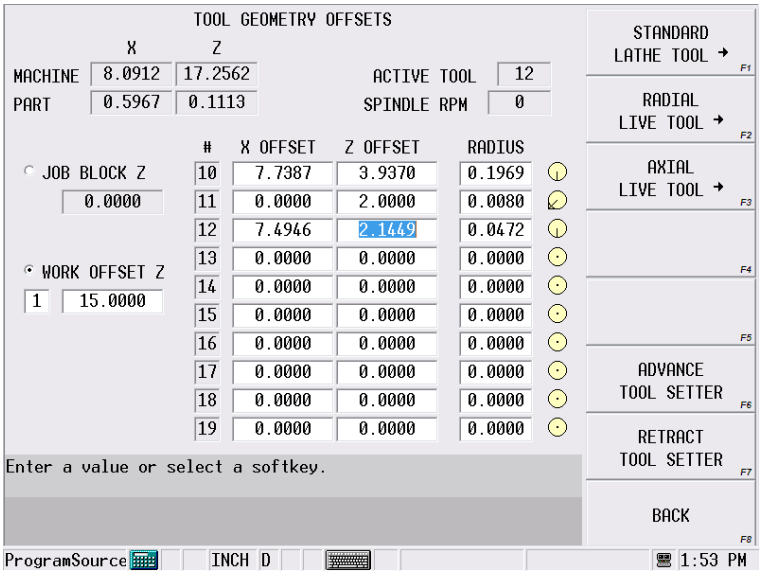


Figure 15–27. Tool Setter softkey menu

Softkeys are defined as follows:

- **STANDARD LATHE TOOL** *F1*—accesses the softkey menu for standard tool calibration.
- **RADIAL LIVE TOOL** *F2*—accesses the softkey menu for radial live tool calibration.
- **AXIAL LIVE TOOL** *F3*—accesses the softkey menu for axial live tool calibration.
- **ADVANCE TOOL SETTER** *F6*—advances the tool setter for calibration.
- **RETRACT TOOL SETTER** *F7*—retracts the tool setter for calibration.
- **BACK** *F8*—returns to the Tool Geometry Offsets screen.

Setting Operation

Follow these steps to use the tool setter to set tools:

1. Position the axes at Home Position.
2. Select the ADVANCE TOOL SETTER *F6* softkey to move the tool setter into position.
3. Manually jog the axes toward the tool setter, positioning the side of the tool to within 10 mm (0.4") of the tool setter head, either the top (X-), bottom (X+), left-hand (Z+), or right-hand (Z-) side. This is the start position required for performing the Tool Setting process.

   If the tool is positioned more than 10 mm (0.4") from the tool setter head, the calibration move may time-out.

4. Select the appropriate TOUCH softkey. The **Start Cycle** button flashes.

   Refer to *Standard Lathe Tools, on page 15 - 37*, *Radial Live Tools, on page 15 - 38*, and *Axial Live Tools, on page 15 - 40* for information about calibrating each tool type.

   To achieve accurate results, the axis must touch the stylus at a slow feedrate.

   - Set the **Axis Feed Rate** percentage between 10% and 20%.
   - Set the **Rapid Override** percentage at 100%.

5. Press the flashing **Start Cycle** button. The selected axis moves slowly toward the tool setter head at 100 mmpm (4 ipm).
   a. When the tool touches the tool setter head, the axis moves at rapid speed at approximately 10 mm in the opposite direction, then touches the tool setter head two more times.
   b. After the third touch, the axis moves at rapid speed to the start position.

   - If the Calibration is successful, the offset is entered into the appropriate Offset field.
   - If the tool was positioned more than 10 mm from the tool setter head, the system will generate a fault. Reposition the tool to within 10 mm of the tool setter head and repeat the steps above.
   - For TMX MY and TMX MYS machines, set the X Offset field in Tool Setup, Geometry Offsets prior to calibrating for the tool radius.
   - If an X Axis entry is in Diameter mode, then double the value before entering it into the Tool Setup, Geometry Offsets, X Offset field.

Standard Lathe Tools

With the tool setter advanced, select the appropriate softkey to set the standard lathe tool in the selected orientation.

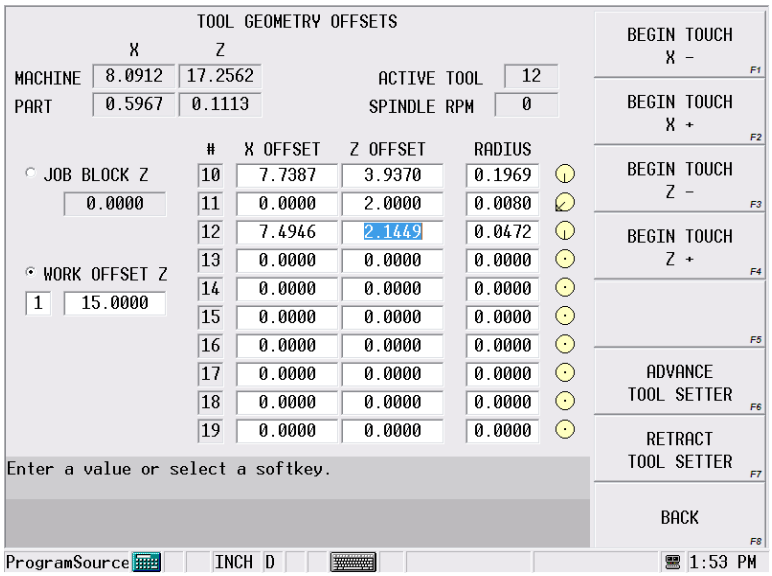


Figure 15–28. Standard Tool Setting screen

- **BEGIN TOUCH X-** *F1*—begins tool calibration in the X- direction.
- **BEGIN TOUCH X+** *F2*—begins tool calibration in the X+ direction.
- **BEGIN TOUCH Z-** *F3*—begins tool calibration in the Z- direction.
- **BEGIN TOUCH Z+** *F4*—begins tool calibration in the Z+ direction.

Radial Live Tools

With the tool setter advanced, select the appropriate softkey to set the tool in the selected orientation.

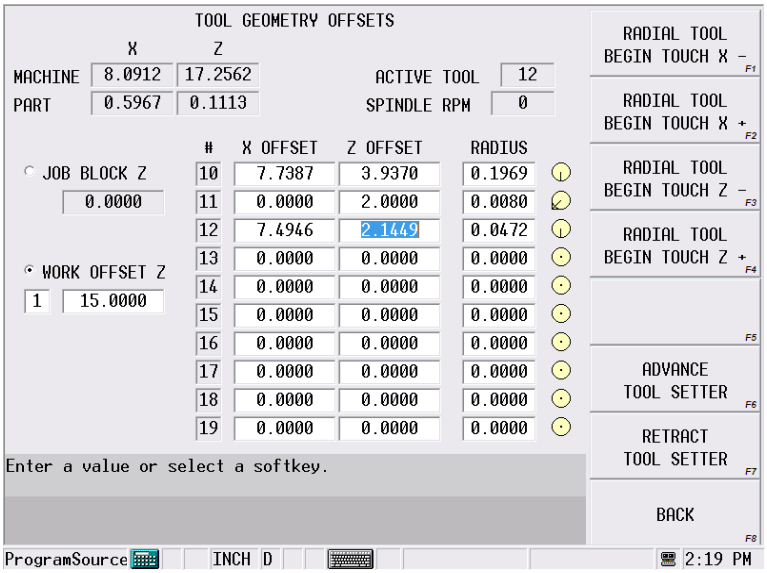


Figure 15–29. Radial Tool Setting screen

- **RADIAL TOOL BEGIN TOUCH X-** *F1*—begins radial tool calibration in the X- direction.
- **RADIAL TOOL BEGIN TOUCH X+** *F2*—begins radial tool calibration in the X+ direction.
- **RADIAL TOOL BEGIN TOUCH Z-** *F3*—begins radial tool calibration in the Z- direction.
- **RADIAL TOOL BEGIN TOUCH Z+** *F4*—begins radial tool calibration in the Z+ direction.

Here is a live radial tool being set in the X- direction:

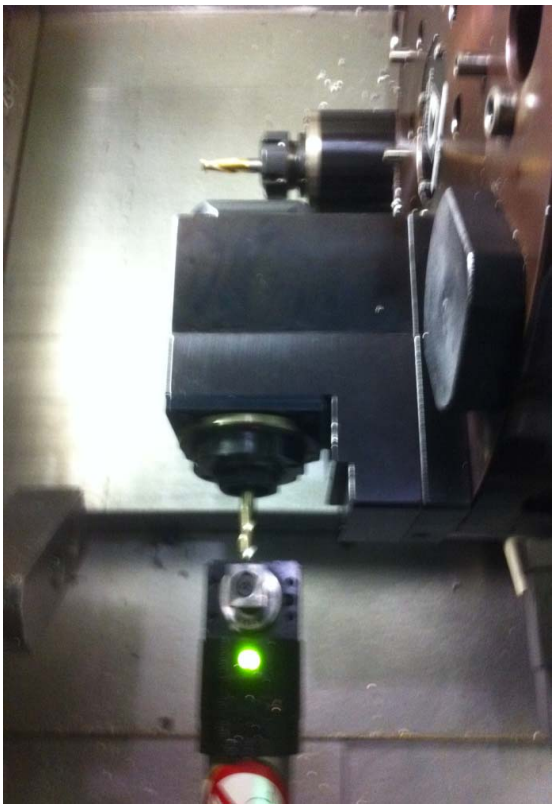


Figure 15–30. Radial live-tool X- Tool calibration

Axial Live Tools

With the tool setter advanced, select the appropriate softkey to set the tool in the selected orientation.

For TMM machines, setting an axial live tool in the -X direction changes the value in the Tool Setup, Geometry Offsets, X Offset field to 0.

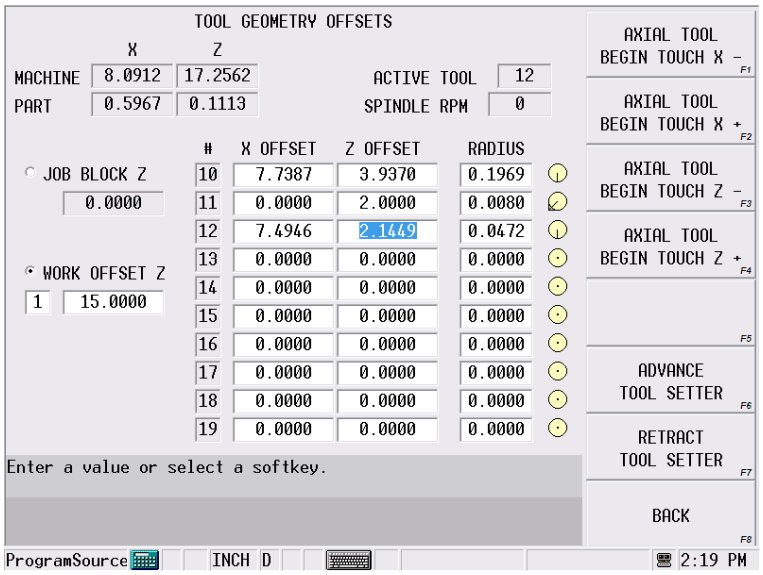


Figure 15–31. Axial Tool Setting screen

- **AXIAL TOOL BEGIN TOUCH X-** *F1*—begins axial tool calibration in the X- direction.
- **AXIAL TOOL BEGIN TOUCH X+** *F2*—begins axial tool calibration in the X+ direction.
- **AXIAL TOOL BEGIN TOUCH Z-** *F3*—begins axial tool calibration in the Z- direction.
- **AXIAL TOOL BEGIN TOUCH Z+** *F4*—begins axial tool calibration in the Z+ direction.

Here is a live axial tool being set in the X- direction:

Figure 15–32. Axial live-tool Z- Tool calibration

SUB-SPINDLE TOOL SETTER

The TMX MYS sub-spindle tool setter calibrates the tools with respect to the axes. The removable sub-spindle tool setter consists of the tool setter arm and stylus. A protective cap is securely positioned over the connection point on the sub-spindle housing while the tool setter is not installed. In addition, a storage stand is attached to the machine to hold the tool setter safely while not in use. The tool setter must be installed prior to its use, and subsequently, it must be removed and retuned to its storage stand after use.

This section describes how to install and remove the TMX MYS sub-spindle tool setter, and how to calibrate, align the stylus head, operate, and set tools using the sub-spindle tool setter.

The tool setter needs to be calibrated before you use it for the first time.

The tool setter requires re-calibration whenever

- The stylus is removed or replaced.
- Whenever an axis motor is replaced.
- If the axis home switch or absolute positions are reset or changed.
- If the turret is removed or loosened on the machine.
- If the setter head or arm is replaced.

The Y-axis must be set to 0.000 on the Tool Setter Calibration screen for all of the following procedures.

Installation

To install the sub-spindle tool setter arm,

1. Locate the sub-spindle tool setter cap on the sub-spindle housing.
2. Release the locking arm at the base of the cap to remove the cap.
3. Locate the pin on the sub-spindle tool setter device.
4. Insert the self-aligning pin into the hole made visible by removing the cap.
5. Move the sub-spindle tool setter locking arm into the locked position.

   ⚠ The green light on the back of the stylus illuminates when the device is properly installed.

Removal

Following the Sub-spindle tool setter operation and prior to machine operation, remove the arm from the sub-spindle housing:

1. Move the Sub-spindle tool setter locking arm into the unlocked position.
2. Remove the tool setter.
3. Return the tool setter to its storage stand location on the machine.
4. Replace the tool setter cap on the sub-spindle housing.
5. Secure the tool setter cap in place.

Sub-spindle Tool Setter Stylus Alignment

The TMX MYS Sub-spindle Tool Setter stylus head is located at the end of the tool setter arm. The removable tool setter arm must be attached securely to the Sub-spindle housing. Refer to *Installation, on page 15 - 43* and *Removal, on page 15 - 43* for details.

The sub-spindle tool setter stylus needs to be aligned correctly prior to tool calibration. To align the stylus,

1. Use the center point of a turning tool, positioned near the mid-point of the stylus head in the X-axis.
   a. Loosen the screw at the base of the stylus head to adjust the height so the center point of the tool is at the mid-point vertically of the stylus head.
   b. Tighten the screw when the stylus is positioned properly.
   c. Move the tool away from the stylus head.
2. Use a dial indicator mounted to the turret to position the stylus head so it is parallel to the Z-axis travel.
   a. Jog the axis so the indicator tip runs along the top face of the stylus head to show the stylus is square to 0.002mm (0.0001").
   b. Adjust the position of the stylus head with the two set screws located on the right-hand side of the tool setter arm, below the stylus head.

Sub-spindle Tool Setter Calibration

The TMX MYS Sub-spindle Tool Setter can be calibrated in any axis order (X first, then Z; or Z first, then X).

The tool setter needs to be calibrated before you use it for the first time.

The tool setter requires re-calibration whenever

- The stylus is removed or replaced.
- Whenever an axis motor is replaced.
- If the axis home switch or absolute positions are reset or changed.
- If the turret is removed or loosened on the machine.
- If the setter head or arm is replaced.

The Y-axis must be set to 0.000 on the Tool Setter Calibration screen for all of the following procedures.

Use a dowel pin for calibrating the sub-spindle tool setter.

Use a dowel pin of an appropriate length in order to avoid tools from colliding with the sub-spindle tool setter when the turret is indexed.

Access the Sub-spindle Tool Setter Calibration screen to perform the calibration.

1. Press the **Menu** console button.
2. Select Utilities Screens.
3. Select the SYSTEM CONFIGURATION *F1* softkey.
4. Select the HARDWARE CONFIGURATIONS *F3* softkey. The SUB-SPN TL SETTER CALIBRATION *F2* softkey appears if the sub-spindle tool setter is installed and operational.

   The Hardware Configurations screen appears with a Caution message. The message is there to alert users to be familiar with the operations available on the screen selections that follow.

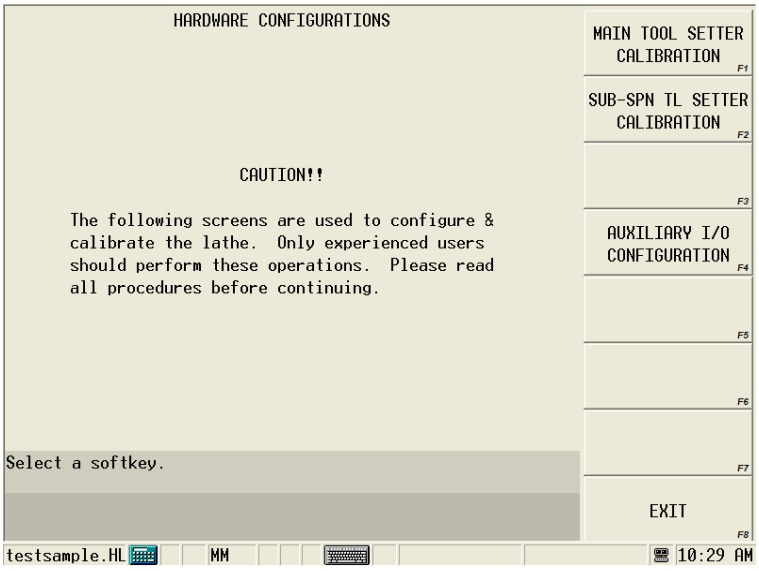


Figure 15–33. Hardware Configurations screen for TMX MYS with Sub-spindle Tool Setter

5. Select the SUB-SPN TL SETTER CALIBRATION *F2* softkey. The Sub-spindle Tool Setter Calibration screen appears:

To operate the Main spindle tool setter, use the Main Tool Setter Calibration *F1* softkey. Refer to *Tool Setter, on page 15 - 1*.

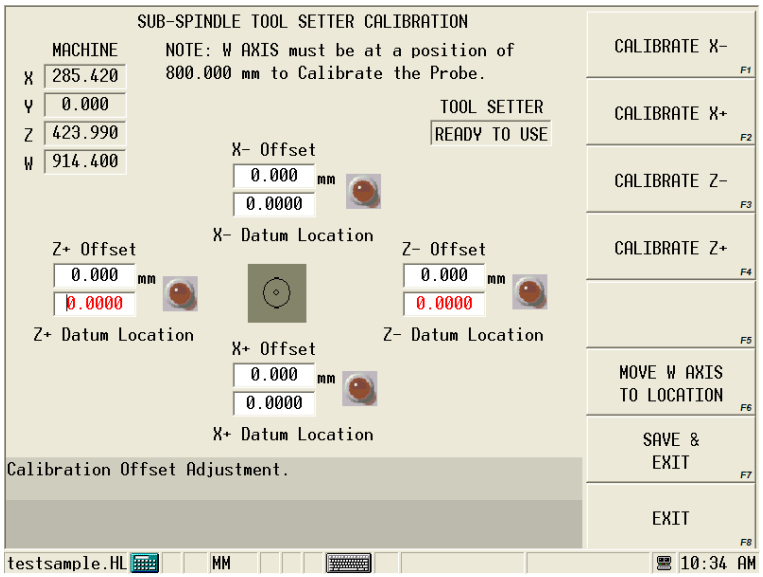


Figure 15–34. Sub-spindle Tool Setter Calibration screen

Position the W Axis

In order to perform sub-spindle tool setter calibration, the Machine W Axis field must be set to 800.000 mm. Follow these steps to achieve the accurate setting:

1. Select the MOVE W AXIS TO LOCATION *F6* softkey. The **Start Cycle** console key flashes.

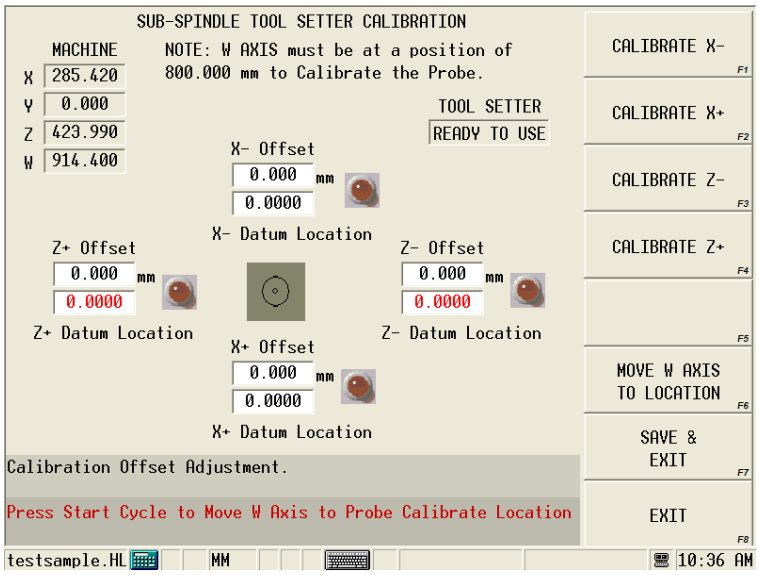


Figure 15–35. Sub-spindle Tool Setter Calibration Move W Axis

2. Press the **Start Cycle** console key. The W Axis moves to the 800.000 mm position, which is then displayed in the Machine W Axis field.

Sub-spindle Tool Setter Calibration Fields

The Sub-spindle Tool Setter Calibration screen fields are defined as follows:

- **Machine X Y Z W**—displays absolute machine position of X, Y, Z, and W axes.

  The Machine Y field must be set to 0.000 during Tool Setter Calibration.

- **Tool Setter**—displays current status of tool setter, either Unknown, Ready to Use, or Not Ready. The tool setter must be Ready to Use to complete calibration. Not Ready displays when the cap is in place on the sub-spindle housing. Unknown appears if the removable tool setter
- **Gray Square**—(center of screen) represents the face of the tool setter stylus. Use this icon as a reference for the Offsets and Datum Locations.
- **X- Offset**—value represents the distance from 0 (the center of the tool holder), equal to the radius of the test bar. The offset must be a positive number in millimeters.

- **X- Datum Location**—displays the machine position of the X-axis face of the top of the stylus head. The default is zero if uncalibrated. If calibrated, the X-Offset is subtracted from the machine position to obtain X- Datum.
- **X+ Offset**—value represents the distance from 0 (the center of the tool holder), equal to the radius of the test bar. The offset must be a positive number in millimeters. The X-Offset should be equal to the X+ Offset.
- **X+ Datum Location**—displays the machine position of the X-axis face of the bottom of the stylus head. The default is zero if uncalibrated. If calibrated, the X+ Offset is subtracted from the machine position to obtain X+ Datum.
- **Z+ Offset**—value represents the distance from the face of the turret to the surface of the plate. The left side of the precision plate must be aligned with the turret face. The offset must be a positive number in millimeters.
- **Z+ Datum Location**—displays the machine position of the left Z-axis face of the stylus head. The default is zero if uncalibrated. If calibrated, the Z+ Offset is subtracted from the machine position to obtain Z+ Datum. A value within the range of 550 mm to 800 mm must be entered.
- **Z- Offset**—value represents the distance from 0, equal to the distance from the face of the turret to the surface of the plate. Set to zero if the left side of the test plate is mounted flush with the turret.
- **Z- Datum Location**—displays the machine position of the Z-axis face of the stylus head on the right side. The default is zero if uncalibrated. If calibrated, the Z- Offset is subtracted from the machine position to obtain Z- Datum. A value within the range of 550 mm to 800 mm must be entered.
- One LED icon appears next to each grouping of Offset fields and Datum Location fields:
  - A green icon means that the calibration process is successful.
  - A red icon means that the calibration process is not successful. If this occurs, repeat the calibration process. If the calibration process continues to be unsuccessful, move the start point closer or further from the stylus head.

X-Axis Sub-spindle Tool Setter Calibration

Calibrate the X- and X+ offset as described below.

The Machine Y field must be set to 0.000 during Tool Setter Calibration.

X- Axis Sub-spindle Tool Setter Calibration

1. Position the axes at Home Position.
2. Use the **Turret -** or **+** key to rotate the turret to the desired location for installing the gauge pin in a boring block.
3. Attach the boring block to the turret and insert and securely attach the dowel pin into the holder.
4. Check the dowel pin for parallelism to the machine. Run an indicator back and forth across the dowel pin, verifying the pin is parallel to the Z-axis.

   The dowel pin should not be out of alignment by any more than a couple of tenths. Make any necessary adjustments before proceeding to the next step.

5. Move the axes back to the Home Position (upper right-hand corner of the machine).
6. Enter the offset into the **X- Offset** field. The offset is the distance from X Zero (the face of the turret with the tool holder mounted) to the center of the tool holder plus the pin radius. In the example below, the X- Offset is: (100mm + Dowel Pin Radius 6.35mm) = 106.35mm.

   The X- Offset value is always a positive number in millimeters.

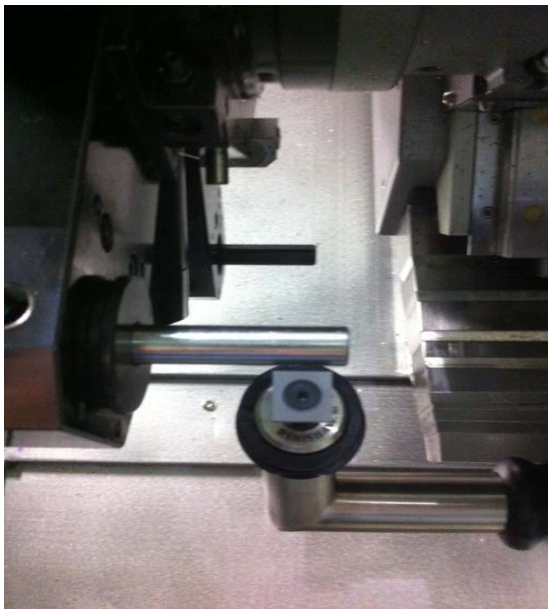


Figure 15–36. X-axis Position for X- Sub-spindle Tool Setter Calibration

7. Manually jog the axes toward the tool setter, positioning the side of the dowel pin to within 10 mm (0.4") of the **top** of the tool setter head. This is the start position required for performing the X- Calibration process.

   ⚠ If the dowel pin is positioned more than 10 mm (0.4") from the tool setter head, the calibration move may time-out.

8. Select the CALIBRATE X- *F1* softkey. The **Start Cycle** button flashes.

   ⚠ To achieve accurate results, the axis must touch the stylus at a slow feedrate.

   - Set the **Axis Feed Rate** percentage between 10% and 20%.
   - Set the **Rapid Override** percentage at 100%.

9. Press the flashing **Start Cycle** button. The X axis moves slowly toward the tool setter head at 100 mmpm (4 ipm).

   a. When the dowel pin touches the tool setter head, the axis moves at rapid speed at approximately 10 mm in the opposite direction, then touches the tool setter head two more times.

   b. After the third touch, the axis moves at rapid speed to the X- Offset Calibration start position.

   ⚠
   - If the X- Offset Calibration is successful, the green LED icon is displayed and the machine position minus the X- Offset is displayed in the X-Datum field.
   - If the X- Offset Calibration is not successful, the red LED icon is displayed
   - If the dowel pin was positioned more than 10 mm from the tool setter head, the system will generate a fault. Reposition the gauge pinto within 10 mm of the tool setter head and repeat the steps above.

X+ Sub-spindle Tool Setter Calibration

1. Position the axes at Home Position.
2. Use the **Turret -** or **+** key to rotate the turret to the desired location for installing the test bar in a boring block.
3. Attach the boring block to the turret and insert and securely attach the dowel pin into the holder.

   Measure and record the diameter of the dowel pin.

4. Check the dowel pin for parallelism to the machine. Run an indicator back and forth across the test bar, verifying the pin is parallel to the Z-axis.

   The dowel pin should not be out of alignment by any more than a couple of tenths. Make any necessary adjustments before proceeding to the next step.

5. Move the axes back to the Home Position (upper right-hand corner of the machine).
6. Enter the offset into the **X+ Offset** field. The offset is the distance from X Zero (the face of the turret with the tool holder mounted) to the center of the tool holder minus the pin radius. In the example below, the X+ Offset is: (100mm - Dowel Pin Radius 6.35mm) = 93.65mm.

   The X+ Offset value is always a positive number in millimeters.

7. Manually jog the axes toward the tool setter, positioning the side of the dowel pin to within 10 mm (0.4") of the **bottom** of the tool setter head. This is the start position required for performing the X+ Calibration process.

   If the dowel pin is positioned more than 10 mm (0.4") from the tool setter head, the calibration move may time-out.

8. Select the CALIBRATE X+ *F2* softkey. The **Start Cycle** button flashes.

   To achieve accurate results, the axis must touch the stylus at a slow feedrate.

   - Set the **Axis Feed Rate** percentage at 100%.
   - Set the **Rapid Override** percentage at 100%.

9. Press the flashing **Start Cycle** button. The X axis moves slowly toward the tool setter head at 60 mmpm (2.362 ipm).
   a. When the dowel pin touches the tool setter head, the axis moves at rapid speed at approximately 10 mm in the opposite direction, then touches the tool setter head two more times.
   b. After the third touch, the axis moves at rapid speed to the X+ Offset Calibration start position.

   - If the X+ Offset Calibration is successful, the green LED icon is displayed and the machine position minus the X+ Offset is displayed in the X- Datum field.
   - If the X+ Offset Calibration is not successful, the red LED icon is displayed
   - If the dowel pin was positioned more than 10 mm from the tool setter head, the system will generate a fault. Reposition the test bar to within 10 mm of the tool setter head and repeat the steps above.

X Calibration Adjustment

The difference between X+ Datum Location and X- Datum Location should be approximately the length of the stylus head in X. To validate the datum,

1. Cut a test part using a Turning operation with a tool that has been calibrated with the tool setter.
2. Measure the turned part and compare to the programmed diameter.
   - If the test part is under-size, increase the X+ or X- Datum values by half the difference of the measured diameter and the programmed diameter.
   - If the test part is over-size, decrease the X+ or X- Datum values by half the difference of the measured diameter and the programmed diameter.

Z-Axis Sub-spindle Tool Setter Calibration

Calibrate the Z- and Z+ offset as described in the appropriate section below:

The Machine Y field must be set to 0.000 during Tool Setter Calibration.

Z- Sub-spindle Tool Setter Calibration

1. Position the axes at Home Position.
2. Use the **Turret -** or **+** key to rotate the turret to the desired location for installing the test plate in a slot in the turret.
3. Check the dowel pin for parallelism to the machine. Run an indicator back and forth across the test bar, verifying the dowel pin is parallel to the X-axis.

   The dowel pin should not be out of alignment by any more than a couple of tenths. Make any necessary adjustments before proceeding to the next step.

4. Move the axes back to the Home Position (upper right-hand corner of the machine).
5. Enter the offset into the **Z- Offset** field. The offset is the centerline of the tool holder, which is equal to the radius of the dowel pin. In this example, the Z- and Z+ Offset is 6.35mm.

   The Z- Offset value is always a positive number in millimeters.

Figure 15–37. Z-axis Position for Z- Sub-spindle Tool Setter Calibration

6. Manually jog the axes toward the tool setter, positioning the side of the dowel pin to within 10 mm (0.4") of the right-hand side of the tool setter head. This is the start position required for performing the Z- Calibration process.

   ⚠ If the dowel bar is positioned more than 10 mm (0.4") from the tool setter head, the calibration move may time-out.

7. Select the CALIBRATE Z- *F3* softkey. The **Start Cycle** button flashes.

   ⚠ To achieve accurate results, the axis must touch the stylus at a slow feedrate.

   - Set the **Axis Feed Rate** percentage between 10% and 20%.
   - Set the **Rapid Override** percentage at 100%.

8. Press the flashing **Start Cycle** button. The Z axis moves slowly toward the tool setter head at 100 mmpm (4 ipm).

   a. When the dowel bar touches the tool setter head, the axis moves at rapid speed at approximately 10 mm in the opposite direction, then touches the tool setter head two more times.

   b. After the third touch, the axis moves at rapid speed to the Z- Offset Calibration start position.

   ⚠
   - If the Z- Offset Calibration is successful, the green LED icon is displayed and the machine position minus the Z- Offset is displayed in the Z- Datum field.
   - If the Z- Offset Calibration is not successful, the red LED icon is displayed
   - If the dowel pin was positioned more than 10 mm from the tool setter head, the system will generate a fault. Reposition the test bar to within 10 mm of the tool setter head and repeat the steps above.

Z+ Sub-spindle Offset Calibration

1. Position the axes at Home Position.
2. Use the **Turret -** or **+** key to rotate the turret to the desired location for installing the test plate in a slot in the turret.
3. Check the dowel pin for parallelism to the machine. Run an indicator back and forth across the test bar, verifying the dowel pin is parallel to the X-axis.

   The dowel pin should not be out of alignment by any more than a couple of tenths. Make any necessary adjustments before proceeding to the next step.

4. Move the axes back to the Home Position (upper right-hand corner of the machine).
5. Enter the offset into the **Z+ Offset** field. The offset is the centerline of the tool holder, which is equal to the radius of the dowel pin. In this example, the Z- and Z+ Offset is 6.35mm.

   The Z+ Offset value is always a positive number in millimeters.

Figure 15–38. Z-axis Position for Z+ Sub-spindle Tool Setter Calibration

6. Manually jog the axes toward the tool setter, positioning the side of the dowel pin to within 10 mm (0.4") of the left-hand side of the tool setter head. This is the start position required for performing the Z- Calibration process.

   If the dowel pin is positioned more than 10 mm (0.4") from the tool setter head, the calibration move may time-out.

7. Select the CALIBRATE Z+ *F4* softkey. The **Start Cycle** button flashes.

   To achieve accurate results, the axis must touch the stylus at a slow feedrate.

   - Set the **Axis Feed Rate** percentage between 10% and 20%.
   - Set the **Rapid Override** percentage at 100%.

8. Press the flashing **Start Cycle** button. The Z axis moves slowly toward the tool setter head at 100 mmpm (4 ipm).

   a. When the dowel pin touches the tool setter head, the axis moves at rapid speed at approximately 10 mm in the opposite direction, then touches the tool setter head two more times.

   b. After the third touch, the axis moves at rapid speed to the Z+ Offset Calibration start position.

   - If the Z+ Offset Calibration is successful, the green LED icon is displayed and the machine position minus the Z+ Offset is displayed in the Z+ Datum field.
   - If the Z+ Offset Calibration is not successful, the red LED icon is displayed
   - If the test bar was positioned more than 10 mm from the tool setter head, the system will generate a fault. Reposition the test bar to within 10 mm of the tool setter head and repeat the steps above.

Z Calibration Adjustment

The difference between Z+ Datum Location and Z- Datum Location should be approximately the width of the stylus head in Z. To validate the datum,

1. Face and cut-off or part-off a test part using a tool that has been calibrated with the tool setter.
2. Measure the length of the part and compare to the programmed length.
   - If the test part is under-sized, increase the Z+ Datum value by the difference of the measured length and the programmed length.
   - If the test part is over-sized, decrease the Z+ Datum value by the difference of the measured length and the programmed length.

Sub-spindle Tool Setter—Setting Tools

With the Sub-spindle Tool Setter option, from the Tool Setup screen, select the Geometry Offsets *F3* softkey followed by the Tool Setter *F6* softkey.

Refer to *Installation, on page 15 - 43* for information about preparing the Tool Setter for operation.

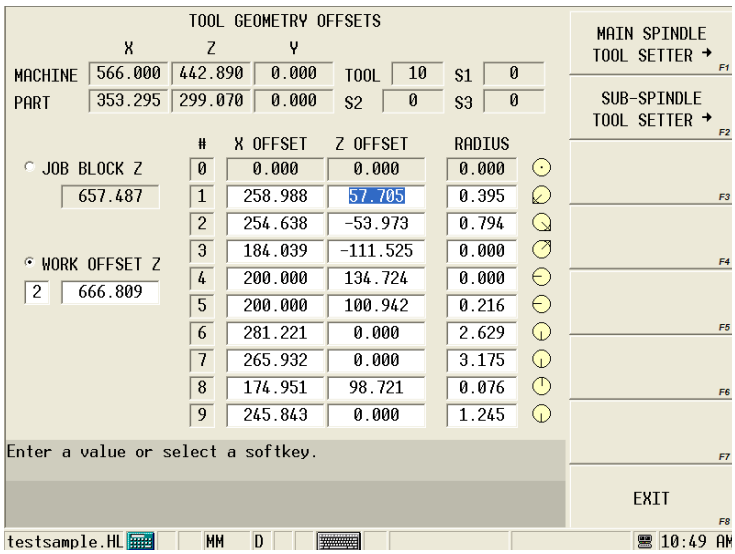


Figure 15–39. Tool Setup Sub-Spindle Tool Setter softkey

Select the SUB-SPINDLE TOOL SETTER Softkey to access the softkey selections. This screen appears:

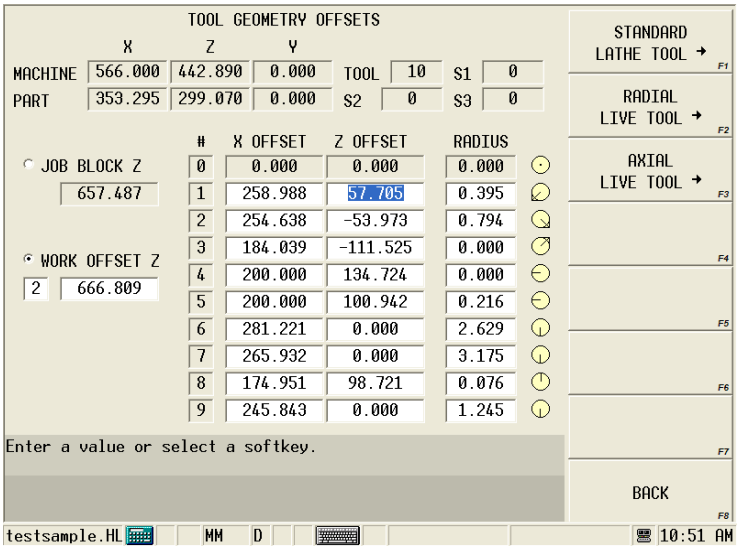


Figure 15–40. Tool Setter softkey menu

Softkeys are defined as follows:

- **STANDARD LATHE TOOL** *F1*—accesses the softkey menu for sub-spindle tool setter, standard tool calibration.
- **RADIAL LIVE TOOL** *F2*—accesses the softkey menu for sub-spindle tool setter, radial live tool calibration.
- **AXIAL LIVE TOOL** *F3*—accesses the softkey menu for sub-spindle tool setter, axial live tool calibration.
- **BACK** *F8*—returns to the Tool Geometry Offsets screen.

Tool Setting Operation

Follow these steps to use the tool setter to set tools:

1. Position the axes at Home Position.
2. Manually jog the axes toward the tool setter, positioning the side of the tool to within 10 mm (0.4") of the tool setter head, either the top (X-), bottom (X+), left-hand (Z+), or right-hand (Z-) side. This is the start position required for performing the calibration process.

   If the tool is positioned more than 10 mm (0.4") from the tool setter head, the calibration move may time-out.

3. Select the appropriate TOUCH softkey. The **Start Cycle** button flashes.

   For information about calibrating each tool type refer to:

   - *Standard Lathe Tools and Sub-spindle Tool Setter, on page 15 - 60*
   - *Radial Live Tools and Sub-spindle Tool Setter, on page 15 - 61*
   - *Axial Live Tools and Sub-spindle Tool Setter, on page 15 - 62*

   To achieve accurate results, the axis must touch the stylus at a slow feedrate.

   - Set the **Axis Feed Rate** percentage between 10% and 20%.
   - Set the **Rapid Override** percentage at 100%.

4. Press the flashing **Start Cycle** button. The selected axis moves slowly toward the tool setter head at 100 mmpm (4 ipm).

   a. When the tool touches the tool setter head, the axis moves at rapid speed at approximately 10 mm in the opposite direction, then touches the tool setter head two more times.

   b. After the third touch, the axis moves at rapid speed to the start position.

   - If the Calibration is successful, the offset is entered into the appropriate Offset field.
   - If the tool was positioned more than 10 mm from the tool setter head, the system will generate a fault. Reposition the tool to within 10 mm of the tool setter head and repeat the steps above.
   - Set the X Offset field in Tool Setup, Geometry Offsets prior to calibrating for the tool radius.
   - If an X Axis entry is in Diameter mode, then double the value before entering it into the Tool Setup, Geometry Offsets, X Offset field.

Standard Lathe Tools and Sub-spindle Tool Setter

Select the appropriate softkey to set the standard lathe tool in the selected orientation. After a softkey is selected the **Start Cycle** button flashes. Refer to *Tool Setting Operation, on page 15 - 59* for details about axis movement.

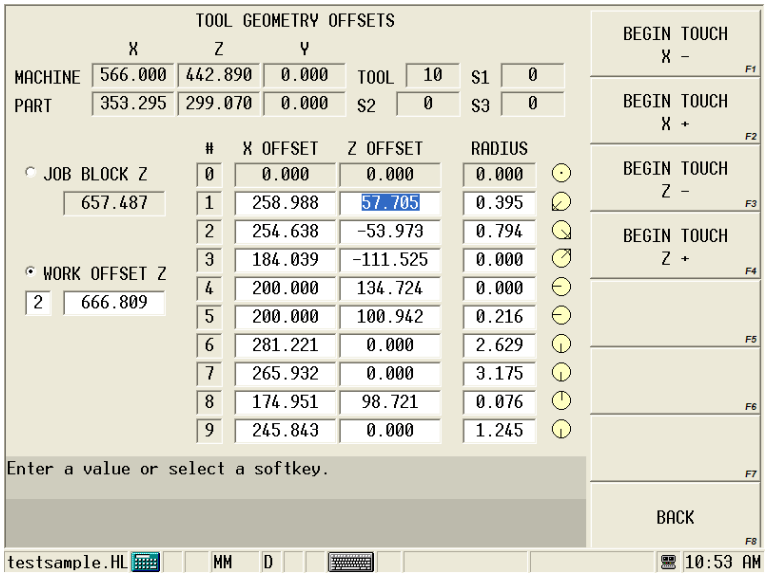


Figure 15–41. Standard Tool Setting screen

- **BEGIN TOUCH X-** *F1*—begins tool calibration in the X- direction.
- **BEGIN TOUCH X+** *F2*—begins tool calibration in the X+ direction.
- **BEGIN TOUCH Z-** *F3*—begins tool calibration in the Z- direction.
- **BEGIN TOUCH Z+** *F4*—begins tool calibration in the Z+ direction.

Radial Live Tools and Sub-spindle Tool Setter

Select the appropriate softkey to set the tool in the selected orientation. After a softkey is selected the **Start Cycle** button flashes. Refer to *Tool Setting Operation, on page 15 - 59* for details about axis movement.

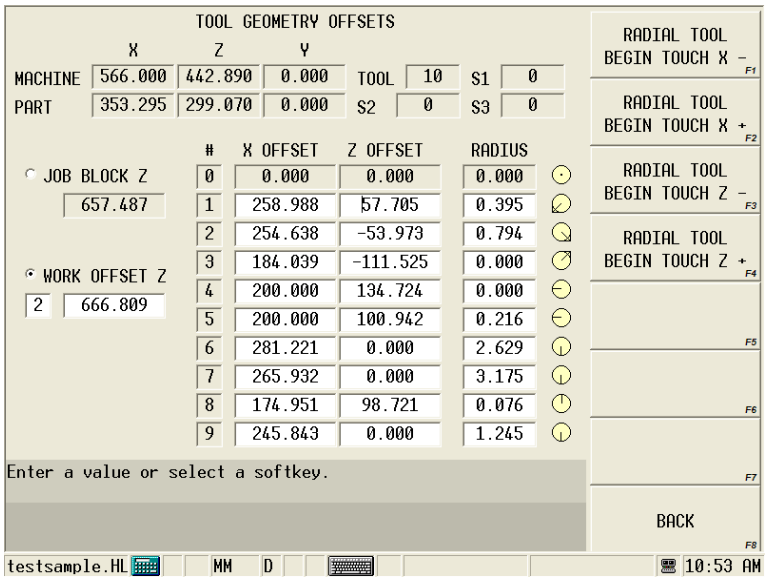


Figure 15–42. Radial Tool Setting screen

- **RADIAL TOOL BEGIN TOUCH X-** *F1*—begins radial tool calibration in the X-direction.
- **RADIAL TOOL BEGIN TOUCH X+** *F2*—begins radial tool calibration in the X+ direction.
- **RADIAL TOOL BEGIN TOUCH Z-** *F3*—begins radial tool calibration in the Z-direction.
- **RADIAL TOOL BEGIN TOUCH Z+** *F4*—begins radial tool calibration in the Z+ direction.

Axial Live Tools and Sub-spindle Tool Setter

Select the appropriate softkey to set the tool in the selected orientation. After a softkey is selected the **Start Cycle** button flashes. Refer to *Tool Setting Operation, on page 15 - 59* for details about axis movement.

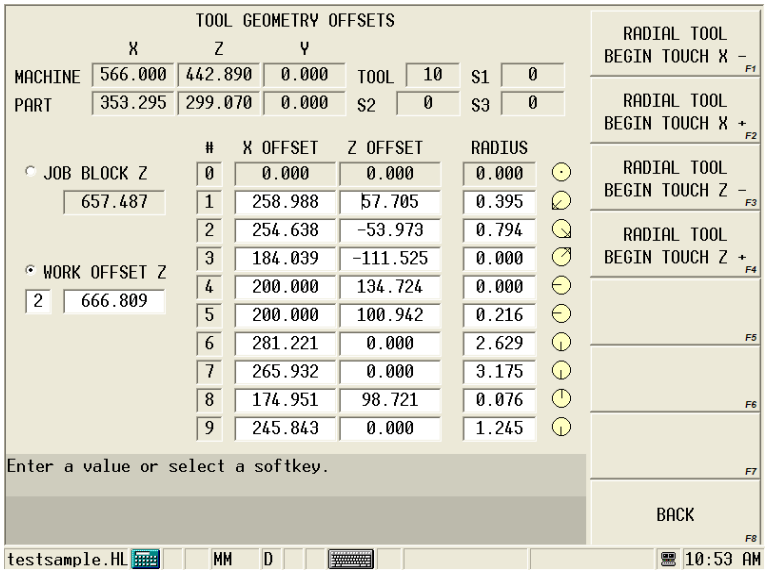


Figure 15–43. Axial Tool Setting screen

- **AXIAL TOOL BEGIN TOUCH X-** *F1*—begins axial tool calibration in the X- direction.
- **AXIAL TOOL BEGIN TOUCH X+** *F2*—begins axial tool calibration in the X+ direction.
- **AXIAL TOOL BEGIN TOUCH Z-** *F3*—begins axial tool calibration in the Z- direction.
- **AXIAL TOOL BEGIN TOUCH Z+** *F4*—begins axial tool calibration in the Z+ direction.

ULTIMONITOR

The UltiMonitor option adds capability and flexibility to your WinMax Lathe operation by providing connection to your Local Area Network (LAN). Using UltiMonitor, you can communicate with other CNCs, and with PCs or file servers connected to your LAN using standard TCP/IP and FTP protocols. UltiMonitor also includes Extended Shop Floor (ESF) for remote machine monitoring and communication.

This section covers the use of the UltiMonitor product. For information about basic system operation, refer to Getting Started with WinMax Lathe*, Programming Basics, on page 4 - 1*.

⇨ Customers using UltiNet should refer to this section for product guidance. The UltiNet option provides the same FTP connectivity as UltiMonitor but does not include ESF.

LAN Requirements

To use UltiMonitor, you must have a Local Area Network (LAN or “network”) that supports the IEEE 802.3 Ethernet hardware standards. Contact your IT provider for guidance about proper LAN design and setup.

Limitations for UltiMonitor

1. Do NOT connect machines to a domain. When connecting a machine to a network, a workgroup must be used. WinMax software runs on an embedded Windows operating system, and any changes to the system could cause unwanted effects.
2. Installing any physical cards, including wireless devices, is NOT supported. WinMax is designed and configured to operate with the existing system hardware.
3. Use of an Ethernet cable connection to a machine is required.

Glossary of Networking Terms

Following are definitions for networking terms used throughout this section.

- **Client / Server—**This is a term commonly used to describe how computers do work for each other. One computer runs server software; the other computer runs client software. The server software waits for requests from the client in order to perform tasks. Many times the server is a background task that runs all the time. A client is a user application on a PC or some other workstation and usually will not run all the time. A user may begin running client software, do some work, then shut down the client when finished. It is important to note that a single server may handle many clients. It is also possible for a computer to function as both a client and a server simultaneously.
- **Ethernet—**Ethernet is a particular networking technology specification. Ethernet uses CSMA/CD to gain access to the network media (cable). CSMA/CD stands for Carrier Sense Multiple Access with Collision Detection. Ethernet sends its data on a carrier wave. Since there are multiple Ethernet adapters attached to the same cable, they must take turns sending data. When ready to transmit, an adapter checks for the presence of a carrier wave. If no carrier is detected, the adapter begins transmission. If another adapter begins transmitting data at the same time, a collision will occur. The Ethernet hardware senses when a collision occurs. The two adapters involved in the collision will stop, delay for a random length of time, and then try to retransmit the data. This process repeats until the data is successfully sent. Data is sent in the form of packets, which have a limited size, allowing other hosts an opportunity to use the network media.
- **FTP—**This is an acronym for File Transfer Protocol. FTP is implemented using the client/server model and uses TCP/IP as its protocols. FTP software contains a standard set of commands that are used to transfer files and manage directory structures.

- **Host—**Any computer connected to a network can be a host, including a PC running DOS, Windows XP/Vista/7, Windows NT; a UNIX workstation; a mainframe computer; a CNC control; or any machine with networking capability.
- **IP Address—**An IP (Internet Protocol) address is used to identify a particular host on a network. Each host on a network must have a unique IP address consisting of a 32-bit number usually presented in dotted decimal format; for example, 200.100.150.1. This format divides the address into four single byte values separated by decimal points. On most networks, the first three bytes represent the network and the last byte is the host. Following this practice, the first three bytes are the same for all hosts and the last byte is different for each host. On the Internet, these cryptic addresses usually are not used—most addresses are represented as plain text, which are converted to IP addresses by DNS, or Domain Name Server. The underlying protocol still uses the unique 32-bit IP addresses.
- **TCP/IP—**This is an acronym for the two major protocols of the Internet, Transmission Control Protocol and Internet Protocol. IP data normally reaches its intended destination, but there is no failure notification for the sender if it does not. TCP, however, has an acknowledgment that the message was received. The TCP layer of a protocol stack uses IP to send a message in the form of packets, or bundles of data. Each packet must be acknowledged or it will be sent again. This process repeats until the entire message, which contains multiple packets, is received. TCP is defined as a reliable stream-oriented protocol. The use of TCP/IP is not limited to the Internet. It is commonly used for communication between hosts on LANs (local area networks). It may be used for any application that requires reliable data transfer.

Configuring a Network

UltiMonitor allows machines to be connected on a Local Area Network (LAN or “network”).

Configuring an IP address for your machine

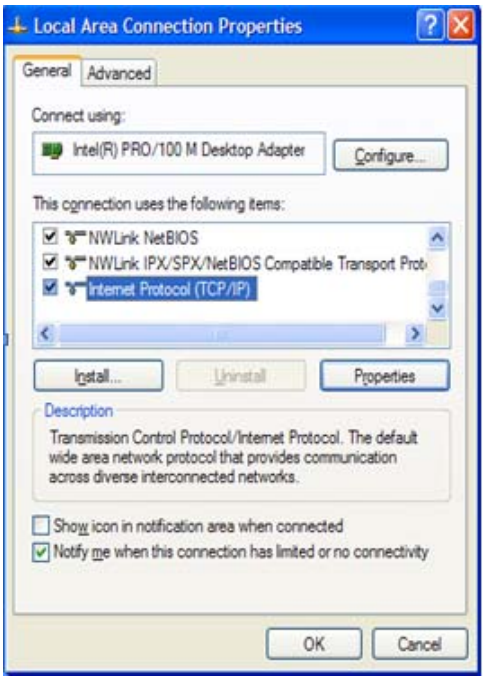
An Internet Protocol (IP) address is a numerical label assigned to each device in a computer network. IP addresses are assigned to a host either at the time of booting, or permanently by fixed configuration of the hardware or software. Persistent configuration is also known as using a static IP address. In contrast, a dynamic IP address occurs when a device’s IP address is assigned with each reboot. All Hurco machines, by default, are configured to use dynamic IP addresses. This is shown in the figure below.

The configuration can be changed to a static IP address, if desired. IP addresses must always be unique to your network; contact your network administrator for the proper IP address values.

To access the LAN and Internet Protocol properties:

1. Select the Windows key (or the Ctrl+Esc keys) on the keyboard to access system settings through the Shell.
2. Expand the left pane of the Shell.
3. Select **Control Panel**.
4. Select **Network Connections**.
5. Enter password.
6. Highlight **Local Area Connection**.
7. Select File/Properties. (See *Figure 16–1. Default LAN and Internet Protocol Properties, on page 16 - 5*.)
8. Highlight **Internet Protocol (TCP/IP)**.
9. Select **Properties**. The IP address configuration can be changed in the TCP/IP Properties box that opens. (See *Figure 16–1. Default LAN and Internet Protocol Properties, on page 16 - 5*.)

a.



b.

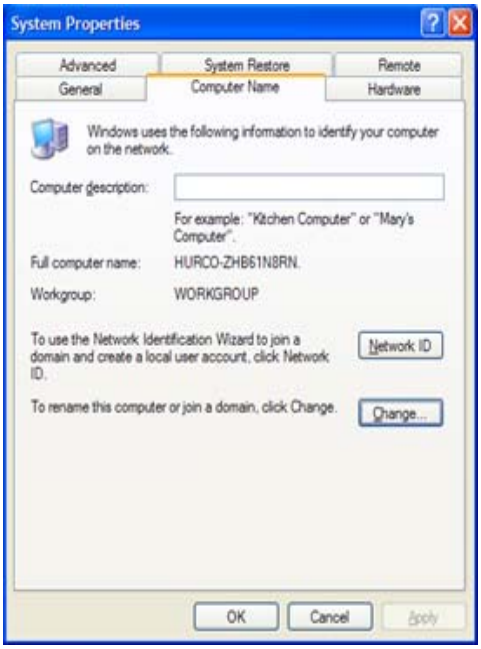
Figure 16–1. Default LAN and Internet Protocol Properties

Configuring the Computer and Workgroup Names

A computer name is an easy way to locate a networked device when connected to a local network, as an alternative to an IP address. Like IP Addresses, it is very important that these names are unique. All Hurco machines are, by default, configured with unique computer names. Although the default name is unique, it is not descriptive or easy to remember, so many users may wish to change this value. The computer name is changed in System Properties. To access:

1. Select the Windows key (or the Ctrl+Esc keys) on the keyboard to access system settings through the Shell.
2. Expand the left pane of Shell.
3. Select **Control Panel**.
4. Select **System**.
5. Enter password.
6. Select **Computer Name** tab. (See *Figure 16–2. Computer Name Changes in System Properties, on page 16 - 6*.)
7. Select **Change**.
8. Enter new name in the **Computer name** field. Select OK. (See *Figure 16–2. Computer Name Changes in System Properties, on page 16 - 6*.)

a.



b.

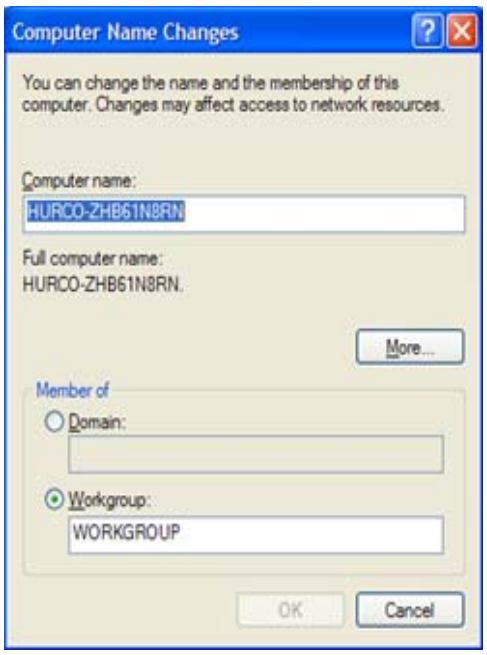


Figure 16–2. Computer Name Changes in System Properties

While in this screen, the name of the workgroup or domain to which this machine is a member may also be changed. Hurco DOES NOT support connections to network domains. A workgroup can be used as a logical grouping of networked devices for a particular purpose. It is easiest to place all devices under a particular workgroup but is not required or necessary. By default, all Hurco controls are configured to be on the workgroup named **WORKGROUP**.

Mapping a Network Drive

A server share is basically a folder on a different computer that is being shared with everyone else. So when you “map a drive,” you are saying that you want access to that folder on your computer also, which is done by mapping it to a letter, for example, F, G, H, etc. To access the Map Network Drive dialog:

1. Select the Windows key (or the Ctrl+Esc keys) on the keyboard to access system settings through the Shell.
2. Expand right pane of Shell.
3. Select **Map Drive**.
4. Enter password. The Map Network Drive dialog box opens:

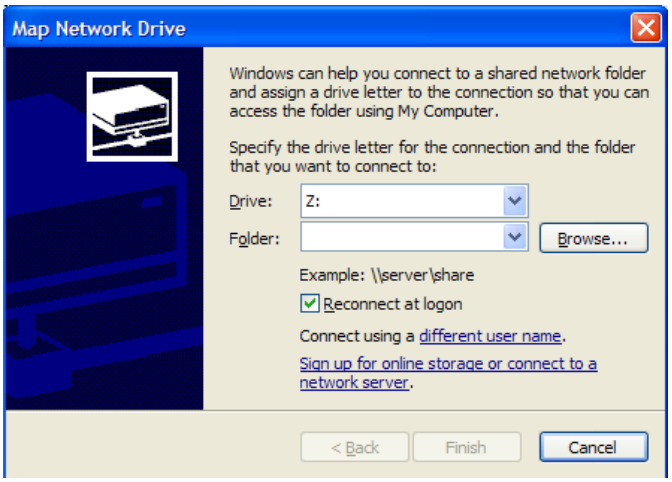


Figure 16–3. Map Network Drive dialog

To map a network drive:

1. Select an unused Drive letter to represent the shared folder and type in the UNC path in the Folder field. UNC path is a special format for pointing to a folder on another computer. The format is **\\computer name\folder name**.

   If you're not sure what the name of the folder is, you can select **Browse...** to find the computer that way. Select **Entire Network**, then **Microsoft Windows Network** and expand the workgroup that your computer is in.

2. Click **Reconnect at logon** to make the connection permanent, which means the drive will still be mapped even after you restart the computer.

   ⇨ If the drive is password protected, use the “Connect using a different user name” link in the mapping window to configure the mapping with the user name and password.

Using FTP

FTP (File Transfer Protocol) is a method of transferring files from one computer to another, using the Internet. For Hurco machining and turning centers, FTP is required for transferring programs between two or more machines. FTP can also be used to transfer programs between a computer and a machine.

FTP Server Settings

To access the FTP Server Settings screen:

1. Press the Menu console key.
2. Select the Utilities softkey to access the Utilities screen and softkey menu.
3. Select the User Preferences softkey.
4. Select the More softkey.
5. Select the FTP Server Settings softkey. The screen is displayed:

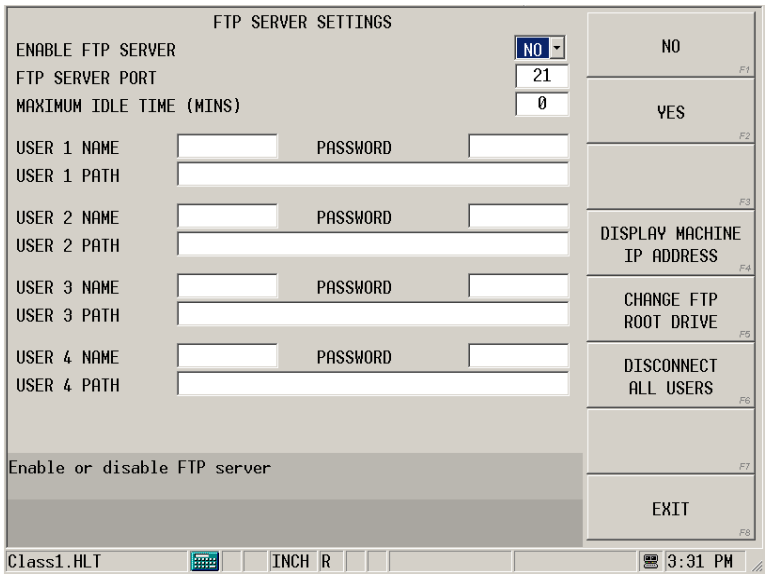


Figure 16–4. FTP Server Settings screen

FTP Server Settings Fields

These fields are available on the FTP Server Settings screen:

- **Enable FTP Server**—select Yes to enable or No to disable the FTP server.
- **FTP Server Port**—enter a number for the FTP server port. The default is 21.
- **Maximum Idle Time (mins)**—enter the number of idle time minutes before the server disconnects. The default is 0 with no time out.

- **User 1-4 Name**—enter each user's logon name. Up to 4 users can logon.
- **User 1-4 Password**—enter each user's logon password. Up to 4 users can logon.

  If the logon name is "anonymous," the password is not required.

- **User 1-4 Path**—enter each user's root path. Up to 4 users can logon.

FTP Server Settings Softkeys

These softkeys are available on the FTP Server Settings screen:

- **Display WinMax IP Address**—displays the IP address list in a pop-up window. Select OK to close the window.

  An IP (Internet Protocol) address is used to identify a particular host on a network. Each host on a network must have a unique IP address consisting of a 32-bit number usually presented in dotted decimal format; for example, 200.10.150.1. This format divides the address into four single byte values separated by decimal points.

  On most networks, the first three bytes represent the network and the last byte is the host. Following this practice, the first three bytes are the same for all hosts and the last byte is different for each host.

  On the Internet, these cryptic addresses usually are not used—most addresses are represented as plain text, which are converted to IP addresses by DNS, or Domain Name Server. The underlying protocol still uses the unique 32-bit IP addresses.

- **Change FTP Root Drive**—displays a pop-up window to browse for a folder. Select the appropriate root drive and select OK to close the window. Select Cancel to stop browsing without changing the root drive.
- **Disconnect All Users**—select Yes in the pop-up window to disconnect all current FTP users; select No to cancel the disconnect operation.

FTP Manager

To manage FTP connections:

1. Press the Input console key.
2. Select the Project Manager softkey.
3. Select the FTP Manager softkey to open the **FTP Host List** screen. From here, you can connect to or disconnect from, add, edit, and delete FTP servers which are identified on this list:

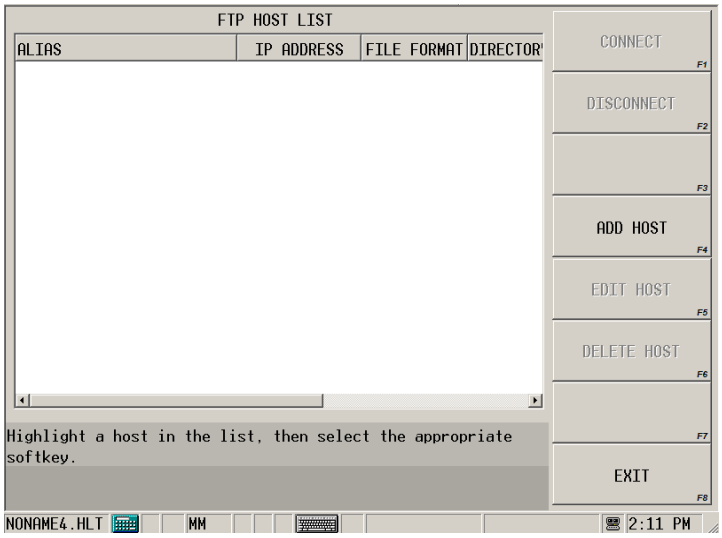


Figure 16–5. FTP Host List screen

To access the FTP Host Properties screen to add a host to the FTP Host List screen:

1. Select the Add Host softkey. The FTP Host Properties screen appears.

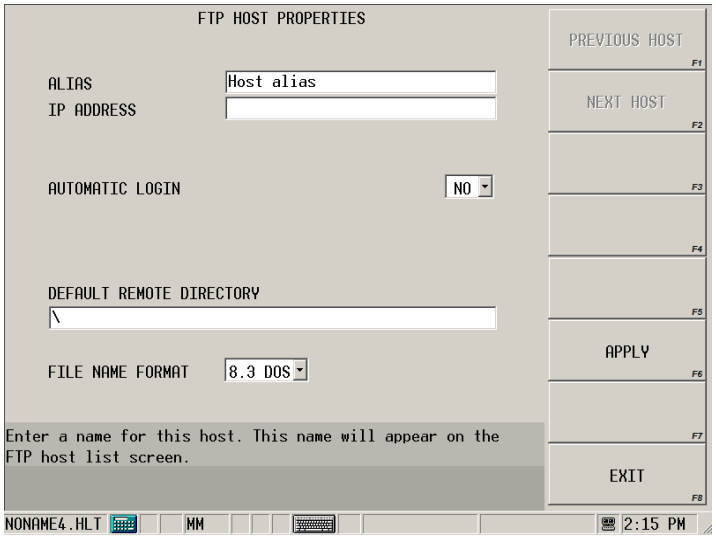


Figure 16–6. FTP Host Properties screen

2. Enter an Alias name to appear on the FTP Host List screen for this host.

3. Enter an IP Address for connecting to the host.
4. Specify Yes or No for Automatic Login:
   - If set to No, the user is required to enter the username and password at the time of connection.
   - If set to Yes, the User Name and Password fields appear. The username and password are stored for automatic connection when the host is selected.
5. Enter the Default Remote Directory to be opened, or leave this field blank for the FTP server's root directory.
6. Select either 8.3 DOS or LONG for the Filename Format field.
   - 8.3—file names with eight characters before the period (.) and three characters for an extension after the period are allowed.
   - Long—the complete path to the file, including the drive letter, server name, folder path, and file name and extension can contain up to 255 characters – however, it will be truncated to the 8.3 format. For example, LongFilename.txt will be truncated to LongFil_.txt
7. Select the Apply softkey to add this host to the FTP Host List screen.

From the FTP Host List screen,

- Select the Connect softkey to connect to a host.
- Select Disconnect to disconnect from a host.
- Select Delete Host to remove a host from the list.
- Select the Exit softkey to return to the Project Manager screen.

Select the File Manager softkey to access a list of system directories and filenames connected with UltiMonitor.

After editing, files may be saved directly to or from the remote location:

1. Select the Save or Save As softkey in Program Manager.
2. Select the FTP Manager softkey. The screen will switch to the remote File Manager, if connected, or to the login screen if not connected.
3. When connected, select the appropriate remote directory and save the file.

Extended Shop Floor (ESF)

Hurco Extended Shop Floor provides a web-based access point for a machine shop owner to communicate and view machine data using a web browser. Additionally, ESF is a distribution point for information about Hurco machine tools and it supports the remote monitoring and diagnosis of machine tools. By default, an ESF connection is made to Hurco when an internet connection is detected.

Here are the ESF connection states:

ESF State	Description
	The ESF Machine Service is not running. Clicking on this icon will turn the ESF Machine Service on.
	Please wait while the ESF Machine Service is starting and attempting to establish a connection with the ESF Server.
	The ESF Machine Service is running and communicating with the ESF Server Clicking on this icon will turn the ESF Machine Service off.
	The ESF Machine Service is running but is unable to communicate with the ESF Server. There may be a problem with the network connection. Clicking on this icon will turn the ESF Machine Service off.

For more information refer to the *ESF User's Guide* found at the ESF website, https://esf.hurco.com; a user account with password is required to access ESF.

ULTIPOCKET OPTION

The UltiPocket™ programming option adds special milling routines for machining pocket boundaries. This option provides complete clean out of pockets. The Pocket Boundary is the outside frame of the part. The basic philosophy of the UltiPocket option is to program the boundary and then tell the system to remove stock within that boundary. This approach eliminates complex calculations and shortens the part programming process.

The pocket feature is available for Mill Contour, Mill Frame, or Mill Circle live-tooling axial or radial program data blocks. For machines configured with the UltiPocket option, the Process tab Milling Type field contains the Pock Boundary (Pocket Boundary) selection. In addition, the Pocket Type field appears with Inward and Outward selections. Select either Arc or Line for the Cutter Comp field in the Axial or Radial Mill Contour or Mill Frame data block.

There are two types of Pocket Boundaries:

- **Outward**—When this routine is selected, the tool begins cutting from the center region of the part outward to pocket the entire programmed boundary. This operation is, therefore, the same as the standard WinMax Pocket selection. With this selection the cutter overlap is controlled by the Pocket Overlap value.
- **Inward**—This selection cuts in from the outside of the defined boundary. When this routine is selected, the tool enters the part and begins following a path formed by offsetting the boundary one-half the tool radius, plus the Pocket Overlap.

To control the percentage of overlap during cutting, enter a value in the Pocket Overlap field. After the first pass, the tool follows a path produced by offsetting the boundary by the tool radius, plus the pocket overlap for each pass.

After pocketing the boundary, the tool then cuts around the inside of the boundary using the selected blend offset and the programmed tool radius.

For TMM, TMX MY, and TMX MYS series machines, the turret moves the tool into position and the spindle turns, meaning the motion is in the X and C axes. The tool cuts the stock moving in the programmed direction.

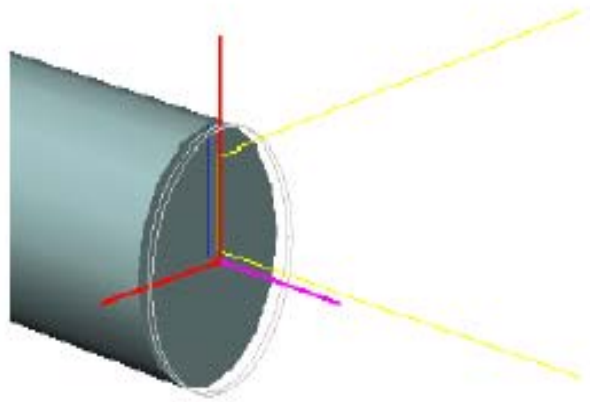


Figure 17–1. Axial Circle Pocket with X- and C-Axis movement

For TMX MY and TMX MYS series machines, the Geometry tab has a Linear Y Motion selection. When this feature is selected, the C Position field appears for entering the angle of the C-Axis position. The turret moves the tool into position, the live-tooling spindle turns, and there is Y-Axis motion using the X and X' axes. The tool cuts the stock moving in the programmed direction.

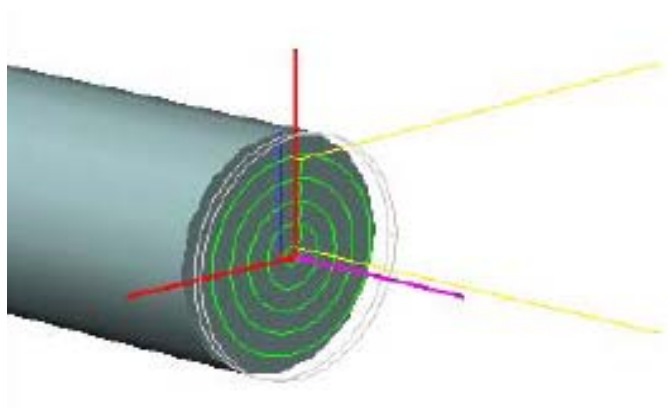


Figure 17–2. Axial Circle Outward Pocket with Linear Y Motion selection

Mill Contours

To create a Mill Contour data block using the UltiPocket option, set up the operation in the start segment (segment zero). As in standard WinMax milling, an UltiPocket Mill Contour block consists of segments beginning with segment zero.

With the cursor in the MILLING TYPE field, select Pock Boundary (Pocket Boundary) in the Start segment to indicate this block is the boundary of the part. The boundary must occur first in the program.

The Pocket Type field appears with Inward and Outward selections. This block is programmed in the same manner as the standard WinMax Mill Contour, with the addition of the Pocket Overlap percentage. Select either Arc or Line for the Cutter Comp field in the Axial or Radial Mill Contour data block.

The segments after the Start segment are programmed in the same manner as standard milling lines and arcs. Automatic calculation of unknown points is available for these data blocks.

Mill Frame

The Mill Frame data block is often used to create the part boundary. With the cursor in the MILLING TYPE field, select Pock Boundary (Pocket Boundary).

The Pocket Type field appears with Inward and Outward selections. This block is programmed in the same manner as the standard WinMax Mill Frame, with the addition of the Pocket Overlap percentage. Select either Arc or Line for the Cutter Comp field in the Axial or Radial Mill Frame data block.

Mill Circle

The Mill Circle data block is used for boundaries. With the cursor in the MILLING TYPE field, select Pock Boundary (Pocket Boundary).

The Pocket Type field appears with Inward and Outward selections. This block is programmed in the same manner as the standard WinMax Mill Circle, with the addition of the Pocket Overlap percentage.

WASHDOWN GUN

An optional washdown gun is available for spraying coolant inside the enclosure to clean the machine. The sprayer is located on the front right side of the machine.

To use the washdown gun,

1. Select the **Manual** console key to access the Manual screen.
2. Select the TOGGLE WASHDOWN GUN *F7* softkey to power the washdown gun located on the front right side of the machine on or off.
   - When on, coolant will be on for the Washdown Gun when the door is open.
   - The Washdown Gun is active when the Primary coolant is on.
   - The selection is modal and remains in its current state until another selection is made.

RECORD OF CHANGES

11.01 v546OP January 2017

Revised by: H.Arle

Approved by: D.Skrzypczak

Changes
Updates to reflect software changes.

704-0115-409, May 2015

Revised by: K.Gross

Approved by: D.Skrzypczak

Changes
704-0115-409 Updates through software v09.02.130.15. Updated Bar Feed Block with Dwell (Sec) field for Tool Not Used and Tool as Bump Stop strategies. Added Max5 UI Graphics Screen option for Max 4 consoles. Added and updated Tool Repair option.

704-0115-408, May 2013

Revised by: K.Gross

Approved by: D.Skrzypczak, May 2013

Changes
704-0115-408 Updates through software v09.02.34. Added Part Ejector information for TMX MYS machines. Added information for Automatic Tailstock Interrupt and Stop Cycle operation.

704-0115-407, October 2012

Revised by: K.Gross

Approved by: D.Skrzypczak, K.Van Blaircum October 2012

Changes
704-0115-407 Updates through software v09.00.43. Added “Select Holes by Diameter” feature to DXF. Added Sub-spindle Tool Setter information to Tool Setter section. Revised with updated Logo and Hurco Brand Standards.

704-0115-406, March 2012

Revised by: K.Gross

Approved by: D.Skrzypczak March 2012

Changes
704-0115-406 Updates through software v8.1.3.48.

704-0115-405, December 2011

Revised by: K.Gross

Approved by: D.Skrzypczak, J.Mulkey, G.Traicoff, K.Van Blaircum

Changes
704-0115-405 Updates through software v8.1.2.30. Updates to Tool Setter section, including Probing Tools. Added Steady Rest section. Added Preliminary information for DXF live-tooling. Added UltiMonitor chapter and removed UltiNet section. Set cross-references between books. Added ScreenPath condition and paragraph tag.

704-0115-404, April 2011

Revised by: K.Gross

Approved by: D.Skrzypczak, J.Mulkey, G.Traicoff, K.Van Blaircum

Changes
• 704-0115-404 Updates through software v8.1.2.

704-0115-403, June 2010, ECN 16538

Revised by: K.Gross

Approved by: D.Skrzypczak, J.Mulkey, G.Traicoff, K.Van Blaircum

Changes
• 704-0115-403 Updates through software v8.1.1. • Added Automatic Door, Oil Skimmer, Parts Conveyor, Ultipockets sections.

704-0115-402, April 2009, ECN 16508

Revised by: K.Gross

Approved by: D.Skrzypczak, J.Mulkey, C.Thale, G.Traicoff

Changes
• 704-0115-402 rB Updates through software v2.02.05. • 704-0115-402 rA Updates through software v2.02.03.

704-0115-401, September 2008, ECN 16508

Revised by: K.Gross

Approved by: D.Skrzypczak, J.Mulkey, C.Thale, G.Traicoff

Changes
• 704-0115-401 rB Added information to front matter about On-screen Help and accessing the On-screen Help in PDF format. • 704-0115-401 rA New manual release, July 2008,

INDEX

A

B

C

D

E

F

G

H

I

J

K

L

M

N

O

P

Q

R

S

T

U

V

W

X

Z