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.

© 2012 Hurco Companies, Inc. All rights reserved.

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](http://www.hurco.com)
# Table of Contents

## WinMax Mill Conversational Programming

### Documentation Conventions
- Console Buttons and Keys
- Icons

### Programming and Operation Information
- Using the On-screen Help
- Printing the Programming Manuals

### Hurco Conversational Overview
- Part Programming
  - Data Blocks
  - Data Block Layout
  - Data Block Creation and Navigation
  - Select Tool from List
  - Temporary Parameter Change
  - Automatic Calculations
- Programming Dual-Zone Machines
- Cutter Compensation
  - Climb Milling (Left)
  - Conventional Milling (Right)
  - Profile Left and Right
  - Cutter Compensation Lookahead
- Calculated Plunge Points for Milling Cycles
- Lead In/Out Moves
  - Lead Angle
  - Lead Length

### Milling Operations
- Creating a Milling Data Block
- Mill Face
- Circle Data Block
- Ellipse Data Block
- Mill Frame
- Finish Tool for a Mill Frame, Mill Circle or Mill Contour
- Stick Lettering
- True-Type Lettering
- Stick Lettering Along Contour
- True-Type Lettering Along Contour
- Serial Number Stick Lettering
  - Text tab
  - Orientation tab
  - Format tab
  - History tab
- HD3 Lettering
- Swept Surface
  - Draw Profile Contour
  - Draw Along Contour
  - Swept Surface
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Graphics On/Off</td>
<td>3</td>
</tr>
<tr>
<td>Change Parameters</td>
<td>5</td>
</tr>
<tr>
<td>Change Parameters (General)</td>
<td>4</td>
</tr>
<tr>
<td>Change Parameters (Milling)</td>
<td>4</td>
</tr>
<tr>
<td>Change Parameters (Holes)</td>
<td>4</td>
</tr>
<tr>
<td>Change Parameters (Probing)</td>
<td>5</td>
</tr>
<tr>
<td>Change Parameters (Performance)</td>
<td>5</td>
</tr>
<tr>
<td>Change Part Setup</td>
<td>6</td>
</tr>
<tr>
<td>Machine Function</td>
<td>7</td>
</tr>
<tr>
<td>Lube Cycle</td>
<td>7</td>
</tr>
<tr>
<td>Comment Block</td>
<td>8</td>
</tr>
<tr>
<td>Insert Block</td>
<td>8</td>
</tr>
<tr>
<td>NC/Conversational Merge</td>
<td>1</td>
</tr>
<tr>
<td>Record of Changes</td>
<td>1</td>
</tr>
<tr>
<td>Index</td>
<td>1</td>
</tr>
</tbody>
</table>
# LIST OF FIGURES

| Figure 1–1. | Length, Angle, and End Point Relationship | 1 - 9 |
| Figure 1–2. | Line Segment with Unknown Y End and XY Angle | 1–10 |
| Figure 1–3. | Coordinates of a Clockwise Arc | 1–11 |
| Figure 1–4. | Arc Segment with Unknown X End and X Center | 1–11 |
| Figure 1–5. | Cutter Compensation | 1–13 |
| Figure 1–6. | Tool Paths for Cutter Compensation | 1–13 |
| Figure 1–7. | Determining Maximum Offset | 1–17 |
| Figure 1–8. | Lead In/Out Moves | 1–20 |
| Figure 2–1. | Face Milling | 2–3 |
| Figure 2–2. | Mill Frame Geometry tab | 2–10 |
| Figure 2–3. | Mill Frame Corners tab | 2–11 |
| Figure 2–4. | Mill Frame Corners example | 2–11 |
| Figure 2–5. | X Reference and Y Reference locations | 2–16 |
| Figure 2–6. | Example of True-Type Lettering | 2–17 |
| Figure 2–7. | X Reference and Y Reference locations | 2–18 |
| Figure 2–8. | Mapping | 2–19 |
| Figure 2–9. | Stick Lettering Along Contour Lettering Parameters screen | 2–21 |
| Figure 2–10. | Stick Lettering Along Contour Start Segment | 2–22 |
| Figure 2–11. | Stick Lettering Along Contour Arc Segment | 2–22 |
| Figure 2–12. | Example of Stick Lettering Along Contour | 2–23 |
| Figure 2–13. | True Type Lettering Along Contour opening screen | 2–24 |
| Figure 2–14. | Mapping | 2–25 |
| Figure 2–15. | True Type Lettering Along Contour Start Segment | 2–25 |
| Figure 2–16. | True Type Lettering Along Contour Arc Segment | 2–26 |
| Figure 2–17. | Example of True-Type Lettering Along Contour | 2–26 |
| Figure 2–18. | HD3 Serial Number Lettering Text tab | 2–27 |
| Figure 2–19. | Serial Number Orientation tab | 2–29 |
| Figure 2–20. | X Reference and Y Reference locations | 2–30 |
| Figure 2–21. | Serial Number Format tab | 2–30 |
| Figure 2–22. | Serial Number History tab | 2–32 |
| Figure 2–23. | X Reference and Y Reference locations | 2–34 |
| Figure 2–24. | Bidirectional Field | 2–38 |
| Figure 2–25. | Stock Allowance | 2–38 |
| Figure 2–26. | Step Size Example | 2–39 |
| Figure 2–27. | Peck Depth Example | 2–39 |
| Figure 2–28. | Line Slot rotated 90° | 2–41 |
| Figure 2–29. | Mill Slot Start tab | 2–42 |
| Figure 2–30. | Mill Slot Geometry tab for a Line slot | 2–43 |
| Figure 2–31. | Mill Slot Geometry tab for an Arc slot | 2–43 |
| Figure 2–32. | Mill Slot Caps tab | 2–44 |
| Figure 2–33. | Append Arc Start Cap and Line End Cap | 2–45 |
| Figure 2–34. | Include Arc Start Cap and Line End Cap | 2–45 |
| Figure 2–35. | Mill Polygon screen | 2–46 |
| Figure 2–36. | Tool movement when Thread Direction is down | 2–49 |
| Figure 2–37. | Tool movement when Thread Direction is up | 2–49 |
| Figure 2–38. | Mill Thread screen | 2–50 |
| Figure 2–39. | Blend Arc Examples | 2–58 |
| Figure 2–40. | 180º 3D Arc | 2–61 |
Figure 3–1. Reference Points Relative to Drill Operations .......................... 3 - 3
Figure 3–2. Countersunk screw head ....................................................... 3 - 6
Figure 3–3. Gun Drilling ................................................................. 3 - 7
Figure 3–4. Gun Drill Positions .......................................................... 3 - 8
Figure 3–5. Custom Drill Tool tab ....................................................... 3 - 10
Figure 3–6. Custom Drill Entry tab ..................................................... 3 - 12
Figure 3–7. Custom Drill Break Out tab ............................................. 3 - 13
Figure 3–8. Custom Drill Bottom tab ................................................... 3 - 14
Figure 3–9. Custom Drill Re-entry tab ................................................. 3 - 15
Figure 3–10. Custom Drill Out tab ..................................................... 3 - 16
Figure 3–11. Custom Drill Retract tab ............................................... 3 - 17
Figure 3–12. Two-Bladed Boring Bar with Adjustment Dial ................. 3 - 19
Figure 3–13. Reaming Tool ............................................................. 3 - 19
Figure 3–14. Two-bladed Boring Bar with Adjustment Dial ............... 3 - 20
Figure 3–15. Reaming Tool ............................................................. 3 - 21
Figure 3–16. Bolt Circle Pattern ....................................................... 3 - 24
Figure 3–17. Bolt Circle Hole or Pattern Locations Copy and Convert softkeys ... 3 - 27
Figure 3–18. Holes to Pattern Locations softkeys ............................... 3 - 28

Figure 4–1. Loop Rectangular ......................................................... 4 - 2
Figure 4–2. Loop Linear ............................................................... 4 - 3
Figure 4–3. Loop Angular ............................................................ 4 - 4
Figure 4–4. Loop Rotate ............................................................. 4 - 5
Figure 4–5. Pattern Scale ............................................................. 4 - 6
Figure 4–6. Mirror Image ............................................................. 4 - 8
DOCUMENTATION CONVENTIONS

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.

Console Buttons and Keys

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

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

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.
Important

→ Ensures proper operation of the machine and control.

Troubleshooting

❓ Steps that can be taken to solve potential problems.

Hints and Tricks

💡 Useful suggestions that show creative uses of the WinMax features.

Where can we go from here?

➕ Lists several possible options the operator can take.

Table of Contents

To assist with onscreen PDF 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.
PROGRAMMING AND OPERATION INFORMATION

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

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

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

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

Using the On-screen Help

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

- Buttons in the upper left-hand corner of the Help screen are used to move through Help topics and print screens.
  - Use the Hide button to hide the navigation pane.
  - Use the Back button to return to the previous Help screen.
  - Use the Print button to print the current displayed Help topic, if a printer is attached and configured. See Printing the Programming Manuals, on page - iv for more information about printing.

- Use the arrow buttons to move between pages within a Help topic and to move through topics.

- Use the Contents tab for a list of information sorted by subject:
  1. Select the “+” to expand the topic and view sub-topics.
  2. Select the topic to display it.

- Use the Index tab to show the Help index:
  1. Quickly scroll to an index topic by typing the topic in the box at the top of the index.
  2. Select a topic and the Display button to view the topic.
• Use the **Search** tab to search the Help for a word or phrase:
  1. Type the search word(s) into the text box at the top of the pane.
  2. Select the List Topics button. A list of topics that contain the search word(s)
     is displayed.
  3. Select a topic and the Display button to view that topic.

• Use the **Favorites** tab to save Help topics for quick access:
  1. Select the Add button at the bottom of the pane to add the current topic.
  2. Select a topic from the Favorites list, and select the Display button to view
     it.
  
  • Select a topic from the Favorites list, and select the Remove button to
    remove it from the list.

## Printing the Programming Manuals

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

1. From the Input screen, select the 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.](image)

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.
Conversational programming allows you to create a part program from a blueprint or program worksheet while working at the machine. Operating selections and prompts on the screen lead you through the steps necessary to enter the data for a part program. Machining operation information is stored within data blocks describing each operation to be performed.

Create and access the data blocks through the **Part Programming** softkey or the Part Programming icon.

- **Part Programming**, on page 1 - 2
- **Milling Operations**, on page 2 - 1
- **Holes Operations**, on page 3 - 1
- **Patterns Operations**, on page 4 - 1
- **Special Operations**, on page 5 - 1
- **NC/Conversational Merge**, on page 6 - 1

These topics are discussed in the Overview section:

- **Part Programming** .......................... 1 - 2
- **Cutter Compensation** ......................... 1 - 13
- **Calculated Plunge Points for Milling Cycles** .......................... 1 - 19
- **Lead In/Out Moves** ........................... 1 - 20
Part Programming

Use part programming to communicate plans from a blueprint to the CNC using data blocks. Data blocks appear in numerical order as they are created. To create a Conversational Part Program:

1. Press the Input console key to display the Input screen. The Input screen allows you to access setup functions including Import Functions, Tool Setup, and Program Parameters. See the Getting Started with WinMax Mill manual for more information.

2. Enter machining operation information in the Part Setup and Tool Setup functions. This information is stored within data blocks describing each operation to be performed. Create and access the data blocks through the Input screen.

3. Select the Part Programming softkey to begin programming. The system displays either the first block of an existing program or a New Block screen for new programs.

Data blocks can be managed with the following programming operations:

- Access softkeys to create a new data block or edit an existing data block in the Input screen.
- View a list summarizing a part program's data blocks on the Program Review screen. See Review Mode in Getting Started with WinMax Mill for more information.
- View the current program's tool list on the Tool Review screen. Refer to Part Program Tool Review in Getting Started with WinMax Mill for more information.
- Use the Insert Block Before softkey to Insert a new data block between two sequentially numbered blocks. The data blocks following the inserted block automatically renumber.
- Holes blocks contain operations, and Mill Contour blocks contain segments.

Part Programming softkeys access the following options:

- **Position**—Insert instructions to move the tool away from the part (or fixture), or to stop a program. A Position block is most often used to move the table to an X-Y location and is normally used at the end of the program and at any time the tool must be moved to the Z Top position of the Safety Work Region. See Position Data Block, on page 5 - 2 for more information.

- **Holes**—Select Drill, Tap, Bore and Ream drilling operations, and Back Spotface and Bolt Circle hole drilling. The locations of these holes on the part can be specified using the Locations operation, and the holes can also be programmed in a Bolt Circle pattern. See Holes Data Block, on page 3 - 2 for more information.
• **Milling**—Program milling operations. See also *Lines/Arcs Data Blocks, on page 2 - 53, Circle Data Block, on page 2 - 5, Mill Frame, on page 2 - 10, Mill Face, on page 2 - 3, Ellipse Data Block, on page 2 - 8, HD3 Lettering, on page 2 - 33, HD3 Lettering, on page 2 - 33, and Swept Surface, on page 2 - 37.* Special milling blocks Triangle, Diamond 1 Face, Diamond 2 Faces, and Hexagon are accessed with the Special ➔ softkey. See *Milling Operations, on page 2 - 1* for more information.

• **Patterns**—Repeat or modify a sequence of one or more data blocks. A program sequence can be repeated in a rectangular, linear, angular, or rotated pattern or at specified locations. This operation is also used to scale or mirror a programmed part. See *Patterns Operations, on page 4 - 1* for more information.

• **Miscellaneous**—Access these softkey functions. Refer to *Temporary Parameter Change, on page 1 - 8* for details.
  - Graphics Off
  - Graphics On
  - Change Parameters
  - Change Part Setup
  - Machine Function
  - Probe Part Setup
  - Lube Cycle
  - Comment
  - Insert
  - Tool Monitoring (Probing)
  - Part Inspection

• **NC Program Call**—Access an NC program from within the conversational program with this softkey.

• **Rotary**—Select Rotary parameters and programming of position blocks, lines and arcs, circles, frames, loops, locations and patterns. Refer to Rotary Programming in *WinMax Mill Options* for more information.

• **Exit**—Returns to the previous screen.
Data Blocks

Use the following basic programming operations when creating data blocks:

- Create a new data block or edit an existing data block through the Input screen.
- Insert a new data block between two sequentially numbered blocks. Data blocks following the inserted block will be automatically renumbered.
- Delete an existing data block. The data blocks following the deleted block renumber automatically.
- View a list of the part program’s data blocks on the Review Program screen.
- Copy, edit, or delete information in a data block using the Block Editor screen.
- Data blocks within a part program contain specific machining information:
  - Operation type (position, holes, milling, patterns).
  - Tool number to be used for the operation.
  - Dimension coordinates relative to X, Y, and Z axes.
  - Feedrate at which the axes move.
  - Locations of operations on the part.

Segments

Mill Contours, 3D Mold Contours, and Rotary Mill Contours are composed of a combination of segments. A segment is a line, arc, blend arc, helix, or 3D arc with programmed single or combined X, Y, Z axis movement at a programmed feedrate. A new segment begins where a different motion from the previous segment is programmed. Segments appear in a numbered list in the Program Review Screen under the data blocks to which they apply.

The order in which segments are programmed determines the direction the tool moves from the start point.

Setting Right (conventional milling) or Left (climb milling) in the Milling Parameters overrides the direction of the programmed path.
Data Block Layout

The layout of most data blocks includes the part geometry at the top of the screen with the tool information below it, as in the following example:

```
<table>
<thead>
<tr>
<th>BLOCK</th>
<th>MILL FRAME</th>
</tr>
</thead>
<tbody>
<tr>
<td>CORNERS</td>
<td></td>
</tr>
<tr>
<td>X CORNER</td>
<td>0.000</td>
</tr>
<tr>
<td>Y CORNER</td>
<td>0.000</td>
</tr>
<tr>
<td>X LENGTH</td>
<td>0.000</td>
</tr>
<tr>
<td>Y LENGTH</td>
<td>0.000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ROUGHING</th>
<th>FINISHING</th>
<th>SFQ</th>
<th>ALLOWANCES</th>
</tr>
</thead>
<tbody>
<tr>
<td>TOOL</td>
<td>0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>MILLING TYPE</td>
<td>ON</td>
<td></td>
<td></td>
</tr>
<tr>
<td>MILL FEED</td>
<td>0</td>
<td>PECK DEPTH</td>
<td>0.000</td>
</tr>
<tr>
<td>SPEED (RPM)</td>
<td>0</td>
<td>PLUNGE FEED</td>
<td>0</td>
</tr>
</tbody>
</table>
```

Information for the tool is imported from Tool Setup; values may be changed/overridden in the data block without affecting the value stored in Tool Setup.

In the WinMax documentation, discussion of the data blocks will mostly focus on the part geometry fields unless there are tool fields that are specific to the type of data block. Information about the common tool fields can be found in Tool Setup, on page 4 - 21, in Getting Started with WinMax Mill.
Data Block Creation and Navigation

These basic concepts are helpful to know when creating and editing data blocks:

- To move through the fields on a data block screen, press the ENTER key, use the up and down console arrows, the up and down arrows on the optional keyboard, or the optional mouse.

- When the cursor is on the Block field in a data block, these softkeys appear: **Previous Block, Next Block, Delete Block**, and **Insert Block Before**.

- If a block should not be used in the program, move the cursor to the Block field and select the **Delete Block** softkey to remove it from the program.

- To insert a data block before an existing block, select the **Insert Block Before** softkey at the beginning of this block. The New Block screen appears with softkey choices.

- To create a data block after the last block in a program, select the **Insert Block Before** softkey from the End of Program screen.

- When the cursor is on the Operation field in a Holes data block, these softkeys appear: **Previous Hole Operation, Next Hole Operation, Insert Hole Operation**, and **Delete Hole Operation**.

- When the cursor is on the Segment field in a Mill Contour data block, these softkeys appear: **Previous Segment, Next Segment, Delete Segment**, and **Insert Segment Before**.

- In Ultimax Classic Edit Mode, to tell the system to read and remember new data that is typed into a field, press the ENTER key. Pressing ENTER also automatically advances the cursor to the next field.

- A field can be skipped (allowed to remain unchanged) by pressing the ENTER key when the cursor is positioned on the field or by using the up and down console or keyboard arrows.

- Use the Page Up or Page Down console keys to move through data blocks and tool setup screens, or press and hold the CTRL key and press the right and left arrow keys.

- Use the optional keyboard’s ALT + SHIFT + left arrow keys to navigate to the previous data block. On the virtual keyboard, sequentially touch the ALT, SHIFT and left arrow keys to move to the previous data block.

- Use the CTRL + ALT + left arrow console keys to navigate to the previous data block.

- Use the optional keyboard’s ALT + SHIFT + right arrow keys to navigate to the next data block. On the virtual keyboard, sequentially touch the ALT, SHIFT and right arrow keys to move to the next data block.

- Use the CTRL + ALT + right arrow console keys to navigate to the next data block.

- Page Down (or CTRL + right arrow) on the last data block adds a new data block.

- Page Down (or CTRL + right arrow) on the last tool adds a new tool.
• While programming a Holes data block, select the **Previous Hole Operation** or the **Next Hole Operation** softkeys to move between the operations within the data block, or use the PAGE UP (CTRL + left arrow) or PAGE DOWN keys (or CTRL + right arrow). These operations describe separate portions of the holes operation.

• Use either the PAGE DOWN key, CTRL + right arrow, or the **Next Hole Operation** softkey to add a new operation when positioned on the last Hole Operation screen.

• When programming a Mill Contour data block, select the **Previous Segment** or **Next Segment** softkeys, or use the PAGE UP (CTRL + left arrow) or PAGE DOWN (or CTRL + right arrow) keys to move between the segments within the data block. These segments describe separate portions of the contour.

• Use either the PAGE DOWN key, CTRL + right arrow, or the **Next Segment** softkey to add a new segment when positioned on the last Mill Contour screen.

• When positioned on the End of Contour, Holes End Block, or End of Program screen, use the PAGE DOWN key or CTRL + right arrow to add a new contour segment, holes operation, or data block.

See *WinMax Interface Environment* in *Getting Started with WinMax Mill* for more details about navigation.

### Select Tool from List

Use the following steps to select a tool using the SELECT TOOL FROM LIST F3 softkey:

1. In a program block screen, place the cursor in the TOOL field and choose the SELECT TOOL FROM LIST F3 softkey.

2. Locate the desired tool using one of the following methods:
   - Scroll through the list of tools and select the desired tool to highlight it.
   - Begin typing the number of the desired tool. A pop-up box will appear when the first digit is entered; enter the entire number and select OK to jump to that tool number in the list.

3. Select the SELECT TOOL F1 softkey, which enters the tool into the data block or segment.
   - If Tool Type Checking is enabled, only tools that are suitable for the programmed operation can be selected.
   - Check the Display Tool Notes checkbox to view the Notes for each tool. Notes can be entered in Part Program Tool Review and Advanced Tool Settings when the Tool & Material Library option is installed.
Temporary Parameter Change

For some programs, you may want to insert a temporary parameter change block between data blocks.

1. Display the data block for which the parameter change will first be used.
2. Select the **Insert Block Before** softkey. The New Block screen appears.
3. Select the **Miscellaneous** softkey. Choose the Change Parameter Softkey.

See *Change Parameters, on page 5 - 3 and Change Part Setup, on page 5 - 6* for more information.

Automatic Calculations

The WinMax AutoCalc feature enables the system to calculate certain dimensions automatically after sufficient data has been programmed. If the system is to calculate a given dimension, leave that data field blank. After sufficient data is entered, the system automatically fills in the blank data field(s).

The screen displays a **Store Calculated Value** softkey for editing a field with a calculated value (indicated by "CAL" preceding the value). Select this softkey or the ENTER key to store the displayed value into the part program. Do not re-enter the data (i.e., 8.9199 Enter); the data is truly 8.9199xxx and will be calculated as such when entered with the softkey or the ENTER key.

To let the system calculate previously entered data clear the old value by following these steps:

1. Use the arrow keys to move the cursor to the field where the automatic calculation is needed.
2. Press the BACKSPACE key. The defined value is cleared from the program and the system-calculated dimension is displayed (or the field is blank waiting for additional data to be entered).
3. In some cases, if you need the system to recalculate the field, you may need to clear other fields that were automatically calculated. This is usually the case if the system calculates erroneous data indicated by "ERR" preceding the value.
**Line Segments**

The software uses data in the Line Segment fields as described below to perform automatic calculations:

- If the X End and Y End coordinates are entered, the system calculates the XY length and the XY angle values.
- The XY Angle is the angle of the line segment (from the start point to the end point), measured counterclockwise from the 3 o'clock position.

![Figure 1–1. Length, Angle, and End Point Relationship](image)

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Start Point</td>
</tr>
<tr>
<td>2</td>
<td>XY Length</td>
</tr>
<tr>
<td>3</td>
<td>End Point</td>
</tr>
<tr>
<td>4</td>
<td>XY Angle</td>
</tr>
</tbody>
</table>

- If both end points are unknown but the XY Length and XY Angle fields are programmed, the system calculates the values for the X End and Y End fields.
- If one end point coordinate and the XY Angle field are programmed, the system calculates the values for the unknown end point and the XY Length fields.
- If one end point coordinate and the XY Length field are programmed, the system calculates the values for the unknown end point and the XY Angle fields. However, unless the XY Angle is known, there are two possible solutions for the unknown end point, and the correct one must be determined for the program.
- When two possible solutions exist, the "Another end point exists" message appears and one of the two possible solutions appears in the unknown field.
Arc Segments

The software uses data in the Arc Segment fields as described below to perform automatic calculations:

- The center points plus the start points or end points provide the arc radius.
- The start points and end points plus the radius provide the two possible center points.
- Either of the end points and the center point provide the value of the other end point and the radius.
- When a known center point, start point, or end point and radius are provided, an unknown center point is provided.
Figure 1–3. Coordinates of a Clockwise Arc

Figure 1–4. Arc Segment with Unknown X End and X Center

1. Start Point
2. End Point
3. Center Point
4. Known Y End and Y Center
5. Unknown X End and X Center
6. Known X Start and Y Start
7. Known Radius
8. Solution #1
9. Center Point #1
10. End Point #1
11. Solution #2
12. Center Point #2
13. End Point #2
**Helix**

The software uses data in the Helix fields as described below to perform automatic calculations:

- The Z End and Sweep Angle provide data to calculate the Lead.
- The Sweep Angle is used to calculate the Z End.
- The X End, Y End, and Lead values provide the Z End.
- The Z End and Lead values provide the X End, Y End, and Sweep Angle values.
- The X End, Y End, X Center, and Y Center values supply the Radius.
- The Sweep Angle and either the X or Y end point provide the unknown X End or Y End.

**Programming Dual-Zone Machines**

With dual-zone machining, milling occurs in one zone while the other zone is being used to set up the part. A Change Part Setup data block is used to switch zones. The Enable Dual Zones parameter must be set for dual zones to use both zones; see the *Machine Parameters* section of *Getting Started with WinMax Mill*.
Cutter Compensation

Cutter Compensation is set in the **Milling Type** field in a milling program block.

Cutter Compensation allows you to choose the side of the contour the tool should begin cutting. The programmed tool automatically follows the finished contour of the part when cutter compensation is selected. Without cutter compensation, the centerline of the programmed tool follows the print line.

![Figure 1–5. Cutter Compensation](image)

1. Programmed path
2. Cutting tool
3. Path compensating for cutter radius
4. Cutter radius = amount of cutter compensation offset

The following diagram shows tool paths using no cutter compensation compared to tool paths using left and right compensation. When either right or left cutter compensation is selected, the tool is offset from the cutting path a distance equal to the tool’s radius. The tool begins cutting at the offset and moves in the selected direction.

![Figure 1–6. Tool Paths for Cutter Compensation](image)

<table>
<thead>
<tr>
<th></th>
<th>No Cutter Compensation</th>
<th>Cutter Compensation to the left</th>
<th>Cutter Compensation to the right</th>
<th>Tool</th>
<th>Centerline of contour</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>2</td>
<td>3</td>
<td>4</td>
<td>5</td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>5</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

![Diagram](image)
The following are the types of cutter compensation used for Mill Contours such as Lines, Arcs, Blend Arcs, Helices, True-Type Lettering, and 3D Blend Arcs:

- **On**—Locates the center of the tool on the programmed contour of the frame.
- **Left**—Performs climb milling. Refer to *Climb Milling (Left), on page 1 - 16*.
- **Right**—Performs conventional milling. Refer to *Conventional Milling (Right), on page 1 - 16*.
- **Profile Left**—Removes material from a contour for climb milling. Refer to *Profile Left and Right, on page 1 - 17*.
- **Profile Right**—Removes material from a contour for conventional milling. Refer to *Profile Left and Right, on page 1 - 17*.

The following are the types of cutter compensation used for milling Circles, Frames, and Ellipses:

- **On**—Locates the center of the tool on the programmed contour.
- **Inside**—the tool enters the part inside the contour and blend into the programmed contour using a 180° arc. Cutter compensation is automatically employed, and the edge of the tool remains inside a programmed contour.
- **Outside**—the tool enters the part outside the programmed contour and follow the outside contour. Cutter compensation is automatically employed, and the edge of the tool remains outside the programmed contour.
- **Inside Tangent**—the tool enters the part inside and tangent to the programmed contour. Cutter compensation is automatically employed, and the direction the tool travels depends upon the Milling Direction. The tool is withdrawn from the part inside and tangent to the programmed contour.
- **Outside Tangent**—the tool enters the part outside and tangent to the programmed contour. Cutter compensation is automatically employed, and the direction the tool travels depends upon the Milling Direction. The tool is withdrawn from the part outside and tangent to the programmed contour.
- **Pocket Boundary**—the tool cuts the part around the programmed boundary and avoid any programmed islands or pockets.
  - **Pocket Type**—The Pocket Type field appears with Inward and Outward softkey and drop-down list box choices when Pocket Boundary or Pocket Island is chosen for Milling Type. The choices define whether spindle movement spirals from inside the pocket or outside the pocket.
    - **Outward**—the tool begins cutting operations in the center region of the pocket and cut outward to the edge of the programmed boundary. Outward is used only for Circle and Frame data blocks without islands.
    - **Inward**—the tool cuts in from the outside of the defined boundary, avoiding the defined islands.
- **Pocket Island**—Defines islands within pockets on a part. As many islands as desired may be defined (subject to available memory), but all must fit within the defined boundary and must allow the tool to completely define the island. A Pocket Island cannot follow an Outward Pocket Boundary.
The order in which segments are programmed determines the direction the tool moves from the start point. Setting Conventional (right) or Climb (left) in **Milling Parameters** overrides the direction of the programmed path.

Unless the **Blend Offset** is set for 0.0 in Milling Parameters, the system automatically creates Lead In and Lead Out arcs for closed contours. Refer to **Lead In/Out Moves, on page 1 - 20**.

<table>
<thead>
<tr>
<th></th>
<th>Center-line</th>
<th>Inside</th>
<th>Outside</th>
<th>Inside Tangent</th>
<th>Outside Tangent</th>
<th>Pocket</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pecks</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Finish Pass Sides 0.008</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Floor 0.004</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cutter Comp</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Blend Offset (0.118)</td>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Blend Overlap (0.118)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pocket Overlap (10%)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>Conventional/Climb Mill</td>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>Circle</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Center Point</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Radius</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>In/Out = 3:00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ellipse X &lt; Y</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Center Point</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>X Radius</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Y Radius</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>In/Out = 3:00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ellipse X &gt; Y</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Center Point</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>X Radius</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Y Radius</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>In/out = 6:00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Frame</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Reference Corner</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>X Length</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Y Length</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>In/Out = 6:00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Climb Milling (Left)

Climb milling is the preferred method of cutter compensation, except when the fixturing is not rigid. In climb milling, the tool cuts in the same direction as the feeding motion. This is also known as "in-cut" or "down milling."

During climb milling, the spindle turns in a clockwise direction. The tool is on the left-hand side of the cut.

The advantages of using climb milling are as follows:

- The chip starts thick and allows easy penetration into the surface of the part, causing less tool wear and less power consumption.
- The tool force cuts in and down on the part, helping to hold the part in the fixture. The more rigid the fixture, the better the hold on the part.
- Chip removal is greater, and there is less re-cutting of chips or marring of the part surface.
- The cutting fluid is more accessible to the cutting surface.

Conventional Milling (Right)

During conventional milling, the cutting teeth move in the opposite direction to the feeding motion. This is known as "out-cut" or "up milling." When using conventional milling with a clockwise spindle direction, the tool is on the right-hand side of the cut.

The advantages of using conventional milling are as follows:

- The chip thickness starts at zero, causing less impact on the cutting teeth. This is ideal for setups that are not very rigid.
- The backlash in older machines is greatly diminished.
The programming sequence of Lines and Arcs segments determines the cutting direction. However, when programming Mill Frames, Circles and Ellipses, the cutting direction is determined by the values entered in Program Parameters.

Profile Left and Right

Profile Left and Profile Right work the same as Left (Climb) and Right (Conventional) milling, except that a **Max Offset** may be added.

**Maximum Offset**

The Max Offset field appears when either **Profile Left** or **Profile Right** is selected. Max Offset allows the cutter to be programmed to start at some specific distance away from the programmed profile and move toward the finished profile, using the the Pocket Overlap parameter, as each pass is completed. It is typically the radius of the largest inscribed circle minus the tool radius. Manually calculate the value and enter it into this field.

![Figure 1–7. Determining Maximum Offset](image)

Drawing the part on the graphics screen is the key to determining an optimum Maximum Offset value.

If a .500" diameter (.25" radius) End Mill is used to machine the part illustrated below (a circle with a 1" radius), the value for the Maximum Offset field is 0.75".

If the tool diameter is changed (i.e., the cutter is sharpened to a smaller diameter), the Maximum Offset value must be manually re-calculated and this new value programmed into Segment 0 of the Mill Contour's Start block. Refer to **Mill Contour, on page 2-53**.

The Tool diameter is programmed in Tool Setup and cannot be changed in Part Programming. See **Tool Setup** in **Getting Started with WinMax Mill** for more information.
Cutter Compensation Lookahead

The Cutcomp Lookahead field appears when the **Milling Type** field is set to Left, Right, Profile Left, or Profile Right. It identifies the number of segments in a contour that are checked to determine if the contour crosses itself and if the tool will fit into the contour. The default value is calculated based on the number of segments and whether the contour is opened or closed:

- **Open contours** - Cutcomp Lookahead value = number of segments - 3
  (If the number of segments is less than 5, the value is set to 2; if the number of segments is greater than 11, the value is set to 8.)
- **Closed contours** - Cutcomp Lookahead value = number of segments - 1
  (If the number of segments is less than 4, the value is set to 3; if the number of segments is greater than 9, the value is set to 8.)

To override the calculated value, type a user-defined value in the Cutcomp Lookahead field.

Select the RECALCULATE LOOKAHEAD softkey to restore the calculated Cutcomp Lookahead value.

CAL to the left of the data field indicates a calculated value; USR indicates a user-defined value.
Calculated Plunge Points for Milling Cycles

The following calculations are for determining the plunge points for milling cycles. The examples are for Climb Milling.

The Blend-in move is 90° for all examples.

The Arc of Radius = Blend Offset for all examples.

<table>
<thead>
<tr>
<th>Circle, Inside</th>
<th>Circle, Outside</th>
</tr>
</thead>
<tbody>
<tr>
<td>Occurs at 3 o’clock position on the circle.</td>
<td></td>
</tr>
<tr>
<td>X Plunge = X Center + Circle Radius - Blend Offset - Tool Radius</td>
<td>X Plunge = X Center + Circle Radius + Blend Offset + Tool Radius</td>
</tr>
<tr>
<td>Y Plunge = Y Center</td>
<td>Y Plunge = Y Center</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Frame, Inside</th>
<th>Frame, Outside</th>
</tr>
</thead>
<tbody>
<tr>
<td>Occurs at 6 o’clock position on the frame.</td>
<td></td>
</tr>
<tr>
<td>X Plunge = X Corner + (X Length/2)</td>
<td>X Plunge = X Corner + (X Length/2)</td>
</tr>
<tr>
<td>Y Plunge = Y Corner + Blend Offset + Tool Radius</td>
<td>Y Plunge = Y Corner - Blend Offset - Tool Radius</td>
</tr>
</tbody>
</table>

The plunge points apply to Circle and Frame milling cycles.
Lead In/Out Moves

With posted NC programs, use Lead In and Lead Out moves to control the tool path before the tool moves into position to cut the part.

A positive Lead Angle starts the tool path away from the programmed path when performing a Lead In move and ends away from the programmed path when performing a Lead Out move. A negative Lead Angle has the opposite effect.

Toolpath Graphics will not draw the Lead Angle or Lead Length motion.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Lead In move</td>
</tr>
<tr>
<td>B</td>
<td>Lead Out move</td>
</tr>
<tr>
<td>1</td>
<td>Programmed paths</td>
</tr>
<tr>
<td>2</td>
<td>Offset tool path</td>
</tr>
<tr>
<td>3</td>
<td>(-) Lead Angle</td>
</tr>
<tr>
<td>4</td>
<td>(+) Lead Angle</td>
</tr>
<tr>
<td>5</td>
<td>Lead Length</td>
</tr>
</tbody>
</table>

Figure 1–8. Lead In/Out Moves

When you select Milling Type Left, Right, Inside, or Outside, and enable the Display APT fields in Editor field on the Utilities—Post Processor screen, Lead Angle and Lead Length fields appear on the Milling data block.

Lead In/Out moves are determined by the lead angle and lead length values. With the exception of open contours, perform a 90° arc Lead In and Lead Out move.

These NC Programming fields operate similarly to Conversational Blend In/Blend Out moves (Blend Offset and Blend Overlap values).
Lead Angle

Lead Angle is used with Lead Length to define Lead In and Lead Out moves. Lead Angle is an angle relative to the direction of the cut, measured 180° from the end of the first segment.

If starting a contour in the middle of a line or arc, set the Lead Angle at 45° or 90° to prevent gouging the contour.

Use caution with Lead Angle, because the tool could gouge the part if left at the default of 0.

Lead Length

Lead Length is used with Lead Angle to define Lead In and Lead Out moves. Lead Length is the length of the Lead In move.

The Lead Length must be larger than the tool's radius. For example, using a 1" diameter End Mill, the Lead Length would be 0.505".
MILLING OPERATIONS

These topics are discussed in this section:

<table>
<thead>
<tr>
<th>Topic</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Creating a Milling Data Block</td>
<td>2 - 2</td>
</tr>
<tr>
<td>Mill Face</td>
<td>2 - 3</td>
</tr>
<tr>
<td>Circle Data Block</td>
<td>2 - 5</td>
</tr>
<tr>
<td>Ellipse Data Block</td>
<td>2 - 8</td>
</tr>
<tr>
<td>Mill Frame</td>
<td>2 - 10</td>
</tr>
<tr>
<td>Finish Tool for a Mill Frame, Mill Circle or Mill Contour</td>
<td>2 - 14</td>
</tr>
<tr>
<td>Stick Lettering</td>
<td>2 - 15</td>
</tr>
<tr>
<td>True-Type Lettering</td>
<td>2 - 17</td>
</tr>
<tr>
<td>Stick Lettering Along Contour</td>
<td>2 - 21</td>
</tr>
<tr>
<td>True-Type Lettering Along Contour</td>
<td>2 - 24</td>
</tr>
<tr>
<td>Serial Number Stick Lettering</td>
<td>2 - 27</td>
</tr>
<tr>
<td>HD3 Lettering</td>
<td>2 - 33</td>
</tr>
<tr>
<td>Swept Surface</td>
<td>2 - 35</td>
</tr>
<tr>
<td>Mill Slot</td>
<td>2 - 41</td>
</tr>
<tr>
<td>Mill Polygon</td>
<td>2 - 46</td>
</tr>
<tr>
<td>Mill Thread</td>
<td>2 - 49</td>
</tr>
<tr>
<td>Serial Number Stick Lettering</td>
<td>2 - 27</td>
</tr>
<tr>
<td>Lines/Arcs Data Blocks</td>
<td>2 - 53</td>
</tr>
<tr>
<td>Mill Contour</td>
<td>2 - 53</td>
</tr>
<tr>
<td>Contour Line</td>
<td>2 - 56</td>
</tr>
<tr>
<td>Contour Arc</td>
<td>2 - 57</td>
</tr>
<tr>
<td>Contour Blend Arc</td>
<td>2 - 58</td>
</tr>
<tr>
<td>Helix</td>
<td>2 - 60</td>
</tr>
<tr>
<td>3D Arc</td>
<td>2 - 61</td>
</tr>
<tr>
<td>Contour End</td>
<td>2 - 63</td>
</tr>
</tbody>
</table>
Creating a Milling Data Block

1. Begin a Mill Frame block by displaying the Input screen and selecting the **Part Programming** softkey or icon. Refer to *Mill Frame, on page 2 - 10*.
2. Select the **Insert Block Before** softkey.
3. When the New Block screen appears, select the **Milling** softkey to display the milling options shown in the seven softkey selections.
4. Select the milling operation and the first data block screen appears. Use **Frame** for this example.
5. Type in the number of the tool that will be used to mill the frame and press the ENTER key to tell the system to read and remember the information in the field.
   - The tool numbers and their descriptions are entered in Tool Setup. If the tool information has not been entered or another tool should be added to the program, select the **Tool Setup** softkey to update the tool descriptions for this program.
   - Entering a number in the Tool field pulls information from the tool setup and automatically loads it into fields on this screen. Note that the Mill Feed and Speed (RPM) fields are filled in based on the information about the tool. For additional assistance to the operator, the system also displays the selected tool's diameter and type as read-only fields.

The fields describe how and where the frame will be milled on the work piece. For a machine equipped with the UltiPocket feature, select a pocket boundary with an inward cutter motion. The Pocket Type field appears when a pocket operation is selected. Many WinMax screens have fields that appear based on selections made in other fields. These are *conditional fields* because they only appear on a screen under certain conditions.

The X and Y Corner fields define a *reference corner* from which the rectangular frame will be milled. The X and Y Length fields define the lengths of the sides of the rectangle.

The fields on the Mill Frame screen are described in detail in Frame. Refer to *Mill Frame, on page 2 - 10*. 
Mill Face

Face milling directs the cutter path so the tool moves over the rectangular face area of the part, removing material using 60% of the tool diameter from the previous pass (but the last pass may be less).

Axis positioning places the tool over the next cutting path, and cutting resumes in the opposite direction.

The starting point is either the lower left or right corner of the rectangular area, depending on the milling direction defined in Program Parameters or Change Parameters. Refer to Getting Started with WinMax Mill for more information.

![Figure 2–1. Face Milling](image)

To access the Mill Face screen, on the New Block screen, select the **Milling** softkey then select the **Face** softkey that appears.

The Face Milling fields are defined as follows:

- **Block**—Identifies the block number for this operation. The system determines the number by the position of this data block in the program.
- **Tool**—Identifies the tool number for this data block and enters that tool's Diameter and Tool Type on this screen.
- **Milling Type**—Defines the type of cutting operation. There are four types of face milling:
- **X Unidirectional**—Directs the tool to cut in one direction, parallel to the X axis, and then move the spindle to the retract clearance value. Axis movement occurs at rapid traverse to return the tool to the start point of the next cutting path, which is determined by the system. The start point for each cut is the tool radius plus blend offset away from the starting edge corner.

- **X Bi-directional**—Directs the tool to cut in one direction from the start point, parallel to the X axis. Axis positioning occurs to place the tool over the next cutting path, and cutting resumes in the opposite direction. The new cutting path for the tool is automatically determined by the system.

- **Y Unidirectional**—Directs the tool to cut in one direction, parallel to the Y axis, and then move the spindle to the retract clearance value. Axis movement occurs at rapid traverse to return the tool to the start point of the next cutting path, which is determined by the system. The start point for each cut is the tool radius plus blend offset away from the starting edge corner.

- **Y Bi-directional**—Directs the tool to cut in one direction from the start point, parallel to the Y axis. Axis positioning occurs to place the tool over the next cutting path and cutting is in the opposite direction. The new cutting path for the tool is automatically determined by the system.

- **X Corner** and **Y Corner**—Identify the X and Y coordinates of any one of the four corners of the face which then becomes the reference corner.

- **X Length** and **Y Length**—The distance and direction of the rectangle along the X axis and Y axis, measured from the reference corner.
  - If the reference corner is at the left side of the rectangle, the X Length is a positive (+) dimension. If the reference corner is at the right side, the X Length is negative (-).
  - The Y Length value is positive (+) if the reference corner is at the lower left or lower right of the rectangular area. It is negative (-) if the reference corner is at the top left or top right of the rectangle.

- **Z Start**—The point where Plunge Feed rate begins for the tool to move down to Z Bottom.

- **Z Bottom**—Identifies the point where the mill feed rate begins.

- **Peck Depth**—Defines the maximum depth to be cut in one pass. If the total depth is greater than this value, multiple cutting passes occur. Entering a zero (0) value causes the total programmed depth to be cut in one pass of the tool.

- **Plunge Feed**—Defines the feed rate for the tool moving from Z Start to Z Bottom.

- **Mill Feed**—Defines the X-Y feedrate. The value initially displayed has been calculated by the control and can be retained or changed to a different value. See *Getting Started with WinMax Mill* for more information.

- **Speed (RPM)**—Determines the spindle speed for the tool calculated in Tool Setup. Entering a value here overrides the Tool Setup value for this data block. See *Getting Started with WinMax Mill* for more information.

- **Tool Diameter**—Contains the diameter entered during Tool Setup. See *Getting Started with WinMax Mill* for more information.
• **Tool Type**—Contains the type entered during Tool Setup. See *Getting Started with WinMax Mill* for more information.

• **Finish Diameter**—Contains the diameter entered during Tool Setup. See *Getting Started with WinMax Mill* for more information.

• **Finish Type**—Contains the type entered during Tool Setup. See *Getting Started with WinMax Mill* for more information.

# Circle Data Block

Use the Circle data block to mill circles.

To create a Circle data block:

1. From the Input screen, select the **Part Programming** softkey or icon.
2. Select the **Insert Block Before** softkey.
3. When the New Block screen appears, select the **Milling** softkey.

From the Milling softkeys, select **Circle**.

Follow this link to the Hurco website to view a Mill Circle video demonstration:

[Video Demonstrations at www.hurco.com](http://www.hurco.com)

Scroll to the bottom of the page and select the video to download. Internet connection is required.

The Mill Circle fields are defined as follows:

• **Block**—Identifies the block number for this operation. The system determines the number by the position of this data block in the program.

• **Tool**—Identifies the tool number for this data block and enters that tool's diameter and type on this screen.

• **Finish Tool**—Identifies the tool number used for finish pass and enters that tool's Tool Diameter and Tool Type on this screen. If a finish tool is selected, the first tool leaves an amount of material selected in the Finish Z field in Milling Parameters. The finish tool removes the remaining stock down to the programmed dimension.

• **Milling Type**—Identifies whether the system should automatically compensate for the diameter of the tool when determining the tool path. With the cursor on the Milling Type field, select different types of Cutter Compensation and pocket type with the softkeys and pull down menu choice. Refer to *Cutter Compensation, on page 1 - 13*.

• **Pocket Type**—Appears when Pocket Boundary or Pocket Island is chosen for Milling Type and defines whether spindle movement is from inside the pocket or outside the pocket.
- **X Center** and **Y Center**—Identify the X and Y coordinates for the center point of the circle.

- **Radius**—Defines (in conjunction with the X and Y center coordinates) the location and size of the circle.

- **Lead Length**—Identifies the Lead Length for this Mill Circle. See *Lead In/Out Moves, on page 1 - 20*. This field only appears for NC posted programs. This field is only available when Milling Type is set to Inside or Outside. Lead In/Out Moves will be ignored in the post unless Enable Blend Moves is set to Yes and the Display Apt Fields in Editor field on the Post Processor Configuration screen is set to Yes.

- **Lead Angle**—Identifies the Lead Angle for this Mill Circle. See *Lead In/Out Moves, on page 1 - 20*. This field only appears for NC posted programs. This field is only available when Milling Type is set to Inside or Outside. Lead In/Out Moves will be ignored in the post unless Enable Blend Moves is set to Yes and the Display Apt Fields in Editor field on the Post Processor Configuration screen is set to Yes.

- **Z Start**—The point where Plunge Feed rate begins for the tool to move down to Z Bottom.

- **Z Bottom**—Identifies the point where the mill feed rate begins.

- **Peck Depth**—Defines the maximum depth to be cut in one pass. If the total depth is greater than this value, multiple cutting passes occur. Entering a zero (0) value causes the total programmed depth to be cut in one pass of the tool.

- **Plunge Feed**—Defines the feed rate for the tool moving from Z Start to Z Bottom.

- **Mill Feed**—Identifies the X-Y feedrate. The value initially displayed has been calculated by the control and can be retained or changed to a different value. See *Getting Started with WinMax Mill* for more information.

- **Speed (RPM)**—Identifies the spindle speed for the tool calculated in Tool Setup, on page 1 - 84. Entering a value here overrides the Tool Setup value for this data block. See *Getting Started with WinMax Mill* for more information.

- **Pocket Overlap**—appears when **Pocket Boundary** Milling Type is selected. Pocket Overlap is the percentage of tool diameter that overlaps for each pass in a pocket milling operation.

If the overlap is less than 50%, the toolpath may make additional moves to clean out the pocket. For example (toolpath is red):
• **Tool Diameter**—Contains the diameter entered during Tool Setup. See *Getting Started with WinMax Mill* for more information.

• **Tool Type**—Contains the type entered during Tool Setup. See *Getting Started with WinMax Mill* for more information.

• **Finish Diameter**—Contains the diameter entered during Tool Setup. See *Getting Started with WinMax Mill* for more information.

• **Finish Type**—Contains the type entered during Tool Setup. See *Getting Started with WinMax Mill* for more information.

If you change the Pocket Overlap using this field and save the file as an HD3 file, a Change Parameters data block will be inserted into the HD3 file to make the Pocket Overlap change. The Change Parameters block changes only the Pocket Overlap, leaving all other parameters as they were.
**Ellipse Data Block**

To access the Mill Ellipse screen on the New Block screen, select the **Milling** softkey then select the **Ellipse** softkey that appears. Use this function to mill an ellipse.

Follow this link to the Hurco website to view a Ellipse programming video demonstration:

[Video Demonstrations at www.hurco.com](http://www.hurco.com)

Scroll to the bottom of the page and select the video to download. Internet connection is required.

The Ellipse fields are defined as follows:

- **Block**—Identifies the block number for this operation. The system determines the number by the position of this data block in the program.

- **Tool**—Identifies the tool number for this data block and enters that tool's diameter and type on this screen. If a finish tool is selected, the first tool leaves an amount of material selected in the Finish Z field in Milling Parameters. The finish tool removes the remaining stock down to the programmed dimension.

- **Finish Tool**—Identifies the finish tool number and enters that tool's diameter and type on this screen.

- **Milling Type**—Identifies whether the system should automatically compensate for the diameter of the tool when determining the tool path. With the cursor in the Milling Type field, select different types of Cutter Compensation and pocket type with the softkeys and pull down menu. Refer to *Cutter Compensation, on page 1 - 13*.

- **Pocket Type**—Appears when Pocket Boundary or Pocket Island is chosen for Milling Type and defines whether spindle movement is from inside the pocket or outside the pocket.

- **X Center** and **Y Center**—Identify the X and Y coordinates for the center point of the ellipse.

- **X Radius** and **Y Radius**—The distance along the X axis from X Center, or along the Y axis from Y Center, to the edge of the ellipse. Ellipse axes must be programmed parallel to the respective axes of the machine.

The major axis is the longer or wider axis and can be either the X or Y axis. The minor axis is the smaller axis. Program an ellipse with the major and minor axes non-parallel to the machine using a Pattern Loop Rotate routine and rotating the ellipse after it has been programmed. Refer to *Loop Rotate, on page 4 - 5*.

- **Z Start**—The point where Plunge Feed rate begins for the tool to move down to Z Bottom.

- **Z Bottom**—Identifies the point where the mill feed rate begins.

- **Peck Depth**—Defines the maximum depth to be cut in one pass. If the total
depth is greater than this value, multiple cutting passes occur. Entering a zero (0) value causes the total programmed depth to be cut in one pass of the tool.

- **Plunge Feed**—Defines the feed rate for the tool moving from Z Start to Z Bottom.

- **Mill Feed**—Identifies the X-Y feedrate. The value initially displayed has been calculated by the system and can be retained or changed to a different value. See *Getting Started with WinMax Mill* for more information.

- **Speed (RPM)**—Identifies the spindle speed for the tool calculated in Tool Setup. Entering a value here overrides the Tool Setup value for this data block. See *Getting Started with WinMax Mill* for more information.

- **Pocket Overlap**—Appears when Pocket Boundary Milling Type is selected. Pocket Overlap is the percentage of tool diameter that overlaps for each pass in a pocket milling operation.

  If the overlap is less than 50%, the toolpath may make additional moves to clean out the pocket. For example (toolpath is red):

  ![Additional corner moves for Pocket Overlap of 25%](image)

  If you change the Pocket Overlap using this field and save the file as an HD3 file, a Change Parameters data block will be inserted into the HD3 file to make the Pocket Overlap change. The Change Parameters block changes only the Pocket Overlap, leaving all other parameters as they were.

- **Tool Diameter**—Contains the value entered during Tool Setup. See *Getting Started with WinMax Mill* for more information.

- **Tool Type**—Contains the type entered during Tool Setup. See *Getting Started with WinMax Mill* for more information.

- **Finish Diameter**—Contains the value entered during Tool Setup. See *Getting Started with WinMax Mill* for more information.

- **Finish Type**—Contains the type entered during Tool Setup. See *Getting Started with WinMax Mill* for more information.
Mill Frame

Mill Frame data block supports the creation of frames with or without uniform corners. To access the Mill Frame screen, on the New Block screen select the Milling softkey then select the Frame softkey that appears.

Follow this link to the Hurco website to view a Mill Frame programming video demonstration:

[Video Demonstrations at www.hurco.com](http://www.hurco.com)

Scroll to the bottom of the page and select the video to download. Internet connection is required.

The Mill Frame block contains fields on two tabs: Geometry and Corners. The geometry parameters are entered on the Geometry tab:

![Figure 2–2. Mill Frame Geometry tab](image)

These are the Geometry field definitions:

- **Block**—Identifies the block number for this operation. The system determines the number by the position of this data block in the program.
- **X Corner** and **Y Corner**—Identifies the X and Y coordinates of any one of the four corners of the frame which then becomes the reference corner.
- **X Length** and **Y Length**—The length of the rectangle along the X axis or Y axis, measured from the reference corner. This value is positive or negative relative to the reference corner. If the reference corner is at the left side of the rectangle, the X or Y Length is a positive (+) dimension. If the reference corner is at the right side, this field's value is negative (-).
- **Z Start**—point where Plunge Feed rate begins, moving the tool to Z Bottom.
- **Z Bottom**—Identifies the point where the mill feed rate begins.
- **Corner Radius**—Identifies the radius of the reference corner. Use this field if all fours corners of the frame will have the same radius. To program unique corners (with different radii), do not set a value here; instead, set the parameters on the Corners tab.
Each corner of a frame can be programmed with a radius or chamfer of a different size, on the Corners tab:

![Corner Radius Field](image)

**Figure 2–3. Mill Frame Corners tab**

Select **Line** or **Arc** for each corner:

- Specify radius for arcs.
- Specify length and angle for lines.

If the corner should have neither, the geometry should be left as an arc with radius of 0.00, which is the default.

Here is an example of a Frame using the corner measurements above:

![Frame Example](image)

**Figure 2–4. Mill Frame Corners example**

Specifying a value in the Corner Radius field will reset all of the corners set as Arcs in the Corners tab to that radius. If you are creating unique corners, do not use this Corner Radius field.
Roughing and Finishing tool parameters are:

- **Tool**—Identifies the tool(s) to be used for roughing and finishing in this data block.
- **Milling Type**—Identifies whether the system should automatically compensate for the diameter of the tool when determining the tool path. With the cursor in the Milling Type field, select different types of Cutter Compensation or pocket type with the softkeys and pull-down menu. Refer to *Cutter Compensation, on page 1 - 13*
- **Pocket Type**—Appears when Pocket Boundary is chosen for Milling Type and defines whether spindle movement is from inside the pocket or outside the pocket.
- **Pocket Overlap**—Appears when **Pocket Boundary** Milling Type is selected. Pocket Overlap is the percentage of tool diameter that overlaps for each pass in a pocket milling operation.

If the overlap is less than 50%, the toolpath may make additional moves to clean out the pocket. For example (toolpath is red):

Additional corner moves for Pocket Overlap of 25%

If you change the Pocket Overlap using this field and save the file as an HD3 file, a Change Parameters data block will be inserted into the HD3 file to make the Pocket Overlap change. The Change Parameters block changes only the Pocket Overlap, leaving all other parameters as they were.

- **Mill Feed**—The X-Y feedrate. The value initially displayed has been calculated by the control and can be retained or changed to a different value. See *Getting Started with WinMax Mill* for more information.
- **Peck Depth**—Defines the maximum depth to be cut in one pass. If the total depth is greater than this value, multiple cutting passes occur. Entering a zero (0) value causes the total programmed depth to be cut in one pass of the tool.
- **Speed (RPM)**—The spindle speed for the tool calculated in Tool Setup.
Entering a value here overrides the Tool Setup value for this data block. See *Getting Started with WinMax Mill* for more information.

- **Plunge Feed**—Defines the feed rate for the tool moving from Z Start to Z Bottom.
Finish Tool for a Mill Frame, Mill Circle or Mill Contour

When a finish tool is used in a Mill Frame data block, and its Speed (on the Tool Setup screen) and Surface Speed (on the Milling Parameters screen) values are zero (0):

1. If the finish tool is the same as the Tool specified in Part Setup, the finish speed and finish feed is calculated as a percentage of the Speed (RPM) and Feed (IPM or MMPM) values in the Part Program data block. The new values used are determined by percentages set for the Finish Speed (%) and Finish Feed (%) in the Milling Parameters screen.

   For example, if the Speed (RPM) value for the data block is 2000 and the Finish Speed (%) value is 130, the calculated finish speed is: 2000 x .130 = 2600.

2. If the finish tool is different from the Tool specified in Part Setup:
   - If the finish tool is a ball end mill, face mill, or flat end mill, the Spindle speed value entered on the Tool Setup screen will be used as the finish speed.
   - If the finish tool is a ball end mill, face mill or flat end mill, and a Spindle speed value was not entered on the Tool Setup screen, WinMax attempts to calculate the finish speed based on the tool's diameter, surface speed, number of flutes and chip load. If the calculation can not be made, the finish speed is calculated as a percentage of the Speed RPM value (same as "1" above).

3. If the finish tool is a ball end mill, face mill or flat end mill, WinMax will attempt to calculate the finish feed using the tool’s diameter, surface speed, number of flutes and chip load. If the calculation can not be made, the finish feed is calculated by multiplying the Mill Feed and the Finish Feed (%) values on the Milling Parameters screen.

   When importing an older part program into WinMax, you may need to clear the Speed (RPM) and/or Surface Speed values for the data block to run as anticipated.
Stick Lettering

The Stick Lettering block supports the HD3 lettering (military text) character set. Text is sized by specifying total length and height. The block also includes an orientation angle.

<table>
<thead>
<tr>
<th>BLOCK</th>
<th>MILL STICK LETTERING</th>
</tr>
</thead>
<tbody>
<tr>
<td>X REF LOCATION</td>
<td>START</td>
</tr>
<tr>
<td>X REFERENCE</td>
<td>5.000</td>
</tr>
<tr>
<td>Y REF LOCATION</td>
<td>BOTTOM</td>
</tr>
<tr>
<td>Y REFERENCE</td>
<td>2.000</td>
</tr>
<tr>
<td>X LENGTH</td>
<td>15.000</td>
</tr>
<tr>
<td>Y LENGTH</td>
<td>1.000</td>
</tr>
<tr>
<td>TEXT</td>
<td>HURCO</td>
</tr>
</tbody>
</table>

- **X Ref Location**—specifies Start, End, or Center of the character for the X reference point. Default is Start.
  - Start—the lettering is left-justified; the first character begins at the X reference point.
  - Center—the lettering is centered left and right of the X reference point.
  - End—the lettering is right-justified; the last character ends at the X reference point.
- **X Reference**—indicates the X coordinate of the point where text begins/ends, relative to the X Ref Location.
- **Y Ref Location**—specifies Bottom, Top, or Center of the character for the Y reference point. Default is Bottom.
  - Bottom—the bottom of the letter is aligned with the Y reference point.
  - Top—the top of the letter is aligned with the Y reference point.
• **Center**—the letter is centered above and below the Y reference point.

• **Y Reference**—indicates the Y coordinate of the point where text is aligned relative to the Y Ref Location.

![Y Reference Diagram](image)

**Figure 2–5. X Reference and Y Reference locations**

• **X Length**—X axis length of the Sizing Box.

• **Y Length**—Y axis length of the Sizing Box.

• **Text**—the text to be milled.

• **Z Start**—the Z coordinate at which plunge feedrate begins.

• **Z Bottom**—Z Bottom coordinate.

• **Orientation**—changes the angle of the text. The center of rotation is located at the reference point. Leaving the field at zero specifies no rotation. A negative value will angle the end of the text string in an upward direction; a positive value will angle the end of the text string in a downward direction.

![X Reference Diagram](image)

**Recovery Restart** is supported from a specific letter, specified as character number. For example, to restart from the "R" in HURCO, you would specify character 3 as the starting character. Recovery Restart is launched from the Auto mode screen.
True-Type Lettering

The Mill True-Type Lettering block mills True-Type fonts. These True-Type fonts are available in different styles and are block-type letters with closed profiles for milling, which allow cutter compensation - i.e., inside, outside, and pocket milling. Supported characters include the full character sets for most Western European languages.

<table>
<thead>
<tr>
<th>BLOCK</th>
<th>MILL TRUE-TYPE LETTERING</th>
</tr>
</thead>
<tbody>
<tr>
<td>X REF LOCATION</td>
<td>START</td>
</tr>
<tr>
<td>1.000</td>
<td>Z BOTTOM</td>
</tr>
<tr>
<td>Y REF LOCATION</td>
<td>BOTTOM</td>
</tr>
<tr>
<td>1.000</td>
<td>MAPPING</td>
</tr>
<tr>
<td>X LENGTH</td>
<td>5.000</td>
</tr>
<tr>
<td>Y LENGTH</td>
<td>1.000</td>
</tr>
<tr>
<td>TEXT</td>
<td>Hurco</td>
</tr>
</tbody>
</table>

Figure 2–6. Example of True-Type Lettering

Mill True-Type Lettering fields are:

- **X Ref Location**—specifies Start, End, or Center of the character for the X reference point. Default is Start.
  - Start—the lettering is left-justified; the first character begins at the X reference point.
  - Center—the lettering is centered left and right of the X reference point.
  - End—the lettering is right-justified; the last character ends at the X reference point.
- **X Reference**—indicates the X coordinate of the point where text begins/ends, relative to the X Ref Location.
• **Y Ref Location**—specifies Bottom, Top, or Center of the character for the Y reference point. Default is Bottom.
  
  • Bottom—the bottom of the letter is aligned with the Y reference point.
  • Top—the top of the letter is aligned with the Y reference point.
  • Center—the letter is centered above and below the Y reference point.

• **Y Reference**—indicates the Y coordinate of the point where text is aligned relative to the Y Ref Location.

![Diagram of Y Reference](image)

**Figure 2–7. X Reference and Y Reference locations**

• **X Length**—X axis length of the Sizing Box.

• **Y Length**—Y axis length of the Sizing Box.

• **Z Start**—the Z coordinate at which plunge feedrate begins.

• **Z Bottom**—Z Bottom coordinate.

• **Orientation**—changes the angle of the text. The center of rotation is located at the reference point. Leaving the field at zero specifies no rotation. A negative value will angle the end of the text string in an upward direction; a positive value will angle the end of the text string in a downward direction.

• **Mapping**—specifies the size of the character in Y:
  
  • Body Only—defines the height of the body of the lettering. Ascenders and descenders are calculated automatically.

  • Descended—defines the height of the body and descenders (the portion of some characters that extends below the baseline) of the lettering. Ascenders are calculated automatically.

  • Ascended—defines the height of the body and ascenders (the portion of some characters that extends above the x-height) of the lettering. Descenders are calculated automatically.

  • Full Font—defines the height of the body, ascenders, and descenders of the lettering. No part of the lettering is calculated automatically.
Font—the lettering font. The font selection dialog box is opened with the Select New Font F3 softkey, when the cursor is in the Text field.

<table>
<thead>
<tr>
<th>Common Fonts Available with Mill True-Type Lettering</th>
</tr>
</thead>
<tbody>
<tr>
<td>Arial (default)</td>
</tr>
<tr>
<td>Arial Black</td>
</tr>
<tr>
<td>Comic Sans MS</td>
</tr>
<tr>
<td>Courier New</td>
</tr>
<tr>
<td>Franklin Gothic Medium</td>
</tr>
<tr>
<td>Georgia</td>
</tr>
<tr>
<td>Impact</td>
</tr>
<tr>
<td>Lucida Console</td>
</tr>
<tr>
<td>Lucida Sans Unicode</td>
</tr>
<tr>
<td>Marlett</td>
</tr>
</tbody>
</table>

- **Text**—the text to be milled.
- **Tool**—Identifies the tool number for this data block and enters that tool's diameter and type on this screen.
- **Peck Depth**—Defines the maximum depth to be cut in one pass of the tool. If the total depth is greater than this value, multiple tool passes occur. Entering the value zero (0) causes the total programmed depth to be cut in one pass of the tool.
- **Plunge Feed**—Defines the feed rate for the tool moving from Z Start to Z Bottom.
- **Mill Feed**—The X-Y feedrate. See Getting Started with WinMax Mill for more information.
- **Speed (RPM)**—Defines the spindle speed for the tool calculated in Tool Setup. Entering a value here overrides the Tool Setup value for this data block. See Getting Started with WinMax Mill for more information.
- **Pocket Overlap**—Appears when **Pocket Boundary** Milling Type is selected. Pocket Overlap is the percentage of tool diameter that overlaps for each pass in a pocket milling operation.

  If the overlap is less than 50%, the toolpath may make additional moves to clean out the pocket. For example (toolpath is red):

  ![Additional corner moves for Pocket Overlap of 25%](image-url)
Stick Lettering Along Contour

With **Stick Lettering Along Contour**, the text is oriented along a programmed contour, which determines the size and orientation. The contour is grouped with the data block and programmed like the contours in a 3D mold or Swept Surface data block.

First, specify how the width of the text is determined, in **the Width Method** field.

- **Use Contour**—the text will be distributed evenly along the contour.
- **Specify Width**—specify the width measurement in the Text Width field (appears when Specify Width is selected).

The **Text Width**, **Text Height**, and **Character Spacing** is set in this screen.

- **Text Width**—field appears when Specify Width is selected as Width Method. Specify the total width of the text along the contour.
- **Text Height**—specifies the height of the text.
- **Char Spacing**—specifies the amount of additional space to add between each character. Leaving at zero will use the default character spacing for the selected font.
- **Z Start**—specifies the Z coordinate at which plunge feedrate begins.
- **Z Bottom**—specifies the Z Bottom coordinate.
- **Text**—specifies the text to be milled.

![Figure 2–9. Stick Lettering Along Contour Lettering Parameters screen](image)

<table>
<thead>
<tr>
<th>BLOCK</th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td>STICK LETTERING ALONG CONTOUR</td>
<td></td>
</tr>
<tr>
<td>WIDTH METHOD</td>
<td>USE CONTOUR</td>
</tr>
<tr>
<td>Z START</td>
<td>1.0000</td>
</tr>
<tr>
<td>Z BOTTOM</td>
<td>1.0000</td>
</tr>
<tr>
<td>TEXT HEIGHT</td>
<td>2.0000</td>
</tr>
<tr>
<td>CHAR SPACING</td>
<td>0.0000</td>
</tr>
<tr>
<td>TEXT</td>
<td>HURCO</td>
</tr>
</tbody>
</table>
The contour start segment includes fields to specify which side of the contour the text is positioned relative to. The text offset specifies distance between the contour and the reference point of the text.

Use **Edit Along Contour** softkey to open the Start segment:

![Figure 2–10. Stick Lettering Along Contour Start Segment](image)

The contour start segment includes fields to specify which side of the contour the text is positioned relative to. The text offset specifies distance between the contour and the reference point of the text.

Select the **Next Segment** softkey to lay the text out on an arc by programming a contour with a single arc segment:

![Figure 2–11. Stick Lettering Along Contour Arc Segment](image)
Recovery Restart is supported from a specific letter, specified as character number. For example, to restart from the "R" in HURCO, you would specify character 3 as the starting character. Recovery Restart is launched from the Auto mode screen.
True-Type Lettering Along Contour

With True-Type Lettering Along Contour, the text is oriented along a programmed contour, which determines the size and orientation. The contour is grouped with the data block and programmed like the contours in a 3D mold or Swept Surface data block.

![Image of True-Type Lettering Along Contour](image)

**Figure 2–13. True Type Lettering Along Contour opening screen**

First, program the text specifications on the Along Contour opening screen:

- **Width Method**—specifies the method to be used to determine the width of the text:
  - **Use Contour**—the text will be distributed evenly along the contour.
  - **Specify Width**—specify the width measurement in the Text Width field (appears when Specify Width is selected).

- **Text Width**—field appears when Specify Width is selected as Width Method. Specify the total width of the text along the contour.

- **Text Height**—specifies the height of the text.

- **Char Spacing**—specifies the amount of additional space to add between each character. Leaving at zero will use the default character spacing for the selected font.

- **Blank Spacing**—specifies the size of the blank character (space). Leave at zero to use the default spacing for the selected font.

- **Z Start**—specifies the Z coordinate at which plunge feedrate begins.

- **Z Bottom**—specifies the Z Bottom coordinate.

- **Mapping**—specifies the size of the character in Y:
  - **Body Only**—defines the height of the body of the lettering. Ascenders and descenders are calculated automatically.
  - **Descended**—defines the height of the body and descenders (the portion of some characters that extends below the baseline) of the lettering. Ascenders are calculated automatically.
  - **Ascended**—defines the height of the body and ascenders (the portion of some characters that extends above the x-height) of the lettering. Descenders are calculated automatically.
  - **Full Font**—defines the height of the body, ascenders, and descenders of
the lettering. No part of the lettering is calculated automatically.

![Figure 2–14. Mapping](image)

- **Font**—the lettering font. The font selection dialog box is opened with the **Select New Font F3** softkey, when the cursor is in the Text field.

<table>
<thead>
<tr>
<th>Common Fonts Available with Mill True-Type Lettering</th>
</tr>
</thead>
<tbody>
<tr>
<td>Arial (default)</td>
</tr>
<tr>
<td>Arial Black</td>
</tr>
<tr>
<td>Comic Sans MS</td>
</tr>
<tr>
<td>Courier New</td>
</tr>
<tr>
<td>Franklin Gothic Medium</td>
</tr>
<tr>
<td>Georgia</td>
</tr>
<tr>
<td>Impact</td>
</tr>
<tr>
<td>Lucida Console</td>
</tr>
<tr>
<td>Lucida Sans Unicode</td>
</tr>
<tr>
<td>Marlett</td>
</tr>
</tbody>
</table>

- **Text**—the text to be milled.

Use the **Edit Along Contour** softkey to open the Start segment:

![Figure 2–15. True Type Lettering Along Contour Start Segment](image)
The contour start segment includes fields to specify which side of the contour the text is positioned relative to. The text offset specifies distance between the contour and the reference point of the text.

Select the **Next Segment** softkey to lay the text out on a line or arc:

<table>
<thead>
<tr>
<th>BLOCK</th>
<th>LETTERING ALONG CONTOUR</th>
</tr>
</thead>
<tbody>
<tr>
<td>SEGMENT</td>
<td>ARC</td>
</tr>
<tr>
<td>DIRECTION</td>
<td>CH</td>
</tr>
<tr>
<td>X END</td>
<td>CAL</td>
</tr>
<tr>
<td>Y END</td>
<td>2.000</td>
</tr>
<tr>
<td>X CENTER</td>
<td>CAL</td>
</tr>
<tr>
<td>Y CENTER</td>
<td>0.000</td>
</tr>
</tbody>
</table>

*Figure 2–16. True Type Lettering Along Contour Arc Segment*

*Figure 2–17. Example of True-Type Lettering Along Contour*

**Recovery Restart** is supported from a specific letter, specified as character number. For example, to restart from the “R” in HURCO, you would specify character 3 as the starting character. Recovery Restart is launched from the Auto mode screen.
Serial Number Stick Lettering

HD3 Serial Number Lettering mills a sequence of unique serial numbers that increment each time the block is run. The characters are HD3-style (non-true type) and can be alphabetic A-Z, numeric 0-9, and special characters @#!%^&*()-[]. Alpha characters are all upper case. The last character in the serial number must be numeric and is incremented by a specified amount.

- Users can manually provide serial number data, which includes a starting sequence, such as 0001 or ABC001, and an increment which advances the serial number for each subsequent run.
- Data can originate from a file in which the serial numbers are listed. Each time the block is run, the next serial number is extracted from the file. The format of this file must be one serial number per line, each line must end in a carriage return, line feed sequence, or Return key.

To create an HD3 Serial Number Lettering block from the New Block Screen:

1. Select the Milling softkey.
2. Select the More softkey twice.
3. Select the HD3 Serial Number Lettering softkey on New Block screen 3.

The HD3 Serial Number Lettering screen uses three tabs: Text, Orientation, and Format.

Text tab

The text of the serial number is set up in the Text tab:

![Image](image)

Figure 2–18. HD3 Serial Number Lettering Text tab

Fields are:

- **X Reference**—indicates the X coordinate of the point where text begins.
• **Y Reference**—indicates the Y coordinate of the point where text begins.

• **Character Height**—indicates the height of the text.

• **Character Width**—indicates the character width. The width includes the character left-justified, plus some spacing between it and the next character to be milled. Spacing between the characters is equal to the tool's diameter.

  Because cutter compensation is not used in this routine and letter contouring follows the center of the tool, character spacing can be adjusted by adjusting the tool diameter in Tool Setup.

• **Z Start**—indicates the point where the Z-axis plunge feedrate begins.

• **Z Bottom**—indicates the location where the Z-axis plunge feedrate ends and mill feed begins.

• **Data Origin**—specifies the source of the serial number data, either User-provided or From File. Default is User-Provided.
  - **User-provided**—the serial numbers are constructed with information provided by the user. When selected, the Starting Sequence and Increment fields appear.
  - **From File**—the serial numbers are populated from a source file. When selected, the File field and Locate... button appear.

• **Starting Sequence**—specifies the characters for the first serial number. The sequence can contain any combination of alpha or numeric characters, but the last character in the sequence must be a digit (number). This field appears only when User-provided Data Origin is selected. Default is 1.

• **Increment**—specifies the amount to increment for each new serial number. This field appears only when User-provided Data Origin is selected. Default is 1.

• **File**—specifies the source file for serial numbers that are generated from a file. This field appears only when From File Data Origin is selected.

The file must be located on the hard drive or an accessible network drive, or can be on a floppy disk or USB memory device that has been loaded onto the control. To select the file:

1. Select the Locate... button to browse for the file.
2. Navigate to the location of the file in the dialog box that opens.
3. Select the file.
4. Select the Open button on the dialog box. The file and path appear in the File field.

**Orientation tab**

The Orientation fields specify the orientation of the X and Y reference points.
Figure 2–19. Serial Number Orientation tab

Fields are:

- **X Ref Location**—specifies Start, End, or Center of the character for the X reference point. Default is Start.
  - Start—the serial number is left-justified; the first character begins at the X reference point.
  - Center—the entire serial number is centered left and right of the X reference point.
  - End—the serial number is right-justified; the last character ends at the X reference point.

- **Y Ref Location**—specifies Bottom, Top, or Center of the character for the Y reference point. Default is Bottom.
  - Bottom—the bottom of the serial number is aligned with the Y reference point.
  - Top—the top of the serial number is aligned with the Y reference point.
  - Center—the serial number is centered above and below the Y reference point.
Format tab

The Format fields indicate whether the serial number should have a minimum length and what that length should be.

<table>
<thead>
<tr>
<th>TEXT</th>
<th>ORIENTATION</th>
<th>FORMAT</th>
<th>HISTORY</th>
</tr>
</thead>
<tbody>
<tr>
<td>ASSURE MINIMUM LENGTH</td>
<td></td>
<td>YES</td>
<td></td>
</tr>
<tr>
<td>MINIMUM LENGTH</td>
<td></td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>LEADING SYMBOL</td>
<td></td>
<td>0</td>
<td></td>
</tr>
</tbody>
</table>

Figure 2–21. Serial Number Format tab

Fields are:

- **Assure Minimum Length**—specifies Yes or No. When Yes is selected, the Minimum Length and Leading Symbol fields appear so that a placeholder can be used to achieve consistent serial number length. Default is Yes.

- **Minimum Length**—specifies the length (in number of characters) when Assure Minimum Length field is set to Yes. This field only appears when Assure Minimum Length is Yes.

- **Leading Symbol**—specifies the character that is added to the beginning of the serial number as a placeholder to achieve the minimum length. This can be an alpha or a numeric character. This character will be repeated in the serial number as many times as necessary to achieve the specified minimum length with the given starting sequence. See example below. This field only appears when Assure Minimum Length is Yes.

Leading Symbol Example:
<table>
<thead>
<tr>
<th>Starting Sequence</th>
<th>001</th>
</tr>
</thead>
<tbody>
<tr>
<td>Increment</td>
<td>1</td>
</tr>
<tr>
<td>Assure Minimum Length</td>
<td>Yes</td>
</tr>
<tr>
<td>Minimum Length</td>
<td>5</td>
</tr>
<tr>
<td>Leading Symbol</td>
<td>A</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>First Serial Number will be</th>
<th>AA001</th>
<th>NOTE: A second &quot;A&quot; is added to starting sequence of 001 to assure the minimum length of 5.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Second Serial Number will be</td>
<td>AA002</td>
<td></td>
</tr>
<tr>
<td>999th Serial Number will be</td>
<td>AA999</td>
<td></td>
</tr>
<tr>
<td>1000th Serial Number will be</td>
<td>A1000</td>
<td>NOTE: The second &quot;A&quot; is no longer needed to assure minimum length. If serial number &quot;AA1000&quot; is required, the Starting Sequence would be AA001 and Assure Minimum Length would be set to No.</td>
</tr>
</tbody>
</table>

Format tab only assures a **minimum** number of characters. The serial number will continue to increment and expand with continuous running of the block.
History tab

The History tab displays the last used and next serial numbers and contains the Reset Numbering button:

<table>
<thead>
<tr>
<th>TEXT</th>
<th>ORIENTATION</th>
<th>FORMAT</th>
<th>HISTORY</th>
</tr>
</thead>
<tbody>
<tr>
<td>LAST USED SERIAL NUMBER:</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>NEXT SERIAL NUMBER:</td>
<td></td>
<td>A1000</td>
<td></td>
</tr>
<tr>
<td>RESET NUMBERING</td>
<td>Reset</td>
<td>Numbering</td>
<td></td>
</tr>
</tbody>
</table>

*Figure 2–22. Serial Number History tab*

Fields are:

- **Last Used Serial Number**—displays the most recently milled serial number.
- **Next Serial Number**—displays the serial number that will be milled on the next cycle.

  The Last Used and Next Serial Numbers are saved in the part program after each cycle so milling can restart from where it was stopped when the part program was closed.

- **Reset Numbering**—resets the serial number back to the first string of the sequence. Select the Reset Numbering button or the Reset Numbering softkey.
HD3 Lettering

The HD3 Lettering function mills lettering into a part.

Follow this link to the Hurco website to view an HD3 Lettering programming video demonstration:

[Video Demonstrations at www.hurco.com](http://www.hurco.com)

Scroll to the bottom of the page and select the video to download. Internet connection is required.

Cutter Compensation is not available for HD3 Lettering. Milling is performed using the centerline of the tool. Refer to *Cutter Compensation, on page 1 - 13.*

The fields are defined as follows:

- **X Ref Location**—specifies Start, End, or Center of the character for the X reference point. Default is Start.
  - Start—the lettering is left-justified; the first character begins at the X reference point.
  - Center—the lettering is centered left and right of the X reference point.
  - End—the lettering is right-justified; the last character ends at the X reference point.
- **X Reference**—indicates the X coordinate of the point where text begins.
- **Y Ref Location**—specifies Bottom, Top, or Center of the character for the Y reference point. Default is Bottom.
  - Bottom—the bottom of the letter is aligned with the Y reference point.
  - Top—the top of the letter is aligned with the Y reference point.
  - Center—the letter is centered above and below the Y reference point.
- **Y Reference**—indicates the Y coordinate of the point where text begins.
Figure 2–23. X Reference and Y Reference locations

- **Character Height**—Determines the height of the character set.
- **Character Width**—Determines the character size. The width includes the character left-justified, plus some spacing between it and the next character to be milled. Spacing between the characters is equal to the tool's diameter.

Because cutter compensation is not used in this routine and letter contouring follows the center of the tool, character spacing can be adjusted by adjusting the tool diameter in Tool Setup. See *Getting Started with WinMax Mill* for more information.

- **Text**—Contains characters to be milled into the part. This field will support up to 30 block characters.
- **Z Start**—The point where Plunge Feed rate begins for the tool to move down to Z Bottom.
- **Z Bottom**—Identifies the point at which the mill feed rate begins. The value initially displayed has been calculated by the control and can be retained or changed to a different value.
- **Peck Depth**—Defines the maximum depth to be cut in one pass of the tool. If the total depth is greater than this value, multiple tool passes occur. Entering the value zero (0) causes the total programmed depth to be cut in one pass of the tool.
- **Plunge Feed**—Defines the feed rate for the tool moving from Z Start to Z Bottom.
- **Mill Feed**—Defines the X-Y feedrate. See *Getting Started with WinMax Mill* for more information.
- **Speed (RPM)**—Identifies the spindle speed for the tool calculated in Tool Setup. Entering a value here overrides the Tool Setup value for this data block. See *Getting Started with WinMax Mill* for more information.
- **Tool Diameter**—Contains the diameter entered during Tool Setup. See *Getting Started with WinMax Mill* for more information.
Swept Surface

The Swept Surface option in WinMax allows you to program draft angles for pocket or island walls with greater flexibility, including Spiral, Constant Z level, and Cusp Height cutting strategies.

Follow this link to the Hurco website to view a Swept Surface programming video demonstration:

[Video Demonstrations at www.hurco.com](http://www.hurco.com)

Scroll to the bottom of the page and select the video to download. Internet connection is required.

Swept Surface builds on the WinMax 3D Mold option. The basic elements in Swept Surface programming are:

- Draw Profile Contour—build the XZ or YZ shape of the part
- Draw Along Contour—build the shape of the contour
- Swept Surface—provide details of how to cut the part

To begin a new Swept Surface program block, perform the following steps:

1. From a New Block screen, select the MILLING F3 softkey.
2. Select the More F7 softkey.
4. Use the drop-down list to select the type of contour in the TYPE field: Draw Profile, Draw Along, or Swept Surface.
   
   A Swept Surface program block must contain all three types of contours: Draw Profile, Draw Along, and Swept Surface.

5. To program Draw Profile Contours or Draw Along Contours, place the cursor in the TYPE field, and select the EDIT ALONG CONTOUR F6 or the EDIT PROFILE CONTOUR F7 softkey.

Draw Profile Contour

Set the TYPE field to DRAW PROFILE(S) and select the EDIT PROFILE CONTOUR F7 softkey. Segment types are:

- Line
- Arc
- Blend Arc
- Helix
- 3D Arc

The Draw Profile Contour screen is displayed as Swept Surface (Profile Contour).
Draw Along Contour

Set the TYPE field to DRAW ALONG and select the EDIT ALONG CONTOUR F6 softkey. Segment types are:

- Line
- Arc
- Blend Arc
- Helix
- 3D Arc

The Draw Along Contour screen is displayed as Swept Surface (Along Contour).

The **Surface Side** field (present in Segment 0) specifies on which side of the Along contour the profile is cut:

- When set to **Right**, the profile is cut to the right of the contour forming a surface to the right of the along contour.
- When set to **Left**, the PROFILE is cut to left of the contour creating a surface to the left of the along contour.
Swept Surface

Swept Surface parameters establish the machining characteristics. Fields are:

- **TYPE**—specifies the type of contour:
  - **DRAW PROFILE(S)**—program the XZ or YZ part dimensions.
  - **DRAW ALONG**—program the XY contour that the XZ or YZ profile follows.
  - **SWEPT SURFACE**—program the details of the Swept Surface.
- **TYPE OF CORNERS**—the type of corner (rounded, extended, or truncated) to insert when there is a change of direction in the Draw Along contour.
- **INDIVIDUAL PROFILES**—allows different profile contours to be used for individual segments of the Draw Along contour.
- **ROLL START POINT**—when YES, sweeps the profile down to the horizontal plane at the start point.
- **ROLL END POINT**—when YES, sweeps the profile down to the horizontal plane at the end point.
- **Z START**—the Z coordinate at which plunge feedrate begins.
- **STEP CONNECT TYPE**—the type of segment (arc or line) to connect passes of normal cut direction.
- **USE CUSP HEIGHT**—when YES, step over is calculated from the specified cusp height. Used only with flat and ball end mills.
- **MIN CUSP OVERLAP (%)**—minimum cutter overlap used for cusp height (percent of cutter diameter) with a flat end mill. The USE CUSP HEIGHT field must be set to YES.
- **POCKET FIRST**—select YES if pocketing should be done before milling the swept surface.

Tabs at the bottom of the screen contain fields for Roughing, Finishing, Pocket, Pocket Finishing, and SFQ:

- **TOOL**—specifies the tools to be used for roughing, finishing, pocketing; select from list or enter a number.
- **CUT DIRECTION**—controls the tool path while the part is machined. There are three choices for the CUT DIRECTION field:
  - **With Contour** - machines the contour using the profile contour originally programmed.
  - **Normal** - tool path follows the part at right angles to the profile contour.
  - **Spiral** - machines a continuous tool path resulting in a smoother surface.
- **BIDIRECTIONAL**—specifies the direction of the tool path while the part is being machined. There are two choices for the BIDIRECTIONAL field:
  - **Yes** - causes the tool to machine in both directions without retracting the tool until the entire contour is complete.
  - **No** - causes the tool to machine in one direction, based on the direction of the contour definition, while retracting between each pass.
Figure 2–24. Bidirectional Field

- **MILL FEED**—defines the X-Y feedrate. The value initially displayed has been calculated by the control and can be retained or changed to a different value.

- **SPEED (RPM)**—spindle speed for the tool, calculated in Tool Setup. Entering a value here overrides the Tool Setup value for this data block.

- **STOCK ALLOWANCE**—leaves or removes extra material on the surface of the part. Stock Allowance can be used for roughing, undersizing, or oversizing a surface. A Ball-Nosed End Mill must be used to maintain a uniform stock allowance dimension over the complete surface. A positive stock allowance value programmed using a Flat End Mill leaves sufficient material for a finishing pass.

  ![Bidirectional Field Diagram]

<table>
<thead>
<tr>
<th></th>
<th>Yes</th>
<th>No</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Print Dimension</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>+ Stock Allowance</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>- Stock Allowance</td>
<td></td>
</tr>
</tbody>
</table>

Figure 2–25. Stock Allowance

When machining harder materials, progressively smaller stock allowances can be used in a series of finishing blocks to optimize cutting efficiency.

- **MIN Z**—limits the negative Z motion to the Centerline Z value when set to Yes.
• **STEP SIZE**—the distance on the surface between cutter passes. Ultimately, this dimension determines the surface finish of the part. A larger step size machines faster but leaves a rougher surface. A smaller step size machines more slowly but leaves a smoother surface. Step size significantly affects the drawing speed of the graphics screen.

![Step Size Example](image)

Figure 2–26. Step Size Example

• **CUSP HEIGHT**—available when the **Use Cusp Height** field is set to Yes. Specifies the step size for ball and flat end mills.

• **PECK DEPTH**—Defines the maximum depth to be cut in one pass. If the total depth is greater than this value, multiple cutting passes occur. Entering a zero (0) value causes the total programmed depth to be cut in one pass of the tool.

![Peck Depth Example](image)

Figure 2–27. Peck Depth Example

• **PLUNGE FEED**—Z feedrate for plunge in inches per minute.

• **Z ROUGHING**—specifies if the Z roughing strategy be applied to the roughing pass.

• **MILLING TYPE**—specifies the pocket milling type: None, Island, Pocket, Island & Pocket.

• **POCKET TYPE**—specifies the pocketing type: inward or outward.
- **POCKET OVERLAP (%)**—Appears when **Pocket Boundary** Milling Type is selected. Pocket Overlap is the percentage of tool diameter that overlaps for each pass in a pocket milling operation.

If the overlap is less than 50%, the toolpath may make additional moves to clean out the pocket. For example (toolpath is red):

Additional corner moves for Pocket Overlap of 25%

- **ROUGH SFQ**—specifies Surface Finish Quality for roughing.
- **FINISH SFQ**—specifies Surface Finish Quality for finishing.
- **POCKET ROUGH SFQ**—specifies Surface Finish Quality for pocket roughing.
- **POCKET FINISH SFQ**—specifies Surface Finish Quality for pocket finishing.
Mill Slot

For Rotary Slot information, see Rotary Slot in the Rotary chapter of WinMax Mill Options.

The Mill Slot block creates a line or arc shape of any width. The slot is defined by a line or an arc segment and a width. The ends of the slot can be round or square.

Milling begins at the center of the bottom wall of the slot, assuming the shape is parallel to the X axis (Angle field is set to zero). The same relative “bottom” wall is maintained when the shape is rotated, so when the Angle field is nonzero, milling begins on the side that would be parallel with the X-axis if the Angle was zero:

To access the Mill Slot block:

1. From a New Block screen, select the Milling softkey.
2. Select the More softkey.
3. Select the Slot softkey.

A Mill Slot data block will be converted to a Mill Contour data block if the program us saved as an HD3 program.

The Mill Slot screen uses three tabs: Start, Geometry, and Caps.
Start tab

The Mill Slot block Start tab fields define the slot shape, width, and common starting information.

![Mill Slot Start tab](image)

**Figure 2–29. Mill Slot Start tab**

Fields are:

- **Shape**—specifies the shape of the slot, either line or arc.
- **Width**—indicates the width of the slot. The slot width determines how far to expand the centerline or center arc segment programmed in the Geometry tab. Half of the programmed width is applied to each side of the line or arc.
- **X Start**—indicates the starting point of the line or arc segment in X.
- **Y Start**—indicates the starting point of the line or arc segment in Y.
- **Z Start**—indicates the plunge feedrate start point.
- **Z Bottom**—indicates the plunge bottom point.
Geometry tab

The Geometry tab allows the user to program a single line or arc segment. The fields that appear depend on the slot shape selected in the Start tab:

![Geometry Tab Example](image)

**Line geometry fields are:**

- **X End**—indicates the end point of the line segment in X.
- **Y End**—indicates the end point of the line segment in Y.
- **XY Length**—indicates the distance from the start point to the end point.
- **XY Angle**—indicates the angle in degrees of the line segment measured from the three o’clock position.

⇒ Some fields may be automatically calculated when sufficient data has been entered. The CAL notation appears before the field.

![Arc Geometry Tab Example](image)

**Arc geometry fields are:**

- **X End**—indicates the end point of the arc segment in X.
- **Y End**—indicates the end point of the arc segment in Y.
- **X Center**—indicates the center point of the arc, in X.
- **Y Center**—indicates the center point of the arc, in Y.
- **Sweep Angle**—indicates the angular distance from the start point to the end point around the center point. The range is 0° to 360°.
- **Radius**—indicates the distance from the center point to start and end points.
- **Direction**—indicates the direction of rotation from the start point to the end point around the center point.
Caps tab

The Caps tab contains fields that define the ends of the slot.

![Figure 2–32. Mill Slot Caps tab](image)

Fields are:

- **Start Cap**—specifies the shape that is used to close the slot at the specified start point:
  - **Line**—flat edge connects the top and bottom edges of the slot, passing through the programmed start point.
  - **Append Arc**—rounded edge; center of semi-circle is programmed start point.
  - **Include Arc**—rounded edge; outermost edge of semi-circle is programmed start point.

- **End Cap**—specifies the shape that is used to close the slot at the specified end point:
  - **Line**—flat edge connects the top and bottom edges of the slot, passing through the programmed endpoint.
  - **Append Arc**—rounded edge; center of semi-circle is programmed endpoint.
  - **Include Arc**—rounded edge; outermost edge of semi-circle is programmed endpoint.

- **Corner Radius**—specifies the blend radius for the corners of the Start Cap or End Cap if either or both are set to Line.

  > If a Corner Radius exceeds 1/2 of the slot width, WinMax will find the intersection point to close the contour.
Figure 2–33. Append Arc Start Cap and Line End Cap

Figure 2–34. Include Arc Start Cap and Line End Cap
Mill Polygon

For Rotary Polygon information, see *Rotary Polygon* in the Rotary chapter of *WinMax Mill Options*.

The Mill Polygon data block mills a multi-sided contour with equal-length sides. The bottom of the polygon is parallel to the X axis at 0° orientation (three o’clock position):

Milling begins on the side of the polygon that is parallel to X at 0° prior to rotation. Rotation is achieved with the Orientation Angle field.

To access the Mill Polygon block:

1. From a New Block screen, select the Milling softkey.
2. Select the More softkey.
3. Select the Polygon softkey. The Mill Polygon block opens:

   ![Figure 2–35. Mill Polygon screen](image)

The fields are:

- **Number Of Sides**—specifies the number of sides the polygon will have, from three to 100.
- **X Center**—identifies the X-axis coordinate for the center of the polygon.
• **Y Center**—identifies the Y-axis coordinate for the center of the polygon.

• **Sizing Method**—specifies how the polygon size is established:
  - **Outer Diameter**—the diameter of a circle that encompasses the outside of the polygon, touching it at the corners:

![Outer Diameter Diagram]

• **Inner Diameter**—the diameter of a circle contained within the polygon, touching at the center of each edge:

![Inner Diameter Diagram]

• **Side Length**—the length of one side of the polygon:

![Side Length Diagram]

• **Side Length**—specifies the length of the side. This field is available when the Side Length Sizing Method is selected.

• **Sizing Diameter**—specifies the diameter of the circle. This field is available when the Outer or Inner Diameter Sizing Method is selected.

• **Orientation Angle**—rotates the part. This field specifies an angle relative to the X axis that determines where the starting side occurs. An orientation of 0° indicates that the starting side is at the bottom of the polygon, parallel to the X axis. For example:
• **Z Start**—identifies the point at which the Plunge Feedrate begins.
• **Z Bottom**—identifies the point at which the Mill Feedrate begins.
• **Corner Radius**—identifies the radius of each corner of the polygon.

> A Mill Polygon data block will convert to a Mill Contour data block if saved as an HD3 program.
Mill Thread

The Mill Thread data block mills a part with internal or external threads. Milling can be clockwise or counterclockwise, and occur from top to bottom or bottom to top.

Example 1 - Thread Direction Down

The following diagram shows the Z positions, and the arrows indicate tool movement when the Thread Direction is Down. In this case, the tool moves at rapid feedrate to the Z Plunge Start position, then feeds to the Z Top position. The tool mills down to the Z Bottom position. It then moves at rapid feedrate out of the hole to Z Plunge Start + Z retract clearance height (set in Program Parameters).

![Figure 2–36. Tool movement when Thread Direction is down](image)

Example 2 - Thread Direction Up

The following diagram shows the Z positions, and the arrows indicate tool movement when Thread Direction is Up. In this case, the tool moves at rapid feedrate to the Z Plunge Start position, then feeds down to the Z Bottom position, and mills up to the Z Top position. The tool then moves at rapid to Z Plunge Start and retracts to the Z retract clearance height (set in Program Parameters).

![Figure 2–37. Tool movement when Thread Direction is up](image)
To access the Mill Thread block:

1. From a New Block screen, select the Milling softkey.
2. Select the More softkey twice.
3. Select the Thread softkey. The Mill Thread block opens:

   ![Mill Thread block](image)

**Figure 2–38. Mill Thread screen**

**Thread tab**

- **Radius**—specifies the radius of the major threads. When tapered threads are enabled, this is the radius at thread top.
- **TPI**—specifies the threads per inch. This field is available only when units are imperial (inch). This field is read-only for multi-cutter thread mills; the value comes from Tool Setup. When this value changes, the pitch is recalculated.
- **Pitch**—specifies the distance between threads. This field is read-only for multi-cutter thread mills; the value comes from Tool Setup. If this value is changed by the operator, the TPI is recalculated.
- **Orientation**—specifies Inside or Outside threads.
- **Start Angle**—specifies the starting angle for the threading operation in the XY plane. Zero Angle is at the 3 o’clock position.
- **Taper Angle**—specifies the angle for tapered threads. External threads must be positive, and internal threads must be negative. Enter zero for non-tapered threads. This field is available only when the tool is a single-cutter thread mill.
Location tab

- **X Center**—specifies the X-axis coordinate for the center of the thread.
- **Y Center**—specifies the Y-axis coordinate for the center of the thread.
- **Z Plunge Start**—specifies the Z position at which plunge feed begins.
- **Thread Top**—specifies the Z position for the top of the thread.
- **Thread Bottom**—specifies the Z position for the bottom of the thread.
- **Direction**—specifies the direction of the threads (clockwise or counterclockwise) and the manner in which they are cut (up or down).
Process tab

The fields are:

- **Enable Cutter Comp**—specifies whether the tool path is offset by the tool radius. Default is Yes. When enabled (Yes), the value of the Thread Diameter (set in Tool Setup) is used to determine the radius.

- **Blend Type**—specifies the type of blend move to make at the beginning and end of the threading operation. Choices are line, ramp, arc, helix.

- **Start at Center**—specifies if the threading operation should start at the center of the threads. Default is Yes. If set to No, the Blend Offset value from Program Parameters is used. This field is available only when the Internal Thread field is set to Yes.
Lines/Arcs Data Blocks

Use segments and cutter compensation to program lines and arcs which create a Mill Contour. The Contour End block marks the end of the programmed contour. Refer to *Contour End, on page 2 - 63*. These are the types of segments used to create a Mill Contour:

- *Contour Line, on page 2 - 56*
- *Contour Arc, on page 2 - 57*
- *Contour Blend Arc, on page 2 - 58*
- *Helix, on page 2 - 60*
- *3D Arc, on page 2 - 61*

Select these segments from the Mill Contour screen by moving the cursor to the Segment field and selecting the *Insert Segment Before* softkey or pressing the PAGE DOWN key. The *Line, Arc, Blend Arc, Helix, and 3D Arc* softkeys appear.

Mill Contour

A series of line and arc segments can be programmed in a single data block to form a complete contour.

Follow this link to the Hurco website to view a Mill Contour programming video demonstration:

[Video Demonstrations at www.hurco.com](http://www.hurco.com)

Scroll to the bottom of the page and select the video to download. Internet connection is required.

The Mill Contour fields are defined as follows:

- **Block**—Identifies the block number for this Mill Contour data block. The system determines the number by the data block's position in the program.
- **Segment**—Identifies the Segments number within the Mill Contour operation. Refer to *Segments, on page 1 - 4*.
- **Tool**—Identifies the number of the tool used to cut the segments in this contour.
- **Finish Tool**—Identifies the tool number used for the finish pass. If a finish tool is selected, the first tool leaves the amount of material selected in the Finish Z field in Milling Parameters. The finish tool removes the remaining stock down to the programmed dimension.
- **Milling Type**—Identifies whether the system should automatically compensate for the diameter of the tool when determining tool path. With the cursor in the Milling Type field, select different types of Cutter Compensation.
or pocket type with the softkeys or drop-down list box. Refer to Cutter Compensation, on page 1 - 13.

- **Pocket Type**—Appears with Inward and Outward softkey and drop-down list box choices when Pocket Boundary or Pocket Island is chosen for Milling Type. The choices define whether spindle movement spirals from inside the pocket or outside the pocket.

- **Enable Blend Moves**—Appears when either Left or Right is selected in the Milling Types field. When set to Yes, automatic arc blend-in and blend-out is performed. Default is Yes.

  Also see Climb Milling (Left), on page 1 - 16 and Conventional Milling (Right), on page 1 - 16 for additional information.

- **X Start** and **Y Start**—Identify the X and Y starting point coordinates of the first segment.

- **Maximum Offset**—Appears when either Profile Left or Profile Right Milling Type is selected. The value is the distance between the programmed contour and the tool center during the first pass. The number entered here must be manually calculated. Refer to Maximum Offset, on page 1 - 17 for details.

- **Pocket Overlap**—Appears when Pocket Boundary Milling Type is selected. Pocket Overlap is the percentage of tool diameter that overlaps for each pass in a pocket milling operation.

  If the overlap is less than 50%, the toolpath may make additional moves to clean out the pocket. For example (toolpath is red):

  ![Additional corner moves for Pocket Overlap of 25%](image)

  If you change the Pocket Overlap using this field and save the file as an HD3 file, a Change Parameters data block will be inserted into the HD3 file to make the Pocket Overlap change. The Change Parameters block changes only the Pocket Overlap, leaving all other parameters as they were.

- **Lead Length**—Identifies the Lead Length for this Mill Contour. See Lead In/Out Moves, on page 1 - 20. This field only appears for NC posted programs. This field is only available when Milling Type is set to Left or Right. Lead In/Out Moves will be ignored in the post unless Enable Blend Moves is set to Yes and the Display Apt Fields in Editor field on the Post Processor Configuration screen is set to Yes.

- **Lead Angle**—Identifies the Lead Angle for this Mill Contour. See Lead In/Out
Moves, on page 1 - 20. This field only appears for NC posted programs. This field is only available when Milling Type is set to Left or Right. Lead In/Out Moves will be ignored in the post unless Enable Blend Moves is set to Yes and the Display Apt Fields in Editor field on the Post Processor Configuration screen is set to Yes.

- **Tool Diameter**—Contains the value entered during Tool Setup. See *Getting Started with WinMax Mill* for more information.

- **Tool Type**—Contains the type of tool entered during Tool Setup. See *Getting Started with WinMax Mill* for more information.

- **Z Start**—The point where Plunge Feed rate begins for the tool to move down to Z Bottom.

- **Z Bottom**—Identifies the point where the mill feed rate begins.

- **Peck Depth**—Defines the maximum depth to be cut in one pass. If the total depth is greater than this value, multiple cutting passes occur. Entering a zero (0) value causes the total programmed depth to be cut in one pass of the tool.

- **Plunge Feed**—Defines the feed rate for the tool moving from Z Start to Z Bottom.

- **Mill Feed**—Identifies the X-Y feedrate. The value initially displayed has been calculated by the control and can be retained or changed to a different value. See *Getting Started with WinMax Mill* for more information.

- **Speed (RPM)**—Determines the spindle speed for the tool calculated in Tool Setup. Ent ering a value here overrides the Tool Setup value for this data block. See *Getting Started with WinMax Mill* for more information.

- **Reverse Mirrored**—Appears if the contour block will be cut with a mirror pattern, and the MILLING TYPE field is not set to Pocket boundary or Pocket Island. If this field value is No, the contour will always be milled from the contour start block to the contour end block, and helixes can not be cut in the reverse direction. If the field value is Yes in the original contour, the cutting direction is determined by whether the contour is mirrored. The default value is Yes.

- **Finish Diameter**—Contains the tool diameter entered for the Finish Tool during Tool Setup. See *Getting Started with WinMax Mill* for more information.

- **Finish Type**—Contains the tool type entered for the Finish Tool during Tool Setup. See *Getting Started with WinMax Mill* for more information.
Contour Line

For line segments, the Auto-Calc feature can automatically calculate certain unknown dimensions after you enter sufficient data. Refer to *Automatic Calculations, on page 1 - 8*.

Line segment fields are defined as follows:

- **Block**—Identifies the block number for this data block. The system determines the number by the position of this data block in the program.
- **Segment**—Identifies the segment number within the Mill Contour operation. The system determines the number by the position of this segment in the contour.
- **X End, Y End, and Z End**—Identify the X, Y, and Z End coordinates. Refer to *Automatic Calculations, on page 1 - 8* to see how entering data affects this field.
- **XY Length**—Identifies the XY length. Refer to *Automatic Calculations, on page 1 - 8* to see how entering data affects this field.
- **XY Angle**—Identifies the XY Angle or the angle of the line segment (from the start point to the end point), measured counterclockwise from the 3 o'clock position. Refer to *Automatic Calculations, on page 1 - 8* to see how entering data affects this field.
- **Feed**—Identifies the feedrate the control will try to achieve while milling this segment. This value is carried forward from the Mill Contour screen's Mill Feed field. See also *Getting Started with WinMax Mill*.
- **X Start, Y Start, and Z Start**—Identify the X, Y, and Z starting points. The Start fields are carried forward from the previous segment's end points.
Contour Arc

For arc segments, the Auto-Calc feature can automatically calculate certain unknown dimensions after you enter sufficient data.

The Arc fields are defined as follows:

- **Block**—identifies the block number for this data block. The system determines the number by the position of this data block in the program.

- **Segment**—identifies the segment number within the Mill Contour operation. The system determines the number by the position of this segment in the contour.

- **Direction**—determines the direction of the arc from the start point (clockwise or counter-clockwise).

- **X End, Y End, and Z End**—identify data coordinates (values for X End, Y End, and Z End) used in the Automatic Calculations, on page 1 - 8.

- **X Center, Y Center, and Z Center**—identify data coordinates (values for X Center, Y Center, and Z Center) used in the Automatic Calculations, on page 1 - 8.

- **Radius**—identifies the radius (used in the Automatic Calculations, on page 1 - 8).

- **Sweep Angle**—identifies the angular distance in degrees from the start point of the arc to the end point. The range is -360° to 360°.

- **Feed**—identifies the rate at which the tool initially enters the part. This value is carried forward from the Mill Contour screen’s Mill Feed field. See Getting Started with WinMax Mill for more information.

- **X Start, Y Start, and Z Start**—define the X, Y, and Z starting points of this segment. The Start fields are carried forward from the previous segment’s end points.
Contour Blend Arc

Follow this link to the Hurco website to view a Blend Arc programming video demonstration:

[Video Demonstrations at www.hurco.com](http://www.hurco.com) Scroll to the bottom of the page and select the video to download. Internet connection is required.

A blend arc is an arc that joins two other segments and is tangent to both. Use a blend arc to join two line segments, to join a line segment and an arc segment, or to join two arc segments. The segments to be joined must have a theoretical point of intersection.

![Blend Arc Examples](image)

If the only information known about an arc is its radius, it is easier to program it as a blend arc if the segments intersect.

The Contour Blend Arc fields are defined as follows:

- **Block**—Identifies the block number for this data block. The system determines the number by the position of this data block in the program.
- **Segment**—Identifies the segment number within the Mill Contour operation. The system determines the number by the position of this segment in the contour.
- **Radius**—Identifies the radius of the arc.
- **Feed**—Identifies the feed value carried forward from the Mill Contour screen's Mill Feed field. See *Getting Started with WinMax Mill* for more information.
• **Direction**—The direction of the arc from the start point (clockwise or counterclockwise).

• **X Start** and **Y Start**—The X Start and Y Start coordinates of the first segment.

• **X End** and **Y End**—The X End and Y End coordinates.

• **X Center** and **Y Center**—The X and Y coordinates used to define the circular path of the blend arc.

Reference values programmed in a previous segment to a blend arc define the start point of this segment and are displayed in parentheses. These values can only be changed in the segment in which they were created.

Some guidelines that must be followed when creating a blend arc:

• The first or last segment of a Mill Contour data block cannot be blend arc segments.

• Blend arc segments cannot be adjacent to one another in a program cannot be blend arc segments. For example, if segment #2 is a blend arc, neither segment #1 nor #3 can be blend arc segments.

• Segments that are adjacent to the blend arc segment must intersect at some point in their theoretical plane. Therefore, if segment #2 is a blend arc, segments #1 and #3 must theoretically intersect at some projected point.

• The Radius of a blend arc segment cannot be too large to be tangent to both of the adjoining segments.

• If any coordinate (start point, center point, or end point) is important to the construction of the two segments to be blended, the segment must be programmed as an arc and not as a blend arc.

• The Feed field is initially displayed with a value carried forward from the previous segment. This value can be accepted or changed.

⇒ A series of arcs and lines can be programmed in a single data block to form a complete contour. Press the right Arrow key to program additional Line and Arc segments for the current data block.
Helix

For Helix data blocks, the Auto-Calc feature can automatically calculate certain unknown dimensions after you enter sufficient data.

The Helix fields are defined as follows:

- **Block**—Identifies the block number for this data block. The system determines the number by the position of this data block in the program.

- **Segment**—Identifies the segment number within the Mill Contour operation. The system determines the number by the position of this segment in the contour.

- **Direction**—Defines the direction of a helix from the starting point. There are two softkey choices when the cursor is on the Direction field.
  - **CW** - the helix will be cut in a clockwise direction.
  - **CCW** - the helix will be cut in a counterclockwise direction.

- **X End, Y End, and Z End**—Identify the X, Y, and Z coordinates for the end point of the helix segment. Refer to *Automatic Calculations, on page 1 - 8* to see how entering data affects this field.

- **X Center and Y Center**—Identify the X and Y coordinate for the center point around which the helical pattern is drawn. Refer to *Automatic Calculations, on page 1 - 8* to see how entering data affects this field.

- **Radius**—Defines the length of the dimension from the center point of the helical pattern to its outer edge.

- **Lead**—Defines the change in the Z axis dimension for each $360^\circ$ of revolution of the helix. This value provides information for the calculation of the X End, Y End, Z End, and Sweep Angle fields.

- **Sweep Angle**—Defines the total number of degrees in the helical arc to be cut. This number can be greater than 360. This value provides information for the calculation of the Z End field and the Lead fields.

- **Feed**—Identifies the rate from the Mill Contour screen’s Mill Feed field. See *Getting Started with WinMax Mill* for more information.

- **X Start, Y Start, and Z Start**—Define the X, Y, and Z starting points of this segment. The Start fields are carried forward from the previous segment’s end points.
3D Arc

Use 3D Arc to create a three-dimensional arc segment.

Do not confuse the 3D Arc feature with the 3D Mold Option. The 3D Arc feature is always included in the WinMax software. Refer to WinMax Mill Options for more information on the 3D Mold option.

The following fields define a 3D Arc:

- **X, Y, and Z End**—define the end point for the arc segment. These fields are required when defining a 3D Arc.
- **X, Y, and Z Center**—define the center point for the arc segment. These fields are required when defining a 3D Arc.
- **X, Y, and Z Point**—define the plane of the arc when it is 180° or greater.
- **Feed**—rate from the previous segment. This value can be accepted or changed.

This diagram shows the relationships of the coordinates to each other:

![Diagram of 3D Arc](image)

Figure 2–40. 180° 3D Arc
To calculate the centerline of cutter movement, remember these points. Refer to *Cutter Compensation, on page 1 - 13* for information about cutter compensation.

- Depending on the axis being worked, X can be interchanged with Y or Z.
- The X and Z points must be calculated for an X-Z arc.
- The Y and Z points must be calculated for a Y-Z arc (all arcs travel through at least two axes).
- To calculate the centerline in the X axis for a ball-nosed end mill, use this formula:

\[
X_{a} = \frac{R_{i}}{R} (X_{s} - X_{c}) + X_{c}
\]

Here are the elements of this formula:

- \(X_{a}\) = Actual centerline dimension of cutter in X axis
- \(X_{s}\) = Arc reference starting point in the X axis
- \(X_{c}\) = X center point
- \(R_{i}\) = Radius of arc minus ½ cutter diameter
- \(R\) = Radius of arc

Follow these *guidelines* when programming 3D arcs:

- Never program cut right or cut left cutter compensation into a 3D arc block.
- Always program the centerline of cutter movement.
- Lower the tool calibration point to one-half the cutter diameter by manually changing the Tool Zero reference point.

For example, when using a 1/4" ball-nosed end mill with Tool Zero of 2.2500" (ball tip touched to work surface), the new zero calibration is 2.3750". Remember this change when entering Z Up and Z Start dimensions (which must include the value of this manual change to the reference point).
Contour End

This block marks the end of the programmed contour. To view the previous segment, select the **Previous Segment** softkey. If there are no existing segments, select the **Insert Segment Before** softkey or the PAGE DOWN key to create a new segment.

The Contour End fields are defined as follows:

- **Block**—Identifies the block number for this data block. The system determines the number by the position of this data block in the program.
- **Segment**—Identifies the segment number within the Mill Contour operation. The system determines the number by the position of this segment in the contour.

The Contour End screen is shared by Lines and Arcs, Rotary Lines and Arcs, and 3D Mold.
HOLES OPERATIONS

These topics are discussed in this section:

Holes Data Block .......................................................... 3 - 2
Drill Overview ............................................................... 3 - 3
Tap Operations ............................................................... 3 - 18
Back Spotface ............................................................... 3 - 23
Bolt Circle ................................................................. 3 - 24
Locations ................................................................. 3 - 25
Bolt Circle to Holes Locations or Pattern Locations Conversion ................................ 3 - 27
Holes to Pattern Locations Conversion ................................ 3 - 28
Holes End Block .......................................................... 3 - 28
Holes Data Block

1. Begin a Holes Data Block by displaying the Input screen and selecting the **Part Programming** softkey or icon.

2. Select the **Insert Block Before** softkey.


4. Select the appropriate softkey for the type of operation.

5. Select the tool for the first operation in the data block.
   - The structure of a hole drilling data block is more complex than other conversational data blocks. To describe a hole drilling process in a data block, enter different types of instructions.
   - Each type of drilling instruction requires creating a separate operation within the data block.

6. Program the operation.

7. After completing an operation, place the cursor in the Operation field and the following softkeys are available:
   - **Next Hole Operation** - places a new Holes operation into the data block after the operation currently displayed on the screen.
   - **Delete Operation** - removes the currently displayed operation from the holes data block.
   - **Insert Operation Before** - places another operation into the data block before the operation currently displayed on the screen.

That when the cursor is at the Peck Type field, the softkeys change to Standard and Chip Breaker. When this program is run on the control, the drill pulls out to the Z Start after reaching the Peck Depth for Standard Peck type and dwells at the Peck Depth for Chip Breaker peck type.

The New Hole Operation menu includes Bolt Circle and Locations softkeys. Select the **Locations** softkey, and the system displays columns for entering the X and Y coordinates of each location.

When the description of a drilling operation is complete, these softkeys are available:

   - **Delete Location** - remove the location coordinates of the hole location currently marked with the cursor.
   - **Insert Location Before** - place another pair of location coordinates before the location currently marked with the cursor.

Choose the Drill softkey if you want to pick a true center drill. Center Drill is just for "Spot" operations.
Drill Overview

See also:

*Drill, on page 3 - 6*

*Center Drill, on page 3 - 6*

*Counterbore, on page 3 - 6*

*Countersink, on page 3 - 6*

*Spotface, on page 3 - 6*

*Gun Drill, on page 3 - 6*

*Custom Drill, on page 3 - 10*

Drill Operations drill different types of holes. The Drill Operation types are Drill, Center Drill, Counterbore, Spotface, Countersink, and Gun Drill. The fields do not differ with the types of operation, except Gun Drill.

The diagram below shows the relationships of the field names to positions of the tool to the work piece:

![Diagram of drill field names and positions](image)

<table>
<thead>
<tr>
<th></th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Head</td>
</tr>
<tr>
<td>2</td>
<td>Part</td>
</tr>
<tr>
<td>3</td>
<td>Table</td>
</tr>
<tr>
<td>4</td>
<td>Z Start Plane</td>
</tr>
<tr>
<td>5</td>
<td>Rapid Traverse</td>
</tr>
<tr>
<td>6</td>
<td>Retract Clearance Plane</td>
</tr>
<tr>
<td>7</td>
<td>Plunge Feed</td>
</tr>
<tr>
<td>8</td>
<td>Z Bottom</td>
</tr>
</tbody>
</table>

*Figure 3–1. Reference Points Relative to Drill Operations*

Be aware of the drill bits' characteristics. The system does not keep track of the type and length of flutes on the bits or the shapes of drills with pilots or other multiple-diameter drills.
This motion occurs in a drilling operation:

1. The table moves at the rapid traverse rate to the programmed X and Y dimensions.
2. The Z axis moves at the rapid traverse rate to the programmed retract clearance.
3. When these coordinates are reached, the Z axis moves at the rapid traverse rate to Z Start.
4. Then the Z axis moves at the programmed plunge feed rate until reaching the Z Bottom dimension.
5. The Z axis dwells the time specified and rapid traverses to the retract clearance.

The rapid traverse rate, retract clearance, and drill dwell are selected in Program Parameters or Change Parameters.

When all values are programmed, the Bolt Circle and Locations softkeys are available. See Bolt Circle, on page 3 - 24 and Locations, on page 3 - 25 for more information.

- **Delete Operation**—Clears this programmed Drill operation.
- **Insert Operation Before**—Adds another Hole operation before the current one.
- **Exit**—Returns to the previous screen level.

Additional operations for this block can be programmed, such as another Drill Operation (i.e., a Center Drill, Counterbore, or Spotface), a Tap Operation, Bore and Ream Operations, Bolt Circle Pattern, or Hole Locations.

**Drill or Bore and Ream Operations**

Select the Drill Operations softkey on the New Hole Operation screen to select the type of drilling operation. The appropriate screen appears with fields describing the operation.

Select the Bore and Ream Operations softkey on the New Hole Operation screen to select the type of boring and reaming operation. Refer to Bore and Ream Operations, on page 3 - 19. The appropriate screen appears with fields describing the operation.

- **Block**—Identifies the block number for this operation.
- **Operation**—Identifies the operation number for this function. The system determines the number by the position of this function in the data block.
- **Hole Operation Type** (not labeled) - allows you to select a different hole operation type from the drop-down list. You may also use the softkeys to select another hole operation type. Select the More ➔ softkey to access more tool type softkeys.

Drill Operations drill different types of holes. The Drill Operation types are Drill, on page 3 - 6, Center Drill, on page 3 - 6, Counterbore, on page 3 - 6, Spotface, on page 3 - 6, Countersink, on page 3 - 6, and Gun Drill, on page 3 - 7. The fields do not differ with the types of operation, except Gun Drill.

- **Tool**—Identifies the tool number for this data block.
• **Z Start**—Identifies the point where the spindle begins to rotate. The Z Start field is the dimension where the Z axis plunge feed rate begins.

  The Z Start value entered in the first operation of a holes block is carried over to new operations in that same block. The value can be changed manually in any operation.

• **Z Bottom**—Identifies the bottom of the finished hole.

• **Peck Type**—Used only with Drill. Two softkey choices appear: Standard and Chip Breaker.
  
  • **Standard** - the tool retracts to Z Start or the location specified in the Pecking Retract Clearance field between pecks.
  
  • **Chip Breaker** - the tool dwells at each peck level before continuing to feed to the next peck level. (Dwell time is the time selected in Program Parameters or Change Parameters (Holes). See *Getting Started with WinMax Mill* for more information.

• **Peck Depth**—Used only with Drill. The distance the tool drills down into the part before stopping to clear out or break the chips. If used, this parameter is usually not larger than the tool's diameter. If a specific number of pecks is desired, the value for Peck Depth can be calculated by determining the absolute distance between Z Start and Z Bottom and dividing that value by the desired number of pecks. For example: a distance of 1.0000 inch between Z Start and Z Bottom divided by a peck depth of 0.2500 inch requires four (4) pecks to drill the hole.

• **Pecking Retract Clearance**—Used only with Drill/Custom Drill. Specifies an incremental distance above or below Z Start to which the tool retracts between pecks.
  
  • Field is only enabled when Peck Type is Standard.
  
  • Defaults to 0.0, in which case the tool returns to Z Start between pecks.
  
  • This value may be negative, indicating a peck retract plane below Z Start.
  
  • If Peck Depth = 0.0, this field is disabled.

  This field applies only to Holes drilling operations and has no relationship to the Pecking Retract Clearance field in Program Parameters, which applies only to Milling operations.

• **Speed (RPM)**—Identifies the speed at which the spindle rotates to drill the hole, calculated in Tool Setup. Entering a value here overrides the Tool Setup value for this data block. See *Getting Started with WinMax Mill* for more information.

• **Plunge Feed**—identifies the rate at which the tool initially enters the part. The Z Start parameter is the dimension where the Z axis plunge feed rate begins. The Z Bottom parameter is the point of maximum Z-down and the dimension where the plunge feed rate ends.
**Counterbore**

Use Counterbore to enlarge the end of a previously drilled hole. This operation creates an enlarged area at the end of the hole to accommodate a bolt, cap screw, or pin. These square-shoulder fasteners can then be inserted flush with the top of the part or slightly below the surface of the material.

To avoid unneeded tool changes, use Counterbore before drilling the hole.

**Countersink**

Countersink enlarges the top end of a hole to a cone shape to accommodate the head of a flat or oval headed machine screw. The hole is enlarged so that when the screw is inserted, the screw head will be flush with or slightly below the top of the work piece surface.

*Figure 3–2. Countersunk screw head*

**Drill**

Use a drill to create holes that may be complete or may be used as the starting points for additional drilling operations.

**Center Drill**

Select Center Drill when the work piece is very rigid to create a guide hole for the drilling tool that will be used to complete the hole. Center drilling helps keep holes in their proper locations and prevent runout.

Center drills are special tools with short flutes, but a standard drilling tool may be used to create these starting holes.

**Spotface**

Spotface smooths and squares the surface around a previously drilled hole to provide a seat for a bolt head, a nut, or the shoulders on mating members. The spotfacing operation is a shallow counterboring operation. Spotfacing is often used when the surface of the part is uneven.

To avoid unnecessary tool changes, Spotface can be used before drilling the hole.
Gun Drill

Select the Gun Drill feature when using a very long tool and drilling deep holes, as if drilling out the center of a gun barrel. A long, rotating tool may whip off center when approaching the cutting surface, so a drilling cycle that controls the approach to the work piece and corrects positioning of the tool is required.

The Gun Drill selection positions the tool without rotating the spindle. This allows the tip of the tool to position precisely in a pre-drilled starting hole, as shown in the diagram below.

To use the Gun Drill features, perform these tasks:

- Describe the gun drill tool during Tool Setup.
- Create a starting hole for the drill as the first step in part programming.
- Create a Gun Drill data block in the part program.

Each Gun Drill data block must contain at least two operations: one describing the starting hole and one for the gun drill hole.
To create a Gun Drill data block, begin on a New data block screen and follow these general steps:

1. Press the Drill Operations softkey.
2. Create the first operation of the data block by describing the starting hole portion of the data block. Select either the Drill softkey or the Center Drill softkey. Type in the description of the starting hole into the fields displayed on the screen.
3. When all of the starting hole data have been entered, begin the second operation in this data block.
4. Press the Drill Operations softkey and then the Gun Drill softkey.

The following illustration shows the Gun Drill program fields as they relate to a part and the drilling tool:

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Work Piece</td>
</tr>
<tr>
<td>2</td>
<td>Move at Rapid Traverse Feedrate down to Z Top</td>
</tr>
<tr>
<td>3</td>
<td>Z Top</td>
</tr>
<tr>
<td>4</td>
<td>Z Top Feed—feedrate between Z Top and Z Start</td>
</tr>
<tr>
<td>5</td>
<td>Z Start—spindle begins to rotate at Speed (RPM)</td>
</tr>
<tr>
<td>6</td>
<td>Z Bottom</td>
</tr>
</tbody>
</table>

*Figure 3–4. Gun Drill Positions*
The Gun Drill fields are defined as follows:

- **Block**—Identifies the block number for this operation.
- **Operation**—Identifies the operation number for this function. The system determines the number by the position of this function in the data block.
- **Tool**—Identifies the tool number for this data block and enters that tool's Diameter and Tool Type on this screen.
- **Z Top**—Identifies the starting point of the gun drill cycle. The tool moves rapidly to this point and then slowly moves into the pre-drilled starting hole.
- **Z Start**—Identifies the position inside the pre-drilled hole where the spindle begins to rotate.
- **Z Bottom**—Identifies the bottom of the finished hole.
- **Z Top Feed**—Identifies the rate at which the tool moves from the Z Top position to the Z Start position.
- **Plunge Feed**—Identifies the feedrate between the Z Start and the Z Bottom positions.
- **Speed (RPM)**—Defines the speed at which the spindle rotates to drill the hole. See also he time selected in Program Parameters or Change Parameters (Holes). See *Getting Started with WinMax Mill* for more information.
- **Spindle Stop**—Identifies the point at which the spindle stops rotating after drilling the hole. This field may be set to be the Z Bottom or the Z Start position.
- **Peck Depth**—Identifies the distance the tool drills down into the part before stopping to clear out or break the chips. If this field is used, it is usually not larger than the tool's diameter.
- **Peck Type**—Select whether the drill pulls out to the Z Start position after reaching Peck Depth (Standard) or whether the drill dwells at the Peck Depth (Chip Breaker).
  - **Tool Diameter**—Contains the value entered during Tool Setup. Refer to Tool Setup in *Getting Started with WinMax Mill* for more information.
  - **Tool Type**—Contains the type entered during Tool Setup. Refer to Tool Setup in *Getting Started with WinMax Mill* for more information.
Custom Drill

The Custom Drill feature can be used to adjust the feedrate and speed for each step in the drill cycle: entry into hole, break out through bottom of material, bottom, re-entry into hole from bottom, out move at top of hole, and the move to retract clearance plane. Custom Drill is typically used for long hole drill cycles but can be used to customize any drill cycle.

Tool Tab

The Custom Drill screen is organized into tabs where the feed, speed, and other parameters are entered for each step of the drill cycle. The first tab is the Tool tab:

![Custom Drill Tool tab](image)

A list of all hole operations are displayed on the Tool tab to show the relation of the Custom Drill cycle with other operations in the block. Fields are:

- **Tool**—specifies the tool to be used for the operation.
- **Peck Type**—specifies whether the drill pulls out to the Z Start position after reaching Peck Depth (Standard) or if it dwells at the Peck Depth (Chip Breaker). Default is Standard.
- **Peck Depth**—specifies the distance the tool drills down into the part before stopping to clear out or break the chips. If this field is used, it is usually not larger than the tool’s diameter. If Peck Depth is zero, Peck Type is ignored.
- **Pecking Retract Clearance**—Used only with Drill/Custom Drill. Specifies an incremental distance above or below Z Start to which the tool retracts between pecks.
  - Field is only enabled when Peck Type is Standard.
  - Defaults to 0.0, in which case the tool returns to Z Start between pecks.
  - This value may be negative, indicating a peck retract plane below Z Start.
• If **Peck Depth** = 0.0, this field is disabled.

This field applies only to Holes drilling operations and has no relationship to the Pecking Retract Clearance field in Program Parameters, which applies only to Milling operations.
Entry Tab

The Entry step establishes the speed and feed for the approach move starting at Z Start and ending at Z Top. The image shows the move from Z Start to Z Top:

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Z Start</td>
<td>specifies the feed starting point. The move to Z Start is at rapid feedrate from the retract clearance plane.</td>
</tr>
<tr>
<td>Z Top</td>
<td>specifies the position of the entry move into the material at the specified speed and feed.</td>
</tr>
<tr>
<td>Speed</td>
<td>specifies the spindle speed for the move from Z Start to Z Top.</td>
</tr>
<tr>
<td>Feed</td>
<td>specifies the plunge feedrate for the move from Z Start to Z Top.</td>
</tr>
</tbody>
</table>

![Figure 3–6. Custom Drill Entry tab](image-url)
Break Out Tab

The Break Out tab allows the user to set a feed and speed for the drill operation that differs from the feed and speed used for the move breaking through the bottom of the material (typically to avoid whipping). This step is optional and can be skipped by setting the Enable field to No. When Enable is No, the feed and speed from the Bottom tab are used for the move from Z Top to Z Bottom.

The image shows the plunge move from Z Top to Z Break Out, if enabled:

![Figure 3–7. Custom Drill Break Out tab](image)

Fields are:

- **Enable**—specifies Yes or No to enable or disable this step.
- **Z Break Out**—specifies the Z end position for the break out step at the specified feed and speed.
- **Speed**—specifies the spindle speed for the move from Z Top to Z Break Out.
- **Feed**—specifies the plunge feedrate for the move from Z Top to Z Break Out.
**Bottom Tab**

The final move through the hole from Z Break Out to Z Bottom is set in the Bottom tab. If the previous Break Out step is not enabled, then the feed and speed set in this step are used for the entire move from Z Top to Z Bottom.

The image shows the motion for the plunge move from Z Break Out to Z Bottom:

![Image showing motion from Z Break Out to Z Bottom](image)

**Figure 3–8. Custom Drill Bottom tab**

Fields are:

- **Z Bottom**—specifies the Z bottom of the finished hole.
- **Speed**—specifies the spindle speed for the move from Z Break Out to Z Bottom.
- **Feed**—specifies the plunge feedrate for the move from Z Break Out to Z Bottom.
Re-entry Tab

The Re-entry step allows the user to set a speed and feed for the retract move from Z Bottom to a Z position back inside the hole (Z Re-entry). This step is optional and can be skipped by setting the Enable field to No. When Enable is No, the move occurs from Z Bottom to the next enabled step (Out or Retract). If there are no other enabled steps, the spindle moves at rapid out of the hole to the Z start location.

The image shows the move from Z Bottom to Z Re-entry, if enabled:

Fields are:

- **Enable**—specifies Yes or No to enable or disable this step.
- **Z Re-entry**—specifies the Z end position of the re-entry move back into the hole at the specified speed and feed.
- **Speed**—specifies spindle speed for the move from Z Bottom to Z Re-entry.
- **Feed**—specifies the retract feedrate for the move from Z Bottom to Z Re-entry.
Out Tab

The Out step allows the user to set the speed and feedrate for the retract move from Z Re-entry to Z Out. This step is optional and can be skipped by setting the Enable field to No. When Enable is No, the move occurs from the last enabled step (Bottom or Re-entry) to the Retract step, if it is enabled. If the Retract step is not enabled, the spindle moves at rapid from the last enable step (Bottom or Re-entry) out of the hole to the Z start location.

The image shows the move from Z Re-entry to Z Out, if enabled:

![Figure 3–10. Custom Drill Out tab](image)

Fields are:

- **Enable**—specifies Yes or No to enable or disable this step.
- **Z Out**—specifies the Z end position of the retract move from Z Re-entry.
- **Speed**—specifies the spindle speed for the move from Z Re-entry to Z Out.
- **Feed**—specifies the retract feedrate for the move from Z Re-entry to Z Out.
**Retract Tab**

The Retract step allows the user to set the feed and speed for the move from Z Out to Z Retract. This step is optional and can be skipped by setting the Enable field to No. When Enable is No, the tool moves at rapid feedrate from the last enabled step (Bottom, Re-entry, or Out) to the Z Start position.

From Z Retract, the tool moves at rapid feedrate to the retract clearance plane; therefore, Z Retract should be less than the retract clearance plane.

The image shows the retract move from Z Out to Z Retract:

![Figure 3–11. Custom Drill Retract tab](image)

Fields are:

- **Enable**—specifies Yes or No to enable or disable this step.
- **Z Retract**—specifies the Z position of the retract move at the specified speed and feed. Z Retract can be greater than, less than, or equal to Z Start.
- **Speed**—specifies the spindle speed for the move from Z Out to Z Retract.
- **Feed**—specifies the retract feedrate for the move from Z Out to Z Retract.
Tap Operations

Select the **Tap Operations** softkey on the New Hole Operation screen.

**Tap** and **Rigid Tap** softkeys appear.

- **Tap** - programs a standard tapping sequence.
- **Rigid Tap** - programs a tapping sequence in which the same hole is tapped repeatedly with precision.

The Tap/Rigid Tap fields are defined as follows:

- **Block** — Identifies the block number for this operation. The system determines the number by the position of this data block in the program.
- **Operation** — Identifies the operation number for this function. The system determines the number by the position of this function in the data block.
- **Tool** — Identifies the tool number for this data block and enters that tool's diameter and type on this screen
- **Z Start** — Identifies the point above the part where the spindle begins to rotate.
- **Z Bottom** — Identifies the bottom of the hole.
- **Speed** — Identifies the spindle speed for the tool, calculated in Tool Setup. Entering a value here overrides the Tool Setup value for this data block.
- **Peck Depth** — Identifies the distance the tool will drill down into the part before stopping to clear out or break the chips. The Peck Depth is usually smaller than the tool's diameter. Peck Depth is used with Rigid Tap.
- **Dwell Time** — Determines the pause in seconds before the tool retracts at the bottom of a drill operation. Dwell Time is used with Rigid Tap.
- **Tool Diameter** — Contains the value entered during Tool Setup.
- **Tool Type** — Contains the type entered during Tool Setup.
- **Pitch** — Shows the distance between threads. Defined in Tool Setup for Tap tool types, this value is calculated automatically, based on values entered in Tool Setup.
- **Plunge Feed** — Identifies the rate at which the tool enters the part. This value is calculated automatically, based on values entered in Tool Setup. The formulas for this calculation are as follows:

  For imperial units, Plunge Feed = RPM / TPI.

  For metric units, Plunge Feed = RPM x Pitch.

See also Tool Setup and Feed and Speed in *Getting Started with WinMax Mill* for more information.
Bore and Ream Operations

The Bore and Ream Operation types are Bore, Ream, Bore Rapid, Bore Orient, and Ream Rapid. Motion varies according to the Bore and Ream operation type.

Use a bore when a straight and accurately round hole is needed. A boring bar usually has one or, at the most, two blades projecting out from the shank. The blades are adjustable on many types of boring bars.

Use a ream to size a previously drilled hole. A reaming tool has multiple flutes that run vertically along the shank of the tool. During reaming, the cutter follows the angle of the existing hole and cannot be used to straighten the angle of the hole.

The Bore and Ream fields are defined as follows:

- **Block**—Identifies the block number for this operation.
- **Operation**—Identifies the operation number for this function. The system determines the number by the position of this function in the data block.
- **Z Start**—Identifies the point where the spindle begins to rotate. The Z Start field is the dimension where the Z axis plunge feed rate begins.
- **Z Bottom**—Identifies the bottom of the finished hole.
- **Tool**—Identifies the tool number for this data block.
- **Plunge Feed**—Identifies the feed rate between the Z Start and the Z Bottom positions.
- **Speed (RPM)**—Identifies the speed at which the spindle rotates to drill the hole, entered in Tool Setup. Entering a value here overrides the Tool Setup value for this data block.

**Bore**

Use a bore when a straight and accurately round hole is needed. A *boring bar* is a tool with typically one or two blades projecting out from the shank. The blades on many boring bars can be adjusted to increase or decrease the size of the hole. The illustration below shows a two-bladed boring bar.

Insert the boring tool into the spindle with the cutting face in line with the tool holder’s orientation hole, opposite the spindle key. WinMax uses this oriented spindle key angle to determine the bore orient retract direction.

![Figure 3–14. Two-bladed Boring Bar with Adjustment Dial](image)

<table>
<thead>
<tr>
<th></th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Boring Bar</td>
</tr>
<tr>
<td>2</td>
<td>Shank</td>
</tr>
<tr>
<td>3</td>
<td>Blade depth adjustment dial</td>
</tr>
<tr>
<td>4</td>
<td>Replaceable blades</td>
</tr>
</tbody>
</table>

To access the Bore screen:

1. Press the Input console key to access the Input screen.
2. Select the Part Programming softkey.
3. Select the Holes softkey.
4. Select the Bore and Ream Operations Softkey softkey
5. Select the Bore softkey.
Ream

Use the Ream feature to size a previously drilled hole. Reaming clears burrs and other particles left from a drilling or boring operation. During reaming, the cutter follows the angle of the existing hole and cannot be used to straighten the angle of the hole.

A reaming tool has multiple flutes that run vertically along the shank of the tool as shown in the following illustration:

![Figure 3–15. Reaming Tool](image)

To access the Ream screen:

1. Press the Input console key to access the Input screen.
2. Select the Part Programming softkey.
3. Select the Holes softkey.
4. Select the Bore and Ream Operations Softkey softkey.
5. Select the Ream softkey.

→ Pecks are not allowed with the reaming feature.

Bore Rapid

Use Bore Rapid when the tool should retract at the Rapid Traverse feedrate programmed in General Parameters or Change Parameters.

To access the Bore Rapid screen:

1. Press the Input console key to access the Input screen.
2. Select the Part Programming softkey.
3. Select the Holes softkey.
4. Select the Bore and Ream Operations Softkey softkey.
5. Select the Bore Rapid softkey.
Bore Orient

Use Bore Orient to orient the bore at Z Bottom and move the boring bar. The cutting edge of the boring bar moves away from the surface of the bored hole so that during the next step the cutting edge will not scrape the walls of the hole. The distance the boring bar moves is set in the Bore Orient Retract field on the Holes Parameters screen.

Insert the boring tool into the spindle with the cutting face in line with the tool holder’s orientation hole, opposite the spindle key. WinMax uses this oriented spindle key angle to determine the bore orient retract direction.

To access the Bore Orient screen:

1. Press the Input console key to access the Input screen.
2. Select the Part Programming softkey.
3. Select the Holes softkey.
4. Select the Bore and Ream Operations Softkey softkey.
5. Select the Bore Orient softkey.

Ream Rapid

Use Ream Rapid when the tool should retract at Rapid Traverse feedrate programmed in General Parameters or Change Parameters.

To access the Ream Rapid screen:

1. Press the Input console key to access the Input screen.
2. Select the Part Programming softkey.
3. Select the Holes softkey.
4. Select the Bore and Ream Operations Softkey softkey.
5. Select the Ream Rapid softkey.
Back Spotface

Use the Back Spotface function to cut out the underside of holes that must be cut through the workpiece.

This motion sequence occurs with Back Spotface:

1. Rapid to Z Start minus cutter offset and set the spindle speed to a plunge speed.
2. Move to Z Plunge minus cutter offset at a closing feedrate to close the cutter.
3. Rapid to Z Bottom minus cutter offset.
4. Dwell for a specified reverse dwell amount, reverse the spindle direction and ramp to cutting speed to open the cutter.
5. Move up to Z Depth minus cutter offset at a cutting feedrate.
6. Dwell for the drill dwell time (from program parameters).
7. Rapid down a Z Clearance amount.
8. Dwell for the Reverse Dwell time and reverse the spindle direction to plunge speed to close the cutter.
9. Move to Z Retract minus cutter offset at a closing feedrate to insure the cutter is closed.
10. Rapid out of the hole.

The Back Spotface fields are defined as follows:

- **Block**—Identifies the block number for this operation. The system determines the number by the position of this data block in the program.
- **Operation**—Identifies the operation number for this function. The system determines the number by the position of this function in the data block.
- **Tool**—Identifies the tool number for this data block and enters that tool's diameter and type on this screen.
- **Z Start**—Identifies the point above the part where the spindle begins to rotate.
- **Z Plunge**—Identifies the point within the hole to which the tool feeds before rapid movement to Z Bottom. This action assures that the cutter is fully closed, preventing damage to the tool and part.
- **Z Bottom**—Identifies the bottom of the finished hole.
- **Z Depth**—Identifies the depth to which the tool feeds.
- **Z Clearance**—Determines the incremental distance to rapid the tool away from the cutting distance. At this point, the tool pauses and the spindle returns to its original direction to close the cutter.
- **Z Retract**—Determines the distance the tool feeds back up through the hole to be certain that the tool is properly closed before rapid movement to the Retract Clearance height.
- **Closing Feed**—Determines the feed at which the tool enters the hole at the top and bottom to be certain that the tool is closed.
• **Cutting Feed**—Determines the feed the tool uses while milling.
• **Plunge Speed**—Identifies the speed of the spindle with the tool closed while entering or exiting the hole.
• **Cutting Speed**—Identifies the speed of the spindle with the tool open.
• **Reverse Dwell**—Determines the number of seconds that the spindle dwells when it is reversing to give the tool time to open or close.
• **Tool Diameter**—Contains the value entered during Tool Setup.
• **Tool Type**—Contains the type entered during Tool Setup.

## Bolt Circle

For each hole data block, you must include a Locations or a Bolt Circle operation. Refer to *Locations, on page 3 - 25*. A Bolt Circle operation executes a series of equally spaced holes in a common circle. This operation allows skipping holes in the bolt circle.

All Locations programmed in a block containing a Bolt Circle are executed in addition to the Bolt Circle pattern. All operations programmed in a data block are performed at all locations specified in that block.

The **Locations** and **Bolt Circle** softkeys are not available for the first operation in a Holes block because they require prior operations to identify the type of operation, the tool, reference points, and machining information.

Move to the next Block or the next Operation by placing the cursor in the desired field and selecting the appropriate softkey (**Next Block**, **Next Hole Operation**, etc.). See also *Data Block Creation and Navigation, on page 1 - 6.*
The Bolt Circle fields are defined as follows:

- **Block**—Identifies the block number for this operation. The system determines the number by the position of this data block in the program.
- **Operation**—Identifies the operation number for this function. The system determines the number by the position of this function in the data block.
- **Number of Holes**—Determines the number of holes in the pattern, including those skipped. Up to 250 holes can be programmed in one operation.
- **X Center** and **Y Center**—Identify the X and Y coordinates for the center of the Bolt Circle.
- **Radius**—Determines the radius of the circle between the coordinate center and the center of the cutter's starting point.
- **Start Angle**—Determines the angle of the circle measured counter-clockwise from the 3 o'clock position to the first hole. This angle may be specified to the nearest one-thousandth of a degree.
- **Skip List**—Enter up to 24 positions that should be skipped (not drilled). In the example above, there are 6 holes in the pattern. If you do not want to drill holes for positions 2 and 5, enter 2 and 5 in the Skip List field.

**Locations**

For each hole data block, you must include a location, rotary location, or a bolt circle operation. Refer to *Bolt Circle, on page 3 - 24*.

- All Locations programmed in a block containing a Bolt Circle are executed in addition to the Bolt Circle pattern.
- All operations programmed in a data block are performed at all locations specified in that block.

The **Locations**, **Rotary Locations**, and **Bolt Circle** softkeys are not available for the first operation in a Holes block because they require prior operations to identify the type of operation, the tool, reference points, and machining information.

With the cursor in a Location X or Y field, the **Add Location** and **Insert Location Before** softkeys appear. Select one of these softkeys to either add or insert fields in the list for entering data. The **Delete Location** softkey appears when at least one location is in the list.

Up to 10 locations can be displayed on the screen. If there are more than 10 locations in the data block, use the CTRL key in combination with either the up or down arrow keys, the PAGE UP or PAGE DOWN keys, or the HOME or END keys to access the remaining locations. If you press ENTER, the cursor moves to the next field; if you press ENTER with the cursor in the last Location field, the cursor moves to the Block field.

Move to the next Block or the next Operation by placing the cursor in the desired field and selecting the appropriate softkey (*Next Block*, *Next Hole Operation*, etc.). See also *Data Block Creation and Navigation, on page 1 - 6*. 
The Holes Locations fields are defined as follows:

- **Block**—Identifies the block number for this operation. The system determines the number by the position of this data block in the program.
- **Operation**—Identifies the operation number for this function. The system determines the number by the position of this function in the data block.
- **X and Y**—Identify the X and Y coordinates of the location.
Bolt Circle to Holes Locations or Pattern Locations Conversion

A Bolt Circle data block can be copied or converted to either a Holes Locations or Pattern Locations block. Place the cursor in a Bolt Circle field and the Copy and Convert softkeys appear:

![Figure 3–17. Bolt Circle Hole or Pattern Locations Copy and Convert softkeys](image)

Softkeys are:

- **Copy to Hole Locations**—a Holes Locations block is created with the Bolt Circle coordinates, and is inserted before the Bolt Circle operation in the program. The Bolt Circle operation is maintained in the program. A Holes End block is inserted after the Hole Locations block.

- **Convert to Hole Locations**—a Holes Locations block is created with the Bolt Circle coordinates, and replaces the Bolt Circle operation in the program. The Bolt Circle operation is removed from the program. A Holes End block is inserted after the Hole Locations block.

- **Copy to Pattern Locations**—a Pattern Locations block is created with the Bolt Circle coordinates, and is inserted before the Holes block in the program. The Bolt Circle operation is maintained in the program. A Pattern End block is inserted after the Holes block.

- **Convert to Pattern Locations**—a Pattern Locations block is created with the Bolt Circle coordinates. The Bolt Circle operation is removed from the program, and a Holes Location block with 0.0 coordinates is inserted for a reference point. A Pattern End block is also inserted after the Holes block.
Holes to Pattern Locations Conversion

A Holes Locations data block can be copied or converted to a Pattern Locations block. Place the cursor in a field and the Copy and Convert softkeys appear:

![Figure 3–18. Holes to Pattern Locations softkeys](image)

Softkeys are:

- **Copy to Pattern Locations**—a Pattern Locations block is created with the Holes locations, and is inserted before the Holes block in the program. The Holes operation is maintained in the program. A Pattern End block is inserted after the Holes block.

- **Convert to Pattern Locations**—a Pattern Locations block is created with the Holes locations. The Holes Locations block remains in the program with 0.0 coordinates as a reference point. A Pattern End block is inserted after the Holes block.

Holes End Block

The WinMax software inserts a Holes End Block screen at the end of each Holes data block.

The fields on the Holes End Block screen are defined as follows:

- **Block**—Identifies the block number for this data block. The system determines the number by the position of this data block in the program.

- **Operation**—Identifies the operation number for this Holes data block. The system determines the number by its position in the data block.
PATTERNS OPERATIONS

Patterns operations repeat or modify a sequence of data blocks. Save programming time by duplicating part geometry to complete a part program or create multiple parts from one program.

Patterns may only be nested 10 levels deep.

These topics are discussed in this section:

- Loop Rectangular ................................................................. 4 - 2
- Loop Linear ........................................................................... 4 - 3
- Loop Angular ............................................................... 4 - 4
- Loop Rotate ................................................................. 4 - 5
- Pattern Locations .......................................................... 4 - 6
- Scale ................................................................................. 4 - 6
- Mirror Image ................................................................. 4 - 8
- Pattern End ....................................................................... 4 - 9
Loop Rectangular

This routine repeats a pattern a specified number of times along lines parallel to the X and Y axes. The original pattern is always milled at its programmed location. Always program a Pattern End data block following a Loop Rectangular block.

![Diagram of Loop Rectangular](image)

The Loop Rectangular fields are defined as follows:

- **Block**—Identifies the block number for this operation. The system determines the number by the position of this data block in the program.
- **X Number**—Determines the number of times the programmed routine will be repeated along a line parallel to the X axis.
- **Y Number**—Determines the number of times the programmed routine will be repeated along a line parallel to the Y axis.
- **X Distance**—Determines the distance between the patterns along the X axis lines. (Negative values indicate direction.)
- **Y Distance**—Determines the distance between the patterns along the Y axis lines. (Negative values indicate direction.)
Loop Linear

This routine repeats a pattern a specified number of times along a line defined in the X-Y plane. Even though the defined line of this pattern is not parallel to the X or Y axes, the original pattern is always milled at its programmed location and orientation does not change with respect to the X and Y axes. Always program a Pattern End data block following a Loop Linear block.

Figure 4–2. Loop Linear

The Loop Linear fields are defined as follows:

- **Block**—Identifies the block number for this operation. The system determines the number by the position of this data block in the program.
- **Number**—Determines the number of times the pattern will be repeated along the defined line.
- **X Distance** and **Y Distance**—Determine the distances between the repeated pattern in the direction of the X and Y axes (+ or -).
- **Angle**—Determines the angle (in degrees) between the defined line and the X positive axis. A positive value is counterclockwise (CCW) from the 3 o'clock position, and a negative value is clockwise (CW) from the 3 o'clock position.
- **Distance**—Determines the dimensional value between repeated patterns along the defined line.

If the X or Y Distance measurement is entered, the control automatically calculates the values for the Angle and Distance fields. The calculated values, which appear grayed out, may be stored by pressing the **Store Value** softkey or the ENTER key when the cursor is on the field. If you do not want to use the calculated value, type in a different value.
Loop Angular

This routine repeats a programmed pattern a specified number of times along a circular path. Pattern orientation does not change with respect to the X and Y axes. The original programmed pattern is not executed at its original position unless the routine places it at that location. The pattern is only shown in the locations specified by the routine. Always program a Pattern End data block following a Loop Angular block.

Figure 4–3. Loop Angular

The Loop Angular fields are defined as follows:

- **Block**—Identifies the block number for this operation. The system determines the number by the position of this data block in the program.
- **Number**—Determines the number of times the pattern is repeated along the specified circular path.
- **X Center** and **Y Center**—Identify the X and Y coordinates of the reference point about which the pattern is rotated.
- **Start Angle**—Identifies the angle value between the original pattern and the location of the first pattern created by this routine.
- **Rotate Angle**—Identifies the angle between the repeated patterns.
- **X Reference** and **Y Reference**—Identify the X and Y coordinates of the reference point (the only point in the pattern always on the circular path defined by this routine).
Loop Rotate

This routine repeats a pattern along a circular path. Loop Rotate moves the pattern around the X-Y center point and executes the pattern only at the programmed locations.

The original programmed pattern is not executed at its original position, unless this routine places it at that location. The pattern is only shown in the specified locations. Always program a Pattern End data block following a Loop Rotate block.

The Loop Rotate fields are defined as follows:

- **Block**—Identifies the block number for this operation. The system determines the number by the position of this data block in the program.
- **Number**—Determines the number of times the pattern will be repeated along the specified circular path.
- **X Center** and **Y Center**—Determine the X and Y coordinates used to define the circular path of the repeated pattern.
- **Start Angle**—Determines the angle value between the original pattern and the location of the first pattern created by this routine. If this location is the same as the original programmed pattern, the value in this field will be zero.
- **Rotate Angle**—Identifies the angle between the repeated patterns being executed by this routine.
Pattern Locations

Enter a Pattern Location block each time a programmed pattern is to be repeated. The pattern is repeated by offsetting the pattern with each displacement specified in the Pattern Location block. Always program a Pattern End data block following a Pattern Location block.

- **Block**—Identifies the block number for this operation. The system determines the number by the position of this data block in the program.
- **X Offset, Y Offset, and Z Offset**—Identify the location where the pattern is to be executed in the X, Y, and Z axes. If this is Location #1 and the pattern is to be at its original location, this coordinate must be zero. The value for this field is the offset for the X or Y axis.

Scale

This routine scales a programmed pattern down or up in a range of 0.100 to 10.000 (10% to 1000%) respectively. Always program a Pattern End data block following a Scale block.

![Figure 4–5. Pattern Scale](image)

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>X Axis</td>
</tr>
<tr>
<td>2</td>
<td>Y Axis</td>
</tr>
<tr>
<td>3</td>
<td>Original Pattern</td>
</tr>
<tr>
<td>4</td>
<td>Scaled Pattern</td>
</tr>
<tr>
<td>5</td>
<td>XY Reference</td>
</tr>
<tr>
<td>6</td>
<td>X Scale = 3; Y Scale = 2; Z Scale = not shown</td>
</tr>
</tbody>
</table>

The Scale fields are defined as follows:

- **Block**—Identifies the block number for this operation. The system determines the number by the position of this data block in the program.
- **X Reference, Y Reference, and Z Reference**—Determine the X, Y, and Z coordinates of the point from which the scaling will be performed.
• **X Scale, Y Scale, and Z Scale**—Determine the scaling factor from the X, Y, and Z axes. If scaling is not executed from the axis, enter 1.0000.
  
  • A value that is less than 1.0000 scales down the pattern.
  
  • A value that is greater than 1.0000 scales up the pattern.

When you use *Cutter Compensation, on page 1 - 13* in the program, the X and Y scale factors must be equal, except for these selections where the X and Y scale factors can be different:

- Circle (ON)
- Frame (ON)
- Ellipse (ON)
- Contour (ON)
- True-Type Font (ON)
- HD3 Lettering

When using Z scaling, the Peck Depth, Retract Clearance Plane, Peck Clearance Plane, and Z Safety Plane values are not affected.

Z Bottom and Z Start are the values commonly affected by Z scaling.
Mirror Image

This pattern programs a part as a mirror image of an existing programmed part. The routine can execute the mirror image alone or the original part and the mirror image. Always program a Pattern End data block following a Mirror Image block.

![Diagram of Mirror Image](image)

**Table 4–6. Mirror Image**

<p>| | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>X Axis</td>
<td>2</td>
<td>Y Axis</td>
<td>3</td>
</tr>
<tr>
<td>4</td>
<td>Angle</td>
<td>5</td>
<td>XY Point on Mirror Line</td>
<td></td>
</tr>
</tbody>
</table>

The Mirror fields are defined as follows:

- **Block**—Identifies the block number for this operation. The system determines the number by the position of this data block in the program.
- **Keep Original**—Select Yes for this field if the original will be cut as well as the mirror image. If only the mirror image is to be produced, select No for this field.
- **X and Y**—Identify the X and Y coordinates for a point on a line about which this routine occurs as measured from part zero to the mirror line.
- **Angle**—Enter the orientation in degrees of the mirror line (which passes through the X-Y dimensions above). This is measured from the 3 o'clock position. A positive value is CCW.
Pattern End

There are no fields to enter in this data block. It is a marker at the end of the sequence of data blocks used in a Pattern operation. Press the Review List softkey to view a part program summary and determine the position of the Pattern End data blocks.

There must be an equal number of Pattern blocks and Pattern End data blocks. Use the right and left arrow keys to scroll through the blocks, press the Insert Before softkey to insert a pattern End block before a displayed data block.
## SPECIAL OPERATIONS

Special data blocks can be inserted between operational data blocks to change the position of the table, automatically stop a program, or temporarily change the part program parameters or set a new part zero.

<table>
<thead>
<tr>
<th>Special Operation</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Position Data Block</td>
<td>5 - 2</td>
</tr>
<tr>
<td>Graphics On/Off</td>
<td>5 - 3</td>
</tr>
<tr>
<td>Change Parameters</td>
<td>5 - 3</td>
</tr>
<tr>
<td>Change Part Setup</td>
<td>5 - 6</td>
</tr>
<tr>
<td>Machine Function</td>
<td>5 - 7</td>
</tr>
<tr>
<td>Lube Cycle</td>
<td>5 - 7</td>
</tr>
<tr>
<td>Comment Block</td>
<td>5 - 8</td>
</tr>
<tr>
<td>Insert Block</td>
<td>5 - 8</td>
</tr>
</tbody>
</table>
Position Data Block

Use a Position data block to move the table or to stop motion so you can adjust the workpiece or to gain clearance around the tool or work piece. See also Part Setup Safety Work Region.

Use a Position data block instead of a Change Parameters data block in these instances:

- Between operations, when a clamp or raised portion of the part is higher than the programmed relative clearance.
- When cutting occurs inside a cavity and is followed by work inside another cavity in which an optimal Retract Clearance for operations in the recessed areas is below the surface of the part.
- The Z Retract is the same as the Z safety plane height in a Position data block.

Refer to *Change Parameters, on page 5 - 3* for more information.

The fields for a Position data block are defined below:

- **Block**—Identifies the block number for this operation. The system determines the number by the position of this data block in the program.
- **Tool**—Identifies the tool number for this data block and enters that tool's diameter and type on this screen.
- **X, Y, Z**—Determine the X and Y coordinates for the position to which the tool should move (after Z Top position is reached). This dimension is measured relative to Part Zero.
- **Stop**—Determines whether the program will stop after executing the current Position block. When the cursor advances to the Stop field, **Yes** and **No** softkeys appear. Press the **Yes** softkey if you want the program to pause (the spindle will pause also). Press the flashing Start Cycle button to continue with the part program. If this Position block is not a stop block, press the **No** softkey.

  ![If you want to open the CE Safety enclosure doors...](image)

  If you want to open the CE Safety enclosure doors after the Position data block executes (the Stop field is Yes), press the Machine Mode Interrupt console key, then press the Start Cycle button. The enclosure doors can be opened and the axes jogged. To resume program execution, close the enclosure doors and press the Start Cycle button.

- **Index Pulses**—Contains the number of positioning movements that the indexer head will make when rotating through an operation. It is used only when indexer equipment is attached to the machine. A new Position block must be put into the part program each time the part needs to move to a new indexer position.
- **Tool**—Identifies the number and type of tool.
Graphics On/Off

Graphics On and Graphics Off program blocks allow you hide selected operations in the graphics display. For example, setting a Graphics Off block as block 3 in a part program and Graphics On as block 10 will hide blocks 4-9 when the part is viewed using Solid or Toolpath graphics.

⇒ All blocks that are hidden by this operation are still cut when the program is run.

Change Parameters

Use this feature to change data stored within the Program Parameters screens (Refer to Program Parameters in Getting Started with WinMax Mill):

- Change Parameters (General), on page 5 - 4
- Change Parameters (Milling), on page 5 - 4
- Change Parameters (Holes), on page 5 - 4
- Change Parameters (Probing), on page 5 - 5
- Change Parameters (Performance), on page 5 - 5

When a part program is executed, the data selected in the Program Parameters are in effect until the system encounters another Change Parameters data block or until the end of the program.

When planning a change to the Retract Clearance parameter, consider the following points:

1. This parameter is defined relative to the Z Start dimension and can optimize Z axis tool motion between localized operations.

2. If all of the operations are at the same Z Start plane, there is no difference between a relative and an absolute clearance. However, when groups of cutting operations are at different Z levels on the part, programming a relative Retract Clearance can be more efficient than an absolute retract dimension. It is possible, using a relative Retract Clearance, to eliminate unnecessary tool motion and save execution time.

3. If the Z Start dimension of the next operation is different from that of the current operation, the control uses the higher of the two values for Retract Clearance before positioning the part in the X-Y axes.
4. To achieve a Retract Clearance close to the part may require the use of a Position data block. Programming a Position block causes the tool to retract to the Z Top (+) of the Safety Work Region.

Examples of areas in which a Position data block must be used rather than a simple parameter change are as follows:

- Between operations, a clamp or raised portion of the part is higher than the programmed relative clearance.
- When cutting occurs inside a cavity and is followed by work inside another cavity in which an optimal Retract Clearance for operations is below the surface of the part.

Refer to Position Data Block, on page 5 - 2 for information about when to use a Position block instead of changing parameters.

Change Parameters (General)

Use this feature to change the General Parameters for the program you are working on.

⇒ The Change Parameters (General) fields are the same as General Parameters 1 and General Parameters 2. Refer to General Parameters 1 and General Parameters 2 in Getting Started with WinMax Mill.

Refer to Position Data Block, on page 5 - 2 for information about when to use a Position block instead of changing parameters.

Change Parameters (Milling)

Use this feature to change Milling Parameters for the program you are working on.

⇒ The Change Parameters (Milling) fields are the same as Milling Parameters 1 and Milling Parameters 2. Refer to Milling Parameters 1 and Milling Parameters 2 in Getting Started with WinMax Mill for more information.

Refer to Position Data Block, on page 5 - 2 for information about when to use a Position block instead of changing parameters.

Change Parameters (Holes)

Use this feature to set the Holes Parameters for the program you are working on.

⇒ The Change Parameters (Holes) fields are the same as Holes Parameters. Refer to Holes Parameters in Getting Started with WinMax Mill.

Refer to Position Data Block, on page 5 - 2 for information about when to use a Position block instead of changing parameters.
Change Parameters (Probing)

Use this feature to set the Probing Parameters for the program you are working on.

⇒ The Change Parameters (Probing) fields are the same as Probing Parameters. Refer to Probing Parameters in *Getting Started with WinMax Mill*.

Refer to *Position Data Block, on page 5 - 2* for information about when to use a Position block instead of changing parameters.

Change Parameters (Performance)

Use this feature to change Performance Parameters for the program you are working on.

⇒ The Change Parameters (Performance) fields are defined the same as Performance Parameters. Refer to Performance Parameters in *Getting Started with WinMax Mill*.

Refer to *Position Data Block, on page 5 - 2* for information about when to use a Position block instead of changing parameters.
Change Part Setup

Use this function to change the Import Functions part zero coordinates and the Offset Z value. Change Part Setup values remain in effect until the next Change Part Setup block. The Auto screen will show the part position relative to the current part setup block. To change the part setup back to the original parameters, enter another Change Part Setup screen with the original parameters at the point in the program where the setup must be reset.

Part Zero A and Centerline A fields appear on the screen when the Rotary option is installed. When the program will be used to cut more than one part, insert another Change Part Setup data block at the end of the program to restore the original (default) part setup.

To create a new Change Part Setup data block:

1. From the Input screen, select the Part Programming softkey or icon.
2. Select the Insert Block Before softkey.
3. When the New Block screen appears, select the Miscellaneous softkey to display the Miscellaneous options shown in the softkey selections.
4. Select the Change Part Setup softkey.

The Change Part Setup fields are defined as follows:

- **Block**—identifies the block number for this operation. The system determines the number by the position of this data block in the program.
- **Part Zero X/Y/Z**—defines the X and Y locations in machines coordinates for part zero. (Part Zero Z appears only in Absolute Tool Length mode).
- **Part IV/V**—defines the first (IV) and second (V) alphabetic axis, if present.
- **Offset Z (Zero Calibration mode)**—defines the Z dimension offset for part zero. This field is usually left at 0 and the Tool Calibration field in Tool Setup is used to determine each tool's part zero.
- **Z Table Offset (Zero Calibration mode)**—specifies that part offset in Z direction relative to the table top.
- **Probe Z**—the distance from Z zero to the top of the part (i.e., the height, or Z Plane), including the reference tool or part probe in the spindle.
- **X/Y Skew (DEG)**—represents, in degrees, how far the part is from perfect alignment with the table. Refer to the “Part Skew Probing Cycles” section of this chapter for more information about X/Y skew.
- **Zone (dual-zone machines)**—specify which zone for the change part setup.
  - 1 = Zone 1 (the left-side zone when facing the machine).
  - 2 = Zone 2 (the right-side zone when facing the machine).
  - 0 = the current zone. Zone 0 can be used when dual zones are enabled but you do not need to run the program in a particular zone. Setting the Zone field to 0 will run the program in the zone that the spindle currently occupies.
Machine Function

To include this data block in a Conversational part program, perform the following steps:

1. From the New Block screen, select the Miscellaneous softkey.
2. Select the Machine Function softkey. You may include any of the functions appearing in the Machine Function list in your program.

⇒ If the Machine Function softkey appears “grayed out,” the machine does not have auxiliary machine or equipment specific M codes defined.

The following M codes appear in the M code list:

- M51—CYCLE PALLET CHANGER
- M52—M52 AUX OUTPUT #1
- M53—M53 AUX OUTPUT #2
- M54—M54 AUX OUTPUT #3
- M55—M55 AUX OUTPUT #4
- M56—CYCLE APC W/O CONFIRM
- M57—APC POS 1 W/O CONFIRM
- M58—APC POS 2 W/O CONFIRM
- M59—CHIP REMOVAL FWD
- M60—CHIP REMOVAL REV
- M61—CHIP REMOVAL OFF
- M62—M62 AUX OUT #1 OFF
- M63—M63 AUX OUT #2 OFF
- M64—M64 AUX OUT #3 OFF
- M65—M65 AUX OUT #4 OFF
- M68—WASHDOWN ENABLE
- M69—WASHDOWN DISABLE

Lube Cycle

If additional lubrication is needed, insert this block into your program to pulse the lube pump. The current lube cycle will not be affected. Check your machine manual to see if your lube reservoir/pump supports programmed inputs. If your lube pump and or reservoir is actuated by a timer, this option will not function. Most Hurco machines made after 1998 have timer actuated lube, but the Lube Cycle option is provided for backward compatibility with older machines. This option is not needed in newer machines due to higher quality components and adequate lube application.
Comment Block

Comment blocks are useful for the programmer to outline tooling requirements and part setup data. Comment blocks will display text to the operator during program execution when the block is encountered in the program. The amount of time a comment block will be visible can be programmed in the comment data block. Subsequent machining data blocks will continue to executed during this time.

When posting Conversational programs to NC, Comment Blocks explain the program code or provide information, such as setup instructions.

Comment Block fields are as follows:

- **Block**—Identifies the block number for this operation. The system determines the number by the position of this data block in the program.
- **Stop**—Allows the program operation to be paused until further input from the operator. If the Stop field is set to Yes when the program is running on the machine, the spindle goes to its Z Top position and the coolant shuts off.
- **Display Time**—The amount of time the comment will be displayed when this block is encountered during program run.
- **Line 1-10**—Enter up to 10 lines of information to display. Each line holds up to 50 characters.

Insert Block

When posting Conversational programs to NC, use an Insert Block to insert up to ten lines of NC code (Text Option) or up to ten lines of APT code (APT option).

Insert Block fields are as follows:

- **Block**—Identifies the block number for this operation. The system determines the number by the position of this data block in the program.
- **Type**—Choose either Text or APT
- **Line 1-10**—Enter up to 10 lines of text. Each line holds up to 50 characters.
NC/CONVERSATIONAL MERGE

WinMax allows you to execute an ISNC G-code (NC) program from within a Conversational part program with NC/Conversational merge. G-code programs can be used and re-used within the Conversational program.

To include this data block in a Conversational program, select the NC Program Call softkey. Use the Program Number field to initiate the NC program (program number must not exceed 6999). For programs that use variables, the Argument Type field allows you to enter argument variables (String or List) to pass to a subprogram. See WinMax Mill NC Programming for more information about variables and arguments.

To invoke a G-Code program from within a Conversational part program, the program must be loaded in Program Manager. The first line of NC code following the percent (%) sign must contain the program number preceded by the letter “O” (not a zero) or a colon (:), for example, O1234 or :1234. There cannot be any other information on this line and “O” or “:” must be the first character. The program must end with an M99 to allow other Conversational program operations after the NC program is complete, as in the following program (O number and M99 are bolded for emphasis):

Example:
%
O5085
(#1 IS THE START ANGLE)
(#2 IS THE NUMBER OF GEAR TEETH)
(#3 IS THE OUTSIDE RADIUS)
(#11 IS THE INSIDE RADIUS)
(#4 IS THE GEAR CENTER PT X COORD)
(#5 IS THE GEAR CENTER PT Y COORD)
(#6 IS THE GEAR CENTER PT Z COORD)
(#19 IS THE TOOTH TO SKIP)
(#18 IS THE TOOTH RATIO)
/
T1 M06
M03G00G21G90X0Y0Z0S1800
(VARIABLE #4006 - INCHES/METRIC)
IF[#4006EQ20]GOTO10
IF[#4006EQ21]GOTO15
N10#850=25.4
GOTO20
N15#850=1.0
N20G0X-3Y-5
Y5
X8
Y-5
#30=[360.0/#2]
#31=0
#22=[#30*#18]
#23=#30-#22
#24=#11*#850
#25=#3*#850
#26=#20*#23
X#4 Y#5
G90G00G16X#25Y#1
G01Z-.25F20.
WHILE[#31LT#2]DO250
#1=[#1+
#22]
G03G16X#25Y[#1]R#3
G01X#24Y[#1+#26]
#1=[#1+
#23]
G03X#24Y[#1-#26]R#11
G01X#25Y[#1]
G15
N200#31=#31+1
N400END250
M99
RECORD OF CHANGES

04-0116-215 rB, Nov 2012
  Revised by: H.Arle
  Approved by: D.Skrzypczak, Nov 2012

Changes
  • Updates to reflect software changes, including:
    • new probing setup fields
    • new field in Stick Lettering Along Contour

04-0116-215 rA, Aug 2012
  Revised by: H.Arle
  Approved by: D.Skrzypczak, Aug 2012

Changes
  • Updates to reflect software changes.

04-0116-210 rL, July 2012
  Revised by: H.Arle
  Approved by: D.Skrzypczak, July 2012

Changes
  • Updates to reflect software changes, including:
    • added new lettering data blocks
    • updated Mill Frame block
    • added new Pecking Retract Clearance field
<table>
<thead>
<tr>
<th>Date</th>
<th>Revision</th>
<th>Revised by</th>
<th>Approved by</th>
</tr>
</thead>
<tbody>
<tr>
<td>04-0116-210 rK, June 2012</td>
<td></td>
<td>H.Arle</td>
<td>D.Skrzypczak, June 2012</td>
</tr>
<tr>
<td></td>
<td>Changes</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• Updates to reflect software changes.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>04-0116-210 rJ, December 2011</td>
<td></td>
<td>H.Arle</td>
<td>D.Skrzypczak, December 2011</td>
</tr>
<tr>
<td></td>
<td>Changes</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• Updates to reflect software changes.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>04-0116-210 rI, October 2011</td>
<td></td>
<td>H.Arle</td>
<td>D.Skrzypczak, October 2011</td>
</tr>
<tr>
<td></td>
<td>Changes</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• Updates to reflect software changes.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>04-0116-210 rH, August 2011</td>
<td></td>
<td>H.Arle</td>
<td>D.Skrzypczak, August 2011</td>
</tr>
<tr>
<td></td>
<td>Changes</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• Added Peck Clearance Plane Holes parameter.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• Other updates to reflect software changes.</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### Changes

- Updates to Cutter Compensation section.
- Updates to reflect software changes.

---

704-0116-210rE, October 2010

**Revised by:** H.Arle  
**Approved by:** D.Skrzypczak, October 2010

**Changes**

- Update Tool Change Optimization (preliminary).
- Updates to reflect software changes.

---

704-0116-210rD, August 2010

**Revised by:** H.Arle  
**Approved by:** D.Skrzypczak, August 2010

**Changes**

- New features (Slot, Thread, Polygon, Tool Change Optimization, Serial Number) moved from Option to Conversational manual.
- Updated to reflect software changes.
Changes

- Updated to reflect software changes.

Changes

- Added Holes to Pattern Locations Conversion.
- Updated to reflect software changes.

Changes

- Added Contour Arc content.
- Updated to reflect software changes from v7.3.
INDEX

Numerics
3D Arc Data Block 2 - 61

A
Add Location softkey 3 - 25
Along Contour 2 - 36
Angle field
  Loop Linear data block 4 - 3
  Mirror Image 4 - 8
Arc Segments 1 - 10
Assure 2 - 30
Assure Minimum Length
  HD3 Serial Number Lettering Format tab 2 - 30
AutoCalc 1 - 8
Automatic Calculations 1 - 8

B
Back Spotface 3 - 23
Bidirectional field
  Swept Surface 2 - 37
Blend Arc 2 - 58
Blend Type field
  Thread Mill block 2 - 52
Block field
  Back Spotface 3 - 23
  Bolt Circle data block 3 - 25
  Bore and Ream Operations 3 - 4
  Boring and Reaming Operations 3 - 19
  Change Part Setup 5 - 6
  Comment data block 5 - 8
  Contour Arc 2 - 57
  Contour Blend Arc 2 - 58
  Contour End 2 - 63
  Ellipse data block 2 - 8
  Face Milling 2 - 3
  Gun Drill data block 3 - 9
  Helix data block 2 - 60
  Line segment 2 - 56
  Loop Angular data block 4 - 4
  Loop Linear data block 4 - 3
  Loop Rectangular data block 4 - 2
  Loop Rotate data block 4 - 5
  Mill Circle data block 2 - 5
  Mill Contour block 2 - 53
  Mill Frame data block 2 - 10
  Mirror Image 4 - 8
  Pattern Scale data block 4 - 6
  Position data block 5 - 2
  Tap Operations 3 - 18
  Bolt Circle 3 - 24
  Bolt Circle to Holes Locations Conversion 3 - 27
  Bolt Circle to Pattern Locations Conversion 3 - 27
  Bore 3 - 20
    boring bar 3 - 20
    orient 3 - 22
    rapid 3 - 21
  Bore and Ream Operations 3 - 4
  Boring and Reaming Operations 3 - 19
  Bottom Tab, Custom Drill 3 - 14
  Break Out Tab, Custom Drill 3 - 13

C
Calculations, automatic 1 - 8
Center Drill 3 - 3, 3 - 6
Change Parameters 5 - 3
Change Parameters (General) 5 - 4
Change Parameters (Holes) 5 - 4
Change Parameters (Milling) 5 - 4
Change Parameters (Performance) 5 - 5
Change Parameters (Probing) 5 - 5
Change Parameters (Program) 5 - 3
Change Part Setup 5 - 6
Character Height
  HD3 Serial Number Lettering Text tab 2 - 28
  Character Height field, HD3 Lettering 2 - 34
Character Width
  HD3 Serial Number Lettering Text tab 2 - 28
  Character Width field, HD3 Lettering 2 - 34
Circle Data Block 2 - 5
Climb Milling 1 - 16
Closing Feed field, Back Spotface 3 - 23
Comment Block 5 - 8
Constant Z level, in Swept Surface 2 - 35
Contour
  Arc 2 - 57
  Blend Arc 2 - 58
  Contour Arc 2 - 57
  Contour Blend Arc 2 - 58
  Contour Line 2 - 56
  End 2 - 63
  Line 2 - 56
  Mill Contour 2 - 53
Conventional Milling 1 - 16
Conversational Overview 1 - 1
Conversational Part Program creation 1 - 2
Corner Radius field
  Mill Polygon block 2 - 48
Mill Slot Caps tab 2 - 45
Counterbore 3 - 3, 3 - 6
Countersink 3 - 3, 3 - 6
Cusp Height
   in Swept Surface 2 - 35
CUSP HEIGHT field
   Swept Surface 2 - 39
Custom Drill 3 - 10
Cut Comp Enable field
   Thread Mill block 2 - 52
Cut Direction field
   Swept Surface 2 - 37
cutter compensation 1 - 13
Cutter Compensation Lookahead 1 - 18
Cutter Type field
   Thread Mill block 2 - 52
Cutting Feed field, Back Spotface 3 - 24
Cutting inside a cavity 5 - 4
Cutting Speed field, Back Spotface 3 - 24

D
Data block 1 - 2, 1 - 4, 5 - 3
Data Block Creation and Navigation 1 - 6
Data Origin
   HD3 Serial Number Lettering Text tab 2 - 28
Delete Location softkey 3 - 25
Delete Operation softkey 3 - 2
Direction field
   Contour Arc 2 - 57
   Contour Blend Arc 2 - 59
   Helix data block 2 - 60
   Mill Slot Geometry tab 2 - 43
   Thread Mill block 2 - 51
Display Time field, Comment data block 5 - 8
Display Tool Notes 1 - 7
Distance field, Loop Linear data block 4 - 3
Draw Along Contour 2 - 36
Draw Profile Contour 2 - 35
Drill 3 - 3, 3 - 4, 3 - 6
Drill Overview 3 - 3
Dwell field, Tap Operations 3 - 18

E
EDIT ALONG CONTOUR softkey 2 - 36
EDIT PROFILE CONTOUR softkey 2 - 35
Ellipse
   Data Block 2 - 8
Enable Blend Moves field, Mill Contour block 2 - 54
Enable field
   Custom Drill Break Out tab 3 - 13
Custom Drill Out tab 3 - 16
Custom Drill Re-entry tab 3 - 15
Custom Drill Retract tab 3 - 17
End Cap field
   Mill Slot Caps tab 2 - 44
Entry Tab, Custom Drill 3 - 12
Exit softkey
   Part Programming 1 - 3

F
Face Milling 2 - 3
Feed field
   3D Arc 2 - 61
   Contour Arc 2 - 57
   Contour Blend Arc 2 - 58
   Custom Drill Break Out tab 3 - 13
   Custom Drill Entry Tab 3 - 12
   Custom Drill Out tab 3 - 16
   Custom Drill Re-entry tab 3 - 15
   Custom Drill Retract tab 3 - 17
   Custom Tool Bottom tab 3 - 14
   Helix data block 2 - 60
   Line segment 2 - 56
Finish Diameter field
   Ellipse data block 2 - 9
   Face Milling 2 - 5
   Mill Circle data block 2 - 7
   Mill Contour block 2 - 55
Finish SFQ field
   Swept Surface 2 - 40
Finish Tool field
   Ellipse data block 2 - 8
   Mill Circle data block 2 - 5
   Mill Contour block 2 - 53
Finish Tool for a Mill Frame, Mill Circle or Mill Contour 2 - 14
Finish Type field
   Ellipse data block 2 - 9
   Face Milling 2 - 5
   Mill Circle data block 2 - 7
   Mill Contour block 2 - 55
Font field, True-Type Lettering 2 - 19
Fonts, True-Type 2 - 19, 2 - 25
Frame Data Block 2 - 10

G
g code
   NC/Conversational Merge 6 - 1
Graphics On/Off 5 - 3
Gun Drill 3 - 3, 3 - 7
H
HD3 Lettering 2 - 33
Helix 1 - 12
Helix Data Block 2 - 60
Holes Data Block 3 - 2
Holes End Block 3 - 28
Holes softkey
  Part Programming 1 - 2
Holes to Pattern Locations Conversion 3 - 28

I
icons - i
Increment
  HD3 Serial Number Lettering Text tab
  2 - 28
Index Pulses field, Position data block 5 - 2
Individual Profiles field, Swept Surface 2 - 37
Inner Diameter, Mill Polygon 2 - 47
Insert Block 5 - 8
Insert Location Before softkey 3 - 25
Insert Operation Before softkey 3 - 2
Insert Segment Before softkey 2 - 63
ISNC, in NC/Conversationl Merge 6 - 1

K
Keep Original field, Mirror Image 4 - 8

L
Lead Angle 1 - 20, 1 - 21
Lead Angle field
  Mill Circle data block 2 - 6
  Mill Contour block 2 - 54
Lead field, Helix data block 2 - 60
Lead In/Out Moves 1 - 20
Lead Length 1 - 21
  field, Mill Circle data block 2 - 6
  field, Mill Contour block 2 - 54
Leading Symbol
  HD3 Serial Number Lettering Format
  tab 2 - 30
Lettering
  HD3 2 - 33
  True-Type 2 - 33
Line Segments 1 - 9
Lines/Arcs
  Data Blocks 2 - 53
Locations 3 - 25
  Holes Locations 3 - 25
Loop Angular 4 - 4
Loop Linear blocks 4 - 3
Loop Rectangular block 4 - 2
Loop Rotate 4 - 5
Lube Cycle 5 - 7

M
Machine Function 5 - 7
Maximum Offset 1 - 17
  field, Mill Contour block 2 - 54
Mill 2 - 46
Mill Contour 2 - 53
Mill Face 2 - 3
Mill Feed field
  Ellipse data block 2 - 9
  Face Milling 2 - 4
  HD3 Lettering 2 - 34
  Mill Circle data block 2 - 6
  Mill Contour block 2 - 55
  Mill Frame data block 2 - 12
  Swept Surface 2 - 38
  True-Type Letting 2 - 19
Mill Frame data block 2 - 10
Mill Polygon 2 - 46
Mill Slot 2 - 41
Mill Thread data block 2 - 49
Mill True-Type Lettering 2 - 33
Milling
  3-D arc
    calculate centerline 2 - 62
    fields 2 - 61
    guidelines 2 - 62
    data blocks 2 - 2
  Milling softkey 2 - 3
  Part Programming 1 - 3
  Milling Type 1 - 20
Mill Type field 1 - 13
  Ellipse data block 2 - 8
  Face Milling 2 - 3
  Mill Circle data block 2 - 5
  Mill Contour block 2 - 53
  Mill Frame data block 2 - 12
  Swept Surface 2 - 39
Min Cusp Overlap (%) field, Swept Surface 2 - 37
Min Z field
  Swept Surface 2 - 38
Minimum Length
  HD3 Serial Number Lettering Format
  tab 2 - 30
Mirror Image 4 - 8
Miscellaneous softkey, Part Programming 1 - 3
multiple parts 5 - 6
NC Program Call softkey, Part Programming 1 - 3
NC/Conversational Merge 6 - 1
Next Hole Operation softkey 3 - 2
Number field
  Loop Angular data block 4 - 4
  Loop Linear data block 4 - 3
  Loop Rotate data block 4 - 5
Number of Holes field, Bolt Circle data block 3 - 25
Number Of Sides field
  Mill Polygon block 2 - 46

Offset
  maximum 1 - 17
Offset Z field
  Change Part Setup 5 - 6
Operation field
  Back Spotface 3 - 23
  Bolt Circle data block 3 - 25
  Bore and Ream Operations 3 - 4
  Boring and Reaming Operations 3 - 19
  Gun Drill data block 3 - 9
  Tap Operations 3 - 18
Orientation Angle field
  Mill Polygon block 2 - 47
Orientation field
  Mill Stick Lettering 2 - 16
  True-Type Lettering 2 - 18
  True-Type Letting 2 - 16, 2 - 18
Orientation field, Mill Thread block 2 - 50
Out Tab, Custom Drill 3 - 16
Outer Diameter, Mill Polygon 2 - 47

Parameter Change 1 - 8
parameters 5 - 3
  retract clearance 5 - 3
Part IV/V field, Change Part Setup 5 - 6
Part Programming 1 - 1, 1 - 2
  softkey 1 - 2
Part Zero X field, Change Part Setup 5 - 6
Part Zero Y field, Change Part Setup 5 - 6
Pattern End 4 - 9
Pattern Locations 4 - 6
Pattern Scale 4 - 6
Patterns Operations 4 - 1
Patterns softkey, Part Programming 1 - 3
Peck Depth field
  Bore and Ream Operations 3 - 5
  Custom Drill Tool Tab 3 - 10
  Ellipse data block 2 - 8
  Face Milling 2 - 4
  Gun Drill data block 3 - 9
  HD3 Lettering 2 - 34
  Mill Circle data block 2 - 6
  Mill Contour block 2 - 55
  Mill Frame data block 2 - 12
  Swept Surface 2 - 39
  Tap Operations 3 - 18
  True-Type Lettering 2 - 19
Peck Type field
  Bore and Ream Operations 3 - 6
  Custom Drill Tool Tab 3 - 10
  Gun Drill data block 3 - 9
Pecking Retract Clearance
  Bore and Ream Operations 3 - 5
  Custom Drill Tool Tab 3 - 10
Pitch field
  Thread Mill block 2 - 50
Pitch field, Tap Operations 3 - 18
Plunge Feed
  Bore and Ream Operations 3 - 5
Plunge Feed field
  Boring and Reaming Operations 3 - 20
  Ellipse data block 2 - 9
  Face Milling 2 - 4
  Gun Drill data block 3 - 9
  HD3 Lettering 2 - 34
  Mill Circle data block 2 - 6
  Mill Contour block 2 - 55
  Mill Frame data block 2 - 13
  Swept Surface 2 - 39
  Tap Operations 3 - 18
  True-Type Letting 2 - 19
Plunge Speed field, Back Spotface 3 - 24
Pocket Finish SFQ field, Swept Surface 2 - 40
Pocket First field, Swept Surface 2 - 37
Pocket Overlap (%) field, Swept Surface 2 - 40
Pocket Overlap field
  Ellipse data block 2 - 9
  Mill Circle data block 2 - 6
  Mill Contour block 2 - 54
  True-Type Letting 2 - 20
Pocket Rough SFQ field, Swept Surface 2 - 40
Pocket Type field
  Ellipse data block 2 - 8
  Mill Circle data block 2 - 5
  Mill Contour block 2 - 54
  Mill Frame data block 2 - 12
  Swept Surface 2 - 39
  Polygon Milling Data Block 2 - 46
Position block 5 - 4
Position data block 5 - 2
Position softkey
  Part Programming 1 - 2
Previous Segment softkey 2 - 63
Probe Z
  Change Part Setup 5 - 6
Profile Contour 2 - 35
Profile Left 1 - 17
Profile Right 1 - 17
Program# 5 - 3

R
Radius field
  Bolt Circle data block 3 - 25
  Contour Arc 2 - 57
  Contour Blend Arc 2 - 58
  Helix data block 2 - 60
  Mill Circle data block 2 - 6
  Mill Slot Geometry tab 2 - 43
  Thread Mill block 2 - 50
Ream 3 - 4, 3 - 21
  rapid 3 - 22
  reaming tool 3 - 21
Reaming Operations 3 - 19
Re-entry Tab, Custom Drill 3 - 15
  reference corner 2 - 2
  Retract Clearance 5 - 3
  Retract Tab, Custom Drill 3 - 17
  Reverse Dwell field, Back Spotface 3 - 24
  Reverse Mirrored field, Mill Contour block 2 - 55
Rigid Tap softkey 3 - 18
Roll End Point field, Swept Surface 2 - 37
Roll Start Point field, Swept Surface 2 - 37
  Rotary softkey, Part Programming 1 - 3
  Rotate Angle field
    Loop Angular data block 4 - 4
    Loop Rotate data block 4 - 5
Rough SFQ field
  Swept Surface 2 - 40

S
Scale 4 - 6
Screens 5 - 3
Segment field
  Contour Arc 2 - 57
  Contour Blend Arc 2 - 58
  Contour End 2 - 63
  Helix data block 2 - 60
  Line segment 2 - 56
  Mill Contour block 2 - 53
Segments 1 - 4
  arc 1 - 10

  line 1 - 9
programming 1 - 9
  Select New Font softkey 2 - 19, 2 - 25
  Select Tool from List 1 - 7
  Serial Number block 2 - 27
  Serial Number Stick Lettering block 2 - 27
Shape field
  Mill Slot Start tab 2 - 42
  Side Length field
    Mill Polygon block 2 - 47
  Side Length, Mill Polygon 2 - 47
  Sizing Diameter field
    Mill Polygon block 2 - 47
  Sizing Method field
    Mill Polygon block 2 - 47
  Skip List field, Bolt Circle data block 3 - 25
  Slot Milling Data Block 2 - 41
Speed (RPM) field
  Bore and Ream Operations 3 - 5
  Boring and Reaming Operations 3 - 20
  Ellipse data block 2 - 9
  Face Milling 2 - 4
  Gun Drill data block 3 - 9
  HD3 Lettering 2 - 34
  Mill Circle data block 2 - 6
  Mill Contour block 2 - 55
  Mill Frame data block 2 - 12
  Swept Surface 2 - 38
  Tap Operations 3 - 18
  True-Type Letting 2 - 19
Speed field
  Custom Drill Break Out tab 3 - 13
  Custom Drill Entry Tab 3 - 12
  Custom Drill Out tab 3 - 16
  Custom Drill Re-entry tab 3 - 15
  Custom Drill Retract tab 3 - 17
  Custom Tool Bottom tab 3 - 14
  Spindle Stop field, Gun Drill data block 3 - 9
  Spiral, in Swept Surface 2 - 35
  Spotface 3 - 3, 3 - 6
  Start Angle field
    Bolt Circle data block 3 - 25
    Loop Angular data block 4 - 4
    Loop Rotate data block 4 - 5
    Thread Mill block 2 - 50
  Start At Center field
    Thread Mill block 2 - 52
  Start at Center field, Mill Thread block 2 - 52
  Start Cap field
    Mill Slot Caps tab 2 - 44
  Starting Sequence
    HD3 Serial Number Lettering Text tab 2 - 28
Step Connect Type field, Swept Surface 2 - 37
Step Size field
  Swept Surface 2 - 39
Stick Lettering Along Contour 2 - 21
Stock Allowance field
  Swept Surface 2 - 38
Stop field
  Comment data block 5 - 8
  Position data block 5 - 2
Store Calculated Value 1 - 8
Sweep Angle
  Contour Arc block 2 - 57
Sweep Angle field
  Mill Slot Geometry tab 2 - 43
Sweep Angle field, Helix data block 2 - 60
Swept Surface 2 - 35, 2 - 37

T
Tap Operations 3 - 18
Tap softkey 3 - 18
Taper Angle field
  Thread Mill block 2 - 50
Temporary Parameter Change 1 - 8
Text
  Mill Stick Lettering 2 - 16
  True-Type Lettering 2 - 19
Text field
  HD3 Lettering 2 - 34
  True-Type Lettering 2 - 19
Thread Bottom field
  Thread Mill block 2 - 51
Thread Mill 2 - 49
Thread Milling Data Block 2 - 49
Thread Top field
  Thread Mill block 2 - 51
Tool
  5 - 3
Tool Changes 5 - 3
Tool Diameter field
  Back Spotface 3 - 24
  Boring and Reaming Operations 3 - 20
  Ellipse data block 2 - 9
  Face Milling 2 - 4
  Gun Drill data block 3 - 9
  HD3 Lettering 2 - 34
  Mill Circle data block 2 - 7
  Mill Contour block 2 - 55
  Tap Operations 3 - 18
Tool field
  Back Spotface 3 - 23
  Bore and Ream Operations 3 - 4
  Boring and Reaming Operations 3 - 19
  Custom Drill Tool Tab 3 - 10

Ellipse data block 2 - 8
Face Milling 2 - 3
Gun Drill data block 3 - 9
Mill Circle data block 2 - 5
Mill Contour block 2 - 53
Mill Frame data block 2 - 12
Position data block 5 - 2
Swept Surface 2 - 37
Tap Operations 3 - 18
True-Type Letting 2 - 19
Tool Selection List 1 - 7
Tool Tab, Custom Drill 3 - 10
Tool Type Checking 1 - 7
Tool Type field
  Back Spotface 3 - 24
  Ellipse data block 2 - 9
  Face Milling 2 - 5
  Gun Drill data block 3 - 9
  Mill Circle data block 2 - 7
  Mill Contour block 2 - 55
  Tap Operations 3 - 18
TPI field
  Thread Mill block 2 - 50
True-Type Fonts 2 - 19, 2 - 25
True-Type Lettering Along Contour 2 - 24
Type field
  Swept Surface 2 - 37
Type Of Corners field, Swept Surface 2 - 37

U
UltiPocket 2 - 2
Use Cusp Height field, Swept Surface 2 - 37

W
Width field
  Mill Slot Start tab 2 - 42

X
X Center field
  3D Arc 2 - 61
  Bolt Circle data block 3 - 25
  Contour Arc 2 - 57
  Contour Blend Arc 2 - 59
  Ellipse data block 2 - 8
  Helix data block 2 - 60
  Loop Angular data block 4 - 4
  Loop Rotate data block 4 - 5
  Mill Circle data block 2 - 6
  Mill Polygon block 2 - 46
  Mill Slot Geometry tab 2 - 43
  Thread Mill block 2 - 51
X Corner field
  Face Milling 2 - 4
  Mill Frame data block 2 - 10
X Distance field
  Loop Linear data block 4 - 3
  Loop Rectangular data block 4 - 2
X End field
  3D Arc 2 - 61
  Contour Arc 2 - 57
  Contour Blend Arc 2 - 59
  Helix data block 2 - 60
  Line segment 2 - 56
  Mill Slot Geometry tab 2 - 43
  Mill Slot Geometry tab 2 - 43
X field
  Mirror Image 4 - 8
  Position data block 5 - 2
X Length
  Mill Stick Lettering 2 - 16
  True-Type Lettering 2 - 18
X Length field
  Face Milling 2 - 4
  Mill Frame data block 2 - 10
  True-Type Lettering 2 - 16, 2 - 18
X Number field, Loop Rectangular data block 4 - 2
X Point field, 3D Arc 2 - 61
X Radius field, Ellipse data block 2 - 8
X Ref Location
  HD3 Lettering block 2 - 33
  HD3 Serial Number Lettering
  Orientation tab 2 - 29
  Mill Stick Lettering block 2 - 15
  Mill True-Type Lettering block 2 - 17
X Reference
  HD3 Serial Number Lettering Text tab 2 - 27
X Reference field
  HD3 Lettering block 2 - 33
  Loop Angular data block 4 - 4
  Mill Stick Lettering block 2 - 15
  Mill True-Type Lettering block 2 - 17
  Pattern Scale data block 4 - 6
X Scale field, Pattern Scale data block 4 - 7
X Start field
  Contour Arc 2 - 57
  Contour Blend Arc 2 - 59
  Helix data block 2 - 60
  Line segment 2 - 56
  Mill Contour block 2 - 54
  Mill Slot Start tab 2 - 42
X/Y Skew (DEG)
  Change Part Setup 5 - 6
X/Y Angle field
  Line segment 2 - 56

Mill Slot Geometry tab 2 - 43

XY Length field
  Line segment 2 - 56
  Mill Slot Geometry tab 2 - 43

Y
Y Center field
  3D Arc 2 - 61
  Bolt Circle data block 3 - 25
  Contour Arc 2 - 57
  Contour Blend Arc 2 - 59
  Helix data block 2 - 60
  Loop Angular data block 4 - 4
  Loop Rotate data block 4 - 5
  Mill Circle data block 2 - 6
  Mill Polygon block 2 - 47
  Mill Slot Geometry tab 2 - 43
  Thread Mill block 2 - 51
Y Corner field
  Face Milling 2 - 4
  Mill Frame data block 2 - 10
Y Distance field
  Loop Linear data block 4 - 3
  Loop Rectangular data block 4 - 2
Y End field
  3D Arc 2 - 61
  Contour Arc 2 - 57
  Contour Blend Arc 2 - 59
  Helix data block 2 - 60
  Line segment 2 - 56
  Mill Slot Geometry tab 2 - 43
Y field
  Mirror Image 4 - 8
  Position data block 5 - 2
Y Length field
  Face Milling 2 - 4
  Mill Frame data block 2 - 10
  True-Type Lettering 2 - 16, 2 - 18
Y Mapping
  True-Type Lettering 2 - 18
Y Number field, Loop Rectangular data block 4 - 2
Y Point field, 3D Arc 2 - 61
Y Radius field, Ellipse data block 2 - 8
Y Ref Location
  HD3 Lettering block 2 - 33
  HD3 Serial Number Lettering
  Orientation tab 2 - 29
  Mill Stick Lettering block 2 - 15
  Mill True-Type Lettering block 2 - 18
Y Reference
- HD3 Serial Number Lettering Text tab
  - 2 - 28
Y Reference field
- HD3 Lettering block 2 - 33
- Loop Angular data block 4 - 4
- Mill Stick Lettering block 2 - 16
- Mill True-Type Lettering block 2 - 18
- Pattern Scale data block 4 - 6
Y Scale field
- Pattern Scale data block 4 - 7
Y Start field
- Contour Arc 2 - 57
- Contour Blend Arc 2 - 59
- Helix data block 2 - 60
- Line segment 2 - 56
- Mill Contour block 2 - 54
- Mill Slot Start tab 2 - 42

Z
Z Bottom
- HD3 Serial Number Lettering Text tab
  - 2 - 28
Z Bottom field
- Back Spotface 3 - 23
- Bore and Ream Operations 3 - 5
- Boring and Reaming Operations 3 - 19
- Custom Tool Bottom tab 3 - 14
- Ellipse data block 2 - 8
- Face Milling 2 - 4
- Gun Drill data block 3 - 9
- HD3 Lettering 2 - 34
- Mill Circle data block 2 - 6
- Mill Contour block 2 - 55
- Mill Frame data block 2 - 10
- Mill Polygon block 2 - 48
- Mill Slot Start tab 2 - 42
- Mill Stick Lettering 2 - 16
- Tap Operations 3 - 18
- True-Type Lettering 2 - 18
- True-Type Letting 2 - 16, 2 - 18
Z Break Out field
- Custom Drill Break Out tab 3 - 13
Z Center field
- 3D Arc 2 - 61
- Contour Arc 2 - 57
Z Clearance field, Back Spotface 3 - 23
Z Depth field
- Back Spotface 3 - 23
Z End field
- 3D Arc 2 - 61
- Contour Arc 2 - 57
- Helix data block 2 - 60
- Line segment 2 - 56
Z field, Position data block 5 - 2
Z Out field
- Custom Drill Out tab 3 - 16
Z Plunge field, Back Spotface 3 - 23
Z Plunge Start field
- Thread Mill block 2 - 51
Z Point field, 3D Arc 2 - 61
Z Re-entry field
- Custom Drill Re-entry tab 3 - 15
Z Reference field, Pattern Scale data block
  - 4 - 6
Z Retract field
- Custom Drill Retract tab 3 - 17
Z Retract field, Back Spotface 3 - 23
Z Roughing field, Swept Surface 2 - 39
Z Scale field, Pattern Scale data block 4 - 7
Z Start
- HD3 Serial Number Lettering Text tab
  - 2 - 28
Z Start field
- Back Spotface 3 - 23
- Bore and Ream Operations 3 - 5
- Boring and Reaming Operations 3 - 19
- Contour Arc 2 - 57
- Custom Drill Entry Tab 3 - 12
- Ellipse data block 2 - 8
- Face Milling 2 - 4
- Gun Drill data block 3 - 9
- HD3 Lettering 2 - 34
- Helix data block 2 - 60
- Line segment 2 - 56
- Mill Circle data block 2 - 6
- Mill Contour block 2 - 55
- Mill Frame data block 2 - 10
- Mill Polygon block 2 - 48
- Mill Slot Start tab 2 - 42
- Mill Stick Lettering 2 - 16
- Swept Surface 2 - 37
- Tap Operations 3 - 18
- True-Type Lettering 2 - 18
- True-Type Letting 2 - 16, 2 - 18
Z Start parameter 5 - 3
Z Table Offset
- Change Part Setup 5 - 6
Z Top field, Gun Drill data block 3 - 9
Z Top field
- Custom Drill Entry Tab 3 - 12
Z Top field
- Gun Drill data block 3 - 9
Zone field
- Change Part Setup 5 - 6