WINMAX MILL OPTIONS

Dual Screen and Max Consoles
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
# TABLE OF CONTENTS

WinMax Mill Options

Documentation Conventions .................................................. xiii
  Console Buttons and Keys .................................................. xiii
  Icons ........................................................................ xiii

Programming and Operation Information ............................... xv
  Using the On-screen Help ...................................................... xv
  Printing the Programming Manuals ...................................... xvi

3D Mold .............................................................................. 1
  3D Mold Parameters .......................................................... 1
  3D Mold Contour ............................................................... 9
  3D Mold Line ................................................................. 10
  3D Mold Arc .................................................................. 11
  3D Mold Blend Arc ........................................................... 12
  Roughing and Finishing Tools .......................................... 13
  Roughing and Finishing Passes ......................................... 16

DXF Option .......................................................................... 2
  DXF Overview ................................................................. 2
  DXF Build Data Block ....................................................... 3
  DXF Parameters ............................................................... 5
  Zoom Window .................................................................... 5
  Edit Drawing ..................................................................... 6
    DXF Edit Modify - Arc .................................................... 8
    DXF Edit Modify - Line ................................................... 8
    DXF Edit Modify - Point ................................................ 8
  DXF Layers ....................................................................... 9

Helical Plunge Option .......................................................... 3
  Helical Plunge Milling Parameter Fields .............................. 2
  Helical Plunge (Inside/Outside) for Mill Frames, Mill Circles and Ellipses ......................................................... 3
  Helical Plunge with UltiPocket ......................................... 4
  Helical Plunge with Operator Specified Location .................. 4
  Helical Plunge in the Center of a Pocket ......................... 4
  Helical Plunge with Outward Pocketing ............................. 4
  Helical Plunge of Mill Frame Inside with No Pecking and Blend Offset ............................................................. 5
  Helical Plunge of Mill Frame Inside with Pecking and Straight Plunge Finish Pass and Blend Offset .............................. 6
  Helical Plunge with Lines and Arcs ................................... 8
    Helical Plunge with 3-D Part Programming Option ............. 8

Insert Pockets ...................................................................... 4
  Mill Triangle Data Block ................................................... 2
  Mill Diamond Data Block ................................................... 4
  Mill Hexagon Data Block .................................................... 7

Rotary .................................................................................. 5
  Rotary Overview .............................................................. 2
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Transform Plane</td>
<td>5 - 5</td>
</tr>
<tr>
<td>Rotary Part Setup</td>
<td>5 - 8</td>
</tr>
<tr>
<td>Rotary Centerline</td>
<td>5 - 8</td>
</tr>
<tr>
<td>Rotary Part Programming</td>
<td>5 - 10</td>
</tr>
<tr>
<td>Rotary New Block</td>
<td>5 - 10</td>
</tr>
<tr>
<td>Rotary Position Block</td>
<td>5 - 11</td>
</tr>
<tr>
<td>Rotary Milling New Block</td>
<td>5 - 12</td>
</tr>
<tr>
<td>Rotary Patterns</td>
<td>5 - 12</td>
</tr>
<tr>
<td>Rotary Parameters</td>
<td>5 - 13</td>
</tr>
<tr>
<td>Rotary A and Rotary A Tilt B Configuration</td>
<td>5 - 14</td>
</tr>
<tr>
<td>Rotary Position</td>
<td>5 - 14</td>
</tr>
<tr>
<td>Rotary Lines and Arcs</td>
<td>5 - 14</td>
</tr>
<tr>
<td>Rotary Circle</td>
<td>5 - 16</td>
</tr>
<tr>
<td>Rotary Frame</td>
<td>5 - 16</td>
</tr>
<tr>
<td>Rotary Patterns</td>
<td>5 - 17</td>
</tr>
<tr>
<td>Rotary Parameters</td>
<td>5 - 19</td>
</tr>
<tr>
<td>Tilt A Rotary C Configuration</td>
<td>5 - 20</td>
</tr>
<tr>
<td>Rotary Position</td>
<td>5 - 20</td>
</tr>
<tr>
<td>Rotary Lines and Arcs</td>
<td>5 - 20</td>
</tr>
<tr>
<td>Rotary Circle</td>
<td>5 - 22</td>
</tr>
<tr>
<td>Rotary Frame</td>
<td>5 - 22</td>
</tr>
<tr>
<td>Rotary Patterns</td>
<td>5 - 23</td>
</tr>
<tr>
<td>Rotary Parameters</td>
<td>5 - 25</td>
</tr>
<tr>
<td>Rotary B Configuration</td>
<td>5 - 26</td>
</tr>
<tr>
<td>Rotary Position</td>
<td>5 - 26</td>
</tr>
<tr>
<td>Rotary Milling</td>
<td>5 - 26</td>
</tr>
<tr>
<td>Rotary Patterns</td>
<td>5 - 26</td>
</tr>
<tr>
<td>Rotary Parameters</td>
<td>5 - 28</td>
</tr>
<tr>
<td>Tilt B Rotary C Configuration</td>
<td>5 - 29</td>
</tr>
<tr>
<td>Rotary Position</td>
<td>5 - 29</td>
</tr>
<tr>
<td>Rotary Lines and Arcs</td>
<td>5 - 29</td>
</tr>
<tr>
<td>Rotary Circle</td>
<td>5 - 31</td>
</tr>
<tr>
<td>Rotary Frame</td>
<td>5 - 31</td>
</tr>
<tr>
<td>Rotary Patterns</td>
<td>5 - 32</td>
</tr>
<tr>
<td>Rotary Parameters</td>
<td>5 - 34</td>
</tr>
<tr>
<td>Universal Rotary Configuration</td>
<td>5 - 35</td>
</tr>
<tr>
<td>Rotary Position Block</td>
<td>5 - 35</td>
</tr>
<tr>
<td>Rotary Lines and Arcs</td>
<td>5 - 36</td>
</tr>
<tr>
<td>Rotary Circle</td>
<td>5 - 38</td>
</tr>
<tr>
<td>Rotary Frame</td>
<td>5 - 38</td>
</tr>
<tr>
<td>Rotary True Type Font</td>
<td>5 - 40</td>
</tr>
<tr>
<td>Rotary Slot</td>
<td>5 - 41</td>
</tr>
<tr>
<td>Rotary Polygon</td>
<td>5 - 43</td>
</tr>
<tr>
<td>Rotary Stick Lettering</td>
<td>5 - 45</td>
</tr>
<tr>
<td>Rotary Patterns</td>
<td>5 - 46</td>
</tr>
<tr>
<td>Calculating X- and Z-Axis Positions After a Tilt-Axis Move</td>
<td>5 - 49</td>
</tr>
<tr>
<td>Part Zero Calibrated at Zero Degree Vertical</td>
<td>5 - 49</td>
</tr>
<tr>
<td>Part Zero Calibrated at Zero Degree Horizontal</td>
<td>5 - 50</td>
</tr>
<tr>
<td>Example Programs</td>
<td>5 - 51</td>
</tr>
<tr>
<td>Rotary Programming Examples</td>
<td>5 - 51</td>
</tr>
<tr>
<td>Tilt-Axis Programming Examples</td>
<td>5 - 60</td>
</tr>
<tr>
<td>Transform Plane Example Programs</td>
<td>5 - 62</td>
</tr>
</tbody>
</table>
# Table of Contents

**Post Processor (Desktop Only)** ............................................. 6 -  1

**Part Probing Option** .......................................................... 7 -  1
- Part Setup—Part Probe Parameters ........................................ 7 -  2
- Part Probe Calibration and Cycles .......................................... 7 -  5
  - Part Probe Deflection Offset Calibration .............................. 7 -  5
  - Conversational Part Probing Cycles ...................................... 7 -  9
- Part Quality Verification ..................................................... 7 - 46
  - Part Inspection ............................................................... 7 - 46

**Tool Probing Option** .......................................................... 8 -  1
- Tool Probing in Absolute Tool Length Mode .............................. 8 -  2
  - Set the Probing Parameters—Absolute Tool Length Mode .......... 8 -  2
  - Run the Probe Cycle ....................................................... 8 -  5
- Tool Probing in Zero Calibration Mode ................................... 8 -  8
  - Set the Probing Parameters—Zero Calibration mode ............... 8 -  8
  - Determine Probe Z .......................................................... 8 -  9
  - Run the Probe Cycle ....................................................... 8 - 11
- Tool Quality Monitoring ..................................................... 8 - 14
  - Probe Tool Monitoring Data Block ...................................... 8 - 14
  - Automatic Tool Monitoring Parameter .................................. 8 - 18
- Appendix A: Tool Probe Setup .............................................. 8 - 19
  - Touch Probe Parameters .................................................. 8 - 19
  - Laser Probe Parameters .................................................. 8 - 21
  - Probe Deflection Offset Calibration .................................... 8 - 26
- Appendix B: Tool Probe Calibration ........................................ 8 - 29
  - Probe Calibration—Absolute Tool Length mode ...................... 8 - 29
  - Probe Calibration—Zero Calibration mode ............................. 8 - 30
- Appendix C: Probing Parameter Definitions .............................. 8 - 33

**Tool Change Optimization** .................................................. 9 -  1
- Using Tool Change Optimization ............................................ 9 -  1
- Tool Change Review ........................................................... 9 -  5
- Hole Blocks and Tool Change Optimization .............................. 9 -  6

**Tool Fixture (TPS) Option** .................................................. 10 -  1
- Tool Fixture Overview ....................................................... 10 -  2
- Automatic Tool Removal Using TPS ........................................ 10 -  3
- Automatic Tool Change Using TPS .......................................... 10 -  4
- Bypass TPS in an Automatic Tool Change ................................. 10 -  5

**UltiMotion** ........................................................................ 11 -  1
- How to Select the Motion System ........................................... 11 -  2

**UltiMonitor** ........................................................................ 12 -  1
- LAN Requirements ............................................................... 12 -  2
- Limitations for UltiMonitor .................................................... 12 -  2
- Glossary of Networking Terms ............................................... 12 -  2
- Configuring a Network .......................................................... 12 -  4
  - Configuring an IP address for your machine .......................... 12 -  4
  - Configuring the Computer and Workgroup Names ................... 12 -  6
  - Mapping a Network Drive ................................................... 12 -  7
- Using FTP ............................................................................ 12 -  9
  - FTP Server Settings ........................................................... 12 -  9
  - FTP Manager ...................................................................... 12 - 11
<table>
<thead>
<tr>
<th>Section</th>
<th>Page(s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Extended Shop Floor (ESF)</td>
<td>12-13</td>
</tr>
<tr>
<td>UltiPockets Option</td>
<td>13-1</td>
</tr>
<tr>
<td>Pocket Boundary</td>
<td>13-2</td>
</tr>
<tr>
<td>Spiral Outward - No Islands</td>
<td>13-2</td>
</tr>
<tr>
<td>Spiral Inward</td>
<td>13-2</td>
</tr>
<tr>
<td>Programming Islands</td>
<td>13-2</td>
</tr>
<tr>
<td>Mill Contours</td>
<td>13-2</td>
</tr>
<tr>
<td>Mill Frame</td>
<td>13-3</td>
</tr>
<tr>
<td>Mill Circle</td>
<td>13-3</td>
</tr>
<tr>
<td>Pattern</td>
<td>13-3</td>
</tr>
<tr>
<td>Helical Plunge with UltiPocket Option</td>
<td>13-4</td>
</tr>
<tr>
<td>Helical Plunge Using Operator Specify Pocket Start</td>
<td>13-5</td>
</tr>
<tr>
<td>Record of Changes</td>
<td>1</td>
</tr>
<tr>
<td>Index</td>
<td>IX-1</td>
</tr>
</tbody>
</table>
LIST OF FIGURES

Figure 1–1. Increased Radius ............................................................... 1 - 3
Figure 1–2. Zero Radius .................................................................. 1 - 3
Figure 1–3. Convex Contour Below the Part Surface ....................... 1 - 4
Figure 1–4. Counterclockwise and Clockwise Motion ...................... 1 - 4
Figure 1–5. Y Start and Y End Fields (XZ Translated in Y) ............... 1 - 5
Figure 1–6. Bidirectional field .......................................................... 1 - 6
Figure 1–7. Step Size ..................................................................... 1 - 7
Figure 1–8. Peck Depth .................................................................. 1 - 7
Figure 1–9. Stock Allowance ............................................................. 1 - 8
Figure 1–10. Flat End Mill on a Contour ......................................... 1 - 13
Figure 1–11. Ball-Nosed End Mill on a Contour ............................. 1 - 13
Figure 1–12. Flat End Mill ............................................................... 1 - 14
Figure 1–13. Ball-Nosed End Mill .................................................. 1 - 15
Figure 1–14. Roughing and Finishing Passes ................................. 1 - 16

Figure 2–1. Extended Lines and Arcs ............................................. 2 - 6
Figure 2–2. Joined Lines and Arcs .................................................. 2 - 7

Figure 3–1. Helical Plunge with No Pecking and Blend Offset (Isometric View) 3 - 5
Figure 3–2. First Peck (Isometric View) .......................................... 3 - 6
Figure 3–3. Second Peck (Isometric View) ....................................... 3 - 7
Figure 3–4. Finish Pass (Isometric View) ........................................ 3 - 7
Figure 3–5. Helical Plunging with Lines/Arcs (Isometric View) ......... 3 - 8

Figure 4–1. Insert Pockets Softkeys .............................................. 4 - 1
Figure 4–2. Triangle Programming Diagram .................................. 4 - 2
Figure 4–3. Diamond 1 Face Diagram ........................................... 4 - 4
Figure 4–4. Diamond 2 Faces Diagram .......................................... 4 - 5
Figure 4–5. Hexagon Diagram ....................................................... 4 - 7

Figure 5–1. Axes Movement ............................................................. 5 - 2
Figure 5–2. Rotary Centerline .......................................................... 5 - 9
Figure 5–3. Rotary Centerline .......................................................... 5 - 9
Figure 5–4. Universal Rotary Frame Geometry tab ....................... 5 - 38
Figure 5–5. Universal Rotary Frame Corners tab ........................... 5 - 39
Figure 5–6. Universal Rotary Frame example ............................... 5 - 39
Figure 5–7. Y Mapping ................................................................. 5 - 40
Figure 5–8. Rotary Stick Lettering screen ...................................... 5 - 45
Figure 5–9. Vertical Tilt-Axis Example .......................................... 5 - 49
Figure 5–10. Horizontal Tilt-Axis Example ..................................... 5 - 50
Figure 5–11. Rotary Mill Frame Part Drawing (cylinder shown for reference) 5 - 51
Figure 5–12. Threading Part Drawing (cylinder shown for reference) 5 - 52
Figure 5–13. Single Hole Part Drawing (cylinder shown for reference) 5 - 54
Figure 5–14. Rotary Pattern Loop Part Drawing (cylinder shown for reference) 5 - 55
Figure 5–15. 4-axis Part Created Using Transform Plane ................ 5 - 57
Figure 5–16. Bolt Holes on Tilted Cylindrical Part ......................... 5 - 60
Figure 5–17. Cylindrical Part Tilted 45° Between Holes ................. 5 - 61
Figure 5–18. Same Part Zero for Transformed Plane, VMX42 SR Machine 5 - 62
Figure 5–19. Part Created on VMX42SR Machine, Using Transform Plane 5 - 64
Figure 6–1. Utilities screen with Post Processor softkey .......................... 6 - 1
Figure 6–2. Post Processor Configuration screen ....................................... 6 - 1

Figure 8–1. Tool Probing screen in Absolute Tool Length mode ............... 8 - 2
Figure 8–2. Tool Probing Cycle Defaults screen in Absolute Tool Length mode . 8 - 4
Figure 8–3. Tool Probing screen in Zero Calibration mode ......................... 8 - 8
Figure 8–4. Tool Probing Cycle Defaults screen in Zero Calibration mode .... 8 - 10
Figure 8–1. Touch Probe Parameters ......................................................... 8 - 19
Figure 8–2. Laser Probe Parameters ......................................................... 8 - 21
Figure 8–3. Typical Laser Probe Calibration Tool ...................................... 8 - 23
Figure 8–4. Typical Laser Probe Calibration Tool Motion ......................... 8 - 25
Figure 8–5. Tool Probe Deflection Offsets ................................................. 8 - 26
Figure 8–6. Tool Measurement Screen ..................................................... 8 - 30
Figure 8–7. Tool Setup Probing Parameters in Zero Calibration mode ........ 8 - 30

Figure 9–1. Tool Change Optimization parameter in Program Parameters ...... 9 - 1
Figure 9–2. Tool Change Optimization On block ....................................... 9 - 2
Figure 9–3. Placement of Tool Change Optimization blocks .................... 9 - 4
Figure 9–4. Tool Change Review Screen ................................................... 9 - 5

Figure 10–1. Tool Fixture ........................................................................... 10 - 2

Figure 12–1. Default LAN and Internet Protocol Properties ..................... 12 - 5
Figure 12–2. Computer Name Changes in System Properties .................. 12 - 6
Figure 12–3. Map Network Drive dialog ................................................... 12 - 7
Figure 12–4. FTP Server Settings screen .................................................. 12 - 9
Figure 12–5. FTP Host List screen ............................................................ 12 - 11
Figure 12–6. FTP Host Properties screen .................................................. 12 - 11
### List of Tables

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5-1</td>
<td>Rotary Configurations Available on Hurco Machines</td>
<td>3</td>
</tr>
<tr>
<td>5-1</td>
<td>Rotary Mill Frame Part Program Summary</td>
<td>52</td>
</tr>
<tr>
<td>5-2</td>
<td>Threading Part Program Summary</td>
<td>53</td>
</tr>
<tr>
<td>5-3</td>
<td>Hole Part Program Summary</td>
<td>54</td>
</tr>
<tr>
<td>5-4</td>
<td>Rotary Pattern Loop Part Program Summary</td>
<td>56</td>
</tr>
<tr>
<td>5-5</td>
<td>4-Axis Transform Plane Program Summary</td>
<td>59</td>
</tr>
<tr>
<td>5-6</td>
<td>Bolt Holes on Tilted Cylindrical Part Program Summary</td>
<td>60</td>
</tr>
<tr>
<td>5-7</td>
<td>Cylindrical Part Tilted 45° Between Holes Part Program Summary</td>
<td>61</td>
</tr>
<tr>
<td>5-8</td>
<td>Same Part Zero for Transformed Plane Part Program Summary, VMX42SR Machine</td>
<td>63</td>
</tr>
</tbody>
</table>
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 - xvi 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.

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.
3D MOLD

These topics are discussed in this section:

3D Mold Parameters ................................................................. 1 - 2
3D Mold Contour ............................................................... 1 - 9
3D Mold Line .......................................................... 1 - 10
3D Mold Arc .......................................................... 1 - 11
3D Mold Blend Arc .......................................................... 1 - 12
Roughing and Finishing Tools .................................................. 1 - 13
Roughing and Finishing Passes .................................................. 1 - 16
3D Mold Parameters

To create a three-dimensional (3D) part, define a two-dimensional (2D) profile in either the XY or XZ plane. Repeat the 2D profile along a straight line (translate) or repeat it around a centerline (revolve) to produce the final 3D shape. Choose Draw 2D Contour to draw the original 2D contour that will be manipulated using the 3D operations.

To program a 3D Mold data block from the Part Programming screen, select the Insert Block Before softkey then select the Milling Softkey that appears. On the Milling softkeys, select 3D Mold.

Combine of any of the three types into composite contours to machine complex parts:

- **Y Revolved about X**—Use a 2D contour programmed in the XY plane and revolve it about a centerline on the X axis to produce the finished 3D contour.
- **XZ Revolved about Z**—Use a 2D contour programmed in the XZ plane and revolve it about a centerline on the Z axis to produce the finished 3D contour.
- **XZ Translated in Y**—Use a 2D contour programmed in the XZ plane and translate it in the Y axis.

Access the 3D Mold Contour screens by selecting the Edit 3D Mold Contour softkey. This softkey is not available when the cursor is in the Block field. When you select the Edit 3D Mold Contour softkey, it changes to Edit 3D Mold Parameters so you can return to the parameters screen. The Edit 3D Mold Parameters softkey is not available when the cursor is in either the Block or Segment field.

The 3D Mold Parameter 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 finish tool number for this data block and enters that tool's diameter and type on this screen.
- **Type**—Defines the type of 3D operation. There are four drop-down list box and softkey choices when the cursor is on the Type field.
  - **Draw 2D Contour**
  - **Y Revolved about X**—Use a 2D contour programmed in the XY plane and revolve it about a centerline on the X axis to produce the finished 3D contour.
  - **XZ Revolved about Z**—Use a 2D contour programmed in the XZ plane and revolve it about a centerline on the Z axis to produce the finished 3D contour.
  - **XZ Translated in Y**—Use a 2D contour programmed in the XZ plane and translate it in the Y axis.
- **Centerline Y** and **Centerline Z**—Determine the coordinate points of the center of the part on the Y and Z axes.

![Figure 1–1. Increased Radius](image1)

<table>
<thead>
<tr>
<th></th>
<th>Y Centerline</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>5</td>
</tr>
</tbody>
</table>

Figure 1–1. Increased Radius

![Figure 1–2. Zero Radius](image2)

<table>
<thead>
<tr>
<th></th>
<th>Y Centerline</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>0</td>
</tr>
</tbody>
</table>

Figure 1–2. Zero Radius
The Centerline Z field determines the Z axis position of the X axis centerline. Changing the Z axis centerline moves the X axis centerline above or below the part surface. This alters the depth of the 3D contour. The Z axis centerline is used only for XY Revolved About X.

To machine the 3D contour below the part surface, enter a negative value in the Centerline Z field. This value is equal to the radius of the part measured from the Y centerline.

Here is a convex contour programmed below the part surface:

![Convex Contour Below the Part Surface](image)

**Figure 1–3. Convex Contour Below the Part Surface**

- **Start Angle** and **End Angle**—Determine the starting and ending values of the angle of revolution for XY Revolved About X and XZ Revolved about Z. When determining Start and End Angles, remember that 0º is where the contour begins and is located at the 3 o’clock position.
- The difference between the start and end angle determines the degrees that the 2D profile revolves about the axis.
- Start and End angles can be entered as positive or negative numbers. CCW motion is programmed as a positive number; CW motion is programmed as a negative number:

![Counterclockwise and Clockwise Motion](image)

**Figure 1–4. Counterclockwise and Clockwise Motion**
• **Y Start** and **Y End**—Determine the length of the 3D contour along the Y axis for XZ Translated in Y, as shown in the example below:

![Diagram of Y Start and Y End fields](image)

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Part Zero</td>
</tr>
<tr>
<td>2</td>
<td>Y Start</td>
</tr>
<tr>
<td>3</td>
<td>Y End</td>
</tr>
</tbody>
</table>

*Figure 1–5. Y Start and Y End Fields (XZ Translated in Y)*

• **Cut Direction**—Controls the tool path while the 3D contour is machined. There are two choices for the Cut Direction field:
  - **With Contour** - machines the 3D contour using the tool path originally programmed.
  - **Normal** - tool path follows the part at right angles to the original two-dimensional profile.
- **Bidirectional**—Determines the direction of the tool path while the part is being machined. There are two choices for the Bidirectional field:
  - **No** - causes the tool to machine in one direction, based on the direction of the contour definition.
  - **Yes** - causes the tool to machine in both directions without retracting the tool until the entire contour is complete.

![Diagram](image)

*Figure 1–6. Bidirectional field*
- **Step Size**—Determines the distance 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.

![Figure 1–7. Step Size](image)

- **Finish Step Size**— Determines the distance between cutter passes for the finish tool.
- **Min Z**—Limits negative Z motion to the Centerline Z value when set to Yes.
- **Z Start**—Identifies the point above the part where the spindle begins to rotate.
- **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.

![Figure 1–8. Peck Depth](image)
• **Plunge Feed**—Identifies the rate at which the tool initially enters the part.

• **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.

• **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.

• **Stock Allowance**—Leaves or removes extra material on the surface of the 3D contour. 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 3D contour. A positive stock allowance value programmed using a Flat End Mill leaves sufficient material for a finishing pass.

![Diagram](image)

<table>
<thead>
<tr>
<th></th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Print Dimension</td>
</tr>
<tr>
<td>2</td>
<td>+ Stock Allowance</td>
</tr>
<tr>
<td>3</td>
<td>- Stock Allowance</td>
</tr>
</tbody>
</table>

*Figure 1–9. Stock Allowance*

• **Tool Diameter**—Contains the tool diameter entered for the tool during Tool Setup.

• **Tool Type**—Contains the tool type entered for the tool during Tool Setup.

• **Finish Diameter**—Contains the tool diameter entered for the finish tool during Tool Setup.

• **Finish Tool Type**—Contains the tool type entered for the finish tool during Tool Setup.
3D Mold Contour

For Swept Surface programming, see *Swept Surface* in the Milling chapter of the Conversational Part Programming manual.

Used in conjunction with 3D Mold Parameters to mill a 3D Mold, program the part surfaces as a 2D profile in either the XY or XZ plane.

The Start Segment number is always 0. Use segments to program lines and arcs which create a contour. Repeat the 2D profile along a straight line (translate) or repeat it around a centerline (revolve) to produce the final 3D shape. The Contour End block marks the end of the programmed contour.

Select the Edit 3D Mold Parameters softkey to access the parameters screen. The **Edit 3D Mold Parameters** softkey is not available when the cursor is in either the Block or Segment field. When you select the **Edit 3D Mold Parameters** softkey, it changes to **Edit 3D Mold Contour**. This softkey is not available when the cursor is in the Block field.

The 3D Mold Start Segment fields are defined as follows:

- **Block**—Identifies the block number for the 3D Mold data block. The system determines the number by the position of this data block in the program.
- **Segment**—Identifies the segment number for this operation.
- **X Start** and **Y Start** or **X Start** and **Z Start**—Identify the starting location for the X and Y or X and Z coordinates. The X and Y or X and Z coordinates are carried forward to the next Segment.

Continue programming the contour by using the Page Down key or by selecting the **Next Segment** softkey.

Line, Arc, and Blend Arc softkey choices appear.

WinMax allows you to paste a contour into a 3D Mold Block on the Program Review screen. This allows you to use the same segments for a Mill Contour Block and a 3D Mold Block without entering each segment twice.

- To copy a contour, select the contour that you wish to copy on the Program Review Screen and use the MULTIPLE BLOCK FUNCTIONS softkey to access the COPY softkey.
- To paste a contour, select the 3D Mold Block that you wish to paste the contour into on the Program Review screen and use the MULTIPLE BLOCK FUNCTIONS softkey to access the PASTE softkey.
3D Mold Line

For Swept Surface programming, see Swept Surface in the Milling chapter of the Conversational Part Programming manual.

Some 3D Mold Line fields are automatically calculated with the Auto-Calc feature.

The 3D Mold Line fields are defined as follows:

- **Block**—Identifies the block number for the 3D Mold data block. The system determines the number by the position of this data block in the program.

- **Segment**—Identifies the segment number for this operation. The system determines the number by the position of this segment in the program.

The following fields change depending on the type of contour selected:

- **X End and Y End or X End and Z End**—Identify the X End and Y End or X End and Z End coordinates. If two End coordinates are entered (X/Y or X/Z), the control automatically calculates the XY (or XZ) Length and the XY (or XZ) Angle fields. Use the **Store Calculated Value** softkey to retain the calculated value.

- **XY Length or XZ Length**—Identify the XY Length or XZ Length. If two End coordinates are entered (X End and Y End or X End and Z End), the control automatically calculates the XY Length and the XY Angle fields. Use the **Store Calculated Value** softkey to retain the calculated value.

- **XY Angle or XZ Angle**—Identify the XY Angle or XZ Angle or the angle of the line segment (from the start point to the end point), measured counterclockwise from the 3 o'clock position. If two End coordinates are entered (X End and Y End or X End and Z End), the control automatically calculates the XY Length and the XY Angle fields. Use the **Store Calculated Value** softkey to retain the calculated value.

- **X Start and Y Start or X Start and Z Start**—Define the starting points of this segment. The Start fields are carried forward from the previous segment's end points.

Continue programming the contour by using the PAGE DOWN key or by selecting the **Next Segment** softkey.

Line, Arc, and Blend Arc softkey choices appear.
3D Mold Arc

For Swept Surface programming, see Swept Surface in the Milling chapter of the Conversational Part Programming manual.

Some 3D Mold Arc fields are automatically calculated with the Auto-Calc feature.

The 3D Mold Arc fields are defined as follows:

- **Block**—Identifies the block number for the 3D Mold data block. The system determines the number by the position of this data block in the program.

- **Segment**—Identifies the segment number for this operation. The system determines the number by the position of this segment in the program.

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

The following fields change depending on the type of contour selected:

- **X End** and **Y End** or **X End** and **Z End**—Identify data coordinates (values for X End and Y End or X End and Z End) used in the automatic calculations. Use the **Store Calculated Value** softkey to retain the calculated value.

- **X Center** and **Y Center** or **X Center** and **Z Center**—Identify data coordinates (values for X Center and Y Center or X Center and Z Center) used in the automatic calculations. Use the **Store Calculated Value** softkey to retain the calculated value.

- **Radius**—Identifies the value for the Radius. The radius is used in the automatic calculations. Use the **Store Calculated Value** softkey to retain the calculated value.

- **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°.

- **X Start** and **Y Start** or **X Start** and **Z Start**—Define the starting points of this segment. The Start fields are carried forward from the previous segment's end points.

Continue programming the contour by using the PAGE DOWN key or by selecting the **Next Segment** softkey.

Line, Arc, and Blend Arc softkey choices appear.
3D Mold Blend Arc

For Swept Surface programming, see Swept Surface in the Milling chapter of the Conversational Part Programming manual.

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.

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 3D Mold Blend Arc 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.
- **Segment**—Identifies the segment number for this operation. The system determines the number by the position of this segment in the contour.
- **Radius**—Identifies the radius of the arc.
- **Direction**—Identifies the direction of the arc from the start point (clockwise or counterclockwise).

The following fields change depending on the type of contour selected:

- **X Start** and **Y Start** or **X Start** and **Z Start**—Define the starting points of this segment. The Start fields are carried forward from the previous segment's end points.
- **X End** and **Y End** or **X End** and **Z End**—Identify the End coordinates.
- **X Center** and **Y Center** or **X Center** and **Z Center**—Identify the X Center and Y Center or X Center and Z Center coordinates used to define the circular path of the blend arc.

Continue programming the contour by using the PAGE DOWN key or by selecting the Next Segment softkey.

Line, Arc, and Blend Arc softkey choices appear.
Roughing and Finishing Tools

In many applications, a Flat End Mill can be used for roughing, followed by a Ball-Nosed End Mill, which is required for cutting the finished surface.

---

**Figure 1–10. Flat End Mill on a Contour**

1. First Data Block Using a Flat End Mill
2. Programmed Contour
3. Material to be Removed by Ball-Nosed End Mill

---

**Figure 1–11. Ball-Nosed End Mill on a Contour**

1. Second Data Block Using a Ball-Nosed End Mill
2. Programmed Contour (Same Contour as Data Block 1)
3. Material to be Removed by Ball-Nosed End Mill

---

- **Flat End Mill** - the cutter path is computed as if a Ball-Nosed End Mill is
used. This computation allows a Flat End Mill to be used for roughing without gouging the part, and in most cases leaves enough material to be removed for the finished surface using a Ball-Nosed End Mill.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Tool Calibrated on Tip</td>
</tr>
<tr>
<td>2</td>
<td>Tool Calculated as center of imaginary ball nose</td>
</tr>
<tr>
<td>3</td>
<td>Tool Zero</td>
</tr>
</tbody>
</table>

*Figure 1–12. Flat End Mill*
- **Ball-Nosed End Mill** - the system computes the compensated cutter path of the ball center:

![Diagram of Ball-Nosed End Mill](image)

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Tool Calibrated on Tip</td>
</tr>
<tr>
<td>2</td>
<td>Tool Calculated as center of ball nose</td>
</tr>
<tr>
<td>3</td>
<td>Tool Zero</td>
</tr>
</tbody>
</table>

*Figure 1–13. Ball-Nosed End Mill*

Special consideration must be taken when using a Finish Tool for a Mill Frame, Mill Circle or Mill Contour.
Roughing and Finishing Passes

The Flat End Mill tool path is calculated as a Ball-Nosed End Mill for the roughing pass. The maximum additional material remaining on the overall 3D contour will not exceed the tool's radius.

1. Roughing Contour Using Flat End Mill
2. Finishing Contour Using Ball-Nosed End Mill
3. Extra Material

*Figure 1–14. Roughing and Finishing Passes*
DXF OPTION

These topics are discussed in this section:

DXF Overview ................................................................. 2 - 2
DXF Build Data Block ....................................................... 2 - 3
DXF Parameters .............................................................. 2 - 5
Edit Drawing ............................................................... 2 - 6
DXF Layers ............................................................... 2 - 9
**DXF Overview**

WinMax DXF offers greater flexibility, improved compatibility with AutoCad®, and the ability to save DXF changes directly to file (avoiding the need to send DXF files back to the CAD system for editing). The DXF file translation software is compatible with DXF files generated with Autocad version 12 and earlier.

DXF files are loaded directly into WinMax as follows:

1. Press the AUXILIARY console button.
2. Select the DXF icon on the Auxiliary screen.
3. Find the DXF file on the Load DXF File screen, and select it to highlight.
4. Select the LOAD F1 softkey.

The DXF program blocks are displayed in the Program Review Screen. On single-screen machines, use F+Draw (console key) to display the DXF drawing.

On dual-screen machines, the DXF drawing is displayed on the graphics screen. After building the data blocks, the part can be viewed in Solid or Toolpath graphics using the Draw console key. Switch the screen back to the DXF drawing using F + Draw (console keys). You can switch the screen back to the previously drawn graphic (without causing it to be redrawn) with the Draw console key. To redraw the graphic, select the Draw console key a second time (or access the DRAW OPTIONS F1 softkey).

DXF units of measurement (INCH or MM) match global WinMax units. When a data block is built from a CAD drawing, the data block adopts the unit displayed on the WinMax status bar. In Build DB, changing the WinMax unit will result in a different sized element, for example, a segment that is 3 inches or 3 mm in length.

Here are the softkeys on the DXF screen:

- Parameters—see *DXF Parameters*, on page 2 - 5.
- Build DB—see *DXF Build Data Block*, on page 2 - 3.
- Zoom Window—see *Zoom Window*, on page 2 - 5.
- Edit Drawing—see *Edit Drawing*, on page 2 - 6.
- Save DXF—saves the DXF file.
- Part Programming—goes to the active part program.
- Quit CAD—exits the DXF CAD option.

The **Part Programming** softkey or icon toggles back to Part Programming without closing the DXF file. You may return to DXF at any time by selecting the DXF icon.
DXF Build Data Block

The **Build DB** softkey accesses the automatic data block building features. The system creates milling, holes, position, or pattern locations data blocks.

**Milling** - the Milling operation softkeys perform these functions in Lines/Arcs, Circles, Frame, 3D Mold, or Ellipse data blocks:

- **Accept**—loads the entity into the data block.
- **Zoom Window**—refer to *Zoom Window, on page 2 - 5*.
- **Edit Drawing**—refer to *Edit Drawing, on page 2 - 6*.
- **Reverse**—reverses the contour direction.
- **AutoChain**—defines contours by autochaining individual segments together.
- **Default Radius**—inserts the value of the default radius set in the Frame screen’s Corner Radius field.
- **Exit/Cancel**—cancels the operation and returns to the previous screen.

If you are using AutoCAD 14, set the registers to generate Polylines and Ellipses so they are saved as pline entity types and not splines.

**Holes** - Holes data blocks are built using the Hole Location Method (F1) or the Hole Pattern Method (F2):

- **Use Hole Location Method**—builds Holes Locations blocks from selected points on the drawing.
- **Use Hole Pattern Method**—builds Holes Pattern blocks from selected points on the drawing.

For either softkey, select holes on the drawing with one of three methods:

- Select individual holes on the touchscreen.
- Choose the Window Select softkey and drag across an area of the screen to select a group of holes.
- Use the Intersect softkey to select two intersecting lines. The point of intersection becomes the center of the hole.

Select the Accept softkey to create the data blocks.

These are the softkeys in the Holes menu:

- **Accept**—loads the entity into the data block.
- **Zoom Window**— refer to *Zoom Window, on page 2 - 5*.
- **Edit Drawing**—refer to *Edit Drawing, on page 2 - 6*.
- **Window Select**—selects a group of holes on the drawing.
- **Intersect**—draws a hole at the intersection of two selected lines and represents the center with a highlighted plus (+).
- **Default Order**—orders the holes as they were selected in the Auto CAD
Position - creates a Position data block from the DXF drawing. These are the softkeys on the Position menu:

- **Accept**—loads the entity into the data block.
- **Zoom Window**—refer to *Zoom Window, on page 2 - 5*.
- **Edit Drawing**—refer to *Edit Drawing, on page 2 - 6*.
- **Window Select**—selects a group of holes on the drawing.
- **Intersect**—draws a hole at the intersection of two selected lines and represents the center with a highlighted plus (+).
- **Default Order**—orders the holes as they were drawn in the original AutoCAD drawing.
- **Exit/Cancel**—cancels the operation and returns to the previous screen.

Pattern Locations—builds a Pattern data block. To use, select the softkey and then select points on the drawing to serve as pattern locations, using one of the following methods:

- Select individual holes on the touchscreen.
- Choose the Window Select softkey and drag across an area of the screen to select a group of holes.
- Use the Intersect softkey to select two intersecting lines. The point of intersection becomes the center of the hole.

Select the Accept softkey to create an empty Pattern Locations data block. Additional data can then be added either manually or from the DXF drawing.

These are the softkeys on the Pattern Locations menu:

- **Accept**—loads the entity into the data block.
- **Zoom Window**—refer to *Zoom Window, on page 2 - 5*.
- **Edit Drawing**—refer to *Edit Drawing, on page 2 - 6*.
- **Window Select**—selects a group of holes on the drawing.
- **Intersect**—specifies a pattern location at the intersection of two selected lines.
- **Default Order**—orders the holes as they were selected in the AutoCAD drawing.
- **Exit/Cancel**—cancels the operation and returns to the previous screen.
DXF Parameters

These parameters link contour segments, define part zero within the drawing, and set the radius for frame corners.

Use the Move Zero and Select Value softkeys to change the location of part zero. The Exit softkey returns to the DXF softkeys.

The fields on the DXF Parameters dialog box are defined as follows:

- **Endpoint Tolerance**—determines when the endpoints of segments are close enough to be considered equal (or coincident).

- **Part X Offset** and **Part Y Offset**—define part zero within the drawing. All dimensions are calculated from this point. The part zero symbol is a circle with crosshairs. To change this location manually, move the cursor to the field for Part X or Y Offset and enter the X or Y Offset values. To change this location graphically and automatically, use the **Move Zero** softkey.

- **Frame Radius**—sets a default corner radius to be used in Build DB. If the corners of a frame do not have the same radius, the user is prompted to either select a corner radius on the drawing or use the default value entered in this field for the radius.

- **Hole Diameter**—determines the default diameter for a hole.

Use the touchscreen to turn the remaining DXF Parameter fields on or off:

- **Display Geometry**—shows selected lines on the graphic display in a color other than black, illustrating which elements have been selected. Colors can be changed with the Choose Colors softkey on the Parameters screen.

- **Autochain Contours**—allows autochaining to be turned off so that a contour may be created by individually selected segments into a chained contour. By default, segments are automatically chained to create contours.

- **Select Holes by Diameter**—selects holes with the diameter specified in the Hole Diameter field (defined above) when the WINDOW SELECT softkey is used. This selection allows you to order the hole selection by size, which optimizes tool changes.

Zoom Window

Use the **Zoom Window** softkey to enlarge an area of the drawing or zoom out to see a full view. Use the pointer to touch an area on the screen and drag across the screen to enlarge an area of the drawing. When an area is enlarged, use the following softkeys:

- **Zoom Out**—pulls back from the drawing incrementally to the previous magnification level without re-centering the part in the drawing.

- **Fit to View**—gives a full scale of the drawing with the part in the drawing auto-centered.

- **Pan**—relocates the center of the drawing on the Graphic display.

- **Exit**—returns to the previous menu.
Edit Drawing

Use the Edit drawing feature to extend, join, modify, or split segments that need to be edited in order to create the proper geometry for the part program.

- **Extend**—locates the intersection of two lines and extends one or both of the lines to the intersection point.

To extend lines, select the EDIT DRAWING softkey and then the **Extend** softkey. Select the two lines that need to be extended. Both lines are highlighted when selected and extended to their points of intersection as shown in the examples below:

```
A
1

A
2

A
3

A
4

B

B
5
```

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

*Figure 2–1. Extended Lines and Arcs*
• **Join**—moves a selected line endpoint to the endpoint of a line or arc segment. Always select as the first endpoint the point that will be joined to the second endpoint. The endpoint at the opposite end of the first selected segment remains stationary and becomes a pivoting point. Both segments are highlighted when selected, and the screen is redrawn to reflect the joining of the two segments.

• **Modify**—is used to view or modify the actual geometry data of segments. Choose the SELECT POINT softkey to view the segment data.
  - The DXF Edit Modify - Arc dialog box appears if you select an arc. Refer to *DXF Edit Modify - Arc, on page 2 - 8*.
  - The DXF Edit Modify - Line dialog box appears if you select a line. Refer to *DXF Edit Modify - Line, on page 2 - 8*.
  - The DXF Edit Modify - Point dialog box appears if you select a point. Refer to *DXF Edit Modify - Point, on page 2 - 8*.
  - The segment appears gray on the Graphic display.

Choose the **Accept** softkey to retain the changes in the control's memory.

![Figure 2–2. Joined Lines and Arcs](image)
• **Split**—use to divide segments for selection, de-selection, and chaining. Segments may be split at midpoint or any point of intersection with other segments.

To split a segment, first select the segment and then select the point where the segment will be divided. When a segment is selected for splitting, the midpoint and all intersection points with the other segments are indicated with crosshair markers. Follow the directions in the Prompt display.

• **Delete**—deletes a programmed endpoint.

• **Trim**—trims a selected segment.

• **Explode PCurve**—shows an exploded view of a selected PolyCurve. If you are using Auto CAD 14, set the registers to generate Polylines and Ellipses so they are saved as pline entity types and not splines.

• **Exit/Cancel**—return to the Main DXF menu.

### DXF Edit Modify - Arc

The DXF Edit Modify - Arc dialog window contains these fields:

- **Start Angle**—defines the starting point of the angle.
- **Sweep Angle**—defines the total number of degrees in the arc to be cut. This number can be greater than 350.
- **Direction**—identifies the direction of the arc from the start point.
- **Radius**—identifies the radius of the arc.
- **Center X** and **Center Y**—identify the X and Y coordinates for the center point of the arc.

### DXF Edit Modify - Line

The DXF Edit Modify - Line dialog window contains these fields:

- **Endpoint1 X** and **Endpoint1 Y**—define the first endpoints for the X and Y coordinates.
- **Endpoint2 X** and **Endpoint2 Y**—define the second endpoints for the X and Y coordinates.
- **Length**—identifies the line length.
- **XY Angle**—identifies the angle of the XY coordinate.

### DXF Edit Modify - Point

The DXF Edit Modify - Point dialog window contains these fields:

- **X Value**—identifies the X location for the selected point.
- **Y Value**—identifies the Y location for the selected point.
DXF Layers

Many DXF drawings use layers - an electronic method of representing transparent acetate overlays used in hand-drawn drafting work.

- **Select Layer**—toggles the highlighted layer on and off.
- **All On**—turns on all of the layers.
- **All Off**—turns off all of the layers.
- **Exit**—return to the Main DXF menu.
HELICAL PLUNGE OPTION

The Helical Plunge programming option provides helical plunge as an alternative machining strategy. Helical plunge and straight plunges can be used separately for roughing and finishing phases, or they can be used together for the same operation. For example, you can rough with a helical plunge and finish with a straight plunge.

In the Helical Plunge option, the tool rotates around the cut and moves down the Z-axis. The cutting tool is continuously cutting deeper and enters and exists the machined part only once.

To use Helical Plunge, set the Mill or Finish Plunge Type in Milling Parameters to Helix.

Helical Plunge uses the feedrate programmed by the operator on the mill circle, mill frame, ellipse, or mill contour start screens.

Pattern blocks can be used with helical plunging. A scaled-up or scaled-down pattern will not affect the diameter of the helix plunge. When using a mirror image pattern, the helical plunge will be in the negative Z direction.

Helical Plunge for rotary mill frames, mill circles, and mill contours is similar to helical plunging for non-rotary mill frames, mill circles, and mill contours.
Helical Plunge Milling Parameter Fields

On the Milling Parameters screen, choose the Mill Plunge Type as Helix. If the Helical Plunge option is enabled, the fields specifying the plunging parameters appear. If the fields do not appear, the Helical Plunge option has not been installed on the control. Call Hurco or a Hurco distributor to purchase the option.

- **Mill Plunge Type**—specifies the plunging strategy to use for the milling pass. Choose Straight or Helix. The default setting is Straight.

- **Mill Plunge Ramp Slope**—defines the slope of the helical ramp for the milling tool. The range is 1° to 90°. Choosing 90° will result in a Straight Plunge. The default value is 10°.

- **Mill Plunge Helix Radius**—used for specifying the Helical Plunge radius as a percentage of the tool diameter. The range is from 0% to 100%. Choosing 0% results in a Straight Plunge. If a value of 50% or less is chosen, it will prevent a post (a thin cylinder of material formed after helical plunging) from being formed by the Helical Plunge. The default setting is 25%.

- **Finish Plunge Type**—specifies the plunging strategy to use for the finish phase. Choose Straight or Helix. The default setting is Straight.

- **Finish Plunge Ramp Slope**—defines the slope of the Helical Ramp for the finishing tool. Range is 1° to 90°. Choosing 90° will result in a Straight Plunge. The default setting is 25°.

- **Finish Plunge Helix Radius**—defines the value of the Helical Plunge radius as a percentage of the diameter of the finishing tool. The range is 0% to 100%. If a value of 50% or less is chosen, it will prevent a post from being formed by the Helical Plunge. The default setting is 25%.

- **Operator Specify Pocket Start**—if Yes, the pocket start location fields will appear on pocket boundary screens, when **Pocket Type** Inward is selected. The default setting is No. The value of Pocket Plunge Near Center is ignored.

- **Pocket Plunge Near Center**—if Yes, UltiPocket will attempt to perform a plunge at the approximate center of the pocket and then move to the start point of the pocket. If the software detects an interference with plunging in the center, the plunge will be made at the start of the tool path. The default setting is No.

The Pocket Plunge Near Center parameter may be used with frames, circles, ellipses, and contours with **Pocket Type** Inward, and contours with **Pocket Type** Outward.

Frames, circles, and ellipses with **Pocket Type** Outward automatically attempt to plunge to the center.

- **Allow Plunge Outside Pocket**—if Yes, UltiPocket will plunge outside the pocket, moving through the open side of the pocket boundary. May be used with open contours only; cannot be applied to frames, circles, or ellipses because they cannot be programmed with open contours.

Helical Plunge Milling Parameter fields are available only in Conversational Programming.
The **Operator Specify Pocket Start** parameter must also be set to Yes.

The contour **Pocket Type** must be set to Inward.

The plunge point should be placed near the open side of the contour outside the pocket. The tool moves through this opening near the approximate center. The software verifies the tool will fit through the opening and generates an error message if it will not.
Helical Plunge (Inside/Outside) for Mill Frames, Mill Circles and Ellipses

Helical Plunge is similar for milling the inside or outside of mill frames, circles, and ellipses (with or without blend-in moves). The center location of the helical plunge is the same as a straight plunge. The direction of the helical plunge (clockwise, CW, or counter clockwise, CCW) will be determined by the tool spin direction (CW or CCW) and the milling direction (climb or conventional). If Blend Offset is used, the helical plunge will be centered about the plunge point of the Blend Offset.

The helical plunge direction that provides a smooth transition to the tool path will be chosen. Helical plunge is not allowed when Milling Type On is selected.

Helical Plunge with UltiPocket

The Helical Plunge option is used with the UltiPocket option to define the plunging location when inward pocketing. The operator can specify the pocket plunge location using the Operator Specify Pocket Start function, or start the pocket plunge near the center by using the Inward Pocket Plunge Near Center function. See the UltiPocket Option chapter for more information on using the Helical Plunge option with UltiPocket.

When both the Operator Specify Pocket Start and the Inward Pocket Plunge Near Center are set to No, the plunge locations are used that would have been used without the Helical Plunge option.

⇒ An error message appears if the plunge point specified would violate the programmed part surface.

Helical Plunge with Operator Specified Location

When the operator specifies the plunge point, all of the helix plunge moves will occur at that location, even for the pocket boundary.

Helical Plunge in the Center of a Pocket

When the Inward Pocket Plunge Near Center field value is Yes (Operator Specify Pocket Start value must be No), a plunge point near the center of the pocket will be chosen. Islands near the center will impact upon the plunge point’s location.

💡 When machining a part with a lot of webbing (many small pockets separated by walls), it may be desirable to disable helical plunging.
Helical Plunge with Outward Pockinging

Helical plunging occurs near the center of the pocket when used with Outward Pockinging, and only one plunge location is needed. The Operator Specify Pocket Start and the Inward Pocket Plunge Near Center fields have no effect on Helical Plunge with Outward Pockinging.

Helical Plunge of Mill Frame Inside with No Pecking and Blend Offset

The tool will helical plunge to the Z Bottom level and then perform the normal blend-in move. The direction of the helical plunge arc will smoothly transition to the blend-in arc. For instance, if the blend-in arc is CCW, the direction of the helical plunge will be CW. The following isometric views are created by setting the Draw Plunge Moves graphics parameter field to Yes.

⇒ This graphical representation is only for informational purposes and cannot be viewed with WinMax.

The graphics show the finish pass only if a finish tool is specified in the frame block, and do not show individual peck levels.

![Figure 3–1. Helical Plunge with No Pecking and Blend Offset (Isometric View)](image-url)
Helical Plunging of Mill Frame Inside with Pecking and Straight Plunge Finish Pass and Blend Offset

The following example is a Mill Frame Block Type with Inside Milling Type. The finish tool specified and the peck depth is set to 0.6 inches.

First Peck - The tool helical plunges down to the first peck depth, then mills another full circle to ensure that all material down to the first peck is removed. After the full circle is completed, a 180° blend in arc is performed. The direction of the helical plunge will always be in the opposite direction of the blend-in arc.

⇒ These graphical representations are only for informational purposes and cannot be viewed with WinMax.

Second Peck and All Pecks to Finish Peck Depth - The tool will rapid down to the previous peck depth plus the peck clearance plane. The tool then helical plunges down to the next peck depth or the Z Bottom plus the Finish Z. If the tool is at a peck depth, a 360° arc will be machined. The 360° arc will not be machined if it is the last peck depth. Instead, a blend-in arc will be machined. The blend-in arc for the last peck depth will be similar to the one machined for the first peck.
Finish Pass - The tool will rapid down to the Z Start and plunge feed down to the Z Bottom. A blend-in move is performed before milling the frame contour.

When helical plunge is used for roughing passes, a large amount of material is removed around the point of entry. Therefore, using helical plunging for the finish passes is probably not necessary.

If the finish tool is larger than the roughing tool, helical plunges should also be performed for the finish pass. If a post was created by the roughing tool (the Helix plunge radius was greater than 50 percent) the finish tool may be cutting into the post.
Helical Plunge with Lines and Arcs

Helical plunge with lines and arcs occurs at the start of the contour.

The following example shows helical plunging with right cutter compensation of a contour.

An error message will be displayed if the helical plunge would cut into a part surface. If this error message appears, move the starting location of the contour to an area that will not cause interference.

Helical Plunge with 3-D Part Programming Option

Helical plunging is supported in the 3-D Part Programming option. A helical plunge can be performed when the Mill Plunge Type field is set to Helix.

Only one helical plunge occurs when using a bi-directional tool path. When performing helical plunge using unidirectional tool path, a helical plunge occurs for each cutter pass.

When programming complex 3-D parts, the operator should review helical plunge placements in graphical form before milling to ensure helical plunges from one block do not interfere with neighboring blocks.

If Mill Plunge Type field is set to Helix, but the helical plunge does not appear on the graphics screen, check the Z Start field on the Mill 3-D block screen. The plunges may not appear if Z Start value is too low.
INSERT POCKETS

Cutter insert manufacturers use the Insert Pockets milling routines to mill pockets in triangular, diamond, and hexagon shapes. These routines are sold as a WinMax option and can be defined in one program data block.

To access the Insert Pockets features, select the Milling softkey in a New Block screen, then Select the More softkey. If the option is installed the next screen will have a Special softkey. Select that softkey to display the following screen:

![Figure 4-1. Insert Pockets Softkeys](image)

Choose Triangle, Diamond 1 Face, Diamond 2 Faces, or Hexagon milling. These data blocks are described in the sections that follow:

*Mill Triangle Data Block, on page 4 - 2*

*Mill Diamond Data Block, on page 4 - 4*

*Mill Hexagon Data Block, on page 4 - 7*
Mill Triangle Data Block

The triangle pocket shape has three equal 60° angles with one open face as shown below. A relief cut can be programmed in the corner at point 1.

![Triangle Programming Diagram](image)

*Figure 4–2. Triangle Programming Diagram*

Fields in the Mill Triangle Data Block are:

- **Block**—Identifies the block number for this operation. The system determines the number by the position of this data block in the program.
- **X Center**—identifies the X-axis coordinate for the center of the triangle.
- **Y Center**—identifies the Y-axis coordinate for the center of the triangle.
- **Radius**—identifies the value for the radius of the inscribed circle.
- **Orientation**—identifies the angle to rotate the pocket about the center.
- **Z Start**—identifies 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.
- **Relief 1**—identifies the direction of corner relief cut and the depth past the corner of the two walls. Provides the option of a right, center, or left relief cut (or none) in the corner at point 1.
- **Tool**—identifies the tool number for this data block and enters that tool's diameter and type on this screen.
- **Milling Type**—defines cutter motion using these options:
  - **Inside** - cuts just the non-open faces of the pocket, including the relief cuts.
  - **Inside 2 Passes** - cuts the same as Inside, except this selection uses a roughing pass and a finishing pass. During roughing, material is left for the finish pass.
  - **Pocket Outside In** - cuts along the faces of the insert, including the relief cuts and then steps inward and cleans out the entire insert by executing smaller versions of the shape until the center point is reached.
- **Pocket 2 Passes Outside In** - cuts in the same manner as Pocket Outside In, except this selection uses a roughing and a finishing pass.

- **Pocket Inside Out** - plunges tool at X-Y center of the insert and cuts outward, executing larger versions of the shape until the faces and relief cuts are milled.

- **Pocket 2 Passes Inside Out** - cuts the same as Pocket Inside Out, except this selection uses a roughing and a finishing pass.

- **Mill Feed**—defines the X-Y feedrate.

- **Speed (RPM)**—identifies the speed at which the spindle rotates to machine the part.

- **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. Also used when plunging down in Z between peck depth passes.
Mill Diamond Data Block

The diamond pocket shapes have four sides with opposite angles equal.

The **Diamond 1 Face** has one open face and the option of a right, center, or left relief cut in the corners at point 1 and point 2 as shown below:

![Diamond 1 Face Diagram](image)

*Figure 4–3. Diamond 1 Face Diagram*
The **Diamond 2Faces** pocket has two open faces and the option of a right, center, or left relief cut in the corner at point 1 as shown below:

![Figure 4–4. Diamond 2 Faces Diagram](image)

**Fields are:**

- **Block**—Identifies the block number for this operation. The system determines the number by the position of this data block in the program.
- **X Center**—identifies the X-axis coordinate for the center of the triangle.
- **Y Center**—identifies the Y-axis coordinate for the center of the triangle.
- **Radius**—identifies the value for the radius of the inscribed circle of the diamond.
- **Orientation**—identifies the angle to rotate the pocket about the center.
- **Shape Angle**—identifies the angle formed by points 1 and 3 defining the shape with a range from 20° to 160°.
- **Z Start**—identifies 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 feedrate begins.
- **Relief 1**—identifies the direction of corner relief cut and the depth past the corner of the two walls. Provides the option of a right, center, or left relief cut (or none) in the corner at point 1.
- **Relief 2**—for Diamond 1 Face only, provides the option of a right, center, or left relief cut (or none) in the corner at point 2.
- **Tool**—identifies the tool number for this data block and enters that tool’s diameter and type on this screen.
- **Milling Type**—defines cutter motion using these options:
  - **Inside** - cuts just the non-open faces of the pocket, including the relief cuts.
  - **Inside 2 Passes** - cuts the same as Inside, except this selection uses a
roughing pass and a finishing pass. During roughing, material is left for the finish pass.

- **Pocket Outside In** - cuts along the faces of the insert, including the relief cuts and then steps inward and cleans out the entire insert by executing smaller versions of the shape until the center point is reached.

- **Pocket 2 Passes Outside In** - cuts in the same manner as Pocket Outside In, except this selection uses a roughing and a finishing pass.

- **Pocket Inside Out** - plunges tool at X-Y center of the insert and cuts outward, executing larger versions of the shape until the faces and relief cuts are milled.

- **Pocket 2 Passes Inside Out** - cuts the same as Pocket Inside Out, except this selection uses a roughing and a finishing pass.

- **Mill Feed**—defines the X-Y feedrate.

- **Speed (RPM)**—identifies the speed at which the spindle rotates to machine the part.

- **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. Also used when plunging down in Z between peck depth passes.
Mill Hexagon Data Block

This pocket shape has six sides and every other angle equal. In the diagram below, the shape has two open faces and the option of a relief cut in the corners at point 1, point 2, and point 3, and the option of a face relief distance between points 0 and 1 and points 3 and 4.

**Figure 4–5. Hexagon Diagram**

<table>
<thead>
<tr>
<th>P-0</th>
<th>Point 0</th>
</tr>
</thead>
<tbody>
<tr>
<td>P-1</td>
<td>Point 1</td>
</tr>
<tr>
<td>P-2</td>
<td>Point 2</td>
</tr>
<tr>
<td>P-3</td>
<td>Point 3</td>
</tr>
<tr>
<td>P-4</td>
<td>Point 4</td>
</tr>
<tr>
<td>P-5</td>
<td>Point 5</td>
</tr>
<tr>
<td>R-1</td>
<td>Relief 1</td>
</tr>
<tr>
<td>R-2</td>
<td>Relief 2</td>
</tr>
<tr>
<td>R-3</td>
<td>Relief 3</td>
</tr>
</tbody>
</table>
Fields are:

- **Block**—Identifies the block number for this operation. The system determines the number by the position of this data block in the program.
- **X Center**—identifies the X-axis coordinate for the center of the triangle.
- **Y Center**—identifies the Y-axis coordinate for the center of the triangle.
- **Radius**—identifies the value for the radius of the inscribed circle.
- **Orientation**—identifies the angle to rotate the pocket about the center.
- **Shape Angle**—identifies the angle formed by points 1 and 3 defining the shape with a range from 20° to 160°.
- **Z Start**—identifies 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 feedrate begins.
- **Relief 1**—identifies the direction of corner relief cut and the depth past the corner of the two walls. Provides the option of a right, center, or left relief cut (or none) in the corner at point 1, or face relief distance between points 0 and 1.
- **Relief 2**—provides the option of a right, center, or left relief cut (or none) in the corner at point 2.
- **Relief 3**—provides the option of a right, center, or left relief cut (or none) in the corner at point 3 or the option of a face relief distance between points 3 and 4.
- **Tool**—identifies the tool number for this data block and enters that tool's diameter and type on this screen.
- **Milling Type**—defines cutter motion using these options:
  - **Inside** - cuts just the non-open faces of the pocket, including the relief cuts.
  - **Inside 2 Passes** - cuts the same as Inside, except this selection uses a roughing pass and a finishing pass. During roughing, material is left for the finish pass.
  - **Pocket Outside In** - cuts along the faces of the insert, including the relief cuts and then steps inward and cleans out the entire insert by executing smaller versions of the shape until the center point is reached.
  - **Pocket 2 Passes Outside In** - cuts in the same manner as Pocket Outside In, except this selection uses a roughing and a finishing pass.
  - **Pocket Inside Out** - plunges tool at X-Y center of the insert and cuts outward, executing larger versions of the shape until the faces and relief cuts are milled.
  - **Pocket 2 Passes Inside Out** - cuts the same as Pocket Inside Out, except this selection uses a roughing and a finishing pass.
- **Mill Feed**—defines the X-Y feedrate.
- **Speed (RPM)**—identifies the speed at which the spindle rotates to machine the part.
• **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. Also used when plunging down in Z between peck depth passes.
ROTARY

These topics are discussed in this section:

Rotary Overview .................................................. 5 - 2
Rotary Part Setup .................................................. 5 - 8
Rotary Part Programming ....................................... 5 - 10
Rotary A and Rotary A Tilt B Configuration .............. 5 - 14
Tilt A Rotary C Configuration ................................. 5 - 20
Rotary B Configuration ........................................... 5 - 26
Tilt B Rotary C Configuration ................................. 5 - 29
Universal Rotary Configuration ............................... 5 - 35
Calculating X- and Z-Axis Positions After a Tilt-Axis Move ............... 5 - 49
Example Programs .............................................. 5 - 51
Rotary Overview

Rotary-axis and tilt-axis operations and hardware (e.g., rotary table, tilt table) are typically used to cut around a cylindrical part while it is turning. When your machine is equipped with rotary-axis and/or tilt-axis hardware, you can mill more diverse parts than you can using 3-axis machining. Hurco’s Conversational rotary part programming allows you to easily drill and contour cylindrical, curved, odd-shaped or asymmetrical parts using rotary-axis and/or tilt-axis hardware.

A rotary axis can move in a 360° rotation in positive or negative directions. See Rotary Axis, on page 5 - 3.

A tilt axis moves less than a 360° rotation. The machine configuration determines the limits of tilt-axis rotation. See Tilt Axis, on page 5 - 3.

In non-rotary linear milling operations, the machine table and spindle move in the X, Y, and Z axes. In rotary-axis and/or tilt-axis milling operations, the A, B, and/or C axes replace the corresponding linear axes.

- A axis rotates/tilts around the X axis.
- B axis rotates/tilts around the Y axis.
- C axis rotates/tilts around the Z axis.

The configuration of rotary and/or tilt hardware installed in your machining center will determine which axes are available for rotary-axis and/or tilt-axis milling operations.
Rotary Axis

Rotary-axis operations in a Conversational rotary part program are similar to standard milling operations, except that the rotary milling feature (e.g., rotary circle, rotary frame) is wrapped around a cylinder. Rotary-axis operations can be performed on a 4-axis or 5-axis machine where the part is fixtured to the centerline of the rotary-axis.

⇒ A rotary-axis table can move in a 360° rotation in both positive and negative directions.

Tilt Axis

A pivoting rotary table allows you to tilt the table to a specific angle during machining. Any rotary program or standard milling operation can be executed at the tilt position, but the tilt-axis will remain stationary while the data block is being machined. If you want to machine a part while moving the tilt axis, you must use NC programming. Refer to the WinMax Mill NC Programming manual for more information.

⇒ A tilt-axis moves less than a 360° rotation. The machine configuration determines the limits of tilt-axis rotation.

See Calculating X- and Z-Axis Positions After a Tilt-Axis Move, on page 5 - 49 for additional information.

Configuration of Hurco Machining Centers

Hurco machining centers provide a variety of machine configurations. Rotary and/or tilt tables can be installed in Hurco machining centers, either at the factory or on-site. Some Hurco machine models come with rotary hardware as standard equipment. The table below shows configurations available for Hurco machining centers.

<table>
<thead>
<tr>
<th>Configuration</th>
<th>Machine Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotary A</td>
<td>4-axis rotary machining</td>
</tr>
<tr>
<td>Rotary A/Tilt B</td>
<td>4-axis rotary and 5-axis rotary or tilt machining</td>
</tr>
<tr>
<td>Tilt A/Rotary C</td>
<td>4-axis rotary and 5-axis rotary or tilt machining on trunion table machines</td>
</tr>
<tr>
<td>Rotary B</td>
<td>4-axis rotary machining on horizontal (HMX) machines</td>
</tr>
<tr>
<td>Tilt B/Rotary C</td>
<td>4-axis rotary and 5-axis rotary or tilt machining on SR machines</td>
</tr>
<tr>
<td>Universal Rotary</td>
<td>All axis configurations and machines</td>
</tr>
</tbody>
</table>

Table 5–1. Rotary Configurations Available on Hurco Machines
Setting the Axis Configuration

You must set the axis configuration in the Conversational part program before you program rotary or tilt operations. Follow these steps to set the axis configuration:

1. Press the Auxiliary console button to access the Auxiliary screen.
2. Select the Utilities icon from the Auxiliary screen.
3. Select the USER PREFERENCES F2 softkey.
4. Select the CONVERSATIONAL SETTINGS F2 softkey.
5. Select the axis configuration from the Default Conversational Program Type drop-down list or softkeys on the Conversational Settings screen (configurations listed below are not available for all Hurco models):
   - Rotary A—Rotary axis
   - Rotary A Tilt B—Rotary and/or tilt axis
   - Tilt A Rotary C—Rotary and/or tilt axis
   - Rotary B—Rotary axis
   - Tilt B Rotary C—SR only
   - Universal Rotary—all configurations

Select the axis configuration for all of the axes on your machine, even if you don’t think they will be used in the program. Rotary- and/or tilt-axis data blocks can be added to an existing program as long as the rotary-axis and/or tilt-axis configuration was selected when the program was first created.

Once an axis configuration is set for a program, it cannot be changed. However, you can use an existing Conversational program to create a new program that contains rotary-axis and/or tilt-axis programming data blocks. Follow these steps to create the new program:

1. Create a new Conversational program with the rotary-axis or tilt-axis configuration for your machine.
2. Select the IMPORT FUNCTIONS F6 softkey from the Input screen to pull in all data blocks and setups from an existing Conversational part program. Refer to Getting Started with WinMax Mill for more information.
3. Add rotary-axis or tilt-axis operations in the new program.
4. Save the new program under a new name. You now have a rotary-axis and/or tilt-axis version and a standard version of the same Conversational part program.
Transform Plane

Hurco’s Transform Plane feature can be used with 4-axis or 5-axis machining centers to machine non-rotary milling or drilling features on multiple sides (i.e., planes) of a workpiece. Part zero is reestablished for each transformed plane in a Transform Plane Reference Points data block; the tool automatically moves so that it is always perpendicular to the transformed plane.

Transform Plane can only be used in a rotary part program.

Transform Plane can be used in a WinMax Mill 4-axis or 5-axis part program or as part of a VMX42SR part program. Using Transform Plane has the following benefits:

- Reduces setup time and potential positioning errors because the part doesn’t have to be refixtured each time the plane is transformed.
- Eliminates programming errors because the machine reestablishes part zero for each transformed plane.
- Allows a non-rotary milling or drilling feature to be machined on one or more sides of an irregularly-shaped workpiece.

Rotary Transform Plane Block

Transform Plane is used in a Conversational rotary program to position part zero for a non-rotary data block to any orientation of any plane of your rotary- or tilt-axis machine. For example, use Transform Plane when you want to repeat a non-rotary feature (e.g., mill contour, holes) on multiple sides of a part.

Rotary Pattern Loop and Rotary Pattern Locations data blocks cannot be used within a Transform Plane operation.

Non-rotary patterns cannot be in effect (i.e., must be closed with a Pattern End data block) prior to a Transform Plane operation.

These are the Rotary Transform Plane fields:

- **Location**—The location number of the transformed plane.
- **X**—the X offset value.
- **Y**—the Y offset value.
- **Z**—the Z offset value.

Follow these steps to use Transform Plane in a rotary part program:

1. Create the non-rotary milling or drilling feature on the X-Y plane (i.e., create the milling or drilling feature as a 3-axis part) in a Conversational rotary part program.
2. Insert a Rotary Position data block to define the orientation of the transformed plane (i.e., position of the part) for the non-rotary data block.
The Rotary Position data block must be placed immediately prior to the Transform Plane Reference Points data block (see step 4) in the part program.

Select the Rotary Position F1 softkey to access the Rotary Position data block from a rotary data block.

3. On the Rotary Position data block, select Yes for the Transform Part Zero field to activate Transform Plane for non-rotary data blocks in the rotary part
   - **Use Offset Z** (Zero Calibration mode only)—determines whether or not the value in the Offset Z field on the Part Setup screen will be used by the machine in a transformed plane. The default value is No. Refer to **Rotary Part Setup, on page 5 - 8** for information about the Offset Z field.
     * When set to Yes, the value in the Offset Z field on the Part Setup screen will be used to establish the part zero coordinate system in Transform Plane operations.
     * When set to No, the value in the Offset Z field on the Part Setup screen will be added or subtracted from the Z calibration point during Transform Plane operations.

   - Although the Transform Plane Reference Points data block resembles a Pattern Locations data block, it behaves differently. The X, Y, and Z fields on a Transform Plane Reference Points data block establishes part zero for the non-rotary feature on the current transformed plane.
   - Follow these steps to access the Transform Plane Reference Points data block from a rotary data block:
     a. Select the Rotary Patterns F6 softkey.
     b. Select the Transform Plane F3 softkey.

5. Add a Transform Plane End data block to end the Transform Plane operation. There are no fields in the Transform Plane End data block.
   - Follow these steps to access the Transform Plane End data block from a rotary data block:
     a. Select the Rotary Patterns F6 softkey.
     b. Select the Transform Plane End F4 softkey.

   After the Transform Plane End data block has executed, the orientation of the part will remain at the position defined in the most recent Rotary Position data block.

   If the part program continues beyond the Transform Plane operation, another Rotary Position data block will be required if the part must be reoriented to a different position.
Rotary Transform Plane End Block

The Transform Plane End block ends the Transform Plane operation.

After the Transform Plane End data block has executed, the orientation of the part will remain at the position defined in the most recent Rotary Position data block.

If the part program continues beyond the Transform Plane operation, another Rotary Position data block will be required if the part must be reorientated to a different position.

See *Transform Plane Example Programs, on page 5 - 62* for more information.
Rotary Part Setup

In addition to standard Part Setup fields, Part Setup for rotary programming includes part zero for the rotary axis and the rotary centerline fields.

- Manually type the values directly into the Part Setup fields, or use the jog unit to set the values. Follow these steps to use the jog unit to enter the Part Setup values:
  1. Use the jog unit to move the appropriate axis until it is in the position that you want to define as part zero.
  2. Select the **Store Machine Position** softkey or the **Store Position** button on the jog unit.

For explanation of the Part Setup fields, see *Part Setup, on page 4 - 3."

Rotary Centerline

Hurco recommends automatically calculating the rotary centerline. Use the following steps:

1. Enter Part Zero X and Part Zero Y field values, either manually or by using the jog unit to store the machine position.
2. Place the cursor in the Part Zero X, Part Zero Y, or Offset Z field to make the Calculate Rotary Offsets softkey available.
3. Select the Calculate Rotary Offsets softkey. The machine automatically calculates the values for the rotary centerline fields.
When the part is fixtured to the center of the rotary-axis table, the rotary centerline is the Y-Z center point of the part.
Rotary Part Programming

Part program type is dependent upon machine configuration:

- Rotary A—4-axis rotary machining
- Rotary A Tilt B—4-axis rotary and 5-axis rotary or tilt machining
- Tilt A Rotary C—4-axis rotary and 5-axis rotary or tilt machining on trunion table machines
- Rotary B—4-axis rotary machining on horizontal machines
- Tilt B Rotary C—4-axis rotary and 5-axis rotary or tilt machining on SR machines
- Universal—all axis configurations and machines

See Configuration of Hurco Machining Centers, on page 5 - 3 for information about setting the program type.

The following sections contain details about the program blocks for each configuration:

- Rotary A and Rotary A Tilt B Configuration .......................... 5 - 14
- Tilt A Rotary C Configuration ............................................. 5 - 20
- Rotary B Configuration ..................................................... 5 - 26
- Tilt B Rotary C Configuration ............................................. 5 - 29
- Universal Rotary Configuration .......................................... 5 - 35

Rotary New Block

Rotary New Block is the same for all configurations. When you select the Rotary softkey a new data block will appear with the following softkeys:

- **Rotary Position**—moves all axes, including the rotary axis. See Rotary Position Block, on page 5 - 11.

- **Rotary Milling**—access the Rotary Lines and Arcs, Circle, Frame, True-Type Font, Slot, and Polygon data blocks. See Rotary Milling New Block, on page 5 - 12.

- **Rotary Patterns**—access the Rotary Pattern Loop, Rotary Pattern Locations, Rectangular, Mirror, and Transform Plane data blocks to repeat a rotary feature on a cylinder. See Rotary Patterns, on page 5 - 12.

- **Rotary Parameters**—define the parameters of the cylinder. See Rotary Parameters, on page 5 - 13.

Tool changes are completed before each rotary data block is executed. The tool change will be automatic or the machine will prompt for manual insertion of the tool specified in the data block.

Refer to Example Programs, on page 5 - 51 for sample part programs.
Rotary Position Block

A Rotary Position data block is used to move the machine’s axes to a specific location at Rapid feedrate. A new Rotary Position data block is required each time the rotary-axis must be repositioned in the part program.

The tool will move to the Retract Plane, Safety Plane, or Home Z position before the table rotates into position each time a Rotary Position data block with rotary-axis positioning is executed in a Conversational part program.

⇒ A Rotary Position data block should be included as the first data block in a Conversational rotary part program to define the initial orientation of the rotary axis.

Rotary Position Block fields are dependent upon machine axis configuration. Fields for all configurations except Universal are listed below. See Rotary Position Block, on page 5 - 35 for Universal Rotary Position block fields.

- **Stop**—when set to Yes, pauses program execution after the Rotary Position data block is executed. To restart the program, press the Start Cycle console button.
- **X Position**—X-axis coordinate relative to Part Zero X.
- **Y Position**—Y-axis coordinate relative to Part Zero Y.
- **A Angle, B Angle, or C Angle**—rotary-axis coordinate (angle) relative to Part Zero A, Part Zero B, or Part Zero C, depending on the configuration of your machine.
- **Rotary Safety Move**—when set to Yes, moves the X and Y axes to the locations defined in X and Y Safety Position fields (described below) before rotary-axis orientation.

  Set the Rotary Safety Move field to Yes if there is a potential for collision between the rotary axis and the spindle.

- **X, Y Safety Position**—when the Rotary Safety Move field (described above) is set to Yes, defines the location to which X and Y axes will move before the rotary-axis orients.
- **Transform Part Zero**—when set to Yes, used to activate Transform Plane for a non-rotary data block machined on multiple sides of a part. Refer to Rotary Transform Plane, on page 5 - 48 for information about using Transform Plane in a Conversational rotary part program.
- **Use Offset Z** (Zero Calibration mode only)—available when Transform Part Zero is Yes. Determines whether or not the value in the Offset Z field on the Part Setup screen will be used by the machine in a transformed plane. The default value is No. Refer to Rotary Part Setup, on page 5 - 8 for information about the Offset Z field.

  - When set to Yes, the value in the Offset Z field on the Part Setup screen will be used to establish the part zero coordinate system in Transform Plane operations.
  - When set to No, the value in the Offset Z field on the Part Setup screen will be added or subtracted from the Z calibration point during
Transform Plane operations.

- **Z Move Type**—select the level to which Z-axis retracts at the beginning of the Rotary Position data block.
- **Z Home**—Z-axis will move at Rapid feedrate to the position defined in Z Machine field.
- **Z Safety**—Z-axis will move at Rapid feedrate to the position defined in the Part Limit Z (+) field on the Part Setup screen.
- **Tool**—number of the tool that will cut the part, entered on the Tool Setup screen.

**Rotary Milling New Block**

Rotary Milling New Block is the same for all configurations. These softkeys appear on the Rotary Milling New Block screen:

- **Rotary Lines And Arcs**—access the Rotary Mill Contour data block to create lines and arcs on a cylinder.
- **Rotary Circle**—access the Rotary Mill Circle data block to create a circle on a cylinder.
- **Rotary Frame**—access the Rotary Mill Frame data block to create a frame on a cylinder.
- **Rotary True Type Font** (available with Universal Rotary configuration only)—access the Rotary True Type font data block to create text on a cylinder.
- **Rotary Slot** (available with Universal Rotary configuration only)—access the Rotary Slot data block to create a slot on a cylinder.
- **Rotary Polygon** (available with Universal Rotary configuration only)—access the Rotary Polygon data block to create a polygon on a cylinder.
- **Rotary Stick Lettering** (available with Universal Rotary configuration only)—access the Rotary Stick Lettering data block to create text on a cylinder.

**Rotary Patterns**

Rotary patterns allow you to repeat a mill feature (e.g., rotary circle, rotary frame, holes) on the part. These are the softkeys on the Rotary Patterns data block:

- **Rotary Loop**—repeats a pattern along the X-A plane on a cylindrical part.
- **Rotary Locations**—repeats a mill feature anywhere on a cylindrical part.
- **Rotary Rectangular** (available only with Universal Rotary configuration)—repeats a rectangle on a cylindrical part.
- **Rotary Mirror** (available only with Universal Rotary configuration)—repeats a pattern as a mirror image.
- **Transform Plane**—activates Transform Plane feature.
- **Transform Plane End**—ends a Transform Plane operation.
- **Pattern End**—ends a rotary pattern loop or rotary locations operation.
Rotary Parameters

The Rotary Parameters data block is used to set the radius of the cylindrical part and to define the Y Off of Centerline parameter.

These are the fields on the Rotary Parameters data block:

- **Cylinder Radius** (all configurations except Tilt B Rotary C)—radius of the cylindrical part. If the radius is a negative number, the tool will cut the rotary mill feature (e.g., rotary circle, rotary frame) from the top to bottom.

  If the radius of the cylindrical part is not defined in the Rotary Parameters data block and Cutter Compensation is used in rotary lines and arcs, the machine will define the part's radius as the distance from Z Start to the rotary centerline.

- **Y Off Of Centerline** (Rotary A and Rotary A Tilt B configurations only)—determines where the tool will locate the Y coordinate. The default selection is No.

  - If set to No, the tool will position the Y coordinate (specified in the A Centerline Y field on the Part Setup screen) directly over the centerline of the rotary axis when cutting a rotary mill feature (e.g., rotary circle, rotary frame).

  - If set to Yes, the tool will stay at the current Y coordinate (specified in the A Centerline Y field on the Part Setup screen) when cutting a rotary mill feature (e.g., rotary circle, rotary frame).

- **Rotary Centerline X** (Tilt B Rotary C configuration only)—the distance from part zero to the vertical axis around which the part will rotate.

- **Rotary Centerline Y** (Tilt B Rotary C configuration only)—the distance from part zero to the horizontal axis around which the part will rotate.
Rotary A and Rotary A Tilt B Configuration

Rotary Position

The Rotary Position data block is available with multiple rotary configurations; see Rotary Position Block, on page 5 - 11 for an explanation of the softkeys on this screen.

Rotary Lines and Arcs

Rotary line and arc segments are used to create a rotary mill contour. A rotary mill contour is similar to a standard mill contour, except the rotary contour is wrapped around a cylinder.

Start Segment

These are the fields on the Rotary Mill Contour Start Segment data block:

- **X Start**—X-axis coordinate for first segment’s starting point.
- **A Angle**—rotary-axis coordinate (angle) for the first segment’s starting point.
- **Z Start**—point where the Z-axis plunge feedrate begins.
- **Z Bottom**—the bottom of the finished segment and the location where the Z-axis plunge feedrate ends.

Line Segment

These are the fields on the Rotary Mill Contour Line segment data block:

- **X End**—X-axis coordinate for ending point of the line.
- **A End**—rotary-axis coordinate (angle) for the ending point of the line.
- **Z End**—Z-axis coordinate for the ending point of the line. The Z End value is the depth of the cut at the line end point and is carried forward from the previous segment. Retain the Z End value or type in a new value.
- **X Start**—X-axis coordinate start point programmed in the previous segment. The value of X Start can be changed only in the segment in which it was created.
- **A Start**—rotary-axis start point coordinate programmed in the previous segment. The value of A Start can be changed only in the segment in which it was created.
- **Z Start**—point where the Z-axis plunge feedrate begins.

Arc Segment

These are the fields on the Rotary Mill Contour Arc segment data block:

- **Direction**—the direction of the arc from the start point, Clockwise or Counter Clockwise.
- **X End**—X-axis coordinate of the arc’s end point.
• **A End**—rotary-axis coordinate (angle) of the arc’s end point.
• **Z END**—point where the Z-axis plunge into the cylinder ends.
• **X Center**—X-axis coordinate of the arc’s center point.
• **A Center**—rotary-axis coordinate (angle) of the arc’s center point.
• **Radius**—the radius of the arc, if known.
• **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°.
• **X Start**—X-axis coordinate start point programmed in the previous segment. The value of X Start can be changed only in the segment in which it was created.
• **A Start**—rotary-axis coordinate start point programmed in the previous segment. The value of A Start can be changed only in the segment in which it was created.
• **Z Start**—Z-axis coordinate start point programming in the previous segment. The value of Z Start can be changed only in the segment in which it was created.

**Blend Arc Segment**

These are the fields on the Rotary Mill Contour Blend Arc segment data block:
• **Radius**—the radius of the arc, if known.
• **X End**—X-axis coordinate of the arc’s end point.
• **A End**—rotary-axis coordinate (angle) of the arc’s end point.
• **X Center**—X-axis coordinate of the arc’s center point.
• **A Center**—rotary-axis coordinate (angle) of the arc’s center point.
• **Direction**—the direction of the arc from the start point, Clockwise or Counterclockwise.
• **X Start**—X-axis coordinate start point programmed in the previous segment. The value of X Start can be changed only in the segment in which it was created.
• **A Start**—rotary-axis coordinate start point programmed in the previous segment. The value of A Start can be changed only in the segment in which it was created.

**Mill Contour End Block**

A Contour End data block is required to end rotary Mill Contour operations. There are no fields in the Contour End data block.
Rotary Circle

A rotary mill circle is similar to a standard mill circle, except the rotary mill circle is wrapped around a cylinder. These are the fields on the Rotary Mill Circle data block:

These are the Rotary Circle fields for Rotary A and Rotary A Tilt B:

- **X Center**—X-axis coordinate for the center point of the circle.
- **A Center**—rotary-axis coordinate (angle) for the center point of the circle.
- **Radius**—radius of the milled circle.
- **Z Start**—location where the Z-axis plunge feedrate begins.
- **Z Bottom**—location where the Z-axis plunge feedrate ends.

Rotary Frame

A rotary mill frame is similar to a mill frame, except the rotary mill frame is wrapped around a cylinder.

These are the Rotary Mill Frame fields for Rotary A and Rotary A Tilt B:

- **X Corner**—X-axis coordinate of any one of the four corners of the frame. This corner becomes the reference corner.
- **A Corner**—rotary-axis coordinate (angle) of the reference corner of the frame.
- **X Length**—X-axis coordinate measured from the reference corner.
  - X Length is positive if the reference corner is at the left side of the rectangle.
  - X Length is negative if the reference corner is at the right side of the rectangle.
- **A Length**—rotary-axis coordinate (angle) from the reference corner.
  - A Length is positive if the location of the frame is clockwise from the reference corner.
  - A Length is negative if the location of the frame is counter clockwise from the reference corner.
- **Z Start**—start point for the Z-axis plunge feedrate.
- **Z Bottom**—end point for the Z-axis plunge feedrate.
- **Corner Radius**—radius of the reference corner. This value will be the same for all corners of the rotary frame.
Rotary Patterns

The Rotary Patterns data block is available with multiple rotary configurations; see Rotary Patterns, on page 5 - 12 for an explanation of the softkeys on this screen. The specific pattern screens for Rotary A and Rotary A Tilt B are explained below.

Rotary Loop

A rotary loop defines the number and locations a mill feature will be repeated on a cylinder. The initial mill feature is located at part zero and all subsequent copies are at a specific distance along the X-A axis of the cylinder.

These are the Rotary Pattern Loop fields for Rotary A and Rotary A Tilt B:

- **Number**—total number of times the mill feature will be cut on the cylinder.
- **X Distance**—the distance on the X-axis between each repetition of the mill feature. The right-hand rule determines if X Distance is positive or negative.
- **A Distance**—the distance (angle) on the rotary-axis between each repetition of the mill feature.
- **B Distance** (Rotary A Tilt B configuration only)—the distance (angle) on the tilt axis between each repetition of the mill feature.

A rotary Pattern End data block is required to end the sequence repeated in the Rotary Pattern Loop data block.

Rotary pattern and standard pattern data blocks can be nested (i.e., entirely contained within another pattern). The order of execution for nested rotary pattern and standard pattern data blocks is from the inside to the outside.

This is an example of nested data blocks in a program (the indentations are used illustrate the order of execution; your Conversational part program will not have these indentations).

In the following example, this is the order of execution:

1. Holes and rotary Mill Frame operations contained within the Pattern Loop Rotate data block.
2. Standard Mill Circle operation contained within the Rotary Pattern Loop data block.

Rotary Pattern Loop

<table>
<thead>
<tr>
<th>Standard Mill Circle (or any other rotary or non-rotary data blocks)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pattern Loop Rotate</td>
</tr>
<tr>
<td>Holes operation</td>
</tr>
<tr>
<td>Rotary Mill Frame (or other rotary or non-rotary data blocks)</td>
</tr>
<tr>
<td>Pattern End (ends the Pattern Loop Rotate)</td>
</tr>
<tr>
<td>Pattern End (ends the Rotary Pattern Loop)</td>
</tr>
</tbody>
</table>

Only one Rotary Position data block is required for nested Rotary Pattern Loop data blocks.
Rotary Locations

The Rotary Pattern Locations data block is used to create a list of locations of a mill feature repeated on a cylinder. Up to 999 copies can be located anywhere on the part.

Each set of locations listed on the Rotary Patterns Location data block is a specific location on the cylinder, relative to part zero. If you want the mill feature to be cut at part zero, you must define the coordinates of part zero in one of the sets of locations.

These are the Rotary Pattern Locations fields for Rotary A and Rotary A Tilt B:

- **X**—X-axis coordinate where the mill feature will be located on the part.
- **Y**—Y-axis coordinate where the mill feature will be located on the part.
- **Z**—offset for the Z axis. Z is used when a vise or other fixture holds multiple parts at different levels.
- **A**—rotary-axis coordinate (angle) where the mill feature will be located on the part.
- **B**—tilt-axis coordinate (angle) where the mill feature will be located on the part.

A rotary Pattern End data block is required to end the sequence in the Rotary Pattern Locations data block.

After the rotary Pattern End data block has executed, the orientation of the machine axes will remain at the position defined in the most recent Rotary Position data block.

If the part program continues beyond the Rotary Pattern operation, another Rotary Position data block will be required if the axes must be reoriented to a different position.

Rotary Transform Plane

The Rotary Transform Plane data block is available with multiple rotary configurations; see *Rotary Transform Plane Block, on page 5 - 5* for an explanation of the fields on this screen.
Pattern End

A Pattern End data block is required to end rotary Pattern Loop and rotary Pattern Locations operations. There are no fields in the Pattern End data block.

After the rotary Pattern End data block has executed, the orientation of the machine axes will remain at the position defined in the most recent Rotary Position data block.

If the part program continues beyond the Rotary Pattern operation, another Rotary Position data block will be required if the axes must be reoriented to a different position.

- A Rotary Pattern Loop data block only modifies the rotary operations contained within the pattern.
- The subsequent Rotary Position data block defines the orientation of the machine axes after the pattern is executed, and can provide a reference point if Recovery Restart is used.
- If the Rotary Pattern End data block is the last data block of the part program, a subsequent Rotary Position data block is not necessary.
- If a non-rotary operation data block (e.g., holes operation) is used in a rotary loop, a Rotary Position data block is required before the non-rotary operation data block to position the axes before the operation is executed in the loop.

Rotary Parameters

The Rotary Parameters data block is available with multiple rotary configurations; see *Rotary Parameters, on page 5 - 13* for an explanation of the fields on this screen.
Tilt A Rotary C Configuration

Rotary Position

The Rotary Position data block is available with multiple rotary configurations; see Rotary Position Block, on page 5 - 11 for an explanation of the fields on this screen.

Rotary Lines and Arcs

Rotary line and arc segments are used to create a rotary mill contour. A rotary mill contour is similar to a standard mill contour, except the rotary contour is wrapped around a cylinder.

Start Segment

These are the fields on the Rotary AC Mill Contour Start segment data block:

- **C Angle**—rotary-axis coordinate (angle) for the first segment’s starting point.
- **Y Start**—Y-axis coordinate for first segment’s starting point.
- **Z Start**—point where the Z-axis plunge feedrate begins.
- **Z Bottom**—the bottom of the finished segment and the location where the Z-axis plunge feedrate ends.

Line Segment

These are the fields on the Rotary AC Mill Contour Line segment data block:

- **C End**—C-axis coordinate for ending point of the line.
- **Y End**—rotary-axis coordinate (angle) for the ending point of the line.
- **Z End**—Z-axis coordinate for the ending point of the line. The Z End value is the depth of the cut at the line end point and is carried forward from the previous segment. Retain the Z End value or type in a new value.
- **C Start**—C-axis coordinate start point programmed in the previous segment. The value of C Start can be changed only in the segment in which it was created.
- **Y Start**—rotary-axis start point coordinate programmed in the previous segment. The value of Y Start can be changed only in the segment in which it was created.
- **Z Start**—point where the Z-axis plunge feedrate begins.

Arc Segment

These are the fields on the Rotary AC Mill Contour Arc segment data block:

- **Direction**—the direction of the arc from the start point, Clockwise or Counter Clockwise.
- **C End**—rotary-axis coordinate (angle) of the arc’s end point.
• **Y End**—Y-axis coordinate of the arc’s end point.
• **Z End**—point where the Z-axis plunge into the cylinder ends.
• **C Center**—rotary-axis coordinate (angle) of the arc’s center point.
• **Y Center**—Y-axis coordinate of the arc’s center point.
• **Radius**—the radius of the arc, if known.
• **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°.
• **C Start**—rotary-axis coordinate start point programmed in the previous segment. The value of C Start can be changed only in the segment in which it was created.
• **Y Start**—Y-axis coordinate start point programmed in the previous segment. The value of Y Start can be changed only in the segment in which it was created.
• **Z Start**—Z-axis coordinate start point programming in the previous segment. The value of Z Start can be changed only in the segment in which it was created.

**Blend Arc**

These are the fields on the Rotary AC Mill Contour Blend Arc segment data block:

• **Radius**—the radius of the arc, if known.
• **C End**—rotary-axis coordinate (angle) of the arc’s end point.
• **Y End**—Y-axis coordinate of the arc’s end point.
• **C Center**—rotary-axis coordinate (angle) of the arc’s center point.
• **Y Center**—Y-axis coordinate of the arc’s center point.
• **Direction**—the direction of the arc from the start point, Clockwise or Counterclockwise.
• **C Start**—rotary-axis coordinate start point programmed in the previous segment. The value of C Start can be changed only in the segment in which it was created.
• **Y Start**—Y-axis coordinate start point programmed in the previous segment. The value of Y Start can be changed only in the segment in which it was created.

**Mill Contour End Block**

A Contour End data block is required to end rotary Mill Contour operations. There are no fields in the Contour End data block.
Rotary Circle

A rotary mill circle is similar to a standard mill circle, except the rotary mill circle is wrapped around a cylinder. These are the fields on the Rotary Mill Circle data block:

These are the fields on the Rotary AC Mill Circle data block:

- **C Center**—rotary-axis coordinate (angle) for the center point of the circle.
- **Y Center**—Y-axis coordinate for the center point of the circle.
- **Radius**—radius of the milled circle.
- **Z Start**—location where the Z-axis plunge feedrate begins.
- **Z Bottom**—location where the Z-axis plunge feedrate ends.

Rotary Frame

A rotary mill frame is similar to a mill frame, except the rotary mill frame is wrapped around a cylinder.

These are the fields on the Rotary AC Mill Frame data block:

- **C Corner**—rotary-axis coordinate (angle) of the reference corner of the frame.
- **Y Corner**—Y-axis coordinate of any one of the four corners of the frame. This corner becomes the *reference corner*.
- **C Length**—rotary-axis coordinate (angle) from the reference corner.
  - C Length is positive if the location of the frame is clockwise from the reference corner.
  - C Length is negative if the location of the frame is counter clockwise from the reference corner.
- **Y Length**—Y-axis coordinate measured from the reference corner.
  - Y Length is positive if the reference corner is at the left side of the rectangle.
  - Y Length is negative if the reference corner is at the right side of the rectangle.
- **Z Start**—start point for the Z-axis plunge feedrate.
- **Z Bottom**—end point for the Z-axis plunge feedrate.
- **Corner Radius**—radius of the reference corner. This value will be the same for all corners of the rotary frame.
Rotary Patterns

The Rotary Patterns data block is available with multiple rotary configurations; see *Rotary Patterns, on page 5 - 12* for an explanation of the softkeys on this screen. The specific pattern data blocks for Tilt A Rotary C are explained below.

Rotary Loop

A rotary loop defines the number and locations a mill feature will be repeated on a cylinder. The initial mill feature is located at part zero and all subsequent copies are at a specific distance along the X-A axis of the cylinder.

These are the fields on the Rotary Pattern Loop data block for Tilt A Rotary C:

- **Number**—total number of times the mill feature will be cut on the cylinder.
- **Z Distance**—the distance on the Z-axis between each repetition of the mill feature. The right-hand rule determines if Distance is positive or negative.
- **A Distance**—the distance (angle) on the rotary-axis between each repetition of the mill feature.
- **C Distance**—the distance (angle) on the tilt axis between each repetition of the mill feature.

A rotary Pattern End data block is required to end the sequence repeated in the Rotary Pattern Loop data block.

Rotary pattern and standard pattern data blocks can be nested (i.e., entirely contained within another pattern). The order of execution for nested rotary pattern and standard pattern data blocks is from the inside to the outside.

This is an example of nested data blocks in a program (the indentations are used illustrate the order of execution; your Conversational part program will not have these indentations).

In the following example, this is the order of execution:

1. Holes and rotary Mill Frame operations contained within the Pattern Loop Rotate data block.
2. Standard Mill Circle operation contained within the Rotary Pattern Loop data block.

Rotary Pattern Loop

- Standard Mill Circle (or any other rotary or non-rotary data blocks)
- Pattern Loop Rotate
- Holes operation
- Rotary Mill Frame (or other rotary or non-rotary data blocks)
- Pattern End (ends the Pattern Loop Rotate)
- Pattern End (ends the Rotary Pattern Loop)

Only one Rotary Position data block is required for nested Rotary Pattern Loop data blocks.
Rotary Locations

The Rotary Pattern Locations data block is used to create a list of locations of a mill feature repeated on a cylinder. Up to 999 copies can be located anywhere on the part.

Each set of locations listed on the Rotary Patterns Location data block is a specific location on the cylinder, relative to part zero. If you want the mill feature to be cut at part zero, you must define the coordinates of part zero in one of the sets of locations.

These are the fields on the Rotary Pattern Locations data block for Tilt A Rotary C:

- **X**—X-axis coordinate where the mill feature will be located on the part.
- **Y**—Y-axis coordinate where the mill feature will be located on the part.
- **Z**—offset for the Z axis. Z is used when a vise or other fixture holds multiple parts at different levels.
- **A**—rotary-axis coordinate (angle) where the mill feature will be located on the part.
- **C**—rotary-axis coordinate (angle) where the mill feature will be located on the part.

A rotary Pattern End data block is required to end the sequence in the Rotary Pattern Locations data block.

After the rotary Pattern End data block has executed, the orientation of the machine axes will remain at the position defined in the most recent Rotary Position data block.

If the part program continues beyond the Rotary Pattern operation, another Rotary Position data block will be required if the axes must be reoriented to a different position.

Rotary Transform Plane

The Rotary Transform Plane data block is available with multiple rotary configurations; see *Rotary Transform Plane Block, on page 5 - 5* for an explanation of the fields on this screen.
**Pattern End**

A Pattern End data block is required to end rotary Pattern Loop and rotary Pattern Locations operations. There are no fields in the Pattern End data block.

After the rotary Pattern End data block has executed, the orientation of the machine axes will remain at the position defined in the most recent Rotary Position data block.

If the part program continues beyond the Rotary Pattern operation, another Rotary Position data block will be required if the axes must be reoriented to a different position.

- A Rotary Pattern Loop data block only modifies the rotary operations contained within the pattern.
- The subsequent Rotary Position data block defines the orientation of the machine axes after the pattern is executed, and can provide a reference point if Recovery Restart is used.
- If the Rotary Pattern End data block is the last data block of the part program, a subsequent Rotary Position data block is not necessary.
- If a non-rotary operation data block (e.g., holes operation) is used in a rotary loop, a Rotary Position data block is required before the non-rotary operation data block to position the axes before the operation is executed in the loop.

**Rotary Parameters**

The Rotary Parameters data block is available with multiple rotary configurations; see *Rotary Parameters, on page 5-13* for an explanation of the fields on this screen.
Rotary B Configuration

Rotary B configuration is used with horizontal (HMX) machines.

Rotary Position

The Rotary Position data block is available with multiple rotary configurations; see Rotary Position Block, on page 5 - 11 for an explanation of the fields on this screen.

Rotary Milling

For rotary milling programs on an HMX machine, Universal Rotary configuration must be used; see Universal Rotary Configuration, on page 5 - 35.

Rotary Patterns

The Rotary Patterns data block is available with multiple rotary configurations; see Rotary Patterns, on page 5 - 12 for an explanation of the softkeys on this screen. The specific pattern data blocks for Rotary B configuration are explained below.

Rotary Loop

A rotary loop defines the number and locations a mill feature will be repeated on a cylinder. The initial mill feature is located at part zero and all subsequent copies are at a specific distance along the X-A axis of the cylinder.

These are the fields on the Rotary Pattern Loop data block for Rotary B:

- **Number**—total number of times the mill feature will be cut on the cylinder.
- **Z Distance**—the distance on the Z-axis between each repetition of the mill feature. The right-hand rule determines if Distance is positive or negative.
- **B Distance**—the distance (angle) on the rotary axis between each repetition of the mill feature.

A rotary Pattern End data block is required to end the sequence repeated in the Rotary Pattern Loop data block.
Rotary pattern and standard pattern data blocks can be nested (i.e., entirely contained within another pattern). The order of execution for nested rotary pattern and standard pattern data blocks is from the inside to the outside.

This is an example of nested data blocks in a program (the indentations are used illustrate the order of execution; your Conversational part program will not have these indentations).

In the following example, this is the order of execution:

1. Holes and rotary Mill Frame operations contained within the Pattern Loop Rotate data block.
2. Standard Mill Circle operation contained within the Rotary Pattern Loop data block.

Rotary Pattern Loop
Standard Mill Circle (or any other rotary or non-rotary data blocks)
Pattern Loop Rotate
Holes operation
Rotary Mill Frame (or other rotary or non-rotary data blocks)
Pattern End (ends the Pattern Loop Rotate)
Pattern End (ends the Rotary Pattern Loop)

Only one Rotary Position data block is required for nested Rotary Pattern Loop data blocks.

Rotary Locations

The Rotary Pattern Locations data block is used to create a list of locations of a mill feature repeated on a cylinder. Up to 999 copies can be located anywhere on the part.

Each set of locations listed on the Rotary Patterns Location data block is a specific location on the cylinder, relative to part zero. If you want the mill feature to be cut at part zero, you must define the coordinates of part zero in one of the sets of locations.

These are the fields on the Rotary Pattern Locations data block for Rotary B:

- **X**—X-axis coordinate where the mill feature will be located on the part.
- **Y**—Y-axis coordinate where the mill feature will be located on the part.
- **Z**—offset for the Z axis. Z is used when a vise or other fixture holds multiple parts at different levels.
- **B**—rotary-axis coordinate (angle) where the mill feature will be located on the part.

A rotary Pattern End data block is required to end the sequence in the Rotary Pattern Locations data block.
After the rotary Pattern End data block has executed, the orientation of the machine axes will remain at the position defined in the most recent Rotary Position data block.

If the part program continues beyond the Rotary Pattern operation, another Rotary Position data block will be required if the axes must be reoriented to a different position.

**Rotary Transform Plane**

The Rotary Transform Plane data block is available with multiple rotary configurations; see *Rotary Transform Plane Block, on page 5 - 5* for an explanation of the fields on this screen.

**Pattern End**

A Pattern End data block is required to end rotary Pattern Loop and rotary Pattern Locations operations. There are no fields in the Pattern End data block.

After the rotary Pattern End data block has executed, the orientation of the machine axes will remain at the position defined in the most recent Rotary Position data block.

If the part program continues beyond the Rotary Pattern operation, another Rotary Position data block will be required if the axes must be reoriented to a different position.

- A Rotary Pattern Loop data block only modifies the rotary operations contained within the pattern.
- The subsequent Rotary Position data block defines the orientation of the machine axes after the pattern is executed, and can provide a reference point if Recovery Restart is used.
- If the Rotary Pattern End data block is the last data block of the part program, a subsequent Rotary Position data block is not necessary.
- If a non-rotary operation data block (e.g., holes operation) is used in a rotary loop, a Rotary Position data block is required before the non-rotary operation data block to position the axes before the operation is executed in the loop.

**Rotary Parameters**

The Rotary Parameters data block is available with multiple rotary configurations; see *Rotary Parameters, on page 5 - 13* for an explanation of the fields on this screen.
Tilt B Rotary C Configuration

Rotary Position

The Rotary Position data block is available with multiple rotary configurations; see Rotary Position Block, on page 5 - 11 for an explanation of the fields on this screen.

Rotary Lines and Arcs

Rotary line and arc segments are used to create a rotary mill contour. A rotary mill contour is similar to a standard mill contour, except the rotary contour is wrapped around a cylinder.

Start Segment

These are the fields on the Rotary BC Mill Contour Start segment data block:

- **Z Start**—Z-axis coordinate for first segment’s starting point.
- **C Angle**—rotary-axis coordinate (angle) for the first segment’s starting point.
- **Radius Start**—point where the Z-axis plunge feedrate begins.
- **Radius Bottom**—the bottom of the finished segment and the location where the Z-axis plunge feedrate ends.

Line Segment

These are the fields on the Rotary BC Mill Contour Line segment data block:

- **Z End**—Z-axis coordinate for the ending point of the line. The Z End value is the depth of the cut at the line end point and is carried forward from the previous segment. Retain the Z End value or type in a new value.
- **C End**—C-axis coordinate for ending point of the line.
- **Radius End**—rotary-axis coordinate (angle) for the ending point of the line.
- **Z Start**—point where the Z-axis plunge feedrate begins.
- **C Start**—C-axis coordinate start point programmed in the previous segment. The value of C Start can be changed only in the segment in which it was created.
- **Radius Start**—rotary-axis start point coordinate programmed in the previous segment. The value of Radius Start can be changed only in the segment in which it was created.

Arc Segment

These are the fields on the Rotary BC Mill Contour Arc segment data block:

- **Direction**—the direction of the arc from the start point, Clockwise or Counter Clockwise.
- **Z End**—point where the Z-axis plunge into the cylinder ends.
- **C End**—rotary-axis coordinate (angle) of the arc’s end point.
• **Radius End**—Y-axis coordinate of the arc’s end point.
• **Z Center**—Z-axis coordinate of the arc’s center point.
• **C Center**—rotary-axis coordinate (angle) of the arc’s center point.
• **Arc Radius**—the radius of the arc, if known.
• **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°.
• **Z Start**—Z-axis coordinate start point programming in the previous segment. The value of Z Start can be changed only in the segment in which it was created.
• **C Start**—rotary-axis coordinate start point programmed in the previous segment. The value of C Start can be changed only in the segment in which it was created.
• **Radius Start**—Y-axis coordinate start point programmed in the previous segment. The value of Radius Start can be changed only in the segment in which it was created.

**Blend Arc Segment**

These are the fields on the Rotary BC Mill Contour Blend Arc segment data block:

• **Arc Radius**—the radius of the arc, if known.
• **Z End**—Z-axis coordinate of the arc’s end point.
• **C End**—rotary-axis coordinate (angle) of the arc’s end point.
• **Z Center**—Z-axis coordinate of the arc’s center point.
• **C Center**—rotary-axis coordinate (angle) of the arc’s center point.
• **Direction**—the direction of the arc from the start point, Clockwise or Counterclockwise.
• **Z Start**—Z-axis coordinate start point programmed in the previous segment. The value of Z Start can be changed only in the segment in which it was created.
• **C Start**—rotary-axis coordinate start point programmed in the previous segment. The value of C Start can be changed only in the segment in which it was created.

**Mill Contour End Block**

A Contour End data block is required to end rotary Mill Contour operations. There are no fields in the Contour End data block.
## Rotary Circle

A rotary mill circle is similar to a standard mill circle, except the rotary mill circle is wrapped around a cylinder. These are the fields on the Rotary Mill Circle data block:

These are the fields on the Rotary BC Circle data block:

- **Z Center**—Z-axis coordinate for the center point of the circle.
- **C Center**—rotary-axis coordinate (angle) for the center point of the circle.
- **Radius**—radius of the milled circle.
- **Radius Start**—location where the Z-axis plunge feedrate begins.
- **Radius Bottom**—location where the Z-axis plunge feedrate ends.

## Rotary Frame

A rotary mill frame is similar to a mill frame, except the rotary mill frame is wrapped around a cylinder.

These are the fields on the Rotary BC Mill Frame data block:

- **Z Corner**—Z-axis coordinate of any one of the four corners of the frame. This corner becomes the reference corner.
- **C Corner**—rotary-axis coordinate (angle) of the reference corner of the frame.
- **Z Length**—Z-axis coordinate measured from the reference corner.
  - Z Length is positive if the reference corner is at the left side of the rectangle.
  - Z Length is negative if the reference corner is at the right side of the rectangle.
- **C Length**—rotary-axis coordinate (angle) from the reference corner.
  - C Length is positive if the location of the frame is clockwise from the reference corner.
  - C Length is negative if the location of the frame is counter clockwise from the reference corner.
- **Radius Start**—start point for the Z-axis plunge feedrate.
- **Radius Bottom**—end point for the Z-axis plunge feedrate.
- **Corner Radius**—radius of the reference corner. This value will be the same for all corners of the rotary frame.
Rotary Patterns

The Rotary Patterns data block is available with multiple rotary configurations; see Rotary Patterns, on page 5 - 12 for an explanation of the softkeys on this screen. The specific pattern data blocks for Tilt B Rotary C configuration are explained below.

Rotary Loop

A rotary loop defines the number and locations a mill feature will be repeated on a cylinder. The initial mill feature is located at part zero and all subsequent copies are at a specific distance along the X-A axis of the cylinder.

These are the fields on the Rotary Pattern Loop data block for Tilt B Rotary C:

- **Number**—total number of times the mill feature will be cut on the cylinder.
- **Z Distance**—the distance on the Z-axis between each repetition of the mill feature. The right-hand rule determines if Distance is positive or negative.
- **C Distance**—the distance (angle) on the C-axis between each repetition of the mill feature.

A rotary Pattern End data block is required to end the sequence repeated in the Rotary Pattern Loop data block.

Rotary pattern and standard pattern data blocks can be nested (i.e., entirely contained within another pattern). The order of execution for nested rotary pattern and standard pattern data blocks is from the inside to the outside.

This is an example of nested data blocks in a program (the indentations are used illustrate the order of execution; your Conversational part program will not have these indentations).

In the following example, this is the order of execution:

1. Holes and rotary Mill Frame operations contained within the Pattern Loop Rotate data block.
2. Standard Mill Circle operation contained within the Rotary Pattern Loop data block.

Only one Rotary Position data block is required for nested Rotary Pattern Loop data blocks.
**Rotary Locations**

The Rotary Pattern Locations data block is used to create a list of locations of a mill feature repeated on a cylinder. Up to 999 copies can be located anywhere on the part.

Each set of locations listed on the Rotary Patterns Location data block is a specific location on the cylinder, relative to part zero. If you want the mill feature to be cut at part zero, you must define the coordinates of part zero in one of the sets of locations.

These are the fields on the Rotary Pattern Locations data block for Tilt B Rotary C:

- **X**—X-axis coordinate where the mill feature will be located on the part.
- **Y**—Y-axis coordinate where the mill feature will be located on the part.
- **Z**—offset for the Z axis. Z is used when a vise or other fixture holds multiple parts at different levels.
- **B**—rotary-axis coordinate (angle) where the mill feature will be located on the part.
- **C**—rotary-axis coordinate (angle) where the mill feature will be located on the part.

A rotary Pattern End data block is required to end the sequence in the Rotary Pattern Locations data block.

After the rotary Pattern End data block has executed, the orientation of the machine axes will remain at the position defined in the most recent Rotary Position data block.

If the part program continues beyond the Rotary Pattern operation, another Rotary Position data block will be required if the axes must be reoriented to a different position.

**Rotary Transform Plane**

The Rotary Transform Plane data block is available with multiple rotary configurations; see *Rotary Transform Plane Block, on page 5 - 5* for an explanation of the fields on this screen.
Pattern End

A Pattern End data block is required to end rotary Pattern Loop and rotary Pattern Locations operations. There are no fields in the Pattern End data block.

After the rotary Pattern End data block has executed, the orientation of the machine axes will remain at the position defined in the most recent Rotary Position data block.

If the part program continues beyond the Rotary Pattern operation, another Rotary Position data block will be required if the axes must be reoriented to a different position.

- A Rotary Pattern Loop data block only modifies the rotary operations contained within the pattern.
- The subsequent Rotary Position data block defines the orientation of the machine axes after the pattern is executed, and can provide a reference point if Recovery Restart is used.
- If the Rotary Pattern End data block is the last data block of the part program, a subsequent Rotary Position data block is not necessary.
- If a non-rotary operation data block (e.g., holes operation) is used in a rotary loop, a Rotary Position data block is required before the non-rotary operation data block to position the axes before the operation is executed in the loop.

Rotary Parameters

The Rotary Parameters data block is available with multiple rotary configurations; see Rotary Parameters, on page 5 - 13 for an explanation of the field on this screen.
Universal Rotary Configuration

Rotary Position Block

A Rotary Position data block is used to move the machine’s axes to a specific location at Rapid feedrate. A new Rotary Position data block is required each time the rotary-axis must be repositioned in the part program.

The tool will move to the Retract Plane, Safety Plane, or Home Z position before the table rotates into position each time a Rotary Position data block with rotary-axis positioning is executed in a Conversational part program.

A Rotary Position data block should be included as the first data block in a Conversational rotary part program to define the initial orientation of the rotary axis.

These are the fields on the Rotary Position data block:

- **Stop**—when set to Yes, pauses program execution after the Rotary Position data block is executed. To restart the program, press the Start Cycle console button.

First Move (Z Retract)

- **Retract Type**—select the Z level to which Z-axis retracts at the beginning of the Rotary Position data block.
  - **Z Home**—Z-axis will move at Rapid feedrate to the position defined in Z Machine field.
  - **Z Position**—Z-axis will move at Rapid feedrate to the position entered in the Z Position field.
  - **Z Position**—the position to which Z-axis retracts when Z Position is selected for the Retract Type.

Second Move

- **Enable**—specify YES to enable axis movement
- **X Position**—defines the location to which the X axis will move before the rotary-axis orients.
- **Y Position**—defines the location to which the Y axis will move before the rotary-axis orients.

Third Move (End Position)

- **X Position**—X-axis coordinate relative to Part Zero X. Enabled when the Enable field is set to YES.
- **Y Position**—Y-axis coordinate relative to Part Zero X. Enabled when the Enable field is set to YES.
- **IV Angle**—rotary-axis coordinate (angle) relative to Part Zero A or Part Zero C, depending on the configuration of your machine. Enabled when the Enable field is set to YES.
• **V Angle**—rotary-axis coordinate (angle) relative to Part Zero A or Part Zero C, depending on the configuration of your machine. Enabled when the Enable field is set to YES.

**Rotary Lines and Arcs**

Rotary line and arc segments are used to create a rotary mill contour. A rotary mill contour is similar to a standard mill contour, except the rotary contour is wrapped around a cylinder.

**Start Segment**

These are the fields on the Universal Rotary Mill Contour Start segment data block:

- **Axis Start**—axis coordinate for first segment’s starting point.
- **Angle Start**—rotary-axis coordinate (angle) for the first segment’s starting point.
- **Radius Start**—point where the Z-axis plunge feedrate begins.
- **Radius Bottom**—the bottom of the finished segment and the location where the Z-axis plunge feedrate ends.

**Line Segment**

These are the fields on the Universal Rotary Mill Contour line segment data block:

- **Axis End**—axis coordinate for the ending point of the line. The Axis End value is the depth of the cut at the line end point and is carried forward from the previous segment. Retain the Axis End value or type in a new value.
- **Angle End**—axis coordinate for ending point of the line.
- **Radius End**—rotary-axis coordinate (angle) for the ending point of the line.
- **Axis Start**—axis coordinate start point programmed in the previous segment. The value of Axis Start can be changed only in the segment in which it was created.
- **Angle Start**—point where the Z-axis plunge feedrate begins.
- **Radius Start**—rotary-axis start point coordinate programmed in the previous segment. The value of Radius Start can be changed only in the segment in which it was created.

**Arc Segment**

These are the fields on the Universal Rotary Mill Contour Arc segment data block:

- **Direction**—the direction of the arc from the start point, Clockwise or Counter Clockwise.
- **Axis End**—rotary-axis coordinate (angle) of the arc’s end point.
- **Angle End**—Y-axis coordinate of the arc’s end point.
- **Radius End**—point where the Z-axis plunge into the cylinder ends.
- **Axis Center**—Z-axis coordinate of the arc’s center point.
- **Angle Center**—rotary-axis coordinate (angle) of the arc’s center point.
• **Radius**—the radius of the arc, if known.

• **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°.

• **Axis Start**—axis coordinate start point programmed in the previous segment. The value of Axis Start can be changed only in the segment in which it was created.

• **Angle Start**—rotary-axis coordinate start point programmed in the previous segment. The value of Angle Start can be changed only in the segment in which it was created.

• **Radius Start**—Z-axis coordinate start point programming in the previous segment. The value of Z Start can be changed only in the segment in which it was created.

**Blend Arc Segment**

These are the fields on the Universal Rotary Mill Contour Blend Arc segment data block:

• **Direction**—the direction of the arc from the start point, Clockwise or Counter Clockwise.

• **Axis End**—rotary-axis coordinate (angle) of the arc’s end point.

• **Angle End**—Y-axis coordinate of the arc’s end point.

• **Axis Center**—Z-axis coordinate of the arc’s center point.

• **Angle Center**—rotary-axis coordinate (angle) of the arc’s center point.

• **Radius**—the radius of the arc, if known.

• **Axis Start**—axis coordinate start point programmed in the previous segment. The value of Axis Start can be changed only in the segment in which it was created.

• **Angle Start**—rotary-axis coordinate start point programmed in the previous segment. The value of Angle Start can be changed only in the segment in which it was created.
Rotary Circle

A rotary mill circle is similar to a standard mill circle, except the rotary mill circle is wrapped around a cylinder. These are the fields on the Rotary Mill Circle data block:

These are the fields on the Universal Rotary Mill Circle data block:

- **Axis Center**—Z-axis coordinate for the center point of the circle.
- **Angle Center**—rotary-axis coordinate (angle) for the center point of the circle.
- **Radius**—radius of the milled circle.
- **Radius Start**—location where the Z-axis plunge feedrate begins.
- **Radius Bottom**—location where the Z-axis plunge feedrate ends.

Rotary Frame

A rotary mill frame is similar to a mill frame, except the rotary frame is wrapped around a cylinder. Rotary Frame block supports the creation of frames with or without uniform corners.

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

- **Axis Start**—axis coordinate of any one of the four corners of the frame. This corner becomes the reference corner.
- **Angle Start**—rotary-axis coordinate (angle) of the reference corner of the frame.
- **Axis Length**—axis coordinate measured from the reference corner.
  - Axis Length is positive if the reference corner is at the left side of the rectangle.
  - Axis Length is negative if the reference corner is at the right side of the rectangle.
- **Angle Length**—rotary-axis coordinate (angle) from the reference corner.
  - Angle Length is positive if the location of the frame is clockwise from the reference corner.
  - Angle Length is negative if the location of the frame is counter clockwise from the reference corner.
• **Radius Start**—start point for the Z-axis plunge feedrate.

• **Radius Bottom**—end point for the Z-axis plunge feedrate.

• **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.

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.

Each corner of a frame can be programmed with a radius or chamfer of a different size, on the Corners tab:

```
<table>
<thead>
<tr>
<th>GEOMETRY</th>
<th>CORNERS</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>ARC</td>
</tr>
<tr>
<td>2</td>
<td>ARC</td>
</tr>
<tr>
<td>3</td>
<td>LINE</td>
</tr>
<tr>
<td>4</td>
<td>ARC</td>
</tr>
</tbody>
</table>
```

**Figure 5–5. Universal Rotary 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 Rotary Frame:

**Figure 5–6. Universal Rotary Frame example**
Rotary True Type Font

Rotary True Type Font is available with Universal Rotary configuration only. These are the True Type Lettering fields:

- **Axis Ref Location**—the location of the reference point in the text:
  - Start of text
  - End of text
  - center of text

- **Axis Reference**—the reference coordinate of the Sizing Box.

- **Angle Ref Location**—the location of the reference point in the text:
  - Bottom of text
  - Top of text
  - Center of text

- **Angle Reference**—the reference coordinate of the Sizing Box.

- **Text Width**—the width of the text.

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

- **Text**—the text to be milled.

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

- **Radius Bottom**—Z bottom coordinate

- **Orientation**—the angle to which the text should be rotated.

- **Mapping Mode**—specifies the height dimension of the lettering:
  - 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 of the lettering. Ascenders are calculated automatically.
  - Ascended—defines the height of the body and ascenders 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 5–7. Y Mapping](image)
• **Font**—the lettering font. The font selection dialog box is opened with the Select New Font softkey, when the cursor is in the Text field.

<table>
<thead>
<tr>
<th>Available Fonts for 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>

• **Outside Cylinder**—specifies if text is milled on the inside or outside of a cylinder:
  • Yes—text is milled on the outside of a cylinder.
  • No—text is milled on the inside of a cylinder.

**Rotary Slot**

The Rotary Slot block creates a line or arc shape of any width on a cylinder.

*See Mill Slot, on page 2 - 41 of the Conversation Part Programming manual for additional information.*

**Start tab**

Fields on the Start tab 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 centerarc segment programmed in the Geometry tab. Half of the programmed width is applied to each side of the line or arc.

• **Axis Start**—indicates the axis coordinate for the starting point of the line or arc segment.

• **Angle Start**—indicates the rotary-axis coordinate (angle) for the starting point of the line or arc segment.

• **Radius Start**—indicates the plunge feedrate start point.

• **Radius Bottom**—indicates the plunge bottom point.
Geometry tab

Line geometry fields are:

- **Axis End**—indicates the axis end point of the line segment.
- **Angle End**—indicates the angle end point of the line segment.

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

Arc geometry fields are:

- **Axis End**—indicates the axis end point of the line segment.
- **Angle End**—indicates the angle end point of the line segment.
- **Axis Center**—indicates the axis center point of the arc.
- **Angle Center**—indicates the angle center point of the arc.
- **Sweep Angle**—indicates the angular distance from the start point to the end point around the center point. The range is $-360^\circ$ to $360^\circ$.
- **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

Caps tab 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**—indicates the blend radius for the corners of the Start Cap or End Cap when set to Line.
Rotary Polygon

The Rotary Polygon data block mills a multi-sided contour with equal-length sides on a cylinder.

See Mill Polygon, on page 2 - 46 of the Conversational Part Programming manual for more information.

The fields are:

- **Number Of Sides**—specifies the number of sides the polygon will have, from three to 100.
- **Axis Center**—identifies the axis coordinate for the center of the polygon.
- **Angle Center**—identifies the angle 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:
  - **Inner Diameter**—the diameter of a circle contained within the polygon, touching at the center of each edge:
  - **Side Length**—the length of one side of the polygon:
  - **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:

- **Radius Start**—identifies the point at which the Plunge Feedrate begins.
- **Radius Bottom**—identifies the point at which the Mill Feedrate begins.
- **Corner Radius**—identifies the radius of each corner of the polygon.
Rotary Stick Lettering

The Rotary Stick Lettering block allows text to be wrapped around a cylinder.

<table>
<thead>
<tr>
<th>BLOCK</th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td>X REF LOCATION</td>
<td>START</td>
</tr>
<tr>
<td>AXIS REFERENCE</td>
<td>2.0000</td>
</tr>
<tr>
<td>ANGLE REF LOCATION</td>
<td>BOTTOM</td>
</tr>
<tr>
<td>ANGLE REFERENCE</td>
<td>57.296</td>
</tr>
<tr>
<td>TEXT WIDTH</td>
<td>5.0000</td>
</tr>
<tr>
<td>TEXT HEIGHT</td>
<td>1.0000</td>
</tr>
<tr>
<td>TEXT</td>
<td>HURCO</td>
</tr>
</tbody>
</table>

The Rotary Stick Lettering block allows text to be wrapped around a cylinder.

Fields are:

- **Axis Ref Location**—the location of the reference point in the text:
  - Start of text
  - End of text
  - Center of text
- **Axis Reference**—the reference coordinate of the Sizing Box.
- **Angle Ref Location**—the location of the reference point in the text:
  - Bottom of text
  - Top of text
  - Center of text
- **Angle Reference**—the reference coordinate of the Sizing Box.
- **Text Width**—the width of the text.
- **Text Height**—the height of the text.
- **Text**—the text to be milled.
- **Radius Start**—the coordinate at which plunge feedrate begins.
- **Radius Bottom**—the end point for the plunge feedrate.
- **Orientation**—the angle to which the text should be rotated.

*Figure 5–8. Rotary Stick Lettering screen*
Rotary Patterns

The Rotary Patterns data block is available with multiple rotary configurations; see Rotary Patterns, on page 5 - 12 for an explanation of the softkeys on this screen. The specific pattern data blocks for Universal Rotary are explained below.

Rotary Loop

A rotary loop defines the number and locations a mill feature will be repeated on a cylinder. The initial mill feature is located at part zero and all subsequent copies are at a specific distance along the X-A axis of the cylinder.

These are the fields on the Universal Rotary Pattern Loop data block:

- **Number**—total number of times the mill feature will be cut on the cylinder.
- **Axis Distance**—the distance on the X-axis between each repetition of the mill feature. The right-hand rule determines if Distance is positive or negative.
- **Angle Distance**—the distance (angle) on the rotary-axis between each repetition of the mill feature.

A rotary Pattern End data block is required to end the sequence repeated in the Rotary Pattern Loop data block.

Rotary pattern and standard pattern data blocks can be nested (i.e., entirely contained within another pattern). The order of execution for nested rotary pattern and standard pattern data blocks is from the inside to the outside.

This is an example of nested data blocks in a program (the indentations are used illustrate the order of execution; your Conversational part program will not have these indentations).

In the following example, this is the order of execution:

1. Holes and rotary Mill Frame operations contained within the Pattern Loop Rotate data block.
2. Standard Mill Circle operation contained within the Rotary Pattern Loop data block.

Only one Rotary Position data block is required for nested Rotary Pattern Loop data blocks.
Rotary Locations

The Rotary Pattern Locations data block is used to create a list of locations of a mill feature repeated on a cylinder. Up to 999 copies can be located anywhere on the part.

Each set of locations listed on the Rotary Patterns Location data block is a specific location on the cylinder, relative to part zero. If you want the mill feature to be cut at part zero, you must define the coordinates of part zero in one of the sets of locations.

These are the fields on the Universal Rotary Pattern Locations data block:

- **Axis**—Axis coordinate where the mill feature will be located on the part.
- **Angle**—Angle coordinate where the mill feature will be located on the part.

A rotary Pattern End data block is required to end the sequence in the Rotary Pattern Locations data block.

After the rotary Pattern End data block has executed, the orientation of the machine axes will remain at the position defined in the most recent Rotary Position data block.

If the part program continues beyond the Rotary Pattern operation, another Rotary Position data block will be required if the axes must be reoriented to a different position.

Rotary Rectangular

Rotary Rectangular Pattern Block is available only with the Universal Rotary configuration. Fields are:

- **Axis Number**—the number of times the programmed routine will be repeated along a line parallel to the rotary axis.
- **Angle Number**—the number of times the programmed routine will be repeated along the rotary axis.
- **Axis Distance**—the distance between the patterns parallel to the rotary axis.
- **Angle Distance**—the distance between the patterns along the rotary axis.

Rotary Mirror

Rotary Mirror Pattern Block is available only with the Universal Rotary configuration. Fields are:

- **Keep Original**—indicate if you want to machine the original pattern in addition to the mirrored pattern.
  - Yes—machine original pattern
  - No—machine only mirrored pattern
- **Cylinder Angle**—the coordinate of a point on the mirror line.
- **Cylinder Axis**—the coordinate of a point on the mirror line.
- **Angle**—the orientation of the mirror line which passes through the point defined above (measured in XY from 3 o’clock).
Rotary Transform Plane

Transform Plane is used in a Conversational rotary program to position part zero for a non-rotary data block to any orientation of any plane of your rotary- or tilt-axis machine. For example, use Transform Plane when you want to repeat a non-rotary feature (e.g., mill contour, holes) on multiple sides of a part.

Rotary Transform Plane End

The Transform Plane End block ends the Transform Plane operation.

After the Transform Plane End data block has executed, the orientation of the part will remain at the position defined in the most recent Rotary Position data block.

If the part program continues beyond the Transform Plane operation, another Rotary Position data block will be required if the part must be reorientated to a different position.

Pattern End

A Pattern End data block is required to end rotary Pattern Loop and rotary Pattern Locations operations. There are no fields in the Pattern End data block.

After the rotary Pattern End data block has executed, the orientation of the machine axes will remain at the position defined in the most recent Rotary Position data block.

If the part program continues beyond the Rotary Pattern operation, another Rotary Position data block will be required if the axes must be reoriented to a different position.

- A Rotary Pattern Loop data block only modifies the rotary operations contained within the pattern.
- The subsequent Rotary Position data block defines the orientation of the machine axes after the pattern is executed, and can provide a reference point if Recovery Restart is used.
- If the Rotary Pattern End data block is the last data block of the part program, a subsequent Rotary Position data block is not necessary.
- If a non-rotary operation data block (e.g., holes operation) is used in a rotary loop, a Rotary Position data block is required before the non-rotary operation data block to position the axes before the operation is executed in the loop.

Rotary Parameters

The Rotary Orientation field defines the axis configuration: Rotary A/AB, Rotary B, Rotary AC, Rotary BC, and User Defined. User Defined orientation requires the Axis of Rotation (Cylinder Axis vector) and Zero Angle vector. The Zero Angle vector describes the zero-degree Cylindrical Point vector direction with respect to the current coordinate system. These vectors must be perpendicular otherwise the control will throw an error. Other Rotary Orientation selections (Rotary A/AB, Rotary B, Rotary AC, Rotary BC) are available to automatically configure the Axis of Rotation and Zero Angle vectors to describe rotary features for Hurco’s current program type configurations.
Calculating X- and Z-Axis Positions After a Tilt-Axis Move

The initial location of a point on a cylindrical workpiece will be at different X and Z coordinates after the tilt-axis is moved into position. Some tilt-axis tables have part zero calibrated at 0° vertical, and some have part zero calibrated at 0° horizontal. Use the formulas below that correspond to the part zero calibration of your machine.

Part Zero Calibrated at Zero Degree Vertical

Follow these steps to determine the relative location of X and Z when the tilt-axis is positioned at 0° in the vertical plane (as shown in the figure below):

1. Use the formula \( L = Z_1 - Z_3 \) to calculate \( L \), the distance between the point’s initial location and the location of the same point after the workpiece is positioned to a known angle on the tilt-axis.

2. Use the formula \( X_2 = L \sin B \) to determine the location of \( X_2 \), the location of the X coordinate on the workpiece after the tilt-axis has been moved.

3. Use the formula \( Z_2 = L - (L \cos B) \) to determine \( Z_2 \), the location of the Z coordinate on the workpiece after the tilt-axis has been moved.

\[
\begin{array}{l|l}
1 & Z_1 \\
2 & Z_2 \\
3 & B \text{ (tilt-axis)} = 0^\circ \\
4 & Z_3 \\
5 & L \\
6 & X_2 \\
\end{array}
\]

*Figure 5-9. Vertical Tilt-Axis Example*
Part Zero Calibrated at Zero Degree Horizontal

Follow these steps to determine the relative location of X and Z when the tilt-axis is positioned at 0° in the horizontal plane (as shown in the figure below).

1. Use the formula \( L = X_1 - X_3 \) to calculate \( L \), the distance between the point’s initial location and the location of the same point after the workpiece is positioned to a known angle on the tilt-axis.

2. Use the formula \( X_2 = L - (L \cos B) \) to determine the location of \( X_2 \), the location of the X coordinate on the workpiece after the tilt-axis has been moved.

3. Use the formula \( Z_2 = L \sin B \) to determine the location of \( Z_2 \), the location of the Z coordinate on the workpiece after the tilt-axis has been moved.

---

**Figure 5–10. Horizontal Tilt-Axis Example**

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>( X_1 )</td>
</tr>
<tr>
<td>2</td>
<td>( X_2 )</td>
</tr>
<tr>
<td>3</td>
<td>( B ) (tilt-axis) = 0°</td>
</tr>
<tr>
<td>4</td>
<td>( X_3 )</td>
</tr>
<tr>
<td>5</td>
<td>( L )</td>
</tr>
<tr>
<td>6</td>
<td>( Z_2 )</td>
</tr>
</tbody>
</table>
Example Programs

Rotary Programming Examples

The figure below shows a Rotary Mill Frame created on a cylinder.

<table>
<thead>
<tr>
<th>No.</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>PART ZERO</td>
</tr>
<tr>
<td>2</td>
<td>A LENGTH (arrow indicates positive axis movement)</td>
</tr>
<tr>
<td>3</td>
<td>Reference Corner (X Corner/A Corner)</td>
</tr>
<tr>
<td>4</td>
<td>X LENGTH</td>
</tr>
</tbody>
</table>

*Figure 5–11. Rotary Mill Frame Part Drawing (cylinder shown for reference)*
This is a summary of the program used to create the part shown in the figure above:

<table>
<thead>
<tr>
<th>DB</th>
<th>Data Block/Field Name</th>
<th>Value</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>ROTARY PARAMETERS</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>CYLINDER RADIUS</td>
<td>2.0000</td>
<td>Defines the cylinder.</td>
</tr>
<tr>
<td>2</td>
<td>ROTARY POSITION</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>0.0000</td>
<td>X-axis position for the rotary operation.</td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>31.0352</td>
<td>Y-axis position for the rotary operation.</td>
</tr>
<tr>
<td></td>
<td>A</td>
<td>0.000</td>
<td>Rotary-axis angle for the rotary operation.</td>
</tr>
<tr>
<td>3</td>
<td>ROTARY MILL FRAME</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>X CORNER</td>
<td>-1.0000</td>
<td>X-axis coordinate for Reference Corner.</td>
</tr>
<tr>
<td></td>
<td>A CORNER</td>
<td>-90.000</td>
<td>Rotary-axis angle for Reference Corner.</td>
</tr>
<tr>
<td></td>
<td>X LENGTH</td>
<td>2.000</td>
<td>X coordinate length from Reference Corner.</td>
</tr>
<tr>
<td></td>
<td>A LENGTH</td>
<td>75.00</td>
<td>Rotary-axis length from Reference Corner.</td>
</tr>
<tr>
<td></td>
<td>CORNER RADIUS</td>
<td>0.2500</td>
<td>Radius of each of the corners.</td>
</tr>
</tbody>
</table>

*Table 5–1. Rotary Mill Frame Part Program Summary*

The figure below shows a helix created from a cylinder using Rotary Mill Contour. The threads shown in the drawing below are on the outside of the cylinder.

*Figure 5–12. Threading Part Drawing (cylinder shown for reference)*
This is a summary of the program used to create the part shown in the figure above:

<table>
<thead>
<tr>
<th>DB</th>
<th>Data Block/Field Name</th>
<th>Value</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>ROTARY PARAMETERS</td>
<td></td>
<td>Defines the cylinder.</td>
</tr>
<tr>
<td></td>
<td>CYLINDER RADIUS</td>
<td>2.0000</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>ROTARY POSITION</td>
<td></td>
<td>Moves the machine axes.</td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>0.000</td>
<td>X-axis position for the rotary operation.</td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>0.0000</td>
<td>Y-axis position for the rotary operation.</td>
</tr>
<tr>
<td></td>
<td>A</td>
<td>0.000</td>
<td>Rotary-axis angle for the rotary operation.</td>
</tr>
<tr>
<td>3</td>
<td>ROTARY MILL CONTOUR</td>
<td></td>
<td>Creates the rotary mill contour.</td>
</tr>
<tr>
<td></td>
<td>Segment 0 Start</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>X START</td>
<td>0.000</td>
<td>Initial X-axis location of the tool on the part.</td>
</tr>
<tr>
<td></td>
<td>A START</td>
<td>0.000</td>
<td>Initial rotary-axis location of tool on the part.</td>
</tr>
<tr>
<td>3.1</td>
<td>Segment 1 Line</td>
<td></td>
<td>The helix that creates the thread.</td>
</tr>
<tr>
<td></td>
<td>X END</td>
<td>-6.000</td>
<td>Final X-axis location at the end of the helix.</td>
</tr>
<tr>
<td></td>
<td>A END</td>
<td>1080.000</td>
<td>Rotary-axis location after the helix is cut. A END is number of degrees of rotation for rotary axis: (3 rotations x 360° =1080°).</td>
</tr>
</tbody>
</table>

*Table 5–2. Threading Part Program Summary*

The figure below shows a single hole drilled into a cylinder.
This is a summary of the program used to create the part shown in the figure above:

<table>
<thead>
<tr>
<th>No.</th>
<th>Description</th>
<th>Value</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>PART ZERO</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>A (arrow indicates positive axis movement)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>DB</th>
<th>Data Block/Field Name</th>
<th>Value</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>ROTARY PARAMETERS</td>
<td></td>
<td>Defines the cylinder.</td>
</tr>
<tr>
<td></td>
<td>CYLINDER RADIUS</td>
<td>2.0000</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>ROTARY POSITION</td>
<td></td>
<td>Moves the machine axes.</td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>-1.5000</td>
<td>X-axis position for the hole location on the cylinder. X is -1.5000 because the part must move in the negative direction from part zero.</td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>0.0000</td>
<td>Y-axis position for the hole location on the cylinder.</td>
</tr>
<tr>
<td></td>
<td>A</td>
<td>30.000</td>
<td>Rotary-axis angle for the hole location on the cylinder.</td>
</tr>
<tr>
<td>3</td>
<td>HOLES</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3.1</td>
<td>DRILL</td>
<td></td>
<td>Specifies the hole operations (same as for 3-axis milling, drilling, tap etc. operations).</td>
</tr>
</tbody>
</table>

Table 5–3. Hole Part Program Summary
The figure below shows a pattern of holes on a cylinder created with a Rotary Pattern Loop.

<table>
<thead>
<tr>
<th>No.</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>PART ZERO</td>
</tr>
<tr>
<td>2</td>
<td>A DISTANCE (arrow indicates positive axis movement)</td>
</tr>
<tr>
<td>3</td>
<td>X DISTANCE</td>
</tr>
</tbody>
</table>

*Figure 5–14. Rotary Pattern Loop Part Drawing (cylinder shown for reference)*
This is a summary of the program used to create the part shown in the figure above:

<table>
<thead>
<tr>
<th>DB</th>
<th>Data Block/Field Name</th>
<th>Value</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>ROTARY PARAMETERS</td>
<td></td>
<td>Defines the cylinder.</td>
</tr>
<tr>
<td></td>
<td>CYLINDER RADIUS</td>
<td>2.0000</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>ROTARY PATTERN LOOP</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>X DISTANCE</td>
<td>0.0000</td>
<td>Distance on the X-axis between each hole.</td>
</tr>
<tr>
<td></td>
<td>A DISTANCE</td>
<td>40.000</td>
<td>Angle on the rotary-axis between each hole.</td>
</tr>
<tr>
<td>3</td>
<td>ROTARY POSITION</td>
<td></td>
<td>Defines the position of machine axes.</td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>-1.5000</td>
<td>X-axis position during pattern execution.</td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>0.0000</td>
<td>Y-axis position during pattern execution.</td>
</tr>
<tr>
<td></td>
<td>A</td>
<td>-10.000</td>
<td>Rotary-axis position (angle) during pattern execution.</td>
</tr>
<tr>
<td>4</td>
<td>HOLES</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4.1</td>
<td>DRILL</td>
<td></td>
<td>Specifies hole operation. This is same data block used for 3-axis drilling, tap, etc.</td>
</tr>
<tr>
<td>4.2</td>
<td>LOCATIONS</td>
<td></td>
<td>Location of the initial hole on the cylinder.</td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>-1.5000</td>
<td>X-axis position of initial hole on the part.</td>
</tr>
<tr>
<td></td>
<td>A</td>
<td>0.0000</td>
<td>Rotary-axis position of initial hole on part.</td>
</tr>
<tr>
<td>5</td>
<td>PATTERN END</td>
<td></td>
<td>Ends rotary pattern loop. The machine’s axes will return to the position they were at before this program was executed.</td>
</tr>
</tbody>
</table>

*Table 5–4. Rotary Pattern Loop Part Program Summary*
The figure below shows a rectangular workpiece created on a WinMax Mill 4-axis machine using the Transform Plane feature. Standard mill contour and Rotary Position data blocks were used to remove the material from the cylinder to make the rectangular part.

![4-axis Part Created Using Transform Plane](image)

<table>
<thead>
<tr>
<th>No.</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>PART ZERO</td>
</tr>
<tr>
<td>2</td>
<td>Initial non-rotary contour in X-Y plane to create first side of part</td>
</tr>
<tr>
<td>3</td>
<td>Non-rotary contour repeated in first transformed plane to create second side of part</td>
</tr>
<tr>
<td>4</td>
<td>Non-rotary contour repeated in second transformed plane to create third side of part</td>
</tr>
<tr>
<td>5</td>
<td>Non-rotary contour repeated in third transformed plane to create fourth side of part</td>
</tr>
</tbody>
</table>

**Figure 5–15. 4-axis Part Created Using Transform Plane**

This is a summary of the program used to create the part shown in the figure above:

<table>
<thead>
<tr>
<th>DB</th>
<th>Data Block/Field Name</th>
<th>Value</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>ROTARY PARAMETERS</td>
<td></td>
<td>Defines the cylinder.</td>
</tr>
<tr>
<td></td>
<td>CYLINDER RADIUS</td>
<td>85.195</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>ROTARY POSITION</td>
<td></td>
<td>Moves the machine axes.</td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>0.000</td>
<td>X-axis position of rotary table.</td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>0.000</td>
<td>Y-axis position of rotary table.</td>
</tr>
<tr>
<td></td>
<td>A</td>
<td>0.000</td>
<td>Rotary-axis remains at machine zero.</td>
</tr>
<tr>
<td>3</td>
<td>MILL CONTOUR</td>
<td></td>
<td>Non-rotary mill contour subprogram to create the letter R on the X-Y plane of part.</td>
</tr>
<tr>
<td>DB</td>
<td>Data Block/Field Name</td>
<td>Value</td>
<td>Notes</td>
</tr>
<tr>
<td>----</td>
<td>-----------------------</td>
<td>-------</td>
<td>-------</td>
</tr>
<tr>
<td>4</td>
<td>ROTARY POSITION</td>
<td></td>
<td>Moves the machine axes</td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>0.000</td>
<td>X-axis position of rotary table for second side of part.</td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>0.000</td>
<td>Y-axis position of rotary table for second side of part.</td>
</tr>
<tr>
<td></td>
<td>A</td>
<td>270.000</td>
<td>Rotary-axis moves 270° from machine zero for second side of the part.</td>
</tr>
<tr>
<td></td>
<td>TRANSFORM PART ZERO</td>
<td>YES</td>
<td>Activates Transform Plane.</td>
</tr>
<tr>
<td>5</td>
<td>TRANSFORM PLANE REFERENCE POINTS</td>
<td></td>
<td>Positions part zero for the transformed plane for second side of the part.</td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>-38.100</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Z</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>MILL CONTOUR</td>
<td></td>
<td>Non-rotary mill frame with part zero at location defined in data block #5.</td>
</tr>
<tr>
<td>7</td>
<td>TRANSFORM PLANE END</td>
<td></td>
<td>Ends the Transform Plane.</td>
</tr>
<tr>
<td>8</td>
<td>ROTARY POSITION</td>
<td></td>
<td>Moves the machine axes.</td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>0.000</td>
<td>X-axis position of rotary table for third side of part.</td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>0.000</td>
<td>Y-axis position of rotary table for third side of part.</td>
</tr>
<tr>
<td></td>
<td>A</td>
<td>180.000</td>
<td>Rotary-axis moves 180° from machine zero for third side of part.</td>
</tr>
<tr>
<td>9</td>
<td>TRANSFORM PART ZERO</td>
<td>YES</td>
<td>Activates Transform Plane.</td>
</tr>
<tr>
<td>10</td>
<td>TRANSFORM PLANE REFERENCE POINTS</td>
<td></td>
<td>Positions part zero for the transformed plane for third side of part.</td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Z</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>MILL CONTOUR</td>
<td></td>
<td>Non-rotary mill frame block with part zero at location defined in data block #10.</td>
</tr>
<tr>
<td>12</td>
<td>TRANSFORM PLANE END</td>
<td></td>
<td>Ends the Transform Plane.</td>
</tr>
<tr>
<td>DB</td>
<td>Data Block/Field Name</td>
<td>Value</td>
<td>Notes</td>
</tr>
<tr>
<td>-----</td>
<td>----------------------------</td>
<td>--------</td>
<td>----------------------------------------------------------------------</td>
</tr>
<tr>
<td>13</td>
<td>ROTARY POSITION</td>
<td>0.000</td>
<td>Moves the machine axes.</td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>0.000</td>
<td>X-axis position of rotary table for final side of part.</td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>0.000</td>
<td>Y-axis position of rotary table for final side of part.</td>
</tr>
<tr>
<td></td>
<td>A</td>
<td>90.000</td>
<td>Rotary-axis position at 90° from machine zero for final side of part.</td>
</tr>
<tr>
<td></td>
<td>TRANSFORM PART ZERO</td>
<td>YES</td>
<td>Activates Transform Plane.</td>
</tr>
<tr>
<td>14</td>
<td>TRANSFORM PLANE</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>REFERENCE POINTS</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>X0</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Y 38.100</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Z 0.000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>PATTERN LOOP ROTATE</td>
<td></td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>MILL CONTOUR</td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>PATTERN END</td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>TRANSFORM PLANE END</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

*Table 5–5. 4-Axis Transform Plane Program Summary*
Tilt-Axis Programming Examples

The figure below shows a bolt hole drilled into a cylindrical part tilted 90°.

![Figure 5–16. Bolt Holes on Tilted Cylindrical Part](image)

This is a summary of the program used to create the part shown in the figure above:

<table>
<thead>
<tr>
<th>No.</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>PART ZERO, center of cylinder, and center of rotary table</td>
</tr>
<tr>
<td>2</td>
<td>Bolt hole</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>DB</th>
<th>Data Block/Field Name</th>
<th>Value</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>ROTARY PARAMETERS</td>
<td>CYLINDER RADIUS</td>
<td>2.0000</td>
</tr>
<tr>
<td>2</td>
<td>ROTARY POSITION</td>
<td>X</td>
<td>0.000</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Y</td>
<td>0.000</td>
</tr>
<tr>
<td></td>
<td></td>
<td>A</td>
<td>0.000</td>
</tr>
<tr>
<td></td>
<td></td>
<td>B</td>
<td>90.000</td>
</tr>
<tr>
<td>3</td>
<td>HOLES</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

*Table 5–6. Bolt Holes on Tilted Cylindrical Part Program Summary*
The figure below shows two holes drilled 45° apart on a cylindrical part.

![Cylindrical Part Tilted 45° Between Holes](image)

This is a summary of the program used to create the part shown in the figure above:

<table>
<thead>
<tr>
<th>No.</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>PART ZERO, center of cylindrical part and center of rotary table</td>
</tr>
<tr>
<td>2</td>
<td>Initial bolt hole</td>
</tr>
<tr>
<td>3</td>
<td>Bolt hole drilled after part is tilted 45°</td>
</tr>
</tbody>
</table>

**Figure 5–17. Cylindrical Part Tilted 45° Between Holes**

This is a summary of the program used to create the part shown in the figure above:

<table>
<thead>
<tr>
<th>DB</th>
<th>Data Block/Field Name</th>
<th>Value</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>ROTARY PARAMETERS</td>
<td></td>
<td>Defines the cylinder.</td>
</tr>
<tr>
<td></td>
<td>CYLINDER RADIUS</td>
<td>2.0000</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>ROTARY POSITION</td>
<td></td>
<td>Moves the machine axes.</td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>0.000</td>
<td>Initial X-axis position.</td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>0.000</td>
<td>Initial Y-axis position.</td>
</tr>
<tr>
<td></td>
<td>A</td>
<td>0.000</td>
<td>Initial rotary-axis position (angle).</td>
</tr>
<tr>
<td></td>
<td>B</td>
<td>0.000</td>
<td>Initial tilt-axis position (angle).</td>
</tr>
<tr>
<td>3</td>
<td>HOLES</td>
<td></td>
<td>Specifies hole operation. This is same data block used for 3-axis drilling, tap, etc.</td>
</tr>
<tr>
<td>4</td>
<td>ROTARY POSITION</td>
<td></td>
<td>Moves the machine axes.</td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>0.000</td>
<td>X-axis position prior to second hole operation.</td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>0.000</td>
<td>Y-axis position prior to second hole operation.</td>
</tr>
<tr>
<td></td>
<td>A</td>
<td>-45.000</td>
<td>Rotary-axis moves to this angle prior to second hole operation.</td>
</tr>
<tr>
<td>5</td>
<td>HOLES</td>
<td></td>
<td>Specifies hole operation.</td>
</tr>
</tbody>
</table>

**Table 5–7. Cylindrical Part Tilted 45° Between Holes Part Program Summary**
Transform Plane Example Programs

**Part Zero Does Not Change for Transformed Plane**

The figure below shows a milling feature created using a non-rotary Conversational mill frame and mill contour that is repeated in a transformed plane. In this example, part zero for the non-rotary milling feature does not change in the transformed plane. The part was created on a VMX42SR machine.

<table>
<thead>
<tr>
<th>Number</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>PART ZERO</td>
</tr>
<tr>
<td>2</td>
<td>Initial part created in X-Y plane</td>
</tr>
<tr>
<td>3</td>
<td>Part repeated in transformed plane with same Part Zero</td>
</tr>
</tbody>
</table>

*Figure 5–18. Same Part Zero for Transformed Plane, VMX42 SR Machine*
This is a summary of the program used to create the part in the figure above:

<table>
<thead>
<tr>
<th>DB</th>
<th>Data Block/Field Name</th>
<th>Value</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>ROTARY POSITION</td>
<td></td>
<td>Defines the position of all machine axes.</td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>0.0000</td>
<td>X-axis positioning of rotary-axis table.</td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>0.0000</td>
<td>Y-axis positioning of rotary-axis table.</td>
</tr>
<tr>
<td></td>
<td>B</td>
<td>0.0000</td>
<td>B-axis positioning at 0° from machine zero.</td>
</tr>
<tr>
<td></td>
<td>C</td>
<td>0.0000</td>
<td>C-axis positioning at 0° from machine zero.</td>
</tr>
<tr>
<td></td>
<td>TRANSFORM PART ZERO</td>
<td>No</td>
<td>Transform Plane not activated.</td>
</tr>
<tr>
<td>2</td>
<td>MILL CONTOUR</td>
<td></td>
<td>Non-rotaory mill contour subprogram to create an R on the X-Y plane of part.</td>
</tr>
<tr>
<td>3</td>
<td>ROTARY POSITION</td>
<td></td>
<td>Defines the position of all machine axes.</td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>0.0000</td>
<td>X-axis move prior to positioning rotary-axis table for Transform Plane operation.</td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>0.0000</td>
<td>Y-axis move prior to positioning rotary-axis table for Transform Plane operation.</td>
</tr>
<tr>
<td></td>
<td>B</td>
<td>-90.00</td>
<td>B-axis will move -90° from machine zero for transformed plane.</td>
</tr>
<tr>
<td></td>
<td>C</td>
<td>0.0000</td>
<td>C-axis will move 0° from machine zero for transformed plane.</td>
</tr>
<tr>
<td></td>
<td>TRANSFORM PART ZERO</td>
<td>YES</td>
<td>Activates Transform Plane.</td>
</tr>
<tr>
<td>4</td>
<td>TRANSFORM PLANE REFERENCE POINTS</td>
<td></td>
<td>Positions part zero for the transformed plane. For this program, part zero for the transformed plane is the same as initial part zero.</td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Z</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>MILL CONTOUR</td>
<td></td>
<td>Non-rotaory mill contour with part zero at location defined in data block #4.</td>
</tr>
<tr>
<td>6</td>
<td>TRANSFORM PLANE END</td>
<td></td>
<td>Ends the Transform Plane in data block #5.</td>
</tr>
</tbody>
</table>

*Table 5–8. Same Part Zero for Transformed Plane Part Program Summary, VMX42SR Machine*
Part Zero and Orientation Different for Transformed Planes

The figure below shows a non-rotary mill contour created on five sides of a part with different part zeroes and axis orientation for each of the transformed planes. The part was developed on a VMX42SR machine.

![Diagram of part creation](image)

<table>
<thead>
<tr>
<th>Number</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Initial PART ZERO</td>
</tr>
<tr>
<td>2</td>
<td>Initial non-rotary contour in X-Y plane to create first side of part</td>
</tr>
<tr>
<td>3</td>
<td>Non-rotary contour repeated in first transformed plane to create second side of part</td>
</tr>
<tr>
<td>4</td>
<td>Non-rotary contour repeated in second transformed plane to create third side of part</td>
</tr>
<tr>
<td>5</td>
<td>Non-rotary contour repeated in third transformed plane to create fourth side of part</td>
</tr>
<tr>
<td>6</td>
<td>Non-rotary contour repeated in fourth transformed plane to create fifth side of part</td>
</tr>
</tbody>
</table>

*Figure 5–19. Part Created on VMX42SR Machine, Using Transform Plane*
This is a summary of the program used to create the part shown in the figure above:

<table>
<thead>
<tr>
<th>DB</th>
<th>Data Block/Field Name</th>
<th>Value</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>ROTARY POSITION</td>
<td></td>
<td>Defines position of machine axes.</td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>B</td>
<td>0.0000</td>
<td>Tilt-axis will move 0° from machine zero.</td>
</tr>
<tr>
<td></td>
<td>C</td>
<td>0.0000</td>
<td>Rotary-axis will move 0° from machine zero.</td>
</tr>
<tr>
<td></td>
<td>TRANSFORM PART ZERO</td>
<td>NO</td>
<td>Transform Plane not activated.</td>
</tr>
<tr>
<td>2</td>
<td>MILL CONTOUR</td>
<td></td>
<td>Non-rotary mill contour to create the letter R on the X-Y plane of the part.</td>
</tr>
<tr>
<td>3</td>
<td>ROTARY POSITION</td>
<td></td>
<td>Redefines position of machine axes.</td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>0.0000</td>
<td>X-axis move prior to positioning rotary table for Transform Plane operation.</td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>0.0000</td>
<td>Y-axis move prior to positioning rotary table for Transform Plane operation.</td>
</tr>
<tr>
<td></td>
<td>B</td>
<td>-90.000</td>
<td>Tilt-axis will move -90° from machine zero for Transform Plane operation.</td>
</tr>
<tr>
<td></td>
<td>C</td>
<td>0.0000</td>
<td>Rotary-axis will move 0° from machine zero for Transform Plane operation.</td>
</tr>
<tr>
<td></td>
<td>TRANSFORM PART ZERO</td>
<td>YES</td>
<td>Activates Transform Plane.</td>
</tr>
<tr>
<td>4</td>
<td>TRANSFORM PLANE</td>
<td></td>
<td>Repositions part zero on transformed plane (i.e., second side of cube).</td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Z</td>
<td>-100.000</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>MILL CONTOUR</td>
<td></td>
<td>Non-rotary mill contour with part zero repositioned at location in data block #4.</td>
</tr>
<tr>
<td>6</td>
<td>TRANSFORM PLANE END</td>
<td></td>
<td>Ends Transform Plane.</td>
</tr>
<tr>
<td>DB</td>
<td>Data Block/Field Name</td>
<td>Value</td>
<td>Notes</td>
</tr>
<tr>
<td>----</td>
<td>----------------------------</td>
<td>---------</td>
<td>------------------------------------------------------------</td>
</tr>
<tr>
<td>7</td>
<td>ROTARY POSITION</td>
<td></td>
<td>Redefines position of machine axes.</td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>0.000</td>
<td>X-axis move prior to positioning rotary table for Transform Plane operation.</td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>0.000</td>
<td>Y-axis move prior to positioning rotary table for Transform Plane operation.</td>
</tr>
<tr>
<td></td>
<td>B</td>
<td>-90.000</td>
<td>Tilt-axis remains at -90° from machine zero for Transform Plane operation.</td>
</tr>
<tr>
<td></td>
<td>C</td>
<td>90.000</td>
<td>Rotary-axis moves 90° from machine zero for Transform Plane operation.</td>
</tr>
<tr>
<td></td>
<td>TRANSFORM PART ZERO</td>
<td>YES</td>
<td>Activates Transform Plane.</td>
</tr>
<tr>
<td>8</td>
<td>TRANSFORM PLANE</td>
<td></td>
<td>Repositions part zero for transformed plane (i.e., third side of cube).</td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>100.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Z</td>
<td>-100.000</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>MILL CONTOUR</td>
<td></td>
<td>Non-rotary mill contour with part zero repositioned at location in data block #8.</td>
</tr>
<tr>
<td>10</td>
<td>TRANSFORM PLANE END</td>
<td></td>
<td>Ends Transform Plane.</td>
</tr>
<tr>
<td>11</td>
<td>ROTARY POSITION</td>
<td></td>
<td>Redefines position of machine axes.</td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>0.0000</td>
<td>X-axis move prior to positioning rotary table for Transform Plane operation.</td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>0.0000</td>
<td>Y-axis move prior to positioning rotary table for Transform Plane operation.</td>
</tr>
<tr>
<td></td>
<td>B</td>
<td>-90.000</td>
<td>Tilt-axis will move -90° from machine zero for Transform Plane operation.</td>
</tr>
<tr>
<td></td>
<td>C</td>
<td>180.000</td>
<td>Rotary-axis will move 180° from machine zero for Transform Plane operation.</td>
</tr>
<tr>
<td></td>
<td>TRANSFORM PART ZERO</td>
<td>YES</td>
<td>Activates Transform Plane.</td>
</tr>
<tr>
<td>12</td>
<td>TRANSFORM PLANE</td>
<td></td>
<td>Repositions part zero on transformed plane (i.e., fourth side of cube).</td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>100.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>100.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Z</td>
<td>-100.000</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>MILL CONTOUR</td>
<td></td>
<td>Non-rotary mill contour with part zero repositioned at location in data block #12.</td>
</tr>
<tr>
<td>14</td>
<td>TRANSFORM PLANE END</td>
<td></td>
<td>Ends Transform Plane.</td>
</tr>
</tbody>
</table>
POST PROCESSOR (DESKTOP ONLY)

The Post Processor option allows Conversational part programs to be converted to NC (M and G code) programs, using third-party Post Processor software.

To convert a program, select the Post Processor icon on the Utilities screen:

![Figure 6–1. Utilities screen with Post Processor softkey](image)

The Post Processor Configuration screen appears:

![Figure 6–2. Post Processor Configuration screen](image)
The fields on the Post Processor Configuration screen are:

- **Output Arcs As**—specify if arcs should be output as arc interpolation commands or a series of lines.
- **Output Canned Cycles As**—specify if canned cycles should be output as canned cycle commands or as a series of lines.
- **Output Tool Path As**—specify if tool paths are output as centerline data (no G41 or G42) or as part surfaces with cutter compensation.
- **Display APT Fields in Editor**—specify Yes or No to display APT fields in the NC Editor.
- **APT Output Units**—specify the units for the APT data output: inches, millimeters, or Program Output (units selected in Program Properties).

Select the Post to NC softkey. The Post to file dialog opens:

Specify the location to save the file in the Post to File field. Click on the folder button to browse for the location. Choose the control type using the drop-down menu in the Control Type field. Click OK to post the file.
The PostHASTE copyright page appears:

Some post formats request information through dialog boxes; respond to prompts as they appear. A message that the file saved successfully appears when the process is complete.
PART PROBING OPTION

Part probing is used during Part Setup to probe the part’s edge, corner, bore, boss, rectangular pocket, or rectangular solid. The probe collects information about the part’s center position and diameter. This information is used to set Part Zero, making part setup faster and more accurate. Part probing can be used in Manual or Auto mode and with either Conversational or NC programming. Part probing can also be used for part quality verification.

It is the operator’s responsibility to set safe travel limits for the part probe as described in this chapter.

Manual Mode

Probing in Manual Mode is used to set part zero and/or determine skew:

2. Calibrate the probe and enter the deflection offsets. See Part Probe Calibration and Cycles, on page 7 - 5.

Auto Mode

Probing in Auto Mode is used to set part zero and/or determine skew within the part program (rather than in Part Setup as in Manual Mode). It can also be used to gather data about part quality.

2. Calibrate the probe and enter the deflection offsets. See Part Probe Calibration and Cycles, on page 7 - 5.
3. Probe the part to set part zero and/or determine skew. This is done within the program; see Automatic Mode, on page 7 - 42.

Additionally, part inspection is performed in Auto Mode. See Part Quality Verification, on page 7 - 46.
Part Setup—Part Probe Parameters

The part probing parameters are accessed through Part Setup. These parameters must be adjusted when a new stylus is installed in the part probe, when the probe work region is changed, or when probing feedrates are changed.

To access the Part Probe Parameters:

1. Select the Input console key to access the Input screen.
2. Press the Part Setup softkey.
3. Press the Part Probing softkey. The Part Setup screen is displayed with the Part Probing menu.
4. Select the Part Probe Parameters softkey.

![Figure 1. Probing Part Setup Softkey Menu](image)
The Part Setup Probe Parameters screen appears:

![Part setup screen with probe parameters](image)

*Figure 2. Part Setup Screen with Part Probe Parameters*

These fields are defined as follows:

- **Present**—indicates the presence of a part probe. Select the Yes or No softkey to indicate whether the probe is present. If not present, then the remaining part probe parameters are not used.

- **Stylus Diameter**—contains the stylus tip diameter, available on the specification sheet for the probing equipment.

- **Fast Start Feed**—specifies the feed rate to use for auto part probing start movement.

- **Approach Feed**—specifies the feed rate to use for the initial touch of the part. No measurement is taken at this feed rate, it is simply used to locate the part feature to be probed.

- **Measurement Feed**—specifies the feed rate to use when measuring the part feature.

- **Repetitions**—specifies the number of Slow Feed touches when measuring the part feature. The user may program up to 99 repetitions; the default value is 3.

- **X Min, X Max; Y Min, Y Max**—specifies the working envelope the probe tip uses to search for part features in X and Y machine coordinates. This envelope also helps protect against crashing the probe. These fields are similar to the safety work region in Part Setup.

- **Minimum Z**—specifies the working envelope the probe tip uses to search for part features in Z machine coordinates.
When the cursor is in any of these fields, the RESET PROBE WORK REGION TO MAXIMUM (F4) and STORE MACHINE POSITION (F7) softkeys become active.

- The RESET PROBE WORK REGION TO MAXIMUM (F4) softkey allows you to expand the working envelope for a new part.
- You can jog the axes and use the STORE MACHINE POSITION (F7) softkey to record the positions.

- Max Spread—the deviation (difference) between the minimum and maximum probe readings. If the deviation is greater than the Max Spread value, an error message appears.

- Circular Passes—the number of times to automatically repeat the probe cycle of a circular feature. After each pass the probe positions to the new center location and runs the cycle again. Multiple passes will provide better data because each pass starts closer to the true center. A poor starting location yields inaccuracies due to the diameter of the stylus and not being on line with the center. The default value is 2.

This information is specific to each installation and is stored on the hard drive. If the hard drive is replaced or formatted, the above information must be restored. Refer to the *Getting Started with WinMax Mill* manual for information about restoring parameters.

When the part probe is used, it must be activated by the control. For this to occur, the control needs to know when the probe is in the spindle. WinMax provides a Probe tool type in the Tool Setup screen. Before using the part probe, enter the part probe as a tool in Tool Setup:

1. Determine which tool number to assign to the probe.
2. Switch to the Tool Setup screen.
3. Enter the probe’s tool number in the Tool field.
4. Select Probe for the tool type.

The control activates the probe hardware when the number in the Tool In Spindle field matches the probe’s Tool number.

- Retract INIT—scales the initial retract move after a deflection. Increase the value if experiencing repeatability problems.

- Retract INCR—scales the incremental moves that may be required if the probe is still deflected after the initial move.
Part Probe Calibration and Cycles

This section describes the probe calibration and cycles available with the Probing Option. Once the probing equipment is calibrated, the tool and part can be calibrated using the probing equipment. This information can be stored as appropriate in Tool or Part Setup, or in a data block to be executed with the part program.

Probe calibration is only required on systems that also have a tool probe installed. Calibration methods vary between systems with tool probing only, versus systems with both tool and part probing. See *Probe Calibration—Absolute Tool Length mode*, on page 7 - 2 or *Probe Calibration—Zero Calibration mode*, on page 7 - 7 of *Tool Probing* for details.

Part Probe Deflection Offset Calibration

Part probe deflection offsets are the difference between the contact point of the probe and the actual receipt of a probe deflection signal. The offsets may vary for each direction of deflection.

These offsets need to be adjusted during an initial probe installation, a new stylus installation, or for centering or re-centering a stylus. They do not need to be performed each time the control is reset.

Access the Part Probe Deflection Offset screen from the Part Setup screen. Select the **Part Probing** softkey followed by the **Part Probe Deflection Offsets** softkey. This screen appears with softkeys for selecting the method to use for determining offsets.

![Figure 3. Part Probe Deflection Offset](image)

The sections that follow describe the procedures to follow for each method: *Ring Gauge*, on page 5 and *Reference Block*, on page 7.

**Ring Gauge**
The Ring Gauge method probes in a circular pattern. Select the PART PROBE DEFLECTION OFFSETS softkey followed by the RING GAUGE softkey, and this screen appears:

Follow these steps to determine a Ring Gauge Deflection Offset:

1. Enter the Diameter of the part.
2. Use an indicator or some similar method to determine the center of the gauge. Enter the Center of the gauge in the Center X and Center Y fields.
3. Jog the spindle to a point in Z where the probe just touches the part and enter the value in the Datum Z field. This value can be entered manually or by pressing the Store Position key.
4. Position the probe tip inside the gauge at the desired depth and select the Use Gauge To Get X&y Offsets softkey. The Start button begins to flash.
5. Press the Start button to begin the cycle.
6. The probe touches the part at 36 points (10° increments) inside the gauge to automatically calculate the X and Y offsets. To determine the Z offset, position the probe tip above the chosen datum point and select the Use Datum Point To Get Z Offset softkey. The Start button begins to flash.
7. Press the Start button to begin the cycle.
8. The probe touches off the datum point and calculates the offset.
9. The Offset values appear on the screen and are stored in memory.

This offset will be used anytime the control uses a probe location.

Offset values may be entered manually by the operator. If you know the readings are off by a certain amount, you can make adjustments without even using the probe.

The sign of the offset should be + for plus axis deflections and - for minus axis deflections.
The **Apply G Code Offset** parameter applies the deflection offsets to G31 commands when conversational and NC probing are used together.

### Reference Block

The Reference Block method probes in the + or - X or Y direction. Select the **Part Probe Deflection Offsets** softkey followed by the **Reference Block Method** softkey, and this screen appears:

#### Figure 5. Reference Block Deflection Offset

Follow these steps to determine the Reference Block Deflection Offsets for the X or Y axes:

1. Enter X or Y values manually in the Reference Block X or Y fields.
2. Position the cursor under the Deflection Offset column at the offset to be determined.
3. Manually jog the machine so the part probe is in the proper location to touch off the reference block for the desired axis and direction.
4. Select the **Use Probe To Determine Offset** softkey, and the **Start** button begins to flash.
5. Press the **Start** Button to start the cycle. The probe touches off the reference block. The offset values are calculated, appear on the screen, and are stored in memory.

This offset is used when the control uses a probe location.

Follow these steps to determine the offset for the Z axis:

1. Jog the probe to the top of the reference block.
2. Use a feeler gauge to determine where the tip of the stylus would touch the reference block. Enter this value in the -Z field in the Reference Block column.
3. Select the **Use Probe To Determine Offset** softkey. The probe touches off the part in the Z axis to determine the offset.

Offset values may be entered manually by the operator. If you know the readings are off by a certain amount, you can make adjustments without using the probe.

The sign of the offset should be + for plus axis deflections and - for minus axis deflections.

- The Ring Gauge method is more accurate than the Reference Block method.

- The probe measures the part at 36 points in 10° increments on the ring gauge.

- The 4 points measured on the reference block correspond to the 0°, 90°, 180°, and 270° values on the ring gauge. The remaining 32 values are estimated using the 4 actual measurements to fill in the 10° incremental offsets.

- The **Apply G Code Offset** parameter applies the deflection offsets to G31 commands when conversational and NC probing are used together.
Conversational Part Probing Cycles

Part probing is used for locating the position and alignment of the part on the table. Inserting a Probe Part Setup data block to run from the part program allows you to probe multiple parts and locate them at run time. This section describes probing cycles which are used for creating Probe Part Setup data blocks.

The software uses information programmed in the Part Setup screen to perform the Probing Cycles in Manual or Automatic mode.

There are two types of cycles available for probing different types of part features: Part Setup Probing Cycles and Part Skew Probing Cycles.

To select the Part Probe cycle type:

1. Select the Input console key to access the Input screen.
2. Press the Part Setup softkey.
3. Press the Part Probing softkey. The Part Setup screen is displayed with the Part Probing menu.

- Select the Part Zero Probe Cycles softkey for the Part Setup Probing Cycles. Refer to Part Setup Probing Cycles, on page 11 for details about programming these Manual Mode cycles. Refer to Automatic Mode, on page 42 for information about programming these Auto Mode cycles.
- Select the Part Skew Probe Cycles softkey to access the Part Skew Probing Cycles. Refer to Part Skew Probing Cycles, on page 34 for details about programming these Manual Mode cycles. Refer to Automatic Mode, on page 42 for information about programming these Auto Mode cycles.

Select the probing cycle type from the Part Setup screen with the Part Probing softkey. The cycles provide a method for allowing the software to automatically enter the Part Zero X, Part Zero Y, Probe Z, and X/Y Skew (deg) fields in the Part Setup screen.

Part Setup Screen

In addition to the standard Part Setup fields defined in the Getting Started with WinMax Mill manual, the software updates these Part Setup fields with data obtained during the probing cycles:

- **Part Zero X; Y**—store the part zero values established during the Part Setup Probing Cycles. Refer to Part Zero Storage, on page 33 for details about the Part Zero X and Y fields.
- **Probe Z**—represents the distance from Z zero to the top of the part (i.e., the height, or Z Plane), with the reference tool or part probe in the spindle.
  - Z zero is the Z calibration point of the Z axis (Z = 0.0000).
  - The software adjusts tool length values automatically and Tool Zero Calibration is recalculated anytime Probe Z is changed.

Refer to Determine Probe Z, on page 7 - 9 of Tool Probing for more details about these calculations.

- **X/Y Skew (deg.)**—represents, in degrees, how far the part is from perfect alignment.
alignment with the table. Refer to the “Part Skew Probing Cycles” section of this chapter for more information about X/Y skew.

Part probing may be run either from Manual Mode or from Auto Mode inside the part program. The sections that follow describe the different types of Manual Mode Probing Cycles. Auto Mode probing is described at the end of this chapter.

**Part Probe Deflection**

During the probing cycles described in this section, the part probe positions and moves at Approach Feed until it reaches the geometry. Then it backs up and moves again at Measurement Feed until it deflects the geometry a second time. The Measurement Feed touches are repeated a total of Repetitions times and the average is used.

**Part Probe Working Envelope**

The part probe cycles allow the part probe to operate within the constraints set in the Part Setup—Part Probe Parameters, on page 2. A working envelope containing safe part probe travel limits is stored in the Part Probing Parameters.

The working envelope represents the area on the machine table in which the probe can search for geometric features. The travel limits mentioned in each of the Manual Mode Part Setup Probing Cycles and the Manual Mode Part Skew Probing Cycles are set in the working envelope. This area is determined by these fields in the Part Probing Parameters screen: X Min, X Max, Y Min, Y Max, and Z Min. Z Min is a location above the table. The X and Y parameters are illustrated in the figure below:

\[ \begin{array}{c|c}
1 & \text{Y Max} \\
2 & \text{Y Min} \\
3 & \text{X Min} \\
4 & \text{X Max} \\
\end{array} \]

*Figure 6. Part Probe Working Envelope’s X and Y Parameters*

⇒ If the probe reaches any part probe travel limit before reaching the part feature, a fault occurs, motion stops, and an error message appears on the screen.
Each cycle’s *feed rate* is determined by the value set in the Part Probing Parameters Approach Feed and Measurement Feed fields.

**Part Setup Probing Cycles**

The table below describes the process for each Part Setup Probing Cycle:
<table>
<thead>
<tr>
<th>Cycle Type</th>
<th>Cycle Parameter Input</th>
<th>Automatic Execution</th>
<th>Results Displayed</th>
<th>Optional Storage</th>
</tr>
</thead>
<tbody>
<tr>
<td>Edge</td>
<td>Axis Probe Direction</td>
<td>Approach edge.</td>
<td>Deflection</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Positive or Negative</td>
<td>Retract. Return to</td>
<td>Position X, Y, or</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Preset X or Y</td>
<td>Start Position.</td>
<td>Z</td>
<td></td>
</tr>
<tr>
<td>Hole Circle</td>
<td>Start Angle 1, 2, 3</td>
<td>Approach 3 Circle</td>
<td>Center X</td>
<td>Center is Part</td>
</tr>
<tr>
<td>Pocket</td>
<td>Preset X</td>
<td>Points from inside.</td>
<td>Center Y</td>
<td>Zero X and</td>
</tr>
<tr>
<td></td>
<td>Preset Y</td>
<td>Position Probe</td>
<td>Diameter</td>
<td>Part Zero Y.</td>
</tr>
<tr>
<td>Cylinder</td>
<td>Probing Radius</td>
<td>Approach 3 Cylinder</td>
<td>Center X</td>
<td>Center is Part</td>
</tr>
<tr>
<td></td>
<td>Start Angle 1, 2, 3</td>
<td>Points from outside.</td>
<td>Center Y</td>
<td>Zero X and</td>
</tr>
<tr>
<td></td>
<td>Z Depth</td>
<td>Position Probe</td>
<td>Diameter</td>
<td>Part Zero Y.</td>
</tr>
<tr>
<td></td>
<td>Preset X</td>
<td>above cylinder</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Preset Y</td>
<td>Center.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rectangular</td>
<td>Preset X</td>
<td>Approach the 4</td>
<td>Center (X)</td>
<td>Center is Part</td>
</tr>
<tr>
<td>Pocket</td>
<td>Preset Y</td>
<td>pocket walls from</td>
<td>Center (Y)</td>
<td>Zero X and</td>
</tr>
<tr>
<td></td>
<td></td>
<td>inside.</td>
<td>Length (X)</td>
<td>Part Zero Y.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Position Probe</td>
<td>Length (Y)</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>into pocket Center.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rectangular</td>
<td>Probing Length (X)</td>
<td>Cycle Start</td>
<td>Center (X)</td>
<td>Center is Part</td>
</tr>
<tr>
<td>Solid</td>
<td>Probing Length (Y)</td>
<td>Approach the 4</td>
<td>Center (Y)</td>
<td>Zero X and</td>
</tr>
<tr>
<td></td>
<td>Z Depth</td>
<td>Rectangle walls</td>
<td>Length (X)</td>
<td>Part Zero Y.</td>
</tr>
<tr>
<td></td>
<td>Preset X</td>
<td>from outside.</td>
<td>Length (Y)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Preset Y</td>
<td>Position Probe</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>above rectangle</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Center.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Plane</td>
<td>X Probe Direction</td>
<td>Cycle Start</td>
<td>Intersection</td>
<td>Intersection</td>
</tr>
<tr>
<td>Intersection</td>
<td>Offset 1</td>
<td>Approach 2 points</td>
<td>point of the two</td>
<td>point is</td>
</tr>
<tr>
<td></td>
<td>Offset 2</td>
<td>on each of the two</td>
<td>planes in the X/</td>
<td>Part Zero X and</td>
</tr>
<tr>
<td></td>
<td>Y Probe Direction</td>
<td>planes. Return to</td>
<td>Y coordinate</td>
<td>Part Zero Y.</td>
</tr>
<tr>
<td></td>
<td>Offset 1</td>
<td>Start Position.</td>
<td>system</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Offset 2</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Preset X</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Preset Y</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

*Table 1. Part Setup Probing Cycles*
Manual Mode Part Setup Probing Cycles

During a Part Setup Probing cycle, the probe moves to specified points on the part, deflects, and stops in the center or at the Start Position, depending on the part feature. From the Part Setup screen, use the PROBING (F5) softkey to access the Probing Cycles.

Each cycle is described in detail in the following sections. When the cycle is finished, the software displays values representing the desired features. The fields for each cycle vary and are defined with each cycle description.

You can accept these values by pressing the ACCEPT POSITION AS PART ZERO softkey when it appears. If you have entered Preset X or Preset Y offsets, these offsets are subtracted from the probed Part Zero values, and the new Part Zero values appear after pressing the ACCEPT POSITION AS PART ZERO softkey.

Follow these steps to access the Part Probe Cycles:

1. From the Part Setup screen, select the **Part Probing** softkey.
2. Select the **Part Zero Probe Cycles** softkey. The probe cycle softkeys appear. Select a softkey to select a cycle.

Depending on the Probing Cycle selected, different probing fields appear on the Part Setup screen.

The following sections describe how to program each of the Part Setup Probing Cycles.
**Edge**

An Edge Cycle is used for determining the location of a specified edge of the part. During an Edge Cycle, the part probe moves to the X, Y, or Z edge of the part and records the deflection position.

The figure below shows part probe movement during X, Y, and Z Edge Cycles:

**Figure**
In addition to the standard Part Setup fields which are defined in the *Getting Started with WinMax Mill* manual, these Part Setup fields appear for the Edge Probing Cycle:

- **Part Zero X, Y; Probe Z**—contains the machine coordinate position of the Edge Cycle. These fields appear upon completion of the Edge Cycle.
- **Part Zero Cycle**—contains the type of cycle chosen from the Part Zero Probe Cycles softkey menu.
- **Probing Axis**—identifies the axis to move toward the edge of the part: X Axis, Y Axis, or Z Axis.
- **Probing Direction**—contains the direction to probe: Positive or Negative. This field appears when the Probing Axis is X or Y. It is not available for Probing Axis Z.
- **Preset X** or **Preset Y**—define an offset for Part Zero X and/or Part Zero Y. Entering offsets in either of these fields is optional.

The offset(s) will be subtracted from the center point of the circle and applied to Part Zero X and Y if you select the ACCEPT POSITION AS PART ZERO softkey, which appears after the cycle has been run. This field appears when the Probing Axis is X or Y. It is not available for Probing Axis Z.

Follow these steps to program an Edge Probing Cycle:

1. From the Part Zero Probe Cycles softkey menu, select Edge.
2. In the Probing Axis field, select the axis to move toward the edge of the part: X Axis, Y Axis, or Z Axis.
3. In the Probing Direction field, select the direction to probe: Positive or Negative. This field appears when the Probing Axis is X or Y. It is not available for Z Probing Axis.
4. If you want to program an offset from Part Zero X or Part Zero Y, enter the offset value in the Preset X or Preset Y field. This field appears when the Probing Axis is X or Y. It is not available for Z Probing Axis.

When the Part Setup fields have been entered, start the Edge Probing Cycle:

1. Place the part probe in the spindle and jog the probe to the Start Position, near the edge to be probed.
2. Press the **Start Probing Cycle** softkey. The **Start Cycle** button flashes.
3. Press the flashing **Start Cycle** button.
   a. The probing axis moves in the specified direction until the probe is deflected.
      - If no deflection occurs before the probe reaches the probe’s travel limit, the cycle is stopped and an error message appears.
      - To clear the error message and return to the Part Setup screen, press any key.
      - Check the Part Probe Parameters and the part fixturing. Make adjustments as necessary and re-start the cycle.
b. The deflection position appears in the Edge (X,Y, or Z) field when the cycle is finished.

c. The probing axis returns to the Start Position.

4. The **Accept Position As Part Zero** and **Do Not Accept** softkeys appear. Press the appropriate softkey.

   - The **Accept Position As Part Zero** softkey accepts the edge position and subtracts the Preset X or Y value to determine Part Zero.
   - The **Do Not Accept** softkey ignores the edge position value and the Preset X or Y value. Part Zero remains unchanged.

Refer to *Part Zero Storage*, on page 33 for more information.

5. The initial Part Setup screen appears with Part Zero entries established during the cycle.
**Hole or Circle Pocket**

The Hole or Circle Pocket Cycle is used for determining the center location and the diameter of a hole or pocket. During a Hole or Circle Pocket Cycle, the part probe moves from the inside of the circle out to three points on the edge, touches at each point, and returns to the Start Position within the circle after each touch. The software records each deflection position and calculates the center location. The probe positions in the center of the pocket.

The figure below shows part probe movement during a Hole or Circle Pocket Cycle:

![Diagram of Hole or Circle Pocket Probing Cycle](image)

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Start Position</td>
</tr>
<tr>
<td>2</td>
<td>Start Angle 1</td>
</tr>
<tr>
<td>3</td>
<td>Start Angle 2</td>
</tr>
<tr>
<td>4</td>
<td>Start Angle 3</td>
</tr>
<tr>
<td>5</td>
<td>Start Position</td>
</tr>
<tr>
<td>6</td>
<td>Start Angle 1</td>
</tr>
</tbody>
</table>

*Figure 8. Hole or Circle Pocket Probing Cycle*
In addition to the standard Part Setup fields which are defined in the *Getting Started with WinMax Mill* manual, these *Part Setup fields* appear for the Hole or Circle Pocket Probing Cycle:

- **Center X**—identifies the Center X machine coordinate location. This field appears upon completion of the cycle.
- **Center Y**—identifies the Center Y machine coordinate location. This field appears upon completion of the cycle.
- **Diameter**—identifies the diameter of the hole or circle pocket. This field appears upon completion of the cycle.
- **Part Probe Cycle**—contains the type of cycle chosen from the Part Zero Probe Cycles menu.
- **Start Angle 1; Start Angle 2; Start Angle 3**—define the three probe deflection points. Relative to the 3:00 position, and increasing counter-clockwise as viewed from above the part, the default angles are 0, 120, and 240, respectively. Change the angles if the part contains geometry that interferes with the defaults.
- **Preset X** and **Preset Y**—define an offset for Part Zero X and/or Part Zero Y. Entering offsets in either of these fields is optional.

The offset(s) will be subtracted from the center point of the circle and applied to Part Zero X and Y if you select the ACCEPT POSITION AS PART ZERO (F1) softkey, which appears after the cycle has been run.

Follow these steps to program a Hole or Circle Pocket Probing Cycle:

1. From the Part Zero Probe Cycles softkey menu, select Hole or Circle Pocket.
2. In the Start Angle fields, enter the desired approach angles.
3. If you want to program an offset from Part Zero X or Part Zero Y, enter the offset value in the Preset X or Preset Y field.

When the Part Setup fields have been entered, start the Hole or Circle Pocket Probing Cycle:

1. Place the part probe in the spindle and jog the probe to the Start Position, into the pocket and below the surface.
2. Press the START PROBING CYCLE (F1) softkey. The **Start Cycle** button flashes.
3. Press the **Start Cycle** button.
   a. The probe moves along Start Angle 1 until it is deflected at the edge of the hole or circle pocket.
      - If no deflection occurs before the probe reaches its travel limit, the cycle is stopped and an error message appears.
      - To clear the error message and return to the Part Setup screen, press any key.
      - Check the Part Probe Parameters and the part fixturing. Make adjustments as necessary and re-start the cycle.
b. After the first deflection, the probe returns to the Start Position.
c. The probe moves along Start Angle 2 to a second contact point.
d. After the deflection, the probe returns to the Start Position.
e. The probe moves along Start Angle 3 to a third contact point.
f. After the deflection, the probe returns to the Start Position.
g. Using the three contact points, the control calculates the diameter and the center (X and Y) of the hole.
h. The probe moves to the center of the hole or circle pocket. The results appear in the Center X, Center Y, and Diameter fields.

4. The ACCEPT POSITION AS PART ZERO (F1) and DO NOT ACCEPT (F2) softkeys appear. Press the appropriate softkey.
   - The ACCEPT POSITION AS PART ZERO (F1) softkey accepts the center position and subtracts the presets to determine Part Zero.
   - The DO NOT ACCEPT (F2) softkey ignores the center value and the Preset X or Y value. Part Zero remains unchanged.

Refer to Part Zero Storage, on page 33 for more information.

5. The initial Part Setup screen appears with Part Zero entries established during the cycle.
**Cylinder**

The Cylinder Cycle is used for determining the center location and the diameter of a cylinder. During a Cylinder Cycle, the part probe moves from the Start Position above the cylinder, out and down to three points around the diameter. The probe touches at each point and returns up and over to the Start Position above the cylinder after each touch. The software determines the diameter and the center location.

The figure below shows part probe movement during a Cylinder Cycle:

![Figure 9. Cylinder Probing Cycle](https://example.com/figure9.png)

<table>
<thead>
<tr>
<th></th>
<th>Start Position</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
</tr>
<tr>
<td>4</td>
<td>4</td>
</tr>
<tr>
<td>5</td>
<td>5</td>
</tr>
<tr>
<td>6</td>
<td>6</td>
</tr>
</tbody>
</table>

*Figure 9. Cylinder Probing Cycle*
In addition to the standard Part Setup fields which are defined in the *Getting Started with WinMax Mill* manual, these *Part Setup fields* appear for the Cylinder Cycle:

- **Center X**—identifies the Center X machine coordinate location. This field appears upon completion of the cycle.
- **Center Y**—identifies the Center Y machine coordinate location. This field appears upon completion of the cycle.
- **Diameter**—identifies the diameter of the cylinder. This field appears upon completion of the cycle.
- **Part Probe Cycle**—contains the type of cycle chosen from the Part Zero Probe Cycles softkey menu.
- **Probing Radius**—defines a value for the probe search radius. This value is used for determining the point at which the probe stops horizontal travel and begins to move downward.
- **Start Angle 1; Start Angle 2; Start Angle 3**—define the three probe deflection points. Relative to the 3:00 position and increasing counterclockwise as viewed from above the part, the default angles are 0, 120, and 240, respectively. Change the angles if the part contains geometry that interferes with the defaults.
- **Z Depth**—defines the distance, relative to the Start Position, the Z axis moves downward before changing direction and searching horizontally for each contact point on the cylinder’s diameter.

There should be no deflection during the Z move.

- **Preset X** and **Preset Y**—define an offset for Part Zero X and/or Part Zero Y. Entering offsets in either of these fields is optional.

The offset(s) will be subtracted from the center point of the circle and applied to Part Zero X and Y if you select the ACCEPT POSITION AS PART ZERO (F1) softkey, which appears after the cycle has been run.

Follow these steps to program a Cylinder Probing Cycle:

1. From the Part Zero Probe Cycles softkey menu, select Cylinder.
2. In the Probing Radius field, enter the probe search radius.
3. In the Start Angle fields, enter the desired approach angles.
4. In the Z Depth field, enter the distance the Z axis moves down before changing direction and searching horizontally for each contact point.
5. If you want to program an offset from Part Zero X and Part Zero Y, enter the offset value in the Preset X and Preset Y field.
When the Part Setup fields have been entered, start the Cylinder Probing Cycle:

1. Place the part probe in the spindle and jog the probe to the Start Position, above the cylinder.

2. Press the START PROBING CYCLE (F1) softkey. The **Start Cycle** button flashes.

3. Press the flashing **Start Cycle** button.
   a. The probe moves along Start Angle 1 until it reaches the Probing Radius.
      - If no deflection occurs before the probe reaches the probe's travel limit, the cycle is stopped and an error message appears.
      - To clear the error message and return to the Part Setup screen, press any key.
      - Check the Part Probe Parameters and the part fixturing. Make adjustments as necessary and re-start the cycle.
   b. The probe moves down until it reaches Z Depth.
      - If deflection occurs during the Z move, axis motion stops and an error message appears.
      - To clear the error message and return to the Part Setup screen, press any key.
      - Check the Part Probe Parameters and the part fixturing. Make adjustments as necessary and re-start the cycle.
   c. The probe moves backwards in the X/Y plane toward the Start Position in order to find a contact point.
   d. After deflection, the probe moves up and over to the Start Position above the cylinder.
   e. The probe moves along Start Angle 2 until it reaches the Probing Radius. The same motion occurs as with Start Angle 1.
   f. After the deflection, the probe moves up and over to the Start Position above the cylinder.
   g. The probe moves along Start Angle 3 until it reaches the Probing Radius. The same motion occurs as with Start Angles 1 and 2.
   h. After the third deflection, the control uses the three contact points and calculates the diameter and center (X and Y) of the cylinder.
   i. The probe moves up and over to the center above the Z Plane of the cylinder. The results appear in the Center X, Center Y, and Diameter fields.

4. The ACCEPT POSITION AS PART ZERO (F1) and DO NOT ACCEPT (F2) softkeys appear. Press the appropriate softkey.
   - The ACCEPT POSITION AS PART ZERO (F1) softkey accepts the center and subtracts the presets to determine part zero.
   - The DO NOT ACCEPT (F2) softkey ignores the center values and the Preset X or Y value. Part Zero remains unchanged.

Refer to **Part Zero Storage**, on page 33 for more information.

5. The initial Part Setup screen appears with Part Zero entries established during
the cycle.

**Rectangular Pocket Inside**

The Rectangular Pocket Inside (or Rectangular Pocket) Cycle is used for determining the center location of the pocket and the X and Y length of the rectangle. During a Rectangular Pocket Cycle, the part probe moves from inside the pocket out to a point on each edge of the rectangle, touches at each point, and returns to the Start Position after each touch. The software records each deflection position and calculates the center location and lengths.

The figure below shows part probe movement during a Rectangular Pocket Cycle.

![Rectangular Pocket Probing Cycle](image)

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Start Position</td>
</tr>
<tr>
<td>2</td>
<td>Point 1</td>
</tr>
<tr>
<td>3</td>
<td>Point 2</td>
</tr>
<tr>
<td>4</td>
<td>Point 3</td>
</tr>
<tr>
<td>5</td>
<td>Point 4</td>
</tr>
<tr>
<td>6</td>
<td>Start Position</td>
</tr>
<tr>
<td>7</td>
<td>Point 1</td>
</tr>
</tbody>
</table>

*Figure 10. Rectangular Pocket Probing Cycle*
In addition to the standard Part Setup fields which are defined in the Programming Basics chapter of the Getting Started with WinMax manual, these Part Setup fields appear for the Rectangular Pocket Cycle:

- **Center X**—identifies the Center X machine coordinate location. This field appears upon completion of the cycle.
- **Center Y**—identifies the Center Y machine coordinate location. This field appears upon completion of the cycle.
- **Length (X)**—identifies the X length of the pocket. This field appears upon completion of the cycle.
- **Length (Y)**—identifies the Y length of the pocket. This field appears upon completion of the cycle.
- **Part Probe Cycle**—contains the type of cycle chosen from the Part Zero Probe Cycles softkey menu.
- **Preset X** or **Preset Y**—define an offset for Part Zero X and/or Part Zero Y. Entering offsets in either of these fields is optional.

The offset(s) will be subtracted from the center point of the pocket and applied to Part Zero X and Y if you select the ACCEPT POSITION AS PART ZERO (F1) softkey, which appears after the cycle has been run.

Follow these steps to program a Rectangular Pocket Cycle:

1. From the Part Zero Probe Cycles softkey menu, select Rectangular Pocket Inside.
2. If you want to program an offset for Part Zero X or Part Zero Y, enter the offset value in the Preset X or Preset Y field.

When the Part Setup fields have been entered, start the Rectangular Pocket Cycle:

1. Place the part probe in the spindle and jog the probe to the Start Position, into the pocket and below the surface.
2. Press the START PROBING CYCLE (F1) softkey. The **Start Cycle** button flashes.
3. Press the **Start Cycle** button.
   a. The probe moves in the positive X direction until it is deflected at the edge of the pocket.
      - If no deflection occurs before the probe reaches its horizontal travel limit, the cycle is stopped and an error message appears.
      - To clear the error message and return to the Part Setup screen, press any key.
      - Check the Part Probe Parameters and the part fixturing. Make adjustments as necessary and re-start the cycle.
   b. After the first deflection, the probe returns to the Start Position.
   c. The probe moves in the negative X direction, reaches a second contact point, and returns to the Start Position.
d. The probe moves in the positive Y direction, reaches a third contact point, and returns to the Start Position.

e. The probe moves in the negative Y direction, reaches a fourth contact point, and returns to the Start Position.

f. Using the four contact points, the control calculates the length of the pocket in the X and Y directions and the center (X and Y) of the pocket.

g. The probe moves to the center of the rectangle. The results are displayed on the screen in the Center X, Center Y, Length (X), and Length (Y) fields.

4. The ACCEPT POSITION AS PART ZERO (F1) and DO NOT ACCEPT (F2) softkeys appear. Press the appropriate softkey.

   • The ACCEPT POSITION AS PART ZERO (F1) softkey accepts the center and subtracts the presets to determine part zero.
   
   • The DO NOT ACCEPT (F2) softkey ignores the center and length values and the Preset X or Y value. Part Zero remains unchanged.

Refer to Part Zero Storage, on page 33 for more information.

5. The initial Part Setup screen appears with Part Zero entries established during the cycle.
Rectangular Solid Outside

A Rectangular Solid Outside (or Rectangular Solid) Cycle is used for determining the center location of the pocket and the X and Y length of the rectangle. During a Rectangular Solid Cycle, the part probe moves from above the rectangle out and down to a point on each of the four walls, touches at each point, and returns to the Start Position after each touch. The software records each deflection position and calculates the center position and lengths.

The figure below shows part probe movement during a Rectangular Solid Cycle:

![Rectangular Solid Probing Cycle](image)

<table>
<thead>
<tr>
<th></th>
<th>Start Position</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>Point 1</td>
</tr>
<tr>
<td>3</td>
<td>Point 2</td>
</tr>
<tr>
<td>4</td>
<td>Point 3</td>
</tr>
<tr>
<td>5</td>
<td>Point 4</td>
</tr>
<tr>
<td>6</td>
<td>Start Position</td>
</tr>
<tr>
<td>7</td>
<td>Point 1</td>
</tr>
</tbody>
</table>

*Figure 11. Rectangular Solid Probing Cycle*
In addition to the standard Part Setup fields which are defined in the Programming Basics chapter of the Getting Started with WinMax manual, these Part Setup fields appear for the Rectangular Solid Cycle:

- **Center X**—identifies the Center X machine coordinate location. This field appears upon completion of the cycle.
- **Center Y**—identifies the Center Y machine coordinate location. This field appears upon completion of the cycle.
- **Length (X)**—identifies the X length of the rectangle. This field appears upon completion of the cycle.
- **Length (Y)**—identifies the Y length of the rectangle. This field appears upon completion of the cycle.
- **Part Probe Cycle**—contains the type of cycle chosen from the Part Zero Probe Cycles softkey menu.
- **Probing Length X**—identifies a maximum value of the estimated X length. Half of this value is used to determine the point at which the Z axis begins to move downward; i.e., its horizontal travel limit for X.
- **Probing Length Y**—identifies a maximum value of the estimated Y length. Half of this value is used to determine the point at which the Z axis begins to move downward; i.e., its horizontal travel limit for Y.
- **Z Depth**—defines the distance, relative to the Start Position, the Z axis moves downward before changing direction and searching horizontally for each contact point on the cylinder’s diameter.

> There should be no deflection during the Z move.

- **Preset X** and **Preset Y**—define an offset for Part Zero X and/or Part Zero Y. Entering offsets in either of these fields is optional.

The offset(s) will be subtracted from the center point of the rectangle and applied to Part Zero X and Y if you select the ACCEPT POSITION AS PART ZERO (F1) softkey, which appears after the cycle has been run.

Follow these steps to program a Rectangular Solid Cycle:

1. From the Part Setup Probe Cycles softkey menu, select Rectangular Solid Outside.
2. In the Probing Length X field, enter an estimate for the X length.
3. In the Probing Length Y field, enter an estimate for the Y length.
4. In the Z Depth field, enter the distance the Z axis should move down.
5. If you want to program an offset for Part Zero X and Part Zero Y, enter the offset value in the Preset X and Preset Y field.
When the Part Setup fields have been entered, start the Rectangular Solid Cycle:

1. Place the part probe in the spindle and jog the probe to the Start Position, above the middle of the rectangle.

2. Press the START PROBING CYCLE (F1) softkey. The **Start Cycle** button flashes.

3. Press the **Start Cycle** button.
   a. The probe moves in the positive X direction, until it reaches its horizontal travel limit determined by the Probing Length X field.
      - If no deflection occurs before the probe reaches the probe's travel limit, the cycle is stopped and an error message appears.
      - To clear the error message and return to the Part Setup screen, press any key.
      - Check the Part Probe Parameters and the part fixturing. Make adjustments as necessary and re-start the cycle.
   b. The probe moves down until it reaches Z Depth.
      - If deflection occurs during the Z move, axis motion stops and an error message appears.
      - To clear the error message and return to the Part Setup screen, press any key.
      - Check the Part Probe Parameters and the part fixturing. Make adjustments as necessary and re-start the cycle.
   c. The probe moves backwards in the X/Y plane toward the Start Position and deflects.
   d. The probe returns up and over to the Start Position.
   e. The probe moves in the negative X direction until it reaches its horizontal travel limit determined by the Probing Length X field.
   f. The probe moves down until it reaches Z Depth.
   g. The probe moves backwards in the X/Y plane toward the Start Position and deflects.
   h. The probe moves up and over to the Start Position.
   i. The probe moves in the positive Y direction until it reaches its horizontal travel limit determined by the Probing Length Y field.
   j. The probe moves down until it reaches Z Depth.
   k. The probe moves backwards in the X/Y plane toward the Start Position and deflects.
   l. The probe moves up and over to the Start Position.
   m. The probe moves in the negative Y direction until it reaches its horizontal travel limit determined by the Probing Length Y field.
   n. The probe moves down until it reaches Z Depth.
   o. The probe moves backwards in the X/Y plane toward the Start Position and deflects.
   p. The probe moves up and over to the Start Position.
   q. After the last deflection, the control calculates the X and Y lengths of the solid and the center (X and Y) of the rectangle.
r. The results appear in the Center X, Center Y, Length (X), and Length (Y), fields. The probe moves to the center above the rectangle.

4. The ACCEPT POSITION AS PART ZERO (F1) and DO NOT ACCEPT (F2) softkeys appear. Press the appropriate softkey.
   - The ACCEPT POSITION AS PART ZERO (F1) softkey accepts the center and subtracts the presets to determine part zero.
   - The DO NOT ACCEPT (F2) softkey ignores the center and length values and the Preset X or Y value. Part Zero remains unchanged.

Refer to Part Zero Storage, on page 33 for more information.

5. The initial Part Setup screen appears with Part Zero entries established during the cycle.
**Plane Intersection (Non-Rectangular Corner)**

A Plane Intersection Cycle is used for determining an X and Y intersection for a non-rectangular corner. During a Plane Intersection Cycle, the part probe moves from an offset position to two points in the X direction and two points in the Y direction to determine an X and Y intersection point.

The Plane Intersection cycle can be used with solid or pocket geometry. The figure below shows part probe movement with the two types of geometry:

---

**Table**

<table>
<thead>
<tr>
<th></th>
<th>Solid Geometry Start Position</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>Solid Geometry Point 1</td>
</tr>
<tr>
<td>3</td>
<td>Solid Geometry Point 2</td>
</tr>
<tr>
<td>4</td>
<td>Solid Geometry Point 3</td>
</tr>
<tr>
<td>5</td>
<td>Solid Geometry Point 4</td>
</tr>
<tr>
<td>6</td>
<td>Pocket Geometry Start Position</td>
</tr>
<tr>
<td>7</td>
<td>Pocket Geometry Point 1</td>
</tr>
<tr>
<td>8</td>
<td>Pocket Geometry Point 2</td>
</tr>
<tr>
<td>9</td>
<td>Pocket Geometry Point 3</td>
</tr>
<tr>
<td>10</td>
<td>Pocket Geometry Point 4</td>
</tr>
</tbody>
</table>

*Figure 12. Plane Intersection Probing Cycle*
In addition to the standard Part Setup fields which are defined in the *Programming Basics* chapter of the *Getting Started with WinMax* manual, these Part Setup fields appear for the Plane Intersection Cycle:

- **Corner X**—defines the X position of the intersection. This field appears when the cycle is finished.
- **Corner Y**—defines the Y position of the intersection. This field appears when the cycle is finished.
- **Probing Direction X**—select the X direction to probe: Positive or Negative.
- **Offset 1**—defines the first Y offset position.
- **Offset 2**—defines the second Y offset position.
- **Probing Direction Y**—select the Y direction to probe: Positive or Negative.
- **Offset 1**—defines the first X offset position.
- **Offset 2**—defines the second X offset position.
- **Preset X** and **Preset Y**—define an offset for Part Zero X and/or Part Zero Y. Entering offsets in either of these fields is optional.

The offset(s) will be subtracted from the corner point and applied to Part Zero X and Y if you select the ACCEPT POSITION AS PART ZERO (F1) softkey, which appears after the cycle has been run.

Follow these steps to program a Plane Intersection Cycle:

1. From the Part Zero Probe Cycles softkey menu, select Plane Intersection.
2. In the Probing Direction X field, select Positive or Negative.
3. In the Offset 1 field, enter the position for the first Y Offset, relative to the Start Position.
4. In the Offset 2 field, enter the position for the second Y Offset, relative to the Start Position.
5. In the Probing Direction Y field, select Positive or Negative.
6. In the Offset 1 field, enter the position for the first X Offset, relative to the Start Position.
7. In the Offset 2 field, enter the position for the second X Offset, relative to the Start Position.
8. If you want to program an offset from Part Zero X or Part Zero Y, enter the offset value in the Preset X or Preset Y field.

When the Part Setup fields have been entered, start the Plane Intersection Cycle:

1. Place the part probe in the spindle and jog the probe into the Start Position, near the non-rectangular corner.
2. Press the START PROBING CYCLE (F1) softkey. The **Start Cycle** button flashes.
3. Press the flashing **Start Cycle** button.
   a. The probe moves the direction specified in Probing Direction X the distance specified in Offset 1.
      - If no deflection occurs before the probe reaches the probe’s travel limit, the cycle is stopped and an error message appears.
      - To clear the error message and return to the Part Setup screen, press any key.
      - Check the Part Probe Parameters and the part fixturing. Make adjustments as necessary and re-start the cycle.
   b. The probe reaches the first contact point with the first edge in the X direction, deflects, and moves to the position specified in Offset 2.
   c. The probe reaches the second contact point in the X direction and deflects.
   d. The probe returns to the starting point.
   e. The probe moves the direction specified in Probing Direction Y the distance specified in Offset 1.
   f. The probe reaches the first contact point with the first edge in the Y direction, deflects, and moves to the position specified in Offset 2.
   g. The probe reaches the second contact point in the Y direction and deflects.
   h. The probe returns to the Start Position.
   i. Using the four contact points, the control calculates the X and Y intersection points. The results appear in the Corner X and Corner Y fields.

4. The ACCEPT POSITION AS PART ZERO (F1) and DO NOT ACCEPT (F2) softkeys appear. Press the appropriate softkey.
   - The ACCEPT POSITION AS PART ZERO (F1) softkey accepts the corner position and subtracts the Preset X or Y value to determine Part Zero.
   - The DO NOT ACCEPT (F2) softkey ignores the corner values and the Preset X or Y value. Part Zero remains unchanged.

Refer to *Part Zero Storage*, on page 33 for more information.

5. The initial Part Setup screen appears with Part Zero entries established during the cycle.
**Part Zero Storage**

At the end of each Probing Cycle, the results of the cycle are displayed on the screen. These results are displayed in machine coordinates and do not include the Preset values.

<table>
<thead>
<tr>
<th>Cycle</th>
<th>Results</th>
</tr>
</thead>
<tbody>
<tr>
<td>Edge</td>
<td>Contact Position (X, Y, or Z)</td>
</tr>
<tr>
<td>Hole or Circle Pocket</td>
<td>Diameter, Center (X and Y)</td>
</tr>
<tr>
<td>Cylinder</td>
<td>Diameter, Center (X and Y)</td>
</tr>
<tr>
<td>Rectangular Pocket</td>
<td>Lengths (X and Y), Center (X and Y)</td>
</tr>
<tr>
<td>Rectangular Solid</td>
<td>Lengths (X and Y), Center (X and Y)</td>
</tr>
<tr>
<td>Plane Intersect</td>
<td>Corner (X and Y)</td>
</tr>
</tbody>
</table>

Selecting the ACCEPT POSITION AS PART ZERO (F1) softkey accepts the probed values and subtracts the presets to determine part zero. The new Part Zero values appear in the Part Zero X and Part Zero Y fields.

With an Edge probing cycle the probe moves only one axis. Therefore, only one of Part Zero X, Part Zero Y, or Probe Z is set. With all other cycles, both Part Zero X and Part Zero Y are determined at the same time.
### Part Skew Probing Cycles

The following table describes the process of the Skew Probing Cycles:

<table>
<thead>
<tr>
<th>Skew Cycle Type</th>
<th>Cycle Parameter Input</th>
<th>Automatic Execution</th>
<th>Display of the results</th>
<th>Optional storage</th>
</tr>
</thead>
<tbody>
<tr>
<td>Edge</td>
<td>Axis X, Y, or Z</td>
<td><strong>Cycle Start</strong></td>
<td>Deflection Position X, Y, or Z Skew Angle (deg)</td>
<td>Skew Angle</td>
</tr>
<tr>
<td></td>
<td>Probe Direction:</td>
<td>Approach edge.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Positive or Negative</td>
<td>Retract.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Preset X or Preset Y</td>
<td>Return to Start</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Position.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Hole or Circle</td>
<td>Start Angle 1, 2, 3</td>
<td><strong>Cycle Start</strong></td>
<td>Center X Center Y</td>
<td>Skew Angle</td>
</tr>
<tr>
<td>Pocket</td>
<td>Preset X Preset Y</td>
<td>Approach 3 Circle</td>
<td>Diameter Skew Angle (deg)</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Points from inside.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Position Probe into</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Center.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cylinder</td>
<td>Probing Radius</td>
<td><strong>Cycle Start</strong></td>
<td>Center X Center Y</td>
<td>Skew Angle</td>
</tr>
<tr>
<td></td>
<td>Z Depth</td>
<td>Approach 3 Cylinder</td>
<td>Diameter Skew Angle (deg)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Start Angle 1, 2, 3</td>
<td>Points from outside.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Preset X Preset Y</td>
<td>Position Probe above cylinder Center.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rectangular</td>
<td>Preset X Preset Y</td>
<td><strong>Cycle Start</strong></td>
<td>Center X Center Y</td>
<td>Skew Angle</td>
</tr>
<tr>
<td>Pocket</td>
<td></td>
<td>Approach the 4</td>
<td>Diameter Skew Angle (deg)</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>pocket walls from</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>inside.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Position Probe into</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>pocket Center.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rectangular</td>
<td>Probing Length X</td>
<td><strong>Cycle Start</strong></td>
<td>Center X Center Y</td>
<td>Skew Angle</td>
</tr>
<tr>
<td>Solid</td>
<td>Probing Length Y</td>
<td>Approach the 4</td>
<td>Diameter Skew Angle (deg)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Z Depth</td>
<td>Rectangle walls from</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Preset X Preset Y</td>
<td>outside.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Position Probe above</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>rectangle Center.</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

*Table 2. Part Skew Probing Cycles*
Manual Mode Part Skew Probing Cycles

From the Part Setup screen, use the PROBING (F5) softkey to access the Probing Cycles. During a Part Skew Probing cycle the probe moves to specified points on the part, deflects, and stops in the center of the part feature. These cycles detect and compensate for X/Y skew in the workpiece. X/Y skew represents, in degrees, how far the part is from perfect alignment with the table.

- A positive skew angle means the part is rotated in a counterclockwise direction from the machine axes (as viewed from above the part).
- A negative skew angle indicates a clockwise rotation from the machine axes.

Although this angle, if known, may be manually typed in using the keypad, it is easier and more accurate to let the probe find the skew angle and automatically enter it.

For best results, it is recommended that you use the probe for determining Part Zero X, Part Zero Y, and X/Y Skew. Entering values for any of these fields with the keypad reduces the amount of information available for skew calculations. Also, the software must make assumptions that may reduce accuracy.

After probing the Part Zero position as a reference, the skew cycles allow you to probe a second feature on the part and adjust all machining operations by the skew angle to exactly match the part.

When the cycle is finished, the software displays values representing the desired features. The fields for each cycle vary and are defined with each cycle description.

You can accept these values by pressing the Accept X/Y Skew Angle softkey when it appears. Preset X or Preset Y offsets are used in the skew calculation. The presets are subtracted from the probed Part Zero values, and the new Part Zero values appear after pressing the ACCEPT POSITION AS PART ZERO (F1) softkey.
The figure below illustrates a skewed workpiece with a Preset Y offset.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Part Zero (corner, first reference)</td>
</tr>
<tr>
<td>2</td>
<td>Hole Centerpoint, used as second reference for skew compensation</td>
</tr>
<tr>
<td>3</td>
<td>Preset Y</td>
</tr>
</tbody>
</table>

**Figure 13. Example of a Skewed Workpiece**

Enter offsets for Preset X and Preset Y for a precise Skew Angle. If only one Preset value is entered, the skew angle will be approximate and should not exceed 3 degrees.

Skew Compensation is intended to be used for correcting a slight misalignment. If the skew angle contains only one Preset value and is greater than approximately 3 degrees, then Part Skew probing may not be exact, especially for Edge and Rectangular cycles.
Follow these steps to perform a Skew Probe Cycle for workpiece skew compensation:

1. Perform the Part Zero Probe Cycle. Refer to the appropriate Part Probe Cycle in this chapter (i.e., Edge, Cylinder, etc.) or more information.
2. From the Part Setup screen, select the PROBING softkey followed by the PART SKEW PROBE CYCLES softkey to access the Part Skew Probe Cycles.
3. The Skew Probe Cycle Type softkeys appear. Select the appropriate softkey for the desired cycle.

Depending on the Skew Probing Cycle selected, different probing fields appear on the Part Setup screen.

- The following sections describe how to program each of the Part Skew Probing Cycles.

**Edge Skew**

Follow these steps to program an Edge Skew Probing Cycle:

1. From the Part Setup Skew Probe Cycle softkey menu, select Edge.
2. In the Probing Axis field, select the axis to move toward the edge of the part: X Axis, Y Axis, or Z Axis.
3. In the Probing Direction field, select the direction to probe: Positive or Negative. Z Axis always probes in the Negative direction.
4. Program an offset in the Preset X or Preset Y field, if desired.

When the Part Setup fields have been entered, start the Edge Skew Probing Cycle:

1. Place the part probe in the spindle and jog the probe to the Start Position, near the edge to be probed.
2. Press the START PROBING CYCLE (F1) softkey. The Start Cycle button flashes.
3. Press the Start Cycle button.

   The tool motion for an Edge Skew Cycle is the same as the motion described for an Edge Cycle. The results appear in the Edge (X,Y, or Z) and Skew Angle (Deg) fields. The probe returns to the Start Position.

4. The ACCEPT X/Y SKEW ANGLE (F1) and DO NOT ACCEPT (F2) softkeys appear.
   - The ACCEPT X/Y SKEW ANGLE (F1) softkey accepts the skew position and subtracts the Preset X or Y value to determine Part Zero.
   - The DO NOT ACCEPT (F2) softkey ignores the edge position value and the Preset X or Y value. Part Zero remains unchanged.

   Refer to Part Zero Storage, on page 33 for more information.

5. The initial Part Setup screen appears with the skew value established during the cycle stored in the X/Y Skew (deg) field, if accepted.
**Hole or Circle Pocket Skew**

Follow these steps to program a Hole or Circle Pocket Skew Probing Cycle:

1. From the Part Setup Skew Probe Cycle softkey menu, select the HOLE OR CIRCLE POCKET (F2) softkey.
2. In the Start Angle 1, Start Angle 2, and Start Angle 3 fields, enter the desired approach angles.
3. Program an offset in the Preset X and/or Preset Y field(s), if desired.

When the Part Setup fields have been entered, start the Hole or Circle Skew Probing Cycle:

1. Place the part probe in the spindle and jog the probe to the Start Position, inside the pocket near the center.
2. Press the START PROBING CYCLE (F1) softkey. The Start Cycle button flashes.
3. Press the Start Cycle button.

The tool motion for a Hole or Circle Pocket Skew Probing Cycle is the same as the motion described for a Hole or Circle Pocket Cycle. The results appear in the Center X, Center Y, Diameter, and Skew Angle (deg) fields. The probe stops in the center of the pocket.

4. The ACCEPT X/Y SKEW ANGLE (F1) and DO NOT ACCEPT (F2) softkeys appear.
   - The ACCEPT X/Y SKEW ANGLE (F1) softkey accepts the skew position and subtracts the Preset X or Y value to determine Part Zero.
   - The DO NOT ACCEPT (F2) softkey ignores the edge position value and the Preset X or Y value. Part Zero remains unchanged.

Refer to *Part Zero Storage*, on page 33 for more information.

5. The initial Part Setup screen appears with the skew value established during the cycle stored in the X/Y Skew (deg) field, if accepted.
**Cylinder Skew**

Follow these steps to program a Cylinder Skew Probing Cycle:

1. From the Part Setup Skew Probe Cycle softkey menu, select Cylinder.
2. In the Probing Radius field, enter the probe search radius.
3. In the Start Angle fields, enter the desired approach angles.
4. In the Z Depth field, enter the distance the Z axis moves down before changing direction and searching horizontally for each contact point.
5. Program an offset in the Preset X and/or Preset Y field(s), if desired.

When the Part Setup fields have been entered, start the Cylinder Skew Probing Cycle:

1. Place the part probe in the spindle and jog the probe to the Start Position, above the cylinder near the center.
2. Press the START PROBING CYCLE (F1) softkey. The **Start Cycle** button flashes.
3. Press the **Start Cycle** button.

   The tool motion for a Cylinder Skew Probing Cycle is the same as the motion described for a Cylinder Cycle. The results appear in the Center X, Center Y, Diameter, and Skew Angle (deg) fields. The probe stops in the center above the cylinder.

4. The ACCEPT X/Y SKEW ANGLE (F1) and DO NOT ACCEPT (F2) softkeys appear.
   - The ACCEPT X/Y SKEW ANGLE (F1) softkey accepts the skew position and subtracts the Preset X or Y value to determine Part Zero.
   - The DO NOT ACCEPT (F2) softkey ignores the edge position value and the Preset X or Y value. Part Zero remains unchanged.

Refer to *Part Zero Storage*, on page 33 for more information.

5. The initial Part Setup screen appears with the skew value established during the cycle stored in the X/Y Skew (deg) field, if accepted.
**Rectangular Pocket Skew**

Follow these steps to program a Rectangular Pocket Skew Cycle:

1. From the Part Setup Skew Probe Cycle softkey menu, select Rectangular Pocket Inside.
2. Program an offset for the Preset X and/or Preset Y field, if desired.

When the Part Setup fields have been entered, start the Rectangular Pocket Skew Probing Cycle:

1. Place the part probe in the spindle and jog the probe to the Start Position, inside the rectangular pocket.
2. Press the START PROBING CYCLE (F1) softkey. The **Start Cycle** button flashes.
3. Press the **Start Cycle** button.

   The tool motion for a Rectangular Pocket Skew Probing Cycle is the same as the motion described for a Rectangular Pocket Cycle. The results are displayed on the screen in the Center X, Center Y, Length (X), Length (Y) and Skew Angle (deg) fields. The probe stops in the center of the pocket.

4. The ACCEPT X/Y SKEW ANGLE (F1) and DO NOT ACCEPT (F2) softkeys appear.
   - The ACCEPT X/Y SKEW ANGLE (F1) softkey accepts the skew position and subtracts the Preset X or Y value to determine Part Zero.
   - The DO NOT ACCEPT (F2) softkey ignores the edge position value and the Preset X or Y value. Part Zero remains unchanged.

Refer to *Part Zero Storage*, on page 33 for more information.

5. The initial Part Setup screen appears with the skew value established during the cycle stored in the X/Y Skew (deg) field, if accepted.
Rectangular Solid Skew

Follow these steps to program a Rectangular Solid Skew Cycle:

1. From the Part Setup Skew Probe Cycle softkey menu, select Rectangular Solid Outside.
2. In the Probing Length X field, enter the pocket’s estimated X Length.
3. In the Probing Length Y field, enter the pocket’s estimated Y Length.
4. In the Z Depth field, enter the distance the Z axis moves downward before changing direction and moving to the edges for deflection.
5. Program an offset for the Preset X and/or Preset Y field, if desired.

When the Part Setup fields have been entered, start the Rectangular Solid Skew Probing Cycle:

1. Place the part probe in the spindle and jog the probe to the Start Position, above the rectangle near the center.
2. Press the START PROBING CYCLE (F1) softkey. The Start Cycle button flashes.
3. Press the Start Cycle button.

The tool motion for a Rectangular Solid Skew Probing Cycle is the same as the motion described for a Rectangular Solid Cycle. The results appear in the Center X, Center Y, Length (X), Length (Y), and Skew Angle (deg) fields. The probe moves to the center above the rectangle.

4. The ACCEPT X/Y SKEW ANGLE (F1) and DO NOT ACCEPT (F2) softkeys appear.
   - The ACCEPT X/Y SKEW ANGLE (F1) softkey accepts the skew position and subtracts the Preset X or Y value to determine Part Zero.
   - The DO NOT ACCEPT (F2) softkey ignores the edge position value and the Preset X or Y value. Part Zero remains unchanged.

Refer to Part Zero Storage, on page 33 for more information.

5. The initial Part Setup screen appears with the skew value established during the cycle stored in the X/Y Skew (deg) field, if accepted.
**Automatic Mode**

To locate part zero and to determine skew in the XY plane as part of the program instead of manually during Part Setup, the Probe Part Setup conversational data block can be used to automatically perform this function.

Access the Probe Part Setup data block from the Part Programming screen as a Miscellaneous Data Block. Select the Miscellaneous (F5) softkey from the New Block screen and the following screen appears:

*Figure 14. New Block Screen with Probe Part Setup Softkey*
**Probe Part Setup Data Block**

From the New Block screen, press the **Probe Part Setup** softkey. The Probe Part Setup data block screen appears with fields for programming cycles to determine part zero alone or in addition to determining the skew in the XY plane.

The fields on the left-hand side of the screen and the XYZ Start fields apply to the Part Zero Cycle; the fields on the right-hand side of the screen and the Skew Start fields apply to the Skew Cycle:

![Probe Part Setup Data Block](image)

*Figure 15. Probe Part Setup Data Block*

**Probe Part Setup Fields**

The Probe Part Setup screen contains fields for the Part Zero Cycle and the Skew Cycle. The fields change depending on the selected cycle.

**Part Zero Cycles**

The fields listed below apply to Part Zero Cycles indicated and are defined as follows:

- **Tool (All cycles)**—the tool number of the part probe.
- **Part Zero Cycle (All cycles)**—contains the geometric feature for this Part Zero cycle. Softkey choices appear for selecting the appropriate feature when the cursor is in this field.
- **Probe Axis (Edge, Slot, and Web cycles)**—specifies the X, Y, or Z (Edge cycle) axis to probe.
- **Preset X** and **Preset Y (All cycles)**—define an offset for Part Zero X and/or Part Zero Y from the probed feature.
- **Probe Direction X** and **Probe Direction Y (Corner and Plane Intersection cycles)**—defines which direction, positive or negative, the probe moves when looking for the part.
- **Probing Length X** and **Probing Length Y (Rect. Pocket and Rect. Solid cycles)**—identifies the X or Y length of the geometry.
  - For Pocket this defines maximum travel.
  - For Solid this defines a clearance area to move the probe outside the Solid.

- **Probing Radius (Cylinder and Hole or Circle cycles)**—defines the radius of the geometry.
  - For Cylinder this defines a clearance range to move the probe outside the Cylinder.
  - For Hole or Circle this defines maximum travel.

- **Z Drop Down Depth (Cylinder and Rectangular Solid cycles)**—defines the distance, relative to the Start Position, the Z axis moves downward before changing direction and searching horizontally for each contact point on the geometry’s surface.

- **Offset X** and **Offset Y (Plane Intersection cycles)**—defines the X or Y offset positions from the Start location.

- **Start Angle 1; Start Angle 2; Start Angle 3 (Cylinder and Hole or Circle cycles)**—define the three probe deflection points. Relative to the 3:00 position and increasing counter-clockwise as viewed from above the part, the default angles are 0, 120, and 240, respectively. Change the angles if the part contains geometry that interferes with the defaults.

- **X, Y, Z Start Positions (All cycles)**—define where to begin the probing process. The easiest method for entering the start positions is to jog the probe down to the desired start location and press the Store Position key. A single push of this key automatically enters all three coordinates (X, Y, and Z) for one Start position.

**Skew Cycles**

These fields apply to the Skew Cycle indicated and are defined as follows:

- **Tool (All cycles)**—the tool number of the part probe.

- **Skew Cycle (All Skew cycles)**—the geometric feature for this Part Skew cycle. Softkey choices appear for selecting the appropriate feature when the cursor is in this field.

- **Skew Axis (All Skew cycles)**—define either the X or Y axis as the skew axis. There is no skewing in Z.

- **Preset X or Preset Y (All Skew cycles)**—define an offset from Part Zero X or Part Zero Y of the feature to be probed. NOTE: Preset X and Preset Y are available in all cases. For better and more accurate skew compensation, the user should enter values for both fields.

- **Z Depth (Cylinder and Rect. Solid cycles)**—defines the distance, relative to the Start Position, the Z axis moves downward before changing direction and searching horizontally for each contact point on the geometry’s surface.
• **Probing Length X and Probing Length Y (Rect. Pocket and Rect. Solid cycles)**—identifies the X or Y length of the geometry.
  • For Pocket this defines maximum travel.
  • For Solid this defines a clearance area to move the probe outside the Solid.

• **Probing Radius (Cylinder and Hole or Circle cycles)**—defines the radius of the geometry.
  • For Cylinder this defines a clearance range to move the probe outside the Cylinder.
  • For Hole or Circle this defines maximum travel.

• **Probe Direction X or Y (depending on Skew Axis selection) (Edge cycles)**—defines which direction, positive or negative, the probe moves when looking for the part.

• **Skew Start Positions (All Skew cycles)**—define where to begin the probing process. The easiest method for entering the start positions is to jog the probe down to the desired start location and press the Store Position key. A single push of this key automatically enters all three coordinates (X, Y, and Z) for one Start position.

**Probe Part Setup Data Block Execution**

When the program executes this data block, it will automatically probe the part and update Part Zero X, Part Zero Y, Probe Z and X/Y Skew. The data block may be placed anywhere in the part program except within a Pattern.
Part Quality Verification

Part Inspection data blocks are available with the Probing Option for performing part quality verification. This section describes the Part Inspection data blocks.

Part Inspection

A Part Inspection data block is available to monitor real-time data for selected probing cycles. When the data block is executed, the software creates two files: progname.txt and progname.dat, where “progname” is the name of the part program.

The files are stored on the hard drive in the same sub-directory as the part program file and contain time-stamped information about the selected geometry.

The information in the files can be used for reports and part quality verification. Both files contain the same information and are available for you to view.

- progname.txt - contains ASCII text, viewable with any editor.
- progname.dat - contains a comma delimited file that may be imported to a spreadsheet.

Part Inspection Cycles

To access the Part Inspection cycles, follow this softkey sequence from the Input screen:

1. Select PART PROGRAMMING.
2. Select MISCELLANEOUS.
3. Select MORE. This screen appears:

![Figure 16. New Block Screen with Part Inspection Softkey](image)

Figure 16. New Block Screen with Part Inspection Softkey
Select the PART INSPECTION (F5) softkey and this screen appears:

![Part Inspection Screen]

**Figure 17. Part Inspection Screen**

**Part Inspection Fields**

The fields on the Part Inspection screen change depending on the selected cycle and are defined as follows:

- **Tool (All cycles)**—the part probe tool number.
- **Inspection Cycle (All cycles)**—the type of cycle to be inspected.
- **Probe Axis (Single Point cycle)**—the axis to be probed.
- **Probe Direction X or Probe Direction Y (Single Point and Plane Intersection)**—based on the Probe Axis field, defines which direction, positive or negative, the probe moves when looking for the part.
- **Probing Length X or Probing Length Y (Rectangular Pocket and Rectangular Solid cycles)**—identifies the X or Y length of the geometry.
  - For Pocket this defines maximum travel.
  - For Solid this defines a clearance area to move the probe outside the Solid.
- **Probing Radius (Cylinder and Hole or Circle cycles)**—defines the radius of the geometry.
  - For Cylinder this defines a clearance range to move the probe outside the Cylinder.
  - For Hole or Circle this defines maximum travel.
• **Z Depth (Cylinder and Rectangular Solid cycles)**—defines the distance, relative to the Start Position, the Z axis moves downward before changing direction and searching horizontally for each contact point on the geometry’s surface.

• **Offset X** or **Offset Y (Plane Intersection cycle)**—defines the X or Y offset positions from the Start location.

• **Start Angle1; Start Angle2; Start Angle3 (Cylinder and Hole or Circle cycles)**—define the three probe deflection points. Relative to the 3:00 position and increasing counter-clockwise as viewed from above the part, the default angles are 0, 120, and 240, respectively. Change the angles if the part contains geometry that interferes with the defaults.

• **X, Y, Z Start Positions (All cycles)**—define where to begin the probing process. The easiest method for entering the start positions is to jog the probe down to the desired start location and press the Store Position key. A single push of this key automatically enters all three coordinates (X, Y, and Z) for one Start position.

**Part Inspection Programming**

Follow these steps to program a Part Inspection data block:

1. Enter the part probe tool number in the tool field.
2. When the cursor is on the Inspection Cycle field, the softkeys change. Select a cycle type for the Inspection Cycle field.
3. Program the remaining fields as described in the “Part Inspection Fields” section. The data block is stored with the program and executed automatically.

**Part Inspection Files**

When the Part Inspection data block is executed, the software automatically creates the part inspection files. The position data is presented in part relative coordinates.

• To view the files on your PC, first copy them to a disk in the floppy drive. Follow the steps for saving files to disk in the Programming Basics chapter of the Getting Started with WinMax manual.

→ It is not possible to view the Part Inspection files on the CNC.
Here is a sample Part Inspection Probe.txt file:

******************************************
*       PART INSPECTION DATA      *
******************************************
Cylinder inspection (block # 3) executed 15:46:58   8/10/2000
Part Count = 4
Center            Diameter
X  16.7168 inches ( 424.607 mm)1.8878 inches ( 47.951 mm)
Y  10.0995 inches ( 256.527 mm)

Part Count = 5
Point
X  10.3365 inches ( 262.548 mm)
Y  10.3882 inches ( 263.860 mm)
Z  14.2154 inches ( 361.070 mm)

Hole/circle inspection (block # 4) executed  9:50:45   8/11/2000
Part Count = 5
Center            Diameter
X  9.3299 inches ( 236.978 mm)2.7577 inches ( 70.045 mm)
Y  15.4572 inches ( 392.612 mm)

Cylinder inspection (block # 5) executed  9:51:02   8/11/2000
Part Count = 5
Center            Diameter
X  16.7171 inches ( 424.613 mm)1.8883 inches ( 47.962 mm)
Y  10.0996 inches ( 256.531 mm)

Rectangular pocket inspection (block # 6) executed  9:51:24   8/11/2000
Part Count = 5
Center            Length
X  14.4527 inches ( 367.099 mm)X  4.7259 inches ( 120.037 mm)
Y  14.3001 inches ( 363.223 mm)Y  4.7258 inches ( 120.034 mm)

Part Count = 5
Center            Length
X  9.4447 inches ( 239.895 mm)X  3.8176 inches ( 96.967 mm)
Y  12.1092 inches ( 307.575 mm)Y  1.2629 inches ( 32.079 mm)

Plane intersect inspection (block # 8) executed  9:52:08   8/11/2000
Part Count = 5
Corner
X  7.7663 inches ( 197.264 mm)
Y  13.2841 inches ( 337.417 mm)
TOOL PROBING OPTION

Tool Probing measures **Tool Cal Length** (in Absolute Tool Length mode) or **Zero Calibration** (in Zero Calibration Mode) and **Diameter/Diameter Wear** using either a touch probe or a laser probe.

If the probe is newly installed or has been relocated, it must be calibrated and the parameters set before probing. See **Appendix A: Tool Probe Setup, on page 8 - 19** and **Appendix B: Tool Probe Calibration, on page 8 - 29**. These need to be done only when the probe is installed, replaced, or relocated, NOT each time it is used.

To get started, select the calibration mode you use:

- **Tool Probing in Absolute Tool Length Mode, on page 8 - 2**
- **Tool Probing in Zero Calibration Mode, on page 8 - 8**

Calibration mode is set in Utilities/User Preferences/Tool Utilities and Settings. For more information, see **Tool Calibration Modes, on page 4 - 16** in *Getting Started with WinMax Mill*.

Included in this chapter:

- **Tool Probing in Absolute Tool Length Mode, on page 8 - 2**
- **Tool Probing in Zero Calibration Mode, on page 8 - 8**
- **Tool Quality Monitoring, on page 8 - 14**
- **Appendix A: Tool Probe Setup, on page 8 - 19**
- **Appendix B: Tool Probe Calibration, on page 8 - 29**
- **Appendix C: Probing Parameter Definitions, on page 8 - 33**
Tool Probing in Absolute Tool Length Mode

In **Absolute Tool Length** mode, the tool is probed and the **Tool Cal Length** and **Diameter Wear** values are measured.

⇒ The probe must be setup and calibrated before proceeding. This is done only once, unless the probe is replaced or relocated. For more information see Appendix A: Tool Probe Setup, on page 8 - 19 and Appendix B: Tool Probe Calibration, on page 8 - 29.

Set the Probing Parameters—Absolute Tool Length Mode

The tool information for the tool(s) to be probed and the probing cycle information is entered in Tool Setup:

1. From the **Input** screen, select the **Tool Review** softkey, then the **Tool Setup** softkey.
2. Select the number of the tool you will be probing, or if it is a new tool, enter the number and type.
3. Enter the tool diameter in the **Diameter** field.
4. Enter an approximate tool length in the **Tool Cal Length** field. Find this approximate length by measuring the tool in the spindle from the spindle nose to the tip of the tool. This value will be updated when the tool is probed.
5. Enter other parameters as necessary to define the tool. For more information, see Tool Setup Fields, on page 4 - 21 in Getting Started with WinMax Mill.
6. Select the **More** softkey to access the second set of Tool Setup softkeys.
7. Select the **Tool Probing** softkey. The Tool Probing screen opens:

![Tool Probing screen in Absolute Tool Length mode](image)

*Figure 8–1. Tool Probing screen in Absolute Tool Length mode*
The current tool is displayed in the Tool field. The cycle parameters are displayed on the Main tab, and are read-only. Default values are set in the Tool Probing Cycle Defaults screen, see Tool Probing Cycle Defaults, on page 8 - 4. If you need to change one or more parameters for the current tool, check the Edit Parameters check box. (If you need to change a parameter for all tools, go back to the Tool Probing Cycle Defaults screen and make the change there.) The defaults should be set before proceeding with the probing cycle.

To set the remaining cycle parameters on the Main tab:

1. Select the type of probing cycle from the drop-down list in the Cycle field:
   - **Length**—measures the Tool Cal Length. This value is automatically calculated by subtracting the probed Z location of the tool in spindle from the Z location of the probe stored in the Tool Measurement screen.
   - **Diameter**—measures the actual tool diameter, and uses this value to determine the Diameter Wear and Radius Offset. For more information, see the Diameter Wear definition in Tool Setup Fields, on page 4 - 21 in Getting Started with WinMax Mill.
   - **Length & Diameter**—measures both length and diameter.

2. Indicate Yes or No for Multi Tool Probing. Yes will include the current tool in multi-tool probing cycle.

3. Specify a Sister Tool to be used as a replacement if tool wear is out of tolerance. See Tool Quality Monitoring, on page 8 - 14.

The Length and Diameter tabs specify where these values are stored. These are also set in the Tool Probing Cycle Defaults screen, but can be changed here by selecting the Edit Parameters check box:

- **Length**—results are stored as either the Tool Length, or as a Tool Length Offset, or both. When stored as an offset, the Offset Number field appears where you can indicate the number in the Tool Offset table where the value is to be stored.
- **Diameter**—results are stored as either Diameter Wear, Tool Radius Offset, Diameter, Diameter Wear and Radius Offset, or Diameter and Radius Offset. When stored as an offset, the Offset Number field appears where you can indicate the number in the Tool Offset table where the value is to be stored.
Tool Probing Cycle Defaults

These are the Tool Probing Cycle defaults in Absolute Tool Length mode. For Zero Calibration Mode, see Tool Probing Cycle Defaults, on page 8 - 10.

Defaults for probing cycle parameters that infrequently need to be changed are set and stored in the Tool Probing Cycle Defaults screen. Once these parameters are set, they are automatically applied to each tool and do not have to be accessed again. However, if it is necessary to change one or more parameters for a specific tool, this is done in the Tool Probing screen. To access the Tool Probing Cycle Defaults screen:

1. From the Tool Probing screen, select the Tool Probe Setup softkey. The Tool Probe Parameters screen opens.
2. Select the Tool Probing Cycle Defaults softkey. The Tool Probing Cycle Defaults screen opens. Set the default values in this screen.

3. Select the Spindle Usage. This is the direction the spindle turns during the probe cycle.
4. Enter a Spindle Speed, if Spindle Usage is CW or CCW.
5. Specify the Fast Feed and Slow Feed. These are the feedrates of the initial touch (Fast) and measurement touch (Slow).
6. Enter a value in the Rapid Clearance field. This is the distance above probe stylus or beam that determines Rapid Z Position.
7. Specify the Min Length Delta. This is the distance below the probe stylus or beam that the Z-axis is allowed to travel; determines the Min Z Position.
8. Specify the X and Y Length Offsets, if probing for length of a tool with its cutter offset from the center, for example, a face mill. Otherwise, leave at 0.0.
9. Specify Z Drop Down Depth for diameter. This is the distance the tool drops down from the point where the tool tip touches the top of the probe, when probing for diameter. Value is always negative.
10. Specify Spindle Clearance, the distance between the tool and the probe.
when the tool drops down for diameter probing.

11. Specify how to store the Length and Diameter results:
   - **Length** can be stored as Length, Offset, or both.
   - **Diameter** can be stored as:
     - Diameter Wear
     - Radius offset
     - Diameter
     - Diameter Wear and Radius Offset
     - Diameter and Radius Offset

Select the **Apply Defaults to Current Tool** softkey to apply these defaults to the tool currently being edited. You only need to do this when you first set the defaults or when you change any of the values.

Select the **Apply Defaults to All Tools** softkey to apply these defaults to all tools in the active program’s tool setup, or to the entire tool library if using the Tool and Material Library option.

Select **Exit** twice to return to the Tool Probing screen.

**Run the Probe Cycle**

When the cycle parameters are set, you are ready to run the probe cycle:

1. Select the **Probe Single Tool** softkey.
   - If the tool that you wish to probe is already in the spindle, select the **Probe Current Tool Now** softkey.
   - To probe multiple tools, see *Probe Multiple Tools, on page 8 - 7*.

2. Enter the Tool Number to probe. The following sequence occurs for length probing:
   a. The **Start Cycle** button flashes and a prompt to press Start appears.
   b. Press the **Start Cycle** button to continue.
      - If the tool to be probed is not in the spindle, a tool change occurs.
      - If the tool to be probed is not in the magazine, the software prompts for the tool.
      - The spindle operates as specified in the **Spindle Usage** field in Tool Setup.
   c. The Z axis moves downward at rapid feed until it reaches **Rapid Z Position**. Axis Feedrate Override is active during this move.
      - The Z axis continues moving at **Fast Feed** until a probe deflection occurs.
      - If the probe reaches **Min Z Position** prior to deflection, an error message appears. The value may need to be adjusted to correct the problem.
d. The tool touches the probe (or breaks the beam for laser).

e. For a touch probe:
   - The tool retracts slightly (at Slow Feed) and makes three touches (deflections) at Slow Feed. The average length of these deflections is used to determine the tool length.

For a laser probe:
   - The tool retracts up out of the beam (at Slow Feed) until the beam is uninterrupted. The measurement is always taken when the tool moves out of the beam. The average of these readings is used to determine the tool length.

3. The Probe Single Tool cycle is now complete for length. Select the Exit softkey to return to the initial Tool Setup screen. The Tool Cal Length field is updated and the “P” designator appears.

Continue with the next step to probe the tool’s diameter.

4. If you entered an estimate of the tool’s diameter in the Diameter field and selected Diameter or Length & Diameter in the Cycle field on the Tool Probing screen, the probe cycle continues with the tool Diameter sequence:
   a. The tool retracts just above the probe and moves to one side at Spindle Clearance.
   b. The tool drops down to Z Drop Down Depth, below the top of the tool probe stylus or beam, and moves toward the probe.
   c. The tool touches the probe (or breaks the beam for a laser).
   d. For a touch probe:
      - The tool retracts slightly (at Slow Feed) and makes three touches (deflections) at Slow Feed. The average of these deflections is stored for calculation after the other side of the tool is probed.

For a laser probe:
   - The tool retracts out of the beam (at Slow Feed) until the beam is uninterrupted. The measurement is always taken when the tool moves out of the beam. The average of these readings is stored for calculation after the other side of the tool is probed.

   e. The tool moves up and over to the other side of the stylus or beam (to Spindle Clearance), and steps a-d are repeated from the other side of the probe.
   f. The two readings (one from each side of the probe) are used to determine the actual diameter of the tool.

5. The actual probed diameter is subtracted from the tool diameter entered in Tool Setup, and this value appears in the Diameter Wear field on the Tool Setup screen with the “P” designator.
Probe Multiple Tools

The Probe Multiple Tools cycle determines Tool Cal Length (Length) and/or Diameter/Diameter Wear based on the setting of the probe cycle field. All tools that are defined in Tool Setup and have Multi-Tool Probing field set to Yes will be included in the multiple tool probing cycle. Any tool that has Multi Tool Probing set to no will be skipped but may still be probed using the single tool cycle.

The tools must be entered in Tool Setup and the probe cycle parameters set; see Set the Probing Parameters—Absolute Tool Length Mode, on page 8 - 2.

Follow these steps to perform a Probe Multiple Tools cycle:

1. Select the **Probe Multiple Tools** softkey.
2. The following sequence occurs:
   a. The **Start Cycle** button flashes and a prompt requests you to press **Start** to initiate the multiple tool probing cycle.
   b. Press the **Start Cycle** button to continue. The first tool to be calibrated is determined by the current tool in the spindle, the **Cycle** field and the value of the Multi Tool Probing field. The control starts with the current tool in spindle and scans in ascending order. A tool must have a Cycle defined (Length, Diameter, or both) and the Multi Tool Probing field set to Yes. The first tool to meet these requirements is placed in the spindle.
      1. The tool is probed as described in Run the Probe Cycle, on page 8 - 5.
      2. The tools list is scanned again in ascending order until the next tool to meet the multi tool probing criteria is found. A tool change occurs and the tool is probed as described in Run the Probe Cycle, on page 8 - 5.
      3. This process repeats until there are no more tools to probe. During the tool scanning when the highest number tool is reached the list wraps around to tool one. The scan continues until the original tool in spindle is reached. When this occurs the multi tool probing cycle is complete.

Note that once this process begins the operator is no longer required. The entire process is automatic (as long as the tools are in the ATC).
Tool Probing in Zero Calibration Mode

In **Zero Calibration** mode, the tool is probed and the **Zero Calibration** and **Diameter Wear** values are calculated.

![Diagram](image)

The probe must be setup and calibrated before proceeding. This is done only once, unless the probe is replaced or relocated. For more information see, *Appendix A: Tool Probe Setup*, on page 8 - 19 and *Appendix B: Tool Probe Calibration*, on page 8 - 29.

Set the Probing Parameters—**Zero Calibration** mode

The tool information for the tool(s) to be probed and the probing cycle information is entered in Tool Setup:

1. From the **Input** screen, select the **Tool Review** softkey, then the **Tool Setup** softkey.
2. Select the number of the tool you will be probing, or enter a number and type, if it is a new tool.
3. Enter other parameters as necessary to define the tool. For more information, see *Tool Setup Fields*, on page 4 - 21.
4. Select the **More** softkey to access the second set of Tool Setup softkeys.
5. Select the **Tool Probing** softkey. The Tool Probing screen opens:

![Tool Probing Screen](image)

The current tool is displayed in the **Tool** field. The cycle parameters are displayed but are read-only. Parameter default values are set in the Tool Probing Cycle Defaults screen, see *Tool Probing Cycle Defaults*, on page 8 - 10. If you need to change one or more parameters for the current tool, check the **Edit Parameters** check box. (If you need to change a parameter for all tools, go back to the Tool Probing Cycle Defaults screen and make the change there.) The defaults should be set before proceeding with the probing cycle.
To set the remaining cycle parameters for the current tool:

1. Select the type of probing cycle from the drop-down list in the **Cycle** field:
   - **Length**—measures the tool length (as **Zero Calibration**). This value is automatically calculated using the stored internal value and the Probe Z value obtained from probe calibration.
   - **Diameter**—measures the actual tool diameter, and uses this value to determine the Diameter Wear. For more information, see the Diameter Wear definition in Tool Setup Fields, on page 4 - 21.
   - **Length & Diameter**—measures both length and diameter.

2. Ensure the correct value has been entered in the **Rapid Z Position** field. This is the position just above the contact point of the probe that the tool rapid down to before continuing downward at Fast Feed for the initial touch.

   To set Rapid Z Position, use the remote jog unit to jog the part probe to a point just above the contact point of the tool probe and select the **Set Position** softkey, or type in a value for the Rapid Z Position field. (With the cursor in this field, the **Position Tool Over Probe** softkey can be used to position the part probe over the tool probe before jogging Z to the desired position.)

3. Specify the **Min Z Position**. This is the lowest position that Z will be allowed to travel during the probe cycle.

4. Indicate Yes or No for **Multi Tool Probing**. Yes will include the current tool in multi-tool probe cycle.

5. Specify a **Sister Tool** to be used for as a replacement if tool wear is out of tolerance. See Tool Quality Monitoring, on page 8 - 14.

6. The **Length** and **Diameter** tabs contain the values for these parameters set in the Tool Probing Cycle Defaults screen, and may be changed here for the current tool; check the Edit Parameters check box.

### Determine Probe Z

The **Probe Z** value of the part needs to be set whenever a new part is placed on the table. The **Zero Calibration** value of the tool is calculated using the new Probe Z value, the stored internal value (from tool probe calibration), and the probed tool length.

If you are using a reference tool to calibrate the tool probe:

1. From the Input screen, select the **Part Setup** softkey to open the Part Setup screen.

2. With the cursor in the **Probe Z** field, touch off the top of the part with the part probe or reference tool.

3. Select the Store Machine Position softkey. This value is stored in Probe Z.

If you are using a part probe to calibrate the tool probe, use the part probe **Z Edge cycle** to determine Probe Z. See Edge, on page 7 - 14 in Part Probing.
Tool Probing Cycle Defaults

Defaults for probing cycle parameters that infrequently need to be changed are set and stored in the Tool Probing Cycle Defaults screen. Once these parameters are set, they are automatically applied to each tool and do not have to be accessed again. However, if it is necessary to change one or more parameters for a specific tool, this is done in the Tool Probing screen. To access the Tool Probing Cycle Defaults screen:

1. From the Tool Probing screen, select the Tool Probe Setup softkey. The Tool Probe Parameters screen opens.
2. Select the Tool Probing Cycle Defaults softkey. The Tool Probing Cycle Defaults screen opens. Set the default values in this screen.

![Tool Probing Cycle Defaults screen in Zero Calibration mode](image)

3. Select the Spindle Usage. This is the direction the spindle turns during the probe cycle.
4. Enter a Spindle Speed, if Spindle Usage is CW or CCW.
5. Specify the Fast Feed and Slow Feed. These are the feedrates of the initial touch (Fast) and measurement touch (Slow).
6. Specify the X and Y Length Offsets, if probing for length of a tool with its cutter offset from the center, for example, a face mill. Otherwise, leave at 0.0.
7. Specify Z Drop Down Depth for diameter. This is the distance the tool drops down from the point where the tool tip touches the top of the probe, when probing for diameter. Value is always negative.
8. Specify Spindle Clearance, the distance between the tool and the probe when the tool drops down for diameter probing.

Select the Apply Defaults to Current Tool softkey to apply these defaults to the tool currently being edited. You only need to do this when you first set the defaults or when you change any of the values.

Select Exit twice to return to the Tool Probing screen.
Run the Probe Cycle

When the **Probe Z** value is set, return to the Tool Setup/Tool Probing screen

1. Select the **Probe Single Tool** softkey.

   ![Diagram]

   If the tool that you wish to probe is already in the spindle, select the **Probe Current Tool** softkey.

   To probe multiple tools, see *Probe Multiple Tools, on page 8 - 7*.

2. Enter the Tool Number to probe. The following sequence occurs for length probing:
   a. The **Start Cycle** button flashes and a prompt to press Start appears.
   b. Press the **Start Cycle** button to continue.
      - If the tool to be probed is not in the spindle, a tool change occurs.
      - If the tool to be probed is not in the magazine, the software prompts for the tool.
      - The spindle operates as specified in the **Spindle Usage** field in Tool Setup.
   c. The Z axis moves downward at rapid feed until it reaches **Rapid Z Position**. Axis Feedrate Override is active during this move.
      - The Z axis continues moving at **Fast Feed** until a probe deflection occurs.
      - If the probe reaches **Min Z Position** prior to deflection, an error message appears. The value may need to be adjusted to correct the problem.
   d. The tool touches the probe (or breaks the beam for laser).
   e. For a touch probe:
      - The tool retracts slightly (at **Slow Feed**) and makes three touches (deflections) at **Slow Feed**. The average length of these deflections is used to determine the tool length.

   For a laser probe:
   - The tool retracts up out of the beam (at **Slow Feed**) until the beam is uninterrupted. The measurement is always taken when the tool moves out of the beam. The average of these readings is used to determine the tool length.

3. The Probe Single Tool cycle is now complete for length. Select the **Exit** softkey to return to the initial Tool Setup screen. The **Zero Calibration** field is updated and the “P” designator appears.

   Continue with the next step to probe the tool’s diameter.
4. If you entered an estimate of the tool’s diameter in the Diameter field and selected Diameter or Length & Diameter in the **Cycle** field on the Tool Probing screen, the probe cycle continues with the tool Diameter sequence:

   a. The tool retracts just above the probe and moves to one side at **Spindle Clearance**.

   b. The tool drops down to **Z Drop Down Depth**, below the top of the tool probe stylus or beam, and moves toward the probe.

   c. The tool touches the probe (or breaks the beam for a laser).

   d. For a touch probe:

      - The tool retracts slightly (at **Slow Feed**) and makes three touches (deflections) at **Slow Feed**. The average of these deflections is stored for calculation after the other side of the tool is probed.

   For a laser probe:

      - The tool retracts out of the beam (at **Slow Feed**), until the beam is uninterrupted. The measurement is always taken when the tool moves out of the beam. The average of these readings is stored for calculation after the other side of the tool is probed.

   e. The tool moves up and over to the other side of the stylus or beam (to **Spindle Clearance**), and steps a-d are repeated from the other side of the probe.

   f. The two readings (one from each side of the probe) are used to determine the actual diameter of the tool.

5. The actual probed diameter is subtracted from the tool diameter entered in Tool Setup, and this value appears in the **Diameter Wear** field on the Tool Setup screen with the “P” designator.
**Probe Multiple Tools**

The Probe Multiple Tools cycle determines Zero Calibration and/or Diameter Wear (Diameter) based on the setting of the probe cycle field. All tools that are defined in Tool Setup and have **Multi-Tool Probing** field set to Yes will be included in the multiple tool probing cycle. Any tool that has Multi Tool Probing set to no will be skipped but may still be probed using the single tool cycle.

The tools must be entered in Tool Setup and the probe cycle parameters set; see *Set the Probing Parameters—Zero Calibration mode, on page 8-8*.

Follow these steps to perform a Probe Multiple Tools cycle:

1. Select the **Probe Multiple Tools** softkey.
2. The following sequence occurs:
   a. The **Start Cycle** button flashes and a prompt requests you to press **Start** to initiate the multiple tool probing cycle.
   b. Press the **Start Cycle** button to continue. The first tool to be calibrated is determined by the current tool in the spindle, the **Cycle** field and the value of the **Multi Tool Probing** field. The control starts with the current tool in spindle and scans in ascending order. A tool must have a Cycle defined (Length, Diameter, or both) and the Multi Tool Probing field set to Yes. The first tool to meet these requirements is placed in the spindle.
      1. The tool is probed as described in *Run the Probe Cycle, on page 8-11*.
      2. The tools list is scanned again in ascending order until the next tool to meet the multi tool probing criteria is found. A tool change occurs and the tool is probed as described in *Run the Probe Cycle, on page 8-11*.
      3. This process repeats until there are no more tools to probe. During the tool scanning when the highest number tool is reached the list wraps around to tool one. The scan continues until the original tool in spindle is reached. When this occurs the multi tool probing cycle is complete.

Note that once this process begins the operator is no longer required. The entire process is automatic (as long as the tools are in the ATC).
Tool Quality Monitoring

Tool Monitoring is available to automatically monitor calibrated tools and detect breakage or wear. The software compares the current tool dimensions to the calibrated dimensions stored in Tool Setup for the programmed tool. If the current dimensions deviate from the defined tolerance programmed in the tool monitoring menus, the tool is defective.

This section describes the ways to perform Tool Monitoring:

*Probe Tool Monitoring Data Block*, on page 8 - 14

*Automatic Tool Monitoring Parameter*, on page 8 - 18

**Probe Tool Monitoring Data Block**

It is possible to program a *spare tool* (Sister Tool) to automatically replace a defective monitored tool. If a Sister Tool is not programmed, or if there is no ATC, axis motion stops and the following message appears on the screen:

“Tool # x is defective, no more tools to substitute.”

To access the Probe Tool Monitoring data block, follow this softkey sequence from the Input screen:

1. Select **Part Programming**.
2. Select **Miscellaneous**.
3. Select **More**.
4. Select **Tool Monitoring (Probing)**.

![Probe Tool Monitoring Screen](image)

*Figure 9. Probe Tool Monitoring Screen*
From the Probe Tool Monitoring screen, select the type of measuring cycle by selecting the appropriate softkey. The measuring cycles are described in the following sections.

As with any data block, use the Insert Block Before and Delete Block softkeys available for each measuring data block to add and delete measuring cycles from the program.

**Tool Breakage Detection**

Tool Breakage Detection monitors the tool for breakage. Follow these steps to program a Tool Breakage Detection Cycle:

1. From the Probe Tool Monitoring screen, select **Tool Breakage Detection** in the **Probe Cycle Type** field.
2. Enter the tool to be monitored, in the **Tool** field. The tool must be programmed in Tool Setup, and it must be probed.
3. If desired, adjust the **Speed (RPM)** value.
4. Enter the **Breakage Tolerance**. This is the amount of deviation from the tool length programmed in the Tool Cal Length field in Tool Setup.

When the data block is executed in the part program, the current tool length is measured and compared with the **Tool Cal Length/Zero Calibration** in Tool Setup. The figure below illustrates the tolerance for tool breakage:

![Figure 10. Tool Breakage Tolerance](image)

The software monitors the tool and determines if the tool is within the Breakage Tolerance. If the tool is shorter than the programmed tolerance, the tool is broken. If the tool is broken, the software checks for a spare tool.

- If a Sister (spare) Tool has been entered in Tool Setup for this tool, axis motion stops and a tool change automatically occurs.
If there is no Sister Tool programmed for this tool, or if there is no ATC, axis motion will stop and a message appears telling you to change tools.

These are the fields when Tool Breakage Detection is selected for Probe Cycle Type:

- **Probe Cycle Type**—contains the type of cycle selected from the Probe Tool Monitoring menu.
- **Tool**—the number and description of the tool to be monitored.
- **Breakage Tolerance**—the amount of deviation from the tool length programmed in the Tool Cal Length field in Tool Setup.
- **Speed (RPM)**—contains the spindle speed value defined in Tool Setup for this tool. This value can be adjusted in Tool Monitoring.
- **Direction**—contains the direction the spindle will turn.
- **Probing Method**—displays the method used: Length, Diameter, Length & Diameter. Field is read-only.
- **Spindle Usage**—the spindle direction. Field is read-only.
- **Sister Tool**—contains the number and description of the tool defined in Tool Setup as the Sister (spare) for this tool. This field is read-only.

**Tool Wear Detection**

To monitor tool length and/or diameter wear, select from the following Probe Cycle Type choices:

- Tool Length Wear Detection
- Tool Diameter Wear Detection
- Tool Length & Diameter Wear Detection

Follow these steps to program a Tool Wear Detection Cycle:

1. From the Probe Tool Monitoring screen, select the **Probe Cycle Type**.
2. Enter the **Tool** to be monitored. The tool must be programmed in Tool Setup, and it must be calibrated.
3. Enter Length Tolerance or Diameter Tolerance, or both if monitoring both length and diameter.

When the data block is executed in the part program, the current tool length is measured and compared with the tool length or diameter tolerance, or both if monitoring both length and diameter. The figure below illustrates the tool wear tolerances:
The software monitors the tool and determines if the tool is within the Length Tolerance or the Diameter Tolerance, or both if monitoring both length and diameter.

- If the tool is shorter than the Length Tolerance value minus the length, the tool is worn.
- If the tool’s diameter is less than the Diameter minus the programmed tolerance, the tool is worn.
- If the tool is worn, the software checks for a sister tool.
- If a Sister (spare) Tool has been entered in Tool Setup for this tool, axis motion stops and a tool change automatically occurs.
- If there is no Sister Tool programmed for this tool, or if there is no ATC, axis motion will stop and a message appears telling you to change tools.

The fields on the Wear Detection screens are defined as follows:

- **Probe Cycle Type**—contains the type of cycle selected from the Probe Tool Monitoring menu.
- **Tool**—the number and description of the tool to be monitored.
- **Length Tolerance**—the amount of deviation from the tool length programmed in the Tool Cal Length/Zero Calibration field in Tool Setup. This field is not available in the Diameter Wear Detection screen.
- **Diameter Tolerance**—the amount of deviation from the tool diameter programmed in the Diameter field in Tool Setup screen. This field is not available in the Length Wear Detection screen.
- **Probing Method**—displays the Probing Method selected on the Tool Probing screen. Field is read-only.
- **Spindle Usage**—displays the Spindle Usage selected in Probing Parameters screen. Field is read-only.
- **Sister Tool**—contains the number and description of the tool defined in Tool Setup as the Sister (spare) for this tool. This field is read-only.

**Automatic Tool Monitoring Parameter**

Another method of monitoring tool quality is the Probing Parameter’s **Automatic Tool Monitoring** field. This setup automatically checks every probed tool for wear immediately after the tool change.

To set this parameter to automatically monitor tool quality:

1. From the Input screen, select the **Program Parameters** softkey.
2. Select the **Probing** tab.
3. Set the **Automatic Tool Monitoring** parameter to Yes.
4. Specify the **Length Tolerance**. This is the amount of deviation from the tool length programmed in Tool Setup screen.
5. Specify the **Diameter Tolerance**. This is the amount of deviation from the tool diameter programmed in Tool Setup screen.

The software monitors the tool and determines if the tool is within the Length Tolerance or the Diameter Tolerance, or both if monitoring both length and diameter.

- If the tool is shorter than the Length Tolerance value minus the length, the tool is worn.
- If the tool’s diameter is less than the Diameter minus the programmed tolerance, the tool is worn.
- If the tool is worn, the software checks for a sister tool.
- If a Sister (spare) Tool has been entered in Tool Setup for this tool, axis motion stops and a tool change automatically occurs.
- If there is no Sister Tool programmed for this tool, or if there is no ATC, axis motion will stop and a message appears telling you to change tools.
Appendix A: Tool Probe Setup

Tool probe setup parameters provide critical information on how the machine will use the probes. The parameters include fields specific to programming a touch probe or a laser probe.

Tool probe setup parameters should only be adjusted when a tool probe is newly installed, relocated, or when probing feedrates are changed.

From Tool Setup, select the Tool Probing softkey, then the Tool Probe Setup softkey. The Tool Probe Parameters screen opens. From here, you can access the parameters for the type of probe you select, either Touch Probe or Laser Probe.

Touch Probe Parameters

Select the Touch Probe Parameters softkey to access the touch tool probe fields. The following screen appears:

![Touch Probe Parameters Screen](image)

Figure 8–1. Touch Probe Parameters

Tool Setup Touch Probe Parameters are defined as follows:

- **Type**—indicates the type of tool probe. If a tool probe is not present, the remaining tool probe parameters are not used.

- **Contact Point X** and **Contact Point Y** (touch tool probe only)—indicates the X and Y location (in machine coordinates) of the tool probe. When the machine is at this location, a tool will touch the center of the tool probe stylus. To enter these values easily, insert a tool in the spindle and jog down to the tip of the probe. When the tool tip is centered over the stylus, press the Store Position key on the jog controls.
• **Setup Fast Feed** — indicates the feed rate to use for setup moves near the probe. For example, when dropping down next to the probe to measure diameter, the drop down move uses this feed rate. This value is also used for calibrating the probe and the initial touch when determining deflection offsets.

• **Retract Feed** — indicates the feed rate to use when retracting away from the probe immediately after a deflection. This value is also used for the slow moves when determining deflection offsets.

• **Repetitions** — specifies the number of Slow Feed touches when touching tools to the probe. You can program up to 99 repetitions to get the average length and diameter of the tool.

• **Probing Axis** — specifies the axis of deflection in the X/Y plane. Orientation of the probe will determine if it deflects along the X axis or the Y axis. It is assumed the probe will always deflect along the Z axis.

• **Stylus Width** (touch tool probe only) — specifies the width of the probe’s stylus along the Probing Axis.

• **Max Spread** — the deviation (difference) between the minimum and maximum probe readings. If the deviation is greater than the Max Spread value, an error message appears.

• **INIT Retract** — scales the initial retract move after a deflection.

• **INCR Retract** — scales the incremental moves that may be required if the probe is still deflected after the initial move.
Laser Probe Parameters

Select the **Laser Probe** softkey to access the laser tool probe fields:

![Laser Probe Parameters](image)

**Figure 8–2. Laser Probe Parameters**

Tool Setup Laser Probe Parameters are defined as follows:

- **Type**—indicates the type of tool probe. If a tool probe is not present, the remaining tool probe parameters are not used.

- **Center Beam X** and **Center Beam Y** (laser tool probe only)—indicates the center location of the beam in X or Y, depending on the orientation of the laser probe. Do not change this value after the Laser Tool Calibration cycle has been run, unless you re-run the cycle. The user enters an approximate value then the laser calibration cycle determines the precise location.

- **Setup Fast Feed**—indicates the feed rate to use for setup moves near the probe. For example, when dropping down next to the probe to measure diameter, the drop down move uses this feed rate. This value is also used for calibrating the probe and the initial touch when determining deflection offsets.

- **Retract Feed**—indicates the feed rate to use when retracting away from the probe immediately after a deflection. This value is also used for the slow moves when determining deflection offsets.

- **Repetitions**—specifies the number of Slow Feedrate touches when touching tools to the probe. You can program up to 99 repetitions (default value is 3 for touch probe and 4 for laser probe) to get the average length and diameter of the tool.

- **Probing Axis**—specifies the axis of deflection in the X/Y plane. Orientation of the probe will determine if it deflects along the X axis or the Y axis. It is assumed the probe will always deflect along the Z axis.
• **Beam Offset** (laser tool probe only)—displays the width of the beam based on + and - trigger points. This field is updated after running the Determine Laser Beam Offset cycle. It may be adjusted by the user to optimize performance.

• **Cal Tool D** (laser tool probe only)—contains the diameter (D) of the Laser Calibration Tool. This value can be obtained by measuring the diameter of a precision dowel or a laser calibration tool.

• **Cal Tool H** (laser tool probe only)—contains the height (H) of the Laser Calibration Tool. This value can be obtained by measuring the height of a precision dowel or a laser calibration tool.

• **Cal Tool L** (laser tool probe only)—contains the length (L) of the Laser Calibration Tool. This value can be obtained by measuring the length of a precision dowel or a laser calibration tool.

• **Max Spread**—The deviation (difference) between the minimum and maximum probe readings. If the deviation is greater than the Max Spread value, an error message appears.

• **INIT Retract**—scales the initial retract move after a deflection.

• **INCR Retract**—scales the incremental moves that may be required if the probe is still deflected after the initial move.

• **Drip Rej. Samples**—enter the number of samples for coolant drip rejection. Coolant drip rejection is the process the control uses to determine if enough coolant has cleared from the laser probe for proper function during tool monitoring. The value entered in the field is the number of probe samples the control takes in a specified period of time to determine if a clean probe state is present. Increase the value for heavy coolant flow.

• **Drip Rej. Delay**—this value is the time in milliseconds between each sample (specified in the **Drip Rej. Samples** field). For example, if Drip Rej. Samples is 200 and Drip Rej. Delay is 5 milliseconds (.005 seconds), the control must see the laser in a clean state for 1 second (200 x 5 msec) before continuing with tool monitoring.
Laser Beam Calibration

The Laser Beam Calibration cycle uses a tool to probe the beam and determine the exact trigger point position in the light beam for X/Y and Z axes so the light beam can effectively measure tools. The **Determine Laser Beam Offset** softkey on the Tool Setup Probe Parameters screen initiates the cycle to set the beam offset value. Refer to *Appendix A: Tool Probe Setup, on page 8 - 19* for field definitions for this screen.

You must calibrate the laser system before using the light beam for measuring tools. The laser calibration tool or precision dowel used for performing calibration is inserted into the spindle just like any tool.

Laser Tool Calibration Calculations

The calibration tool’s dimensions are determined by using a precision dowel or a laser calibration tool, as shown in the figure below. Use this formula to determine the location on the tool’s diameter to interrupt the beam:

\[
\text{Length} + \left(\frac{\text{Height}}{2}\right) = \text{Point on Diameter to Interrupt Beam}
\]

The software uses the Cal Tool D(iameter), H(eight), and L(ength) fields (shown below as D, H, and L) and the trigger points established in this cycle to determine the Center Beam X or Y values, depending on the Probing Axis.

![Figure 8–3. Typical Laser Probe Calibration Tool](image)

<table>
<thead>
<tr>
<th></th>
<th>Donut</th>
<th>Tip</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Laser Tool Calibration Cycle

Follow these steps to run the Laser Calibration Tool cycle:

1. Access the Tool Setup Probe Parameters screen using this softkey sequence:
   a. Tool Setup softkey
   b. More softkey
   c. Tool Probing softkey
   d. Tool Probe Setup softkey

2. In the Type field, select Laser Probe.

3. Enter values in the Cal Tool D, H, or L fields. The DETERMINE LASER BEAM OFFSET softkey appears when the cursor is in any of these fields.

4. Select the DETERMINE LASER BEAM OFFSET softkey and this sequence occurs:
   a. The Start Cycle button flashes.
   b. Press the Start Cycle button to begin the cycle. The tool moves down in the Z Plane at Setup Fast Feed until the beam is interrupted by the tip of the tool.
   c. The tool moves up slightly at Retract Feed until the beam is uninterrupted. This sequence of slow moves into, then out of, the beam repeats for Repetition number of readings. The average switch point coming out of the beam is recorded.
   d. The tool moves up slightly again, moves over in the X/Y Plane, and down until the laser beam is positioned parallel to the center of the tool’s donut.
   e. The tool moves in the X/Y Plane at Fast Setup Feed toward the beam until the beam is interrupted.
   f. The sequence of slow moves into then out of the beam repeats for Repetition number of readings. The average switch point coming out of the beam is recorded.
   g. The process is repeated from the opposite side of the beam.
   h. The Center Beam X or Y field is updated based on the trigger points established in this cycle, depending on the Probing Axis selection.
      - If Probing Axis is X, the Center Beam X field is updated.
      - If Probing Axis is Y, the Center Beam Y field is updated.
   i. The Beam Offset field is updated based on the trigger points established in this cycle.
The figure below illustrates the tool motion during this cycle:

![Figure 8–4. Typical Laser Probe Calibration Tool Motion](image)

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Z Minus</td>
</tr>
<tr>
<td>2</td>
<td>X/Y Plus</td>
</tr>
<tr>
<td>3</td>
<td>X/Y Minus</td>
</tr>
</tbody>
</table>

*Figure 8–4. Typical Laser Probe Calibration Tool Motion*
Probe Deflection Offset Calibration

Tool and part probe deflection offsets are the difference between the contact point of the probe and the actual receipt of a probe deflection signal. The offsets may vary for each direction of deflection. The switch points are repeatable to one micron or less.

These offsets need to be adjusted during an initial probe installation, a new stylus installation, or for centering or re-centering a stylus. They do not need to be performed each time the control is reset.

**Tool Probe Deflection Offset**

Access the Tool Probe Deflection Offset screen from the Tool Setup screen. Select the Tool Probing softkey, the Tool Probe Setup softkey, and the Tool Probe Deflection Offset softkey. This screen appears:

![Tool Probe Deflection Offsets](image)

**Figure 8-5. Tool Probe Deflection Offsets**

The Reference Tool Diameter field holds the diameter of the tool being probed.

The probe orientation determines the offsets used in the Probe Stylus Position fields. The -Z offset is always used along with +/-X or +/-Y, depending from which direction the probe can deflect.

The following sections describe the two methods for determining Tool Probe Deflection Offsets: Absolute Location, on page 8 - 27 or Reference Tool Touch, on page 8 - 27.
**Absolute Location**

Use an edge finder to determine the absolute location of each edge of the probe stylus.

1. Enter the Reference Tool Diameter.
2. Position the cursor on the desired offset field.
3. Position the reference tool to the correct start position.
4. Press the **Use Probe To Determine Offset** softkey. The **Start** Button flashes.
5. Press the flashing **Start** Button.
   a. The cursor position determines which axis is moved and in what direction.
   b. The deflected position is used to calculate the offsets.
   c. The offset value appears in the current field.
   d. The offsets are saved by the system so they are retained after power to the machine is turned off.
   e. Unused fields contain a 0 value.
   f. The sign of the offset is + for plus axis deflections and - for negative axis deflections.

The **Apply G Code Offset** parameter applies the deflection offsets to G31 commands when conversational and NC probing are used together.

**Reference Tool Touch**

Use a feeler gauge to determine the position where the reference tool touches the top and each side of the probe stylus. Follow the prompts on the screen to know which side of the stylus to use. Follow these steps:

1. Enter a 0 for the Reference Tool Diameter.
2. Position the reference tool in the correct start position. Begin with the top of the stylus.
3. Place the cursor in the -Z field of the Probe Stylus column.
4. Press the **USE PROBE TO DETERMINE OFFSET (F1)** softkey. The **Start** Button flashes.
5. Press the flashing **Start** Button.
   a. The cursor position determines which axis is moved and in what direction.
   b. The deflected position is used to calculate the offsets.
   c. The offset value appears in the -Z field.
   d. The offsets are saved by the system so they are retained after power to the machine is turned off.
   e. Unused fields contain a 0 value.
   f. The sign of the offset is + for plus axis deflections and - for negative axis deflections.

6. Repeat these steps for the other two axis positions (+/- X and +/-Y). Position the reference tool appropriately and put the cursor in the appropriate
Probe Stylus field.

You can manually adjust the deflection offsets to optimize performance. By running a probe cycle on a reference tool you may make slight adjustments to the deflection offsets until the cycle returns with the exact value(s) desired.

The **Apply G Code Offset** parameter applies the deflection offsets to G31 commands when conversational and NC probing are used together.
Appendix B: Tool Probe Calibration

The Tool Probe must be calibrated before probing can be performed.

Probes Calibration must be completed before probing can begin, but they only need to be done one time, unless the probe is relocated or replaced, if the Slow Feed value is changed, or if the tool calibration mode is changed.

Probe Calibration—Absolute Tool Length mode

To calibrate the Tool Probe:

1. Access the Tool Measurement screen (Tool Setup/More/Tool Measurement Settings softkey):
2. Specify Probe in the Device field.
3. Advance to Z Reference field.
4. Place a gauge of known height on table top.
5. With a reference tool in spindle, jog Z-axis down and carefully touch reference tool to top of gauge block.
6. Select the Store Machine Position softkey or press the store position button on remote jog unit.
7. Subtract the reference tool length and the gauge height from the stored machine position, and manually enter this value in the Z Reference field.
   
   For example, if the stored machine position is -300mm, and the reference tool length = 150mm, and the gauge height = 50mm, the Z Reference value is -500mm.

8. Access the Tool Setup screen. Enter the exact tool length for the Reference Tool in the Tool Cal Length field.
9. Select the More softkey, and then the Tool Probing softkey. The Tool Probing screen opens.
10. Select Length in the Cycle field.
11. Select the Calibrate the Tool Probe softkey. The control asks for the reference tool (number) to use for calibrating the probe; enter the number and select OK to calibrate the probe.
12. When calibration is complete, the Tool Measurement screen opens and the Height and Z Location fields contain the measured and calculated values:
Now the probe is calibrated and you can proceed to Set the Probing Parameters—Absolute Tool Length Mode, on page 8 - 2.

**Probe Calibration—Zero Calibration mode**

The tool probe can be calibrated (Z height determined) with a **part probe** or with a **reference tool**.

When tools are probed, the Probe Z value and the stored internal value (from the calibration of the tool probe) are used to calculate **Zero Calibration**. When a new part is placed on the table, a new Probe Z value must be determined.
**Calibrate Tool Probe with Part Probe**

The Tool Probe can be calibrated (Z height determined) with a Part Probe.

When using a part probe, **Part Probe Parameters** and **Deflection Offsets** must be set before proceeding with tool probe calibration. *Part Setup—Part Probe Parameters, on page 7 - 2 and Part Probe Deflection Offset Calibration, on page 7 - 3.*

With the part probe in the spindle:

1. From the Tool Setup screen, select the **More** softkey.
2. Select the **Tool Probing** Softkey.
3. Set the **Rapid Z Position**. Use the remote jog unit to jog the part probe to a point just above the contact point of the tool probe and select the **Set Position** softkey, or type in a value for the Rapid Z Position field. (With the cursor in this field, the **Position Tool Over Probe** softkey can be used to position the part probe over the tool probe before jogging Z to the desired position.)
4. Select the **Calibrate the Tool Probe** softkey.
5. Press the flashing Start Cycle button.

The control activates the part probe and the Z axis moves down at rapid, then at Approach Feed until a probe deflection occurs or Minimum Z is reached. (If the probe reaches Minimum Z prior to deflection, an error message appears. Minimum Z is set in the Part Setup Probe Parameters.)

If the appropriate fields are set correctly, then the two probes will touch (or the part probe breaks the beam on a laser tool probe). The part probe will then retract slightly and touch again at the feed rate specified in the Measurement Feed field. The average deflection position is recorded and saved internally for the Zero Calibration value calculation.

Next you must determine the Probe Z value. See *Determine Probe Z, on page 8 - 9.*

**Calibrate Tool Probe with Reference Tool**

The Tool Probe can be calibrated (Z height determined) with a Reference Tool.

1. From the Tool Setup screen, select the **More** softkey.
2. Select the **Tool Probing** Softkey.
3. Set the **Rapid Z Position**. Use the remote jog unit to jog the reference tool to a point just above the contact point of the tool probe and select the **Set Position** softkey, or type in a value for the Rapid Z Position field. (With the cursor in this field, the **Position Tool Over Probe** softkey can be used to position the reference tool over the tool probe before jogging Z to the desired position.)
4. Select the **Calibrate the Tool Probe** softkey.
5. A prompt appears requesting the Reference Tool number. Enter the appropriate tool number and select OK.
6. Press the flashing Start Cycle button.

The Z axis moves downward at rapid then at Fast Feed until a probe deflection occurs or Minimum Z is reached. (If the probe reaches Minimum Z prior to deflection, an error message appears. Minimum Z is set in the Tool Probing screen.)

The reference tool touches the tool probe, retracts slightly, and touches again at the feedrate specified in the Slow Feed field. The number of slow touches is specified by the Repetitions parameter. The average deflection position is recorded and saved internally for the Zero Calibration calculation.

Next you must determine the Probe Z value. See Determine Probe Z, on page 8 - 9.
Appendix C: Probing Parameter Definitions

Tool Probing Parameter Definitions

Definitions for the fields on the Tool Probing screen are:

- **Rapid Clearance (Absolute Tool Length only)**—the distance above the probe stylus or beam that determines **Rapid Z Position** (based on approximate tool length and probe height in Z calibration).

- **Rapid Z Position**—the tool (in the Z axis) rapids down to this position and then continues downward at the speed specified in the **Fast Feed** field. For Zero Calibration mode, use the remote jog unit or type in a value for the Rapid Z Position field; For Absolute Tool Length mode the value is determined by **Rapid Clearance**.

  With the cursor in this field, the Position Tool Over Probe softkey can be used to position the tool over the probe before jogging Z to the desired position.

- **Spindle Usage**—specifies the direction the spindle should turn during the probe cycle. Choices are: manual (free rotating), oriented, clockwise (CW) or counter clockwise (CCW).

  The default value for a touch tool probe is the reverse of the programmed tool. For example, if the tool is CW, Spindle Usage for a touch tool probe will default to CCW. For a laser tool probe, Spindle Usage value defaults to the programmed tool direction.

- **Spindle Speed**—specifies spindle speed when the Spindle Usage field is set to either CW or CCW. When the operator sets the diameter setting for the tool probe, the Spindle RPM field will be set to a suggested value – this value may be overwritten by the operator. The suggested value for RPM is only done when the tool is first entered.

- **Fast Feed**—specifies the feed rate used when making the initial touch of the probe (prior to measurement touches). When the operator sets the diameter setting for the tool only the first time the tool is entered, the Fast Feed field will be updated with a suggested value—you may overwrite this value.

- **Slow Feed**—specifies the feed rate used when taking measurements after the initial deflection. The default value is 4 mm per minute for a touch probe, 25.4 mm per minute for a laser probe.

- **Tool Cal Length (Absolute Tool Length only)**—the Tool Cal Length value entered in Tool Setup.

- **Min Length Delta (Absolute Tool Length only)**—the distance below probe stylus or beam that the Z axis is allowed to travel; determines **Min Z Position**.

- **Min Z Position**—specifies that lowest position that Z will be allowed to travel during the probe cycle. This parameter sets up a safety zone for each tool. This value must be low enough to allow proper deflection of the probe. This is especially important when checking diameter as the tool must drop down next to the probe stylus.

- **Length Offset X**—used to change the X table position when probing – applies only when probing tool length. Required for tools that have a cutter offset from the center.
• **Length Offset Y**—used to change the Y table position when probing – applies only when probing tool length. Required for tools that have a cutter offset from the center.

• **Z Drop Down Depth**—used only when measuring tool diameter. This parameter (a negative value) indicates the distance to drop down from the surface of the probe or beam. For example, if you wish to measure the diameter of the tool ¼" from the tip, this parameter would be set to −0.25".

• **Spindle Clearance**—used only when measuring tool diameter. Specifies an additional distance to leave between the tool and tool probe when determining the tool diameter. This value can be adjusted to optimize probe cycle time.

• **Multi-Tool Probing**—specifies if multiple tools are to be probed. Yes will include current tool in multi-tool probe cycle.

• **Sister Tool**—specifies the spare tool for tool quality monitoring. See *Tool Quality Monitoring*, on page 8 - 14

Definitions for the softkeys on the Tool Probing screen are:

• **Tool Probe Setup**—opens the Tool Probe Parameters screen for probe setup.

• **Calibrate the Tool Probe**—determine the Z location of the tool probe. This value is retained on power down.

• **Probe Single Tool**—determine the length and/or diameter of a single tool.

• **Probe Multiple Tools**—determine the length and/or diameter of multiple tools.

• **Position Tool Over Probe**—move the tool into position over the probe.

• **Probe Current Tool Now**—determine the length and/or diameter of the current tool in spindle.
TOOL CHANGE OPTIMIZATION

Tool Change Optimization reduces the number of tool changes in Conversational programs. In typical Conversational program execution, each data block is completed (roughing and finishing) before the program moves to the next data block. With Tool Change Optimization, operations are completed sequentially by tool; for example, the program completes all operations that use tool 1 before moving to the operations that use tool 2, and so on.

For example:

A Mill Frame with tool 1 for roughing and tool 2 for finishing, patterned three times, with tool 1 in the spindle:

- Without Tool Change Optimization, each frame in the pattern is rough cut and finish cut before moving to the next frame in the pattern. Sequence of tool changes: 1, 2, 1, 2, 1, 2. Total number of tool changes = 5.
- With Tool Change Optimization enabled, all three frames are rough cut, then all are finish cut. Sequence of tool changes: 1, 2. Total number of tool changes = 1.

Using Tool Change Optimization

To use Tool Change Optimization, the Tool Change Optimization parameter must be set to YES (default is Yes). This parameter is found on the General 2 tab in Program Parameters:

![Figure 9–1. Tool Change Optimization parameter in Program Parameters](image)
In addition, the Tool Change Optimization On block must be inserted ahead of the program blocks to which it applies. Both the parameter and the data block are necessary to activate Tool Change Optimization. The Tool Change Optimization Off block is inserted after the last block in which tool optimization is to occur. This allows flexibility in using Tool Change optimization on select blocks while maintaining normal tool change sequence in other blocks of the program.

⚠️ With Tool Change Optimization on, a data block should not be dependent on the finishing pass of the preceding block to clear out stock (material) prior to that data block. With Tool Change Optimization, the finishing pass has not yet run (because all roughing operations are completed first), and therefore the finish material is not yet removed.

To access the Tool Change Optimization On and Off blocks:

1. From a New Block screen, select the Miscellaneous softkey.
2. Select the More softkey twice.
3. Select the Tool Optimize On softkey. The block opens:

![Figure 9–2. Tool Change Optimization On block](image)

*Figure 9–2. Tool Change Optimization On block*
The Tool Change Optimization On block contains the following fields:

- **Maintain Roughing Sequence**—when set to Yes, all roughing operations are performed in block sequence, including any specified tool changes. When set to No, all roughing operations are performed in tool change sequence. Default is No.

  For Holes, the first operation is treated as a roughing tool and will be executed when other roughing tools are executed, based on the maintain roughing sequence. The other hole operations will always follow the first hole operation. If the hole operation should not be started until a previous block has been finished, then be sure to disable Tool Change Optimization prior to hole operation.

- **Maintain Finishing Sequence**—when set to Yes, all finishing operations are performed in block sequence, including any specified tool changes. When set to No, all finishing operations are performed in tool change sequence. Default is No.

  The Maintain Finishing Sequence parameter does not apply to holes operations.

4. Select the Tool Optimize Off softkey. The Tool Change Optimization Off block is used to turn off the Tool Change Optimization feature for a block or group of blocks within a program, after the Tool Change Optimization On block has been used at an earlier point in the program.
Example

*Figure 9–3. Placement of Tool Change Optimization blocks*

In the sample screen above, tool change optimization occurs for blocks 2-6. The sequence with Tool Change Optimization parameter set to Yes and the Tool Optimization On block Maintain Roughing and Maintain Finishing Sequence set to No is shown below:

<table>
<thead>
<tr>
<th>Block</th>
<th>Type</th>
<th>Rough Tool</th>
<th>Finish Tool</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>TCO On</td>
<td>1</td>
<td>3</td>
</tr>
<tr>
<td>2</td>
<td>Mill Contour</td>
<td>2</td>
<td>4</td>
</tr>
<tr>
<td>3</td>
<td>Mill Frame</td>
<td>1</td>
<td>3</td>
</tr>
<tr>
<td>4</td>
<td>Mill Circle</td>
<td>1</td>
<td>3</td>
</tr>
<tr>
<td>5</td>
<td>Mill Ellipse</td>
<td>1</td>
<td>3</td>
</tr>
<tr>
<td>6</td>
<td>Mill Contour</td>
<td>1</td>
<td>3</td>
</tr>
<tr>
<td>7</td>
<td>TCO Off</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Tool Change Sequence**

- Block 2 (Tool 1)
- Block 3 (Tool 1)
- Block 5 (Tool 1)
- Block 6 (Tool 1)
- Block 4 (Tool 2)
- Block 2 (Tool 3)
- Block 3 (Tool 3)
- Block 5 (Tool 3)
- Block 6 (Tool 3)
- Block 4 (Tool 4)

**Tool Changes = 3**

Note in the sequence that all roughing blocks that use the first roughing tool (Tool 1) are executed in block sequence, before switching to the next roughing tool (Tool 2). After all roughing operations are completed, all finishing operations that use the first finishing tool (Tool 3) are executed in block sequence, followed by the block that uses the next finishing tool (Tool 4). This requires 3 tool changes.

⇒ If a finish tool is not specified in a milling block, the tool listed in the Roughing tab is considered a finish tool and will be grouped with the other finish tools.
When Tool change Optimization is not used (no Tool Optimization On block), the sequence of tool changes looks like this:

**Tool Change Sequence**

<table>
<thead>
<tr>
<th>Block</th>
<th>Type</th>
<th>Rough Tool</th>
<th>Finish Tool</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Mill Contour</td>
<td>1</td>
<td>3</td>
</tr>
<tr>
<td>2</td>
<td>Mill Frame</td>
<td>1</td>
<td>3</td>
</tr>
<tr>
<td>3</td>
<td>Mill Circle</td>
<td>2</td>
<td>4</td>
</tr>
<tr>
<td>4</td>
<td>Mill Ellipse</td>
<td>1</td>
<td>3</td>
</tr>
<tr>
<td>5</td>
<td>Mill Contour</td>
<td>1</td>
<td>3</td>
</tr>
</tbody>
</table>

Block 1 (Tool 1)  
Block 1 (Tool 3)  
Block 2 (Tool 1)  
Block 2 (Tool 3)  
Block 3 (Tool 2)  
Block 3 (Tool 4)  
Block 4 (Tool 1)  
Block 4 (Tool 3)  
Block 5 (Tool 1)  
Block 5 (Tool 3)

**Tool Changes = 9**

Note that tool changes occur in block order, resulting in 9 tool changes.

**Tool Change Review**

Tool changes are reviewed in the Tool Change Review Screen, accessed with the Tool Change Review softkey in the Part Program Tool Review menu. Tools are displayed in the order they are used in the program. All tool changes in a part program are displayed, even if the Tool Change Optimization feature is not used.

![Figure 9–4. Tool Change Review Screen](image)

Data blocks are displayed in the order they are executed. 

Tools are displayed in the order they are used in program.

Operation is listed for each block:
- Roughing
- Finishing
- Hole operation by type

Jump to the data block from this screen in one of the following ways:

1. Select the block in the list and press the Enter console key.
2. Select the block in the list and select the Part Programming softkey.
3. Double-click on the block in the list.
Hole Blocks and Tool Change Optimization

If a hole block is within a group of blocks that use Tool Change Optimization, the hole operations are completed as a group. In addition, the first hole operation’s tool is grouped with the roughing tools of the other blocks and is completed in the sequence specified by the Maintain Roughing Sequence:

<table>
<thead>
<tr>
<th>Block</th>
<th>Type</th>
<th>Rough Tool</th>
<th>Finish Tool</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>TCO On</td>
<td>1</td>
<td>2</td>
</tr>
<tr>
<td>2</td>
<td>Mill Frame</td>
<td>1</td>
<td>4</td>
</tr>
<tr>
<td>3</td>
<td>Mill Frame</td>
<td>3</td>
<td>4</td>
</tr>
<tr>
<td>5</td>
<td>Hole</td>
<td>5 (1st Op)</td>
<td>6 (2nd Op)</td>
</tr>
<tr>
<td>6</td>
<td>Mill Frame</td>
<td>1</td>
<td>2</td>
</tr>
<tr>
<td>7</td>
<td>TCO Off</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Tool Change Sequence</th>
<th>Tool Change Sequence</th>
<th>Tool Change Sequence</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maintain Roughing No</td>
<td>Maintain Roughing Yes</td>
<td>Maintain Finishing Yes</td>
</tr>
</tbody>
</table>

- Block 2 (Tool 1)
- Block 3 (Tool 1)
- Block 6 (Tool 1)
- Block 4 (Tool 3)
- Block 5 (Tool 5)

- Block 2 (Tool 2)
- Block 6 (Tool 2)
- Block 3 (Tool 4)
- Block 4 (Tool 4)

<table>
<thead>
<tr>
<th>Tool Changes = 5</th>
<th>Tool Changes = 7</th>
<th>Tool Changes = 9</th>
</tr>
</thead>
</table>

- All Roughing and first operation tools are grouped by tool number and executed in the order tools appear in program.
- *For multiple tool Hole operations, all of the operations are completed in sequence after the first operation is completed.
- All Finishing tools (but not any hole operations) are grouped by tool number and completed in the order they appear in the program.

- All Roughing and first operation tools are grouped by tool number and executed in order of the data blocks.
- *For multiple tool Hole operations, all of the operations are completed in sequence after the first operation is completed.
- All Finishing tools (but not any hole operations) are grouped by tool number and executed in the order they appear in the program.

- Each block is completed before advancing to the next block.
TOOL FIXTURE (TPS) OPTION

These topics are discussed in this section:

Tool Fixture Overview .......................................................... 10 - 2
Automatic Tool Removal Using TPS ....................................... 10 - 3
Automatic Tool Change Using TPS ........................................ 10 - 4
Bypass TPS in an Automatic Tool Change ............................ 10 - 5
Tool Fixture Overview

The Tool Fixture (TPS) option provides an alternative way to manually move tools to and from the spindle during tool setup or program execution. TPS has two components, a tool fixture and an optikey that enables the option. The fixture is attached to the table, near the front enclosure doors. It is not removable, except for complete removal.

![Figure 10–1. Tool Fixture](image)

TPS is used as part of a tool change sequence from Tool Setup or Manual or Auto mode. Tools are laterally loaded into and unloaded from the tool fixture, and use the same spindle orient position as the ATC arm.

Tools used with TPS are limited by the tool fixture’s height and diameter, as well as ATC weight restrictions. Tools inserted into the spindle with TPS must be removed with TPS. You can convert a tool loaded with TPS into an auto tool, as long as it fits the diameter and weight constraints of the magazine. Once the tool becomes an auto tool, it is no longer tracked as a TPS loaded tool.

You can insert a tool or remove a tool using TPS in Tool Setup, Manual Mode or Auto Mode. This section describes tool loading and unloading using TPS in Tool Setup.
Automatic Tool Removal Using TPS

Follow the instructions below to use TPS to automatically remove a tool from the spindle:

1. Select the Input console key to access the Input screen.
2. Select the Tool Setup softkey.
3. With the cursor in the Tool field, enter the number of the tool you want to change into the spindle.
4. Select the Auto Tool Change console button. The Start Cycle button begins flashing.
6. Make certain the Tool Fixture is empty. Press the Start Cycle button and its stops flashing.
7. A "Remove Tool X from Fixture, Close Doors & Press Start" message appears on the screen.
8. Remove the tool from the fixture, close the enclosure doors and Press the Start Cycle button.
9. The tool entered in step #3 is inserted from the magazine into the spindle.
10. The tool removal is complete.

⇒ You must use Manual Mode if the tool you are removing was the last tool in the spindle.
Automatic Tool Change Using TPS

Follow the instructions below to complete an Auto Tool change using TPS

1. Select the Input console button to access the Input screen.
2. Select the Tool Setup softkey to access the Tool Setup screen.
3. With the cursor in the Tool field, enter the number of the tool that will be inserted into the spindle using TPS.
4. Press the Tool Changer Auto console button. The Start Cycle button begins flashing.
5. Press the Start Cycle button to initiate the tool change.
   a. The Z axis rapids to the home position.
   b. The X and Y axes rapid to the Access position. An "Insert Tool X in Fixture. Press Start" message appears. The Start Cycle button flashes again.
      ⇒ If the Tool in Spindle is 0, a message will appear confirming that the spindle is empty. If the Tool in Spindle is a number other than 0, the tool will be removed from the spindle (automatically put into the magazine or you will be prompted to be removed manually)
6. Insert the tool into the fixture.
7. Close the enclosure doors. Press the Start Cycle button.
   a. The Start Cycle button stops flashing.
   b. The X and Y axes rapid to the Fixture position.
   c. The spindle orients (if it is not already oriented).
   d. The spindle unclamps.
   e. The Z axis rapids to the Clear position (its position before the tool change began) - then moves at a fixed, reduced feedrate to the Fixture position. The spindle clamps the tool.
   f. The Z-axis moves to the Fixture position at a reduced feedrate.
   g. The spindle clamps the tool.
   h. The X and Y axes move to the Clear position (their positions before the tool change began) at a reduced feedrate.
   i. The Z axis rapids to the Tool Change position.
8. The tool change is complete.
     ⇒ A message requesting the tool be removed from the spindle using TPS will be displayed for any tool that was inserted into the spindle with TPS. You may use TPS to remove the tool or manually unclamp the tool to remove it from the spindle.
Bypass TPS in an Automatic Tool Change

Follow the instructions below to complete an Auto Tool change and bypass TPS:

1. Select the Input console button to access the Input screen.
2. Select the Tool Setup softkey to access the Tool Setup screen.
3. With the cursor in the Tool field, enter the number of the tool that will be inserted into the spindle.
4. Press the Tool Changer Auto console button; the Start Cycle button begins flashing. Press the Start Cycle button. If there is a tool in the spindle, remove it before proceeding. If the spindle is empty, the machine moves to the Access position.
5. When prompted to insert the tool into the fixture, manually insert the tool into the spindle. To install a tool using TPS, see Automatic Tool Removal Using TPS, on page 10 - 3.
   a. Press the spindle unclamp button and insert the tool. The prompt changes to "Insert Tool XX in Spindle & Press Start Cycle."
   b. Release the unclamp button. The tool is clamped in the spindle.
6. Close the enclosure doors and press the Start Cycle button.
7. The tool change is complete.
   ⇒ Any tool that is inserted into the spindle manually (bypassing TPS) must be removed manually.
ULTIMOTION

Hurco’s UltiMotion system provides the following benefits:

**Improved Surface Quality**

With Hurco's proprietary smoothing, lookahead and motion control technology, the UltiMotion system provides significantly improved part surface quality. You'll see smoother and shinier surfaces, less vibration, less chatter marks, and higher accuracy.

**Higher Throughput**

Compared with a standard motion system, the UltiMotion system's advanced motion planning and rapid corner feature can provide decreased cycle time with the same or even better surface finish quality. This is especially true for parts with complex geometries or large amounts of repeated tasks, such as drilling or tapping.

**Rapid Corner**

For programs with a lot of consecutive rapid moves, the UltiMotion system doesn't stop between two rapids. Instead, it travels through blended corners at very high speed with only negligible deviation. This saves significant amount of time for repeated tasks, such as drilling and tapping.

**Broader Performance Range for Surface Finish Quality**

The Surface Finish Quality (SFQ) mechanism is Hurco's proprietary technology to allow users to choose better surface finish or higher throughput for their applications via a simple one-parameter control mechanism. Winmax control automatically adjusts internal parameters to achieve either better surface quality (smaller SFQ) or higher throughput (higher SFQ). The UltiMotion system extends the performance range for SFQ to have even better surface quality for your finish pass and even more time savings in the roughing or semi-roughing passes.

**Higher Quality Rigid and Regular Tapping**

The UltiMotion system deploys a coordinated motion control mechanism for regular and rigid tapping. You no longer need to worry if your tap will pass go/no-go tests; the UltiMotion system monitors the spindle angle at all times and controls the Z-axis to track the spindle position.

**Smoother Motion Results in Longer Machine Life**

Rough motion, vibration, and bumping increase mechanical system fatigue and shorten machine life. The UltiMotion system's unique acceleration and jerk control, along with its advanced motion planning, have made it possible to achieve faster, yet smoother motion. You will get a faster machine with extended machine life.
Robust and Consistent Performance Over a Wide Range of Working Conditions

Have you ever had trouble maintaining the equivalent performance with work pieces weighing from several ounces to several thousands of pounds? The UltiMotion system's advance and accurate model compensation will help you to maintain the same level of performance over a wide range of working conditions.

Dynamic Variable Lookahead Algorithm

Hurco's proprietary Lookahead doesn't require a fixed number of blocks lookahead like conventional controls; instead the lookahead varies dynamically depending on the geometry and motion profile, but always guarantees there is enough to make optimized maneuvers.

Smooth and Responsive Handwheel Jogging

Compared with standard system, the UltiMotion system’s hand wheel jogging is optimized to reduce bumping and vibration while providing more responsiveness.

How to Select the Motion System

The UltiMotion option must be installed. To turn it on or off:

1. Select the **Auxiliary** console button.
2. Select the **Utility Screen** softkey.
3. Select the **System Configuration** softkey.
4. Select the **WinMax Configuration** softkey.
5. Select the **Motion Configuration** softkey. The Motion Configuration screen opens and the current motion system is displayed.
6. Select either Standard or UltiMotion in the **Motion System** field. The system must be restarted for the change to take effect.
ULTIMONITOR

The UltiMonitor option adds capability and flexibility to your WinMax Mill operation by providing connection to your Local Area Network (LAN). Using UltiMonitor, you can communicate with other CNCs, and with PCs or file servers connected to your LAN using standard TCP/IP and FTP protocols. UltiMonitor also includes Extended Shop Floor (ESF) for remote machine monitoring and communication.

This section covers the use of the UltiMonitor product. For information about basic system operation, refer to the Getting Started with WinMax Mill manual.

⇒ Customers using UltiNet should refer to this section for product guidance. The UltiNet option provides the same FTP connectivity as UltiMonitor but does not include ESF.

LAN Requirements ................................................................. 12 - 2
Configuring a Network ............................................................ 12 - 4
Using FTP ............................................................................ 12 - 9
Extended Shop Floor (ESF) ....................................................... 12 - 13
LAN Requirements

To use UltiMonitor, you must have a Local Area Network (LAN or “network”) that supports the IEEE 802.3 Ethernet hardware standards. Contact your IT provider for guidance about proper LAN design and setup.

Limitations for UltiMonitor

1. Do NOT connect machines to a domain. When connecting a machine to a network, a workgroup must be used. WinMax software runs on an embedded Windows operating system, and any changes to the system could cause unwanted effects.

2. Installing any physical cards, including wireless devices, is NOT supported. WinMax is designed and configured to operate with the existing system hardware.

3. Use of an Ethernet cable connection to a machine is required.

Glossary of Networking Terms

Following are definitions for networking terms used throughout this section.

- **Client/Server**—This is a term commonly used to describe how computers do work for each other. One computer runs server software; the other computer runs client software. The server software waits for requests from the client in order to perform tasks. Many times the server is a background task that runs all the time. A client is a user application on a PC or some other workstation and usually will not run all the time. A user may begin running client software, do some work, then shut down the client when finished. It is important to note that a single server may handle many clients. It is also possible for a computer to function as both a client and a server simultaneously.

- **Ethernet**—Ethernet is a particular networking technology specification. Ethernet uses CSMA/CD to gain access to the network media (cable). CSMA/CD stands for Carrier Sense Multiple Access with Collision Detection. Ethernet sends its data on a carrier wave. Since there are multiple Ethernet adapters attached to the same cable, they must take turns sending data. When ready to transmit, an adapter checks for the presence of a carrier wave. If no carrier is detected, the adapter begins transmission. If another adapter begins transmitting data at the same time, a collision will occur. The Ethernet hardware senses when a collision occurs. The two adapters involved in the collision will stop, delay for a random length of time, and then try to retransmit the data. This process repeats until the data is successfully sent. Data is sent in the form of packets, which have a limited size, allowing other hosts an opportunity to use the network media.

- **FTP**—This is an acronym for File Transfer Protocol. FTP is implemented using the client/server model and uses TCP/IP as its protocols. FTP software contains a standard set of commands that are used to transfer files and manage directory structures.

- **Host**—Any computer connected to a network can be a host, including a PC running DOS, Windows XP/Vista/7, Windows NT; a UNIX workstation; a
mainframe computer; a CNC control; or any machine with networking capability.

- **IP Address**—An IP (Internet Protocol) address is used to identify a particular host on a network. Each host on a network must have a unique IP address consisting of a 32-bit number usually presented in dotted decimal format; for example, 200.100.150.1. This format divides the address into four single byte values separated by decimal points. On most networks, the first three bytes represent the network and the last byte is the host. Following this practice, the first three bytes are the same for all hosts and the last byte is different for each host. On the Internet, these cryptic addresses usually are not used—most addresses are represented as plain text, which are converted to IP addresses by DNS, or Domain Name Server. The underlying protocol still uses the unique 32-bit IP addresses.

- **TCP/IP**—This is an acronym for the two major protocols of the Internet, Transmission Control Protocol and Internet Protocol. IP data normally reaches its intended destination, but there is no failure notification for the sender if it does not. TCP, however, has an acknowledgment that the message was received. The TCP layer of a protocol stack uses IP to send a message in the form of packets, or bundles of data. Each packet must be acknowledged or it will be sent again. This process repeats until the entire message, which contains multiple packets, is received. TCP is defined as a reliable stream-oriented protocol. The use of TCP/IP is not limited to the Internet. It is commonly used for communication between hosts on LANs (local area networks). It may be used for any application that requires reliable data transfer.
Configuring a Network

UltiMonitor allows machines to be connected on a Local Area Network (LAN or “network”).

Configuring an IP address for your machine

An Internet Protocol (IP) address is a numerical label assigned to each device in a computer network. IP addresses are assigned to a host either at the time of booting, or permanently by fixed configuration of the hardware or software. Persistent configuration is also known as using a static IP address. In contrast, a dynamic IP address occurs when a device’s IP address is assigned with each reboot. All Hurco machines, by default, are configured to use dynamic IP addresses. This is shown in the figure below.

The configuration can be changed to a static IP address, if desired. IP addresses must always be unique to your network; contact your network administrator for the proper IP address values.

To access the LAN and Internet Protocol properties:

1. Select the Windows key (or the Ctrl+Esc keys) on the keyboard to access system settings through the Shell.
2. Expand the left pane of the Shell.
3. Select Control Panel.
4. Select Network Connections.
5. Enter password.
6. Highlight Local Area Connection.
7. Select File/Properties. (See Figure 12-1 a.)
9. Select Properties. The IP address configuration can be changed in the TCP/IP Properties box that opens. (See Figure 12-1 b.)
Figure 12–1. Default LAN and Internet Protocol Properties
Configuring the Computer and Workgroup Names

A computer name is an easy way to locate a networked device when connected to a local network, as an alternative to an IP address. Like IP Addresses, it is very important that these names are unique. All Hurco machines are, by default, configured with unique computer names. Although the default name is unique, it is not descriptive or easy to remember, so many users may wish to change this value. The computer name is changed in System Properties. To access:

1. Select the Windows key (or the Ctrl+Esc keys) on the keyboard to access system settings through the Shell.
2. Expand the left pane of Shell.
3. Select Control Panel.
4. Select System.
5. Enter password.
6. Select Computer Name tab. (See Figure 12-2 a.)
7. Select Change.
8. Enter new name in the Computer name field. Select OK. (See Figure 12-2 b.)

While in this screen, the name of the workgroup or domain to which this machine is a member may also be changed. Hurco DOES NOT support connections to network domains. A workgroup can be used as a logical grouping of networked devices for a particular purpose. It is easiest to place all devices under a particular workgroup but is not required or necessary. By default, all Hurco controls are configured to be on the workgroup named WORKGROUP.
Mapping a Network Drive

A server share is basically a folder on a different computer that is being shared with everyone else. So when you “map a drive,” you are saying that you want access to that folder on your computer also, which is done by mapping it to a letter, for example, F, G, H, etc. To access the Map Network Drive dialog:

1. Select the Windows key (or the Ctrl+Esc keys) on the keyboard to access system settings through the Shell.
2. Expand right pane of Shell.
3. Select Map Drive.
4. Enter password. The Map Network Drive dialog box opens:

![Map Network Drive dialog](image)

**Figure 12–3. Map Network Drive dialog**

To map a network drive:

1. Select an unused Drive letter to represent the shared folder and type in the UNC path in the Folder field. UNC path is a special format for pointing to a folder on another computer. The format is `\computer name\folder name`.

   If you're not sure what the name of the folder is, you can select Browse... to find the computer that way. Select Entire Network, then Microsoft Windows Network and expand the workgroup that your computer is in.

2. Click Reconnect at logon to make the connection permanent, which means the drive will still be mapped even after you restart the computer.

   ![Reconnect at logon icon](image)

If the drive is password protected, use the “Connect using a different user name” link in the mapping window to configure the mapping with the user name and password.
Using FTP

FTP (File Transfer Protocol) is a method of transferring files from one computer to another, using the Internet. For Hurco machining and turning centers, FTP is required for transferring programs between two or more machines. FTP can also be used to transfer programs between a computer and a machine.

FTP Server Settings

To access the FTP Server Settings screen:

1. Press the Auxiliary console key.
2. Select the Utilities softkey to access the Utilities screen and softkey menu.
3. Select the User Preferences softkey.
4. Select the More softkey.
5. Select the FTP Server Settings softkey. The screen is displayed:

![FTP Server Settings screen](image)

**Figure 12–4. FTP Server Settings screen**

FTP Server Settings Fields

These fields are available on the FTP Server Settings screen:

- **Enable FTP Server**—select Yes to enable or No to disable the FTP server.
- **FTP Server Port**—enter a number for the FTP server port. The default is 21.
- **Maximum Idle Time (mins)**—enter the number of idle time minutes before the server disconnects. The default is 0 with no time out.
- **User 1-4 Name**—enter each user’s logon name. Up to 4 users can logon.
- **User 1-4 Password**—enter each user’s logon password. Up to 4 users can logon.
If the logon name is "anonymous," the password is not required.

- **User 1-4 Path**—enter each user’s root path. Up to 4 users can logon.

**FTP Server Settings Softkeys**

These softkeys are available on the FTP Server Settings screen:

- **Display WinMax IP Address**—displays the IP address list in a pop-up window. Select OK to close the window.

  An IP (Internet Protocol) address is used to identify a particular host on a network. Each host on a network must have a unique IP address consisting of a 32-bit number usually presented in dotted decimal format; for example, 200.10.150.1. This format divides the address into four single byte values separated by decimal points.

  On most networks, the first three bytes represent the network and the last byte is the host. Following this practice, the first three bytes are the same for all hosts and the last byte is different for each host.

  On the Internet, these cryptic addresses usually are not used—most addresses are represented as plain text, which are converted to IP addresses by DNS, or Domain Name Server. The underlying protocol still uses the unique 32-bit IP addresses.

- **Change FTP Root Drive**—displays a pop-up window to browse for a folder. Select the appropriate root drive and select OK to close the window. Select Cancel to stop browsing without changing the root drive.

- **Disconnect All Users**—select Yes in the pop-up window to disconnect all current FTP users; select No to cancel the disconnect operation.
FTP Manager

To manage FTP connections:

1. Press the Input console key.
2. Select the Program Manager softkey.
3. Select the FTP Manager softkey to open the **FTP Host List** screen. From here, you can connect to or disconnect from, add, edit, and delete FTP servers which are identified on this list:

![FTP Host List](image1)

**Figure 12–5. FTP Host List screen**

To access the FTP Host Properties screen to add a host to the FTP Host List screen:

1. Select the Add Host softkey. The FTP Host Properties screen appears.

![FTP Host Properties](image2)

**Figure 12–6. FTP Host Properties screen**

2. Enter an Alias name to appear on the FTP Host List screen for this host.
3. Enter an IP Address for connecting to the host.

4. Specify Yes or No for Automatic Login:
   - If set to No, the user is required to enter the username and password at the time of connection.
   - If set to Yes, the User Name and Password fields appear. The username and password are stored for automatic connection when the host is selected.

5. Enter the Default Remote Directory to be opened, or leave this field blank for the FTP server’s root directory.

6. Select either 8.3 DOS or LONG for the Filename Format field.
   - 8.3—file names with eight characters before the period (.) and three characters for an extension after the period are allowed.
   - Long—the complete path to the file, including the drive letter, server name, folder path, and file name and extension can contain up to 255 characters – however, it will be truncated to the 8.3 format. For example, LongFilename.txt will be truncated to LongFil_.txt

7. Select the Apply softkey to add this host to the FTP Host List screen.

From the FTP Host List screen,
   - Select the Connect softkey to connect to a host.
   - Select Disconnect to disconnect from a host.
   - Select Delete Host to remove a host from the list.
   - Select the Exit softkey to return to the Project Manager screen.

Select the File Manager softkey to access a list of system directories and filenames connected with UltiMonitor.

After editing, files may be saved directly to the remote location:

1. Select the Save or Save As softkey in Program Manager.
2. Select the FTP Manager softkey. The screen will switch to the remote File Manager, if connected, or to the login screen if not connected.
3. When connected, select the appropriate remote directory and save the file.
Extended Shop Floor (ESF)

Hurco’s Extended Shop Floor provides a web-based access point for a machine shop owner to communicate and view machine data using a web browser. Additionally, ESF is a distribution point for information about Hurco’s machine tools and it supports the remote monitoring and diagnosis of the machine. By default, an ESF connection is made to Hurco servers when an internet connection is present.

Here are the ESF connection states:

<table>
<thead>
<tr>
<th>ESF State</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1" alt="Icon" /></td>
<td>The ESF Machine Service is not running. Clicking this icon will turn the ESF Machine Service on.</td>
</tr>
<tr>
<td><img src="image2" alt="Icon" /></td>
<td>Please wait while the ESF Machine Service is starting and attempting to establish a connection with the ESF Server.</td>
</tr>
<tr>
<td><img src="image3" alt="Icon" /></td>
<td>The ESF Machine Service is running and communicating with the ESF Server. Clicking this icon will turn the ESF Machine Service off.</td>
</tr>
<tr>
<td><img src="image4" alt="Icon" /></td>
<td>The ESF Machine Service is running but is unable to communicate with the ESF Server. There may be a problem with the network connection. Clicking this icon will turn the ESF Machine Service off.</td>
</tr>
</tbody>
</table>

For more information refer to the *ESF User’s Guide* found at the ESF website, https://esf.hurco.com; a user account with password is required to access ESF.
ULTIPOCKETS OPTION

The UltiPocket™ programming option adds special milling routines for machining pocket boundaries with islands. This option provides complete clean out of odd-shaped pockets without cutting the islands programmed inside the boundary. The software automatically calculates the tool path around islands eliminating the long task of plotting these shapes. Islands may also be rotated, scaled, and repeated.

The following drawing shows an inward spiral boundary with three differently shaped islands.

The pocket feature is available for any of these standard WinMax program data blocks, Mill Contour, Mill Frame, Mill Circle, or Patterns.

To use the UltiPocket feature, first establish the cutter compensation parameter and then program any UltiPocket data blocks.

- Refer to Milling Parameters.
- Select either Insert Arc or Insert Line for the Cutter Comp Parameter field. If this field does not appear on the Milling Parameters screen, the UltiPocket option has not been installed on the control.

These topics are discussed in this section:

- Pocket Boundary .............................................................. 13 - 2
- Helical Plunge with UltiPocket Option ................................. 13 - 4
- Helical Plunge Using Operator Specify Pocket Start ............... 13 - 5
Pocket Boundary

The Pocket Boundary is the outside frame of the part. The basic philosophy of the UltiPocket option is to program the boundary and then tell the system which pockets or islands to avoid within that boundary. This approach eliminates complex calculations and shortens the part programming process.

There are two types of Pocket Boundaries: Spiral Outward (no islands) and Spiral Inward.

Spiral Outward - No Islands

This selection is only for Circle and Frame data blocks without islands. When this routine is selected, the tool begins from the center region of the part outward to pocket the entire programmed boundary. This operation is, therefore, the same as the standard WinMax Pocket selection. With this selection the cutter overlap is controlled by the Pocket Overlap value on the Milling Parameters screen, not the Pocket Overlap field on the Mill Circle and Mill Frame screens.

Spiral Inward

This selection cuts in from the outside of the defined boundary avoiding the defined islands. When this routine is selected, the tool enters the part and begins following a path formed by offsetting the boundary one-half the tool radius, plus the pocket overlap.

To control the percentage of overlap during cutting, enter a value in the Pocket Overlap field on the Mill Circle or Mill Frame screen. After the first pass, the tool follows a path produced by offsetting the boundary by the tool radius, plus the pocket overlap for each pass while avoiding islands inside the boundary.

After pocketing the boundary, the tool then cuts around the inside of the boundary and the outside of each island, using the selected blend offset and the programmed tool radius.

Programming Islands

After programming the mill data block for a boundary, an island can be defined by creating a Pocket Island data block. As many islands as desired may be defined (subject to computer memory on the control), but all must fit within the defined boundary and should allow the tool to completely define the island.

The island data block can be a Mill Frame, Circle, or Contour (provided it is a closed contour). The Pocket Island data blocks use the standard milling values from the boundary data block and do not display these parameters on the island programming screen. The pocket overlap percentage was also defined in the boundary data block.

Mill Contours
To create a Mill Contour data block using the UltiPocket option, set up the operation in the start segment (segment zero). As in standard WinMax milling, an UltiPocket Mill Contour block consists of segments beginning with segment zero. With the cursor in the MILLING TYPE field, select one of the Pocket options in the Start segment to indicate whether this block is the boundary of the part or one of the islands within the boundary. The boundary must occur first in the program.

The segments after the Start segment are programmed in the same manner as standard milling lines and arcs. Automatic calculation of unknown points is available for these data blocks.

**Mill Frame**

The Mill Frame data block is often used to create the part boundary.

This block is programmed in the same manner as the standard WinMax Mill Frame, with the addition of the Pocket Overlap percentage.

**Mill Circle**

The Mill Circle data block is used for both boundaries and islands. It is similar to the standard Mill Circle data block except that if this block is used to create an island, it uses the tool from the boundary data block.

**Pattern**

Pattern data blocks can be inserted to rotate, scale, or repeat islands. Only Pattern data blocks can be programmed between a boundary data block and an island. As many islands as desired may be defined (subject to available memory), but all must fit within the defined boundary.
Helical Plunge with UltiPocket Option

The Helical Plunge option is used with the UltiPocket option to define the plunging location when inward pocketing. Islands created using inward pocketing can influence plunge locations. A plunge location is determined that will not interfere with pocket islands.

The plunging location in outward pocketing is the same as a straight plunge. Outward pocketing is used only for mill frame, mill circle, and ellipse pocket boundaries that do not have pocket islands. Since islands are not present with outward pocketing, the plunging location is in the middle of the mill frame, mill circle and ellipse.

When the Helical Plunge option is installed, two additional pocketing-related fields appear on the Milling Parameters screen. These fields are Operator Specify Pocket Start and Inward Pocket Plunge Near Center.

The Operator Specify Pocket Start field takes precedence over the Inward Pocket Plunge Near Center field. If the Operator Specify Pocket Start field is set to Yes, the value of the Inward Pocket Plunge Near Center field is ignored.

When the Operator Specify Pocket Start field is set to No, the value of the Inward Pocket Plunge Near Center field is checked. If the Inward Pocket Plunge Near Center field is set to Yes, a starting location will be determined near the center of the pocket. If both fields are set to No, the default starting position will be used for the pocket.
Helical Plunge Using Operator Specify Pocket Start

To use the Operator Specify Pocket Start function:

1. Set the Type field to Pocket Boundary on the Mill Contour, Mill Frame or Mill Circle screen.
2. Set the Pocket Type field to Inward on the Mill Contour, Mill Frame or Mill Circle screen.
3. Set the Operator Specify Pocket Start field to Yes on the Milling Parameters screen.
4. The Pocket X Start and Pocket Y Start fields appear on the Mill Contour, Mill Frame, or Mill Circle screen. The operator identifies the pocket plunge location with the Pocket X Start and Pocket Y Start fields.

The Pocket X Start and Pocket Y Start fields define the centerline of the plunge path. This is the location where the helix will be centered. If a straight plunge is selected, the straight plunge will occur at the location.

The following figure shows the Pocket X Start and Pocket Y Start fields on the Mill Frame screen. See the "Helical Plunge Option" chapter for an example using Pocket Start X and Pocket Start Y fields.

⇒ For non-rotary blocks, Pocket X Start and Pocket Y Start define the XY starting location. For rotary blocks, these fields define the XA starting location.

An error message will display if the values for Pocket X Start or Pocket Y Start interfere with the pocket island or pocket boundary.
RECORD OF CHANGES

04-0116-415 rB, Nov 2012
   Revised by: H.Arle
   Approved by: D.Skrzypczak, Nov 2012

Changes
   • Updates to reflect software changes.

04-0116-415 rA, Aug 2012
   Revised by: H.Arle
   Approved by: D.Skrzypczak, Aug 2012

Changes
   • Updates to reflect software changes, including:
     • Tool Diameter or Diameter Wear stored with Tool Probing

04-0116-410 rL, July 2012
   Revised by: H.Arle
   Approved by: D.Skrzypczak, July 2012

Changes
   • Updates to reflect software changes, including:
     • several updates to Tool Probing, including new Tool Probe Cycle defaults screen.

04-0116-410 rK, June 2012
   Revised by: H.Arle
   Approved by: D.Skrzypczak, June 2012
Changes

- Updates to reflect software changes.
• Updates to reflect software changes.

Changes

• Tool Probing chapter updates.
  • Other updates to reflect software changes.

Changes

• Preliminary Tool Probing added.
  • Other updates to reflect software changes.

Changes

• Updates to reflect software changes.
Changes

- Updated Probing section (preliminary).
- Updated to reflect software changes.

Changes

- Added Post Processor section.
- Updated to reflect software changes.

Changes

- Updated to reflect software changes.

Changes

- Added UltiMotion.
- Updated to reflect software changes from v7.3.
Changes

- Added Custom Drill.
- Added Mill Polygon.
- Added HD3 Serial Numbeing Lettering.
- Added Contour Arc content.
- Added Lettering Reference Locations content.
- Added Mill Slot.
- Added Thread Mill.
- Updated to reflect software changes from v7.3.
INDEX

Numerics
3D Mold 1 - 1
Blend Arc 1 - 12
Contour 1 - 9
Line 1 - 10
Parameters 1 - 2

A
A Angle field
  Rotary Lines and Arcs Start Segment 5 - 14
  Rotary Position block 5 - 11
A Center field
  Rotary Circle block 5 - 16
  Rotary Lines and Arcs Arc Segment 5 - 15
  Rotary Lines and Arcs Blend Arc Segment 5 - 15
A Corner field
  Rotary Frame block 5 - 16
A Distance field
  Rotary Loop block 5 - 17, 5 - 23
A End field
  Rotary Lines and Arc Blend Arc Segment 5 - 15
  Rotary Lines and Arcs Arc Segment 5 - 15
  Rotary Lines and Arcs Line Segment 5 - 14
A Length field
  Rotary Frame block 5 - 16
A Start field
  Rotary Lines and Arcs Arc Segment 5 - 15
  Rotary Lines and Arcs Blend Arc Segment 5 - 15
  Rotary Lines and Arcs Line Segment 5 - 14
A-axis rotation 5 - 2
  absolute location, tool probing 8 - 27
ACCEPT POSITION AS PART ZERO (F1) softkey
  Cylinder cycle 7 - 16
  Edge cycle 7 - 29, 7 - 33
  Hole or Circle Pocket cycle 7 - 32
  Part Zero storage 7 - 15
  Plane Intersection cycle 7 - 19
  Rectangular Pocket cycle 7 - 25
  Rectangular Solid cycle 7 - 22
  Accept softkey, DXF 2 - 3, 2 - 7
ACCEPT X/Y SKEW ANGLE (F1) softkey
  Cylinder Skew cycle 7 - 39
  Edge Skew cycle 7 - 41
  Hole Skew cycle 7 - 37
  Rectangular Pocket Skew cycle 7 - 40
  Rectangular Solid Skew cycle 7 - 38
All Off, DXF 2 - 9
All On, DXF 2 - 9
Angle Center field 5 - 38
  Arc Segment
    Universal Rotary 5 - 36
    Rotary Polygon block 5 - 43
    Rotary Slot Geometry tab 5 - 42
Angle Distance field 5 - 46
Angle End field 5 - 36
  Rotary Slot Geometry tab 5 - 42
Angle Length field 5 - 38
Angle Start field 5 - 36, 5 - 38
  Rotary Slot Start tab 5 - 41
Approach Feed field, part setup probe parameters 7 - 3
Arc Radius field 5 - 30
Autochain
  Contours, DXF 2 - 5
  DXF 2 - 3
Automatic mode 7 - 42
automatic part probing cycles 7 - 42
automatic part setup data block execution 7 - 45
Automatic Tool Change Using TPS 10 - 4
Automatic Tool Monitoring field 8 - 18
Automatic Tool Monitoring field, Probing Parameters screen 8 - 18
Automatic Tool Removal Using TPS 10 - 3
Axis Center field 5 - 38
  Rotary Polygon block 5 - 43
  Rotary Slot Geometry tab 5 - 42
Axis Configuration 5 - 4
Axis Distance field 5 - 46
Axis End field 5 - 36
  Rotary Slot Start tab 5 - 41
Axis of Rotation 5 - 48
Axis Start field 5 - 36, 5 - 38
  Rotary Slot Start tab 5 - 41
axis, touch tool stylus width 8 - 20, 8 - 21

B
B Distance field 5 - 17, 5 - 26
Ball-Nosed End Mill 1 - 15
  on a Contour 1 - 13
B-axis rotation 5 - 2
Beam Offset, laser probe 8 - 22
Bidirectional field
  3D Mold Parameters 1 - 6
Block field
  3D Mold Arc 1 - 11
  3D Mold Blend Arc 1 - 12
  3D Mold Contour 1 - 9
  3D Mold Line 1 - 10
  3D Mold Parameters 1 - 2
  Diamond data block 4 - 5
  Hexagon data block 4 - 8
  Mill triangle data block 4 - 2
Breakage Tolerance field, Tool Monitoring screen 8 - 15
Breakage Tolerance, Tool Breakage Detection 8 - 16
Build DB softkey 2 - 3
Bypass TPS in an Automatic Tool Change 10 - 5

C
C Angle field 5 - 20, 5 - 29
C Center field 5 - 21
  Arc segment
    AC Configuration 5 - 21
    BC configuration 5 - 30
  Blend Arc segment
    BC Configuration 5 - 30
  Circle block
    AC Configuration 5 - 22
    BC Configuration 5 - 31
C Corner field 5 - 22, 5 - 31
C Distance field 5 - 23, 5 - 32
C End field 5 - 20, 5 - 21, 5 - 29, 5 - 30
C End field, Arc Segment, AC configuration 5 - 20
C End field, Arc segment, BC configuration 5 - 29
C Length field 5 - 22
C Length field, Rotary Frame, BC configuration 5 - 31
C Start field 5 - 20, 5 - 21, 5 - 29, 5 - 30
  Rotary Lines and Arcs Arc Segment 5 - 21, 5 - 30
calculate X and Z-axis positions after a tilt-axis move, tilt-axis program, WinMax Mill 5 - 49
CALIBRATE THE TOOL PROBE (F2) softkey, Tool Setup screen 8 - 34
Calibrate the Tool Probe field 8 - 34
Calibrate the Tool Probe, Tool Setup
  Probing Parameters, Zero Calibration mode 8 - 31
  calibration laser tool 8 - 23
  part probe 7 - 5
C-axis rotation 5 - 2
Center Beam X, laser tool probe only 8 - 21
Center Beam Y, laser tool probe only 8 - 21
Center X field
  Cylinder cycle 7 - 21
  Hole or Circle Pocket cycle 7 - 24
  Rectangular Pocket cycle 7 - 27
  Rectangular Solid cycle 7 - 18
  Ring Gauge Deflection Offset 7 - 6
Center X, DXF 2 - 8
Center Y field
  Cylinder cycle 7 - 18
  Hole or Circle Pocket cycle 7 - 27
  Rectangular Pocket cycle 7 - 24
  Rectangular Solid cycle 7 - 6
  Ring Gauge Deflection Offset 7 - 21
Center Y, DXF 2 - 8
Centerline Y field, 3D Mold Parameters 1 - 3
Centerline Z field, 3D Mold Parameters 1 - 3
CHANGE FTP ROOT DRIVE F5 softkey
  FTP Server Settings screen 12 - 10
Circular Passes field, Part Setup screen 7 - 4
Client definition 12 - 2
configuration of Hurco machining centers 5 - 3
Contact Point X, touch tool probe only 8 - 19
Contact Point Y, touch tool probe only 8 - 19
Conversational part probing cycles 7 - 9
Corner Radius field 5 - 16, 5 - 22, 5 - 31, 5 - 39
  Mill Polygon block 5 - 44
Corner X field, Plane Intersection cycle 7 - 31
Corner Y field, Plane Intersection cycle 7 - 31
Crash prevention while probing 7 - 3
create a rotary-axis or tilt-axis program from an existing Conversational program 5 - 4
Cut Direction field
  3D Mold Parameters 1 - 5
cutter inserts 4 - 1
Cycle field in Probing 8 - 9
Cycle field, Tool Setup
  Probing Parameters, Absolute Tool Length mode 8 - 3
Cylinder
  probing cycle 7 - 20
  probing part setup fields 7 - 21
  skew probing cycle 7 - 39
Cylinder cycle 7 - 20
Cylinder Radius field 5 - 13
Cylinder Skew Probing cycle 7 - 39

D
Datum Z field, Ring Gauge Deflection Offset 7 - 6
Default Order
  softkey 2 - 3, 2 - 4
Default Radius
  DXF 2 - 3
Deflection offset calibration
  probing 7 - 5, 8 - 26
Deflection Offset column, Reference Block
  Deflection Offset 7 - 7
Delete softkey
  DXF 2 - 8
DETERMINE LASER BEAM OFFSET (F1) softkey, Tool Setup screen 8 - 23
DETERMINE LASER BEAM OFFSET softkey,
  Tool Setup screen 8 - 23
Diameter
  Cal Tool D, laser probe, Tool Setup
  screen 8 - 22
  stylus 7 - 3
  tool probing sequence, single tool 8 - 6,
  8 - 12
Diameter field
  Cylinder cycle 7 - 21
  Hole or Circle Pocket cycle 7 - 18
Diameter Tolerance field, Tool Wear Detection screen 8 - 17
Diameter Wear
  Tool Probing 8 - 6, 8 - 12
Diamond Data Block 4 - 4
Direction field 5 - 14, 5 - 15, 5 - 20, 5 - 21,
  5 - 29, 5 - 30, 5 - 36, 5 - 37
  3D Mold Arc 1 - 11
  3D Mold Blend Arc 1 - 12
  DXF 2 - 8
  Mill Slot Geometry tab 5 - 42
  Tool Breakage Detection 8 - 16
DISCONNECT ALL USERS F6 softkey
  FTP Server Settings screen 12 - 10
Display Geometry, DXF 2 - 5
DISPLAY MACHINE IP ADDRESS F4 softkey,
  FTP Server Settings screen 12 - 10
DO NOT ACCEPT (F2) softkey
  Cylinder cycle 7 - 19
  Cylinder Skew cycle 7 - 25
  Edge cycle 7 - 39
  Edge Skew cycle 7 - 38
  Hole or Circle Pocket cycle 7 - 37
  Plane Intersection cycle 7 - 29
  Rectangular Pocket cycle 7 - 40
  Rectangular Pocket Skew cycle 7 - 41
  Rectangular Solid cycle 7 - 16
  Rectangular Solid Skew cycle 7 - 22
Draw 2D Contour 1 - 2
Draw console key 2 - 2
DXF
  Editor 2 - 2
  Layers 2 - 9
  DXF Edit Modify
    Arc 2 - 8
    Line 2 - 8
    Point 2 - 8
  DXF Option 2 - 1

E
Edge (X, Y, or Z) field
  Edge Skew cycle 7 - 37
  Draw 2D Contour 1 - 2
  Draw console key 2 - 2
  End Angle field, 3D Mold Parameters 1 - 4
  End Cap field
    Mill Slot Caps tab 5 - 42
  Endpoint Tolerance, DXF 2 - 5
  Endpoint1 X, DXF 2 - 8
  Endpoint1 Y, DXF 2 - 8
  Endpoint2 X, DXF 2 - 8
  Endpoint2 Y, DXF 2 - 8
  Ethernet
    definition 12 - 2
  Exit/Cancel
    DXF 2 - 3
  Explode PCurve softkey, DXF 2 - 8
  Extend softkey, DXF 2 - 6

F
Fast Feed
  Tool Probing 8 - 5, 8 - 11
  Fast Feed, Tool Setup
    Probing Parameters, Absolute Tool
      Length mode 8 - 4, 8 - 10
  Fast feed, Tool Setup
    Probing Parameters, Zero Calibration
      mode 8 - 33
Fast feedrate, Partl Setup 7 - 3
Fast Start Feed, Probing Part Setup screen 7 - 3
Feedrate
Part probing 7 - 11
Part Setup 7 - 3
fields
Part Setup, rotary-axis program, WinMax Mill 5 - 8
File Transfer Protocol 12 - 9
File transfer protocol definition 12 - 2
Finish Diameter field 3D Mold Parameters 1 - 8
Finish Plunge Helix Radius field, Helical Plunge Milling Parameters 3 - 2
Finish Plunge Ramp Slope field, Helical Plunge Milling Parameters 3 - 2
Finish Plunge Type field, Helical Plunge Milling Parameters 3 - 2
Finish Step Size field, 3D Mold Parameters 1 - 7
Finish Tool field 3D Mold Parameters 1 - 2
Finish Tool Type field, 3D Mold Parameters 1 - 8
Fit to View softkey DXF 2 - 5
Flat End Mill 1 - 13 on a Contour 1 - 13
Frame Radius 2 - 5
FTP definition 12 - 2
FTP Host 12 - 12
FTP Host List screen 12 - 11
FTP Host Properties 12 - 11
FTP Host Properties screen 12 - 11
FTP MANAGER F8 softkey Project Manager screen 12 - 11
FTP SERVER PORT field
FTP Server Settings screen 12 - 9
FTP Server Settings 12 - 9

G
glossary 12 - 2

H
Height, Cal Tool H, laser probe 8 - 22
Helical Plunge (Inside/Outside) for Mill Frames, Mill Circles and Ellipses 3 - 4
Helical Plunge in the Center of a Pocket 3 - 4
Helical Plunge Milling Parameter Fields 3 - 2
Helical Plunge of Mill Frame Inside with No Pecking and Blend Offset 3 - 5
Helical Plunge Option 3 - 1
Helical Plunge Using Operator Specify Pocket Start 13 - 5
Helical Plunge with 3-D Part Programming Option 3 - 8
Helical Plunge with Lines and Arcs 3 - 8
Helical Plunge with Operator Specified Location 3 - 4
Helical Plunge with Outward Pocketing 3 - 5
Helical Plunge with UltiPocket 3 - 4, 13 - 4
Helical Plunging of Mill Frame Inside with Pecking and Straight Plunge Finish Pass and Blend Offset 3 - 6
Helix thread program example, WinMax Mill 5 - 52
Hexagon Data Block 4 - 7
HMX 5 - 26
Hole Diameter 2 - 5
Hole or Circle Pocktcycle probing part setup fields 7 - 18
HOLE OR CIRCLE POCKET (F2) softkey 7 - 38
Hole or Circle Pocket cycle 7 - 17 skew probing cycle 7 - 38
Hole or Circle Pocket Skew Probing cycle 7 - 38
Holes softkey DXF 2 - 3
Host definition 12 - 2
HTX 5 - 26

I
icons - xiii
INCR Retract, laser probe tool setup 8 - 22
INCR Retract, probe part setup 7 - 4
INCR Retract, touch probe tool setup 8 - 20
Increased Radius 1 - 3
INIT Retract, laser probe tool setup 8 - 22
INIT Retract, probe part setup 7 - 4
INIT Retract, touch probe tool setup 8 - 20
Inner Diameter, Mill Polygon 5 - 43
Insert Pockets 4 - 1
inspection, part 7 - 46
Intersect softkey 2 - 3, 2 - 4
IP (Internet Protocol) definition 12 - 10
IP address definition 12 - 3
Island, Pocket 13 - 2
Join softkey, DXF 2-7
Joined Lines and Arcs 2-7

LAN Requirements 12-2
laser calibration tool 8-23
Laser Calibration Tool Probe cycle 8-24
Laser Probe 8-19, 8-21
laser probe 8-6, 8-11
Beam Offset 8-22
diameter, Cal Tool D 8-22
height, Cal Tool H 8-22
length, Cal Tool L 8-22
X & Y center beam location 8-21
LASER PROBE (F3) softkey, Tool Setup screen 8-21
laser tool calibration 8-23
laser tool probe parameters 8-21
length
Cal Tool L, laser probe 8-22
Length (X) field
Rectangular Pocket cycle 7-24
Rectangular Solid cycle 7-27
Length (Y) field
Rectangular Pocket cycle 7-24
Rectangular Solid cycle 7-27
Length field
Mill Slot Geometry tab 5-42
Length Offset X, Tool Setup 8-33
Length Tolerance field, Tool Wear Detection 8-17
Length, DXF 2-8

machining center configuration 5-3
Maintain Finishing Sequence field
Tool Change Optimization 9-3
Maintain Roughing Sequence field
Tool Change Optimization 9-3
Max Spread field, Part Setup screen 7-4,
8-20, 8-22
Max Spread field, Tool Setup screen 7-4
MAXIMUM IDLE TIME (MINS) field
FTP Server Settings screen 12-9
Measurement Feed field, Part Setup screen 7-3
Measurement feedrate, Part Setup screen 7-3
Mill Circle 13-3
Mill Contours 13-2
Mill Feed field
3D Mold Parameters 1-8

Diamond data block 4-6
Hexagon data block 4-8
Mill triangle data block 4-3
Mill Frame 13-3
Mill Hexagon Data Block 4-7
Mill Plunge Helix Radius field, Helical Plunge Milling Parameters 3-2
Mill Plunge Ramp Slope field, Helical Plunge Milling Parameters 3-2
Mill Plunge Type field, Helical Plunge Milling Parameters 3-2
Mill Triangle Data Block 4-2
Milling softkey
DXF 2-3
Milling Type field
Diamond data block 4-5
Hexagon data block 4-8
Mill triangle data block 4-2
Min Length Delta, Tool Setup
Probing Parameters, Absolute Tool Length mode 8-4
Min Length Delta, Tool Setup
Probing Parameters, Zero Calibration mode 8-33
Min Z field
3D Mold Parameters 1-7
Min Z Position
Tool Probing 8-5, 8-11
Min Z Position, Tool Setup
Probing Parameters, Zero Calibration mode 8-9, 8-33
Minimum Z field, Part Setup screen 7-3
Modify softkey, DXF 2-7
Monitor Motion, probe part setup 7-5
Motion System 11-2
Multi Tool Probing
Tool Probing 8-7, 8-13
Multi Tool Probing, Tool Setup
Probing Parameters, Absolute Tool Length mode 8-3
Multi Tool Probing, Tool Setup
Probing Parameters, Zero Calibration mode 8-9
Multi-tool probing, Tool Setup
Probing Parameters, Zero Calibration mode 8-34

nested Pattern Loop data blocks, rotary-axis program 5-17
nested Pattern Loop data blocks, rotary-axis program, WinMax Mill 5-23, 5-27, 5-32, 5-46
networking terms 12-2
Number field 5 - 17, 5 - 23, 5 - 26, 5 - 32, 5 - 46
Number Of Sides field
Mill Polygon block 5 - 43

O
Offset 1 field, Plane Intersection cycle 7 - 31
Offset 2 field, Plane Intersection cycle 7 - 31
Operator Specify Pocket Start field, Helical Plunge Milling Parameters 3 - 2
Orientation Angle field
Mill Polygon block 5 - 44
Orientation field
Diamond data block 4 - 5
Hexagon data block 4 - 8
Mill triangle data block 4 - 2
Outer Diameter, Mill Polygon 5 - 43

P
Pan softkey
DXF 2 - 5
Parameters
part setup 7 - 3
tool setup 8 - 19, 8 - 21
parameters
laser tool probe 8 - 21
part probing 7 - 2
probing option 7 - 3, 7 - 10
touch tool probe 8 - 19
Part Inspection
cycles 7 - 46
fields 7 - 47
files 7 - 48
programming 7 - 48
Part inspection 7 - 46
PART INSPECTION (F5) softkey, New Block screen 7 - 47
Part Inspection cycles 7 - 46
Part Inspection Probe.txt, sample file 7 - 49
part probe
calibration 7 - 5
cycles, accessing 7 - 13
deflection 7 - 10
deflection offset calibration 7 - 5
reference block 7 - 7
ring gauge 7 - 5
Part Probe Cycle field
Cylinder cycle 7 - 21
Edge cycle 7 - 18
Hole or Circle Pocket cycle 7 - 24
Rectangular Pocket cycle 7 - 27
Rectangular Solid cycle 7 - 15
part probe cycles
selecting type 7 - 9
PART PROBE DEFLECTION OFFSETS (F7) softkey, Part Setup screen 7 - 5, 7 - 6, 7 - 7
part probe parameters
accessing 7 - 2
Part Setup screen Probing menu 7 - 10
PART PROBE PARAMETERS softkey, Part Setup screen Probing menu 7 - 2
Part Probe Probe Deflection Offset screen 7 - 5
part probe, tool setup 7 - 4
part probing 7 - 9
automated method 7 - 42
deflection offset calibration 7 - 5
manual mode part setup cycles 7 - 13
manual mode part skew cycles 7 - 35
parameters 7 - 2
X/Y Skew 7 - 9, 7 - 35
part probing automatic cycles
fields 7 - 43
Part Skew 7 - 44
Part Zero 7 - 43
part probing cycles
Cylinder 7 - 20
Cylinder Skew 7 - 39
Edge 7 - 14
Edge Skew 7 - 37
Hole or Circle Pocket 7 - 17
Hole or Circle Pocket Skew 7 - 38
Part Setup fields 7 - 15
Part Zero storage 7 - 33
Plane Intersection 7 - 30
Rectangular Pocket 7 - 23
Rectangular Pocket Skew 7 - 40
Rectangular Solid 7 - 26
Rectangular Solid Skew 7 - 41
Part Probing softkey, Part Setup screen 7 - 49, 7 - 19
Part Quality Verification 7 - 46
Part Setup 7 - 9
manual probing cycles 7 - 13
parameters 7 - 3
probing option 7 - 3
working envelope 7 - 10
PART SETUP (F1) softkey, Part Setup screen 7 - 9
Part Setup cycles, manual method 7 - 13
Part Setup probing cycles 7 - 11
Part Setup probing cycles, processes for each type 7 - 11
Part Setup screen 7 - 9
PART SETUP softkey, Part Setup screen 7 - 2
Part skew
  automatic probing cycles 7 - 42
  manual probing cycles 7 - 35
Part Skew cycles 7 - 34
Part Skew cycles, manual method 7 - 35
PART SKEW PROBE CYCLES (F3) softkey, Part Setup screen 7 - 9, 7 - 37
Part Skew Probing cycle 7 - 35
Part Skew probing cycles, processes for each type 7 - 34
Part X Offset, DXF 2 - 5
Part Y Offset, DXF 2 - 5
Part Zero cycle fields 7 - 43
  part zero different for Transformed Planes program example 5 - 64
  part zero does not change for Transformed Planes program example 5 - 62
PART ZERO PROBE CYCLES (F2) softkey, Part Setup screen Part Probe functions 7 - 9
PART ZERO PROBE CYCLES softkey, Part Setup screen Part Probe functions 7 - 13
Part Zero storage 7 - 33
  Part Zero storage, at end of probing cycles 7 - 33
  Part Zero X and Y 7 - 9
  Part Zero X field 7 - 33
  Part Zero Y, X field
    Edge cycle 7 - 15
  Part Zero Y field 7 - 33
Pattern 13 - 3
  Pattern End, Rotary A and Rotary A Tilr B configurations 5 - 12
  Pattern Locations softkey 2 - 4
  Pattern Loop data blocks, nesting, rotary-axis program, WinMax Mill 5 - 23, 5 - 27, 5 - 32, 5 - 46
Peck Depth field
  3D Mold Paratmeters 1 - 7
  Diamond data block 4 - 6
  Hexagon data block 4 - 9
  Mill triangle data block 4 - 3
Plane Intersection cycle 7 - 30
Plunge Feed field
  3D Mold Paratmeters 1 - 8
  Diamond data block 4 - 6
  Hexagon data block 4 - 9
  Mill triangle data block 4 - 3
Pocket Boundary 13 - 2
Pocket Island 13 - 2
Pocket Plunge Near Center field, Milling 2 Parameters 3 - 2
Position softkey
  DXF 2 - 4
POSITION TOOL OVER THE PROBE (F5) softkey 8 - 9, 8 - 31, 8 - 33, 8 - 34
Present field, Part Setup screen 7 - 3
  preset values
    skew angle 7 - 36
Preset X field
  Cylinder cycle 7 - 18
  Edge cycle 7 - 15
  Hole or Circle Pocket cycle 7 - 21
  Plane Intersection cycle 7 - 24
  Rectangular Pocket cycle 7 - 27
  Rectangular Solid cycle 7 - 31
Preset Y field
  Cylinder cycle 7 - 18
  Edge cycle 7 - 24
  Hole or Circle Pocket cycle 7 - 21
  Plane Intersection cycle 7 - 31
  Rectangular Pocket cycle 7 - 27
  Rectangular Solid cycle 7 - 15
PROBE A SINGLE TOOL (F3) softkey, Tool Setup screen 8 - 34
PROBE A SINGLE TOOL softkey, Tool Setup screen 8 - 34
Probe Calibration
  Zero Calibration mode 8 - 30
    with Part Probe 8 - 31
    with Reference Tool 8 - 31
PROBE CURRENT TOOL NOW (F7) softkey, Tool Setup screen 8 - 34
Probe Cycle Type field, Tool Breakage Detection 8 - 16
Probe Cycle Type field, Tool Monitoring screen 8 - 15
Probe Cycle Type field, Tool Wear Detection 8 - 17
Probe Cycle Type, Tool Wear Detection 8 - 16
PROBE MULTIPLE TOOLS (F4) softkey, Tool Setup screen 8 - 34
Probe Multiple Tools softkey 8 - 7, 8 - 13
PROBE PART SETUP (F6) softkey, New Block screen 7 - 43
Probe Part Setup data block 7 - 43
  accessing 7 - 42
  execution 7 - 45
Probe Part Setup fields 7 - 43
PROBE PART SETUP softkey, New Block screen 7 - 43
Probe Tool Monitoring data block 8 - 14
Probe Z 7 - 9
Probe Z field, Edge cycle 7 - 33
Probing

Probing Parameters, Zero Calibration mode 8 - 9

Probing Tool 8 - 1

PROBING (F2) softkey, Tool Setup screen 8 - 26

PROBING (F5) softkey, Part Setup screen 7 - 5, 7 - 13

Probing Axis field, Part Setup screen 7 - 15

Probing Cycle field 8 - 3, 8 - 9

Probing Direction field, Part Setup screen 7 - 15

Probing Direction X field, Part Setup screen 7 - 31

Probing Direction Y field, Part Setup screen 7 - 27

Probing Length X field, Part Setup screen 7 - 27

Probing Length Y field, Part Setup screen 7 - 27

Probing Method

Tool Breakage Detection 8 - 16

Tool Wear Detection 8 - 17

Probing Parameters

feedrate 7 - 11

Probing Radius field, Part Setup screen 7 - 21

progrname.dat 7 - 46

progrname.txt 7 - 46

R

Radius 5 - 38

Radius Bottom field 5 - 29, 5 - 31, 5 - 36, 5 - 38, 5 - 39

Rotary Polygon block 5 - 44

Rotary Slot Start tab 5 - 41

Radius End field 5 - 29, 5 - 36

Radius End field, Arc segment, BC configuration 5 - 30

Radius field 5 - 15, 5 - 16, 5 - 21, 5 - 22, 5 - 30, 5 - 31, 5 - 37

3D Mold Arc 1 - 11

3D Mold Blend Arc 1 - 12

Diamond data block 4 - 5

Hexagon data block 4 - 8

Mill triangle data block 4 - 2

Mill triangle data block 4 - 2

Radius Start field 5 - 25, 5 - 31, 5 - 36, 5 - 38, 5 - 39

Rotary Polygon block 5 - 44

Rotary Slot Start tab 5 - 41

Radius Start field, Arc segment, BC configuration 5 - 30

Radius, DXF 2 - 8

Rapid Clearance 8 - 33

Rapid Clearance field, Tool Setup

Probing Parameters, Absolute Tool Length mode 8 - 4

Rapid Z Position 8 - 33

Tool Probing 8 - 5, 8 - 11

Rapid Z Position, Tool Setup

Probing Parameters, Zero Calibration mode 8 - 9

Rectangular Pocket cycle

probing cycle 7 - 23, 7 - 24

skew probing cycle 7 - 40

Rectangular Solid cycle

probing cycle 7 - 26, 7 - 27

skew probing cycle 7 - 41

Reference Block column, Reference Block Deflection Offsets screen 7 - 7

Reference block deflection offset method 7 - 7

REFERENCE BLOCK METHOD (F2) softkey, Reference Block Deflection Offsets screen 7 - 7

Reference Tool Touch, tool probing 8 - 27

Relief 1 field

Diamond data block 4 - 5

Hexagon data block 4 - 8

Mill triangle data block 4 - 2

Relief 2 field

Diamond data block 4 - 5

Hexagon data block 4 - 8

Relief 3 field

Hexagon data block 4 - 8

Repetitions field

Part Setup screen 7 - 3

Tool Setup screen 8 - 21

Repetitions field

Tool Setup screen 8 - 20

RESET PROBE WORK REGION TO MAXIMUM (F4) softkey, Part Setup screen 7 - 4

Retract feed field, Tool Setup 8 - 20, 8 - 21

Reverse, DXF 2 - 3

RING GAUGE (F1) softkey, Part Probe Deflection Offsets screen 7 - 6

ring gauge deflection offset method 7 - 5

Rotary Axis machining 5 - 3

Rotary Centerline 5 - 8

Rotary Circle softkey 5 - 12

Rotary Circle, BC configuration 5 - 31

Rotary Frame softkey 5 - 12

Rotary Lines and Arcs softkey 5 - 12

Rotary Locations, AC configuration 5 - 24

Rotary Locations, Rotary A and Rotary A Tilt B configurations 5 - 12
Spindle Clearance, Tool Setup
Probing Parameters, Absolute Tool Length mode 8 - 4, 8 - 10
Spindle Clearance, Tool Setup
Probing Parameters, Zero Calibration mode 8 - 34
Spindle Speed, Tool Setup
Probing Parameters, Absolute Tool Length mode 8 - 4, 8 - 10
Spindle Speed, Tool Setup
Probing Parameters, Zero Calibration mode 8 - 33
Spindle Usage
Tool Breakage Detection 8 - 16
Tool Probing 8 - 5, 8 - 11
Tool Wear Detection 8 - 17
Spindle Usage, Tool Setup
Probing Parameters, Absolute Tool Length mode 8 - 4, 8 - 10
Spindle Usage, Tool Setup
Probing Parameters, Zero Calibration mode 8 - 33
Split softkey, DXF 2 - 8
Start Angle 1 field
Cylinder cycle 7 - 21
Hole or Circle Pocket cycle 7 - 18
Start Angle 2 field
Cylinder cycle 7 - 18
Hole or Circle Pocket cycle 7 - 21
Start Angle 3 field
Cylinder cycle 7 - 21
Hole or Circle Pocket cycle 7 - 18
Start Angle field
3D Mold Parameters 1 - 4
Start Angle, DXF 2 - 8
Start Cap field
Mill Slot Caps tab 5 - 42
START PROBING CYCLE (F1) softkey
Cylinder cycle 7 - 18
Cylinder Skew cycle 7 - 31
Edge cycle 7 - 39
Edge Skew cycle 7 - 40
Hole or Circle Pocket cycle 7 - 24
Hole or Circle Pocket Skew cycle 7 - 22
Plane Intersection cycle 7 - 28
Rectangular Pocket cycle 7 - 38
Rectangular Pocket Skew cycle 7 - 15
Rectangular Solid cycle 7 - 41
Rectangular Solid Skew cycle 7 - 37
Step Size field
3D Mold Parameters 1 - 7
Stock Allowance field
3D Mold Parameters 1 - 8
Stop field 5 - 11

Store Calculated Value softkey 1 - 10, 1 - 11
STORE MACHINE POSITION (F7) softkey,
Part Setup screen 7 - 4
stylus diameter 7 - 3
Stylus Diameter field, Part Setup screen 7 - 3
Stylus Width field, Tool Setup screen 8 - 20
Sweep Angle
3D mold Arc block 1 - 11
Rotary AC 5 - 21
Rotary BC 5 - 30
Universal Rotary 5 - 37
Sweep Angle field 5 - 15
Mill Slot Geometry tab 5 - 42
Sweep Angle, DXF 2 - 8

T

TCP/IP
definition 12 - 3
tilt axis at 45 degrees program example,
WinMax Mill 5 - 61
Tilt Axis machining 5 - 3
tilt-axis at 90 degrees program example,
WinMax Mill 5 - 60
tilt-axis hardware 5 - 2
tilt-axis table 5 - 3
Tool Rot
Rotary Position, rotary-axis program,
WinMax Mill 5 - 12
tool breakage detection 8 - 15
Tool Breakage Detection cycle 8 - 15
TOOL BREAKAGE DETECTION softkey, Tool Monitoring screen 8 - 15
Tool Cal Length
Tool Probing 8 - 6
Tool Cal Length field, Tool Probing 8 - 2
Tool Change Optimization 9 - 1
Tool Change Optimization Off block 9 - 2, 9 - 3
Tool Change Optimization On block 9 - 2
Tool Change Optimization parameter 9 - 1
Tool Diameter field
3D Mold Parameters 1 - 8
Tool field
3D Mold Parameters 1 - 2
Diamond data block 4 - 5
Hexagon data block 4 - 8
Mill triangle data block 4 - 2
Tool Breakage Detection 8 - 16
Tool Wear Detection 8 - 17
Tool Fixture (TPS) Option 10 - 1
Tool In Spindle field for activating probe hardware, Tool Setup screen 7 - 4
Tool Length, Tool Setup
Probing Parameters, Zero Calibration mode 8 - 33
Tool Measurement screen, use in Tool Probing 8 - 29
tool monitoring 8 - 14
spare tool 8 - 14
tool breakage detection 8 - 15
Tool Monitoring (Probing) softkey 8 - 14
Tool Monitoring Enable, General Parameters screen 8 - 18
tool probe
deflection offset calibration 8 - 26
multiple tool cycle 8 - 7, 8 - 13
Retract Feed 8 - 20, 8 - 21
Setup fast feed 8 - 20, 8 - 21
X & Y location 8 - 19
TOOL PROBE DEFLECTION OFFSET (F7) softkey, Tool Probe Deflection Offsets screen 8 - 26
Tool probe deflection offset calibration 8 - 26
absolute location 8 - 27
reference tool touch 8 - 27
Tool Probe Deflection Offset screen accessing 8 - 26
Tool Probe Parameters
Laser Probe 8 - 21
Touch Probe 8 - 19
Tool Probe Parameters screen 8 - 19
Tool Probe Setup fields 8 - 19, 8 - 21
Tool Probe Setup Parameters 8 - 19, 8 - 21
TOOL PROBE SETUP PARAMETERS (F1) softkey, Tool Setup screen 8 - 34
TOOL PROBE SETUP PARAMETERS softkey, Tool Setup screen 8 - 34
Tool Probe Setup softkey, laser tool calibration 8 - 24
tool probing
Diameter sequence, single tool 8 - 6, 8 - 12
multiple tool cycle 8 - 7, 8 - 13
PROBE MULTIPLE TOOLS (F4) softkey 8 - 7, 8 - 13
Zero Calibration sequence, single tool 8 - 5, 8 - 11
Tool Probing Cycle Defaults 8 - 4
Tool Probing Option 8 - 1
Tool Probing Parameter Definitions
Zero Calibration mode 8 - 33
Tool Probing softkey 8 - 8
Tool Probing softkey, Tool Setup
Probing Parameters, Absolute Tool Length mode 8 - 2
Tool Probing, Tool Setup

Probing Parameters, Zero Calibration mode 8 - 31
Tool Quality Monitoring 8 - 14
Tool Removal 10 - 3
Tool Setup
parameters 8 - 19, 8 - 21
Probing Parameters 8 - 33
Tool Setup Probing Parameters
Zero calibration mode 8 - 8
Tool Type field
3D Mold Parameters 1 - 8
Tool Wear Detection cycle 8 - 16
touch probe 8 - 6, 8 - 11
TOUCH PROBE PARAMETERS softkey, Tool Setup screen 8 - 19
touch tool probe parameters 8 - 19
Transform Part Zero field 5 - 11
Transform Plane 5 - 5
orientation, rotary-axis program,
WinMax Mill 5 - 5
part zero location, rotary-axis program,
WinMax Mill 5 - 5
Transform Plane End, Rotary A and Rotary A Tilt B configurations 5 - 12
Transform Plane program example, WinMax Mill 5 - 57
Transform Plane Reference Points 5 - 6
Transform Plane, Rotary A and Rotary A Tilt B configurations 5 - 12
travel limits 7 - 10
travel limits, part probe 7 - 10
Triangle Data Block 4 - 2
Trim softkey, DXF 2 - 8
True-Type Fonts 5 - 41
Type field
3D Mold Parameters 1 - 2

U
UltiMonitor 12 - 1
UltiMotion 11 - 1
UltiMotion Option 11 - 2
UltiNet 12 - 1
glossary 12 - 2
UltiPockets Option 13 - 1
USE DATUM POINT TO GET Z OFFSET (F2) softkey, Ring Gauge Deflection Offset 7 - 6
USE GAUGE TO GET X&Y OFFSETS (F1) softkey, Ring Gauge Deflection Offset 7 - 6
Use Hole Location Method softkey 2 - 3
Use Hole Pattern Method softkey 2 - 3
Use Offset Z field 5 - 6, 5 - 11
USE PROBE TO DETERMINE OFFSET (F1)softkey, Reference Block Deflection Offsets 7 - 7, 7 - 8
USE PROBE TO DETERMINE OFFSET softkey, Reference Tool Touch 8 - 27
USER 1-4 NAME field
FTP Server Settings screen 12 - 9
USER 1-4 PASSWORD field
FTP Server Settings screen 12 - 9
USER 1-4 PATH field
FTP Server Settings screen 12 - 10

V
verification of tool and part quality 7 - 46

W
Width field
  Rotary Slot Start tab 5 - 41
Window Select softkey 2 - 3, 2 - 4
working envelope 7 - 3, 7 - 10
Working envelope, part probe 7 - 3, 7 - 10

X
X & Y contact point location, tool probe,
  Tool Setup screen 8 - 19
X and Y offsets, Ring Gauge Deflection Offsets 7 - 6
X Center field 5 - 15, 5 - 16
  3D Mold Arc 1 - 11
  3D Mold Blend Arc 1 - 12
  Diamond data block 4 - 5
  Hexagon data block 4 - 8
  Mill triangle data block 4 - 2
X Corner field 5 - 16
X Distance field 5 - 17, 5 - 23
X End field 5 - 14, 5 - 15
  3D Mold Arc 1 - 11
  3D Mold Blend Arc 1 - 12
  3D Mold Line 1 - 10
X Length field 5 - 16
X Length Offset, Tool Setup
  Probing Parameters, Absolute Tool Length mode 8 - 4, 8 - 10
X Min, X Max fields, Part Setup screen 7 - 3
X Position field 5 - 11
X Safety Position field 5 - 11
X Start 5 - 14
X Start field 5 - 14, 5 - 15
  3D Mold Arc 1 - 11
  3D Mold Blend Arc 1 - 12
  3D Mold Contour 1 - 9
  3D Mold Line 1 - 10
X Value, DXF 2 - 8
X/Y skew 7 - 45
X/Y Skew, part probing 7 - 9
XY Angle field
  3D Mold Line 1 - 10
XY Angle, DXF 2 - 8
XY Length field
  3D Mold Line 1 - 10
XZ Angle field, 3D Mold Line 1 - 10
XZ Length field, 3D Mold Line 1 - 10
XZ Revolved about Z 1 - 2
XZ Translated in Y 1 - 2

Y
Y Center field 5 - 21, 5 - 22
  3D Mold Arc 1 - 11
  3D Mold Blend Arc 1 - 12
  Diamond data block 4 - 5
  Hexagon data block 4 - 8
  Mill triangle data block 4 - 2
Y Corner field 5 - 22
Y End Field 5 - 20
Y End field 5 - 21
  3D Mold Arc 1 - 11
  3D Mold Blend Arc 1 - 12
  3D Mold Line 1 - 10
  3D Mold Parameters 1 - 5
Y End, Arc Segment, AC configuration 5 - 21
Y Length field 5 - 22
Y Length Offset, Tool Setup
  Probing Parameters, Absolute Tool Length mode 8 - 4, 8 - 10
Y Length Offset, Tool Setup
  Probing Parameters, Zero Calibration mode 8 - 34
Y Min, Y Max fields, Part Setup screen 7 - 3
Y Off of Centerline field 5 - 13
Y Position field 5 - 11
Y Revolved about X 1 - 2
Y Safety Position field 5 - 11
Y Start field 5 - 20, 5 - 21
  3D Mold Arc 1 - 11
  3D Mold Blend Arc 1 - 12
  3D Mold Contour 1 - 9
  3D Mold Line 1 - 10
  3D Mold Parameters 1 - 5
Y Value, DXF 2 - 8

Z
Z Bottom field 5 - 14, 5 - 16, 5 - 20, 5 - 22
  Diamond data block 4 - 5
  Hexagon data block 4 - 8
Mill triangle data block 4 - 2
Z Center field 5 - 30, 5 - 31
  3D Mold Arc 1 - 11
  3D Mold Blend Arc 1 - 12
Z Center field, Arc segment, BC configuration 5 - 30
Z Corner field 5 - 31
Z Depth field, Cylinder cycle 7 - 21
Z Depth field, Rectangular Solid cycle 7 - 27
Z Distance field 5 - 26, 5 - 32
Z Drop Down Depth
  Tool Probing 8 - 6, 8 - 12
Z Drop Down Depth, Tool Setup
  Probing Parameters, Absolut Tool Length mode 8 - 4, 8 - 10
Z drop down depth, Tool Setup
  Probing Parameters, Zero Calibration mode 8 - 34
Z End field 5 - 14, 5 - 15, 5 - 20, 5 - 29, 5 - 30
  3D Mold Arc 1 - 11
  3D Mold Blend Arc 1 - 12
  3D Mold Line 1 - 10
Z End field, Arc segment, BC configuration 5 - 29
Z End, Arc Segment, AC configuration 5 - 21
Z Home 5 - 12
Z Home field 5 - 35
Z Length field 5 - 31
Z Move Type field 5 - 12
Z offset, Ring Gauge Deflection Offset 7 - 6
Z Safety 5 - 12
Z Start field 5 - 14, 5 - 15, 5 - 16, 5 - 20,
  5 - 21, 5 - 22, 5 - 29, 5 - 30
  3D Mold Arc 1 - 11
  3D Mold Blend Arc 1 - 12
  3D Mold Contour 1 - 9
  3D Mold Line 1 - 10
  3D Mold Parameters 1 - 7
  Diamond data block 4 - 5
  Hexagon data block 4 - 8
  Mill triangle data block 4 - 2
Zero Angle Vector 5 - 48
Zero Calibration
  Tool Probing 8 - 11
zero calibration
  tool probing sequence, single tool 8 - 5,
    8 - 11
Zero Calibration, Tool Setup
  Probing Parameters, Zero Calibration mode 8 - 30
Zero Radius 1 - 3