# HURCO®

# GETTING STARTED WITH WINMAX MILL

## Dual Screen and Max Consoles
### for Hurco Machining Centers

# HURCO

The information in this document is subject to change without notice and does not represent a commitment on the part of Hurco Companies, Inc. (Hurco). The software described in this document is furnished under the License Agreement to customers. It is against the law to copy the software on any medium except as specifically allowed in the license agreement. The purchaser may make copies of the software for backup purposes. No part of this document may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopying, for any purpose without the express written permission of the Hurco machine tool owner.

Hurco Manufacturing Company reserves the right to incorporate any modification or improvements in machines and machine specifications which it considers necessary, and does not assume any obligation to make any said changes in machines or equipment previously sold.

Hurco products and services are subject to Hurco's then current prices, terms, and conditions, which are subject to change without notice.

**Patents:**  U.S. Patents B14,477,754; 5,453,933; Canadian Patent 1,102,434; Japanese Patents 1,649,006 and 1,375,124; other Patents pending.

Hurco, Max, Ultimax, and WinMax are Registered Trademarks of Hurco Companies, Inc.

AutoCAD, Autodesk, and DXF are registered trademarks of Autodesk, Inc.

MS-DOS, Microsoft, and Windows are registered trademarks of Microsoft Corporation.

Many of the designations used by manufacturers and sellers to distinguish their products are claimed as trademarks. Hurco has listed here all trademarks of which it is aware. For more information about Hurco products and services, contact:

**Hurco Companies, Inc.**
One Technology Way
P.O. Box 68180
Indianapolis, IN 46268-0180
Tel    (317) 293-5309 (products)
         (317) 298-2635 (service)
Fax    (317) 328-2812 (service)

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

# TABLE OF CONTENTS

# LIST OF FIGURES

# USING THIS MANUAL

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

Additional information is available on the machine's Documentation CD.

## Sample Screens

Sample screens in this documentation were taken from a WinMax Mill single-screen control. All screens are subject to change. The screens on your system may vary slightly. The sample screen here illustrates softkeys and includes a software version.
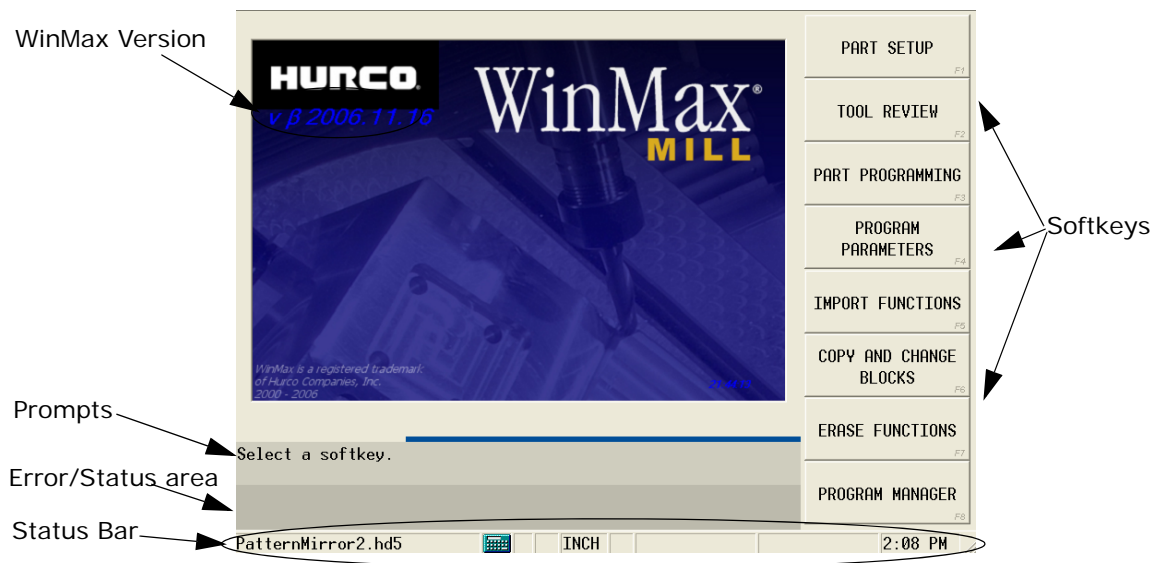


## Softkeys

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 Mill* 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 above 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,* on page xv 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 Your WinMax Mill* 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.

# Printing

To print part or all of this manual from the CD, select **File/Print**. Be sure to review the **Print Range** selections and make the appropriate choice for pages. Select **Properties/ Paper/Quality** and adjust the **Tray Selection/Paper Source** if necessary.

Printing to a Post Script printer provides the best results.

# Icons

This manual may contain the following icons:

### Caution/Warning

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

### Hints and Tricks

Useful suggestions that show creative uses of the WinMax features.

### Important

Ensures proper operation of the machine and control.

### Troubleshooting

**?** Steps that can be taken to solve potential problems.

### Where can we go from here?

Lists several possible options the operator can take.

### Table of Contents

To assist with onscreen viewing, this icon is located on the cover page. Click the icon to access the Table of Contents (TOC).

You can also access many of the same TOC entries from the Adobe Reader bookmarks located on the left side of the PDF page.

# USING AND PRINTING THE HELP

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 either the Help console button or the F console key followed by the 1 key (F1).

- To view the on-screen Help on the desktop software, select either the Help icon in the menu bar or the F1 key on your keyboard.

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 dispalyed Help topic, if a printer is attached and configured. See *Printing the Help* 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 Help

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 PROGRAM MANAGER *F8* softkey.

2. Select the DISK OPERATIONS *F7* softkey.

3. In the left-hand pane, navigate through the folders:

    - For WinMax Mill on a machine, the path is D:\Hurco\Winmax Mill\hlp.

    - For WinMax Desktop on a PC, the path is C:\Program Files\Winmax Mill\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.

# MACHINE AND CONSOLE BASICS

These topics are discussed in this section:

# Machine Components

Before using the machine, you should become familiar with its components. Because of European Committee (CE) requirements, Hurco machines sold in Europe differ somewhat from those sold elsewhere. The figure below identifies some of the easily recognized components of a machine. The console is in front of the machine, facing the operator's area.
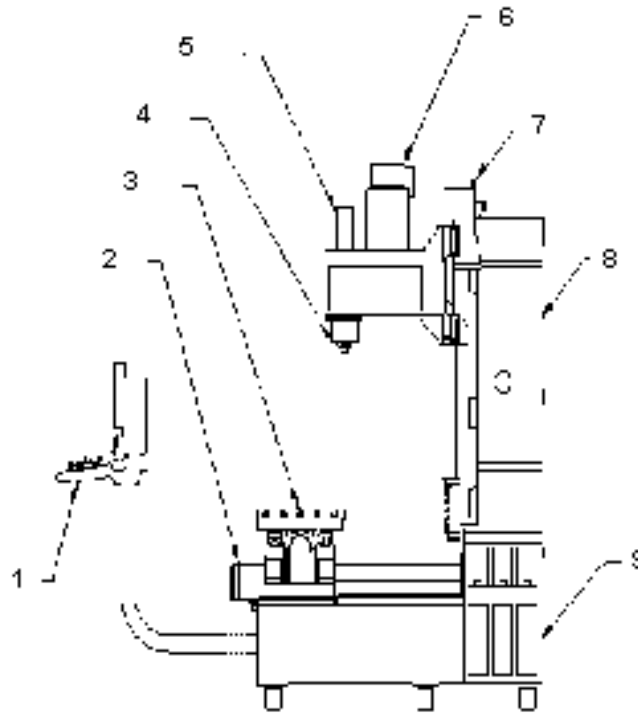
| | |
|---|---|
| **1** | Console |
| **2** | Way Cover |
| **3** | Table |
| **4** | Spindle |
| **5** | Power Drawbar |
| **6** | Spindle Motor |
| **7** | Z-Axis Servo |
| **8** | Column |
| **9** | Base |

*Figure 1–1. Hurco Machine with the Dual-Screen Console*

Hurco machines are available with several hardware and software options.  Information about these options is available from Hurco or your Hurco Distributor.

# Consoles

The dual-screen and single-screen consoles use WinMax software:
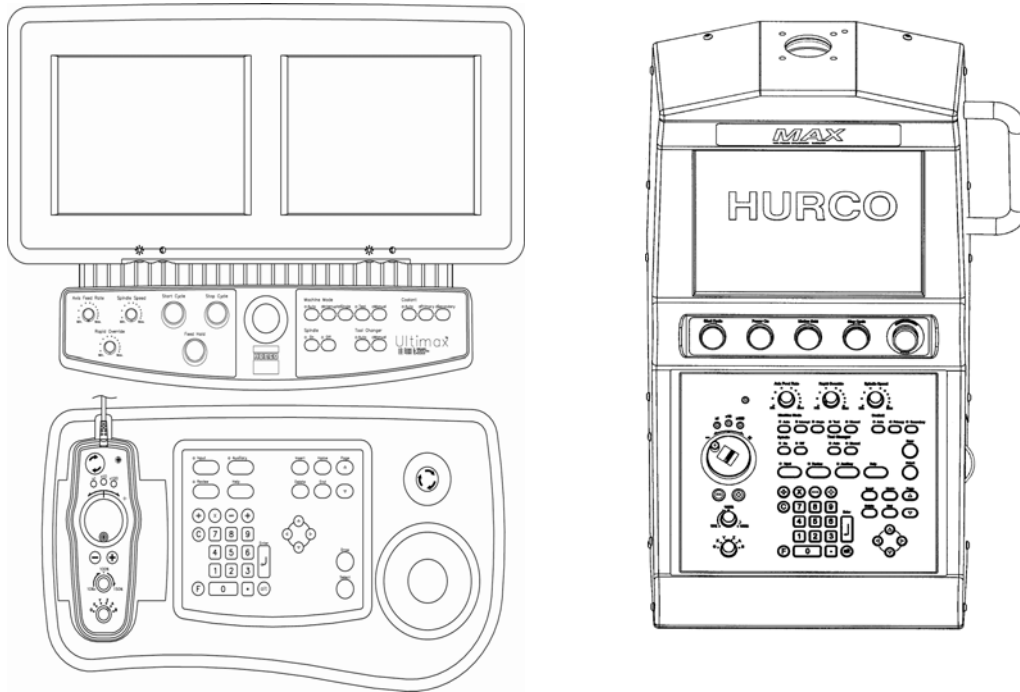


**Figure 1–2. Dual-Screen and Single-Screen Console**

The console, and the electrical components required to operate it, are called the "control" or the "*CNC*" (Computer Numeric Control). Some of the electrical components are built into a separate enclosure kept in the machine's electrical cabinet.

Some of the control's internal components, such as disk drives and memory, are like those in a PC. Disk operations, such as copying, deleting and storing files, are also similar.

The floppy drive is located on the console's right side panel. To protect the drive from debris, the protective floppy drive cover should be closed, except when inserting or removing a diskette.

Information about options is available from Hurco or your Hurco distributor.

# WinMax Interface Environment

There are three ways to navigate and enter data for programming:

- **Touchscreen**—use the stylus or other pointing device to select softkeys and drop-down lists for data entry and programming.

- **Keyboard**—use the function (F1-F12) keys and other keyboard shortcuts for navigation and to call up screens.

- **Ultimax classic edit mode**—use the console keys; for example, use the arrow keys for navigation and the enter key to accept data typed into a field. See *Edit Mode* below for more information.

## Edit Mode

Edit Mode is set in Utilities. Refer to *User Interface Settings, on page 3 - 4*. WinMax has two edit modes:

- Ultimax Classic (default)—in this mode the console arrow keys are used to move between blocks and segments, and the enter key is used to accept data after it is typed into a field.

- Windows Dialog—in this mode the console arrow keys do not navigate through fields and the enter key is not required to accept data typed into a field.

| If you want to... | Ultimax Classic | Windows Dialog |
|---|---|---|
| Enter data into a field | Enter key to accept data | Enter key to accept data<br>OR<br>Down (or Up) arrow to accept data |
| Advance to next block | Right/Left arrows<br>OR<br>Next Block/Previous Block softkeys<br>OR<br>Page Up or Page Down key | Next Block/Previous Block softkeys<br>OR<br>Page Up or Page Down key |
| Advance to next segment or operation | Right/Left arrows<br>OR<br>Next segment/hole softkeys<br>OR<br>Page Up or Page Down key | Next segment/hole softkeys<br>OR<br>Page Up or Page Down key |
| Navigate to next or previous tool in Tool Setup | Left or right arrow keys | Page Up or Page Down keys |

⇨ The right and left arrows will change the selection in fields with drop-down lists (such as Tool Type in Tool Setup) in either Ultimax Classic or Windows Dialog mode.

## Full Precision Editing

Numbers with decimals are rounded to the decimal display limit (three digits for metric and four digits for imperial) when displayed in a field. Full Precision Editing allows you to view the full number (up to 12 digits after the decimal) and/or to edit it to a more precise point. With the cursor in a numeric decimal field, press **F + decimal (.)** on a machine, or **Ctrl + decimal (.)** on the desktop to open a pop-up box containing the number.
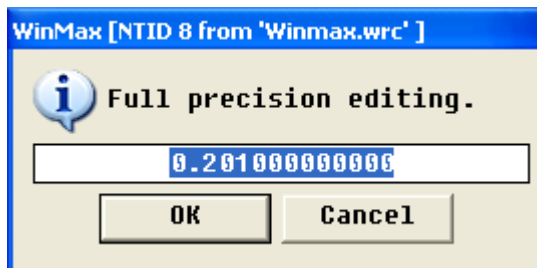


*Figure 1–3. Full Precision Editing Pop-up Box*

Edit the number and select OK to save the changes. To close the pop-up box without saving changes, select Cancel.

⇨ Do not use Full Precision Editing to change numbers that are automatically calculated by WinMax. These numbers are calculated to their 12-digit maximum accuracy and changing them could result in changes to other automatic calculations in the program.

## Softkeys

Softkeys appear as buttons on the screen; their default location is the right side of the screen, but they can also be positioned on the left by changing the setting in **User Preferences** (see *Utilities, on page 3 - 1* for more information). Select a softkey using one of these methods:

- On the screen, select the softkey.

- On the console, simultaneously press the **F** key and the number that corresponds to the softkey (for example, **F** + **1** will select the **F1** softkey). For dual-console machines, **ALT** + the softkey number will select softkeys on the graphics screen.

- On the keyboard (if using), press the corresponding function key (F1, F2, F3, etc).

## Drop-down Lists

Many fields contain a choice of items that are viewed by pulling down a list:



Touch the small arrow to view a drop-down list; use arrow keys to highlight choice and **Enter** console button to select
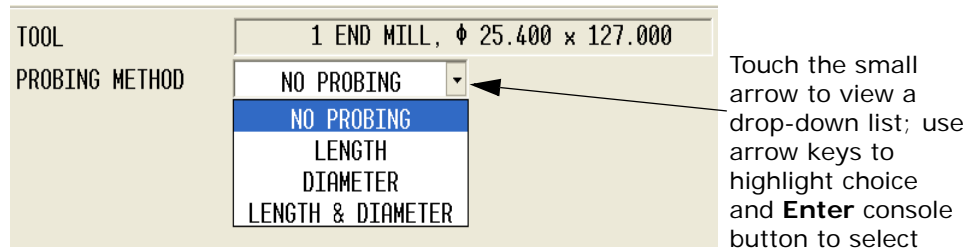
*Figure 1–4. Example of a drop-down list*

## Expand and Collapse Files

In the Program Manager, files can be expanded or collapsed as follows:

Touch the "+" to expand and view the files and folders in a folder

Touch the "-" to collapse (close) and hide the files and folders in a folder
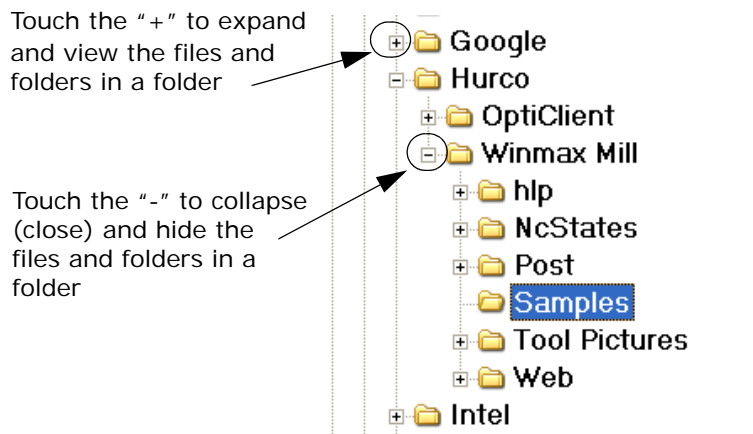


*Figure 1–5. Example of Expand and Collapse Files*

## Sorting and Resizing Columns

On screens that contain columns, data in a column can be sorted ascending to descending or descending to ascending by selecting the column heading. Columns can be resized by dragging the column divider in the header.

Select to sort column

Select and drag to resize column
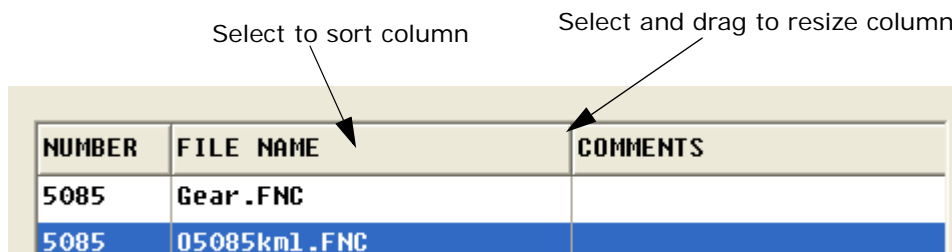


*Figure 1–6. Example of Sorting and Resizing Columns*

## Pop-ups

During machine operation and programming, pop-up boxes may be displayed to convey prompts or status messages. These pop-up boxes can be closed by selecting the appropriate button (i.e., YES or NO or OK). Some pop-up boxes may only provide informational messages and will be displayed for a few seconds before they automatically close.
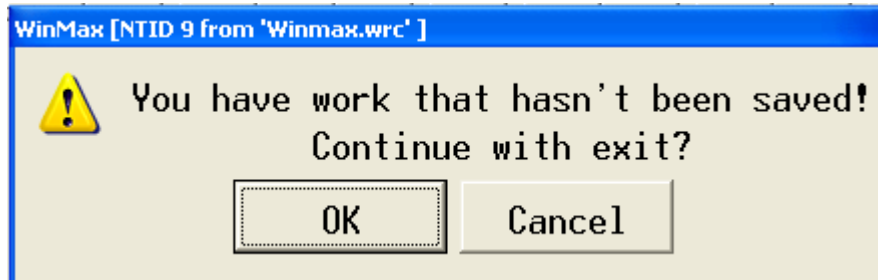


*Figure 1–7. Pop-up message example*

Certain pop-ups, such as the calculator and virtual keyboard, can be minimized by touching the "—" in the upper right corner. These pop-ups will remain open but are "hidden" in the taskbar, and can be viewed again by touching the taskbar button. These pop-ups can also be closed by touching the "X" in the upper right corner:
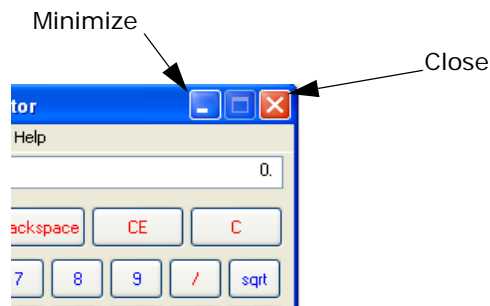


*Figure 1–8. Pop-up showing minimize and close icons*

## Status Bar

The Status Bar appears at the bottom of every WinMax screen. It displays the program name of the current active part program, a calculator icon, the current unit of measure (inch or mm), the keyboard icon, and 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.

- To use the calculator function, touch the calculator icon.

- To use the on-screen keyboard, touch the keyboard icon.

- To change the unit of measure, touch the unit of measure abbreviation.
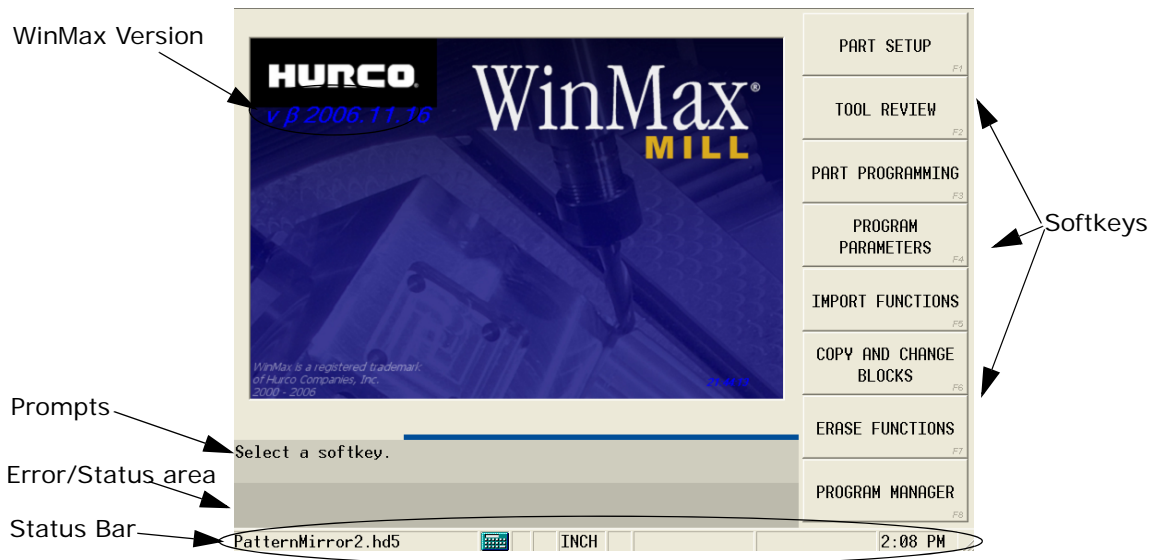


*Figure 1–9. Status Bar and other areas*

## *Calculator*

Select the calculator icon in the Status Bar to open the calculator. The calculator appears on screen and is operated using the stylus to select the calculator keys on screen.
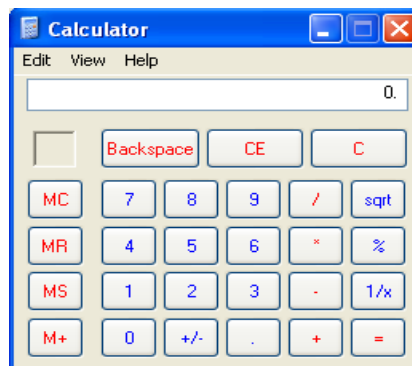


*Figure 1–10. Calculator pop-up*

When the calculator is minimized (with "—" in the upper right), the last calculation is

retained, but when it is closed (with "X" in the upper right), the last calculation is erased.

## 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 *Accessing the WinMax Help in PDF format, on page 1 - 10* 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.

    3. Select a topic from the Favorites list, and select the Remove button to remove it from the list.

## Accessing the WinMax Help in PDF format

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 viewing or printing. Here are the steps to access the PDF files:

1. From the Input screen, select the PROGRAM MANAGER *F8* softkey.

2. Select the DISK OPERATIONS *F7* softkey.

3. In the left-hand pane, navigate through the folders:

   - For WinMax Mill on a machine, the path is D:\Hurco\Winmax Mill\hlp

   - For WinMax Desktop on a PC, the path is C:\Program Files\Winmax Mill\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.

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

# Control Panel Function Groups

The buttons, keys, and knobs on the dual- and single-screen consoles are grouped by their functions. Here are the control panel groups on a dual-screen console:
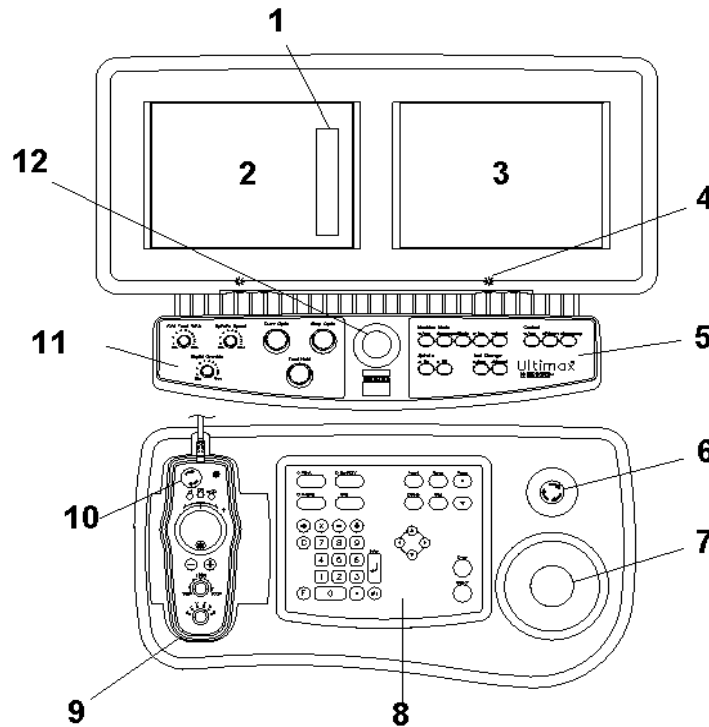


| 1 | Touchscreen softkeys |
|---|---|
| 2 | Text screen |
| 3 | Graphics screen |
| 4 | Brightness control |
| 5 | Machine operation keys |
| 6 | Emergency stop |
| 7 | Trackball |
| 8 | Programming keyboard |
| 9 | Remote jog |
| 10 | Emergency stop |
| 11 | Axis, spindle, and machine control |
| 12 | Power on button |

*Figure 1–11. Dual-Screen Console Panel Groups*

The dual-screen console also has a power on button, emergency stop button, and brightness control thumbwheels for the Text screen and Graphics screen. Some consoles are also equipped with contrast control thumbwheels.

Here are the control panel groups on a single-screen console:
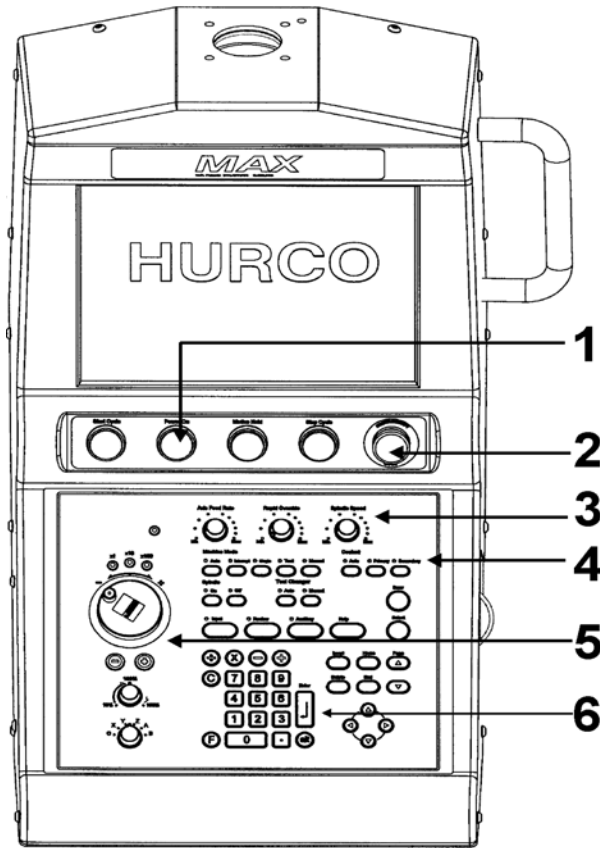


| 1 | Power On Button |
|---|---|
| 2 | Emergency Stop Button |
| 3 | Axis, Spindle, and Machine Control Dials |
| 4 | Machine Operation Keys |
| 5 | Console Jog Unit |
| 6 | Programming Keyboard |

*Figure 1–12. Single-Screen Console Panel Groups*

# Emergency Stop Buttons

There is an Emergency Stop button located on each console and one on the Remote Jog Unit. Press the Emergency Stop button to stop all spindle and table motion. This button locks down when pressed. To release it, twist the button in the direction indicated by the arrows.

⇨ Learn the location of all Emergency Stop buttons on the machining center *before* operating.

⚠ If the Emergency Stop button is pressed during execution of a part program, the tool must be jogged clear of the part before resuming operation.

When the Emergency Stop button is pressed, a special error file is created and saved to the machine hard drive in a folder called NavESTOP. These files record the machine conditions at the time the Emergency Stop button is pressed. These files can be retrieved when necessary for service purposes; refer to *Retrieve Log and Diagnostic Files, on page 3 - 37* for more information.

# Programming Keyboard

Program a job at the machine while reading from a blueprint or program worksheet. The prompts on the Text screen lead you through each element of a part program. Enter machine operations, part dimensions, and other parameters by selecting the appropriate screen softkeys and console buttons.

Set up and run part programs, and manage part program files using the following data entry keys:

- Text Screen Data Entry
- Softkeys
- Numeric Keypad
- Pop-up Text Entry Window
- Graphics Screen Data Entry

## Text Screen Data Entry

Text screen data entry keys are used for entering programming information into the Text screen's fields. These keys are located in the center of the console's Programming Keypad.

### *Programming Mode*

Programming Mode console keys are named for the screens they activate:

- **Input** – displays the main programming screen used to create and edit part programs. From this screen, access Part Setup, Tool Setup, Part Programming, Program Parameters, Copy and Change Blocks, and Restore and Erase menus.

- **Auxiliary** – accesses program storage management, system configuration settings, DXF files, reset master, and the upgrade system files menus.

- **Review** – for Conversational programs, provides an outline view of the blocks currently programmed, including type of block and tool used. Jump to a desired block by typing the block number and pressing Enter. For Numerical Control (NC) programs, the Review key displays or re-displays the NC part program.

- **Help** – displays help text. Place the cursor on a field in question and press the Help key. If help text is available, it will appear in a pop-up window. Help is not available for all screens.

These keys function as they would on a standard PC keyboard:

- **Insert** - type over and replace current text.

- **Delete**  - delete the character to the right of the cursor.

- **Home**  - position the cursor before a line of text.

- **End** - position the cursor at the end of a line of text.

- **Page Up** - position the cursor at the beginning of the previous page.

- **Page Down** - position the cursor at the beginning of the next page.

## *Text Screen Cursor Control Keys*

These keys control cursor movement and perform programming operations:

- **Arrow keys** - move the cursor from one field to the next, or advance a part program to the next data block.

- **Enter key** (↵) - accept the information typed in a text field, or move to the next field.

- Special Function keys

- *C* console key - Use the *C* console  and *Clear* keys to delete the value at the current cursor position.

- *Delete* console key - Use the *F* console and *Delete* keys together to delete the character to the left of the cursor.

- Arrow console key - Use the *F* console key in combination with the arrow keys to quickly move the graphics cursor on the graphics screen.

## Numeric Keypad

The numeric keypad allows you to enter numbers and calculate values in the Text screen. Perform the following operations with this keypad:

- Enter numeric data into fields on the screen.

- Perform calculations using the mathematical symbols ($\div$, $\times$, $-$, $+$) on the keypad.

## *Optional Computer Keyboard*

If the console is equipped with an optional computer keyboard, use it to enter data into a field. Press the Enter key to update a field and advance the cursor.

# Axis, Spindle, and Machine Control

The following keys and knobs are used to control machine movement and adjust the spindle and axes.

## Override Knobs

Three knobs on the upper console allow you to override the programmed axis feedrate, rapid, and spindle speed.

The Override knobs function as follows:

- **Axis Feed Rate** - controls the programmed axis feedrate during an auto run program. Turning the dial to counterclockwise slows the feedrate; turning the dial to clockwise speeds the feedrate. Selecting *Min* slows the spindle to 10% of the nominal value. Selecting *Max* increases the feedrate to 150% of the nominal value.

- **Spindle Speed** - controls the spindle speed. Turning the dial counterclockwise slows the spindle; turning the dial clockwise increases spindle speed. Selecting *Min* slows spindle speed to 640 RPM slower than the nominal value. Selecting *Max* increases spindle speed to 640 RPM faster than the nominal value.

- **Rapid Override** - overrides the programmed rapid traverse; the speed at which the table moves from one point to another. Selecting *Min* slows the table speed to 10% of the nominal value. Selecting *Max* increases the table speed to 150% of the nominal value.

## Jog Unit

Use the Jog Unit to manually jog the axes. The Console Jog Unit is on the Max console. The hand-held Remote Jog Unit can be removed from the console and carried closer to the work piece.
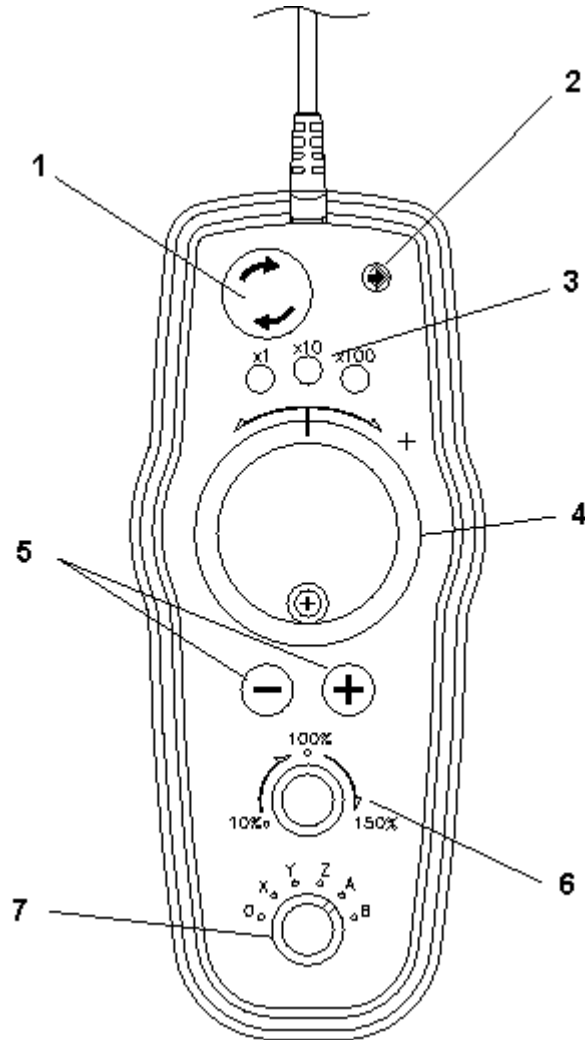
| 1 | Emergency Stop Button |
|---|---|
| 2 | Store Position Key |
| 3 | Hand Wheel Multiplier Keys |
| 4 | Jog Hand Wheel |
| 5 | Jog Feed Keys |
| 6 | Jog Feed Override |
| 7 | Axis Select Switch |

*Figure 1–13. Remote Jog Unit*

Both the Console Jog Unit and the Remote Jog Unit function the same:

1. Select an axis with the Axis Select Switch.

2. Use the Jog Feed Keys:

   a. Select either the + or – Jog Feed Keys.

   b. Adjust Jog Feed Override to override the programmed axis feedrate.

   OR

3. Use the Hand Wheel Multiplier:

   a. Select a hand wheel resolution with one of the Hand Wheel Multiplier Keys.

   b. Rotate the Jog Hand Wheel.

   ⇨  Other than the Emergency Stop button, the Jog Unit does not affect running programs.

The dials on a Jog Unit are defined as follows:

- **Axis Select Switch** – select the axis to jog (0, X, Y, Z, A, B).

- **Jog Feed Override** – control the jog speed (10% to 150%) of the nominal value. Use this dial to touch off the tool and move the X and Y axes to touch off the part for Tool and Part Setup.

- **Jog Feed Keys** - select minus (-) or plus (+) jog direction.

- **Jog Hand Wheel** - select minus  (-) or plus (+) jog direction.

- **Hand Wheel Multiplier Keys** - define the hand wheel resolution.

  - **x1** - defines a one-to-one ratio (each click equals .0001 inch, or .00254 mm).

  - **x10** - defines a 10-to-one ratio (each click equals .001 inch, or .0254 mm).

  - **x100** - defines a 100-to-one ratio (each click equals 0.01 inch, or .254 mm. One full turn equals 1 inch, or 25.4 mm).

- **Store Position Key** - record the axis' current position in the part program's setup screens.

  ⇨  See *Setting Jog Unit Parameters, on page 1 - 18* for instructions on setting the Jog Unit parameters.

Use a Jog Unit to manually jog the axes:

1. Select an axis with the Axis Select Switch.

2. Use the Jog Feed Keys:

   a. Jog axis direction by select either  + or − .
   b. Use the Jog Feed Override to override the programmed axis feedrate.

   OR

Use the Hand Wheel Multiplier:

   a. Select a hand wheel resolution with one of the Hand Wheel Multiplier Keys
   b. Rotate the Jog Hand Wheel to jog axis direction.

## *Setting Jog Unit Parameters*

To access the parameters :

1. Press the Manual Mode console key to display the Manual screen.

2. Press the Manual Function (F2) softkey. The Jog Unit parameters are displayed:

   • **Manual Jog Feed** - enter the desired manual jog axis feedrate. The range is from 0.0 to the machine's maximum feedrate.

     An axis may have a maximum jog feedrate slower than other axes. This slower axis can only move at its maximum jog feedrate (and not the higher feedrate of other axes).

     For example, the X and Y axes on a machine each have a maximum jog feedrate of  787 inches per minute (ipm). The Z axis has a maximum jog feedrate of 100 ipm. Without using the jog feedrate override, the X and Y axes can jog at 787 ipm , but the Z axis is limited at 100 ipm.

   • **Manual Spindle Speed** - enter the spindle speed when the Spindle On console key is pressed. This value can not be greater than the machine's maximum spindle speed. Entering a negative value (e.g., -500) causes the spindle to reverse (turn counterclockwise) at that speed.

## Machine Control

Machine Control buttons start and stop machine operation. The buttons function as follows:

- **Power On** - enables the relay control system. This button must be illuminated to operate the machine, but may be switched off while creating or editing a part program.

- **Start Cycle** - activates machine operation. When the machine is in an active mode, the Start Cycle flashes to indicate the machine is ready. When this button is pressed again, the light switches off.

To turn Control Power On:

1. Press the Power On console button.

2. Press the Manual Mode console key.

3. Press the Start Cycle console button.

   ⇨ Never press the Start Cycle button without knowing exactly what the machine will do.

- **Stop Cycle** - stops axes movement, then stops the spindle.

- **Feed Hold** (**Motion Hold** on Max console) - stops all axes movement, except a tap operation, when the tool is in the programmed feedrate region. Pressing the button a second time allows machine positioning to resume.

To Stop an Automatic Machine Operation:

Press the Stop Cycle button to stop the axis, then the spindle.

⇨ Or

1. Press the Feed Hold (or Motion Hold) console button to stop axis motion.

2. Press the Spindle Off console key to stop the spindle.

3. Press the Feed Hold (or Motion Hold) console button again.

# Machine Operations Keys

The Machine Operations console keys are needed to run part programs and control the machine during cutting. These keys are labeled under the Machine Mode, Spindle, Tool Changer, and Coolant groupings.

## Machine Mode

The Machine Mode console keys have these functions:

- **Auto** - allows you to run a part program automatically. See *Auto Mode, on page 4 - 55* for more information.

- **Interrupt** - halts machine operation during automatic execution of a program to allow manual functions, such as cleaning the work piece. See *Stop Motion, on page 1 - 20* for more information. Press Auto or Single to resume the part program and most cycles that were manually started by the operator (such as Chip Auger or Conveyor).

- **Single** - provides access to the Single Cycle screen. In this mode, the machine stops the axes after each automatic operation. Pressing the flashing Start Cycle button causes the automatic machining operation to resume. For information on the Single Cycle screen softkeys, see the *Auto Mode, on page 4 - 55* section. The softkeys are the same on both screens.

- **Test** - provides access to perform a program test run to identify potential problems before cutting the part. For more information, see *Auto Mode, on page 4 - 55*.

- **Manual -** provides access to manual machine operations that allow axis positioning with the jog unit. See *Manual Mode, on page 4 - 58* for more information.

## *Stop Motion*

If you observe a problem during the cutting process or simply want to stop the machine to make some adjustments, you may press one of three buttons to stop motion:

- **Feed Hold**—stops all axes movement (except a tapping operation in progress), when the tool is in the programmed feedrate region. Pressing the button a second time allows machine positioning to resume without loss of position, provided no other button or key has been pressed.

- **Interrupt Cycle**—halts machine operation during automatic execution of a program to allow manual functions such as cleaning the work piece. Pressing the Interrupt button performs the following actions:

  1. Stops all motion except the spindle.
  2. Pulls the tool straight up and out of the part.
  3. Stops the spindle.
  4. Displays the manual screen on the console.

To restart the part program, press the Auto button followed by the Start Cycle button.

⇨ If any key other than the Auto button is pressed to restart the program, the system cancels the execution of the part program.

- **Emergency Stop**—all motion stops and power is shut off to the spindle, relay control, and servo systems. An emergency stop message with recovery instructions appears at the bottom of the text screen. To release an Emergency Stop button, twist the button in the direction indicated by the arrows and pull the button up.

⇨ After the last choice is selected, the following sequence occurs:

1. A prompt is displayed to press the Start button while the console Start Cycle button flashes.

2. When the Start Cycle button is pressed, the program begins running at the specified point of the Start Block.

3. When the specified End Block is reached, the display indicates that Recovery Restart mode is complete and the Start Cycle button flashes again.

4. If the Start Cycle button is pressed again, then the program will rerun using the same Start and End Blocks.

## Spindle

- **On**—activates the spindle if the machine is in manual mode. The Start Cycle button must also be pushed to start spindle rotation.

- **Off**—stops spindle rotation during manual operation if the Spindle On button was previously pressed.

## Tool Changer

- **Auto**—activates the automatic tool changing function. When selected, all operator-initiated tool changes will be automatic.

- **Manual**—activates the manual tool changing function. When selected, all tools must be manually inserted into or removed from the spindle; bypassing the automatic tool changer.

## Coolant

- **Auto**—activates the selected coolant to spray whenever the tool is below the Retract Clearance plane.  The coolant turns off when the tool moves above the Retract Clearance plane, or during a tool change. This key cannot be activated in Manual mode. Pressing this key a second time turns off the coolant operation.

- **Primary**—functions only on machines equipped with a primary (i.e., flood) coolant system. Activates the primary coolant system when the machine is in Auto or Manual mode, and overrides an Coolant Auto operation. Pressing the Primary key a second time turns off this operation.

- **Secondary**—functions only on machines equipped with a secondary coolant system (e.g., Coolant Through Spindle, CTS). Activates the secondary coolant system when the machine is in Auto or Manual mode and overrides Coolant Auto operation. Pressing the Secondary key a second time turns off this operation.

# Communications Panel

All communication ports are located on the Comm Port panel assembly on machine control cabinet. The following connectors are available:

| Port | Connector Type | Use |
|------|----------------|-----|
| 10-base T | RJ45 | Network (Ethernet) |
| Indexer | 8-pin military | Indexer |
| PORT 1 and Port 2 | 9-pin | RS-232C Serial Communications |
| USB | USB jump drive | Not available for use. |

The communication ports are typically arranged as follows:
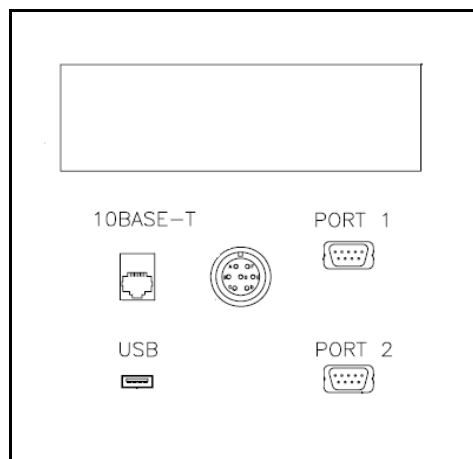


*Figure 1–14. Communications Panel*

## RS-232C Serial Port

The RS-232C serial ports can be used to connect peripherals to the machine. These ports may be addressed separately. The standard baud rates are software-selectable. The ports can be used as an output or input, depending upon the software.

The connector pin designated for the RS-232C signal is shown below:
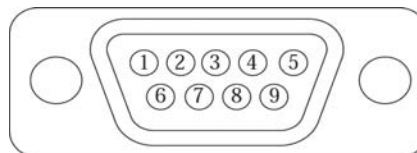


*Figure 1–15. Male 9-Pin D-Type Connector*

While the signals present at the serial port conforms to the RS-232C standard, not all standard RS-232C signals are available. Some peripheral devices may provide RS-232C control signals that are not available at the port described here. However, such devices

can usually be adapted to the port. In some cases, it may be necessary to add jumpers to the connector. Signals available at the serial port are:

| Pin | Signal Name | Signal on the Pin |
|---|---|---|
| 1 | Data Carrier Detect (DCD) | Not used by the control. |
| 2 | Receive Data (RXD) | Data received (by machine) in serial format from peripheral device. |
| 3 | Transmit Data (TXD) | Data transmitted (by machine) to peripheral device in serial format. |
| 4 | Data Terminal Ready (DTR) | Not used by the control. |
| 5 | Signal Ground (SG) | Line establishing the common ground reference potential for all interface lines. |
| 6 | Data Set Ready (DSR) | Signal to notify printer that transmitter is ready for transmission. |
| 7 | Request to Send (RTS) | Line used by control to instruct peripheral device to get ready to receive data. Data can be transmitted after the Clear-To-Send signal is received from connected peripheral device. |
| 8 | Clear to Send (CTS) | Control line used by peripheral device to indicate that it is ready to receive data from machine. |
| 9 | Ring Indicator (RI) | Signal indicates modem has received the ring of an incoming call. |

To connect a peripheral to the machine, fabricate an adapter cable. If a properly shielded low capacitance cable is used, cable lengths of up to 100 feet are permissible.

Be certain that you use the correct cabling before connecting the device to the machine. Consult the peripheral manual to determine whether the peripheral is a Data Terminal Equipment (DTE) or Data Communication Equipment (DCE) device. The Hurco machine is a DTE device, and in most cases, so is a personal computer. A printer may be either a DTE or DCE device.

## Indexer Port

Indexing signals are always present at the Indexer port, so there is no need to turn it on. It is the customer's responsibility to provide a harness from the Indexer to the Indexer port. Before making this harness, see the *Parts Listing and Diagrams Manual* for the correct pin-outs.

## Network Port

The 10baseT (RJ45) connector is used with the Ultinet option. This option requires an ethernet card, cabling from the ethernet card to the communications panel, and an optikey diskette to enable the option.

## USB Port

The USB Port (Universal Serial Bus) is a high-speed port that allows you to connect devices, such as printers and jump drives to the panel. You can use a jump drive to transfer files.

# HURCO

# Automatic Tool Changers

Hurco machining centers use either a Swing-Arm Random Pocket Automatic Tool Changer (ATC) or a Horizontal Chain Type ATC. Both types of ATCs function essentially the same.

Each tool is inserted into a tool holder before being loaded into the spindle. The *orientation hole* in the tool holder must always line up with the *orientation key* in the tool changer. Tool changer stations are numbered to identify and locate each tool.

⇨ Only use tools that are dimensioned for the maximum spindle speed.

## Loading a Tool into the Spindle

Use this procedure to manually load a tool into the spindle:

1. Press the Machine Mode Manual console key to prevent the ATC from moving while you insert a tool. The Manual screen appears.

2. Touch the Orient Spindle softkey to position the spindle for tool insertion. If there is a tool in the spindle, refer to *Removing Tools from the ATC Magazine, on page 1 - 26*.

   ⇨ On some machines the Y-axis moves to the front of the machine for manual tool changes.

3. Insert the tool holder into the spindle. Make sure the tool holder slots align with the spindle head guides.

4. Release the Spindle Unclamp button to secure the tool in the spindle. Be certain that the tool is firmly seated.

5. Touch the Tool In Spindle softkey. The Tool In Spindle field appears.

6. Make sure the tool number in the Tool In Spindle field matches the number of the tool in the spindle. If the numbers do not match, enter the correct tool number.

7. To load a tool into the ATC magazine, see the *Loading Tools into the ATC Magazine, on page 1 - 25* section.

## Unloading a Tool from the Spindle

To manually remove a tool from the spindle, follow these steps:

1. Press the Manual Mode console key. The Manual screen appears.

2. Hold the tool to prevent it from dropping.

3. Press the Spindle Unclamp button. The spindle unclamp button is either on the side of the spindle, or the front of the spindle. Refer to *Parts Listing and Wiring Diagrams Manual* for your machine for a drawing showing the Spindle Unclamp button location.

4. The tool disengages. Pull the tool out of the spindle.

5. Release the Spindle Unclamp button when the tool is free.

# Loading Tools into the ATC Magazine

The ATC takes a tool from the spindle and automatically loads it into the magazine, if space allows. The tool's location in the magazine is recorded in the ATC Map (the Horizontal Chain Type ATC does not use an ATC Map). Before loading a tool into the ATC magazine, the Servo power must be On, and the machine must be calibrated.

⚠ Do not manually load tools directly into the magazine.

To load the tool currently in the spindle into the ATC magazine:

1. Press the Manual Mode console key.

2. Touch the Tool In Spindle softkey. The Tool in Spindle, Next Tool, and ATC Map (Swing-Arm Random Pocket ATC only) fields appear. For more information about the ATC Map, see "The ATC Map" section.

3. Verify that the Tool In Spindle value matches the tool currently in the spindle. If the numbers do not match, enter the correct tool number.

4. Enter the same tool number into the Next Tool field.

5. Press the Tool Changer Auto console key.

6. Enter a new tool number into the Next Tool field. The ATC Map field must be Auto.

7. Press the Tool Changer Auto console key. The Start Cycle light begins flashing.

8. Clear the tool changer and shut the enclosure door. Press the Start Cycle button. The Tool In Spindle field will be updated to the next tool value.

   • If the Next Tool is an Auto tool, it was placed into the magazine when the previous tool was removed from the spindle.

   • If the Next Tool is a Manual tool, you will be prompted to insert it into the spindle.

# Removing Tools from the ATC Magazine

Remove tools from the ATC magazine by following these steps:

1. Press the Manual Mode console key.

2. Touch the Tool In Spindle softkey to access the Tool In Spindle screen. If there is no tool in the spindle, set the Tool In Spindle field to 0 (zero).

   ⇨ The ATC Map field will indicate if the tool selected is in the magazine, and its location.

3. Enter the tool number (of the tool you want to remove in the magazine) into the Next Tool field.

4. Press the Tool Changer Auto console key to move the Next Tool into the spindle.

5. Clear the tool changer area and shut the enclosure door. Press the Start Cycle button to initiate the tool change.

6. Press the Spindle Unclamp button and manually remove the tool from the spindle.

7. Repeat steps 2 through 6, as needed, to remove additional tools from the ATC magazine.

# Large Tools in the ATC Magazine

A part program may require tools with large diameters. These tools can be manually loaded by the operator, or automatically loaded.

⇨ The ATC magazine capacity is reduced by half for tools larger than 80 mm (125 mm for some machines).

Follow these steps to load large tools into the ATC magazine:

1. Touch the ATC Map softkey from the Tool In Spindle screen. The ATC Map appears. See "The ATC Map" section for more information.

2. Touch the Max. Tool Dia. More than XX mm softkey.

3. An "ATC Map will be cleared! Are you sure you want to change Max. Tool Diameter to more than XX mm?" message appears.

   ⇨ Each time you switch between large and small tools, the entire ATC Map will be cleared and the magazine must be reloaded.

4. Select the Yes softkey. The ATC Map will clear, then reappear. Only the odd numbered tool pockets will be available.

5. Reload tools into the magazine using the "Loading a Tool into the Spindle" section.

6. Return to the default setting of Maximum Tool Diameter XX mm or Less by using the previous procedure and touching the Max. Tool Dia. XX mm or Less softkey.

# Machine Start Up

## Calibrate Machine

Calibrating establishes the machine reference point (absolute zero) for each axis. Absolute zero is the location on the table, usually a corner or near a corner, where the X and Y axes intersect. This value does not change.

To calibrate the machine, press the Manual Mode console button. "Uncalibrated" will be in the Cal Status field.

### Establishing Servo Power

Perform the following steps to establish servo power:

1. If necessary, release all Emergency Stop buttons. Twist the button in the direction indicated by the arrows to release it.
2. Press the Manual Mode console button.
3. Press the console Power On button to turn on the machine. The Power On button lights up.
4. If there is a servo or spindle error, press the Reset Servos and Spindle softkey to clear it. The Start Cycle button begins flashing.
5. Press the Start Cycle button to turn on the machine servos. The Start Cycle button stops flashing.

### Axes Calibration

To calibrate all axes, follow these steps:

1. Press the Calibrate Machine softkey. The Start Cycle button begins to flash off and on.
2. Press the flashing Start Cycle button. The Axis Limit Switches field indicates the current status of the machining center's limit switches as each axis calibrates.
3. The machine position display (at the top of the text screen) shows zero (0) for all axes when the calibration process is complete.

## Warm Up Machine

If the machine has been idle for an hour or more, it is recommended that the warm up cycle be run. Warming up an idle machine before part cutting improves component reliability and machine performance. You must be in Manual mode to run a warm up cycle.

Follow these steps to warm up the machining center:

1. The control power must be on and the axes must be calibrated. There must not be any tool in the spindle, and the Tool in Spindle field on the Manual screen must be 0.

2. Select the Warm Up Machine softkey. The Start Cycle button begins flashing.

3. Press the Start Cycle button (to cancel the warm up cycle, press any Mode console button or softkey before pressing the Start Cycle button). The axes slowly move from one end of the machine to the other. The spindle moves at a low RPM for five minutes.

4. The warm up cycle completes in 15 minutes. The Manual screen reappears and axis movement stops.

Follow these steps to *cancel* the warm-up cycle:

1. Press the Feed Hold console button.

2. Press the Spindle Off button.

3. Press the Feed Hold button a second time, or press the Stop Cycle button.

## Reset Master

To restart the control (*reboot* the system) without switching the power to the machine tool off and back on again, follow these steps:

1. Press the Auxiliary console button. The Auxiliary screen appears.

2. Press the More softkey.

3. Press the Reset Master softkey. The Yes and No softkeys appear. When the Yes softkey is pressed, the system will reboot. Pressing the No softkey aborts the reset process.

⇨ Be sure to save any part program you're working on before resetting the master. In NC Programming, an unsaved part program will be permanently erased. In Conversational Programming, the part program will be saved, but any changes made after the last autosave will be lost.

# Recovery and Restart

Restart a part program at almost any point within the program - typically, the point at which the running program was interrupted.

If the Emergency Stop button was used to stop machining, machine power must be restored before restarting the program.

To restart the program, follow these steps:

1. Select the console Auto button. The Start Block default is 1, and the End Block default is the last block of the program.

2. Enter the proper *Start Block* and, optionally, an *End Block* if other than the end of the program.

3. Touch the Recovery Restart softkey.

4. If the Start Block contains multiple choices for restart, then prompts are displayed to select the proper point of restart.

# Stop Machine Operation

There are three methods to stop machine operation without turning off the power:

- Emergency Stop
- Park machine
- Control Power Off Timer parameter

⇨ The methods described here stop machine operation, but do not shutdown the control. See *Shutdown Control, on page 3 - 34* for more information about shutting down the control.

⇨ If the machine will not be used for several days, or the shop has power surges or blackouts, turn off machine power at the main power switch, after stopping machine motion and shutting down the control.

## Emergency Stop

To shut down the machining center quickly, press the Emergency Stop button. All motion stops and power is shut off to the spindle, relay control, way lubrication pumps, and servo systems.

⚠ Do not use Emergency Stop shut down if the machine has a long table with heavy equipment attached to one end. Instead, park the machine so that the weight of the table and any attached equipment will be evenly distributed.

⇨ Before hitting the Emergency Stop button, park the machine or center the table.

When the Emergency Stop button is pressed, a special error file is created and saved to the machine hard drive in a folder called NavESTOP. These files record the machine conditions at the time the Emergency Stop button is pressed. These files can be retrieved when necessary for service purposes; refer to *Retrieve Log and Diagnostic Files, on page 3 - 37* for more information.

## Park Machine

Parking the machine centers the table and places the spindle at the home position. Table and attached equipment weight is distributed evenly when the machine is parked. Before parking the machine, the servo power must be on, the machine must be calibrated and the Tool in Spindle must be 0.

Follow this procedure to park the machine:

1. Press the Manual Mode console button.
2. Select the PARK MACHINE softkey. The Start Cycle button flashes.
3. Press the Start Cycle button. The machine moves to its park position.
4. A message with instructions to return to the power-up condition is displayed.

## Control Power Off Timer

Set the Control Power Off Timer on the Machine Parameters screen. All motion stops and power is shut off to the spindle, relay control, way lubrication pumps, and servo systems.

Where can we go from here?

To restart the machine after the Emergency Stop was pressed, follow this sequence:

1. Twist the Emergency Stop button to release it.
2. Press the Manual Machine Mode button.
3. Press the Power On button.
4. Press the Start Cycle button.

**Or...**

To restart the machine after it was parked, follow this sequence:

1. Press the Manual Machine Mode button.
2. Press the Power On button.
3. Press the Start Cycle button.

**Or...**

If machine will not be used for several days or power surges and blackouts are common, switch off the machine's power. After restarting, calibrate and warm up the machine.

# PROGRAM MANAGER

These topics are discussed in this section:

## Managing Program Files

Program Manager shows all part programs that are in the control's memory to edit or run. Use the Program Manager menu to create, open, save, and close programs. Features of the Program Manager include:

- Ability to load more than one program at a time.

- Ability to load Conversational and NC programs at the same time.

- Ability to copy blocks from one program into another (blocks are copied in Program Review; Program Manager is used to switch between programs to facilitate copy and paste).
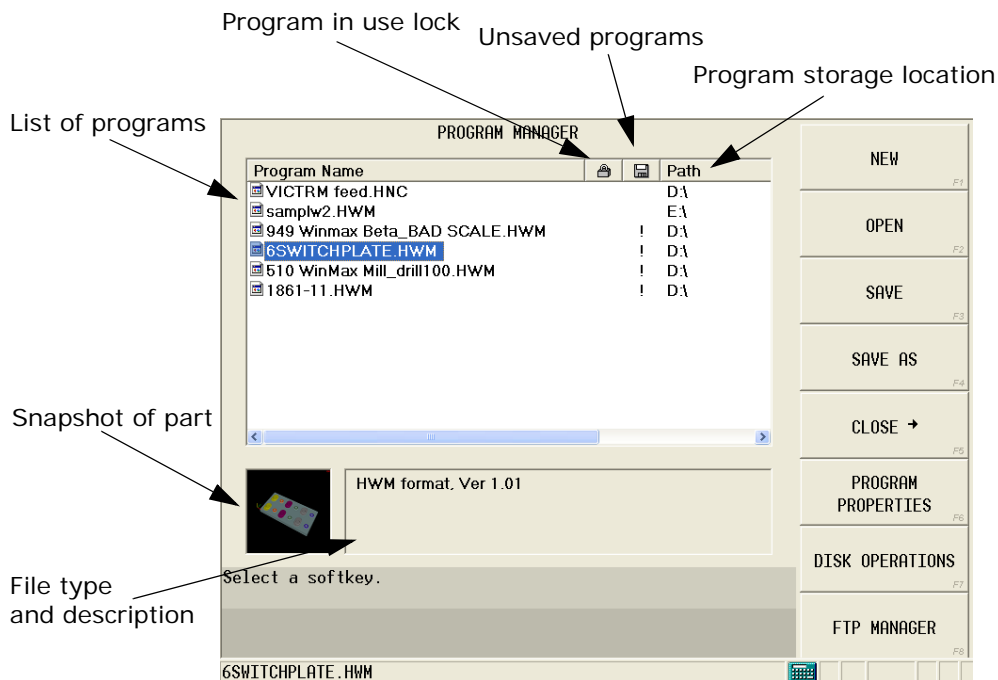


Figure 2–1. WinMax Program Manager

> Sort the program list by Program Name or Path by selecting the corresponding header. Select the Program Name header once to sort the list alphabetically. Select the Program Name header again and sort the list in reverse alphabetical order.

Program Manager softkeys are:

- **NEW** *F1*—creates a new part program. Choose the program type by selecting one of these softkeys:

    - **CONVERSATIONAL PROGRAM** *F1*—creates a new conversational part program.

    - **NC PROGRAM** *F2*—creates a new NC part program (extensions for NC part programs are determined by the NC dialect set in User Preferences, .FNC for Industry Standard and . HNC for Basic NC.).

        > WinMax gives new programs the default name NONAME (for example, NONAME1.HWM). When new programs are created, they should be renamed with a suitable name and be saved to the hard drive or other media.

- **OPEN** *F2*—opens a part program that is saved to the hard drive, network drive, floppy disk, or USB memory device. The Load Program screen opens, where you can locate the program from the list, as in the following example:
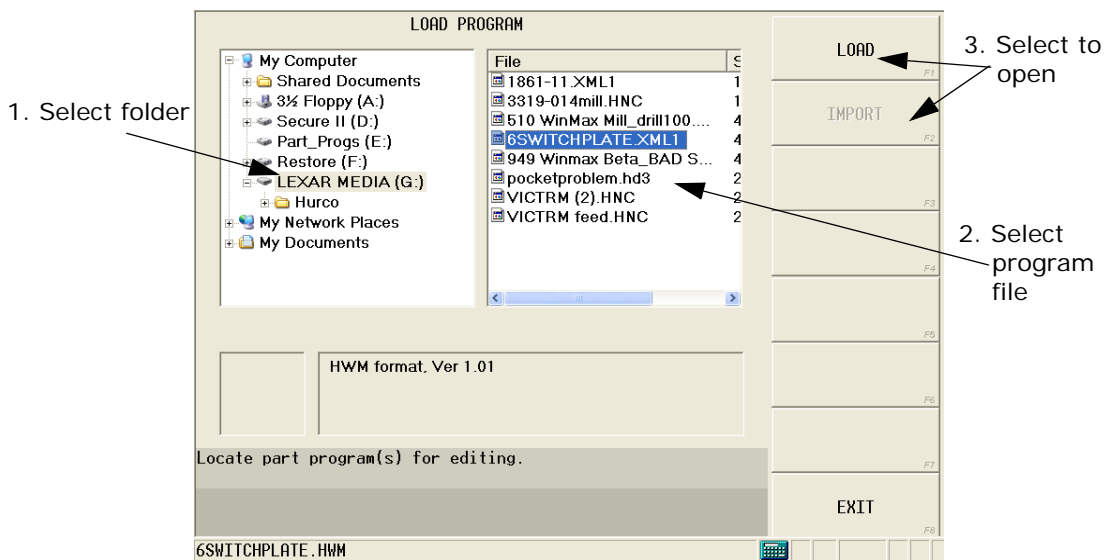


*Figure 2–2. Load Program screen*

To find a program, navigate through the folders in the left pane by selecting "+" or "-" to expand or collapse. Select a folder to view its contents in the right pane.

- **Load** *F1*—opens an HWM file or an XML1 file.

- **Import** *F2*—opens an HD3 file.

⇨ HD3 and XML1 files will be listed with .HWM extension in Program Manager. HWM is Hurco's conversational program file type. **SAVE AS** must be used to resave the program as HD3. Using **SAVE** will save the program with the HWM extension. Programs cannot be saved with the XML1 extension.

💡 Sort the program list by File, Size, or Date by selecting the corresponding header. Programs with the same date will be sorted alphabetically.

Select the File header once to sort the list alphabetically. Select the File header again to sort the list in reverse alphabetical order.

- **SAVE *F3***—saves the current part program to a hard drive, network drive, floppy disk, or USB memory device. If the program has a path indicated in Program Manager, it will be saved to the same location.

💡 Hurco recommends the following storage areas in My Computer:

  - 3½ Floppy (A:)
  - PART_PROGS (D:)
  - USB Device (E:)
  - My Network Places (if the UltiNet option is installed)

💡 Programs cannot be saved to the root level of the My Computer or My Documents directories. You must open these to see the directory listing before you can save to these areas.

- **SAVE AS *F4***—allows the current part program to be saved with a different name, as a different file type, or in a different location. Also used to save files with the default NONAME program name. Type the program name in the File Name box and select the file type, as in the following example:

| FILE NAME | 6SWITCHPLATE.HWM |
|---|---|
| SAVE AS TYPE | HWM Files (*.hwm) ▾ |

To rename a program, highlight the entire name in the FILE NAME field and type or use alpha/numeric console keys to replace the name. To modify character(s) in the name, highlight the name, select again in the field, and use the cursor keys, delete, and the alpha/numeric keys to change the name. When the filename is completely highlighted, the filename extension does not have to be re-entered; the extension selected in the SAVE AS TYPE field will be added to the name entered in the FILE NAME field.

For Conversational programs, the following file types are available from the Save As Type drop-down list:

- .HWM—WinMax Mill conversational format
- .HD5—WinMax Desktop conversational format (desktop only)
- .HD3—Ultimax conversational format

To save as an HWM file, select the **SAVE *F1*** softkey.

To save as an HD3 (or HD5) file, select the **EXPORT** *F2* softkey.

For NC programs, SAVE AS will default extensions to the selected NC dialect, either FNC or HNC. If another extension is desired, include the extension with the filename in the FILE NAME field. For example, SAMPLE. NC.

- **CLOSE** *F5*—opens a menu to close selected program or all programs.
- **Program Properties** *F6*—stores properties for the selected part program.
- **Disk Operations** *F7*—opens the Disk Operations screen to browse available folders and files, and cut, copy, paste, rename, and delete files.
- **FTP Manager** *F8*—displays external network connections (with the Ultinet option).

# Program Properties

Program properties stores and manages properties for the selected part program.

Fields are:

- **DISPLAY UNITS**—the unit of measurement (inches or millimeters) used throughout part programming (this field does not apply to NC programs). These units will be used when the part program is loaded into memory but the display value can be changed using the unit (IN or MM) icon in the status bar.
- **NAME**—the file name of the part program. This field appears as read-only.
- **PATH**—the media and directory path to the saved program. This field appears as read-only.
- **MATERIAL**—the material to be used, if specified (optional). This field appears as read-only; however, the material can be changed with the CHANGE MATERIAL *F3* softkey. See *Tool and Material Database, on page 4 - 37* for more information.
- **DESCRIPTION**—a short text description of the part program.
- **PROGRAM TYPE**—specifies the type of program.  Use the drop-down list to change the program type in this field. Program type may be changed for an exisiting program, and any incompatible data blocks or part setup data is removed from the program.

  The default Conversational Program Type can be set in Coversational Settings, see *Conversational Settings, on page 3 - 5* for more information.

- **WRITE PROTECTION**—prevents changes to the program from being saved when set to ON. Changes can be made if set to OFF.

  A write-protected program can be saved with a different program name. The user will be prompted to do this when attempting to save.
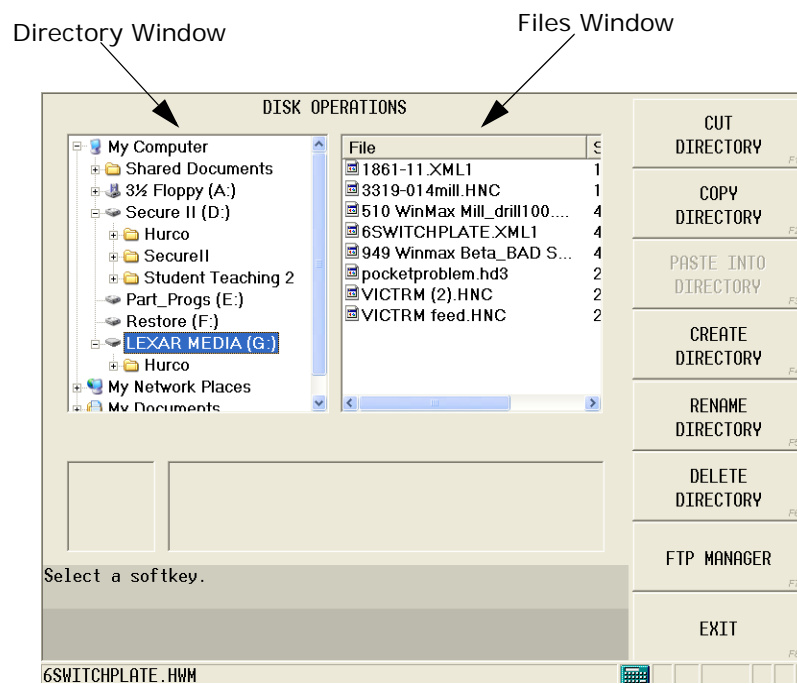
# Disk Operations

Disk Operations displays available directories and files. Cut, copy, paste, rename, and delete program files from Disk Operations.

See also:

*Import Functions, on page 4 - 48*

*NC States, on page 4 - 48*

*Retrieve Log and Diagnostic Files, on page 3 - 37*



**Figure 2–3. Disk Operations**

When a directory is highlighted in the Directory window, the softkeys are:

- **CUT DIRECTORY** *F1*—deletes the directory from one location to be pasted into another location
- **COPY DIRECTORY** *F2*—makes a copy of the directory (but does not delete) to be pasted into another location
- **PASTE INTO DIRECTORY** *F3*—pastes the directory or file that has been cut or copied. For example, to copy a directory and paste it into a new location:
  1. Highlight the directory you wish to copy.
  2. Select the COPY DIRECTORY *F2* softkey.
  3. Highlight the folder in which you wish to place the copied directory.
  4. Select the PASTE INTO DIRECTORY *F3* softkey.

- **CREATE DIRECTORY** *F4*—creates a new directory
- **RENAME DIRECTORY** *F5*—renames a directory
- **DELETE DIRECTORY** *F6*—removes the directory
- **FTP MANAGER** *F7*—displays the external network connections (with Ultinet option)

When a file is highlighted in the Files Window, the softkeys are:

- **CUT** *F1*—deletes the file from one location to be pasted into another location
- **COPY** *F2*—makes a copy of the file (but does not delete) to be pasted into another location
- **PASTE** *F3*—pastes the file that has been cut or copied. For example, to copy a file and paste it into a new location:
  1. Highlight the file you wish to copy.
  2. Select the COPY *F2* softkey.
  3. Highlight the folder in which you wish to place the copied file.
  4. Select the PASTE *F3* softkey.
- **RENAME** *F4*—renames the file
- **DELETE** *F5*—removes the file
- **LOAD** *F6*—loads the file into the Program Manager
- **FTP MANAGER** *F7*—displays the external network connections (with Ultinet option)

# FTP Manager

FTP Manager allows you to transfer programs from a remote location (host), such as a PC or machine, to a local PC or another machine. To use FTP Manager follow these steps:

1. Set up a connection between your machine and a host.
2. Connect to the remote location.
3. Manipulate files/folders between your machine and the host.
4. Disconnect from the host.
5. Use *Program Manager, on page 2 - 1* to access the copied file.

After you have connected to a host:

- Select the FTP Manager softkey to cut, copy, create and delete files/folders to which you have access on the host computer.
- Select the File Manager softkey to cut, copy, create and delete files/folders on the local PC or machine.

## FTP Host List

This screen displays all the remote locations (hosts) to which you can connect. The following softkeys are available:

- **CONNECT—c**onnect to the selected host. This softkey will be grayed out if you are already connected.

- **DISCONNECT—** disconnect from the selected host. This softkey will be grayed out if you are not connected.

- **Add Host—**selecting this softkey takes you to the FTP Host Properties screen and allows you to add a host.

- **Edit Host—**edit the properties of the selected host.

- **DELETE HOST—**remove the selected host from the FTP Host List.

## FTP Host Properties

This screen displays the properties of the remote location (host). The following fields are available:

- **ALIAS—**the name you want displayed in the host list. Choose a meaningful name to distinguish the host.

- **IP ADDRESS—**the IP address of the host.

- **USERNAME—**the name of the user. Access is determined by the FTP server's settings. Usually, access to the data drive is given to the "anonymous" login. Most servers assume a password is not needed for an anonymous login, but that may not be always be the case.

- **PASSWORD—**the password required when connecting to the host. The password must be a minimum of 0 characters and a maximum of 10 characters. If you do not want to require a password, the username should be anonymous.

- **DEFAULT REMOTE DIRECTORY—**the directory on the host you want to access. You can only enter the part of the path to which the server allows you access (e.g., you won't be able to access C:\Payroll if the server only allows you access to the D drive. Also, the default remote directory MUST exist.

> Leave this field empty. It will automatically populate with the root directory on the server to which you have access.

- **FILENAME FORMAT—**choose the format of file names sent from the host computer:

  **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

# UTILITIES

Utilities are accessed with the Auxiliary console key. Select the Auxiliary console key and this window with softkey selections appears on the screen:
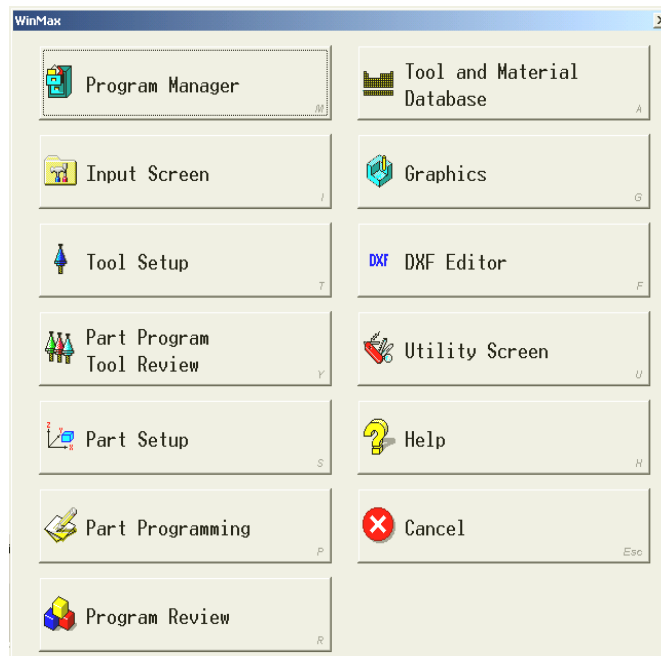


*Figure 1. Auxiliary Screen*

Select the **Utility Screen** softkey to access the Utility screen. These functions are available and are described in this section:

# System Configuration

The SYSTEM CONFIGURATION *F1* softkey on the Utilities screen displays machine, control, and software information. Softkeys on the System Configuration submenu are:

- **Display WinMax Configuration** *F3*—changes WinMax configuration settings.

- **Display Machine Specifications** *F4*—changes machine specifications.

- **Upgrade Motion Control Firmware** *F5*—upgrades firmware.

- **Backup Config & Machine Files** *F6*—copy Configuration and Machine files to a directory or to a USB memory device or floppy disk.

- **Restore Config & Machine Files** *F7*—use saved files to overwrite existing Configuration and Machine files.

## Display WinMax Configuration

The WinMax Configuration screen displays the current full version number of WinMax. This information is also displayed when you select the DISPLAY SOFTWARE VERSION softkey. The following softkeys are also available:

- **Display Software Options** *F2*—displays the current software options available for the machining center.

- **Display Motion Configuration** *F3*—displays information about the motion controller configuration.

- **Display Ladder Configuration** *F4*—Ladder files bridge communication between the machine and the software. The current version of the Ladder files on the machining center, the total number of Ladder files, and any mismatched Ladder files that may cause a software conflict are displayed.

- **Display WRC File Configuration** *F5*—WRC files contain the screen images. The total number of WRC files and any mismatched WRC files that may cause a software conflict are displayed.

- **Display WinMax IP Address** *F7*—Displays the current IP address of the CNC.

## Display Machine Specifications

Displays specific information about the machining center that was entered during software/machine installation.

- **Machine Class**—The physical orientation of the spindle, relative to the table surface. The possible machine classes are vertical, horizontal, and universal.

- **Vertical/Horizontal**—The current orientation of the machine. This field will be vertical for vertical machines and horizontal for horizontal machines. For universal machines, this field can be either vertical or horizontal.

- **Number of Axes Present**—The number of axes present on the machining center.

- **Machine Hour Meter**—The total number of machine hours expended to date.

- **Maximum Spindle Motor Speed**—The maximum acceptable speed for the spindle.

- **Maximum Spindle Tool Speed**—The maximum acceptable speed for a tool in the spindle.

- **Maximum Rapid Traverse Rate**—The maximum rapid traverse rate.

- **Maximum Contouring Rate**—The maximum feedrate for contouring.

- **Minus/Plus X Direction Travel Limit**—The maximum and minimum travel limits for the X axis.

- **Minus/Plus Y Direction Travel Limit**—The maximum and minimum travel limits for the Y axis.

- **Minus/Plus Z Direction Travel Limit**—The maximum and minimum travel limits for the Z axis.

## Backup Config & Machine Files

Backup files can be used to restore corrupted Configuration and Machine files on the hard drive. Select this softkey and choose the location to copy the files. Backup files can be copied to any directory on the hard drive or to a USB memory device or floppy diskette.

## Restore Config & Machine Files

Select this softkey to use saved Configuration and Machine files to overwrite existing files stored on the hard drive.

# User Preferences

The USER PREFERENCES *F2* softkey on the Utilities screen provides access to the following:

- *User Interface Settings, on page 3 - 4*

- *Conversational Settings, on page 3 - 5*

- *NC Settings, on page 3 - 5*

- *Autosave Settings, on page 3 - 6*

- *Tool Utilities and Settings, on page 3 - 6*

- *Machine Parameters, on page 3 - 11*

- *Serial Port Settings, on page 3 - 30*

- *FTP Server Settings, on page 3 - 31*

- *WinMax Uptime, on page 3 - 31*

- *Select Language, on page 3 - 31*

- *Data Logging Filters, on page 3 - 31*

# User Interface Settings

User Interface Settings change the screen display. Fields are:

- **APPLICATION FONT SIZE**—size of text displayed by the application in lists (for example, in Program Manager); default is Large. This field is inactive in WinMax Mill (active in WinMax desktop).

- **SOFTKEY MENU POSITION**—positions the softkey menu to the right or left of the screen; default is Right.

- **EDIT MODE**—select either Ultimax Classic (default) or Windows Dialog. For example, Ultimax Classic requires the Enter key to change the value of a data field, while Windows Dialog will accept a number by pressing the Down arrow.

- **ENABLE PROGRAM RESTORE**—resets previously loaded programs when the application is started after a shutdown; default is No.

- **SHOW ALL FILE TYPES**—view all file extensions when opening files; default is No (only displays HWM, HD3, HD5, FNC, HNC, and NC file types).

- **SCREENSAVER TIMEOUT**—set screensaver timeout in minutes (range is 1-30 minutes); default is 10.

- **LIST CONTROL ROWS**—select the appearance of list controls; default is Lines.

- **LIST ICON SIZE**—changes the size of icons in lists; default is Small.

- **WARN BEFORE SAVING IN OLD FORMAT**—specify if WinMax should should warn that information may be lost when saving a program as HD3.

- **MAX MEMORY LOAD**—controls the amount of system memory that graphics generation uses. The range is 50% to 100%; default is 85%. Graphics generation that uses more than the specified memory will be aborted by the system and an error message will be displayed. Raising the maximum memory load allows large programs to be drawn.

  ⇨ If Max Memory Load is set to 100%, graphics generation will not abort even if all system memory is being used to draw a graphic.

## Conversational Settings

Conversational Settings fields are:

- **MATH ASSIST STYLE**—select either Ultimax Classic or Standard Calculator interface to be used when editing data fields.

  **Ultimax Classic Math Assist**—select operation (=,-,*,/), then number, and then Enter. For example, if the number 5 is displayed as a value, pressing "+" and the number 3 will change the value to 8.

  **Standard Calculator**—select number, then operation (=,-,*,/), then Enter. For example, if the number 5 is displayed as a value, pressing 3 and "+" will change the value to 8.

- **DEFAULT CONVERSATIONAL PROGRAM TYPE**—select the default program type when creating new Conversational programs.

- **HD3 SAVE PROGRAM TYPE**—select the program type used when exporting conversational programs to the HD3 file format.

  ⇨ Choices for HWM and HD3 Program Types are:

  Standard = 3-axis

  Rotary A or Rotary B = 4-axis

  Rotary A, Tilt B, or Tilt A, Rotary C = 5-axis

## NC Settings

NC Settings fields are:

- **NC DIALECT**—select Basic or Industry Standard NC.

- **NC DISPLAY TYPE**—select axes data:

  Standard = 3-axis

  Rotary A or Rotary B = 4-axis

  Rotary A, Tilt B or Tilt A, Rotary C = 5-axis

- **EXPORTED NC DECIMAL PLACES**—indicate the number of decimal places expressed when exporting a conversational program as NC (range is 1-8); desktop version only

- **BPRNT/DPRNT OUTPUT DEVICE**—select where to output BPRINT and DPRINT formatted data; desktop version only.

- **BPRNT/DPRNT OUTPUT FILE**—specify the file path if BPRINT and DPRINT data will be output to a file; desktop version only.

- **CUSTOM NC FILE EXTENSION**—provide a custom filename extension to enable loading of NC files that use the extension.

## Autosave Settings

Autosave Settings fields are:

- **ENABLE AUTOMATIC SAVE**—turn Autosave on or off; if Yes, programs are saved to memory at the time interval specified—it will not save to the source location of the program.

- **SAVE FREQUENCY**—specify number of minutes between each autosave; range is 1-255.

- **SAVE ACTIVE PROGRAM ONLY**—specify Yes to save only the current program; specify No to save all loaded programs; default is No.

## Tool Utilities and Settings

See also *Import and Export, on page 3 - 10*.

The Tool Utilities and Settings screen is accessed from Utilities/User Preferences. Fields vary dependent on whether the Tool and Material Library option is loaded.

With the Tool & Material Library option, fields are:

- **Tool Calibration Mode**—sets the method used to calibrate tools.

  - Zero Calibration—stores the tip of tool dimension relative to home position (machine zero in z axis). See *Zero Calibration Mode, on page 4 - 15* for more information.

  - Absolute Tool Length—stores the actual length of the tool from spindle nose to the tip of tool. See *Absolute Tool Length mode, on page 4 - 15* for more information.

- **Automatically Load Unmatched Tools As Manual**—automatically loads unmatched tools from newly-loaded conversational part programs:

  - YES—new tools from the newly loaded program (that are not in the Tool & Material Library) are automatically added to Manual.

  - NO—requires new tools to be matched (see "Part Program Tool Review" section for more information about matching tools.)

- **Use Tool Type Checking**—only allow the selection of tool types that are valid for the operation in conversational mode:

  - YES—only the tools that are valid for the data block can be selected (i.e., drills and taps displayed for hole operations)

  - NO—allows any tool to be selected for any data block

- **Update Data Blocks With Tool Changes**—Update feeds and speeds in conversational data blocks when tool feeds and speeds are changed.

- **Overwrite Existing Zero Calibration**—when matching tools during a program load, the zero calibration from the program replaces the calibration for the tool on the machine. Default is YES.

⇨ The OVERWRITE EXISTING ZERO CALIBRATION setting applies both to tools that are automatically matched (during a program load), as well as tools that are manually matched.

- **Replace In Files**—specify if new tool feeds and speeds should be updated in current editing file only or in all open files

- **Tool Matching: Maximum Diameter Difference**—the maximum diameter difference for a tool to be considered a match

Softkeys are:

- **Tool Information Printing** *F6*—allows the Tool Library to be printed or saved to a file

- **Import and Export** *F7*—import and export tools to and from the Tool Library. To move the Tool Library from one machine to another, select EXPORT.

Without the Tool & Material Library option, fields are:

- **Tool Calibration Mode**—sets the method used to calibrate tools.

  - Zero Calibration—uses Offset Z and Z Table Offset fields to calibrate tool. See *Zero Calibration Mode, on page 4 - 15* for more information.

  - Absolute Tool Length—stores the actual length of the tool from spindle nose to the tip of tool. See *Absolute Tool Length mode, on page 4 - 15* for more information.

- **Use Tool Type Checking**—only allow the selection of tool types that are valid for the operation in conversational mode

  - YES—only the tools that are valid for the data block can be selected (i.e., drills and taps displayed for hole operations)

  - NO—allows any tool to be selected for any data block

- **Update Data Blocks With Tool Changes**—Update feeds and speeds in conversational data blocks when tool feeds and speeds are changed.

# Tool Information Printing

⚠️ Tool Information Printing is available only with the Tool and Material Library option.

The TOOL INFORMATION PRINTING *F6* softkey on the Tool Utilities and Settings screen allows you to view and print tool information for all tools saved in the Tool Library:
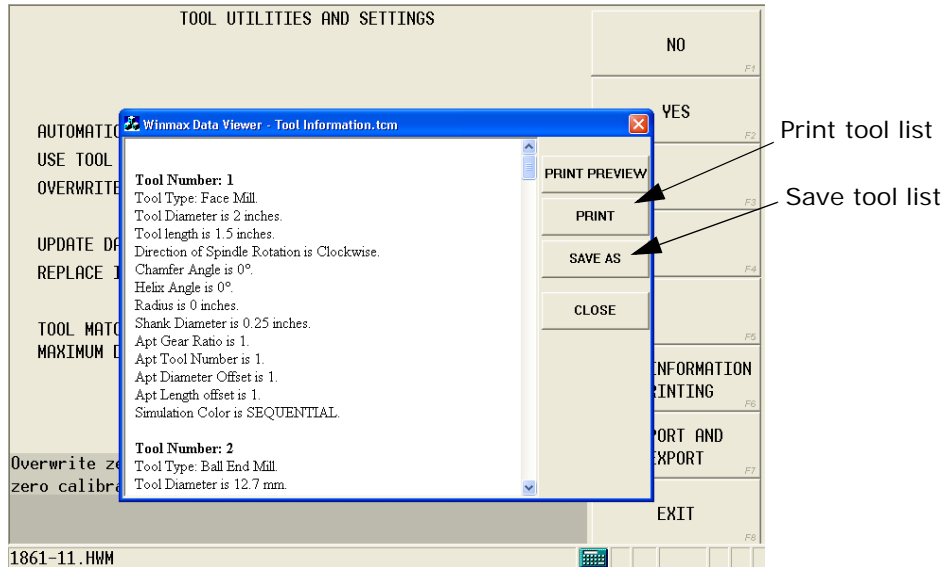


*Tool Information and Printing*

Select the PRINT button on the Tool Information pop-up to print a list of tools and settings, as in the following example:

```
ATC
Tool Number: 23
Tool Type: Ball End Mill
Tool Diameter is 0.0938 inches.
Tool length is 0.5625 inches.
Length of cut is 0.4688 inches.
Number of flutes is 0 inches.
Direction of Spindle Rotation is Clockwise.
Helix Angle is 0°.
Shank Diameter is 0.0938 inches.
Apt Gear Ratio is 1.
Apt Tool Number is 1.
Apt Diameter Offset is 1.
Apt Length offset is 1.
Simulation Color is SEQUENTIAL.
```

MTC
Tool Number: 1
Tool Type: Drill
Tool Diameter is 0.031 inches.
Tool length is 5 inches.
Length of cut is 5 inches.
Number of flutes is 0 inches.
Direction of Spindle Rotation is Clockwise.
Drill Angle is 118.000000°.
Helix Angle is 0°.
Shank Diameter is 0.031 inches.
Apt Gear Ratio is 1.
Apt Tool Number is 1.
Apt Diameter Offset is 1.
Apt Length offset is 1.
Simulation Color is SEQUENTIAL.

Tool Number: 20
Tool Type: Cutting Tap
Tool Diameter is 0.216 inches.
Tool length is 1.296 inches.
Length of cut is 1.08 inches.
Tap Direction: Right Hand.
Tap Chamfer: Bottoming.
Thread Pitch is 0.0417 inches.
Thread Diameter is 0.216 inches.
Helix Angle is 0°.
Flute Style: Spiral Flute.
Shank Diameter is 0.216 inches.
Apt Gear Ratio is 1.
Apt Tool Number is 1.
Apt Diameter Offset is 1.
Apt Length offset is 1.
Simulation Color is SEQUENTIAL.

## Import and Export

⚠️ Import and Export functions are available only with the Tool and Material Library option.

The IMPORT AND EXPORT *F7* softkey in Tool Utilities and Settings allows you to move tool information from the WinMax control to other locations and vice versa.

Import and Export submenu softkeys are:

- **Export Auto And Manual Tools** *F1*—save Auto and Manual tools to a different directory.

- **Export Manual Tool List** *F2*—save only the Manual tools to a different directory.

- **Import Into Manual Tool List** *F3*—load tools into the control from a directory. All tools are imported into the Manual list, even if they were exported as Auto tools (F1).

  ⇨ The ATC positions are not retained when tools are imported.

- **Export Tool and Material Database** *F5*—save tool templates and materials to a different directory.

- **Append Tool and Material Database** *F6*—load tool templates and materials into the control from a directory. These are added to the existing database on the control.

- **Replace Tool and Material Database** *F7*—load tool templates and materials into the control from a directory. The exisiting database is replaced.

# Machine Parameters

The Machine Parameters screens list various parameters, the range of values that can be set for each parameter, and the current set or default value. The values of the Machine Parameters may be changed by the operator.

⚠️ The machine, part, and/or tool may be damaged if parameters are changed without understanding the machine operation that may be affected by the change.

The fields that will be displayed on the Machine Parameters screens are dependent upon machine type. The values shown on the following screen figures are the default factory settings for HTX and VTX series machines.

Follow these steps to access the Machine Parameters screens:

1. Select the AUXILARY console key.
2. Select the Utilities icon.
3. Select the **USER PREFERENCES** *F2* softkey.
4. Select the **MACHINE PARAMETERS** *F6* softkey.

## Machine Parameters Page 1



*Figure 3–1. Machine Parameters, Page 1*

## Machine Parameters - Page 1

| Parameter | Description | Range | Default |
|---|---|---|---|
| Coolant Delay Time | Sets the time the program pauses when the coolant is enabled. | 0-60 seconds | 0 |
| Pulsating or Delay Washdown Enable | Sets the Washdown Coolant pump run cycle; used in conjunction with other washdown coolant parameters. | 0 (continuous) 1 (pulsating) 2 (delay) | 0 or 1 (depends on machine) |
| Alt Washdown Dwell | Controls washdown coolant flow on the right side of the machine enclosure on certain machines. Used in conjunction with other washdown coolant parameters. | 0-32767 (.01 sec) | 500 or 1000 (depends on machine) |
| Alt Dwell Lt Side | Controls washdown coolant flow on the left side of the machine enclosure on certain machines. Used in conjunction with other washdown coolant parameters. | 0-32767 (.01 sec) | 200 |
| Alt Washdown Off Time | Sets the time the washdown coolant flow cycle is paused, on certain machines. | 0-32767 (.01 sec) | 0 or 200 (depends on machine) |
| Washdown On Delay Timer | Defines the time the washdown coolant pump is on; used with Washdown Off Delay Timer. | 0-9999 seconds | 0 |
| Washdown Off Delay Timer | Defines the time the washdown coolant pump is off; used with Washdown On Delay Timer. | 0-9999 seconds | 0 |
| Control Power Off Time | Turns the control off after the specified period of inactivity. | 0-255 minutes | 120 |
| Disable Tool Picker Option | Turns off the Tool Picker option. | 0 (enable) 1 (disable) | 0 |
| Enable Dual Zones | Sets a dual-zone capable machine as either a single long bed machine, or as a dual zone machine. | 0 (1 zone - long bed) 1 (dual zones) | 1 |

# Coolant Parameters

The coolant parameters are used to control coolant flow to the spindle (known as flood coolant), coolant flow in the machine enclosure (known as washdown coolant), or to turn the washdown coolant pump on or off. The Washdown Coolant Pump must be On for Right and Left side operation. Some coolant parameters may not be functional on your machine. For examples of how to use washdown coolant parameters on your Hurco machine, see *Table 3–1. Washdown Coolant Parameter Examples, on page 3 - 19*.

Washdown coolant flow for VMX24, VMX30, VMX42, VMX50, and VMX60 machines is continuos to both sides of the enclosure. The amount of washdown coolant used during machining operations is controlled by cycling the washdown coolant pump.

VMX64 and VMX84 machines are equipped with a butterfly solenoid that alternates washdown coolant flow between the right and left sides of the machine enclosure.

This is the washdown coolant flow cycle for VMX64 and VMX84 machines:

1. Coolant flows on right side of machine enclosure (defined in Alt Washdown Dwell).

2. Pause time of no coolant flow (defined in Alt Washdown Off Time).

3. Coolant flows on left side of machine enclosure (defined in Alt Dwell Lt Side).

4. Pause time of no coolant flow (defined in Alt Washdown Off Time).

The amount of washdown coolant used during machining operations for VMX64 and VMX84 machines is controlled by the time washdown coolant flows/pauses to each side of the machine enclosure, and/or by cycling the washdown coolant pump.

VMX64 and VMX84 machines are equipped with a separate Factory-configured timer to control the Y-axis rear way cover washdown on and off cycles.

These coolant parameters are found on Page 1 of Machine Parameters:

- Coolant Delay Time
- Pulsating or Delay Washdown Enable
- Alt Washdown Dwell
- Alt Dwell Lt Side
- Alt Washdown Off Time
- Washdown On Delay Timer
- Washdown Off Delay Timer

## Coolant Delay Time

The Coolant Delay Time parameter pauses the running part program for the indicated time whenever the program enables the primary or secondary flood coolant. Set the Coolant Delay Time parameter to pause the program long enough to allow flood coolant to pump from the coolant tank to the head-mounted nozzles around the spindle before the tool enters the workpiece.

The default setting for Coolant Delay Time is 0 seconds.

## Pulsating or Delay Washdown Enable

This parameter is used with other washdown coolant features to regulate washdown coolant flow for continuous clearing of chips from the machine enclosure.

Parameter operation is determined by machine type.

- **VMX24, VMX30, VMX42, VMX50, VMX60**—use this parameter to manage washdown coolant flow:

  - When Pulsating or Delay Washdown Enable = 0, washdown coolant will continuously flow on the right and left sides of the machine enclosure. Values in the Alt Washdown Dwell, Alt Dwell Lt Side, or Alt Washdown Off Time will have no effect on washdown coolant operation.

  - When Pulsating or Delay Washdown Enable = 1, do not use this setting for a VMX24, VMX30, VMX42, VMX50, or VMX60 machine. If unexpected performance occurs, refer to the Machine Maintenance section of the *Maintenance and Safety Manual*.

  - When Pulsating or Delay Washdown Enable = 2, the washdown coolant pump will cycle for the times defined in Washdown Delay On Timer and Washdown Delay Off Timer.

  - The default setting for Pulsating or Delay Washdown Enable is 0.

- **VMX64, VMX84**—use this parameter to alternate washdown coolant flow on the machine enclosure:

  - When Pulsating or Delay Washdown Enable = 0, do not use this setting for a VMX64 or VMX84 machine. If unexpected performance occurs, refer to the Machine Maintenance section of the *Maintenance and Safety Manual*.

  - When Pulsating or Delay Washdown Enable = 1, washdown coolant flow will cycle for the times defined in Alt Washdown Dwell, Alt Dwell Lt Side and Alt Washdown Off Time.

  - When Pulsating or Delay Washdown Enable parameter = 2, the washdown coolant pump will cycle for the times defined in Washdown On Delay Timer and Washdown Off Delay Timer, whether or not washdown coolant flow is alternating in the machine enclosure.

  - The default setting for Pulsating or Delay Washdown Enable is 1.

Pulsating or Delay Washdown Enable is not functional for HTX500, VM, and VTXU machines.

## Alt Washdown Dwell

This parameter controls the time washdown coolant flows on the right side of the machine enclosure. Alt Washdown Dwell is used with Alt Washdown Off Time to define the washdown coolant flow cycle on the right side of the machine enclosure. The value for Alt Washdown Dwell in 0.01 sec increments, so a value of 500 equates to 5 sec.

Pulsating or Delay Washdown Enable must be set to 1 or 2 to use Alt Washdown Dwell. Parameter operation is determined by machine type.

- **VMX24, VMX30, VMX42, VMX50, VMX60**

  - When Pulsating or Delay Washdown Enable = 1, do not use this setting for a VMX24, VMX30, VMX42, VMX50 or VMX60 machine. If unexpected performance occurs, refer to the Machine Maintenance section of the *Maintenance and Safety Manual*.

  - When Pulsating or Delay Washdown Enable = 2, Alt Washdown Dwell is not functional; entering a value in Alt Washdown Dwell will have no effect on washdown coolant flow for a VMX24, VMX30, VMX42, VMX50 or VMX60 machine.

  - The default setting for Alt Washdown Dwell is 500 (5 seconds), but set this parameter to 0 to avoid unexpected performance. If unexpected performance occurs, refer to the Machine Maintenance section of the *Maintenance and Safety Manual*.

- **VMX64**—When Pulsating or Delay Washdown Enable = 1 or 2, define the time for washdown coolant to flow on the right side of the machine enclosure.

  - The default setting for Alt Washdown Dwell is 500 (5 seconds).

- **VMX84**—When Pulsating or Delay Washdown Enable = 1 or 2, define the time for washdown coolant to flow on the right side of the machine enclosure.

  - The default setting for Alt Washdown Dwell is 1000 (10 seconds).

Alternating Washdown Dwell is not functional for HTX500, VM, and VTXU machines.

## Alt Dwell Lt Side

On VMX60, VMX64, and VMX84 machines, this parameter controls the time washdown coolant flows on the left side of the machine enclosure. Alt Dwell Lt Side is used with Alt Washdown Off Time to cycle washdown coolant flow on the left side of the machine enclosure. The value for Alt Dwell Lt Side is in 0.01 sec increments, so a value of 500 equates to 5 sec.

Pulsating or Delay Washdown Enable must be set to 1 or 2 to use Alt Dwell Lt Side. Parameter operation is determined by machine type.

- **VMX24, VMX30, VMX42, VMX50, VMX60**—Alt Dwell Lt Side is not functional; entering a value in Alt Dwell Lt Side will not affect washdown coolant flow for a VMX24, VMX30, VMX42, VMX50 or VMX60 machine.

    - The default setting for Alt Washdown Dwell is 200 (2 seconds), but you should set this parameter to 0 to avoid unexpected performance. If unexpected performance occurs, refer to the Machine Maintenance section of the *Maintenance and Safety Manual*.

- **VMX64, VMX84**—define the time for washdown coolant to flow on the left side of the machine enclosure.

    - Alt Dwell Lt Side will only function if the Alt Washdown Dwell is greater than 0. If the value of Alt Washdown Dwell is 0, the machine will ignore Alt Dwell Lt Side and washdown coolant will not flow on either side of the machine enclosure.

    - If Alt Dwell Lt Side = 0 and Alt Washdown Dwell is greater than 0, washdown coolant will flow to the left side of the enclosure for the time defined in Alt Washdown Dwell.

    - The default setting for Alt Dwell Lt Side is 200 (2 seconds).

Alternating Dwell Left Side is not functional for HTX500, VM, and VTXU machines.

## Alt Washdown Off Time

On VMX64 and VMX84 machines, this parameter defines the pause time in the washdown coolant flow cycle. The value is in 0.01 sec increments, so a value of 500 equates to 5 sec.

Pulsating or Delay Washdown Enable must be set to 1 or 2 to use Alt Washdown Off Time. Parameter operation is determined by machine type.

- **VMX24, VMX30, VMX42, VMX50, VMX60 -** Alt Washdown Off Time is not functional; entering a value in Alt Washdown Off Time will not affect washdown coolant flow for a VMX24, VMX30, VMX42, VMX50 or VMX60 machine.
    - The default setting for Alt Washdown Off Time is 0; this parameter is not functional for a VMX24, VMX30, VMX42, VMX50, or VMX60 machine.

- **VMX64, VMX84**—When Pulsating or Delay Washdown Enable = 1 or 2, washdown coolant flow will pause for the time indicated in the Alt Washdown Off Time after coolant flow on each side of the enclosure.
    - If Alt Washdown Off Time is set to 0, the washdown off-time is 0.7 sec.
    - The default setting for Alt Washdown Off Time is 200 (2 seconds).

Alternating Washdown Off Time is not functional for HTX500, VM, and VTXU machines.

## Washdown On Delay Timer

This parameter defines the time (in seconds) the washdown coolant pump is On, and is used with the Washdown Off Delay Timer to manage coolant usage. The default setting for Washdown On Delay Timer is 0 seconds.

Pulsating or Delay Washdown Enable must be set to 2 to use Washdown On Delay Timer.

- **VMX24, VMX30, VMX42, VMX50, VMX60**—washdown coolant flow will be continuous to both sides of the machine enclosure during the time the washdown coolant pump is On.

- **VMX64, VMX84**—the washdown coolant pump is On for the time defined, whether or not washdown coolant flow is alternating in the machine enclosure. *Table 3–1.Washdown Coolant Parameter Examples, on page 3 - 19* shows an example where the washdown coolant pump cycles on and off while washdown coolant flow is alternating on the machine enclosure.

    ⇨ It is recommended that you follow this formula:

    Washdown On Delay Timer = Alt Washdown Dwell + Washdown Off Delay Timer + ALT Dwell Left Side + Washdown Off Delay Timer

    See *Table 3–1.Washdown Coolant Parameter Examples, on page 3 - 19*.

- The default setting for Washdown On Delay Timer is 0 seconds.

This parameter defines the time (in seconds) the washdown coolant pump is On, and is used with the Washdown Off Delay Timer to manage coolant usage. Washdown On Delay Timer is not functional for VM machines.

## Washdown Off Delay Timer

This parameter defines the time (in seconds) the washdown coolant pump is Off, and is used with the Washdown On Delay Timer to manage coolant usage. The functionality of this feature is dependent upon machine type. Washdown Off Delay Timer is not functional for VM machines.

Pulsating or Delay Washdown Enable must be set to 2 to use this parameter.

- **VMX24, VMX30, VMX42, VMX50, VMX60**—defines the time the washdown coolant pump is Off.

- **VMX64, VMX84**—defines the time the washdown coolant pump is Off, whether or not coolant flow is alternating in the machine enclosure.

> It is recommended that you follow this formula:
>
> Washdown Off Delay Timer = Alt Washdown Dwell + Washdown On Delay Timer + Alt Dwell Left Side + Washdown On Delay Timer
>
> See *Table 3–1.Washdown Coolant Parameter Examples, on page 3 - 19*.

> Washdown Off Delay Timer should be calculated using even multiples of its value, increasing it by that value.
>
> For example, if the calculated value is 30, then you can increase it to 60, 90, etc.

- The default setting for Washdown Off Delay Timer is 0 seconds.

## Examples for Coolant Parameters Use

### VMX24, VMX30, VMX42, VMX50, and VMX60
### N95 Codes (Integrator Configuration Parameters)

| 247 | 228 | 248 | 249 | 297 | 298 | Result |
|-----|-----|-----|-----|-----|-----|--------|
| 0 | 0 | 0 | 0 | 0 | 0 | Continuous washdown coolant flow. |
| 2 | 0 | 0 | 0 | 20 | 45 | 1. Washdown coolant pump on for 20 sec.<br>2. Pump off for 45 sec.<br>3. Cycle repeats. |

### VMX64, VMX84
### N95 Codes (Integrator Configuration Parameters)

| 247[1] | 228 | 248 | 249 | 297 | 298 | Result |
|--------|-----|-----|-----|-----|-----|--------|
| 1 | 800 | 600 | 0 | 0 | 0 | 1. Right side washdown coolant for 8 sec.<br>2. No coolant for 0.7 sec.<br>3. Left side washdown coolant for 6 sec.<br>4. No coolant for 0.7 sec.<br>5. Cycle repeats. |
| 1 | 800 | 600 | 250 | 0 | 0 | 1. Right side washdown coolant for 8 sec.<br>2. No coolant for 2.5 sec.<br>3. Left side washdown coolant for 6 sec.<br>4. No coolant for 2.5 sec.<br>5. Cycle repeats. |
| 2 | 1000 | 1000 | 500 | 30 | 60 | 1. Right side washdown coolant for 10 sec.<br>2. No coolant for 5 sec.<br>3. Left side washdown coolant for 10 sec.<br>4. No coolant for 5 sec.<br>5. Washdown coolant pump on for 30 sec during right & left cycles.<br>6. Pump off for 60 sec.<br>7. Cycle repeats. |

1. A value of 3 programmed for N95:247 enables control of the Y-Axis Washdown.

### Key to N95 Code Column Headings for Tables Above

| Integrator Configuration Parameter | Definition |
|------------------------------------|------------|
| N95:247 | Pulsating or Delay WD Enable |
| N95:228 | Alt WD Dwell |
| N95:248 | Alt Dwell Lt Side |
| N95:249 | Alt WD Off Time |
| N95:297 | WD On Delay Timer |
| N95:298 | WD Off Delay Timer |

*Table 3–1.    Washdown Coolant Parameter Examples*

### Control Power Off Time

Turns the control off after the specified period of activity.

The default setting for Control Power Off Time is 120 minutes.

### Disable Tool Picker Option

Turns off the Tool Fixture (TPS) option, if it is present on your VM, VMX, or VTXU machine. Refer to the *Tool Fixture (TPS) Option, on page 12 - 1* in Options.

The default setting for Disable Tool Picker Option is 0.

The Disable Tool Picker Option parameter is not functional on HTX500 machines.

### Enable Dual Zones

This parameter enables zones on a dual-zone machine. 0 sets both zones as a single long bed machine, and 1 sets the zones as separate zones. The default is 1.

## Machine Parameters Page 2



*Figure 3–2. Machine Parameters, HTX500 (left) and VTXU (right), Page 2*

**Machine Parameters - Page 2**

| Parameter | Description | Range | Default |
|---|---|---|---|
| CAL to LS Velocity X, Y, Z | Sets the feedrate for the X, Y, or Z axis as it moves toward the calibration limit switch during a machine calibration cycle. See Cal to LS Velocity Parameters. | 100-2540 MMPM | 1270 |
| CAL to LS Velocity A, B, C | Sets the feedrate for the A, B, or C axis as it moves toward the calibration limit switch during a machine calibration cycle. See Cal to LS Velocity Parameters. | 100-2540 DPM | 0 |
| Auto Balance Enable | Adjusts the balance between the motion card and the servo drives at the start of calibration and run program. | 0 (disable) 1 (enable) | 1 |
| ATC Disable | Disables all automatic tool changer functions. | 0 (enable) 1 (disable) | 0 |
| Rapid Override Disable | Enables or disables the Rapid Override potentiometer on the console. | 0 (enable) 1 (disable) | 0 |
| Tilt Axis Safety Position | Sets the position for the tilt axis during an automatic tool change when the Table Safety Move parameter is set to Yes for an Automatic Tool Change. | 0-360 degrees | 0 |
| X-Axis Safety Position | Sets the absolute X axis machine location to which the table will move when the Table Safety Move parameter is set to Yes for an Automatic Tool Change. See X-Axis and Y-Axis Safety Position. | Travel Limits (mm) | 0 |
| Y-Axis Safety Position | Sets the absolute Y axis machine location to which the table will move when the Table Safety Move parameter is set to Yes for an Automatic Tool Change. See X-Axis and Y-Axis Safety Position. | Travel Limits (mm) | 0 |
| ATC Z-Axis Move to Zero Position | On HTX500 only, moves the Z-axis to zero position at the end of a tool change. | 0 (disable) 1 (enable) | 0 |

# Cal to LS Velocity Parameters

The Cal to LS Velocity parameters set the feed rate for each axis as it is moving towards the calibration limit switch during a machine calibration cycle. Only the axes that are present on the machine need to be set. The units are indicated in parentheses and can be MMPM (millimeters per minute), IPM (inches per minute), or DPM (degrees per minute).

- The default setting for Calibrate to Limit Switch Velocity is 1270 MMPM (50 IPM) for the X, Y, and Z axes. The range for Cal to LS Velocity values for the X, Y, and Z axes is 100 MMPM to 2540 MMPM.

- The default setting for Calibrate to LS Velocity is 0 DPM for the A, B, and C axes. The range for Cal to LS Velocity values for the A, B, and C axes is 100 DPM to 2540 DPM.

- The default setting for Calibrate to Limit Switch Velocity is 1270 mmpm (50 IPM) for X, Y, and Z axes.

- The default setting for Calibrate to Limit Switch Velocity is 1000 mmpm (39.37 IPM) for A axis on a VTXU machine.

- The default setting for Calibrate to Limit Switch Velocity is 2000 (78.74 IPM) mmpm for B axis on an HTX500 machine.

- The default setting for Calibrate to Limit Switch Velocity is 1000 mmpm (50 IPM) for C axis on a VTXU machine.

# Auto Balance Enable

Auto Balance Enable set to 1 will have the logic controller send an auto balance request to adjust the balance between the motion card and the servo drives at the start of calibration and run program. No auto balance request will be sent if the parameter is 0.

The default setting for Auto Balance Enable is 1.

# ATC and Rapid Override Disable Parameters

## ATC Disable

The ATC Disable parameter allows the user to completely disable all automatic tool change functions. Setting the ATC Disable parameter to 1 disables the Auto Tool Change button (for performing tool changes in Manual mode or Tool Setup) and the Auto Tool Change in Auto mode button (for running programs). With ATC Disable set to 1, all tools will have to be inserted and removed manually using the Spindle Clamp/Unclamp button mounted on the head.

⚠ When ATC is disabled, the Control assumes that the ATC is in a safe position to allow normal X, Y, and Z axis movement. Input status for the ATC at home position will not be checked. Failure to ensure that the ATC is in a safe position may result in machine damage.

The default setting for ATC Disable is 0.

## Rapid Override Disable

The Rapid Override Disable parameter allows the user to enable or disable the Rapid Override potentiometer on the console. Rapid Override Disable set to 1 disables the Rapid Override potentiometer so that all rapid movements in an automatic operation will be at the programmed setting. Rapid Override Disable set to 0 enables the Rapid Override so that rapid movements in an automatic operation may be adjusted slower than the programmed setting.

The default setting for Rapid Override Disable parameter is 0.

# Axis Safety Position Parameters

## Tilt Axis Safety Position

The Tilt Axis Safety Position parameter sets the position for the tilt axis on the VTXU machine during an ATC if either one of the following conditions occurs:

- the Move to Safe Pos During TC parameter is set to Yes (for Auto Run mode).
- the Move to Safety Pos Manual Mode ATC parameter (for Manual or Input modes) is set to 1.

After the Z retracts before the ATC, the tilt axis moves to the position indicated in degrees. After the automatic tool change completes, the tilt axis returns to the original position.

The default setting for Tilt Axis Safety Position is 0.

The Tilt Axis Safety Position parameter is not functional for HTX500 machines, VM machines, or VMX machines.

## X-Axis and Y-Axis Safety Position

The X-Axis Safety Position and Y-Axis Safety Position parameters determine the absolute X-axis and Y-axis machine locations, in millimeters, to which the table will move when the Table Safety Move parameter is set to Yes for an Automatic Tool Change. See Program Parameters for more information.During an automatic tool change with the Table Safety Move set to Yes:

1. The Z-axis will retract to maximum position.

2. The X-axis and Y-axis will move at rapid feedrate to the X-Axis Safety Position and the Y-Axis Safety Position, respectively.

3. The automatic tool change will complete.

4. The X-axis and Y-axis will move at rapid feedrate to the previous X-axis and Y-axis positions before the move to the X-Axis and Y-Axis Safety Positions.

The X-Axis Safety Position value must not exceed the minimum or maximum travel limits for the X-axis. The Y-Axis Safety Position value must not exceed the minimum or maximum travel limits for the Y-axis. If the Table Safety Move parameter is set to No, these parameters are not used, and steps 1, 2, and 4 above are not executed for an automatic tool change.

The default setting for X-Axis Safety Position is 0.000 mm. The default setting for Y-Axis Safety Position is 0.000 mm.

The X-Axis and Y-Axis Safety Position parameters are not functional for HTX500 machines.

# ATC Z-Axis Parameter

## ATC Z-Axis Move to Zero Position

This parameter is functional only on an HTX500 machine. The ATC Z-Axis Move to Zero Position parameter moves the Z-axis to the zero (0) position at the end of either an Auto or Manual tool change.

ATC Z-Axis Move to Zero Position set to 0 disables the parameter. ATC Z-Axis Move to Zero Position set to 1 enables the parameter.

The default setting for ATC Z-Axis Move to Zero Position is 0.

ATC Z-Axis Move to Zero Position parameter is not functional for VTXU, VM, or VMX machines.

# Machine Parameters Page 3



```
                   MACHINE PARAMETERS
                        (PAGE 3)                    -PAGE 1-
DESCRIPTION                     RANGE                         F1

AUX OUTPUT 1 CONFIRMATION ENABLE    0-1        [0]   -PAGE 2-
AUX OUTPUT 2 CONFIRMATION ENABLE    0-1        [0]            F2
AUX OUTPUT 3 CONFIRMATION ENABLE    0-1        [0]
AUX OUTPUT 4 CONFIRMATION ENABLE    0-1        [0]   -PAGE 3-
DISABLE AUX OUT 1 DURING INTERRUPT  0-1        [0]            F3
DISABLE AUX OUT 2 DURING INTERRUPT  0-1        [0]
DISABLE AUX OUT 3 DURING INTERRUPT  0-1        [0]   -PAGE 4-
DISABLE AUX OUT 4 DURING INTERRUPT  0-1        [0]            F4
MOVE TO SAFETY POS MANUAL MODE ATC  0-1        [0]
CHIP CONVEYOR ON/OFF DELAY ENABLE   0-1        [0]           F5
CHIP CONVEYOR ON DELAY TIME (SEC)   0-9999   [0]
CHIP CONVEYOR OFF DELAY TIME (SEC)  0-9999   [0]            F6

                                                           F7

                                                    EXIT
                                                           F8
```

*Figure 3–3. Machine Parameters, Page 3*

**Machine Parameter - Page 3**

| Parameter | Description | Range | Default |
|---|---|---|---|
| Aux Output 1, 2, 3, 4 Confirmation Enable | Enables a confirmation signal for completion of each Auxiliary M-code Output. | 0 (disable 1 (enable) | 0 |
| Disable Aux Out 1, 2, 3, 4 During Interrupt | Disables the specified Auxiliary M-code Output when an Interrupt cycle is selected during Auto Run mode. | 0 (enable) 1 (disable) | 0 |
| Move to Safety Pos Manual Mode ATC | Manual Mode ATC operations are performed at Safety Position when enabled. | 0 (disable) 1 (enable) | 0 |
| Chip Conveyor On/Off Delay Enable | Enables the Chip Conveyor Delay. | 0 (disable) 1 (enable) | 0 |
| Chip Conveyor On Delay Time | Sets the time the Chip Conveyor cycles on, when Chip Conveyor On/Off Delay is enabled. | 0-9999 seconds | 0 |
| Chip Conveyor Off Delay Time | Sets the time the Chip Conveyor cycles off, when Chip Conveyor On/Off Delay is enabled. | 0-9999 seconds | 0 |

# Aux Output Parameters

## Aux Output 1, 2, 3, 4 Confirmation Enable

Aux Output Confirmation Enable set to 1 enables a confirmation signal for completion of each Auxiliary M-code Output. The program pauses until the confirmation signal is detected. A setting of 0 will not require a confirmation signal and will not pause the program during the Auxiliary M-code Output.

The default setting for each Auxiliary Output Confirmation Enable is 0.

## Disable Aux Out 1, 2, 3, 4 During Interrupt

When set to 1, the parameter Disable Aux Output During Interrupt disables the specified Auxiliary M-code Output when an Interrupt cycle is selected during Auto Run mode. Upon return to Auto Run mode, the Auxiliary M-code Output will automatically be re-enabled. However, if another mode is selected, then the Auxiliary M-code Output will remain disabled until activated again in Auto Run mode.

A setting of 0 will leave the Auxiliary M-code Output enabled when an Interrupt cycle is selected during Auto Run mode. It will remain enabled after exit from Interrupt cycle, even when accessing a mode other than Auto Run.

The default setting for Disable Aux Out During Interrupt is 0 (enable).

# Move to Safety Pos Manual Mode ATC

When the Move to Safety Post Manual Mode ATC operations parameter is set to 0 for an ATC command, the Z axis moves to tool change height and the ATC is completed at the current X and Y axis position.

The default setting for Move to Safety Pos Manual Mode ATC is 0 (disable).

If this parameter is set to 1, then:

1. The Z axis moves at rapid to zero.

2. The X and Y axes move at rapid to the X- and Y-axis safety positions.

3. The Z axis moves to tool change height.

4. The automatic tool change is completed.

5. The Z axis moves to zero.

6. The X and Y axes move at rapid back to the previous X and Y position.

# Chip Conveyor Parameters

⇨ Chip Conveyor On/Off Delay Enable, Chip Conveyor On Delay Time and Chip Conveyor Off Delay Time parameters are functional only when the **Chip Removal Forward On/Off *F2*** softkey (Automatic Run Mode) is set to On.

## Chip Conveyor On/Off Delay Enable

The Chip Conveyor On/Off Delay Enable parameter is used with the Chip Conveyer On Delay Time and Chip Conveyor Off Delay Time parameters to periodically allow the chip conveyor or auger to remove chips from inside the machine during the part program.

When Chip Conveyor On/Off Delay Enable is set to 0, this parameter is disabled. Values entered into the Chip Conveyor On Delay Time and Chip Conveyor Off Delay Time will not be retained.

When Chip Conveyor On/Off Delay Enable is set to 1, use the Chip Conveyor On Delay Time and Chip Conveyor Off Delay Time parameters to regulate operation of the chip conveyor or auger.

The default setting for Chip Conveyor On/Off Delay Enable is 0 (disable).

## Chip Conveyor On Delay Time

When the Chip Conveyor On/Off Delay Enable parameter is set to 1, the Chip Conveyor On Delay Time parameter defines the time that the chip conveyor or auger cycles On to remove chips from inside the enclosure.

The default setting for Chip Conveyor On Delay Time is 0 seconds.

## Chip Conveyor Off Delay Time

When the Chip Conveyor On/Off Delay Enable parameter is set to 1, the Chip Conveyor

Off Delay Time parameter sets the time that the chip conveyor or auger cycles Off. When the chip conveyor or auger cycles Off, the screen status display is FWD-Delay, not Stopped.

The default setting for Chip Conveyor Off Delay Time is 0 seconds.

## Machine Parameters Page 4



*Figure 3–4. Machine Parameters, Page 4*

## Machine Parameters - Page 4

| Parameter | Description | Range | Default |
|---|---|---|---|
| Warm-Up Cycle Time Per Pass | Sets the time for each step of the warm-up cycle. | 60-600 | 120 |
| Warm-Up Starting Speed | Sets the spindle speed for the initial step of the warm-up cycle. | 0-25% Max RPM | 1000 |
| Warm-Up Speed Steps | Sets spindle speed increments for each step of the warm-up cycle. | 0-25% Max RPM | 2000 |
| Warm-Up Max Spindle Speed | Sets the spindle speed for the final step of the warm-up cycle. | 0-Max RPM | Machine's maximum spindle RPM |
| Warm-Up Axis Feed Rate | Sets the axis feed rate for each step of the warm-up cycle. | 0-Max Rapid | 2920 |
| Disable Auto On Chip Removal | Disables chip conveyor from turning on automatically in AUTO mode. | 0-1 | 0 |
| Axis Feedrate Override Min (%) | Sets the Axis Feedrate Override Minimum value. | 0-99% | 10% |
| Axis Feedrate Override Max (%) | Not accessible by the user. Contact a Hurco Certified Service representative for assistance. | 101-200 | 150 |
| Rapid Feedrate Override Min (%) | Sets the Rapid Feedrate Override Minimum value. | 0-200% | 10% |
| Rapid Feedrate Override Max (%) | Not accessible by the user. Contact a Hurco Certified Service representative for assistance. | 0-200 | 100 |
| Dual Tool Probe Present | Sets tool probe in one or both zones of dual-zone machines. | 0 (one) 1 (both) | 0 |

# Warm Up Parameters

These parameters allow the operator to configure the machine's daily warm-up cycle.

- Warm-up Cycle Time Per Pass

- Warm-up Starting Speed

- Warm-up Speed Steps

- Warm-up Max Spindle Speed

- Warm-up Axis Feed Rate

⚠️ The Z-axis is at zero and does not move during the machine warm-up. Tool In Spindle setting must be 0 before beginning Warm Up Machine. For the VM1G machine, a tool must be in the spindle before beginning Warm Up Machine.

When the warm-up cycle is initiated:

- Spindle speeds up to the Warm-up Starting Speed RPM for the time indicated in the Warm-up Cycle Time per Pass field.

- X and Y axes move diagonally from corner to corner at the Warm-up Axis Feed Rate.

- The machine then steps through the warm-up cycle until the Warm-up Max Spindle Speed is reached. The length of each step is defined by the Warm-Up Cycle Time per Pass. During each step of the cycle, the spindle speed will increase to the Warm-up Speed Steps RPM and the axes will run at the Warm-up Axis Feed Rate.

⚠️ Altering the default daily warm-up parameters may lead to inadequate machine warm up that can negatively impact spindle operations and may void the Hurco warranty.

See Machine Warm-up for warm-up cycles for a new spindle or a machine that was idle 30 days or more.

# Disable Auto On Chip Removal

The Disable Auto On Chip Removal parameter disables the chip conveyor from turning on automatically in Auto Mode. The range is 0 or 1; 0 is the default. Setting the parameter to 1 will disable the Auto On Chip Removal feature.

# Axis Feedrate Override Min (%)

Sets the Axis Feedrate Override Minimum value. This value cannot be greater than or equal to the maximum value.

# Rapid Feedrate Override Min (%)

Sets the Rapid Feedrate Override Minimum value. This value cannot be greater than or equal to the maximum value.

# Dual Tool Probe Present

The Dual Tool Probe Present parameter sets tool probe in one or both zones of dual-zone machines. 0 specifies there is one tool probe; 1 specifies there is a probe in each zone. The default is 0.

# Serial Port Settings

Serial Port Settings allows you to set Port 1 and Port 2 settings for protocol, baud rate, character length, stop bits, and parity. The following fields appear on the Serial Port Settings screen. Refer to your hardware documentation for your equipment's specific settings.

- **Protocol—**The format in which the data is communicated. Choose from CTS/RTS (hardware flow control), Xon/Xoff (software flow control), and Full Handshake

- **Baud Rate—**The communication speed rate. Choose a baud rate that is appropriate for your network from the drop-down list.

- **Character Length—**The maximum number of bits sent back and forth at one time.

- **Stop Bits—**Stop bits signal the end of the transmission of data. Choose the size for the Stop Bit.

- **Parity—**Provides simple error checking for the transmitted data.

  Select Even if the data bits plus the parity bit result in an even number of 1's

  Select Odd if the data bits plus the parity bit result in an odd number of 1's.

  Select None if there is no parity bit included in the transmission. When No is selected, its assumed that there are other forms of checking that will detect any errors in transmission.

  For example, suppose the data bits 01110001 are transmitted to your computer. If even parity is selected, then the parity bit is set to 0 by the transmitting device to produce an even number of 1's. If odd parity is selected, then the parity bit is set to 1 by the transmitting device to produce an odd number of 1's.

## FTP Server Settings

The WinMax console can serve as an FTP Server. The following fields appear on the FTP Server Settings screen. For more information refer to *UltiNet, on page 14 - 1* in Options.

- **Enable FTP Server—**Enable the WinMax Control to act as an FTP Server.
- **FTP Server Port—**The port number for FTP access.
- **Maximum Idle Time (Mins)—**Maximum amount of time before connection is dropped.
- **User Name—**The log in name that will allow users access to the FTP Server.
- **Password—**The password that corresponds to the user name and allows access to the FTP Server.
- **Path—**The file path that the users will be allowed access to.

## WinMax Uptime

The WinMax Uptime screen displays the start date time and runtime for the current session of Winmax.

## Select Language

Select and register a language on the Language Selection screen:

1. Select an installed language. Languages that have not been installed and registered are grayed out on the list.
2. Press the SELECT/LOAD *F1* softkey to install the Language files. You can also press the F1 key to perform this function.

### Language Registration

To register a new language:

1. Press the REGISTRATION *F3* softkey on the Language Selection screen to register alternate languages and enter map and hlp filenames.

   - Define the language to appear on the screen by using the drop-down list or select a softkey.
   - Select the Map filename and HLP filename for the language. WinMax requires the HLP file for the online Help.
   - Select the Register button so the software registers the language selection.

2. Select EXIT to return to the Language Selection screen.

## Data Logging Filters

Used for diagnostic purposes only.

# Additional Utilities Softkeys

- **Printing Setup** *F3*—set printing preferences for program blocks, range of program blocks, program parameters, part setup, and tool setup.

- **Integrator Support Services** *F4*—for diagnostics and machine configuration; password required for access.

- **Restart Control** *F5*—exits all control operations, powers down, then restarts machine and control.

- **Shutdown Control** *F6*—exits all control operations and powers down machine.

- **Serial I/O** *F1*—to begin read/write program operations using screenport 1 or 2.

- **Log Files** *F2*—displays error and status messages that have occurred during normal operation.

# Printing Setup

The Printing Setup F3 softkey on the main Utilities screen provides access to the following printing functions:

- **Part Program Printing** *F1*—print part program elements.
- **Tool Information Printing** *F2*—print the Tool Library or save it to file.
- **Probing Data Printing** *F3*—print probing part inspection files.

## Part Program Printing

In the Part Program Printing screen you can choose to print some or all of the following elements:

- Program Blocks (all or a range)
- Program Parameters
- Part Setup
- Tool Setup

After checking the desired elements, select the PRINTING F3 softkey. A pop-up window appears that contains the selected data. The information can be viewed by selecting the heading to expand the section; select the heading again to hide the information. The following softkeys appear in the pop-up window:

- Print Preview—displays the part program information as it will appear when printed.

- Print—sends the information to the printer.

- Save As—saves the information as a text file to an external drive.

- Close—closes the pop-up window.

## Probing Data Printing

When Probing Part Inspection files are generated, they are saved to the same location and with the same filename as the part program with a .TXT filename extension. For example, the Part Inspection file for the program SAMPLE.HWM is SAMPLE.TXT. Part Inspection files are printed from the control using the PROBING DATA PRINTING softkey:

1. Select the PRINTING SETUP F3 softkey from Utilities.

2. Select the PROBING DATA PRINTING F3 softkey.

3. Locate the Part Inspection file from the Locate Probing Data Files screen.

4. Select the PRINT F1 softkey to print the file.

   ⇨ Subsequent Part Inspection data from the same part program is appended to the existing Part Inspection file for that program.

# Integrator Support Services

⬦ The Integrator Support Services screen requires a password to access and is for Hurco Certified Technicians' use in configuring and setting up the machine.

# Restart Control

The Restart Control command will remove control power, save all open programs in Project Manager, complete an orderly shutdown of the WinMax Mill control, and then restart.

Follow these steps to use the Restart Control command:

1. Press the **Aux/Menu** key followed by the **Utility Screen** softkey to access the Utilities screen and softkey menu.

2. From the Utilities screen, select the Restart Control softkey to access the Restart Control command. A pop-up message appears: "Do you really want to restart the control?"

3. Select Yes or No. When Yes is selected, the Restart Control command is performed. This feature saves time and avoids having to shut down the entire machine when it is only necessary to restart the control.

# Shutdown Control

The Shutdown Control command will remove control power, save all open programs in Project Manager, and then complete an orderly shutdown of the WinMax control.

⇨ Hurco recommends using the Shutdown Control command prior to turning off machine power to ensure that no data is lost.

1. Follow these steps to use the Shutdown Control command:

2. Press the **Aux/Menu** key followed by the **Utility Screen** softkey to access the Utilities screen and softkey menu.

3. From the Utilities screen, select the Shutdown Control softkey to access the Shutdown Control command. A pop-up message appears: "Do you really want to shut down the control?"

4. Select Yes or No. When Yes is selected, the Shutdown Control command is performed.

5. Wait for the shutdown process to finish before shutting off machine power at the Main Disconnect switch.

# Serial I/O

Two serial ports are available on the control. The Serial I/O screen contains Status and Bytes Transferred fields for both ports. In addition, there are read, write, and abort softkeys for both ports.

These are the fields on the Serial I/O screen:

- **Status—**status of serial port.

- **Bytes Transferred—**number of bytes transferred.

These softkeys are available on the Serial I/O screen. Duplicate sets of fields and softkeys are available on the Serial I/O screen for Port 1 and Port 2.

- **Begin Reading from Port**—brings up 2 softkey choices: READ NC FROM PORT and READ CONV FROM PORT to identify the program format to read.

- **Begin Writing to Port**—writes the program to the port.

- **Abort Port Operation**—halts the read or write operation for the port.

# Log Files

WinMax provides two log files that are accessed with the **Log Files** softkey, as well as status and error listings:

- *Active Error Listing, on page 3 - 35*

- *Active Status Listing, on page 3 - 35*

- *Error History, on page 3 - 36*

- *Status History, on page 3 - 36*

- *Retrieve Log and Diagnostic Files, on page 3 - 37*

- *Export Log, on page 3 - 37*

## Active Error Listing

WinMax provides a list of the most recent error messages displayed, up to a maximum of 200 messages. Review messages when troubleshooting or to determine if a problem recurs.

These are the softkeys on the Active Error Listing screen:

- **Previous Page *F1*—**displays the error messages on the previous page of the Active Error Listing.

- **Next Page *F2*—**displays the error messages on the next page of the Active Error Listing.

- **Clear All *F4*—**clears all error messages from the Active Error Listing.

## Active Status Listing

The Active Status Listing screen displays current machine status messages. Each item will include time/date stamp, language file, language file index and the machine status message currently active in the system. Examples are messages such as "MOTION HOLD HAS BEEN DEPRESSED," or "LOW LUBE LEVEL." As each status changes back to a normal state, it will be removed from the list (see *Status History, on page 3 - 36*).

Example:

10:53:39, STATUS.WRC, 32

CHIP DOOR(S) OPEN.

These are the softkeys on the Active Status Listing screen:

- **Previous Page *F1*—**displays the status messages on the previous page of the Active Error Listing.

- **Next Page *F2*—**displays the status messages on the next page of the Active Error Listing.

- **Clear All *F4*—**clears all status messages from the Active Status Listing.

# Error History

The Error History screen displays a list of all system errors since the last power up (see *Active Error Listing, on page 3 - 35*). A plus sign "+" indicates when the error occurred and a minus sign "-" indicates when the error was cleared.

Examples:

Error occurred:

10:53:40, ERROR.WRC, 110, +

SERVO FAULT – Z AXIS.

Error cleared:

10:57:32, ERROR.WRC, 110, -

SERVO FAULT – Z AXIS.

The history will also display part program errors generated during programming (interpreter errors). These errors will not display in the Active Error Listing since they are transient.

Example:

11:39:21, COMPILER.WRC, 3, +

ERROR IN BLOCK 1: TOOL 1 IS AN INVALID TOOL NUMBER.

# Status History

The Status History screen displays a list of all system messages since the last power up (see *Active Status Listing, on page 3 - 35*). A plus sign "+" indicates when the status went active and a minus sign "-" indicates when the status returned to normal.

Examples:

Status went active:

10:53:39, STATUS.WRC, 32, +

CHIP DOOR(S) OPEN.

Status returned to normal: Log F

10:54:27, STATUS.WRC, 32, -

CHIP DOOR(S) OPEN.

## Retrieve Log and Diagnostic Files

When the Retrieve Log and Diagnostic Files softkey is selected, a file manager screen opens with access to Log and Diagnostic files. Directories are displayed in the left pane; these include AtcMapLog, Communications, DumpFiles, NavErr, NavTap, NavESTOP, Profiles, ScreenCaptures, and ThreadMonitor. Select a directory to see its contents (the files) displayed in the right pane. To help distinguish between files, view the date and time they were created, which is displayed in the Date column.

When it is necessary to copy certain files for service purposes, follow these steps:

⇨ Due to the large size of some of the files, it is recommended to copy them from the Hurco machine to a USB flash memory device (128M or larger).

1. Attach the USB flash memory device to the external connector on the communications port on the machine.

2. Select the appropriate directory from the left-hand pane and select the file(s) you wish to copy from the right-hand pane.

3. Select the COPY softkey.

4. In the left-hand pane, select the USB flash memory device (will probably indicate it is E drive in parentheses).

5. Select the PASTE softkey. The files are now on the USB flash memory device, and can be transferred to a PC to be emailed.

## Export Log

The Export Log displays data that is lost or changed when a program is exported from WinMax (.HWM) to another format (.HD3, .FNC, etc.). Each time a program is exported from .HWM to another format, the data from the most recent export is displayed and the older data is erased.

These are the softkeys on the Export Log screen:

- **Previous Page *F1*—**displays the previous page if log is longer than one page.

- **Next Page *F2*—**displays the next page if log is longer than one page.

- **Clear All *F4*—**erases all information from the Export Log.

# PROGRAMMING BASICS

The following sections explain basic programming information for Conversational and NC programming, such as required setup, program checking, editing, and running of the program.

# Input Mode

Input mode is used for part and tool setup, part programming, setting parameters, and other information entry. Press the Input key on the console to access the Input screen.

The following softkeys appear on the Input screen:

- **Import Functions** *F1*—access the Part Setup screen to establish part zero, centerline, offset Z, safety work region, and other parameters.

- **Tool Review** *F2*—access the Tool Review screen for a summary listing of all tools used in active program (see *Part Program Tool Review, on page 4 - 30* for more information).

- **Part Programming** *F3*—access and create data blocks of a part program. The current program's data blocks appear on screen when this softkey is used. Delete, add, edit, and navigate through the data blocks. The NC editor is displayed for NC programs.

- **Program Parameters** *F4*—access General, Milling, Holes, Probing, and Performance parameters. NC parameters are accessible for NC programs.

- **Import Functions** *F5*—import sections of a previously saved program.

- **Copy and Change Blocks** *F6*—make changes to several blocks at one time and/or copy and change blocks within the active program.

- **Erase Functions** *F7*—erase sections of the current program.

- **Program Manager** *F8*—access and manage other part programs.

## Erase Functions

Erase functions erases (deletes) programs or components (part setup, tool setup, and program parameters).

New WinMax features are:

- **ERASE PART SETUP** *F1*—resets Part Setup to the default values.

- **ERASE TOOL SETUP** *F2*—deletes tools in Manual Tools, Auto Tools, and spindle.

  ⚠ The Erase Tool Setup softkey deletes ALL tools from the control, and cannot be undone.

- **RESET PROGRAM PARAMETERS** *F3*—resets Program Parameters to the default values.

- **ERASE PROGRAM** *F4*—deletes the part program data blocks but retains the program name.

- **UNLOAD PROGRAM** *F5*—removes program from the Program Manager, including the filename. This does the same function as CLOSE PROGRAM in the Program Manager.

# Part Setup

See also:

*Part Setup Fields, on page 4 - 8*

*Part Setup Softkeys, on page 4 - 11*

Part setup establishes the locations of part zero in X, Y, and Z relative to the machine's absolute zero. Part Zero may be located anywhere on the fixturing or the part. During machine calibration, each axis moves to its + or - travel limits. Machine zero, identified during machine calibration, is the location to which each axis moves to determine a fixed point where the X, Y, and Z axes become tangent. This value does not change after calibration.



| 1 | Part Zero |
|---|---|
| 2 | Workpiece |
| 3 | Machine Zero |
| 4 | X Axis |
| 5 | Y Axis |
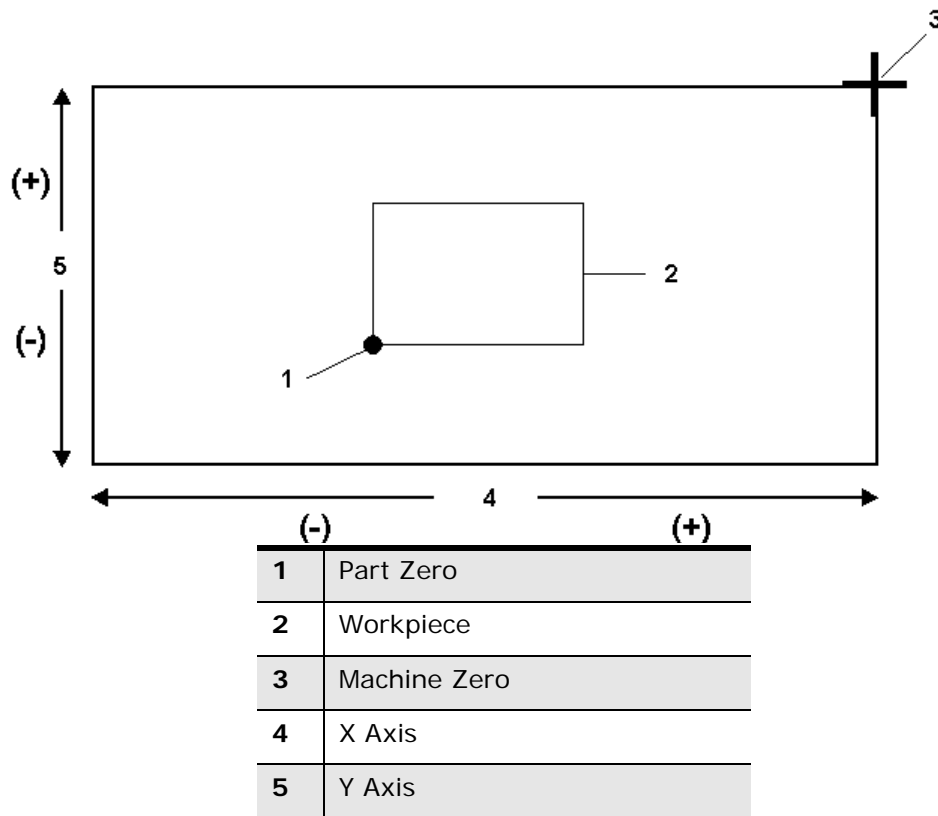
*Figure 4–1. Part Zero Relative to Machine Zero Viewed Looking Down at Table*

Program the axes to move within the coordinate system as shown below:
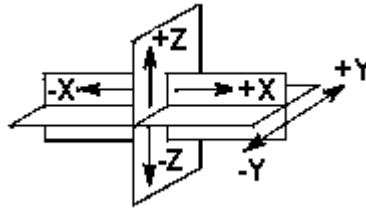


| - X | Table moves to the left |
|-----|-------------------------|
| +X  | Table moves to the right |
| - Y | Table moves toward the operator |
| +Y  | Table moves away from the operator |
| - Z | Spindle moves down and into the part |
| +Z  | Spindle moves up and away from the part |

*Figure 4–2. Axis Motion*

The location of part zero in Z is established during tool calibration. During setup, the tool zero calibration plane is established for each tool. The program dimensions for a part are relative to these points, viewed facing the machine, as shown below:



| 1 | Table |
|---|-------|
| 2 | Part (workpiece) |
| 3 | Machine Absolute - Z Axis |
| 4 | Tool Zero Calibration Plane |
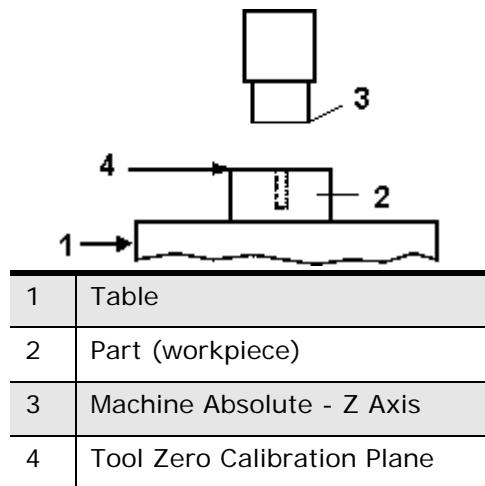
*Figure 4–3. Part Zero in Z (Tool Zero Calibration Plane)*

Select the **Toggle Units** softkey to change between inch and metric units of measurement.

# Part Setup Coordinate Systems

## Machine Coordinate System

The Machine Coordinate System is fixed to the machine frame and does not move when the machine axes move. This coordinate system is located at the spindle nose center when all machine axes are at their zero positions.

## Unrotated Coordinate System

The Unrotated Part Coordinate System is present on a 4-axis or 5-axis machine and is located at the Workpiece Coordinate system when all the machine's axes are at their Part Setup Zero locations. This coordinate system moves with the machine's linear axes, but does not rotate with the rotary axes. The Unrotated Coordinate system is generally used only by NC Post Processors that do not use Tool Center Point Management.

## Workpiece Coordinate System

The Workpiece Coordinate System is fixed to the physical workpiece fixtured to the table; it moves and rotates when the machine axes move. The coordinate system is fixed to the part fixtured on the machining center's table. The Workpiece Coordinate System is typically set in the user's CAD/CAM system or on the part drawing used for programming the tool paths.

The Machine Coordinate, Workpiece Coordinate, and Unrotated Coordinate systems are shown in the figure below:
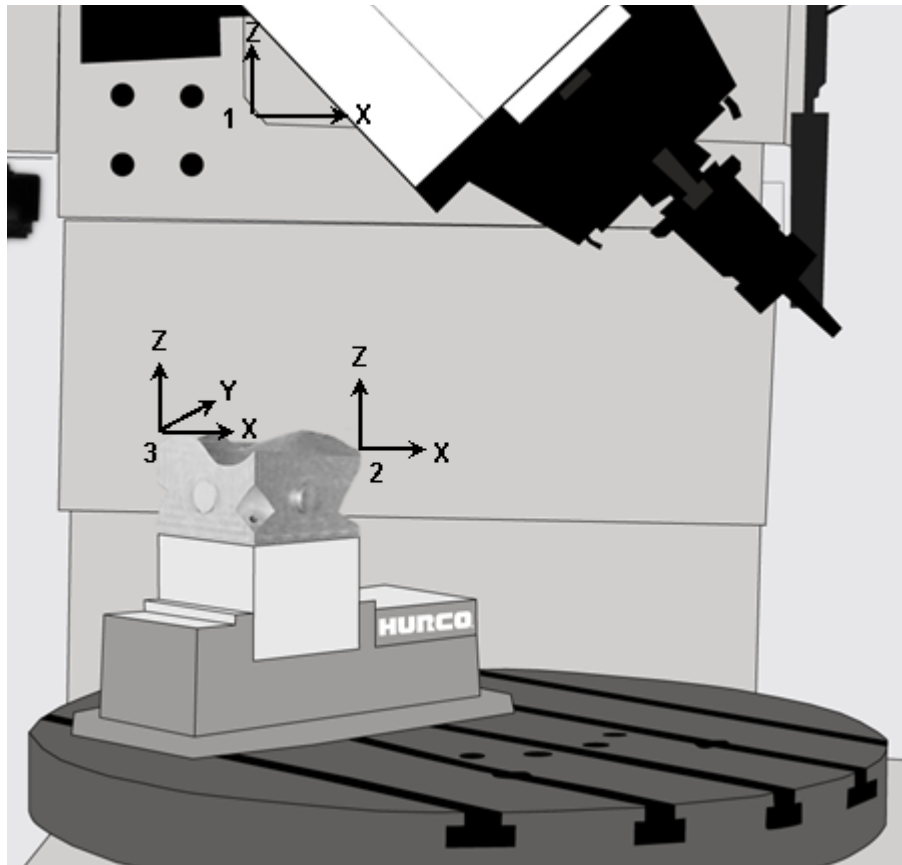


| 1 | Machine Coordinate System |
|---|---|
| 2 | Unrotated Part Coordinate System |
| 3 | Workpiece Coordinate System |

*Figure 4–4. Part Setup Coordinate Systems*

# 5-Axis Part Setup Fields

The Part Setup screen for a 5-axis machine contains these two fields:

- **Z Table Offset**—directional distance from the C-axis table to the X-Y plane of the *Workpiece Coordinate System, on page 4 - 5*. Z Table Offset is positive in the part setup shown in the figure below.

- **Offset Z**—directional distance from Zero Calibration (described below) to the X-Y plane of the *Workpiece Coordinate System, on page 4 - 5*. Offset Z is positive in the part setup shown in the figure below.

- **Zero Calibration**, the tool calibration height, is also used to set up a part on a 5-axis machine. Zero Calibration is defined on the Tool Setup screen. Zero Calibration is positive in the part setup shown in the figure below; however, in the NC Tool Offsets table it is negative.

  ⇨ - Ensure that the B-axis angle is at zero degree when measuring Z Table Offset, Offset Z, and Zero Calibration fields for 5-axis machining.
  - For conversational programs, B-axis Part Setup Offset must be zero degree.
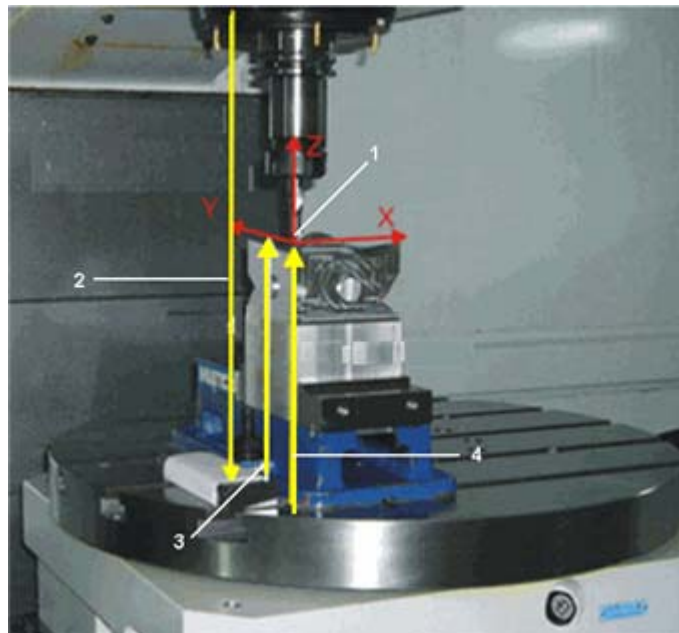


| 1 | Workpiece Coordinate System |
| 2 | Zero Calibration (positive value) |
| 3 | Offset Z (positive value) |
| 4 | Z Table Offset (positive value) |

*Figure 4–5. 5-Axis Part Setup*

# Part Setup Fields

The Part Setup screen contains these fields:

- **Part Zero X / Y—**defines the X and Y locations in machine coordinates for part zero.

- **Part Zero A / B /C—**defines the A, B, and/or C locations of Rotary coordinates for part zero.

- **A /B /C Centerline X/ Y / Z—**defines incremental distances from true part zero. Program these values in order to draw the program properly on the Graphics screen.

    ⇨ For rotary programs, Hurco recommends automatically calculating the rotary centerline. See *Rotary Part Setup, on page 6 - 8* for more information.

- **Part Zero Z (Absolute Tool Length Mode)**—defines the distance from the tool calibration plane to the workpiece zero. This field defaults to zero for all new programs. See *Absolute Tool Length mode, on page 4 - 15* for more information.

    - Touch the top of workpiece with the tip of a calibrated tool and select the Calculate Part Z with Current Tool F6 softkey. The value is automatically entered into the field as the distance from the touch-off gauge block to the workpiece.

    - To add an offset, add or subtract a value from the Part Zero Z value.

    ⚠ If this field is not set before running a part program, tools will plunge to the gauge block height.

- **Offset Z (Zero Calibration Mode)—**defines an offset dimension relative to the tool's Zero Calibration value. This field defaults to 0 so that the tool tip of all calibrated tools will be at the same Z=0 in the program. Offset Z is combined with all of the program's tool calibration points and the Z Top and Z Bottom of the Safety Work Region.

    - To use Store Position softkey or button, calibrated tool previously defined in Tool Setup for this program must be in the spindle.

    - Entering a value in the Offset Z field adds or subtracts the amount of the offset to/from the Tool Calibration value. For example, you can adjust all Tool Calibration values in the Z axis and Z Start to compensate for part thickness by changing the Offset Z value without recalibrating each tool.

    - The Include Offset Z in Tool Zero Cal parameter indicates if Offset Z is added to the Zero Calibration value when tool calibration is set using Store Machine Position. See *General Parameters 2, on page 4 - 40* for more information.

| 1 | Table |
|---|---|
| 2 | Workpiece |
| 3 | Reference gauge block |
| 4 | Offset Z (Zero Calibration)<br>Part Zero Z (Absolute Tool Length) |
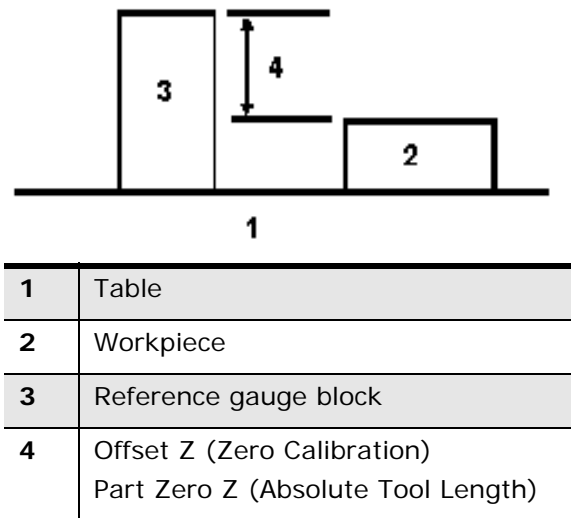
*Figure 4–6. Offset Z or Part Zero Z*

- **Safety Work Region—**defines a part-relative safety area to prevent the cutting tool from colliding with fixtures or other equipment. The Safety Work Region created in Part Setup is saved with a conversational part program; the Safety Work Region is not saved with NC programs.

  Use the Safety Work Region fields on the Part Setup screen to enter values to define the safety region. Type the value in the field, or jog each axis to the desired safety region, and press the Store Position button on the jog unit.

  Out of range entries in the Safety Work Region appear in red. See Machine Specifications for ranges.

  ⚠ Always enter the Z(-) parameter (Z Bottom) to prevent the tool from drilling through the part and into the table.

  Use the Z(+) field (Z Top) to define the clearance above the part and fixture.

  Use the Z(+) field with a Position block. The Z Top value reduces rapid Z motion for Position blocks. The spindle retracts to the Z Top value above the tool zero calibration for Position blocks.

The figure below translates these fields into the Safety Work Region:

| 1 | Part Zero |
|---|-----------|
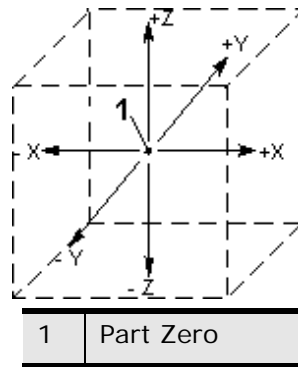
*Figure 4–7. Safety Work Region*

- **Z Table Offset (Zero Calibration Mode)**—specifies the part offset in Z direction relative to the table top.

- **X/Y Skew (Deg)**—represents, in degrees, how far the part is from perfect alignment with the table.

- **Zone (dual-zone machining)**—specifies the zone for dual-zone machines.

# Digital Read Out (DRO)

The DRO is displayed at the top of the Part Setup screen (it also appears at the top of the Tool Setup and Machine Mode screens). The DRO shows the real-time position of the axes and machine status. The current zone is displayed for dual-zone machines. The Machine and Part position information in the DRO will not appear if the machine is not calibrated. These are the DRO fields (fields are dependant on the machine mode displayed) :

- **Zone (dual-zone machines only)—**displays the current zone.

- **Machine and Part Axes**—displays the current position of all axes.

- **Opt Stop On (Off)**—displays the status (on or off) of the Optional Stop for NC programs.

- **Orientation—**the orientation of the axes.

- **Chip Removal**—displays the status of the chip auger.

- **Tool in Spindle**—displays the tool number of the tool in the spindle.

- **Spindle**—displays the current spindle speed.

- **Feed**—displays the current feed rate.

- **Program Run Time**—displays the time the program has been running.

- **F(%)—**displays the current Feedrate per Minute percentage set on the axis Feed Rate knob.

- **R(%)—**displays the current Rapid Traverse Feedrate percentage set on the Rapid override knob.

- **S(%)—**displays the current spindle RPM percentage set on the Spindle Speed override knob.

- **Magazine**—indicates the current pocket.

- **Axes Status**—shows the axes status (calibrated or uncalibrated).

- **Axis Limit Switches**—displays the status of the machine's limit switches as each axis calibrates.

## Part Setup Softkeys

Part Setup contains these softkeys:

- **Work Offsets**—Accesses the work coordinates G54-G59 and a set of shift values for NC programs. These are used to set multiple part zeroes for multiple parts fixtured to the table and milled consecutively using the same program.

- **Tool Setup**—Accesses Tool Setup screen to enter the descriptions of tools that will be used in the part program.

- **Part Programming**—Accesses fields to enter the exact description of how the part will be cut.

- **Program Parameters**—Accesses the Program Parameters screen to specify data common to all program data blocks.

- **Part Probing**—Accesses the Probing parameters screen (available only with the Probing option).

- **Store Machine Position**—Sets the current axis position as a Part Zero location. The cursor location determines which axis (X or Y) will be set.

These are the softkeys on the second Part Setup screen, accessed with the More ➔ softkey:

- **Stock Geometry**—Accesses fields to specify the dimensions of the stock material so it displays properly in graphics.

- **Calculate Rotary Offsets**—Calculates centerline offsets for rotary programs.

- **Orient Spindle**—Allows the spindle to be positioned for tool insertion or removal.

- **Toggle Units**—Changes units from metric to imperial and vice versa.

## Part Fixturing and Tool Loading

Before entering data into the part and tool setup screens for a part program, first fixture the work piece to the table and load a tool into the spindle.

In order to determine the starting point (part zero) in the part, first fixture the work piece (raw material or stock) to the machine tool table. This process, called fixturing, can be accomplished using a variety of clamping devices, such as vises and toe clamps. Select the fixturing device that will hold the work piece securely without getting in the way of the cutting tool while the part is being made.

You may want to insert a tool in the spindle to use as an edge finder when identifying part zero. Press the Manual button in the Machine Mode grouping to display the Manual screen. Use the Tool in Spindle field and softkey to enter the tool number for the edge finder tool.

Insert this tool as described in *Loading a Tool into the Spindle, on page 1 - 24*. The tool will need to be calibrated as described in *Tool Setup, on page 4 - 14*.

# Work Offsets

When in NC mode, the **Work Offsets** softkey displays up to six work coordinates (G54-G59) and a set of shift offset values. These are used to set multiple part zeroes for multiple parts fixtured to the table and milled consecutively using the same program.

The X, Y, Z, and optional Rotary A and B work offset values can be entered for G54 to G59 codes. The coordinates defining G54 are the part zero coordinates for the original part defined on the Part Setup screen. When the G54 coordinates are changed on this screen, the part setup is also changed.

These work offsets are stored in memory, but not with the part program. They are not saved to a disk file and need to be re-entered.

The shift offset coordinates, which follow the six work coordinates on the screen, move all of the part zero coordinates as a group. This incremental value is useful when you place part fixtures on the table in a different location and want to shift all of the work offsets to the newly fixtured location. The G54-G59 offset values do not change on the screen when shift offset values are entered.

For dual-zone machines, specify the zone for each offset:
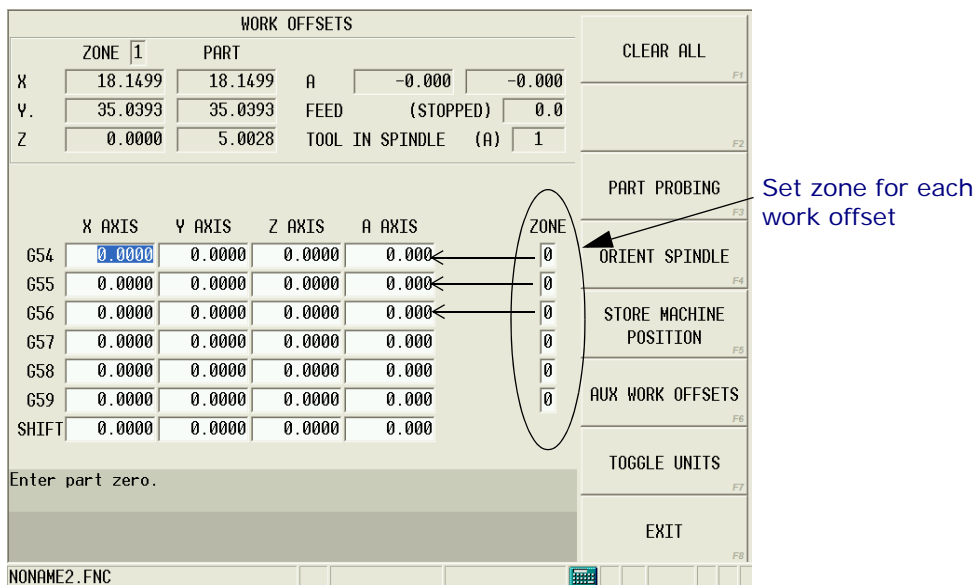


*Figure 4–8. Work Offsets on a dual-zone machine*

Values for the Zone field are:

- 0—zone undefined (the active zone when only one is in use)
- 1—Zone 1
- 2—Zone 2

## Auxiliary Work Offsets

There are 93 additional X, Y, Z, and optional Rotary A and B work offsets available on the AUX WORK OFFSETS screen. These are accessed with the **Aux Work Offsets** *F6* softkey on the WORK OFFSETS screen.

To access any of these offsets call G code **G54.1 Pn**, where **n** is 1 thru 93. For example, to change to auxiliary work offset 46 you would call **G54.1 P46** in your NC program.

To update work offset values, use data setting G code **G10 L20 Pn** to set the Auxiliary work offsets values. For example, to update work offset 46 value call **G10 L20 P46 X12.5 Y3.0 Z-0.5**

Following is a sample NC program that uses Auxiliary Work Offsets:

```
%
G90G80
T1M6F100
G10 L20 P1 X1 Y1 Z.-1
G10 L20 P93 X10 Y10 Z-.1
G54.1 P1 (change work part setup to 1)
G1X0.Y0.Z0.
X1.
Y1.
X0.
Y0.
G54.1 P93 (change work part setup to 93)
G1X0.Y0.Z0.
X1.
Y1.
X0.
Y0.
M02
```

## Stock Geometry

STOCK GEOMETRY *F1* allows you to specify the dimensions of the stock material so it displays properly in graphics (when STOCK OUTLINE is enabled in **Graphics Settings**).

Fields on the Stock Geometry screen are:

- **MANUAL STOCK SIZING**—specify if you will manually set the stock dimensions
  - **NO**—software will automatically size and position the stock. To calculate this, WinMax takes the diameter of the largest tool in the program and adds 7/10ths of that size to the stock diameter X and Y fields (with a minimum addition of .05).
  - **YES**—enter the dimensions manually

- **STOCK TYPE**—specify BOX or CYLINDER
  - **BOX**—enables settings for X length, Y length, Z length of stock box or

cube

- **CYLINDER**—enables direction, length, and radius of cylinder

- **MANUAL BORDER SIZING**—specify if you will manually size the border (available only when MANUAL STOCK SIZING is set to NO)

- **BORDER SIZE**—specify stock border dimension (available only when MANUAL BORDER SIZING is set to YES)

- **ZERO REF**—coordinate system zero from which to reference the stock zero position

- **X REF POSITION**—locate the stock on X axis relative to the ZERO REF; can use Store Position or can be set to Part Zero with softkey

- **Y REF POSITION**—locate the stock on Y axis relative to the ZERO REF; can use Store Position or can be set to Part Zero with softkey

- **Z REF POSITION**—locate the stock on Z axis relative to the ZERO REF; can use Store Position or can be set to Part Zero with softkey

- **BOX LENGTH (X, Y, Z)**—specify lengths when MANUAL STOCK SIZING is YES and STOCK TYPE is BOX

- **CYLINDER—**set dimensions when MANUAL STOCK SIZING is YES and STOCK TYPE is CYLINDER

  - **DIRECTION**—specify POS X, NEG X, POS Y, NEG Y, POS Z, or NEG Z

  - **LENGTH**—specify length along axis from reference position

  - **RADIUS**—specify stock radius

# Tool Setup

Use the Tool Setup screen to describe the tools that will be used for the part program. You can access this screen from either the Input screen or the Part Setup screen by selecting the Tool Setup softkey.

To review all tools currently programmed, go to the [Tool Review](#) screen.

⇨ When running a previously created part program, Tool Setup must be carefully checked to be certain the tools described for the old program match the tools planned for the new program. If a tool breaks or is not available when running a previously created part program, the Tool Setup information must be changed.

See also:

*Tool Setup Fields, on page 4 - 18* and *Tool Setup Softkeys, on page 4 - 22*

## Tool Calibration Modes

Tool measurement and Part Setup Z measurements can be set with one of two modes: Zero Calibration or Absolute Tool Length. The mode is selected in Utilities/User Preferences/Tool Utilities and Settings. See *Tool Utilities and Settings, on page 3 - 6.*

➯ If a program's tools were calibrated using one mode, and that program is loaded into a control that is set to the other mode, a message is displayed that the program's tool calibrations cannot be used. The program data is loaded, but the tool calibration data is not; tool calibration fields are set to zero.

## Zero Calibration Mode

With the Zero Calibration method of tool measurement, tools are calibrated to a plane in the Z-axis as referenced from the Z Home position. Each tool used in a program should be calibrated to the same plane. To use Zero Calibration mode:

1. In the Tool Setup screen, enter the number of the tool to be calibrated into the Tool Number field.

2. If the tool is in spindle, jog Z down to the reference plane or gauge block.

3. Select Store Machine Position softkey or button on the Jog Unit.

   ➯ All tools should be calibrated to the same reference plane.

4. If the reference plane is not the same as Z=Part Zero for program depth:
   a. Select Part Setup softkey.
   b. In Offset Z field, enter distance from reference plane to Part Zero, or use calibrated tool to touch off Z=0 plane and select the Store Machine Position softkey or button on the Remote Jog Unit.

If the plane that all tools are touched off of is the top of the workpiece, all program dimensions for Z will be from Z=0. If the plane is not the top of the workpiece (as when a gauge block is used), the operator can use the Offset Z field in Part Setup to set the distance from the gauge block or other plane to the top of the workpiece plane. See *Part Setup, on page 4 - 3* for more information.

## Absolute Tool Length mode

With the Absolute Tool Length method, the absolute length of the tool from the spindle nose to the tip of the tool is stored in Tool Setup. Each tool used in a program should be calibrated to the same gauge device set on table top. To use Absolute Tool Length Mode:

1. In the Tool Setup screen, enter the number of the tool to be calibrated into the Tool Number field.

2. If the tool is in spindle, jog Z down to the gauge block.

3. Select Store Machine Position softkey or button on the Jog Unit.

4. Part Zero is stored separately in the Part Zero Z field in Part Setup and is the distance from the gauge block to the Z=0 of the workpiece:
   a. Select Part Setup softkey.
   b. In Part Zero Z field, enter distance from gauge device to part zero, or use calibrated tool to touch off Z=0 plane and select the store machine softkey or button on the jog unit.

### *Setup of Gauge Device*

To use Absolute Tool Length mode, you must define the Z location for the table top and

the touch off device(s). This needs to be done only once unless the touch off device changes. Access the Tool Measurement screen with the Tool Measurement Settings softkey on the 2nd set of Tool Setup softkeys (use the More softkey):



*Figure 4–9. Tool Measurement screen*

Tool Measurement fields are:

- **Touch-Off Device**—identifies the device to be used. The control allows up to six devices.

- **Device**—defines the tool setter method: gauge block or tool probe, if equipped.

- **Height**—specifies the height of gauge or probe relative to the Table Top Z.

- **Z Location**— calculates the difference between Table Top Z and the height of the device.

- **Notes**—enter a description of the device specified.

- **Table Top Z**—defines the machine reference position of the table top from machine zero.

To set the Table Top Z position:

1. Advance to Table Top Z field.

2. Place a gauge of known length on table top.

3. Jog Z-axis down and carefully touch spindle nose to top of gauge block.

4. Select Store Z Location softkey or press the store position button on remote jog unit.

5. Subtract the length of the gauge block from the value stored in Table Top Z field.

With Table Top Z field highlighted, you can type in length of the gauge block followed by enter. For example, if Table Top Z measures -562.259, using a 200 length gauge, type "200-" and enter to set Table Top Z to -762.259.

Once Table Top Z is set it does not have to be changed or set again, even if new devices are used. This field and the calculated Z Location fields should not be edited.

To set up a gauge device:

1. In the Touch-Off Device field, select the device number to be used when measuring tool length. Up to six different devices can be set, but only the one currently selected in the Touch-Off Device field is active when setting tool length.

2. Advance to the row number that corresponds to the touch-off device number.

3. In the Device column, select gauge.

4. In the Height column, enter the height of the gauge.

   Z Location field will be automatically calculated by adding the height to the Table Top Z value. Do not edit Z Location column.

5. Multiple gauges can be set up by specifying height in the appropriate row (1-6). However, the active device for measuring tool length is the one specified in the Touch-Off Device field.

   All values set in Tool Measurement Settings are retained in the control and are applicable to all programs. They are not program specific.

To set up a probe device:

1. Follow the steps to set up a single gauge device (for example, set up gauge for device) and set touch-off device to Device 1.

2. In Tool Setup, use any tool to touch-off on the Device 1 gauge.

3. Select Store Machine Position softkey or Store Position button on jog unit.

4. In Tool Measurement Screen, select the touch-off device to be used when probing tool length.

5. In the appropriate row, select Probe as the device from the drop-down Device list.

6. Jog the tool down to the probe until probe is engaged.

7. With cursor in the Z Location column, select Store Position button on jog unit or Store Z Location softkey. This will calculate the height of the probe from Table Top Z.

# Tool Setup Fields

The DRO is displayed at the top of the Tool Setup screen. See *Digital Read Out (DRO), on page 4 - 10* for more information.

The Tool Setup fields are defined as follows (note that not all fields appear on the screen for all tools):

- **Tool Number**—Identifies the number of the programmed tool.

  - For a new program, enter a new number in the Tool Number field and move to another field to display Tool Setup fields.

  - After at least one tool has been programmed, can add another tool:

    - Enter new tool in Tool Number field.

    - Use right arrow to create next available numeric tool number.

  - The range of possible tool numbers is 1 through 9999 and is set in the Tool Changer Specifications function in the Machine Specifications screen in Utilities.

  - The **Next Tool** and **Previous Tool** softkeys become available when more than one tool has been programmed. Use these softkeys as appropriate to access tool setup screens within a program.

    ⇨ Tool numbers do not necessarily correspond exactly to tool pot numbers on the magazine of your ATC.

- **Location—**Identifies the physical location of the tool:

  - Spindle—tool is in spindle

  - Manual—tool is defined

  - Auto—tool is in ATC magazine

  - Pock—displays the tool pocket location of the tool

- **Tool Type—**Identifies the type of tool. Use the appropriate softkey or drop-down list to select a tool type. If you do not see the type of tool that you wish to use, select the **More ➔** softkey to display more Type softkeys. The tool types are:

| Standard Tools | WinMax-Only Tools |
|---|---|
| Drill | Center Drill |
| Tap | Chamfer Mill |
| Boring Head | Bull Nose Mill |
| End Mill | Ream |
| Face Mill | Spot Drill |
| Ball Nose Mill | Forming Tap |
| Back Spotface Drill | Counter Bore |
| Probe | Counter Sink |
| Gun Drill | Keyseat Mill |
| | Thread Mill |
| | Taper Radius End Mill |
| | Corner Round Mill |
| | Dove Tail Mill |
| | Engraving Mill |

- **Diameter—**Identifies the nominal diameter of the tool's cutting surface (i.e., the diameter before the tool suffers from normal wear).

  The diameter range is 0 through 99.9999 inches/2539.997 mm. The Diameter also appears on the Tool Geometry screen.

  ⇨ The programmed diameter is used to automatically determine cutter compensation during milling operations. The system calculates the radius of the cutting tool and automatically allows for this distance when performing milling operations. This means it is not necessary to remember the size of the cutting tool or manually calculate tool offsets when programming the part.

- **Zero Calibration (Zero Calibration Mode)—**identifies Z-axis position of the tool tip when positioned at workpiece or gauge block. A (P) is displayed if the value is set by probing. See *Zero Calibration Mode, on page 4 - 15* for more information.

- **Tool Cal Length (Absolute Tool Length mode)**—identifies the length of the tool from the spindle nose to the tip of the tool. See *Absolute Tool Length mode, on page 4 - 15* for more information.

- **Touch-Off Device (Absolute Tool Length mode)**—displays the touch off device and type (gauge or probe) specified on the Tool Measurement screen. Displayed for reference only; edited on the Tool Measurement screen.

- **Shank Diameter**—Appears when the tool type is Back Spotface and represents the diameter of the tool shank.

- **Thread Diameter**—Appears when the tool type is Tap. Select the Tap icon, located to the right of the field. Choose the appropriate diameter from the pop-up box, as shown below:



Use the slider to select range

Select desired diameter

Touch to select
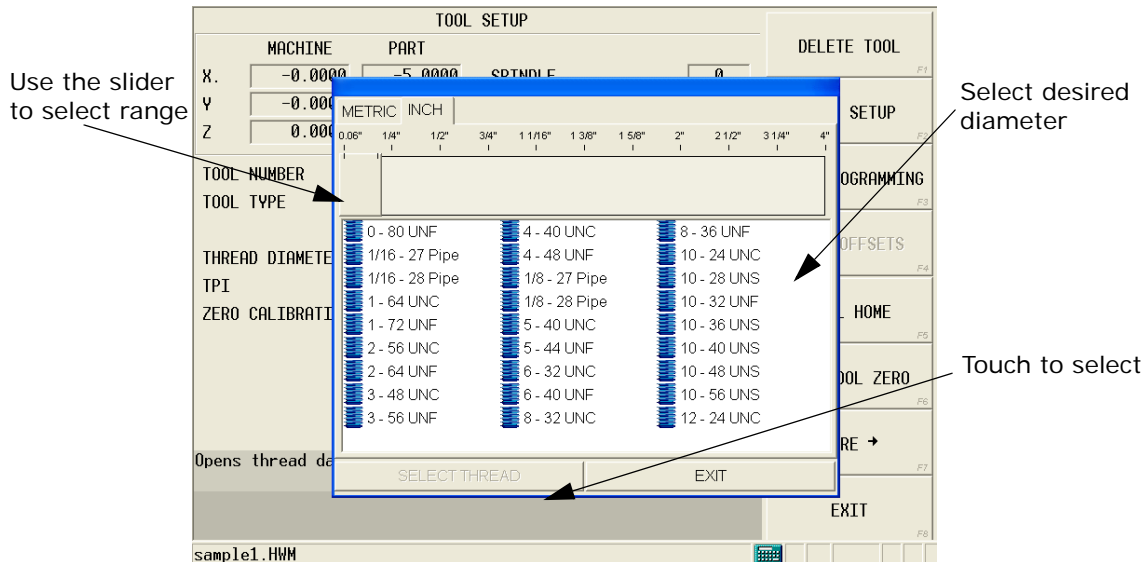
- **TPI (or PITCH)**—Appears when the tool type is Tap. Taps per Inch (TPI) appears when the program's unit of measurement is inch. Pitch appears when millimeter is the selected unit of measurement. The range is 0.0 through 99.99.

  If you enter a value for TPI, WinMax automatically calculates the Pitch. If you enter a value for Pitch, WinMax calculates TPI.

- **Speed (RPM)**—Define the correct speed to be used for this tool, if known. This value will be copied into each new data block in the part program that uses this tool. If necessary, this parameter can be changed within the new data block when programming the part.

  The system uses the Speed (RPM) value to automatically calculate spindle speeds for the tool.

  You can manually change feed and speed values calculated by the control.

  The range is set in *Display Machine Specifications, on page 3 - 2* in Utilities.

- **Coolant**—Define the coolant, if any, to be used for the tool. Coolant is programmable on a tool-by-tool basis. The softkey and drop-down list choices change to **Off**, **Primary**, **Secondary**, or **Both** when the cursor is at the Coolant field. Select **Primary** for machines equipped with a coolant system, **Secondary** for machines equipped with a secondary coolant system (i.e. through spindle coolant), and **Both** for machines with two coolant systems.

- **Surface Speed (FPM)**—Use for milling tools (drill, end mill, face mill, or ball nosed end mill). The Surface Speed, Flutes, and Feed/Flute fields appear. Enter a Surface Speed in feet per minute (or millimeters per minute), and the Flutes and Feed/Flute fields appear. Enter values that indicate the number of

the flutes (teeth) on the tool and the feedrate per flute.

The system uses the values entered in Surface Speed, Flutes, and Feed/Flute to automatically calculate the feeds and speeds for the tool.

You can manually change all feed and speed values calculated by the control.

- **Flutes—**Enter the number of cutting flutes for the tool. This entry will be used to automatically calculate the Mill Feed in all data blocks for this part program using this tool.

  You can manually change all feed and speed values calculated by the control .

- **Feed—**Displays the default Mill Feed or Plunge Feed (depending on the tool type). This value is calculated by the control, based on diameter, feed/flute, and number of flutes for milling tools, or diamter and feed per revolution for hole-making tools. The value can also be entered directly into the field.

- **Feed/Flute (Tooth)—**Enter the tool's chipload. This entry will be used to automatically calculate the Mill Feed in all data blocks for this part program using this tool.

  You can manually change all feed and speed values calculated by the control.

- **Cutting Time**—displays the number of minutes a tool has been running in the spindle (seconds are rounded up to the nearest minute). Starts at zero unless a time is pre-set by the user (if there is already time on the tool).

- **Diameter Wear**—enter a number to compensate for tool wear.

  Diameter Wear = (program tool diameter) - (actual tool diameter)

For example, to adjust a 0.5 inch diameter tool for .001 inch of wear, enter .001 in the Diameter Wear field to set cutter comp diameter to 0.499 inches.

⇨ Compensate for tool wear in this field rather than the Tool Diameter field. This will maintain the tool diameter and ensure accurate tool matching with other part programs that use the same tool.

The Diameter Wear value alters the toolpath for cutter compensation. For example, when milling a circle with Milling Type set to Outside, a positive number in the Diameter Wear field will result in a smaller diameter cut, and a negative number will result in a larger diameter cut.

If a probe is used to determine diameter, the Diameter Wear field will contain the compensated value based on the probed diameter. A (P) appears next to the Diameter Wear field to indicate that the value was derived from the probed diameter.

# Tool Setup Softkeys

The Tool Setup Softkeys are as follows:

- **Delete Tool**—Deletes all program settings for the tool number entered in the Tool Field, for a Manual tool. If the tool is in Spindle or Auto, this softkey is not active.

- **Part Setup**—Accesses Part Setup screen. See *Part Setup, on page 4 - 3* for more information.

- **Part Programming—**Accesses fields to enter the exact description of how the part will be cut. See *Part Programming, on page 1 - 2* for more information.

- **Tool Offsets—**Available only for NC programs. The tool length offsets appear on the screen. Tool offsets are used to compensate tool length without altering the NC program.

    ⇨ Tool offsets are not saved with the NC program.

    - Use the Positive Tool Length Compensation (G43) or Negative Tool Length Compensation (G44) codes. A G49 code specifies tool offset cancel. An H00 also cancels an offset.

    - The G43 and G44 codes set a mode of operation within the program that is in effect until a G49 or H00 is used. If an H code is used without a G43 or G44, in effect, the value stored in the tool length offset table is used as the calibrated tool length.

    - The four keyboard arrows, Page Up, and Page Down keys scroll through the 01 to 200 offsets.

    - When the Store Machine Position softkey is pressed, a negative Z value reflecting the Z axis machine position is entered on the screen.

- **Tool Home**—Allows you to move the tool quickly away from the part. Using this softkey after Tool Calibration is much faster than pressing the axis jog buttons on the jog unit. Press this softkey and then press the Start Cycle button to move the spindle to the change height.

- **Set Tool Zero (Zero Calibration mode)—**stores the Z-axis position of the tool tip when at workpiece or gauge block plane. To use this softkey, carefully jog the tool in the spindle down to the top of the part or to the fixture defined at the Tool Calibration point and then press the softkey. The system stores the position of the tool into the current part program and the number appears in the Zero Calibration field on the Tool Setup screen. This can also be accomplished by pressing the Store Position button on the jog unit. On the screen the part display for "Z" changes to zero. In Zero Calibration mode, all tools used in a part program will need to be calibrated to the same plane.

    ⇨ A warning message is displayed if the edited tool is not the tool in spindle. Select OK to store the current position for the tool that is being edited. Select Cancel if you do not want to set tool zero for that tool.

- **Set Length Using Touch-Off Device (Absolute Tool Length mode)**— stores the tool length when using a gauge or other touch-off device. For a gauge, carefully jog the tool in the spindle down to the top of the gauge and select the softkey. The system stores the tool length in the Tool Cal Length field.

- **Set Tool Zero Using Gauge Block (Zero Calibration mode; Probing only)**—stores the Z-axis position of the tool tip when using a gauge block into the Zero Calibration field and marks the tool as probed so it can be used in Part Probing. To use this softkey, carefully jog the tool in the spindle down to the top of the gauge at the Tool Calibration plane and press the softkey.

- **More ➔**—accesses second set of Tool Setup softkeys.

- **Exit**—exits the tool setup process and allows return to the Input screen.

Second set of Tool Setup softkeys:

- **Advanced Tool Settings**—set Tool Geometry, Feed and Speed, Surface Finish Quality, and other tool information.

- **Change Tool Number**—Allows you to change the tool number for the current tool displayed.

- **Tool Probing**—Accesses the *Probing Parameters, on page 4 - 46*. Probing is available only with BMCs.

- **Program Parameters**—Accesses the program parameters.

- **Part Program Tool Review**—Accesses the tool review screen.

- **Tool Measurement Settings (Absolute Tool Length mode)**—accesses the Tool Measurement screen where touch-off device height is set.

- **More ➔**—Access first set of Tool Setup Softkeys.

- **Exit**—Exits the tool setup process and allows return to the Input screen.

# Advanced Tool Settings

See also:

*Tool Geometry, on page 4 - 24*

*Feed and Speed, on page 4 - 26*

*NC SFQ, on page 4 - 28*

*Supplier, on page 4 - 28*

*Notes, on page 4 - 28*

*Edit Apt Parameters, on page 4 - 29*

Basic tool information is stored in Tool Setup and additional information can be set in the Advanced Tool Settings screens. Tool Geometry, Feed and Speed information, SFQ, and other tool information are set in the Advanced Tool Settings screen.

⇨ Advanced Tool Settings are optional; it is not necessary to adjust these settings in order to run part programs. They are available to simplify programming and increase program efficiency.

## Tool Geometry

Tool dimensions are set in the Geometry tab. Fields denoted with a * are required in order to draw a tool in Solid Graphics; however, these fields are automatically populated with information from Tool Setup, as ratio of entered diameter. A picture of the tool is displayed, and when a field is selected, that area on the picture is denoted.



Tool length field is selected and corresponding area is denoted on picture
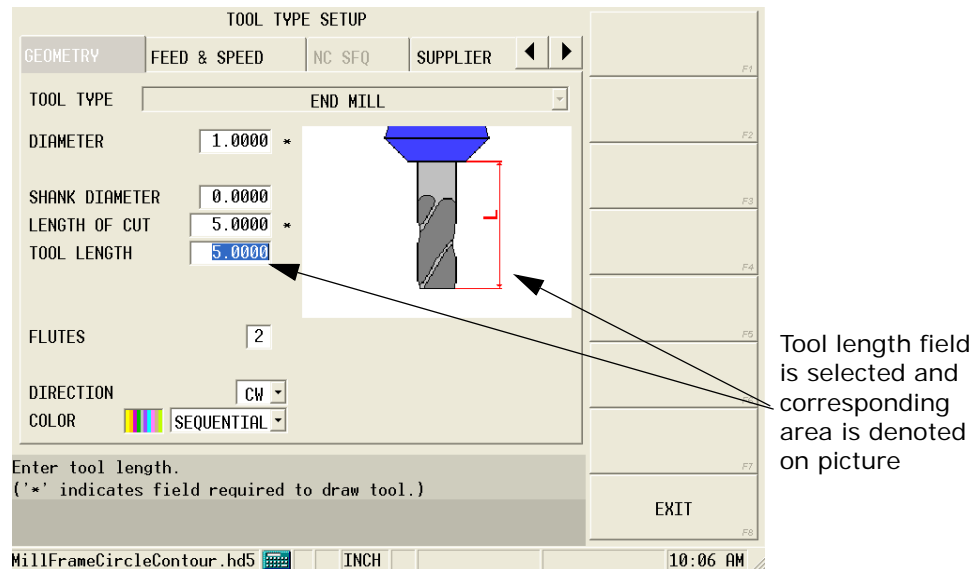
*Figure 4–10. Tool Geometry*

Most fields on the Geometry screen correspond to tool type, see the table on the following page for information about these fields. Additional Geometry fields are:

- **DIRECTION**—spindle direction
- **COLOR**—the color of the path left by the tool in Solid Graphics. The default selection is Sequential, where tools are represented in the following order by color:
    - Yellow—1st tool
    - Orange—2nd tool
    - Violet—3rd tool
    - Green—4th tool
    - Gray—5th tool
    - Blue—6th tool
    - Cyan—7th tool
    - Magenta—8th tool
    - Tan—9th tool
    - Lime—10th tool

    Alternatively, a specific color can be assigned to a specific tool by changing the selection in the field.

The following table provides the tool-type specific fields:

| | Diameter | Shank Diameter | Length of Cut | Tool Length | Flutes | Drill Angle | Thread Diameter | Threads per inch | Radius | Chamfer Angle | Cutting Edges | Stylus length | Drill Angle | Angle | Tip Angle | Tip Length | Tip Diameter | Ream Chamfer | Neck Diameter |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| **Drill** | X | X | X | X | X | X | | | | | | | | | | | | | |
| **Tap (cutting)** | | | X | X | | | X | X | | | | | | | | | | | |
| **Boring head** | X | X | X | X | | | | | X | | | | | | | | | | |
| **End Mill** | X | x | x | x | | | | | | | | | | | | | | | |
| **Face Mill** | x | x | | x | | | | | | X | X | X | | | | | | | |
| **Ball End Mill** | X | X | X | X | X | | | | | | | | | | | | | | |
| **Back Spotface** | X | X | X | X | X | | | | | | | | | | | | | | |
| **Probe** | X | X | | X | | | | | | | | X | | | | | | | |
| **Gun Drill** | X | X | X | X | X | | | | | | | | X | | | | | | |
| **Center Drill** | X | X | | X | X | | | | | | | | | X | X | X | X | | |
| **Chamfer Mill** | X | X | | X | X | | | | | X | | | | | | | X | | |
| **Bull Nose Mill** | X | X | X | X | X | | | | X | | | | | | | | | | |
| **Ream** | X | X | X | X | X | | | | | | | | | | | | | X | |
| **Spot Drill** | X | X | | X | X | | | | | | | | | X | | | | | |
| **Forming Tap** | | X | X | X | | | X | X | | | | | | | | | | | |
| **Counter Bore** | X | X | X | X | X | | | | | | | | | | | X | X | | |
| **Counter Sink** | X | X | | X | X | | | | | | | | | X | | | X | | |
| **Keyseat Mill** | X | X | X | X | X | | | | | | | | | | | | | | X |
| **Thread Mill** | X | X | X | X | | | X | X | | | | | | | | | | | |
| **Taper Radius End Mill** | X | | X | X | X | | | | X | | | | | X | | | X | | |
| **Corner Round Mill** | X | X | X | X | X | | | | X | | | | | | | | X | | |
| **Dove Tail Mill** | X | X | X | X | X | | | | | | | | | X | | | | | X |
| **Engraving Mill** | X | X | | X | | | | | | | | | | X | | | X | | |

⇨ Tip diameter is used to calculate Milling Type offset for Chamfer Mill and Corner Round Mill.

⇨ When Shank Diameter, Length of Cut, and Tool Length are set as a ratio of the diameter, values will be automatically recalculated if the diameter changes until a user-defined value is entered.

## Feed and Speed

⚠ Feed and Speed in Advanced Tool Settings is available only with the Tool and Material Library option.

Feed and Speed settings are carried over from Tool Setup or can be specified in the Feed and Speed tab for specific tool and material combinations in a part program. Choose one of the materials from the work material list for a specific tool and enter the feeds and speeds for both roughing and finishing for that tool and work material combination. This information is saved in the Tool and Material Library and can be utilized in future part programs without re-entering the speeds and feeds: When the Program Parameters for a part program specify a work material, the tool number will recall the feed and speed for that material.

⇨ Regardless of the material set in the Feed and Speed tab, the main Tool Setup screen will always display feeds and speeds for "Unspecified" material.
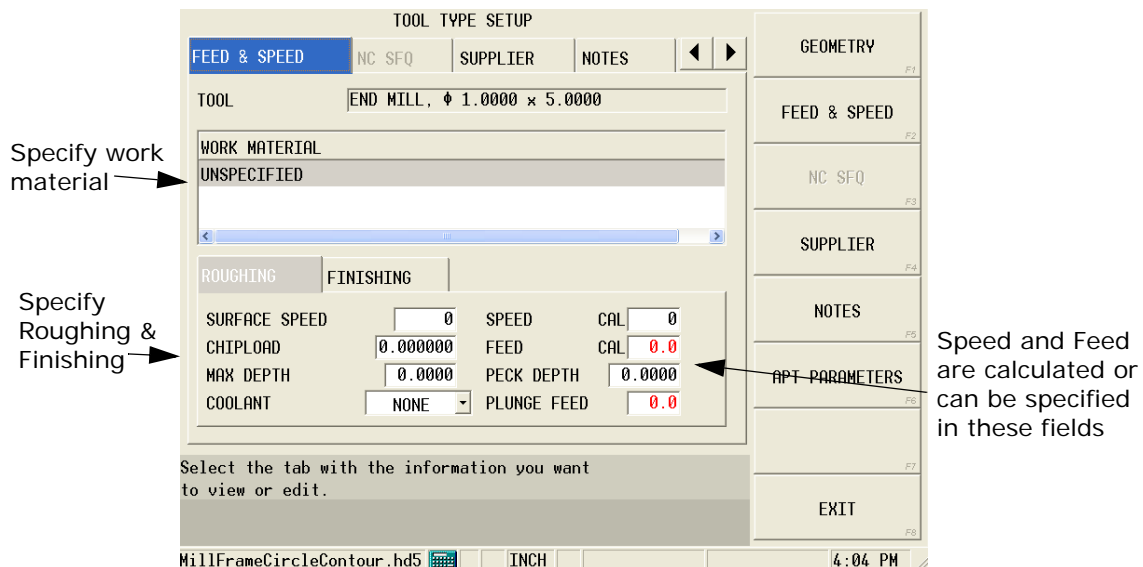


*Figure 4–11. Advanced Tool Settings: Feed and Speed*

Roughing and finishing tool parameters can be specified. Fields are:

- **SURFACE SPEED**—set in Tool Setup, or calculated from speed and diameter
- **CHIPLOAD**—set in Tool Setup, or calculated based on feeds and flutes
- **MAX DEPTH**—set maximum depth of cut (optional)
- **COOLANT**—select coolant type: none, primary, secondary, or both
- **SPEED**—calculated value or may be user-defined
- **FEED**—calculated value or may be user-defined
- **PECK DEPTH**—set peck depth

- **PLUNGE FEED**—set plunge feed, or calculated based on Feed Per Rev for hole-making tools.

⇨ **Feed Per Rev** is used for hole-making tools instead of chipload, independent of the number of flutes.

## Feed and Speed Calculations

The WinMax software automatically performs tool feed and speed calculations for each tool programmed in *Tool Setup, on page 4 - 14*. The calculations are carried over to each data block in the part program using the programmed tool.

The software uses data to perform the calculations from these Tool Setup fields: Diameter, Speed (RPM), Surface Speed (FPM) or (MPM), Flutes, and Feed/Flute (Tooth). Feeds and speeds are related using the following formulas: *Speed Formulas, on page 4 - 27* and *Feed Formulas, on page 4 - 28*.

Calculated values are carried forward to the Mill Feed field and Speed (RPM) field for milling and holes data blocks. You can override automatically calculated values in a data block by inserting a user defined value. If you manually change a calculated value in Tool Setup, you will be prompted to update feeds and speeds for the part program data blocks.

⇨ For fields that display truncated decimal point values, you may view the non-truncated value by pressing and holding the CTRL key and pressing the period (.) key.

## Speed Formulas

The WinMax software uses these formulas to automatically calculate the spindle speeds:

Metric Mode:

Surface Speed x 100

$$\frac{\text{Surface Speed} \times 1000}{\text{Tool Diameter} \times \pi} = \text{RPM}$$

English Units:

$$\frac{\text{Surface Speed} \times 12}{\text{Tool Diameter} \times \pi} = \text{RPM}$$

⚠ If the calculated RPM exceeds the maximum spindle RPM entered in Machine Specifications, the value appears in red font color.

⇨ If you enter the Surface Speed (fpm) value, the Speed (RPM) value will be calculated for you.

If you enter the Speed (RPM) value, the Surface Speed (fpm) value will be calculated for you.

In some cases, calculation rounding may slightly alter a calculated value.

For example, if you enter a value of 7000 RPM for the Speed field in Tool Setup for a Drill operation, the calculated value for Surface Speed is 458 fpm.

However, if you enter a Surface Speed of 458 fpm in the Surface Speed field, the calculated value for Speed is 6997 RPM.

### *Feed Formulas*

For Milling data blocks, specify the number of flutes for the tool in the Flutes field.

The software uses the following formula to calculate the Mill Feed:

$$\text{Feed} = \text{Chipload} \times \text{Flute} \times \text{RPM}$$

For Holes operations, flutes are not specified for the tool.

The software uses this formula to calculate the Mill Feed:

$$\text{Feed} = \text{Chipload} \times \text{RPM}$$

⚠ If the calculated feedrate exceeds the maximum contouring feedrate entered in Machine Specifications, the value appears in red font color.

### NC SFQ

⚠ NC SFQ in Advanced Tool Settings is available only with the NC/Conversational Merge option.

Surface finish quality in NC programming is set in the NC SFQ tab of Advanced Tool Settings. Change Enable Tool SFQ to "YES" and adjust the Tool SFQ. This automatically sets the SFQ value whenever the tool is used in NC programming and will override the SFQ value set in Program Parameters or with the G05.3 setting.

### Supplier

⚠ Supplier in Advanced Tool Settings is available only with the Tool and Material Library option.

The Supplier tab is optionally used to store information about tool supplier and orders in memo-type fields.

### Notes

⚠️ Notes in Advanced Tool Settings is available only with the Tool and Material Library option.

The Notes tab is used to store notes and miscellaneous information about a tool in a memo-type field.

## Edit Apt Parameters

The Edit Apt Parameters fields are defined as follows:

- **Tool—**displays the tool number and type entered on the Tool Setup screen. This field cannot be edited. To change tools, go to *Tool Setup, on page 4 - 14*.

- **NC Gear Range—**define the spindle gear range. 0 = Low; 9 = High.

- **NC Tool Number—**indicate the starting tool in the tool changer. The range is 0 to 999.

- **NC Diameter Offset—**indicate the tool diameter to be used for cutter compensation. This field is used for posting only. This will not affect a file that is saved as an ISNC file.

  ⇨ If Cutter Comp is selected from the Output Tool Path As field on the Post Processor Configuration screen, the value in the NC Diameter Offset field will be put into the APT file. This value will refer to the tool diameter offset on the target machine. If Centerline is selected from the Output Tool Path As field, then the value entered in the NC Diameter Offset field will not affect the APT file.

- **NC Length Offset—**indicate the length offset at the machine tool.

# Change Tool Number

The Change Tool Number softkey allows you to change the number of an existing tool or copy the tool and give it a different number while retaining the tool with its same number.

To change Tool Number or create a copy of a tool, display the Tool Number in Tool Setup and select more to access Change Tool Number softkey.  The Current Tool Description and New Tool Number fields are displayed. Type a number into the New Tool Number field and select one of the following softkeys:

- **Change Tool Number**—changes the number of the tool to the new Tool Number. The old tool number is simply changed to the new number. If the new tool number is already assigned to another tool, then the tool numbers are swapped.

- **Copy Tool**—adds a new tool that is identical to the Current Tool Description.

  If the New Tool Number is not in the current program or Tool & Material Library, the current tool is copied to the New Tool Number. If the New Tool Number is in the current program or Tool & Material Library, a prompt appears that the New Tool Number selected will be replaced with a copy of the current tool.

# Part Program Tool Review

Use the PART PROGRAM TOOL REVIEW softkey on the Program Review screen to review the tools used in a specific part program. This screen lists all tools used in a program, and the program block where they are used. The location column indicates if the tool is in Manual, Auto, or Spindle.

⇨ Part Program Tool Review is available for Conversational Programming only in WinMax Mill versions 7.1.x.

List of tools used in program →

Supplier, Order, and Notes tabs can be used to keep track of other tool information

Program blocks in which highlighted tool is used (highlight block and select the EDIT BLOCK F1 softkey to jump to the program block)

*Figure 5. Part Program Tool Review of a Conversational program*

💡 Jump to a program block by highlighting the block and selecting the EDIT BLOCK *F1* softkey.

Tool information can be entered into the Supplier, Order, and Notes tabs at the bottom of the Part Program Review screen. Information entered on the Notes tab can be viewed in the Select Tool screen (accessed with the Select Tool from List softkey when entering a tool in a program block).

⇨ The Supplier, Order, and Notes tabs are only available with the Tool & Material Library option.

# Part Program Tool Review for NC Programs

Part Program Tool Review in NC programming scans the current NC program for T codes to determine the tools used in the program. T codes are ignored if they are inside a comment (surrounded by parentheses) or if they are an argument in a macro call (preceded by a G65).

Tools are listed by number. When a tool is selected (highlighted), the Block list will display the number and NC text of all blocks that contain the T code for a tool change to the selected tool, as shown below:

List of tools used in program

Program blocks in which highlighted tool is used (highlight block and select the EDIT BLOCK F1 softkey to jump to the program block)



**Figure 6. Part Program Tool Review of an NC program**

Jump to a program block by highlighting the block and selecting the EDIT BLOCK *F1* softkey.

If a tool in the current NC program is not defined in Tool Setup, it appears in the Tool Review list as an unknown tool with a diameter of 0 and an unknown location. This is shown below.
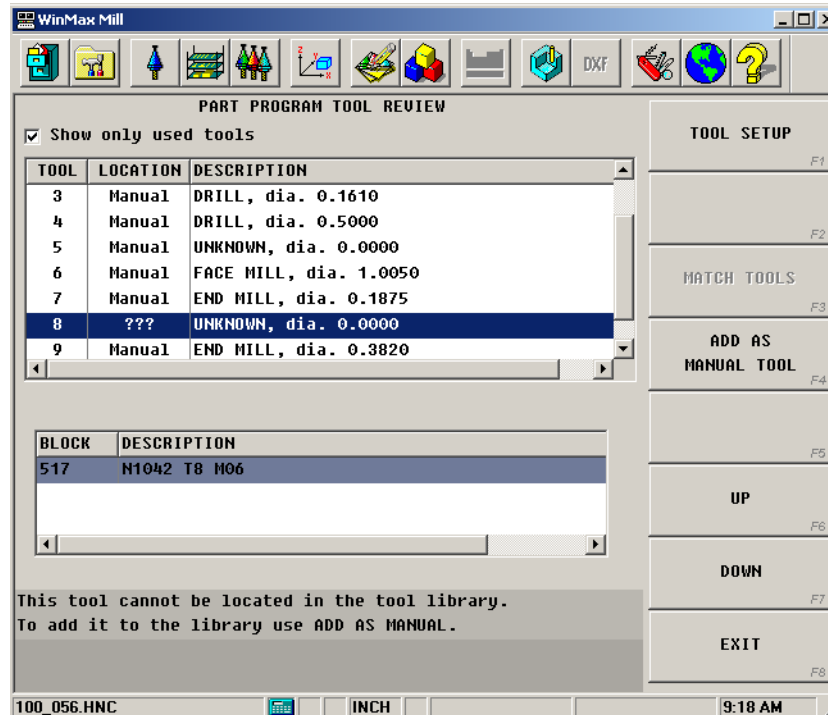
*Figure 7. Tool Review showing Unknown tool in NC program*

An undefined tool can be added to the NC tool setup by selecting the tool in the list, selecting the Tool Setup softkey, and selecting a type or entering data for the new tool. The tool can also be quickly added as an unknown tool with a diameter of 0 by selecting the Add As Manual Tool softkey.

## Part Program Tool Review Softkeys

Softkeys on the Part Program Tool Review screen are:

- **Import Functions *F1*—**access the Part Setup screen to establish part zero, centerline, offset Z, safety work region, and other parameters. See *Import Functions, on page 4 - 48* for more information.

- **Tool Setup *F2*—**access Tool Setup. See *Tool Setup, on page 4 - 14* for more information.

- **Part Programming *F3*—**access and create data blocks of a part program. The current program's data blocks appear on screen when this softkey is used. Delete, add, edit, and navigate through the data blocks. The NC editor is displayed for NC programs. See *Conversational Part Programming* for more information.

- **Program Parameters *F4*—**accesses the Program Parameters screen to specify data common to all program data blocks. See *Program Parameters, on page 4 - 38* for more information.

- **Tool Change Review Screen *F5*—**access the Tool Change Review screen to see all tool changes in the part program. Available with the Tool Change Optimization option. See *Tool Change Review, on page 11 - 3* for more

information.

- **Match Tools** *F6*—access tool matching results. See *Tool Matching, on page 4 - 33* for more information.

- **Add As Manual Tool** *F7*—adds the highlighted unmatched tool as a Manual tool in the Tool & Material Library and assigns it the next available tool number.

## Tool Matching

When a new program is loaded and the Tool & Material Library option is enabled:

1. The new program's tools are compared to the tools in the Tool & Material Library to identify a match by tool type and diameter.

   ⇨ The diameter is set in the Tool Matching: Maximum Diameter Difference field in *Tool Utilities and Settings, on page 3 - 6*.

2. If only one tool match is found, the program will use that tool number, including the Zero Calibration, from the Tool & Material Library. Note that Feed and Speed data will not be replaced in the program.

3. If more than one match exists, WinMax then looks at tool number. If it finds the same tool number, type, and diameter, it will match the tool.

4. If there are no matches, tools are either loaded into the Tool & Material Library as Manual, or users can select from a list of tools in the Tool & Material Library that are similar to the unmatched tool. The AUTOMATICALLY LOAD UNMATCHED TOOLS AS MANUAL field in *Tool Utilities and Settings, on page 3 - 6* controls this:

   - When the field is set to Yes, any unmatched tools from the new part program will be added to the Tool & Material Library and will be assigned to the first available tool number.

     For example, a new program is loaded with tool #12 as a 0.375 inch end mill. If tool #12 in the Tool & Material Library is not a 0.375 inch end mill and no other tool in the Tool & Material Library is a 0.375 inch end mill, then this tool will be added to the Tool & Material Library as a Manual tool and will be assigned the first available tool number.

   - When the field is set to NO, a prompt to match tools appears. Answer YES to the prompt to review the unmatched tools.

Tools that are unmatched are displayed in Tool Matching Results.

### Tool Matching Results

When tools from the part program cannot find a matching tool in the Tool & Material Library, these unmatched tools are listed in the Tool Matching Results screen, accessed with the MATCH TOOLS softkey from the Tool Review screen. Unmatched tools from the current part program are listed in the upper "Tools to be matched" section; as each unmatched tool is highlighted, a list of tools from the Tool & Material Library with the same tool type are listed in the lower, "Closest matches" section. Tools of the same type are listed in order of closest to furthest diameter difference from the highlighted tool.

**Unmatched tools**

**Difference in tool diameter from unmatched tool**

**List of similar tools in Tool Library (list corresponds to the highlighted tool)**
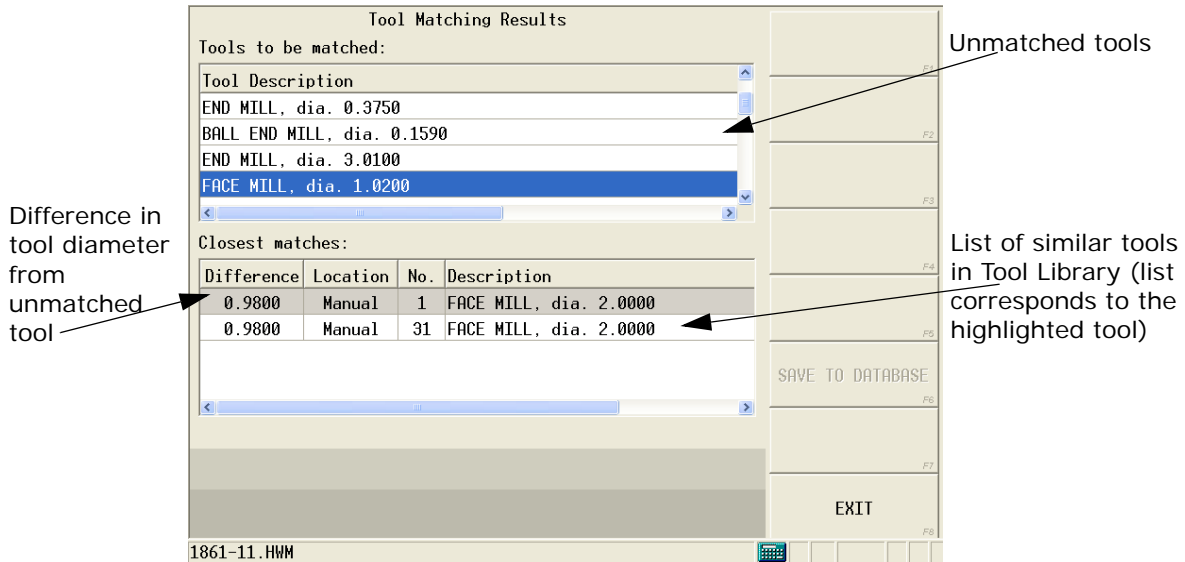
*Figure 4–1. Tool Matching Results*

The CHOOSE AS REPLACEMENT *F4* softkey appears when you select a tool from the Closest matches list. Use this softkey to replace an unmatched tool with the selected tool from the Tool & Material Library.

If there is not a match in "Closest matches," use the EXIT softkey to return to the Part Program Tool Review screen, and use the ADD AS MANUAL TOOL *F4* softkey to add the tool to the Tool & Material Library. Use the TOOL SETUP *F1* softkey to see the tool details.

The SAVE TO DATABASE F6 softkey appears when you select an unmatched tool from the list. Select the softkey to save the tool to the Tool and Material Database.

⇨ Saving a tool to the Tool and Material Database will not match a tool with an existing tool in the Tool & Material Library. The program will still consider the tool unmatched and will not run until a match is selected or a new tool is added to the Tool & Material Library.

# Tool Management

Tool Management screens are accessed in Manual Mode with the TOOL MANAGEMENT *F1* softkey. Tool information can be stored independent of the part program data using the Tool & Material Library option. Program-specific tool information (such as feed rate and speed) created for one program can be reused when tools are selected from the library for a subsequent program. Tool Management tabs are:

- Spindle—shows the tool located in the machine's spindle.

- Auto—lists the tools that are in the machine's tool magazine.

- Manual—lists the tools that may be utilized on the machine but are not currently in the machine's magazine or spindle. The Manual tab is active only with the Tool & Material Library option.

## Spindle

The Spindle tab displays information about the tool in spindle and allows you to set the next tool to execute a tool change, or to change the current tool in spindle.

Fields are:

- **NEXT TOOL**—lists the next tool to call into the spindle

- **TOOL IN SPINDLE**—lists the current tool in the spindle

## Auto

The Auto tab lists the tools in the ATC (automatic tool changer) magazine. Tools are listed by pocket number. The view can be customized to show only occupied pockets or to disable even-numbered pockets (for large tools), using the checkboxes at the top of the screen.

Softkeys on the Tool Library Auto tab are:

- **MOVE TOOL TO SPINDLE *F1***—when spindle is empty, highlighting a tool on the Auto or Manual lists will enable this softkey. Select to confirm and the tool will appear on the SPINDLE tab. The control will prompt to insert the tool into the spindle.

- **SELECT TOOL *F2***—highlight a tool in the list and touch this softkey to select the tool

- **CLEAR POCKET *F3***—removes the selected tool from the pocket

- **CLEAR ALL POCKETS *F4***—removes all tools from the Auto list

# Manual

The Manual tab shows tools available for use that are resident on the control, but are not currently in the spindle or ATC. The Manual tab is active only with the Tool & Material Library option:

Softkeys are:

- **MOVE TOOL TO SPINDLE** *F1*—when spindle is empty, highlighting a tool on the Auto or Manual lists will enable this softkey. Select to confirm and the tool will appear on the SPINDLE tab. The control will prompt to insert the tool into the spindle.

- **INSERT TOOL** *F3*—moves a tool from Manual to Auto

- **Tool Setup** *F4*—accesses the Tool Setup screen

- **CHANGE TOOL NUMBER** *F5*—allows you to change the number of a tool in the list

- **CLEAR TOOLS** *F6*—clears (removes) tools from the list:

  - **CLEAR SELECTED TOOL** *F1*—removes highlighted tool from the Tool Library

  - **CLEAR AUTO AND MANUAL TOOLS** *F2*—removes all tools from the Tool Library

  - **CLEAR AUTO TOOLS** *F3*—removes all tools in Auto from the Tool Library

  - **CLEAR MANUAL TOOLS** *F4*—removes all tools in Manual from the Tool Library

# Tool and Material Database

The Tool and Material Database is available when the Tool & Material Library option is enabled. It is accessed in Auxiliary Mode. Tool and material information can be entered and stored in the database. The materials entered into the database are available in the work material list located on the Feed and Speed tab in Tool Type Setup/Advanced Tool Settings. This information is saved in the material database and can be utilized in future part programs without re-entering the speeds and feeds.

To add a tool to the database:

1. Select the TOOLS tab.

2. Select ADD TOOL *F1 softkey.*

3. Enter the tool information in the fields. See *Advanced Tool Settings, on page 4 - 23* for more information about these fields.

Softkeys on the TOOLS tab are:

- **ADD TOOL *F1*—**accesses Tool Type Setup screen to add a tool to the database

- **EDIT TOOL *F2*—**accesses Tool Setup to edit tool

- **REMOVE TOOL *F3*—**deletes the selected tool from the database

- **RELOAD DATABASE *F4*—**TBD

- **DATABASE SOURCE *F7*—**TBD

To add a material to the database:

1. Select the MATERIALS tab.

2. Select the ADD MATERIAL *F1* softkey.

3. Type the name of the material in the NAME field.

4. Add any notes in the NOTES field.

5. Select the SAVE *F1* softkey.

Softkeys on the MATERIALS tab are:

- **ADD MATERIAL *F1*—**add a new material to the database

- **EDIT MATERIAL *F2*—**change the specifications of a material in the database

- **DELETE MATERIAL *F3*—**delete a material from the database

- **RELOAD DATABASE *F4*—**TBD

- **SELECT MATERIAL FOR PART PROGRAM *F5*—**uses the highlighted material in the current part program

- **DATABASE SOURCE *F7*—**TBD

# Program Parameters

Program parameters are displayed on tabs for General 1, General 2, Milling 1, Milling 2, Holes, Probing, and Performance. The Performance tab in WinMax is active when the SelectSurface Finish Quality option is enabled. The programmer has the option to make changes to any or all of the program parameters and save them as user defaults. The user defaults and original WinMax defaults can be restored by using the appropriate softkey. Parameters can be altered with the Change Parameters data block during program execution. Refer to Conversational Part Programming for details.

Refer to these sections in Program Parameters:

Refer to NC Part Programming for information about NC Parameters.

Softkeys on the Program Parameters screen are:

- **SAVE AS USER DEFAULTS** *F4*—saves the selected field's value as the user-defined default value

- **RESTORE USER DEFAULTS** *F5*—restores the user-defined values to a field that has been populated with other values

- **RESTORE WINMAX DEFAULTS** *F6*—restores the WinMax-defined values to a field that has been populated with other values.

  Change Parameters program blocks load the user-defined parameters set in Program Parameters.

- **NC Parameters** *F7*—accesses NC Configuration Parameters. These parameters are available only with NC Part Programming.

# General Parameters 1

These are the fields on the General 1 tab:

- **Retract Clearance—**Determines the Z coordinate to which the Z axis positions before rapid table positioning. This includes a tool moving from one drilled hole to another, or from one milling operation to another (programmed in separate data blocks or generated as a patterns operation).

    - The default is the maximum programmable Z travel. This is the difference between the Z-Axis MAX Travel and the Z-Axis MIN Travel as indicated on the Machine Specifications screen.

    - The range is 0 through 99.9999 inches (0 through 2514.6 mm).

      If the next operation has a different Z Start value, the CNC always retracts to the highest dimension. When a Position block is programmed, the tool always retracts to The safety plane programmed as Z Top of the Safety Work Region.

- **Rapid Traverse—**Determines the feedrate that the table (X and Y axes) moves between one point in the part program to the next point in the program (rapid table positioning).

    - The default is 400 ipm (10160.0 mm/min).

    - The Range MAX value is user-defined in the Maximum Rapid Traverse Rate field on the Machine Specifications screen. The Range MIN value is 0.1 ipm (2.54 mm/min).

- **Peck Clearance Plane—**Determines the relative distance to the previous peck level. In conversational programming, the tool retracts to Z Start after each peck. The tool then rapids down to a position which is the Peck Clearance distance above the previous peck level before plunging to the next peck level at plunge feedrate.

    Peck Clearance Plane only applies to conversational programs.

    - The default is 0.05 inches (1.27 mm).

    - The range is 0 through 99.9999 inches (0 through 2514.6 mm).

- **Chord Error—**Determines the maximum distance the cutter deviates from the true arc path.

- **Override Lockout—**Disables the Axis Feed dial on the jog unit of Hurco controls when set to On. The default is Off.

# General Parameters 2

These are the fields on the General 2 tab:

- **Depletion Retract—**Specifies the dimension above the part surface to which the Z axis retracts. The Z axis retracts while waiting for additional data to be transmitted into the current program during execution of an NC part program.

    - The default is 0.005 inches (.127 mm).

    - The range is 0 through 99.9999 inches (0 through 2514.6 mm).

        Depletion Retract only applies to NC programs.

- **Interrupt Cycle Z Retract—**Retracts the Z axis to Retract Clearance when you press the Interrupt Cycle console button on a Hurco control.

    - Select No to keep the spindle in its current position when the button is pressed.

    - The default is Yes.

- **First Peck Offset—**Permits modifying the depth of the first peck in milling and hole operations. Use this feature whenever a first peck needs to be deeper or shallower than subsequent pecks.

    The permitted range is –10.0000 to + 10.0000 inches or –254.00 to +254.000 millimeters.

    The First Peck Offset value is added to the operation's peck depth in calculating the first peck only. Use a positive First Peck Offset value for deeper peck and negative value for shallower peck. A First Peck Offset of 0.000 will run the pecks normally, without any First Peck Offset.

    For example, if the peck depth in a drill operation is set to 0.2000 inches and First Peck Offset is set to +0.0500 inches, then the first peck will be 0.2500 inches down from Z start plane and all subsequent pecks will be 0.2000 inches deep. If the First Peck Offset is set to –0.0500 inches, the result is a first peck only 0.1500 inches down from Z start plane and every subsequent peck will be 0.2000 inches deep.

- **Move to Safe Pos During TC—**Indicates whether or not the table will move to the right/front of the machine when the operator is changing a tool. If this field is set to Yes when a part program block calls for a tool change, the table will move out of the way.

- **Include Offset Z in Tool Zero Cal**—Indicates whether or not the Offset Z value in Part Setup is added to the zero calibration value when tool lengths are adjusted. Default is Yes.

    - Yes—Zero Calibration is adjusted by the Offset Z value.

    - No—Zero Calibration is the distance from machine home to the tool tip when positioned at the workpiece or gauge block.

- **Tool Change Optimization**—Indicates if Tool Change Optimization should be enabled for the program. Default is No. (This field is available in Program Parameters only; it is not available in Change Parameters.)

    - No—Any Tool Change Optimization blocks are ignored by the program.

- Yes—Tool Change Optimization blocks are used by the program.

See *Tool Change Optimization, on page 11 - 1* for more information.

# Milling Parameters 1

Milling parameters apply to cutter motions during conversational milling operations only.

Milling 1 Parameters fields are defined as follows:

- **Blend Offset—**Determines the distance from the entry point of the part surface and the Z plunge point where the tool enters the work piece. This field is used in milling circles, frames, and ellipses.

  - The default is 0.1250 inches (3.175 mm).

  - The range is 0 through 1.0 inch (0 through 25.4 mm).

- **Blend Overlap—**Determines the distance the tool travels past the entry point before it is withdrawn from the part. This field is used in milling circles, frames, and ellipses.

  - The default is 0.1250 inches (3.175 mm).

  - The range is 0 through 1.0 inch (0 through 25.4 mm).
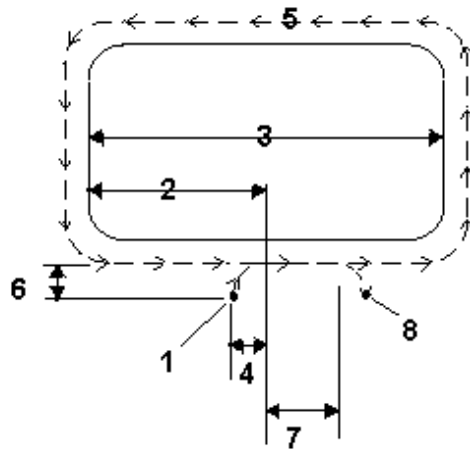


| 1 | Start Point |
|---|---|
| 2 | 1/2 X Length |
| 3 | X Length |
| 4 | Blend Offset |
| 5 | Tool Path |
| 6 | Blend Offset |
| 7 | Blend Overlap |
| 8 | End Point |

*Figure 4–2. Blend Offset and Blend Overlap*

- **Finish Feed (%)**—Allows you to specify a different feed for finishing operations without changing the tool. The specified percentage is a multiplier of the feed entered in Tool Setup. This multiplier is applied whenever the tool is entered for the finishing operation of a milling block. (See example below.)

    ⇨ If this parameter is changed in either Program Parameters or with a Change Parameters block, all existing data blocks that are affected by the multiplier will be updated.

- **Finish Speed (%)**—Allows you to specify a different speed for finishing operations without changing the tool. The specified percentage is a multiplier of the speed entered in Tool Setup. This multiplier is applied whenever the tool is entered for the finishing operation of a milling block. (See example below.)

    ⇨ If this parameter is changed in either Program Parameters or with a Change Parameters block, all existing data blocks that are affected by the multiplier will be updated.

    **Finish Feed / Finish Speed example:**
    Program Parameters: Finish Speed %=120, Finish Feed %=80

    Tool Setup: Speed=5000, Feed=100

    When a program block is created, the speed is automatically set to 5000 and the feed is set to 100. When the tool is entered into the finishing operation, the multipliers are applied, and finish speed is set to 6000 (5000 x 120%), finish feed is set to 80 (100 x 80%).

    ⇨ The Finish Feed and Finish Speed parameters are not applied if the roughing feed or speed in a data block is changed.

    Values entered manually into the Finish Feed or Finish Speed fields in the data block take precedence over these parameters.

    If the Tool & Material Library option is enabled, separate roughing and finishing defaults can be set for each tool. If finishing defaults are defined for a tool, those values take precedence over the Finish Feed and Finish Speed multiplier parameters.

- **Finish XY—**Determines the amount of material in the X-Y axis direction to be removed by the finish pass.

    - The default is 0.2 mm (0.007874 inches).
    - The range is 0 through 25.4 mm (0 through 1.0 inches).

    ⇨ Stock is removed up to a **maximum** of 90% of tool diameter.

    For example, if 25 mm is entered in the Finish XY field, and the tool has a diameter of 10 mm, the XY stock removed on the finish pass will be 9 mm (90% of the 10 mm tool diameter), despite the number entered in the Finish XY field.

- **Finish Z**—Determines the amount of material in the Z axis dimension to be removed by the finish pass.

  - The default is 0.1 mm (0.003937inches).

  - The range is 0 through 1.0 inch (0 through 25.4 mm).

- **Milling Direction**—Determines the milling type. Select Conventional or Climb milling for canned milling cycles (e.g., frame, circle, and ellipse) and for contours (e.g., line, arc).

  - The default is Climb.

  - The choices are Conventional or Climb.

- **Default Pocket Overlap**—Determines the cutter step-over movement in a pocket milling operation. 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 while avoiding islands inside the boundary.

  - The default is 50%.

  - The range is 0 through 99%.

- **Cutter Comp Param**—Determines the programmed tool automatically follows the finished contour of the part with cutter compensation. Without cutter compensation, the center line of the programmed tool follows the print line.

  - **Insert Arc**—Inserts a tangent arc to connect two line segments, or a line segment and an arc segment (when the two cutter compensated segments are offset and do not intersect).



| 1 | Programmed Tool Path |
|---|---|
| 2 | Cutter Compensated Path |
| 3 | Using the Insert Arc Parameter |

*Figure 4–3. Cutter Compensation using the Insert Arc parameter*

  - **Insert Line**—Joins the cutter compensated lines and arcs as described below:

    - Two line segments are extended until they intersect (provided they form a 90° or greater angle). If the lines form an angle of less than 90°, a line is inserted to connect them.

    - Line and arc segments have the line segment extended, and a tangent line to the arc segment inserted and extended until the lines intersect (provided they form a 90° or greater angle). If the segments form an angle of less than 90°, a line is inserted to connect them.

    - Two arc segments have tangent lines (to the arcs) inserted and

extended until the lines intersect (provided the extended tangent lines form a 90° or greater angle). If the extended tangent lines form an angle of less than 90°, a line or arc is inserted to connect them.
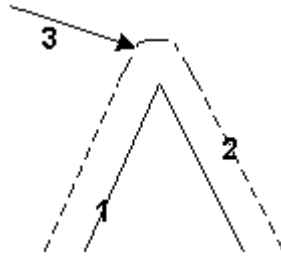


| 1 | Programmed Tool Path |
|---|---|
| 2 | Cutter Compensated Path |
| 3 | Completed Path Using the Insert Line Parameter |

*Figure 4–4. Cutter Compensation using the Insert Line parameter*

# Milling Parameters 2

The Milling 2 Parameters fields are related to the Helical Plunge Option:

- Mill Plunge Type
- Mill Plunge Ramp Slope
- Mill Plunge Helix Radius
- Finish Plunge Type
- Finish Plunge Ramp Slope
- Finish Plunge Helix Radius
- Operator Specify Pocket Start
- Inward Pocket Plunge Near Center

Refer to Helical Plunge in *WinMax Mill Options* for details about these parameters.

# Holes Parameters

The Holes Parameters fields are defined below:

- **Bore Orient Retract—**Determines the distance the boring tool moves away from the part surface at the end of the boring cycle. Used only when a Bore Orient data block is included in the part program.

  - The default is 0.05 mm (0.019685 inches).

  - The range is 0 through 99.9999 inches (0 through 2514.6 mm).

- **Drill Dwell—**Determines the pause (dwell) in seconds before the tool retracts at the bottom of a drill operation. The most often changed Holes Parameter is Drill Dwell. This parameter controls the length of time the drill stays at the bottom of a hole after it has drilled the hole. This parameter is not used for NC programs.

  - The default is 0.5 seconds.

  - The range is 0 through 20 seconds. (Set this to 0.0 seconds, and the drill immediately pulls out of the hole after it is drilled.)

- **Bore Dwell—**Determines the pause in seconds before the tool retracts at the bottom of a Bore operation. This parameter is not used for NC programs.

  - The default is 1.0 seconds.

  - The range is 0 through 20 seconds.

# Probing Parameters

The Probing Parameters are:

- **Automatic Tool Monitoring**—indicate if tools that were calibrated with the probe should be automatically checked with each tool change.

- **Zero Cal (Length) Tolerance**—indicate the zero calibration (tool length) used when checking for a defective tool.

- **Diameter Tolerance**—indicate the diameter tolerance used when checking for a defective tool.

- **Retain Probed Part Setup**—allows the probed part setup and/or tool calibrations to be retained for new program runs. Choices are:

  - **Do Not Retain**—no updates will be made to part zero or tool lengths.

  - **Retain All**—retains part setup and tool calibrations.

    ⇨ Previous versions of WinMax may have included additional selections which have subsequently been removed; older programs that contained a selection other than **Do Not Retain** or **Retain All** will be converted to Retain All upon opening, and a message will appear informing the user.

  For example, the following table shows Part Setup values for a sample program before and after a Probed Part Setup is executed:

| Original Part Setup values | Probed Part Setup values |
| --- | --- |
| Part Zero X = 20.0000 | Part Zero X = 19.1234 |
| Part Zero Y = 10.0000 | Part Zero Y = 11.1111 |
| Probe Z = 15.0000 | Probe Z = 16.5555 |
| Skew Angle = 0.0000 | Skew Angle = 2.0045 |
| Zero Cal = 18.0000 (Tool Setup) | Zero Cal = 17.4444 (Tool Setup) |

  If Retain Probed Part Setup is set to **Do Not Retain**, the original values for Part Setup and Zero Cal will be restored. Part Setup and Tool Setup will contain the original data (20, 10, 15, 0, and 18).

  If Retain Probed Part Setup is set to **Retain All**, Part Setup and Tool Setup will contain the probed values (19.1234, 11.1111, 16.5555, 2.0045, and 17.4444).

    ⇨ OFFSET Z is not affected in any way by the probe block or the parameter setting.

# Performance Parameters

Surface Finish Quality (SFQ) is enabled with the SelectSurface Finish Quality option. SFQ parameters can be modified in either Conversational or NC programming. The default SFQ for roughing is 80 and finishing is 20. Recommended values are:

| SFQ | Desired Result |
|-----|----------------|
| 1-20 | High precision parts /finishing |
| 21-79 | Good surface quality / finishing, semi-finishing |
| 80-100 | High throughput / roughing |

*Table 4–1.    Recommended SFQ values*

⇨ If SelectSurface Finish Quality is not enabled, conversational roughing tools use SFQ of 80 and conversational finishing tools use SFQ of 20; NC default is 50.

Smoothing Tolerance specifies the maximum allowable deviation from the tool path. The range is 0.0000 to 0.0500 inches (0.0000 to 1.2700 mm); default is 0.0005 inches (0.01270 mm). This corresponds to NC code G05.2.

# Import Functions

Import Functions imports Part Setup, Tool Setup, Program Parameters, Part Program information, and/or NC states from an existing Conversational program or NC State file into the active part program, as follows:

1. Select the IMPORT FUNCTIONS *F5* softkey from the Input screen.

2. Select the appropriate softkey for a Conversational or NC program.

3. Choose the file from which components will be imported and select the LOAD *F1* softkey.

4. Choose one or more components from the list and select the BEGIN OPERATION *F2* softkey.

5. Select OK to continue the import operation.

## Conversational Components

These are the Conversational components that can be imported from an existing part program into a new part program:

- **Part Setup**—imports the Part Setup from the selected program into the current part program.

- **Tool Setup**—imports tools from the Tool Setup of the selected part program to the current tool setup on the machine. If the Tool & Material Library option is enabled, unique tools are added as Manual and other tools may be matched (see *Tool Matching, on page 4 - 33*).

- **Program Parameters**—imports the Program Parameters from the selected program into the current part program.

- **Part Program**—imports the part program from the selected program into the current part program.

## NC States

NC components are stored in the NC States file on the control. Only one NC States file exists on a control, and it is updated as changes are made to current or new NC programs. When an NC States file is saved, a corresponding NC Tools file (.nct) is also created with the same name in the same directory. For example, the NC States file Program Name.ncs. will have a corresponding tool file named Program Name.nct. The .nct file must remain in the same directory with the same name as the associated .ncs file. The .nct file can be imported into a program the same way a .ncs file is.

When transferring an NC program from one machine to another the NC States file from the original machine can be imported to another machine using the IMPORT NC STATES softkey. These are the components that can be imported from an NC States file into a new program:

- **Program Parameters**—imports the Program Parameters.

- **Part Setup**—imports the Part Setup.

- **Tool Setup**—imports tools from the Tool Setup of the selected part program to the current tool setup on the machine. If the Tool & Material Library option is enabled, unique tools are added as Manual and other tools may be matched (see *Tool Matching, on page 4 - 33*).

- **Tool Offsets**—imports Tool Offsets.

- **Work Offsets**—imports Work Offsets.

- **NC Parameters**—imports the NC parameters.

- **NC Variables**—imports NC variables.

⚠ The imported components will replace the current components from the open program; they are not merged together.

⇨ Either the .ncs or the .nct file can be selected to import any NC component(s); when either file is selected both are used.

NC States are saved as follows:

1. Select the IMPORT FUNCTIONS *F5* softkey from the Input screen.

2. Select the SAVE NC STATE TO FILE *F6* softkey.

3. Choose the location to which the file will be saved in the SAVE STATE FILE screen.

4. Type the name of the file in the FILE NAME field, or use the default, **ncstate.ncs**, which appears automatically in this field.

   ⇨ It is recommended to rename the NC States file to match the active part program. When renaming, you must retain the .ncs extension at the end of the file name. For example, **Program Name.ncs.**

5. Select the SAVE *F1* softkey to save the NC States file. A message that the file saved successfully will briefly appear on screen.

## Importing NC States into Conversational Programs

NC States information that is valid for Conversational programs (the Part Setup, Tool Setup, and Program Parameters) can be imported into a Conversational program.

1. Make sure the current part program is Conversational.

2. Select the IMPORT FUNCTIONS F5 softkey from the Input screen.

3. Select the IMPORT NC STATE FROM FILE F4 softkey.

4. Choose the .ncs or .nct file from which components will be imported and select the LOAD F1 softkey. Either file may be used to import any component.

5. Choose one or more components from the list and select the BEGIN OPERATION F2 softkey.

6. Select OK to continue the import operation.

# Copy and Change Blocks

To make changes to several data blocks at one time, use the Copy and Change Blocks softkey to access the Block Editor.

These are the softkeys:

- **Copy Blocks—**Duplicate the specified data blocks in another location in the part program.

- **Move Blocks—**Remove the specified data blocks from their current location and transfer them to another location in the part program.

- **Delete Blocks—**Remove the specified data blocks from the part program.

- **Modify Dimensions—**Add offsets to the axes' coordinate dimensions currently stored in the part program for a range of blocks.

- **Change All Feeds, Speeds, and Tools—**Substitutes new feeds and speeds for all tools within the specified range of data blocks.

- **Change Feeds and Speeds by Tool—**Substitutes new speeds and feedrates for a specified tool within a range of data blocks.

    The following fields appear when you select either softkey, except where noted otherwise:

    - **Start Block—**Enter the number of the first block in the program to be changed.

    - **End Block—**Enter the number of the last block in the program to be changed.

    - **Put Block Before—**Move a block in the program before the indicated block.

    - **Change Tool Number—**Identify the tool number. Blocks using this tool number will be changed. (This field appears only for the Change Feeds and Speeds by Tools screen.)

    - **New Speed (RPM)—**Enter a new Speed (RPM).

    - **New Feed—**Enter a new Feed.

    - **New Plunge Feed—**Enter a new Plunge Feed.

    - **New Tool—**Enter a new Tool.

    - **New Finish Tool—**Enter a new Speed (RPM).

Enter the changes in the fields and select the **Make the Change** softkey.

⇨   The **Copy Blocks**, **Move Blocks**, and **Delete Blocks** softkeys are also available for editing a current program.

# Copy, Move, or Delete Blocks

The COPY BLOCKS screen is used to duplicate the specified data blocks in another location in the part program.

The MOVE BLOCKS screen is used to remove the specified data blocks from their current location and transfer them to another location in the part program.

The DELETE BLOCKS screen is used to remove specified data blocks from the part program.

Fields on the screens are:

- **Start Block**—Enter the number of the first block in the program to be changed.
- **End Block**—Enter the number of the last block in the program to be changed.
- **Put Block Before**—Move a block in the program before the indicated block.

# Modify Dimensions

Modify Dimensions allows you to add offsets to the axes' coordinate dimensions currently stored in the part program for a range of blocks.

These softkey choices appear when you select the Modify Dimensions softkey:

- **Start Block** - enter the number of the first block in the program to be modified.
- **End Block** - enter the number of the last block in the program to be modified.
- **X Offset** - enter the offset for the X axis.
- **Y Offset** - enter the offset for the Y axis.
- **Z Offset** - enter the offset for the Z axis.
- **Change Z-Start** - enter the new Z-Start position.
- **A Offset** - enter the offset for the A axis.
- **B Offset** - enter the offset for the B axis.

Enter the modifications in the fields and select the **MAKE THE CHANGE** softkey to modify the programmed dimensions.

# Changing Feeds, Speeds, and Tools

Feeds, Speeds, and Tools can be changed for a range of blocks in a part program with the following softkeys from the Block Editor screen:

- **Change All Feeds Speeds & Tools—**change Feed, Speed, and Tool for specified program blocks. Fields are:

  - **Start Block—**Enter the number of the first block in the program to be changed.

  - **End Block—**Enter the number of the last block in the program to be changed.

  - **New Speed (RPM)—**Enter a new Speed (RPM).

  - **New Feed—**Enter a new Feed.

  - **New Plunge Feed—**Enter a new Plunge Feed.

  - **New Tool—**Enter a new Tool.

  - **New Finish Tool—**Enter a new Speed (RPM).

- **Change Feeds & Speeds By Tool**—Make changes to feeds and speeds for blocks using a specified tool. Fields are:

  - **Start Block—**Enter the number of the first block in the program to be changed.

  - **End Block—**Enter the number of the last block in the program to be changed.

  - **Change Tool**—Enter tool; changes will be made only to blocks using this tool.

  - **New Speed (RPM)—**Enter a new Speed (RPM).

  - **New Feed—**Enter a new Feed.

  - **New Plunge Feed—**Enter a new Plunge Feed.

  - **New Tool—**Enter a new Tool.

# Change Surface Finish Quality

The surface finish quality can be changed for a range of blocks in a part program. Fields are:

- **Start Block—**Enter the number of the first block in the program to be changed.

- **End Block—**Enter the number of the last block in the program to be changed.

- **Change Rough SFQ**—Specify Yes or No.

- **Rough SFQ**—Enter the Rough SFQ for the range of blocks.

- **Change Finish SFQ**—Specify Yes or No.

- **Finish SFQ**—Enter the Finish SFQ for the range of blocks.

# Review Mode

The Program Review screen is accessed with the console Review key. Data blocks and sub blocks of the active part program are displayed in a list, and the tools used in each data block are displayed in the Tool Data list. Program blocks can be moved and added in the Program Review screen.

Sub-blocks within the selected data block

List of program's data blocks

Tools used in the selected data block

(When a tool is highlighted, the EDIT TOOL softkey will access tool setup for that tool)

Select to jump to selected data block or sub-block
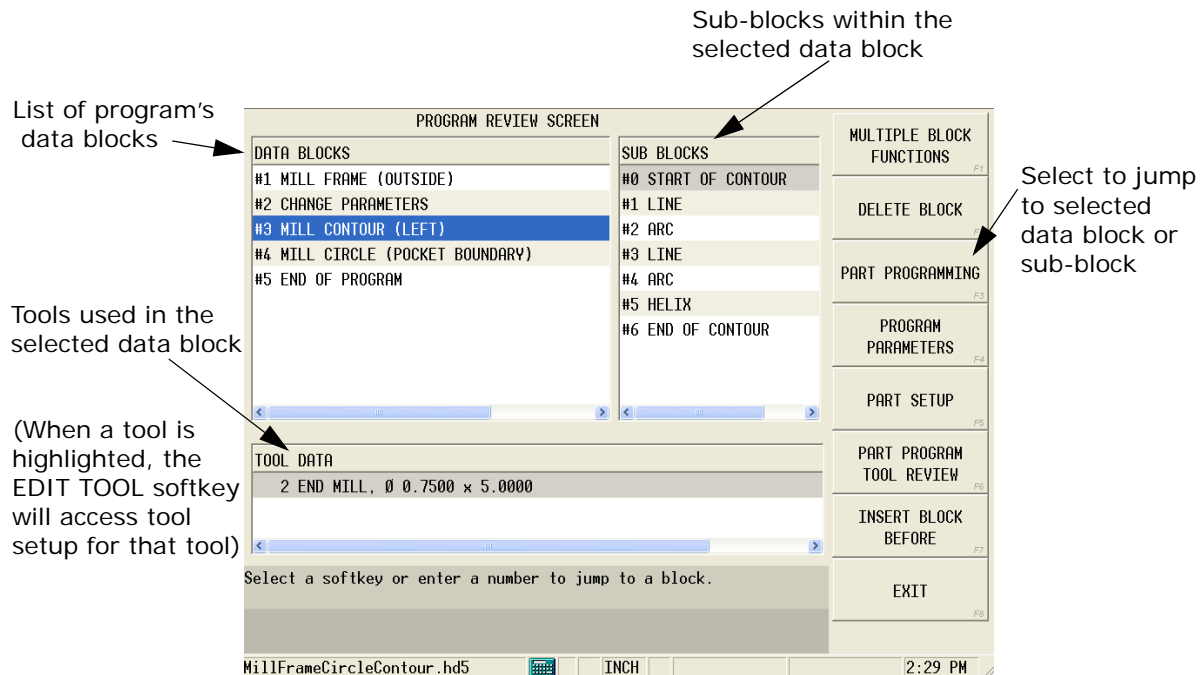


**Figure 4–5. Program Review Screen**

Program Review softkeys are:

- **Multiple Block Functions** *F1*—allows multiple blocks to be cut, copied, or pasted between programs:

  - **Cut** *F1*—highlight data block(s) and touch this softkey to delete from the current program. Block(s) can then be pasted into the same or a different program.

  - **Copy** *F2*—highlight data block(s) and select this softkey to make a copy that can be pasted into the same or a different program.

  - **Paste** *F3*—places previously cut or copied data block(s) above the highlighted data block.

  - **Delete** *F4*—highlight data block(s) and touch this softkey to permanently remove the block from the program.

  Multiple blocks can be selected simultaneously by holding the **F** and **Alt** keys while pressing the up or down arrow keys.

- **Convert to Rotary** *F6*—converts linear dimensions to rotary dimensions; a flat geometry can be wrapped around a cylinder, given a radius. Blocks that can be wrapped are: contour, circle, frame, and TrueType lettering holes (locations).

- **Convert to Linear** *F7*—converts rotary dimensions to linear dimensions.

  ⇨ For WinMax Desktop and WinMax Mill on machines not equipped with rotary, the **Default Conversational Program Type** must be changed to a rotary type in order to convert the program to rotary. This is changed in Conversational Settings in Utilities.

- **Delete Block / Delete Sub Block** *F2*—deletes the highlighted data block or sub-block.

- **Part Programming** *F3*—allows you to edit the selected data block or sub-block.

- **Program Parameters** *F4*—access General, Milling, Holes, Probing, and Performance parameters. NC parameters are accessible for NC programs.

- **Import Functions** *F5*—access the Part Setup screen to establish part zero, centerline, offset Z, safety work region, and other parameters.

- **Part Program Tool Review** *F6*—review the tools used in the part program.

- **Insert Block / Sub Block Before** *F7*—inserts a new program block or sub-block before the selected block or sub-block.

# Auto Mode

Programs are run in Auto Mode. Press the Auto button on the console to access Auto Mode to check for errors, compute estimated run time, recovery/restart, perform a dry run, or run a part program. The following softkeys are available in Auto Mode:

- **Use Editing File *F1***—selects the active program file to run. If this softkey is not selected, WinMax defaults to the last program run. If the last program run does not match the program that is being edited (as indicated in status bar), the operator will be prompted to select which program to run.

- **Feed & Speed Optimization *F2***—fine tunes program execution, using the Axis Feed Rate and Spindle Speed dials to adjust values. This softkey is disabled in Test Mode.

- **Check for Errors *F4***—checks the program from the Start Block through the End Block and displays error status. The number of the data block containing the error is included in the error message.

  The time required for error checking depends upon the program's length and complexity. Select the Abort Operation softkey to stop error checking at any time.

- **Compute Estimated Run Time *F5***—a pop-up window displays an estimate of time it takes to run the program. Pressing any console key will remove the pop-up window. Select the Abort Operation softkey to stop computing estimated run time.

  Error checking automatically occurs during Compute Estimated Run Time.

- **Recovery Restart *F6***—restarts a conversational or NC part program; typically the point at which the program was interrupted. This softkey is disabled in Test Mode.

  - For Conversational Programs—if necessary, Conversational Start and End blocks can be changed from the default.

    For Mill Contour data blocks: Recovery Restart only occurs at segment 0 of a Mill Contour data block, not at a segment within a data block.

    For Pattern data blocks: when Recovery Restart begins at the first block of a pattern, all instances of the pattern are run. If Recovery Restart begins at a block within the pattern, WinMax prompts the user to specify which instance to restart at, and then requests any additional restart information for the start block.

  - For NC Programs—use the following softkeys to restart an NC program after it was aborted either by the machine or the operator. Preparatory functions such as coolant, feeds and speeds, and offsets will be executed before machining resumes, and Tool Vector input is checked.

    - **Set Restart Marker**—manually sets the restart marker. An 'r' will appear to the left of the block selected, and will clear if the program successfully runs.

    - **Auto Set Restart Marker**—automatically marks the last block being executed when the program was stopped, or where an error occurred during error checking.

- **Reset Restart Marker**—clears the Restart Marker and cancels the Recovery Restart operation. Can be used on a block after a G41/G42.

- **Dry Run *F7***—active in Test Run mode only, performs a program test run to identify potential problems before cutting the part. Specify all or a portion of the part program that will be tested in the Start and End blocks.

  Trace the tool over the part at the programmed minimum Z level with the Spindle Off. Peck cycles and roughing passes are skipped.

  ⚠️ If the Z-Start value is set below the stock surface, the minimum Z value must be programmed so the tool does not plunge into the part.

- **Run Program *F8***—initiates program execution and displays monitoring information. If the machine is not calibrated, the Manual screen immediately displays. This softkey is disabled in Test Mode.

# Auto Mode Monitoring

The Auto Mode Monitoring screen appears when the RUN PROGRAM softkey is selected and program execution begins. The upper part of the screen displays the DRO. See *Digital Read Out (DRO), on page 4 - 10* for more information. The lower area of the screen displays the Spindle Load Monitor, the data block executed, the type of operation, and the part count. For NC programs, five lines of NC code are shown; to view more NC code as the program runs, select the NC MONITOR *F6* softkey. The bottom portion of the screen is reserved for program status and error messages.

These are the fields on the Auto Mode Monitoring screen:

- **Zone (dual-zone machines only)—**displays the current zone.

- **Machine and Part Axes**—displays the current position of all axes.

- **Opt Stop On (Off)**—displays the status (on or off) of the Optional Stop for NC programs.

- **Chip Removal**—displays the status of the chip auger.

- **Tool in Spindle**—displays the tool number of the tool in the spindle.

- **Spindle**—displays the current spindle speed.

- **Feed**—displays the current feed rate.

- **Program Run Time**—displays the time the program has been running.

- **F(%)—**displays the current Feedrate per Minute percentage set on the axis Feed Rate knob.

- **R(%)—**displays the current Rapid Traverse Feedrate percentage set on the Rapid override knob.

- **S(%)—**displays the current spindle RPM percentage set on the Spindle Speed override knob.

- **Axis Limit Switches**—displays the status of the machine's limit switches as each axis calibrates.

- **Spindle Load Monitor**—displays the percentage of full load on each axis as the program runs. The load is displayed in a bar graph format, with colors progressing from green to yellow to red to indicate the percentage of load

from 0 to 150%.

- **Block**—displays the current program block.
- **Part Count**—displays the number of times the program was executed.



Spindle Load Monitor

5 lines of NC code
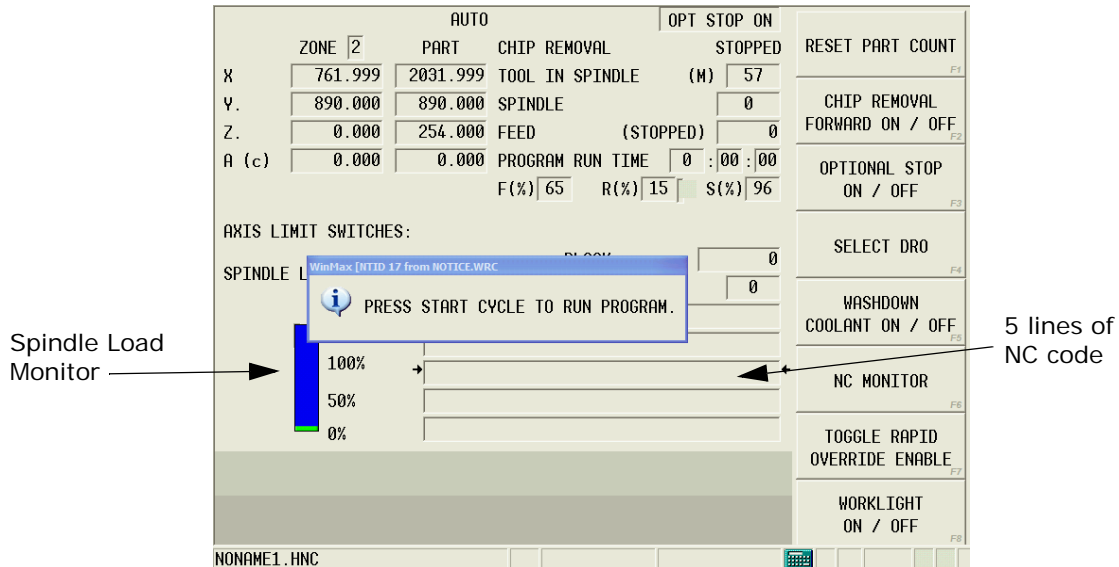
*Figure 4–6. Auto Mode Monitoring screen*

These are the monitoring softkeys that may be available for Auto Cycle mode:

- **Reset Part Count** *F1*—the number of times a program was executed. To return this value to zero when a new program starts, touch the Reset Part Count softkey.

- **Chip Removal Forward On/Off** *F2*— turn the chip auger in the forward (clockwise) direction on or off. This selection is saved when the Interrupt button is pressed or after the program has finished running. If a stop condition or mode change is made before restarting a new part program or exiting the interrupt cycle, the saved information is cleared. This softkey is only available if your machining center uses a chip auger.

- **Optional Stop On/Off** *F3*—pauses the program and shuts off the spindle. This softkey functions only with NC programs. When Optional Stop is On, the M01 code in the NC program will be processed; when Optional Stop is Off, the M01 code is ignored.

- **Select DRO** *F4*—change the size of the digital read out (DRO) on the screen. From the Select DRO screen you can see machine information displayed in either Full Status or Full DRO. Full Status displays the current location of machine and part axes, as well as other machine information. Full DRO displays the current location of machine and part axes, Distance to Go and other machine information (with abbreviated categories). The current zone is also displayed (dual-zone machines only).

- **Coolant Washdown On/Off** *F5*—turn the coolant washdown on or off for washing chips from the enclosure.

- **NC Monitor** *F6*—displays a pop-up window containing the current NC code. This feature allows you to view the code as the program runs; the program

name appears in the Current Program Name box at the bottom of the monitor. The line of code that is currently being machined is identified in red. When the "Show Modals" boxed is checked, the current active modals are shown. As machining progresses, the monitor will scroll through the lines of code. Select the "Close" button at the bottom of the window to close the NC Monitor.

- **Toggle Rapid Override Enable** *F7*—enable or disable the ability to override the programmed rapid traverse using the Rapid Override console knob.

- **Worklight On/Off** *F8*—turn the enclosure worklight on or off. This softkey is only available if your machining center is equipped with an enclosure worklight.

# Manual Mode

Manual mode controls machine settings and operations. The top part of the screen displays the DRO. See *Digital Read Out (DRO), on page 4 - 10* for more information.

These are the softkeys for Manual mode:

- **Tool Management** *F1*—Access tool information for Spindle, Auto Tools, and Manual Tools.

- **Manual Function Setup** *F2*— Set manual spindle speed and jog feed, retract tool, execute a manual rapid move, turn washdown coolant on and off, and turn worklight on and off from this screen.

- **Diagnostics** *F3*— Access the CE Status & Diagnostics screen and the ATC & Machine Diagnostics screen. See *Manual Safety Override Mode, on page 4 - 79* and *ATC and Machine Diagnostics, on page 4 - 65* for more information.

- **Park Machine** *F4*—Center the table and leave the spindle at the home position.

- **Warm Up Machine** *F5*—Warm up an idle machine.

- **Orient Spindle** *F6*—Ensures that the Z axis is at the correct height for a tool to be inserted in the spindle.

    ⇨ The ORIENT SPINDLE softkey will not function unless the enclosure doors are completely closed.

- **Reset Servos and Spindle** *F7*—Activates only to enable recovery from certain types of electronic hardware faults such as overloads.

- **Calibrate Machine** *F8*—Establish absolute zero for each axis on the machining center.

## Manual Function Setup

The Digital Read Out is displayed at the top of the Manual Function screen. See *Digital Read Out (DRO), on page 4 - 10*. Other fields on this screen are:

- **Manual Jog Feed**—enter the jog feedrate.

- **Manual Spindle Speed**—enter the spindle speed.

Softkeys are:

- **Retract Tool**—retracts the tool.
- **Manual Rapid Move**—accesses the manual rapid move parameters.
- **Washdown Coolant On/Off**—turns the washdown coolant on and off.
- **Worklight On/Off**—turns the worklight on and off.

## Manual Rapid Move

Manual Rapid Move executes a single axis rapid move to a specific position at a controlled feedrate. The operator can enter an axis position, either in machine or part coordinates, and initiate the move to that position at a specified feedrate.

The DRO is displayed at the top of the screen; see *Digital Read Out (DRO), on page 4 - 10.* The Manual Rapid Move fields are:

- **Coordinate System**—specifies the coordinate system for the end position of the move, either Part or Machine.
- **Manual Rapid Feed**—specifies the feedrate of a linear axis move.
- **Rotary Rapid Feed**—specifies the feedrate of a rotary axis move.
- **X / Y / Z**—specifies the end position of the move in a linear axis.
- **A / B / C**—specifies the end position of the move in a rotary axis, if equipped (fields are inactive for a non-rotary machine).

  ⇨ More than one axis end position may be entered, but the move occurs only on the axis that is selected with the corresponding Rapid Move softkey.

Softkeys are:

- **Rapid Move X**—enables the rapid move in X.
- **Rapid Move Y**—enables the rapid move in Y.
- **Rapid Move Z**—enables the rapid move in Z.
- **Rapid Move A**—enables the rapid move in A (softkey is inactive if the machine is not equipped with a rotary/tilt A axis).
- **Rapid Move B**—enables the rapid move in B (softkey is inactive if the machine is not equipped with a rotary/tilt B axis).
- **Rapid Move C**—enables the rapid move in C (softkey is inactive if the machine is not equipped with a rotary C axis).

To perform a Manual Rapid Move:

1. In Manual mode, select the Manual Function Setup softkey.
2. Select the Manual Rapid Move softkey. The Manual Rapid Move fields are displayed. (This softkey is grayed out if the machine is not calibrated.)
3. Select Part or Machine coordinate system from the drop-down Coordinate System list.
4. Set the linear axis feedrate in the Manual Rapid Feed field. Alternatively, set

the rotary axis feedrate in the Rotary Rapid Feed field.

5. Specify the end position of the move in the appropriate axis field(s).

6. Select the appropriate Rapid Move softkey to enable the move. When the softkey is selected, a message is displayed that instructs the operator to press the Start Cycle button to move the axis. When the Start Cycle button is pressed, the rapid move is performed.

⇨ Only one axis move can be performed at a time. During the move, all other Rapid Move softkeys are disabled until the move is completed. Once complete, another softkey can be selected, initiating the next move.

⇨ The Rapid Override knob/potentiometer is active during the move.

# Automatic Tool Changer

The ATC for both HTX500 and VTXU machines is a random-pocket tool changer that tracks tools in the magazine using an ATC Map. All ATC operations require that the servo power is on, that the machine is calibrated, and that the ATC is at Home position.

Each VMX or VM machining center is equipped with a swing-arm random pocket Automatic Tool Changer (ATC).

## VM Tool Magazine

The station magazine is positioned vertically on the machine. An electronic motor and helical gear drive the magazine. When activated, the tool magazine operates as described below:

1. The requested tool pocket rotates 90° to make the tool available for the swing arm.

2. The swing arm rotates 60° to simultaneously grab the tool in the pocket and the tool in the spindle.

3. The spindle unclamps.

4. The swing arm moves down to pull the tools out of the pocket and the spindle, and then rotates 180°.

5. The swing arm moves up and swaps the tools.

6. Finally, the swing arm and tool pocket return to their original positions.

## Types of Automatic Tool Changers (ATC)

There are two types of ATCs, which are described in the following sections:

- 40-Taper ATC with 24- or 40-Station Tool Magazine
- 50-Taper ATC with 30- or 32-Station Tool Magazine

### 40-Taper ATC with 24- or 40-Station Tool Magazine

This tool changer features a magazine with either 24 or 40 tool pockets (stations). The magazine is positioned vertically on the machine. An electric motor and helical gear drive the 24- and 40-station magazine. The 24-station magazine is round and the 40-station magazine is elliptical. Refer to the previous table for the number of tool pockets (stations) available for a specific machining center.

### ATC Sequence

This is the basic sequence of operation of the tool magazine, assuming the magazine is positioned to the next tool required:

1. The requested tool pocket rotates down 90° to make the tool available for the swing arm.

2. The swing arm rotates 60° to simultaneously grab both the tool in the pocket and the tool in the spindle.

3. The spindle unclamps.

4. The swing arm moves down to pull the tools out of the pocket and spindle, and then the arm rotates 180°.

5. The swing arm moves up and swaps the tools.

6. The spindle clamps the new tool in the spindle and the swing arm rotates back to 0°.

7. The tool pocket moves back to its original position.
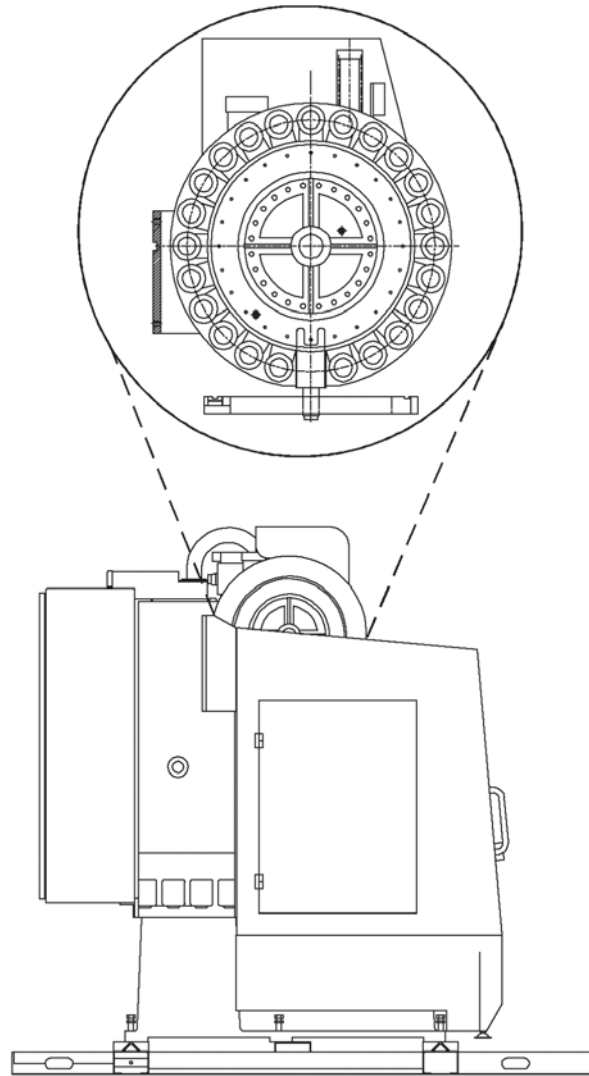


*Figure 4–7. VMX 40-Taper ATC with 24-station Tool Magazine*

## 50-Taper ATC with 30 or 32-Station Tool Magazine

The VMX50/50T uses the 50-Taper ATC with a 30-station tool magazine. The VMX64 uses the 50-Taper ATC with a 32-station tool magazine.

An electric motor, two dogs (cam detents), and three proximity switches control the 50-Taper tool changer motion. For the VMX64, a free-standing hydraulic unit runs the rotation of the magazine, magazine locking cylinder, and the tool pot up and down cylinder. The ATC sequence for the 50-taper ATC is the same as the 40-taper ATC sequence.
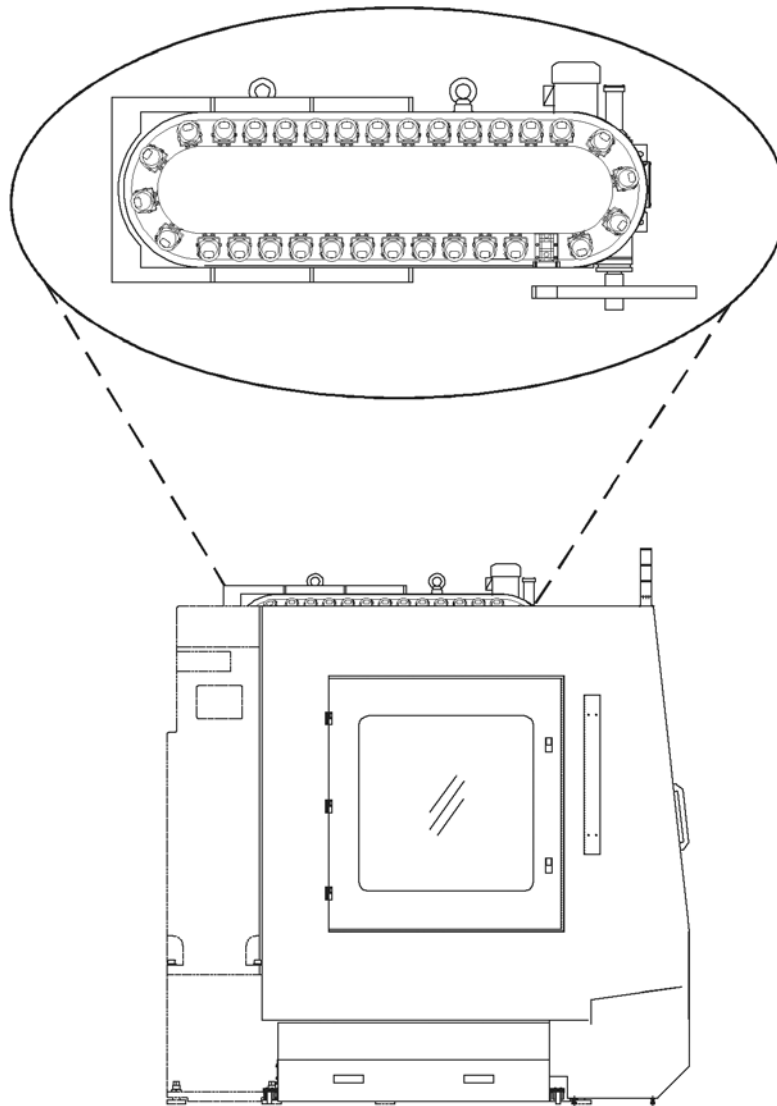


*Figure 4–8. VMX 64/50-Taper, 32 Station ATC*

# HURCO

# HTX500 and VTXU Tool Magazine

The ATC can be operated in Auto or Manual machine mode, using ATC Diagnostics.

## HTX500 Machine

For the HTX500 machine, the following steps explain the basic sequence of operation of the tool magazine, assuming the magazine is positioned to the next tool required.

1. The tool holder rotates to 90°, then the ATC door opens.

2. The exchange arm rotates to 90° and simultaneously grabs the tool in the tool holder, and if present, the tool in the spindle, if the following conditions are met:

   • The X, Y, and Z axes are at tool change position.

   • The spindle is oriented.

3. The spindle unclamps.

4. The exchange arm moves out and simultaneously pulls the tools from the holder and the spindle, then rotates 180°.

5. The exchange arm moves in and swaps the tools.

6. The exchange arm rotates to 90°.

7. The ATC door closes and the tool holder simultaneously returns to its original position.

## VTXU Machine

The following steps describe the basic sequence of operation of the tool magazine in the VTXU magazine, assuming the magazine is positioned to the next tool required.

1. The ATC door opens if the following conditions are met:

   • The X and Y axes are at tool change position.

   • The Z axis is at Height (refer to *ATC and Machine Diagnostics, on page 4 - 65* for more information).

   • The spindle is oriented.

2. The tool pocket rotates from the Up position to the Down position.

3. The swing arm rotates 90° and simultaneously grabs the tool in the tool pocket and, if present, the tool in the spindle.

4. The spindle unclamps.

5. The swing arm moves up and swaps the tools, then rotates 180°.

6. The swing arm and tool pocket return to the original positions.

7. The ATC door closes.

# ATC and Machine Diagnostics

The HTX500 machine has separate screens for Machine Diagnostics and ATC Diagnostics. For all other machines, the ATC and Machine Diagnostics are combined on the ATC & Machine Diagnostics screen.

## HTX500 Machine and ATC Diagnostics

Follow these steps to display the Machine Diagnostic screen for the HTX500 machine:

1. Press the MACHINE MODE MANUAL console key.

2. Select the **DIAGNOSTIC** *F3* softkey.

3. Select the **MACHINE DIAGNOSTICS** *F6* softkey.

```
                *** MACHINE DIAGNOSTICS ***

   SPINDLE STATUS      NOT ORIENTED        CLAMPED                         F1
   TOOL IN SPINDLE     2


   ATC STATUS            HOME             CALIBRATED                        F2


   AIR PRESSURE        PRESSURE OK                                         F3
   WAY LUBE STATUS     LEVEL OK
   COOLANT STATUS      LEVEL OK          FILTER OK                         F4
   HYDRAULIC STATUS    NO ALARM
   SPINDLE CHILLER     NO ALARM                                            F5
   OVERLOAD STATUS     NOT TRIPPED

                                                                           F6


                                                                           F7

                                                           EXIT
                                                                           F8
```
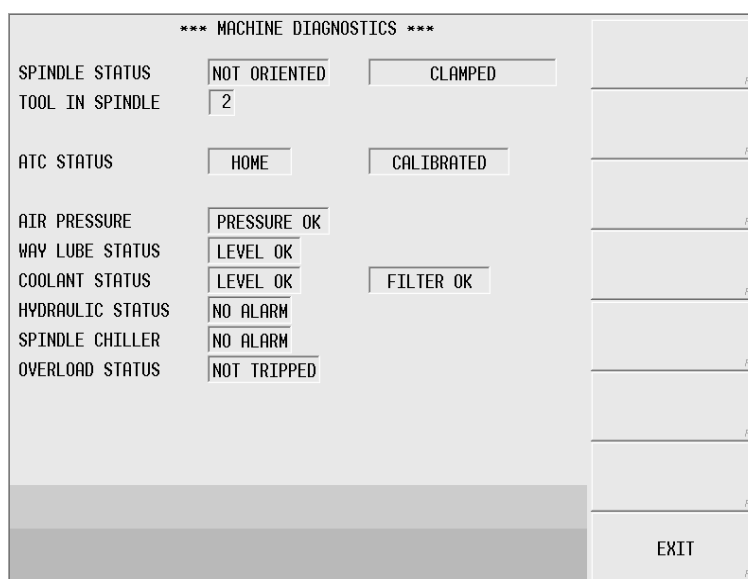
*Figure 4–9. Machine Diagnostics Screen for HTX500 Machine*

These are the status fields on the Machine Diagnostics screen for an HTX500 machine:

- **Spindle Status**—indicates if the Spindle is Oriented or Not Oriented, and clamped or unclamped. Oriented is required for ATC to begin.

- **Tool in Spindle**—indicates the tool number in the Spindle.

- **ATC Status**—indicates if ATC is in Home position. Home is required for ATC to begin. Home is defined as ATC Door closed, Exchange Arm at $0^\circ$, Tool Holder at $0^\circ$, and Magazine In Pos (1).

- **Air Pressure**—indicates Pressure OK or Low Pressure. Pressure OK is required to begin an ATC operation.

- **Way Lube Level**—indicates Level OK or Low Level. A Low Level alarm will not allow programs to start running.

- **Coolant Status**—indicates Level OK and Filter OK. Both are required for tool coolant operation. Available with the Coolant Through Spindle (CTS) option.

- **Hydraulic Status**—indicates No Alarm or Alarm. An alarm will not allow programs to start running.

- **Spindle Chiller**—indicates No Alarm or Alarm. No Alarm is required to begin an ATC operation.

- **Overload Status**—indicates Tripped or Not Tripped. Not Tripped is required to operate the machine.

Display the ATC Diagnostics screen for the HTX500 machine by following these steps:

1. Press the MACHINE MODE MANUAL console key.

2. Select the **DIAGNOSTIC** *F3* softkey.

3. Select the **ATC DIAGNOSTICS** *F7* softkey.

```
                    *** ATC DIAGNOSTICS ***
                                                    ORIENT SPINDLE
   SPINDLE STATUS      NOT ORIENTED      CLAMPED                    F1
   TOOL IN SPINDLE      2
   ATC AXIS POSITIONS  NOT IN POSITION                ATC DOOR
                                                    OPEN / CLOSE
   Z-AXIS POSITION      AT ZERO                                     F2
   MAGAZINE POSITION    14                           TOOL HOLDER
   ATC STATUS        □  HOME          CALIBRATED       0° / 90°
                                                                    F3
   TOOL HOLDER       □ RETRACTED (0D)
   EXCHANGE ARM      □  0°                           EXCHANGE ARM
                                                      0° / 90°
   ATC DOOR          □  CLOSED                                      F4
   MAGAZINE STATUS   □  IN POS #1                      SPINDLE
                                                   CLAMP / UNCLAMP
   MAGAZINE PIN      □  LOCKED                                      F5

                                                   EXCHANGE ARM
   HYDRAULIC STATUS    NO ALARM                         180°
                                                                    F6

                                                      MORE➜
   Select the appropriate softkey.                                 F7

                                                       EXIT
                                                                    F8
```

*Figure 4–10. HTX500 ATC Diagnostics Screen, Status Condition Fields*

## HTX500 ATC Status Condition

These are the status condition fields that appear on the ATC DIAGNOSTICS screen. Green indicators next to fields identify when the status is in the home position.

- **Spindle Status—**indicates if the Spindle is Oriented or Not Oriented, and clamped or unclamped. Oriented is required for ATC to begin.

- **Tool in Spindle**—indicates the tool number in the Spindle.

- **ATC Axis Positions**—indicates At X, At Y, and At Z positions. All three are required for ATC to begin.

- **Z-Axis Position**—indicates At Zero position.

- **Magazine Position**—indicates the current magazine position (note that this may not be the tool number).

- **ATC Status**—indicates if ATC is in Home position. Home is required for ATC to begin. Home is defined as ATC Door closed, Exchange Arm at 0°, Tool Holder at 0°, and Magazine In Pos (1).

- **Tool Holder**—indicates 0° or 90° position. 0° is required for ATC Status Home.

- **Exchange Arm**—indicates 0° or 90° position. 0° is required for ATC Status Home.

- **ATC Door**—indicates Closed or Open position. Closed is required for ATC Status Home.

- **Magazine Status**—indicates the current magazine position (note that this may not be the tool number)

- **Magazine Pin**—indicates Locked or Unlocked Position. Locked is required for ATC Status Home.

- **Hydraulic Status**—indicates No Alarm or Alarm. No Alarm is required to begin an ATC operation.

## HTX500 ATC Diagnostics Softkeys

There are three pages of diagnostic softkeys for controlling the ATC operation for HTX500 machines. Pressing the *-MORE-* softkey changes to either of the other two pages.

When all requirements for a softkey action are met, a prompt displays and the Start Cycle lamp flashes. Pressing the Start Cycle button executes the command. The Start Cycle lamp stops flashing during the execution of the command and is turned off when the command is completed.

Selecting another softkey before pressing the Start Cycle button cancels the pending command and causes the Start Cycle lamp to cease flashing. If the requirements for a softkey action are not met, a message displays indicating why the action cannot be commanded.

- **Orient Spindle** *F1*—sets the command to orient the spindle. Oriented is required to begin an ATC cycle.

- **ATC Door Open/Close** *F2*—sets the command to open or close the ATC Door. Each press of the softkey switches command between open and close. Must be Closed for ATC to begin.

- **Tool Holder 0°/90°** *F3*—sets the command to move the Tool Holder to 0° or 90°. Each press of the softkey switches command between 0° and 90°.

- **Exchange Arm 0° / 90°** *F4*—sets the command to move the Exchange Arm to 0° or 90°. Tool Holder must be at 90°, and ATC Door Open. Once rotated to 90°, the Exchange Arm must be rotated to 180° or jogged back to 0°.

- **Spindle Clamp/Unclamp** *F5*—sets the command to clamp or unclamp the spindle. Each press of the softkey switches the command between clamp and unclamp.

- **Exchange Arm 180°** *F6*—sets the command to move the Exchange Arm out, rotate 180° and in. Tool Holder must be at 90° and ATC Door Open. Must be at 0° for ATC to begin.

- **Move Axes to TC Position** *F1*—sets the command to move all required axes to the ATC position.

- **Move Z-Axis to Zero** *F2*—sets the command to move the Z-axis to the Zero position.

- **Magazine CW** *F3*—sets the command to move the magazine clockwise one position.

- **Magazine CCW** *F4*—sets the command to move the magazine counter-clockwise one position.

- **Calibrate T/C Magazine** *F6*—sets the command to calibrate the magazine. The magazine will rotate to the Reference Pos.

- **IO Maps** *F1*—displays the softkey selections for I/O boards in the control. See below for more information.

- **Exchange Arm Jog Reversal** *F2*—sets the command to pulse jog the Exchange Arm in the reverse direction. Once initiated continue selecting until the Exchange Arm is at 0° or 90°.

## HTX 500 I/O Maps

Selecting the I/O MAPS F1 softkey on the ATC Diagnostics screen displays softkey selections for I/O boards in the control. Each of the softkeys identifies I/O boards in the control. Select one of the softkeys to display a list of inputs and outputs on the board and their active or enabled state. The boards are abbreviated as follows:

- IOIF - Input/Output Interface Board

- DCI/01 - DC Input Output #1 Board

- DCI/02 - DC Input Output #2 Board

- DCI/03 - DC Input Output #3 Board

The Inputs and Outputs are listed by board connector, either J1 or J2. The outputs cannot be activated nor can the inputs be tested.

Each displayed I/O map includes both the input and output signal identifier list. The input and output signals are highlighted when the signal is activated or enabled. Refer to the machine *Parts Listing and Wiring Diagram* manual for schematics and harnesses that identify the signals and boards.

## VM, VMX, and VTXU ATC & Machine Diagnostics

Display the ATC & Machine Diagnostics screen for these machines by following these steps:

1. Press the MACHINE MODE MANUAL console key.

2. Select the **DIAGNOSTIC** *F3* softkey.

3. Select the **ATC & MACHINE DIAGNOSTICS** *F7* softkey.
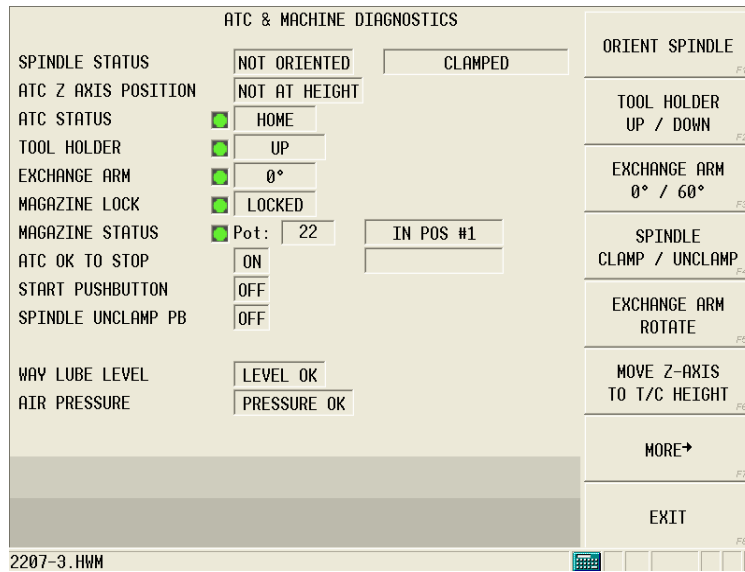


*Figure 4–11. ATC & Machine Diagnostics Screen, Page 1*

## ATC & Machine Diagnostics Input Fields

The following descriptions are for the status of inputs for the ATC for VM, VMX, and VTXU machines.

- **Spindle Status**—indicates if the Spindle is Oriented or Not Oriented. Oriented is required for ATC to begin.

- **Spindle Clamp**—indicates if the Spindle is clamped or unclamped.

- **ATC Z Axis Position**—indicates Z is at Machine Tool Change Height.

- **ATC Position** (VTXU only)—indicates Z At Height and X/Y At Position.

- **ATC Status**—indicates if ATC is in Home position. Home is required for ATC to begin. Home is defined as ATC Door closed (VTXU), Exchange Arm at 0°, Tool Holder at UP, and Magazine In Pos #1.

- **ATC Door Status** (VTXU only)—Indicates the status of the ATC door on the VTXU machine.

- **Tool Holder**—indicates Up or Down position. Up is required for ATC Status Home.

- **Exchange Arm**—indicates 0° or 60° position. 0° is required for ATC Status Home.

  - Cannot move from 0° to 60° unless Z-axis is at Tool Change Height, Tool Pocket is Down, Spindle is Oriented and Clamped.

  - Cannot move from 60° to 0° unless Z-axis is at Tool Change Height, Tool Pocket is Down, and Spindle is Clamped.

- **Magazine Lock**—indicates Locked or Unlocked Position. Locked is required for ATC to begin.

- **Magazine Status**—indicates the current magazine position (note that this may not be the tool number).

- **Magazine In Position**—indicates In Pos (1) or In Pos (2).

  - In Pos (1) is required for ATC Status Home.

  - In Pos (2) is active when magazine is rotating and not in position.

- **Magazine Reference**—indicates Reference Pos when the ATC calibration sensor is detected.

- **ATC OK to Stop**—indicates Off or On. On is set during Load Arm rotation and disables the Load Arm motor and enables the brake. Off should be set when Load Arm is at 0 degrees or 60 degrees.

- **Start Pushbutton**—indicates Off or On. On is displayed when the Start Cycle button is pressed.

- **Spindle Unclamp PB**—indicates Off or On. On is displayed when the Spindle Unclamp button is pressed.

- **Way Lube Level**—indicates OK or Low Level. A Low Level alarm will not allow programs to start running.

- **Air Pressure**—indicates OK or Low Pressure. OK is required to begin an ATC operation.

## VM, VMX, and VTXU ATC & Machine Diagnostics Softkeys

When all requirements for a softkey action are met, a prompt displays and the Start Cycle lamp flashes. Pressing the Start Cycle button executes the command. The Start Cycle lamp stops flashing during the execution of the command and is turned off when the command is completed.

Selecting another softkey before pressing the Start Cycle button cancels the pending command and causes the Start Cycle lamp to cease flashing. If the requirements for a softkey action are not met, a message displays indicating why the action cannot be commanded.

These are the ATC & Machine Diagnostics softkeys. Some softkeys may not be present on all machines or may appear in a different order:

- **Orient Spindle**—sets the command to orient the spindle. Oriented is required to begin an ATC cycle.

- **Tool Holder Up / Down**—sets the command to move the Tool Holder Down or Up. Each press of the softkey switches command between Down and Up.

- **Exchange Arm 0° / 60° (90° for VTXU)**—sets the command to move the Load Arm to 0° or 60° (90°). Tool Pocket must be Z-axis at Tool Change Height, Spindle Clamped, and Oriented Down before moving to 60° (90°). Each press of the softkey switches command between 0° and 60° (90°).

- **Spindle Clamp / Unclamp**—sets the command to clamp or unclamp the spindle. Each press of the softkey switches the command between clamp and unclamp.

- **Exchange Arm Rotate**—sets the command to move the Exchange Arm down, rotate 180° and up. Tool Pocket must be Down, Load Arm must be at 60°, and spindle must be unclamped.

- **Move Z-Axis to T/C Height**—sets the command to move the Z-axis to the Tool Change Height position. Z-axis must be At Height for ATC to begin.

- **Move Z-Axis to Zero**—sets the command to move the Z-axis to the Zero position.

- **Magazine Pin Lock / Unlock**—sets the command to lock or unlock the magazine pin. Locked is required for ATC to begin.

- **Magazine CW**—sets the command to move the magazine clockwise one position.

- **Magazine CCW**—sets the command to move the magazine counter-clockwise one position.

- **Calibrate T/C Magazine**—sets the command to calibrate the magazine. Magazine will rotate to the Reference Pos.

- **Exchange Arm Jog Reverse**—sets the command to pulse jog the exchange arm in the reverse direction. Pulse commands will stop once the Exchange Arm is at 0 degree.

- **ATC Door Open / Close** (VTXU only)—sets the command to open or close the ATC Door on the VTXU machine. Each press of the softkey switches command between open and close. Must be Closed for ATC to begin.

- **Move Axes to T/C Position**—sets the command to move all required axes to the ATC position.

# HTX Automatic Pallet Changer

The Automatic Pallet Changer (APC) system is standard on HTX500 machining centers. Equipped with two pallets, the APC provides the ability to set up a job on one pallet while a job is being machined on the other pallet.

⇨ The APC table must be squared before any operation can occur. Refer to *Squaring the APC Table, on page 4 - 72* for instructions for squaring the APC table.

This is the basic sequence of operation of the APC:

1. The APC Pin unlocks and the APC Clamp unclamps.

2. The APC table raises and rotates 180°. Hydraulic solenoids ensure the table rotates in the proper direction.
   ⇨ The APC table does not rotate through 360°, but rotates in the opposite direction to return to its previous position.

3. The APC table lowers into position.

4. The APC Clamp clamps and the APC Pin locks.
   ⇨ Only a Hurco-certified Service Engineer can make adjustments to the APC Table.

## Squaring the APC Table

The APC table is not automatically squared during machine calibration. Therefore, the APC table must be squared before it can be used. Follow these steps to square the APC table:

1. Press the MACHINE MODE MANUAL console key.

2. Select the **DIAGNOSTICS** *F3* softkey.

3. Select the **PALLET CHANGER DIAGNOSTICS** *F5* softkey to access the APC DIAGNOSTICS screen.

4. Select the **MOVE Z-AXIS TO ZERO** *F1* softkey to position the Z-axis at Zero.

5. Select the **SQUARE PALLET FOR ROTATION** *F2* softkey and press the Cycle Start button. The APC table will rotate into the proper position for the pallet exchange cycle.
   ⇨ If the B-axis is away from the squared position, selecting the **SQUARE PALLET FOR ROTATION** *F2* softkey rotates the table at the maximum RPM back to the squared location.

## Operating the APC

The APC can be operated in Auto mode within a Conversational or NC part program.

Follow these steps to operate the APC in a Conversational program:

1. On any Conversational part programming data block, select the **MISCELLANEOUS** *F5* softkey.

2. Select the **MACHINE FUNCTION** *F5* softkey to display the Machine Function screen.

3. Select the **SHOW M CODE LIST** pop-up box to display the M Code List. The following M Codes are used to operate the APC (use the scroll bar to view more M codes, if necessary):

   • **M51 Cycle Pallet Changer**—requires confirmation from the Operator before an pallet change cycle is initiated (see details below).

   • **M56 Cycle APC W/O Confirm**—complete a pallet change cycle without confirmation from the Operator.

   • **M57 APC Pos 1 W/O Confirm**—place Pallet 1 in the machine without confirmation from the Operator.

   • **M58 APC Pos 2 W/O Confirm**—place Pallet 2 in the machine without confirmation from the Operator.
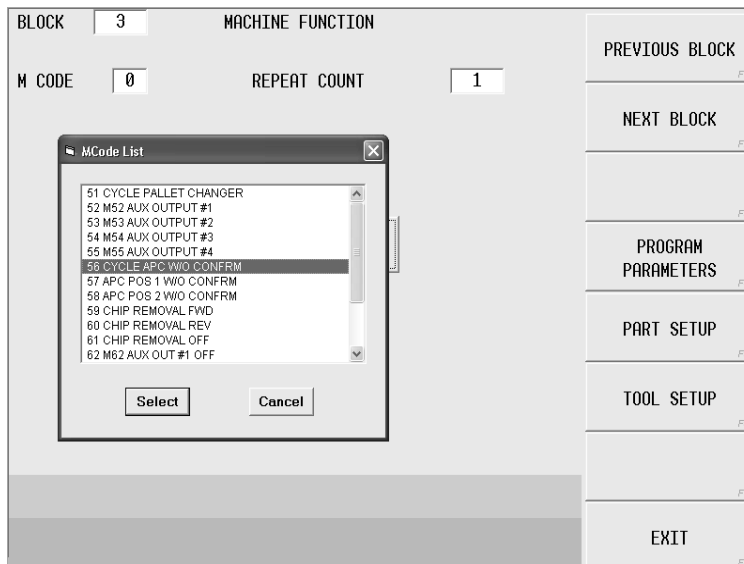


*Figure 4–12. Machine Function Screen with M Code List*

Using M51 does not require indicating whether Pallet 1 or 2 is in the machining setup area. Before M51 will initiate the APC cycle, the front enclosure doors must be closed and the Workzone Confirmation pushbutton pressed.

   • If the front enclosure doors are open when the M51 operation is requested, the program will be in a Feed Hold state and a message to close the front enclosure doors will be displayed. Close the doors and press the Workzone Confirmation pushbutton to continue the part program.

- If the Workzone Confirmation pushbutton has not been pressed when the M51 operation is requested, the program will be in a Feed Hold state with the following message displayed, "Pallet setup confirmation required to cycle pallet changer." Press the Workzone Confirmation and Cycle Start pushbuttons on the remote APC Panel to continue the program.

⇨ Pressing Feed Hold during a pallet change cycle will stop the cycle. Releasing Feed Hold will resume the cycle. However, Feed Hold followed by Spindle Off (or the Stop Cycle button) during a pallet change cycle will abort the program and the pallet change cycle. The APC must be returned to the home position using the Pallet Changer Diagnostics before an Auto program can be run again.

Refer to the *NC Programming Manual* for instructions on operating the APC within an NC program.

## APC Diagnostics

Follow these steps to access the APC Diagnostics screen:

1. Press the MACHINE MODE MANUAL console key.
2. Select the **DIAGNOSTICS** *F3* softkey.
3. Select the **PALLET CHANGER DIAGNOSTICS** *F5* softkey.

```
                *** APC DIAGNOSTICS ***            MOVE Z-AXIS
                                                    TO ZERO
    APC POSITION          DOWN                              F1
    PALLET IN MACHINE     1
    PALLET PIN            LOCKED                   SQUARE PALLET
    APC DECEL SWITCH      1                        FOR ROTATION
    APC REFERENCE         REFERENCE POS.                     F2

                                                  CYCLE PALLET
    HYDRAULIC STATUS      NO ALARM                   CHANGER
    Z-AXIS POSITION       AT ZERO                            F3
    ATC STATUS            HOME
    PALLET STATUS         PALLET NOT SQUARED       RAISE PALLET
                                                    CHANGER
                                                           F4

                                                  ROTATE APC TO
                                                   POSITION 1
                                                           F5

                                                  ROTATE APC TO
                                                   POSITION 2
                                                           F6

                                                  LOWER PALLET
                                                    CHANGER
    Select the appropriate softkey.                        F7

                                                     EXIT
                                                           F8
```
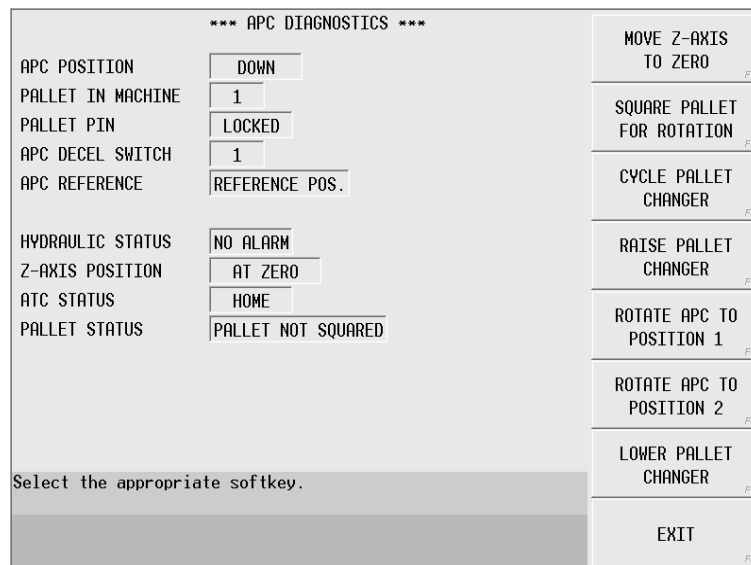
*Figure 4–13. APC Diagnostics Screen*

## APC Status Conditions

The APC Diagnostics screen displays the status of inputs for the APC system.

- **APC Position**—indicates whether APC table is in the Up or Down position.
- **Pallet in Machine**—indicates whether the APC table Position 1 or 2 is in the machining area.

- **APC Pin**—indicates whether the APC Pin is Locked or Unlocked. The APC Pin must Unlock before the APC table can be lifted and rotated and Locked to complete the cycle.

- **APC Decel Sw**—indicates whether the Position 1 or 2 switch was seen during rotation. This switch enables low-speed rotation before the APC table completes the 180° rotation.

- **APC Reference**—indicates whether the Manual Table Release Lever is in the Reference Pos or has been retracted. The Manual Table Release Lever must be at Reference Pos to begin an APC cycle.

- **Hydraulic Status**—indicates either OK or Alarm. Must be OK to begin an APC cycle.

- **Z-Axis Position**—indicates if the Z-axis is At Zero (calibration point). Must be At Zero to begin an APC cycle.

- **ATC Status**—indicates if the ATC table is Home. Must be Home to begin APC cycle.

- **Pallet Status**—indicates Pallet Squared. Must be Pallet Squared to begin an APC cycle.

## APC Diagnostics Softkeys

The APC Diagnostic softkeys are functional only when Servo Power is On and the machine is calibrated. These are the APC Diagnostic softkeys.

- **Move Z Axis to Zero Position** *F1*—sets the command to move the Z-axis to the zero position.

- **Square Pallet for Rotation** *F2*—sets the command to square the pallet.

- **Cycle Pallet Changer** *F3*—sets the command to complete a full rotation of the APC Table (all eight steps of the basic sequence of operation).

- **Raise Pallet Changer** *F4*—sets the command to lift the APC Table to the Up position. The APC table must be squared and the APC table must not be Up.

- **Rotate APC to Position 1** *F5*—sets the command to rotate the APC table to place Position 1 inside the machining area. The APC table must be squared and in the raised position, and Position 1 must not be in the machining area.

- **Rotate APC to Position 2** *F6*—sets the command to rotate the APC table to place Position 2 inside the machining area. The APC table must be squared and in the raised position, and Position 2 must not be in the machining area.

- **Lower Pallet Changer** *F7*—sets the command to lower the APC table to the Down position. APC table must be squared and the table must not be Down.

When all requirements for a softkey action are met, a prompt displays and the Start Cycle lamp flashes. Pressing the Start Cycle button executes the command. The Start Cycle lamp stops flashing during execution of the command and is turned off when the command is completed.

Selecting another softkey before pressing the Start Cycle button cancels the pending command and causes the Start Cycle lamp to cease flashing. If requirements for a softkey action are not met, a message displays indicating why the action cannot be commanded.

## Manually Operate the APC from the Load Station

The remote APC Panel (shown in the figure below) is located on the front of the machine enclosure, below the front enclosure doors. The APC Panel is used to manually raise and lower the APC table, and to rotate between Position 1 and Position 2.
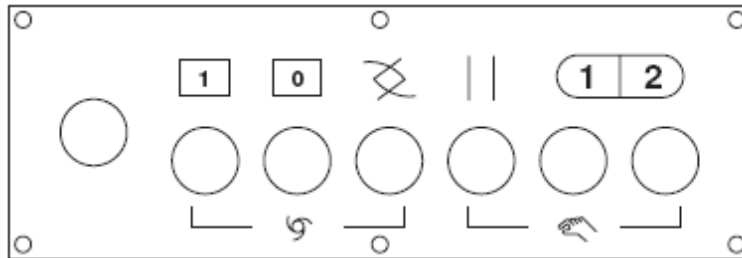


*Figure 4–14. Remote APC Panel*

### Remote APC Panel Pushbuttons

These are the Remote APC Panel pushbuttons, from left to right.

- **Emergency Stop**—disables all axis servo drives and stops all moves of the pallet changer.

- **Cycle Start**—initiates a selected operation that runs until it is completed. The Cycle Start pushbutton flashes while the operation is pending, is steady when the operation is in process, and turns off when the operation is completed.

- **Cycle Stop**—cancels any pending operation (if the Cycle Start lamp is flashing, it will turn off). Pressing the Cycle Start pushbutton after an operation is cancelled by the Cycle Stop pushbutton results in no APC motion.

- **Workzone Confirmation Or Ready to Rotate**—this pushbutton is not functional for manual pallet changer operations.

  ⇨ The console's Start Cycle pushbutton flashes when the Hydraulic Lift, APC Table Rotate to Position 1, or APC Table Rotate to Position 2 pushbutton is illuminated and the Cycle Start pushbutton is flashing. Pressing the console's Stop Cycle pushbutton cancels the operation.

- **Hydraulic Lift**—commands the APC table to be lifted or lowered and displays a message, depending on the current position of the APC table:

  - If the APC table is in the Down position, an "APC unit will raise" message displays on the console.

  - If the APC table is in the Up position, an "APC unit will lower" message displays on the console.

  When the Hydraulic Lift pushbutton is pressed, it illuminates and the Cycle Start lamp flashes. The operation will not begin until the Cycle Start pushbutton is pressed.

  The Hydraulic Lift pushbutton will no longer be illuminated when the operation is completed or if the Cycle Stop pushbutton is pressed.

- **APC Table Rotate to Position 1**—if the external pallet is Pallet 1, commands the APC table to rotate Pallet 1 into the machining area. The APC table must be in the Up position for APC Table Rotate to Position 1 to complete.

  Pressing the APC Table Rotate to Position 1 pushbutton causes the Cycle Start pushbutton to flash and an "APC unit will rotate to position 1" message to display on the console. The operation will not begin until the Cycle Start pushbutton is pressed.

  The APC Table Rotate to Position 1 will no longer be illuminated when the operation is completed or if the Cycle Stop pushbutton is pressed.

- **APC Table Rotate to Position 2**—if the external pallet is Pallet 2, commands the APC table to rotate Pallet 2 into the machining area. The APC table must be in the Up position for APC Table Rotate to Position 2 to complete.

  Pressing the APC Table Rotate to Position 2 pushbutton causes the Cycle Start pushbutton to flash and an "APC unit will rotate to position 2" message to display on the console. The operation will not begin until the Cycle Start pushbutton is pressed.

  The APC Table Rotate to Position 2 will no longer be illuminated when the operation is completed or if the Cycle Stop pushbutton is pressed.

  ⇨ The following conditions must be met before initiating a manual pallet changer operation from the load station:

  - The machine must be calibrated and in Manual machine mode.
  - Servo power must be on.
  - ATC must be in the Home position.
  - Z-axis position must be AT ZERO.
  - HYDRAULIC STATUS must be No Alarm.
  - Front enclosure doors must be closed and the pallet must be squared. Refer to *Squaring the APC Table, on page 4 - 72* for instruction on squaring the APC table.

Follow these steps to initiate a manual pallet changer operation from the load station:

1. Press the Hydraulic Lift button on the Remote APC panel. The button lights and the Cycle Start lamp flashes.

2. Press the Cycle Start button:

   a. The APC table raises and the Cycle Start button stops flashing, but remains lit.

   b. When the APC table is in the Up position, both the Hydraulic Lift and Cycle Start buttons are no longer lit.

3. Press the APC Table Rotate To Position 1 button. The APC Table Rotate To Position 1 button lights and the Cycle Start button flashes.

4. Press the Cycle Start button:

   a. The APC table begins a 180° high-speed rotation, followed by low-speed rotation into position.

b.   The Cycle Start button stops flashing, but remains lit.

c.   When the APC table is in position, both the Cycle Start and APC Table Rotate To Position 1 buttons are no longer lit.

5.   Press the Hydraulic Lift button on the Remote APC panel. The Hydraulic Lift button lights and the Cycle Start button flashes.

6.   Press the Cycle Start button:

a.   The APC table lowers and the Cycle Start button stops flashing.

b.   When the APC is in the Down position, both the Cycle Start and Hydraulic Lift buttons are no longer lit.

## Manual APC Table Rotation

Near the front of the APC table is a Manual Table Release Lever. This lever provides the input for the APC Pin Lock. To manually rotate the table without using the remote APC Panel, pull the Release Lever out. This allows the APC table to be rotated by hand for part setup. After rotating the APC table, the arrows on the pallet must be aligned with the arrows on the APC table's platform before executing a pallet exchange.

⇨   The APC table must be in Down position before using the Manual Table Release Lever to manually rotate the table.

⚠   Failure to align arrows prior to executing a pallet exchange can result in the APC table not positioning correctly in the machining area.

# Manual Safety Override Mode

For machines with CE safety circuit switches enabled, the Manual Safety Override mode allows the enclosure doors to be opened when the system is in Manual mode so the operator or supervisor can perform limited manual machine operations.

While the doors are open, manual jog feed is restricted. The jog feed is limited to 2 meters per minute, or approximately 80 inches per minute, with manual safety override enabled. No jogging is permitted with manual safety override disabled.

⇨ Manual operations that cannot be started with the doors open, regardless of manual safety override mode, are:

- Spindle On
- Park Machine
- Warm Up Machine
- Calibrate Machine
- Spindle Orient
- Chip Removal
- Tool Changer Functions

When the Manual Safety Override mode is enabled, it remains enabled until control power is turned off, the Emergency Stop button is pressed, or machine power is turned off. A fault will disable the Manual Safety Override mode also.

If you try to jog the axes with the doors open without enabling the manual safety override mode, a prompt appears to cycle the keyswitch or enter the access code to enable jog.

## CE Diagnostics

To enable the Safety Override mode, access the CE Status & Diagnostics screen.

1. From the Manual mode screen, select the Diagnostics softkey.
2. Select the CE Diagnostics softkey. The following screen appears:
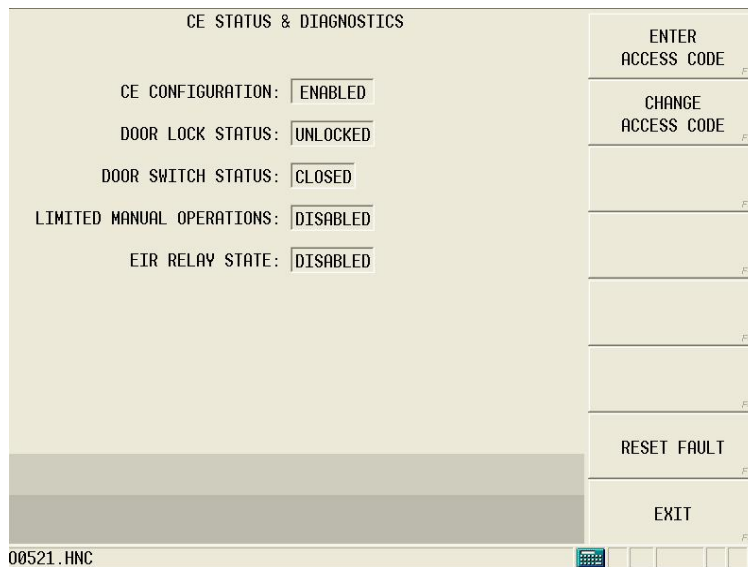
*Figure 4–15. CE Status & Diagnostics Screen*

The fields on the CE Status & Diagnostics screen are defined as follows:

- **CE Configuration**—Displays CE logic configuration. Enabled or disabled in Integrator Support Services.

- **Door Lock Status**—Displays status of all doors with CE safety circuit switches Unlocked or Locked. The Door Switch Status must be Closed when the Door Lock Status is Locked.

- **Door Switch Status**—Displays status as Closed or Open. The switch must be Closed when the Door Lock Status is Locked.

- **Limited Manual Operations**—Displays status as Disabled or Enabled. Status is Enabled after entering an access code. Enabled allows limited safe motion with the doors open.

- **EIR Relay State**—Displays status as Disabled or Enabled. When enabled, the enclosure doors have been locked due to a process request or an active process.

## Entering the Access Code

A numeric access code is required to enable Limited Manual Operations. Follow these steps to enter the access code:

1. Select the Enter Access Code softkey. A message appears with a prompt for entering a 4 digit code.

2. Enter a numeric code using the console keypad. Asterisks (*) appear representing each entered digit. When the fourth digit is entered, the asterisks and message disappear.

   ⇨ Using a non-numeric key cancels the operation.

When the code is successfully entered, the Limited Manual Operations feature is enabled,

as indicated in the status field.

For initial setup of the Access Code, the user is prompted to enter a new code and then to enter the same code a second time before it is accepted.

## Changing an Existing Access Code

To change an existing access code, follow these steps:

1. Select the Change Access Code softkey. A message appears asking you to enter the current code.

2. Enter the current numeric code from the console keypad. A message appears asking you to enter the new code.

3. Enter the new code. A message appears asking you to re-enter the new code for confirmation.

   • If the new code is not re-entered, the code is not changed.

   • When the new code is verified, the code will change and a message appears to confirm the action.

# Graphics

WinMax graphics include *Toolpath and Solid Graphics, on page 4 - 83*. Real-time graphic animation is available with the Runtime tool, which shows a simulated tool cutting the part on screen while the machine is cutting the actual part.

⇨ Solid graphics and Runtime tool are only available with the Advanced Verification Graphics option.

## Graphics Settings

Graphics preferences are set in the Graphics Settings screen, which is accessed with the GRAPHICS SETTINGS *F8* softkey. The Graphics Settings screen shows the following:

### Display Properties

Fields on the Display tab are:

- **SHOW GRAPHICS**—set the type, either Show All, Toolpath, or Solids.
- **TOOL PATH**—specify Yes or No to show the tool path (appears as red dashed line).
- **PART SURFACE**—specify Yes or No to show the part surface (appears as solid cyan line).
- **RAPIDS**—specify Yes or No to show rapid moves (appears as yellow solid lines).
- **STOCK OUTLINE**—specify Yes or No to show the stock outline, which is set in the stock geometry screen (appears as green solid line). Refer to the "Stock Geometry" section in *Part Setup* for more information.
- **PLUNGES**—specify Yes or No to show plunge moves (appears as purple lines).
- **ENABLE RUNTIME TOOL DISPLAY**—specify Yes or No to see simulated tool move around the part while the machine is running; tool also shown when jogging tool near part.
- **BACKGROUND COLOR**—choose Black or White background color.
- **DEFAULT VIEW**—set the default view when WinMax is started, either XY plane, XZ plane, Isometric, or All Views.
- **DEFAULT ZONE (dual-zone machines)**—set the zone initially displayed in graphics when the application is started.

### Performance Properties

Fields on the Performance tab are:

- **SHOW PECKS OF CONTOURS**—specify Yes or No to show all the roughing tool path, applies only to Toolpath graphics.
- **SHOW PECKS OF 3D SURFACES**—specify Yes or No to show all the pecks during milling of 3D mold and swept surface blocks.
- **USE CHORD ERROR FROM PROGRAM**—specify Yes or No to use the chord

error value programmed in general parameters when rendering curves in NC.

- **GRAPHICS CHORD ERROR**—enter the chord error to be used when drawing curves in graphics.

- **SOLIDS GRAPHICS OPTIMIZATION**—specify None, Display Lists, or Vertex Buffer.

- **WIRE GRAPHICS OPTIMIZATION**—specify Display Lists or Vertex Buffer.

- **REFRESH SPEED**—choose the refresh speed for graphics rendering. Choose one of seven options between fastest speed (Fastest Completion) to smoothest rendering (Smoothest).

## Axis Configuration

Select the program type and axis settings in this tab to optimize graphics viewing.

# Toolpath and Solid Graphics

Toolpath graphics displays a wireframe view of the part. Toolpath is also animated to show the tool in motion if the runtime tool is displayed (only available with the Advanced Verification Graphics option).

Solid graphics displays a solid 3-dimensional part; only available with the Advanced Verification Graphics option.
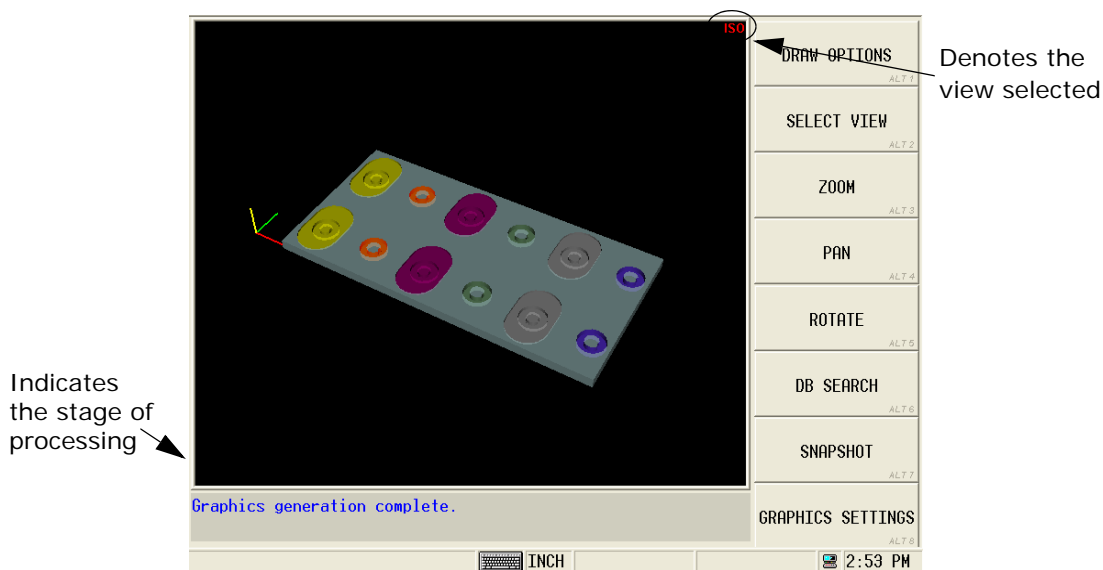


*Figure 4–16. Solid graphic display*

The console Draw button initiates drawing. The Draw Options menu is displayed (for both Toolpath and Solid) to control drawing and animation:

- **DRAW OPTIONS** *F1*—displays the following draw options:

    - **DRAW (PAUSE)** *F1*—starts drawing the part. When drawing is in progress, the softkey displays PAUSE—select it to pause the drawing. When paused, the softkey displays DRAW—select it to resume drawing.

    - **ACCELERATED DRAW** *F2*—displays only the completed drawing without showing each block as it is drawn. There may be a delay before the completed drawing appears on screen; selecting the SHOW PROGRESS F2 softkey displays the drawing at the current point each time the softkey is selected.

    - **SINGLE STEP** *F3*—displays only one step or program block at a time. Subsequent selection of SINGLE STEP will execute the next step or block.

    - **NEXT TOOL CHANGE** *F4*—displays cutting from one tool change to the next.

- **SELECT VIEW** *F2*—choose XY plane, XZ plane, YZ plane, Isometric, or All Views.

- **ZOOM** *F3*—magnifies an area of the graphic. Select the softkey and choose one of the following:

    - **ZOOM IN** *F1*—magnify incrementally by 20% each time the softkey is selected.

    - **ZOOM OUT** *F2*—shrink view incrementally by 10% each time the softkey is selected.

    - **FIT TO VIEW** *F3*—returns drawing to full view.

    - "Touch-and-drag"—place stylus at a point on the screen and drag it to expand the box over the area you wish to magnify. When stylus is lifted from screen the area will be magnified.

- **PAN** *F4*—moves the graphic up, down, left, or right. Use the softkeys or touch the screen and drag the graphic.

- **ROTATE** *F5*—changes the rotation of the graphic.

- **DB SEARCH** *F6*—jumps to a selected data block. Select the softkey, touch an area on the graphic touchscreen, and select the JUMP TO BLOCK softkey.

- **SNAPSHOT** *F7*—creates an image of the solid that is stored with the HWM file. The image is displayed on the Program Manager screen when the program is highlighted.

- **GRAPHICS SETTINGS** *F8*—displays the Graphics Settings screen.

# RECORD OF CHANGES

704-0116-110rB, February 2010

       Revised by: H. Arle

       Approved by: D. Skrzypczak, February 2010

### Changes

- Updates to reflect software changes.


704-0116-110rA, December 2009

       Revised by: H. Arle

       Approved by: D. Skrzypczak, December 2009

### Changes

- Added Tool Change Optimization content.
- Added Manual Rapid Move content.
- Revisions to Absolute Tool section.
- Updates to reflect software changes from v7.3.

# INDEX

## Numerics

## A

## N

## O

## P

HURCO.