WINMAX LATHE CONVERSATIONAL PART PROGRAMMING
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.

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.

© 2018 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

For Hurco subsidiary contact information, go to Hurco’s Web site:

[www.hurco.com](http://www.hurco.com)
TABLE OF CONTENTS

Documentation Conventions .................................................... - xiii

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

Overview ................................................................. 1 -  1
  Conversational Part Programming ...................................... 1 -  2
  Data Blocks .............................................................. 1 -  2
    Types of Data Blocks .................................................. 1 -  3
  Conversational Programming Calculator ............................... 1 -  6
    Automatic Calculations ............................................... 1 -  6
  Program Editing Features ............................................... 1 -  7
    Program Review ....................................................... 1 -  8
      Edit From the Program Review Screen ............................ 1 - 10
      Move a Data Block .................................................. 1 - 10
      Copy a Data Block .................................................. 1 - 10
      Copy a Data Block to Another Part Program .................... 1 - 10
      Delete Blocks ...................................................... 1 - 11
      Insert Blocks ....................................................... 1 - 11
      Create Finish Profile .............................................. 1 - 11

Conversational Part Programming ......................................... 2 -  1
  New Data Block .......................................................... 2 -  2
  Position Data Block .................................................... 2 -  3
    Tool Change Override ................................................ 2 -  8
  Turning Data Blocks .................................................... 2 - 10
  Profile Data Block ...................................................... 2 - 11
    Process Tab .............................................................. 2 - 11
      Turn Rough Operation ............................................... 2 - 14
      Turn Finish Operation .............................................. 2 - 17
      Groove Rough Operation ............................................. 2 - 20
      Groove Finish Operation ............................................. 2 - 22
    Geometry Tab ............................................................ 2 - 24
      Elements ............................................................... 2 - 25
        Turn Element ....................................................... 2 - 28
        Face Element ..................................................... 2 - 29
        Taper Element .................................................... 2 - 30
        Arc Element ....................................................... 2 - 32
        Blend Arc Element ................................................ 2 - 34
        Chamfer Element ................................................... 2 - 36
  Threading Cycle .......................................................... 2 - 38
    Process ................................................................. 2 - 38
    Geometry ............................................................... 2 - 40
      OD Straight Thread Type ............................................ 2 - 41
      OD Taper Thread Type ............................................... 2 - 43
      ID Straight Thread Type ............................................. 2 - 45
      ID Taper Thread Type ................................................ 2 - 47
      Face Thread Type .................................................... 2 - 49
  Grooving Cycle ........................................................... 2 - 51
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Process</td>
<td>2-52</td>
</tr>
<tr>
<td>Geometry</td>
<td>2-54</td>
</tr>
<tr>
<td>Patterns</td>
<td>2-57</td>
</tr>
<tr>
<td>Hole Cycle</td>
<td>2-58</td>
</tr>
<tr>
<td>Cutoff Block</td>
<td>2-62</td>
</tr>
<tr>
<td>Process/Geometry</td>
<td>2-62</td>
</tr>
<tr>
<td>Advanced Pecking</td>
<td>2-62</td>
</tr>
<tr>
<td>Live Tooling Data Blocks</td>
<td>2-66</td>
</tr>
<tr>
<td>Radial Milling</td>
<td>2-67</td>
</tr>
<tr>
<td>Programming Coordinates and Linear Y Motion</td>
<td>2-68</td>
</tr>
<tr>
<td>No Linear Y Motion</td>
<td>2-68</td>
</tr>
<tr>
<td>Y-Axis Coordinate System - Radial Surface of the Part without Linear Y</td>
<td>2-69</td>
</tr>
<tr>
<td>Live Tool Mode Milling with Polar Coordinates</td>
<td>2-69</td>
</tr>
<tr>
<td>Linear Y Motion</td>
<td>2-70</td>
</tr>
<tr>
<td>Radial Lines and Arcs</td>
<td>2-71</td>
</tr>
<tr>
<td>Process Tab</td>
<td>2-71</td>
</tr>
<tr>
<td>Geometry Tab</td>
<td>2-76</td>
</tr>
<tr>
<td>Segments</td>
<td>2-78</td>
</tr>
<tr>
<td>Line</td>
<td>2-79</td>
</tr>
<tr>
<td>Arc</td>
<td>2-83</td>
</tr>
<tr>
<td>Blend Arc</td>
<td>2-86</td>
</tr>
<tr>
<td>Radial Circle</td>
<td>2-89</td>
</tr>
<tr>
<td>Process Tab</td>
<td>2-90</td>
</tr>
<tr>
<td>Calculated Plunge Points for Radial Circles</td>
<td>2-92</td>
</tr>
<tr>
<td>Geometry Tab</td>
<td>2-93</td>
</tr>
<tr>
<td>Radial Frame</td>
<td>2-95</td>
</tr>
<tr>
<td>Process Tab</td>
<td>2-95</td>
</tr>
<tr>
<td>Calculated Plunge Points for Radial Frames</td>
<td>2-99</td>
</tr>
<tr>
<td>Geometry Tab</td>
<td>2-100</td>
</tr>
<tr>
<td>Radial Slot</td>
<td>2-102</td>
</tr>
<tr>
<td>Process Tab</td>
<td>2-102</td>
</tr>
<tr>
<td>Geometry Tab</td>
<td>2-105</td>
</tr>
<tr>
<td>Radial Lettering</td>
<td>2-110</td>
</tr>
<tr>
<td>Radial Holes</td>
<td>2-114</td>
</tr>
<tr>
<td>Radial Hole Cycle</td>
<td>2-115</td>
</tr>
<tr>
<td>Drilling Operation</td>
<td>2-119</td>
</tr>
<tr>
<td>Tapping Operation</td>
<td>2-119</td>
</tr>
<tr>
<td>Reaming Operation</td>
<td>2-120</td>
</tr>
<tr>
<td>Radial Locations</td>
<td>2-121</td>
</tr>
<tr>
<td>Radial Loop Around C-Axis</td>
<td>2-123</td>
</tr>
<tr>
<td>Axial Milling</td>
<td>2-125</td>
</tr>
<tr>
<td>Programming Coordinates and Linear Y Motion</td>
<td>2-125</td>
</tr>
<tr>
<td>No Linear Y Motion</td>
<td>2-126</td>
</tr>
<tr>
<td>Linear Y Motion</td>
<td>2-127</td>
</tr>
<tr>
<td>Axial Lines and Arcs</td>
<td>2-128</td>
</tr>
<tr>
<td>Process Tab</td>
<td>2-128</td>
</tr>
<tr>
<td>Geometry Tab</td>
<td>2-133</td>
</tr>
<tr>
<td>Segments</td>
<td>2-135</td>
</tr>
<tr>
<td>Line</td>
<td>2-136</td>
</tr>
<tr>
<td>Arc</td>
<td>2-140</td>
</tr>
<tr>
<td>Blend Arc</td>
<td>2-143</td>
</tr>
<tr>
<td>Axial Circle</td>
<td>2-147</td>
</tr>
<tr>
<td>Process Tab</td>
<td>2-147</td>
</tr>
<tr>
<td>Calculated Plunge Points for Axial Circles</td>
<td>2 - 150</td>
</tr>
<tr>
<td>------------------------------------------</td>
<td>--------</td>
</tr>
<tr>
<td>Geometry Tab</td>
<td>2 - 151</td>
</tr>
<tr>
<td>Axial Frame</td>
<td>2 - 153</td>
</tr>
<tr>
<td>Process Tab</td>
<td>2 - 153</td>
</tr>
<tr>
<td>Calculated Plunge Points for Axial Frames</td>
<td>2 - 157</td>
</tr>
<tr>
<td>Geometry Tab</td>
<td>2 - 158</td>
</tr>
<tr>
<td>Axial Slot</td>
<td>2 - 160</td>
</tr>
<tr>
<td>Process Tab</td>
<td>2 - 160</td>
</tr>
<tr>
<td>Geometry Tab</td>
<td>2 - 162</td>
</tr>
<tr>
<td>Axial Flats</td>
<td>2 - 168</td>
</tr>
<tr>
<td>Process Tab</td>
<td>2 - 168</td>
</tr>
<tr>
<td>Geometry Tab</td>
<td>2 - 171</td>
</tr>
<tr>
<td>Axial Slot</td>
<td>2 - 175</td>
</tr>
<tr>
<td>Axial Holes</td>
<td>2 - 178</td>
</tr>
<tr>
<td>Axial Hole Cycle</td>
<td>2 - 179</td>
</tr>
<tr>
<td>Drilling Operation</td>
<td>2 - 180</td>
</tr>
<tr>
<td>Tapping Operation</td>
<td>2 - 181</td>
</tr>
<tr>
<td>Reaming Operation</td>
<td>2 - 182</td>
</tr>
<tr>
<td>Axial Locations</td>
<td>2 - 182</td>
</tr>
<tr>
<td>Axial Bolt Circle</td>
<td>2 - 185</td>
</tr>
<tr>
<td>Cutter Compensation</td>
<td>2 - 188</td>
</tr>
<tr>
<td>Climb Milling</td>
<td>2 - 190</td>
</tr>
<tr>
<td>Conventional Milling</td>
<td>2 - 190</td>
</tr>
<tr>
<td>Miscellaneous Data Blocks</td>
<td>2 - 191</td>
</tr>
<tr>
<td>Change Parameters</td>
<td>2 - 192</td>
</tr>
<tr>
<td>Change Part Setup</td>
<td>2 - 195</td>
</tr>
<tr>
<td>Machine Function—M Code</td>
<td>2 - 198</td>
</tr>
<tr>
<td>Bar Feed Block</td>
<td>2 - 200</td>
</tr>
<tr>
<td>Comment Block</td>
<td>2 - 201</td>
</tr>
<tr>
<td>Repeat Start Block</td>
<td>2 - 202</td>
</tr>
<tr>
<td>Repeat End Block</td>
<td>2 - 203</td>
</tr>
<tr>
<td>Repeat Start and Repeat End Block Sample</td>
<td>2 - 204</td>
</tr>
<tr>
<td>Move Tailstock Data Block</td>
<td>2 - 206</td>
</tr>
<tr>
<td>Transfer Data Block</td>
<td>2 - 207</td>
</tr>
<tr>
<td>NC Program Call</td>
<td>2 - 210</td>
</tr>
<tr>
<td>Sample Part Program</td>
<td>3 - 1</td>
</tr>
<tr>
<td>Record of Changes</td>
<td>I</td>
</tr>
<tr>
<td>Index</td>
<td>V</td>
</tr>
</tbody>
</table>
DOCUmentation COvEnTIONS

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

Sample Screens

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

Softkeys

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

Screen Areas

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

Data Entry

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

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

Prompts and Error/Status Area

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

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

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

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

Status Bar

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

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

Console Buttons and Keys

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

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

Using the Touchscreen

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

Icons

This manual may contain the following icons:

Caution/Warning

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

→ Ensures proper operation of the machine and control.

Troubleshooting

❓ Steps that can be taken to solve potential problems.

Hints and Tricks

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

Where can we go from here?

🌐 Lists several possible options the operator can take.
PROGRAMMING AND OPERATION INFORMATION

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

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

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

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

Using the On-screen Help

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

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

- Use the arrow buttons to move between pages within a Help topic and to move through topics.
- Use the Contents tab for a list of information sorted by subject:
  1. Select the “+” to expand the topic and view sub-topics.
  2. Select the topic to display it.

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

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

### Printing the Programming Manuals

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

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

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

   ![The SHOW ALL FILE TYPES field in User Interface Settings must be set to YES (default is NO) in order to see the PDF files in the directory. Access the SHOW ALL FILE TYPES field in Auxiliary Mode, Utilities/ User Preferences/ User Interface Settings.](image)

4. Highlight the PDF file(s) in the right-hand pane, and select the **COPY F2** softkey.
5. Ensure that your media is loaded (either a floppy disk in the disk drive or a USB memory device in the USB port), and navigate to the proper location in the left-hand pane of the DISK OPERATIONS screen (either the floppy drive A: or the USB port E:). Highlight the desired location.
6. Place the cursor in the right-hand pane and select the **PASTE F3** softkey to paste the PDF file(s) to the desired location.

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

This section contains information about these Conversational Part Programming topics:

- Conversational Part Programming .................................................. 1 - 2
- Data Blocks .......................................................................................... 1 - 2
- Conversational Programming Calculator .......................................... 1 - 6
- Automatic Calculations ....................................................................... 1 - 6
- Program Editing Features ................................................................... 1 - 7
Conversational Part Programming

Conversational part programming allows you to create a part program from a blueprint or program worksheet while working at the machine. Operating selections and prompts on the screen lead you through the steps necessary to enter the data for a part program.

To create a Conversational Part Program,

1. Press the **Input** console key to display the Input screen. The Input screen allows you to access setup functions including Part Setup, Tool Setup, and Program Parameters.
2. Enter machining operation information in the Part Setup and Tool Setup functions. This information is stored within data blocks describing each operation to be performed. Create and access the data blocks through the Input screen.
3. Select the PART PROGRAMMING F3 softkey to begin programming. The system displays either the first block of an existing program or a New Block screen for a new program.
4. Select the PROJECT MANAGER F8 softkey to access the file management features, such as Save, Save As, and Open.

Refer to the WinMax Lathe Getting Started Planning, on page 4 - 5 for details about program planning and part and tool setup, setting up directories, and saving files.

Data Blocks

A Conversational Part Program is made up of numerically ordered data blocks that store information specific to the program. Each data block contains all of the information to execute that particular function.

When changing the cutting type (for example, hole drilling or profile turning) for a data block, a different data block must be created to describe the new cutting type. When a different tool is needed in a program, a new data block must be created to describe the actions of that tool. An exception is the Profile data block in which the roughing and finishing tools can be called within the same block.

Use these basic programming operations when creating data blocks:

- Create a new data block or edit an existing one through the Input screen or Review screen.
- Insert a new data block between two sequentially numbered blocks. The data block is inserted in front of the one shown on the screen or highlighted in the Program Review screen. Data blocks following the inserted block will be automatically renumbered.
- Delete an existing data block. The data blocks following the deleted block renumber automatically.
• View a list and navigate through the part program’s data blocks on the Program Review screen.
• View a list and navigate through the part program’s tools on the Tool Review screen.
• Copy, edit, or delete information in a data block while viewing a data block screen.

Data blocks within a part program contain specific machining information:

• Operation type, such as position, profile, threading, groove, hole.
• Tool number to be used for the operation.
• Dimension coordinates relative to X- and Z-axes.
• Speed and Feedrate at which the spindle and axes move.
  • Speed selections are
    • Speed as Constant Surface Speed (CSS) in meters per minute (MPM) or feet per minute (FPM).
    • Speed as RPM.
  • Feedrate selections are
    • Feed per Minute in millimeters per minute (MMPM) and inch per minute (IPM).
    • Feed per Revolution in millimeters per revolution (MMPR) and inch per revolution (IPR).
• Locations of operations on the part.

Types of Data Blocks

The Part Programming softkeys access these data blocks:

• **Position F1**—inserts a block to select a tool and to move the tool to an X-Z location or pause a program for inspection. A Position block is commonly used to move the turret to an X-Z Home location at the end of the program or when the tool must be moved to clearance position around the part. Refer to Position Data Block, on page 2 - 3 for information.

• **TURNING ➔ F2**—accesses this set of softkeys for turning blocks:
  • **PROFILE F1**—describes the part profile, or shape, for roughing and/or finishing operations including tool selection, speed and feed information, clearances and other pertinent information. Each item of the profile is described as an individual element, such as turn, face, arc, blend arc, taper, and chamfer. Refer to Profile Data Block, on page 2 - 11 for information.
  • **THREAD F2**—inserts a block to thread the part, whether OD (outer diameter) Straight, OD Taper, ID (inner diameter) Straight, ID Taper, or Face, including tool selection, speed and feed information, clearances, and other information. Refer to Threading Cycle, on page 2 - 38 for information.
• **GROOVE F3**—inserts a block to put a groove on the part, whether OD, ID, or Face, including tool selection, speed and feed information, clearances, and other information. Refer to *Grooving Cycle, on page 2 - 51* for information.

• **HOLE F4**—inserts a block to drill, center drill, or tap the part, including tool selection, speed and feed information, clearances, and other information. You can select from these cycle types: Standard, Dwell, Peck, Chip Breaker, Center Drill, Decreasing Depth, and Decreasing Chip Breaker. Refer to *Hole Cycle, on page 2 - 58* for information.

• **CUTOFF F5**—inserts a block to cut off the part including tool selection, speed and feed information, clearances, and other information. Refer to *Cutoff Block, on page 2 - 62* for more information.

• **THREAD REPAIR F6**—allows you to re-cut a damaged thread. The Thread Repair Cycle screen and the Threading Cycle screen are the same, except this screen has the Repair tab.

• **LIVE TOOLING → F3**—accesses this set of softkeys for live-tooling blocks:
  - **RADIAL MILLING F1**—accesses a block for Line, Arc, or Blend Arc contour segments for milling around the radius of the stock. Refer to *Radial Milling, on page 2 - 67* for programming information.
  - **RADIAL HOLES F2**—accesses a block for Radial Hole Cycle, Radial Locations, or Radial Loop Around C-Axis drilling around the radius of the stock. Refer to *Radial Holes, on page 2 - 114* for programming information.
  - **AXIAL MILLING F4**—accesses a block for Axial Lines and Arcs, Axial Circle, Axial Frame, Axial Slot, and Axial Flats for milling around the C axis of the stock. Refer to *Axial Milling, on page 2 - 125* for programming information.
  - **AXIAL HOLES F5**—accesses a block for Axial Hole cycle, Axial Locations, or Axial Bolt Circle around the C axis of the stock. Refer to *Axial Hole Cycle, on page 2 - 179* for programming information.

• **MISCELLANEOUS → F4**—accesses another set of softkeys for other blocks:
  - **CHANGE PARAMETERS F1**—changes program parameters set in the Program Parameters screen. Refer to the WinMax Lathe Getting Started *Program Parameters, on page 4 - 82* for information about Program Parameters.
  - **CHANGE PART SETUP F2**—changes work offsets programmed in Part Setup. Refer to the WinMax Lathe Getting Started *Part Setup—Work Offsets, on page 4 - 11* for information about Part Setup.
  - **M CODE F3**—accesses the Machine Function screen. This screen contains M Code selections. Refer to *Machine Function—M Code, on page 2 - 198* for information.
  - **BAR FEED F4**—accesses the Bar Feed Block screen. Refer to *Bar Feed Block, on page 2 - 200* and WinMax Lathe Options *Bar Feed Block, on page 3 - 1* for programming information.
  - **COMMENT F5**—accesses the Comment Block screen which provides you with instructions during part program execution and allows you to program optional stops. Refer to *Comment Block, on page 2 - 201* for information.
• **NC PROGRAM CALL** $F6$—accesses the NC Program Call data block. Refer to *NC Program Call*, on page 2 - 210 for programming information.

• **MORE $\Rightarrow F7$**—accesses the remaining data block types.

• **REPEAT START** $F1$—indicates the start of a block or group of blocks to be repeated in a part program. Refer to *Repeat Start Block*, on page 2 - 202 for information.

• **REPEAT END** $F2$—indicates the end of a block or group of blocks to be repeated in a part program. Refer to *Repeat Start Block*, on page 2 - 202 for information.

• **MORE $\Rightarrow F7$**—returns to the first set of Miscellaneous softkeys.

• **EXIT** $F8$—returns to the first set of part programming softkeys.

• **TRANSFER** $F5$—inserts a block for transferring the stock from the Main spindle to the Sub-spindle. This softkey appears only for TMX MYS series machines. Refer to *Transfer Data Block*, on page 2 - 207 for programming information.

• **MOVE TAILSTOCK** $F5$—The data block allows you to move the Z-Axis to the tailstock and automatically hitch the Z-Axis to the tailstock. Then you can move the Z-Axis and tailstock together to a specified location at a programmed feed rate. This softkey appears only for TM12 and TM18 series machines with the auto-hitch tailstock. Refer to *Move Tailstock Data Block*, on page 2 - 206 for programming information.
Conversational Programming Calculator

To calculate data values for screen fields, follow these steps:

1. Position the cursor on the field you want to calculate.
2. Type the number used in the calculation.
3. Press the console key of the operation you want to perform (+, -, *, or ÷).
4. Type the second number used in the calculation.
5. Press the Enter key.

⇒ You may repeat steps 3 and 4 for a longer equation.

To make a change to an existing field, follow these steps:

1. Select the field.
2. Type the value you want.
3. Select the operation (+, -, *, or ÷).
4. Press Enter.

For example, to add 0.05 to a field containing a value of 1.000, type 0.05 + then Enter. The value in the field changes to 1.050.

Automatic Calculations

The Automatic Calculation feature automatically calculates values for certain fields after sufficient data is known. For example, if the X or Z end points cannot be determined from drawings, leave the fields blank. After you input additional information about an adjacent element the missing data may be calculated.

The screen displays a "CAL" when a calculated parameter is created. If you enter a value and later decide to let the control calculate the data, clear the entered value by following these steps:

1. Use the Arrow keys to move the cursor to the parameter where the automatic calculation is needed.
2. Press the C (Clear) console key.
3. Press the Enter key. The value is cleared from the program and the calculated dimension is displayed in its place (if the field remains blank, additional data must be entered before the control can calculate the value).
Program Editing Features

Several editing features are available with the WinMax Lathe. Keyboard and software features provide you with the editing capabilities to create and update part programs.

The console keyboard contains these keys:

- **Programming Mode keys**—Input, Review, and Menu keys.
- **Standard AT-keyboard keys**—Insert, Delete, Home, End, Page Up, Page Down.

Please refer to Getting Started with WinMax Lathe for details about using these keys for specific functions for Tool Setup—Geometry Offsets, on page 4 - 17 or Tool Review, on page 4 - 77. Refer to Program Review, on page 1 - 8 and for details about using these keys with the Program Review screen.

- **Cursor Control keys**—Arrows, Enter, alt, C, and F function keys.
- **Draw key**—displays the graphic in wireframe view. Refer to Getting Started with WinMax Lathe Verifying Part Programs—Graphics, on page 4 - 86 for more information about Wireframe Graphics.
- **Optional AT-Keyboard**—allows you to enter data using a keyboard.

Please refer to the Getting Started with WinMax Lathe, Consoles, on page 1 - 5 for additional information about the control.

The software provides these features:

- **Pop-up Text Entry window**—allows you to enter text, such as naming a part program, if the console does not have an optional AT-keyboard.
- **Program Review screen**—lists the data blocks in a program and the tools programmed for each block. Refer to Program Review, on page 1 - 8 for information about using this screen.
- **Tool Review screen**—lists the tool number, type, orientation, size, programming mode (diameter or radius), speed, and feed for each tool programmed in the data block. Refer to Getting Started with WinMax Lathe, Tool Review, on page 4 - 77 for information about using this screen.

The on-screen cursor

- **Moves through the fields on a data block screen**—when you press the Enter key, Page Up and Page Down keys, or keyboard arrows.
- **Stores new data typed into a field**—when you type data in the field and press the Enter key. The cursor then advances to the next field.
- **Skips a field**—(allows it to remain unchanged) when you position the cursor on the field and press the Enter key.
- **Moves to other data blocks in the part program**—when the cursor is in the Block field, press the left or right arrow keys or use Page Up and Page Down keys.
Program Review

Press the **Review** console key to display the Program Review screen. A summary of the part program’s data blocks and elements or segments is displayed.

![Program Review Screen](image)

*Figure 1–1. Program Review screen*

Use the left and right Arrow keys to move between the data blocks listed in the left pane and the elements listed in the right pane. To go directly to the block or element, place the cursor on a field, and press the **Enter** key.

Select a data block listed in the left pane to open the data block part programming screen for that block number.

Select an element listed in the right pane to open the element part programming screen for that block number. The right pane is empty for data blocks that do not include elements.
These Standard Console keys perform these functions with the Program Review screen:

- **The Insert** console key functions the same as the INSERT BLOCK BEFORE F7 softkey:
  1. Highlight a data block in the Program Review screen.
  3. Select the softkey for the new data block. The new block appears above the highlighted block.

- **The Delete** console key functions the same as the DELETE BLOCK F2 softkey:
  1. Highlight the data block to be deleted in the Program Review screen.
  2. Press the Delete key. A pop-up message appears asking if you are sure you want to delete the block.
  3. Select OK to delete the block or Cancel to cancel the deletion.

- **The Home** console key jumps to the first data block listed in the Program Review screen.

- **The End** console key jumps to the End of Program data block listed in the Program Review screen.

- **The Page Up** console key jumps to the first data block visible on the screen.

- **The Page Down** console key jumps to the last data block visible on the screen.

When the entire part program contains more data blocks than can be displayed on one screen, use the Page Down and Page Up console keys or the scroll bar located on the right-hand side of the screen to scroll through the listed data blocks.

You can cut, copy, and move data blocks to another location in a part program or even another program.

To select multiple blocks, select the block listed first and move the stylus down to the last block without lifting the stylus.

With the optional AT-keyboard, you can use Ctrl + select (using the stylus) to select non-consecutive data blocks.

The Cut and Copy softkeys function with data blocks but not with elements.
Edit From the Program Review Screen

Position the cursor on the data block or element you wish to edit, and select the MULTIPLE BLOCK FUNCTIONS F1 softkey on the Program Review screen to access these editing features:

Move a Data Block

You can move a data block to a different location within the part program using the CUT F1 and PASTE F3 softkeys. To cut and paste data blocks to another location in a part program from the Program Review screen:

1. Select the data block you want to move.
2. Select CUT F1.
3. Highlight the data block that will follow the pasted block.
4. Select PASTE F3. The data block that was cut appears above the highlighted data block on the Program Review screen.
5. Save the revised program.

Copy a Data Block

You can copy a data block to a different location in the part program using the COPY F2 and PASTE F3 softkeys.

1. Select the data block you want to copy.
2. Select COPY F2.
3. Highlight the data block that will follow the copied block.
4. Select PASTE F3. The data block that was copied from the other program appears above the highlighted data block on the Program Review screen.
5. Save the revised program.

Copy a Data Block to Another Part Program

You can copy a data block to a different part program using the COPY F2 and PASTE F3 softkeys.

1. Select the data block you want to copy.
2. Select COPY F2.
3. Access the Project Manager screen and Open the second part program.
4. Access the Program Review screen for the second part program.
5. Highlight the data block that will follow the copied block.
7. Select PASTE F3. The data block that was copied from the other program appears above the highlighted data block on the Program Review screen.
8. Save the revised program.
**Delete Blocks**

You can permanently remove data blocks from the part program. To delete data blocks from the Multiple Block Functions menu,

1. Select the data block you want to delete.
2. Press the DELETE F4 softkey.
3. Save the revised program.

**Insert Blocks**

To insert a data block or an element into a part program from the Program Review screen,

1. Highlight the data block or element that will follow the inserted block or element.
2. Select the INSERT BLOCK BEFORE F7 softkey or the Insert key.
   - If you are inserting a data block, the softkey menu changes to data block selections.
   - If you are inserting an element, the softkey menu changes to element selections.
   - If you are inserting a segment, the softkey menu changes to segment selections.
3. Select the data block, element type, or segment type. The new data block, element, or segment appears in the Program Review or New Block screen.
4. Highlight the new data block or element and press Enter. The screen changes to the new data block or element. Enter data as appropriate.
5. Save the revised program.

**Create Finish Profile**

To create a finish profile by using entries in the Rough Profile data block (for either a Turn or Groove Profile),

1. Highlight the rough profile data block listed in the Program Review screen.
2. Select the MULTIPLE BLOCK FUNCTIONS F1 softkey.
3. Select the CREATE FINISH PROFILE F7 softkey. A Finish Profile data block is listed below the Rough Profile data block in the Program Review screen. Turn Finish or Groove Finish is entered in the data block Operation field.
   - For more than one finish profile in a data block, repeat the above steps.
   - For Finish Turning data blocks, Cutter Comp is determined based on the Profile geometry. Stock Allowances are automatically set to 0.
CONVERSATIONAL PART
PROGRAMMING

This section describes steps for programming different types of data blocks.

New Data Block ............................................................... 2 - 2
Position Data Block ......................................................... 2 - 3
Turning Data Blocks ........................................................ 2 - 10
  Profile Data Block ......................................................... 2 - 11
  Threading Cycle ........................................................... 2 - 38
  Grooving Cycle ........................................................... 2 - 51
  Hole Cycle ................................................................. 2 - 58
  Cutoff Block ............................................................... 2 - 62
Live Tooling Data Blocks .................................................. 2 - 66
  Radial Milling ............................................................. 2 - 67
  Radial Holes ............................................................... 2 - 114
  Axial Milling .............................................................. 2 - 125
  Axial Hole Cycle ........................................................ 2 - 179
Miscellaneous Data Blocks ................................................ 2 - 191
  Change Parameters ...................................................... 2 - 192
  Change Part Setup ....................................................... 2 - 195
  Machine Function—M Code ........................................... 2 - 198
  Bar Feed Block .......................................................... 2 - 200
  Comment Block .......................................................... 2 - 201
  Repeat Start Block ...................................................... 2 - 202
  Repeat End Block ....................................................... 2 - 203
Move Tailstock Data Block ............................................... 2 - 206
Transfer Data Block ....................................................... 2 - 207
NC Program Call ............................................................ 2 - 210

Refer to Getting Started with WinMax Lathe, Part Setup—Work Offsets, on page 4 - 11 and Tool Setup—Geometry Offsets, on page 4 - 17 for information about part and tool setup.
New Data Block

The Part Programming softkeys access the different types of data blocks from the New Block screen. Refer to *Types of Data Blocks, on page 1-3* for descriptions of data blocks. Detailed programming information for each type of data block follows in this chapter.

To create a new data block,

1. Press the **Input** console key to access the Input screen.
2. Select the **PART PROGRAMMING F3** softkey. The New Block screen appears.
   - The Position F1 and NC Program Call F6 softkeys appear for all machine types.
   - The Turning ➔ F2, Live Tooling ➔ F3, and Miscellaneous ➔ F4 softkeys access additional softkey menus.
   - The Transfer F5 softkey appears only for TMX MYS machines.
   - The Move Tailstock F5 softkey appears only for TM12 and TM18 machines.

![Figure 2–1. TMX MYS Series New Block screen](image)

3. Select a softkey to begin creating a new data block.
Position Data Block

Position data blocks allow you to position the axes. Positions can be relative to Machine Zero or Part Zero. The default is Part Zero. If you select Machine Zero, the control checks the position values to be sure they are not out of travel before the machine will run. Insert Position blocks into a part program to:

- Move the currently active tool to a safe location.
- Automatically stop motion during part cutting to refixture or perform some other manual operation.
- Turn off the coolant and stop the spindle.
- Drive a long tool around obstructions.
- Move the tool around a tailstock.
- Move the turret back at the end of a program to provide easy access to the finished part.
- Move a new tool to a safe starting position after a tool change, but before the tool is used by the subsequent data block.
- Send the last tool used to the home position.
- Change tools at the end of a cycle.

To create a Position Block from the New Block screen, select the POSITION F1 softkey. The cursor is located at the Block field on the Position Block screen. The softkeys change depending on where the cursor is located.

⇒ The fields are displayed based on your machine configuration.

![Figure 2–2. Position Data Block screen, TMX MY Series shown](image)
The following fields appear on the Position Block screen:

- **Tool**—contains the programmed tool number from Tool Setup. To avoid unnecessary tool changes, this tool will either match the tool from the previous block or the following block in the program.

- **Tool Offsets**—contains the programmed tool offset and orientation programmed in Tool Setup. You can accept or change this offset.

- **WHERE**—moves turret or axes to the selected position:
  - **MOVE TO TOOL CHANGE**—moves the turret to the location specified in the Program Parameters User Defined Tool Change Position area.
  - **MOVE TURRET HOME**—moves the turret to the Home position. This field is used in conjunction with the Axis Order field entries (including the C Axis Order field). At least one axis must be selected in Axis Order when Move Turret Home is selected.
  - **MOVE TO POSITION**—moves the axes to the specified Reference, X, Z, W, and Y, and all Axis Order fields (including the C Axis Order field). At least one axis must be selected in Axis Order when Move To Position is selected.
  - **MOVE ALL HOME F4**—moves all axes to the Home position. This field is used in conjunction with the Axis Order X, Z, W, and Y and C Axis Order field entries. At least one axis must be selected in Axis Order when Move All Home is selected. This field appears on TMX, TMX MY, and TMX MYS Series machines and replaces the PROGRAM PARAMETERS F4 softkey.
  - **MOVE W ONLY F5**—moves only the W Axis to the position specified in the W field. This field appears on TMX, TMX MY, and TMX MYS Series machines and replaces the PART SETUP F5 softkey.
  - **MOVE W HOME F6**—moves the W Axis to the Home Position. This field appears on TMX MY and TMX MYS Series machines.
  - **MORE F7**—accesses a second softkey menu. This field appears on TMX MY and TMX MYS Series machines.
  - **MOVE Y ONLY F1**—moves only the Y Axis to the position specified in the W field. This field appears on TMX MY and TMX MYS Series machines.
  - **MOVE Y HOME F6**—moves the Y Axis to the Home Position. This field appears on TMX MY and TMX MYS Series machines.

- **REFERENCE**—contains selections for the axes locations to be relative to Part Zero or Machine Zero. This field is available only when the Where field contains the Move To Position, W Only, or Y Only selections.
- **X (DIA) OR (RAD)**—contains the X diameter or radius coordinate where the turret should move.

  This field is active when either Move Turret Home, Move to Position, or Move All Home is selected in the WHERE field.

  When the cursor is in this field, the STORE MACHINE POSITION F7 softkey appears. Select this softkey to store the current machine position into the field.

- **Z**—contains the Z coordinate where the turret should move.

  This field is active when either Move Turret Home, Move to Position, or Move All Home is selected in the WHERE field.

  When the cursor is in this field, the STORE MACHINE POSITION F7 softkey appears. Select this softkey to store the current machine position into the field.

- **W**—contains the W coordinate where the turret should move. This field appears on TMX, TMX MY, and TMX MYS Series machines.

  This field is active when either Move to Position or Move All Home is selected in the WHERE field.

  When the cursor is in this field, the STORE MACHINE POSITION F7 softkey appears. Select this softkey to store the current machine position into the field.

- **Y**—contains the Y coordinate where the turret should move. This field appears on TMX MY and TMX MYS Series machines.

  This field is active when either Move to Position or Move All Home is selected in the WHERE field.

  When the cursor is in this field, the STORE MACHINE POSITION F7 softkey appears. Select this softkey to store the current machine position into the field.
• **AXIS ORDER**—contains the selection for the order the axes will move:
  
  - **X**—specifies the order the turret moves the X axis from the current position. The default is 0.
  - **Z**—specifies the order the turret moves the Z axis from the current position. The default is 0.
  - **W**—specifies the order the W axis (Programmable Tailstock or Subspindle) moves from the current position. The default is 0. This field appears on TMX, TMX MY, and TMX MYS Series machines.
  - **Y**—specifies the Y-axis motion from the current position. The default is 0. This field appears on TMX MY and TMX MYS Series machines.

  When the cursor is in any of the Axis Order fields, these selections are available for entering the order of axes movement:
  
  - **DON'T MOVE (0)**
  - **FIRST (1)**
  - **SECOND (2)**
  - **THIRD (3)**
  - **FOURTH (4)**
  - **SIMULTANEOUS**

  An axis must be selected in the axis order field when either Move Turret Home, Move To Position, or Move All Home is selected in the Where field.

• **C AXIS ORDER**—contains the order selection for the C Axis to move relative to the other axes. This field is active when either Move Turret Home, Move to Position, or Move All Home is selected in the WHERE field. This field is available for TMM, TMX MY, and TMX MYS Series machines.
  
  - **No C**
  - **C FIRST**
  - **C LAST**
  - **SIMULTANEOUS**

• **C (DEG)**—contains the degree of the angle for the C Axis. This field is active when Move Turret Home, Move to Position, or Move All Home is selected in the Where field and either C First, C Last C Middle, or Simultaneous is selected in the C Axis Order field. This field is available for TMM, TMX MY, and TMX MYS Series machines.
• **RAPID TO POSITION**—contains the selection for X and Z axis feedrate toward the X and Z positions. This field is active when either Move To Position, Move W Only (available for TMX, TMX MY, and TMX MYS Series machines), or Move Y Only (available for TMX MY and TMX MYS Series machines) is selected in the WHERE field.
  - **YES**—moves the X and Z axes at the Rapid Traverse feedrate entered in Program Parameters.
  - **NO**—enables the feedrate to be entered in the FEED (IPM) or (MMPM) field.

• **FEED (IPM) or (MMPM)**—contains the feedrate for the programmed tool.
  - Enter the feedrate into this field when No is selected in the RAPID TO POSITION field.
  - This field is inactive when Yes is selected in the RAPID TO POSITION field.

• **PROGRAM STOP**—contains the selection for a stop block:
  - **NO**—continues the program to the next data block after the axes reach the position specified.
  - **YES**—pauses the program. To continue with the part program after executing this Position block, press the flashing **Start Cycle** button.
  - **OPTIONAL**—pauses the program when the **Opt Stop** console key is enabled. To continue with the part program after executing this Position block, press the flashing **Start Cycle** button.

• **STOP THE SPINDLE**—contains the selection for a stopping the spindle:
  - **YES**—stops the spindle after the axes reach the position specified.
  - **No**—continues running the spindle.

• **TURN OFF COOLANT**—
  - **YES**—turns off the coolant.
  - **No**—continues using the coolant.

• **DWELL (SECS)**—contains the number of seconds the spindle should dwell after the axes reach the position specified.

• **TOOL CHANGE OVERRIDE**—select the Tool Change Override check box (clear is the default) when the tool specified in this data block requires different tool change settings than those made in Program Parameters screen, User Defined Tool Change Position settings.
Tool Change Override

When the tool specified is different from the tool in the previous block or when the tool specified is different from the current tool before the Position block is run, use the Tool Change Override settings, described below.

- If the tool is different than the previous block or the current tool, then the tool change will occur after the axes move to position.
- If the tool is the same as the previous block or the current tool, then this area is ignored.

Clear the Tool Change Override check box to disable the override selections and retain the settings from Program Parameters. Clear is the default for this check box.

- **WHERE**—moves turret to the selected position:
  - **DON’T MOVE**—perform the tool change at current machine position.
  - **MOVE TO HOME**—the turret will move to the home position for the tool change.
  - **MOVE TO XZ**—the turret will move to the defined X, Z position. This field is available for TM, TMM, and TMX Series machines.
  - **MOVE TO XYZ**—the turret will move to the defined X, Y, and Z position. This field is available for TMX MY and TMX MYS series machines.
- **REFERENCE**—contains selections for the axes locations to be relative to Part Zero or Machine Zero. This field is available only when the Tool Change Override WHERE field contains the Move To XZ or Move To XYZ selection.
  - **X (DIA) OR (RAD)**—contains the X diameter or radius coordinate where the turret should move. When the cursor is in this field, the STORE MACHINE POSITION F7 softkey appears. Select this softkey to store the current machine position into the field.
  - **Z**—contains the Z coordinate where the turret should move. When the cursor is in this field, the STORE MACHINE POSITION F7 softkey appears. Select this softkey to store the current machine position into the field.
- **AXIS ORDER**—contains the selection for the order the axes will move:
  - **SIMULTANEOUS F1**—from the current position before the Position block, the turret moves both axes together to the X and Z positions.
  - **FIRST X THEN Z F2**—from the current position before the Position block, the turret moves the X axis first to the X position then the Z axis to the Z position.
  - **FIRST Z THEN X F3**—from the current position before the Position block, the turret moves the Z axis first to the Z position then the X axis to the X position.
• **FIRST X THEN Y THEN Z** \(F4\)—from the current position before the Position block, the turret moves the X axis first to the X position, then the Y axis moves to the Y position, then the Z axis to the Z position. This field is available for TMX MY and TMX MYS Series machines.

• **FIRST Z THEN Y THEN X** \(F5\)—from the current position before the Position block, the turret moves the Z axis first to the Z position, then the Y axis moves to the Y position, then the X axis to the X position. This field is available for TMX MY and TMX MYS Series machines.

• **FIRST Y THEN X THEN Z** \(F6\)—from the current position before the Position block, the Y axis moves first to the Y position, then the turret moves the X axis to the X position, then the Z axis to the Z position. This field is available for TMX MY, and TMX MYS Series machines.

• **FIRST Y THEN Z THEN X** \(F7\)—from the current position before the Position block, the Y axis moves first to the Y position, then the turret moves the Z axis to the Z position, then the X axis to the X position. This field is available for TMX MY and TMX MYS Series machines.

After the tool change is complete, the Position move is run using the other parameters listed.
Turning Data Blocks

A Turning softkey menu is available for all machines. In addition, this softkey menu appears on the WinMax Lathe desktop software.

From the New Block screen or Program Review screen, select the Turning F2 softkey to access the following types of turning data blocks:

Profile Data Block .................................................. 2 - 11
Threading Cycle ....................................................... 2 - 38
Grooving Cycle ....................................................... 2 - 51
Hole Cycle .............................................................. 2 - 58
Cutoff Block ............................................................ 2 - 62
Profile Data Block

Profile data blocks use elements to create different profiles in a part program. An element is any single or combined X-Z axis movement at a programmed feedrate. A series of profiles can be programmed in a single data block, using different elements, to form a complete profile.

To create a Profile from the New Block screen, select the TURNING ➔ F2 softkey followed by the PROFILE F1 softkey.

The Profile Start screen appears with tabs for Process and Geometry. The Process tab contains fields for programming the tool and cutting information; the Geometry tab contains fields for programming the shape of the profile.

If you open a program created using a software version prior to v1.1, the Profile Start screen will retain the Roughing and Finishing fields and values entered in the original program for backwards compatibility.

Process Tab

Program the tool and cutting information using the fields on the Process tab.

![Figure 2–3. Profile Start Process Tab screen](image)

The Profile Start screen opens with the fields ready to accept values. When you enter the tool number for a tool programmed in Tool Setup, a pop-up message appears asking if you want to use the defined feeds and speeds for the tool.

- Select **Yes** and the feed and speed values entered for the tool in Tool Setup are entered for this data block.
- Select **No** and enter the feeds and speeds for the tool.
For Finish passes, another pop-up message appears asking if you want to use the defined offsets in all following elements.

- Select Yes and the offsets defined for this tool in Tool Setup are entered in all elements for this data block.
- Select No and enter the offsets for each element.

You can program separate data blocks to perform some tasks, such as undercuts. You can also rough a part using multiple finish profiles by making multiple finish data blocks, each with different stock allowances. For example, the first finish data block would have a 0.050”, the next a 0.025”, and the last a 0.0” stock allowance.

The start points from the Profile Start screen carry forward to the next Element in the data block. Each Element’s start point is the previous Element’s end point.

The following fields appear on the Profile Start Process Tab screen:

- **TOOL**—contains the programmed tool number from Tool Setup.
- **TOOL OFFSETS**—identifies the programmed tool offset and orientation programmed in Tool Setup that will be used in the Profile for Roughing. You can accept or change this offset.
  
  When the cursor is located on this field, the COPY OFFSETS TO ALL ELEMENTS F6 softkey is available. Select this softkey, and a pop-up message appears asking if you would like to use the defined offsets in all following elements. Select the Yes screen button to use this offset in all elements that follow in this data block.

- **INSERT TIP RADIUS**—displays the insert tip radius programmed in Tool Setup. This field is read-only and can only be changed in Tool Setup for the tool number specified.

- **SPEED**—identifies the RPM value or CSS (FPM) value for this tool. This value can be accepted or changed.
  
  When the cursor is located on this field, the COPY SPEED TO ALL ELEMENTS F6 softkey is available. Select this softkey, and a pop-up message appears asking if you would like to use the defined speed in all following elements. Select the Yes screen button to use this speed in all elements that follow in this data block.

- **FEED**—identifies the programmed feed rate for this tool, either Inches or MM per Minute (IPM or MMPM) or Inches or MM per Revolution (IPR or MMPR). This value can be accepted or changed.
  
  When the cursor is located on this field, the COPY FEED TO ALL ELEMENTS F6 softkey is available. Select this softkey, and a pop-up message appears asking if you would like to use the defined feed in all following elements. Select the Yes screen button to use this feed in all elements that follow in this data block.
• **GOUGE AVOIDANCE**—select either **YES** or **NO** for the leading and trailing angles of the insert to be considered as part of the tool path, for example, turning a thread undercut.
  - Select **NO** and the insert stays inside the programmed geometry when it moves.
  - Select **YES** and the shape, size, and lead angle of the insert are considered when the tool path is determined, so the insert avoids the programmed geometry.

• **OPERATION**—identifies the type of path, rough or finish, with respect to the type of tool used for the profile. Refer to *Turn Rough Operation*, on page 2 - 14, *Turn Finish Operation*, on page 2 - 17, *Groove Rough Operation*, on page 2 - 20, or *Groove Finish Operation*, on page 2 - 22 for programming details.
  - **TURN ROUGH** F1—select if the profile is a rough pass using a turning tool.
  - **TURN FINISH** F2—select if the profile is a finish pass using a turning tool.
  - **GROOVE ROUGH** F3—select if the profile is a rough pass using a grooving tool.
  - **GROOVE FINISH** F4—select if the profile is a finish pass using a grooving tool.

To create a Finish Profile data block,
1. Program a Rough Profile data block (either Turn or Groove).
2. Select the Menu console button and select Program Review.
3. Highlight the Rough Profile data block.
4. Select the **MULTIPLE BLOCK FUNCTIONS** F1 softkey.
5. Select the **CREATE FINISH PROFILE** F7 softkey.

A Finish Profile block is created using the entries in the Rough Profile block with Turn Finish or Groove Finish entered for the Operation field.

For more than one finish profile in a data block, repeat the above steps.

For Turn Finish data blocks, Cutter Compensation is determined based on the Profile geometry. Stock Allowances are automatically set to 0.
If any of these changes are made for the Profile Operation:

- a Turn Rough operation is changed to a Turn Finish,
- a Groove Rough operation is changed to a Groove Finish,
- the CREATE FINISH PROFILE F7 softkey is used with a Turn Rough or Groove Rough operation,

the Finish Speed and Finish Feed values change to 0 for any Element associated with the Profile. The 0 values may be edited, and they are modal (they stay as the current value until changed in a subsequent element).

Finish Speed and Finish Feed values are inactive for Turn Rough or Groove Rough operations and cannot be edited.

The majority of the remaining fields change depending upon the type of operation being performed. The RETRACT ANGLE, RETRACT CLEARANCE, OVERRIDE TOLERANCE, and ELEMENT TOLERANCE fields appear for all operations.

**Turn Rough Operation**

Follow these rules when programming a Turn Rough operation:

- The first element and the last element must be perpendicular.
- The start and end point define the stock geometry.
- With Turn Rough selected, the X Rapid and Z Rapid position selected in the Geometry tab is recommended to be off the corner of the stock.

Refer to *Figure 2–17. Sample OD Operation Elements, on page 2–27* for examples.

The following fields appear when Turn Rough is selected in the Operation field:

*Figure 2–4. Profile Start Process Tab Turn Rough Operation screen*
• **DIRECTION**—identifies the cut direction for the rough pass as X+, X-, Z+, or Z-. This field appears when Turn Rough is selected for the Operation. Axis motion is shown in the figure below:

![Figure 2–5. Axes Motion](image)

<table>
<thead>
<tr>
<th>Chuck, Jaws, and Spindle</th>
</tr>
</thead>
<tbody>
<tr>
<td>2 Tool in turret moves toward part (-Z) or away from part (+Z) in spindle.</td>
</tr>
<tr>
<td>3 Tailstock</td>
</tr>
<tr>
<td>4 Tool in turret moves toward tailstock and chuck (-X) or away from tailstock and chuck (+X).</td>
</tr>
</tbody>
</table>

**Figure 2–6. Finish Tool Cutting Side default values based on Roughing Direction**

<table>
<thead>
<tr>
<th>Roughing Tool Orientation</th>
<th>Roughing Direction</th>
<th>Finish Tool Cutting Side</th>
</tr>
</thead>
<tbody>
<tr>
<td>![image]</td>
<td>Z- Z+</td>
<td>Left Right</td>
</tr>
<tr>
<td>![image]</td>
<td>Z- Z+</td>
<td>Right Left</td>
</tr>
<tr>
<td>![image]</td>
<td>X- X+</td>
<td>Left Right</td>
</tr>
<tr>
<td>![image]</td>
<td>X- X+</td>
<td>Right Left</td>
</tr>
</tbody>
</table>

• **DEPTH OF CUT**—identifies the total depth of cut of the tool into the part. This field appears when Turn Rough is selected for the Operation.
• **APPROACH CLEARANCE**—identifies the clearance of the tool as it approaches the stock. Used with the Approach Angle to determine a vector which defines how the tool moves toward the beginning of the profile. This field appears when Turn Rough or Turn Finish is selected for the Operation.

• **RETRACT ALONG PROFILE**—select either **YES** or **NO** for the tool to follow along the profile when retracting between Steps. Yes is the default. This field appears when Turn Rough is selected for the Operation. The figure below shows the tool path for both selections:

![Retract Along Profile](image)

*Figure 2–7. Tool Paths for Retract Along Profile selections*

• **RETRACT ANGLE**—pulls the tool off the profile between steps along a vector in the direction of retract angle, shown as 2 in *Figure 2–12. Groove Operation Cut Clearance and Retract Clearance example, on page 2 - 21*, if **NO** is selected for RETRACT ALONG PROFILE. This field appears for all rough and finish operations.

• **RETRACT CLEARANCE**—identifies the distance that the tool moves away from the wall, shown as 3 in *Figure 2–12. Groove Operation Cut Clearance and Retract Clearance example, on page 2 - 21*, before moving at Rapid to the start of the next pass. The default is 0.254 mm or 0.010 inch. This field appears for all rough and finish operations.

• **OVERRIDE TOLERANCE**—overrides the default tolerance, 0.00254 mm or 0.0001 inch, for elements in the current profile when Yes is selected. The ELEMENT TOLERANCE field is active when Yes is selected.

• **ELEMENT TOLERANCE**—identifies the amount of adjustment allowed if two elements of a profile fail to meet or intersect. If this occurs, the elements can be joined automatically if the required length is within this amount. The default value is 0.00254 mm (0.0001 in.). The maximum value is 0.0254 mm (0.001 in.). This field is not available when No is selected in the OVERRIDE TOLERANCE field.
**Turn Finish Operation**

A rule to follow when programming a Turn Finish operation is that a Finish Pass can move in any direction, i.e., a Turn Element can follow another Turn Element.

Refer to *Figure 2–17. Sample OD Operation Elements, on page 2-27* for an example of a Turn Element.

The following fields appear when Turn Finish is selected in the Operation field:

*Figure 2–8. Profile Start Process Tab Turn Finish Operation screen*
• **STRATEGY**—select either **ALONG PROFILE** or **ALL FACES FIRST** for the turn finish strategy. This field appears under the Operation field when Turn Finish is selected for the Operation.

• **CUTTING SIDE**—select cutter compensation side of the profile for the tool to cut. This field appears when Turn Finish is selected for the Operation.

  To determine which side is the cutting side, look in the direction the tool is heading and see which side of the line the cutter should be on.

• **NONE**—the tool tip follows the programmed geometry.

• **RIGHT**—OD Profiles typically cut on the Right side.

• **LEFT**—ID Profiles typically cut on the Left side. OD tools cutting the face toward the centerline typically cut on the Left side. Boring Bars also typically cut on the Left side.

The control assigns a default Cutting Side for a Finish Profile following a Roughing Profile based on the selected Direction. When the orientation and roughing direction are identified as shown in the following table, the corresponding cutting side value is automatically entered in the Cutting Side field of the Finish Operation block:

<table>
<thead>
<tr>
<th>Roughing Tool Orientation</th>
<th>Roughing Direction</th>
<th>Finish Tool Cutting Side</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Z- Z+</td>
<td>Left Right</td>
</tr>
<tr>
<td></td>
<td>Z- Z+</td>
<td>Right Left</td>
</tr>
<tr>
<td></td>
<td>X- X+</td>
<td>Left Right</td>
</tr>
<tr>
<td></td>
<td>X- X+</td>
<td>Right Left</td>
</tr>
</tbody>
</table>

*Figure 2–9. Finish Tool Cutting Side default values based on Roughing Direction*
• **SPRING PASSES**—identifies the number of repetitions the finish tool will make at the final finish depth. This field appears when Turn Finish is selected for the Operation.

• **APPROACH ANGLE**—identifies the angle the tool approaches the stock. Used with the Approach Clearance to determine a vector which defines how the tool moves toward the beginning of the profile. This field appears when Turn Finish is selected for the Operation. The figure below shows a 45° approach angle for each Step:

![Figure 2–10. Tool Path 45° Approach Angle](image)

• **APPROACH CLEARANCE**—identifies the clearance of the tool as it approaches the stock. Used with the Approach Angle to determine a vector which defines how the tool moves toward the beginning of the profile. This field appears when Turn Rough or Turn Finish is selected for the Operation.

• **RETRACT ANGLE**—pulls the tool off the profile between steps along a vector in the direction of retract angle if NO is selected for RETRACT ALONG PROFILE, shown as 2 in Figure 2–12. Groove Operation Cut Clearance and Retract Clearance example, on page 2 - 21. The default is 45°. This field appears for all rough and finish operations.

• **RETRACT CLEARANCE**—identifies the distance that the tool moves away from the wall, shown as 3 in Figure 2–12. Groove Operation Cut Clearance and Retract Clearance example, on page 2 - 21, before moving at Rapid to the start of the next pass. The default is 0.254 mm or 0.010 inch. This field appears for all rough and finish operations.

• **OVERRIDE TOLERANCE**—overrides the default tolerance, 0.00254 mm or 0.0001 inch, for elements in the current profile when Yes is selected. The ELEMENT TOLERANCE field is active when Yes is selected.
• **ELEMENT TOLERANCE**—identifies the amount of adjustment allowed if two elements of a profile fail to meet or intersect. If this occurs, the elements can be joined automatically if the required length is within this amount. The default value is 0.00254 mm (0.0001 in.). The maximum value is 0.0254 mm (0.001 in.). This field is not available when No is selected in the OVERRIDE TOLERANCE field.

**Groove Rough Operation**

A rule to follow when programming a Groove Rough operation is that the first element and the last element must be parallel. It is also recommended to have the first and last elements start and end at the same Diameter (or Radius) or Z location.

Refer to *Figure 2–17. Sample OD Operation Elements, on page 2 - 27* for an example of a Turn Element.

The following fields appear when Groove Rough is selected in the Operation field:

- **STEP OVER**—identifies the distance the tool steps over each pass between grooving passes. This field appears when Groove Rough is selected for the Operation.
- **CUT CLEARANCE**—identifies the amount of clearance above the groove for cutting each profile, shown as 1 in the figure below. The default is 1.27 mm or 0.05 inch. This field appears when Groove Rough or Groove Finish is selected for the Operation.
- **RETRACT ANGLE**—pulls the tool off the profile between steps along a vector in the direction of retract angle, shown as 2 in the figure below, if NO is selected for RETRACT ALONG PROFILE. The default is 45°. This field appears for all rough and finish operations.
• **RETRACT CLEARANCE**—identifies the distance that the tool moves away from the wall, shown as 3 in the figure below, before moving at Rapid to the start of the next pass. The default is 0.254 mm or 0.010 inch. This field appears for all rough and finish operations.

![Figure 2–12. Groove Operation Cut Clearance and Retract Clearance example](image)

<table>
<thead>
<tr>
<th></th>
<th>Cut Clearance</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>Retract Angle</td>
</tr>
<tr>
<td>3</td>
<td>Retract Clearance</td>
</tr>
</tbody>
</table>

• **OVERRIDE TOLERANCE**—overrides the default tolerance, 0.0025 mm or 0.0001 inch, for elements in the current profile when Yes is selected. The ELEMENT TOLERANCE field is active when Yes is selected.

• **ELEMENT TOLERANCE**—identifies the amount of adjustment allowed if two elements of a profile fail to meet or intersect. If this occurs, the elements can be joined automatically if the required length is within this amount. The default value is 0.00254 mm (0.0001 in.). The maximum value is 0.0254 mm (0.001 in.). This field is not available when No is selected in the OVERRIDE TOLERANCE field.
Groove Finish Operation

The following fields appear when Groove Finish is selected in the Operation field:

- **STRATEGY**—select either **END TO MIDDLE** or **END TO END** for the groove finish strategy. This field appears under the Gouge Avoidance field when Groove Finish is selected for the Operation.

- **CUT CLEARANCE**—identifies the amount of clearance for cutting each profile, shown as 1 in the figure below. The default is 1.27 mm or 0.05 inch. This field appears when Groove Rough or Groove Finish is selected for the Operation.

- **RETRACT ANGLE**—pulls the tool off the profile between steps along a vector in the direction of retract angle, shown as 2 in the figure below, if **NO** is selected for **RETRACT ALONG PROFILE**. The default is 45°. This field appears for all rough and finish operations.
• **RETRACT CLEARANCE**—identifies the distance that the tool moves away from the wall, shown as 3 in the figure below. The default is 2.54 mm or 0.100 inch, before moving at Rapid to the start of the next pass. The default is 0.254 mm or 0.010 inch. This field appears for all rough and finish operations.

![Figure 2–14. Groove Operation Cut Clearance and Retract Clearance example](image)

1. Cut Clearance
2. Retract Angle
3. Retract Clearance

• **OVERRIDE TOLERANCE**—overrides the default tolerance, 0.0025 mm or 0.0001 inch, for elements in the current profile when Yes is selected. The ELEMENT TOLERANCE field is active when Yes is selected.

• **ELEMENT TOLERANCE**—identifies the amount of adjustment allowed if two elements of a profile fail to meet or intersect. If this occurs, the elements can be joined automatically if the required length is within this amount. The default value is 0.00254 mm (0.0001 in.). The maximum value is 0.0254 mm (0.001 in.). This field is not available when No is selected in the OVERRIDE TOLERANCE field.
**Geometry Tab**

Describe the finish shape with the fields in the Geometry tab.

The program determines how many rough passes to make to remove the material based on the entries in the Process tab fields. Finishing follows the geometry of the profile defined in the Geometry tab.

![Profile Start Geometry Tab screen](image)

*Figure 2–15. Profile Start Geometry Tab screen*

The following fields appear on the Profile Start Geometry Tab screen:

- **X RAPID (D)** or **(R)**—identifies the X axis (DIAMETER) or (RADIUS) starting location, near the material, shown in the figure below.
- **Z RAPID**—identifies the Z axis starting location, near the material, shown in the figure below.

![X Rapid and Z Rapid Location example](image)

*Figure 2–16. X Rapid and Z Rapid Location example*
• **X START (D) or (R)**—defines the X (DIAMETER) or (RADIUS) startpoint for the first element of the profile.

• **Z START**—defines the Z startpoint for the first element of the profile.

• **STOCK ALLOWANCE**—identifies the amount of material to leave on the profile, applying stock to Tapers, Chamfers, and Arcs. The control calculates the TURN ALLOWANCE and FACE ALLOWANCE fields when you enter a value in this field.

• **TURN ALLOWANCE**—identifies the amount of material to leave on horizontal (turn) surfaces. This field is calculated by the control when the Stock Allowance is entered; however, you can edit the value to fine tune the data.

• **FACE ALLOWANCE**—identifies the amount of material to leave on vertical (face) surfaces. This field is calculated by the control when the Stock Allowance is entered; however, you can edit the value to fine tune the data.

### Elements

The first element in a Profile data block is always a Profile Start indicated by the Element zero (0). The last element in a Profile data block is always a Profile End. Element numbers are system generated, and you cannot change them. As you insert elements for the profile, the numbers increment and the Profile End changes to the last number in the sequence.

The Calc-Assist feature calculates certain unknown element dimensions automatically, after sufficient data has been entered. A field with a calculated value is indicated with “CAL” preceding the value. Since elements are dependent on adjacent blocks for calculations or end points, you cannot copy or paste individual elements.

These softkeys appear for creating and editing Elements in a Profile:

- **PREVIOUS ELEMENT F1**—access the previous element in the current Profile data block. This softkey is not active when Element 0 is active.

- **NEXT ELEMENT F2**—accesses the next element in the current Profile data block. This softkey is not active in the Profile End screen.

- **DELETE ELEMENT F3**—deletes the current element from the current Profile data block. This softkey is not active when either Element 0 or Profile End is active.

- **INSERT ELEMENT BEFORE F7**—adds a new element in front of the current element in the current Profile data block. This softkey is not active when Element 0 is active.
To create an Element in the Profile data block,

1. The cursor is initially positioned in the Block field. Press **Enter** or the **Down Arrow** to move to the Element field. The cursor returns to the Element field when the last field is edited on the screen.

2. Select the NEXT ELEMENT F2 softkey. The Element menu softkeys appear: **FACE F1, TAPER F2, TURN F3, ARC F4, BLEND ARC F5, CHAMFER F6**. When Rough Turning, Element 1 must be either a Face or a Turn operation.
   
   - If Element 1 is a Face, the last element before the Profile End must be a Turn.
   - If Element 1 is a Turn, the last element before the Profile End must be a Face.

3. Select one of the Element softkeys. The Profile screen appears with values carried forward from the Profile Start screen or from previous Elements.

Refer to **Turn Element, on page 2 - 28, Face Element, on page 2 - 29, Taper Element, on page 2 - 30, Arc Element, on page 2 - 32, Blend Arc Element, on page 2 - 34, or Chamfer Element, on page 2 - 36** for programming details.

Element operations can be programmed by defining profiles along the X and Z axis. Blend Arcs and Chamfers can be inserted between Turn, Face, and Taper operations.
The following figure illustrates elements for an OD operation:

![Figure 2–17. Sample OD Operation Elements](image)

<table>
<thead>
<tr>
<th></th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Turn</td>
</tr>
<tr>
<td></td>
<td>First Element</td>
</tr>
<tr>
<td></td>
<td>start point</td>
</tr>
<tr>
<td>2</td>
<td>Face</td>
</tr>
<tr>
<td>3</td>
<td>Chamfer</td>
</tr>
<tr>
<td>4</td>
<td>Turn</td>
</tr>
<tr>
<td>5</td>
<td>Blend Arc</td>
</tr>
<tr>
<td>6</td>
<td>Face</td>
</tr>
<tr>
<td>7</td>
<td>Chamfer</td>
</tr>
<tr>
<td>8</td>
<td>Turn</td>
</tr>
<tr>
<td>9</td>
<td>Face</td>
</tr>
<tr>
<td></td>
<td>Last Element</td>
</tr>
<tr>
<td></td>
<td>end point</td>
</tr>
<tr>
<td>10</td>
<td>X, Z Rapid Position</td>
</tr>
<tr>
<td>11</td>
<td>Stock Boundary</td>
</tr>
</tbody>
</table>

*Figure 2–17. Sample OD Operation Elements*
**Turn Element**

A Turn operation cuts the stock horizontally.

![Figure 2–18. Turn Element screen](image)

These fields appear on the Turn Element screen:

- **X END (D) or (R)**—contains the X (DIAMETER) or (RADIUS) coordinate for the ending point of the Turn. The value in this read-only field is carried over from the Profile Start or the previous element.
- **Z END**—identifies the Z coordinate for the ending point.
- **LENGTH**—contains the control-calculated X-Z length of the element. This field is read-only and “CAL” appears next to it.
- **ANGLE**—contains the control-calculated X-Z angle from the start point to the end point, measured counterclockwise from the home position. This field is read-only and “CAL” appears next to it.
- **X START (D) or (R)**—defines the X (DIAMETER) or (RADIUS) startpoint for the element. The value in this read-only field is programmed in the previous element or Profile Start Geometry Tab screen.
- **Z START**—defines the Z startpoint for the element. The value in this read-only field is programmed in the previous element or Profile Start Geometry Tab screen.
- **FINISH SPEED (FPM) or (MPM)**—identifies the speed for the finish pass. This field is only active when Turn Finish or Groove Finish is selected in the Profile Start Process Operation field.
- **FINISH FEED (IPR) or (MMPR)**—identifies the feedrate for the finish pass. This field is only active when Turn Finish or Groove Finish is selected in the Profile Start Process Operation field.
• **FINISH OFFSETS**—identifies the offset and the orientation for the finish pass. This field is only active when Turn Finish or Groove Finish is selected in the Profile Start Process Operation field.

![Warning](https://via.placeholder.com/150)

When the cursor is located on either the active FINISH SPEED (FPM) or (MPM), FINISH FEED (IPR) or (MMPR), or FINISH OFFSETS field, the COPY SPEED TO ALL ELEMENTS F6, COPY FEED TO ALL ELEMENTS F6, or COPY OFFSETS TO ALL ELEMENTS F6 softkey is available, respectively.

Select the softkey, and a pop-up message appears asking if you would like to use the defined speed, feed, or offset in all following elements. Select Yes to use this speed, feed, or offset in all elements that follow in this data block.

**Face Element**

A Face operation cuts the stock vertically.

![Figure 2–19. Face Element screen](https://via.placeholder.com/150)

These fields appear on the Face Element screen:

- **X END** *(D)* or *(R)*—identifies the X (DIAMETER) or (RADIUS) coordinate for the ending point of the Face.
- **Z END**—contains the Z coordinate for the ending point. The value in this read-only field is carried over from the previous element.
- **LENGTH**—contains the control-calculated X-Z length of the element. This field is read-only and “CAL” appears next to it.
- **ANGLE**—contains the control-calculated X-Z angle from the start point to the end point, measured counterclockwise from the 3 o’clock position. This field is read-only and “CAL” appears next to it.
• **X START (D) or (R)**—defines the X (DIAMETER) or (RADIUS) startpoint for the element. The value in this read-only field is carried over from the previous element.

• **Z START**—defines the Z startpoint for the first element of the profile. The value in this read-only field is carried over from the previous element.

• **FINISH SPEED (FPM) or (MPM)**—identifies the speed for the finish pass. This field is only active when Turn Finish or Groove Finish is selected in the Profile Start Process Operation field.

• **FINISH FEED (IPR) or (MMPR)**—identifies the feedrate for the finish pass. This field is only active when Turn Finish or Groove Finish is selected in the Profile Start Process Operation field.

• **FINISH OFFSETS**—identifies the offset and the orientation for the finish pass. This field is only active when Turn Finish or Groove Finish is selected in the Profile Start Process Operation field.

When the cursor is located on either the active FINISH SPEED (FPM) or (MPM), FINISH FEED (IPR) or (MMPR), or FINISH OFFSETS field, the COPY SPEED TO ALL ELEMENTS F6, COPY FEED TO ALL ELEMENTS F6, or COPY OFFSETS TO ALL ELEMENTS F6 softkey is available, respectively.

Select the softkey, and a pop-up message appears asking if you would like to use the defined speed, feed, or offset in all following elements. Select Yes to use this speed, feed, or offset in all elements that follow in this data block.

---

**Taper Element**

A Taper operation cuts the stock at an angle.

![Figure 2–20. Taper Element screen](image-url)
These fields appear on the Taper Element screen:

- **X END (D) or (R)**—identifies the X (DIAMETER) or (RADIUS) coordinate for the ending point of the Taper.

- **Z END**—identifies the Z coordinate for the ending point.

  When the cursor is located in either the **X END (D) or (R)** field or the **Z END** field, the **FIND ANOTHER END POINT F7** softkey appears. Select this softkey and the control calculates a second end point to provide you with an alternative. You can choose the geometry that best suits your needs.

- **LENGTH**—contains the control-calculated X-Z length of the element. This value can be accepted or changed.

- **ANGLE**—contains the control-calculated X-Z angle from the start point to the end point, measured counterclockwise from the 3 o'clock position. This value can be accepted or changed.

- **X START (D) or (R)**—defines the X (DIAMETER) or (RADIUS) startpoint for the element. The value in this read-only field is carried over from the previous element.

- **Z START**—defines the Z startpoint for the first element of the profile. The value in this read-only field is carried over from the previous element.

- **FINISH SPEED (FPM) or (MPM)**—identifies the speed for the finish pass. This field is only active when Turn Finish or Groove Finish is selected in the Profile Start Process Operation field.

- **FINISH FEED (IPR) or (MMPR)**—identifies the feedrate for the finish pass. This field is only active when Turn Finish or Groove Finish is selected in the Profile Start Process Operation field.

- **FINISH OFFSETS**—identifies the offset and the orientation for the finish pass. This field is only active when Turn Finish or Groove Finish is selected in the Profile Start Process Operation field.

  When the cursor is located on either the active **FINISH SPEED (FPM) or (MPM)**, **FINISH FEED (IPR) or (MMPR)**, or **FINISH OFFSETS** field, the **COPY SPEED TO ALL ELEMENTS F6**, **COPY FEED TO ALL ELEMENTS F6**, or **COPY OFFSETS TO ALL ELEMENTS F6** softkey is available, respectively.

  Select the softkey, and a pop-up message appears asking if you would like to use the defined speed, feed, or offset in all following elements. Select Yes to use this speed, feed, or offset in all elements that follow in this data block.
**Arc Element**

An Arc operation cuts the stock in an arc pattern from one point back to another point to form a rounded corner.

![Arc Element screen](image)

*Figure 2–21. Arc Element screen*

These fields appear on the Arc Element screen:

- **X END (D) or (R)**—identifies the X (DIAMETER) or (RADIUS) coordinate for the ending point of the Arc.
- **Z END**—identifies the Z coordinate for the ending point of the Arc.

⚠️ When the cursor is located in either the **X END (D) or (R) field** or the **Z END field**, the **FIND ANOTHER END POINT F7** softkey appears. Select this softkey and the control calculates a second end point to provide you with an alternative. You can choose the geometry that best suits your needs.

- **DIRECTION**—identifies the direction of the arc from the start point. Select CW (clockwise) or CCW (counterclockwise).
- **X CENTER (D) or (R)**—contains the X coordinate of the arc’s center point.
- **Z CENTER**—contains the Z coordinate of the arc’s center point.

⚠️ When the cursor is located in either the **X CENTER (D) or (R) field** or the **Z CENTER field**, the **FIND ANOTHER CENTER POINT F7** softkey appears. Select this softkey and the control calculates a second center point to provide you with an alternative. You can choose the geometry that best suits your needs.
• **RADIUS**—contains the radius of the arc.

⚠ If the only information known about an arc is its radius, it is easier to program it as a blend arc, if the elements intersect. Refer to *Blend Arc Element, on page 2 - 34* for information about the blend arc element.

• **X START (D) or (R)**—defines the X (DIAMETER) or (RADIUS) startpoint for the element. The value in this read-only field is carried over from the previous element.

• **Z START**—defines the Z startpoint for the first element of the profile. The value in this read-only field is carried over from the previous element.

• **FINISH SPEED (FPM) or (MPM)**—identifies the speed for the finish pass. This field is only active when Turn Finish or Groove Finish is selected in the Profile Start Process Operation field.

• **FINISH FEED (IPR) or (MMPR)**—identifies the feedrate for the finish pass. This field is only active when Turn Finish or Groove Finish is selected in the Profile Start Process Operation field.

• **FINISH OFFSETS**—identifies the offset and the orientation for the finish pass. This field is only active when Turn Finish or Groove Finish is selected in the Profile Start Process Operation field.

⚠ When the cursor is located on either the active FINISH SPEED (FPM) or (MPM), FINISH FEED (IPR) or (MMPR), or FINISH OFFSETS field, the COPY SPEED TO ALL ELEMENTS F6, COPY FEED TO ALL ELEMENTS F6, or COPY OFFSETS TO ALL ELEMENTS F6 softkey is available, respectively.

Select the softkey, and a pop-up message appears asking if you would like to use the defined speed, feed, or offset in all following elements. Select Yes to use this speed, feed, or offset in all elements that follow in this data block.
Blend Arc Element

A Blend Arc operation creates a curved line that joins two other elements and is tangent to both.

A blend arc can be used to join these types of elements:

- two Face elements.
- a Face element and a Turn element.
- two Arc elements.
- a Taper to another element.

The elements 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 elements intersect.

Here are some guidelines that must be followed when creating a blend arc:

- The first or last element of a Profile data block cannot be blend arc element.
- Blend arc or chamfer elements cannot be adjacent to one another in a program. For example, if Element 2 is a blend arc, neither Element 1 nor 3 can be blend arc elements.
- Elements that are adjacent to the blend arc element must intersect at some point in their theoretical plane. Therefore, if Element 2 is a blend arc, Elements 1 and 3 must theoretically intersect at some projected point.
- The radius of a blend arc element cannot be too large to be tangent to both of the adjoining elements.
- If any coordinate (start point, center point, or end point) is important to the construction of the two elements to be blended, the element must be programmed as an arc and not as a blend arc.
• The Feed field is initially displayed with a value carried forward from the previous element. This value can be accepted or changed.

These fields are available for programming a Blend Arc Element:

• **X END (D) or (R)**—contains the X (DIAMETER) or (RADIUS) coordinate for the ending point of the blend arc. The value in this read-only field is carried over from the previous element.

• **Z END**—contains the Z coordinate for the ending point of the blend arc. The value in this read-only field is carried over from the previous element.

• **DIRECTION**—contains the direction of the blend arc from the start point, CW (clockwise) or CCW (counterclockwise). The value in this read-only field is carried over from the previous element.

• **X CENTER (D) or (R)**—contains the X (DIAMETER) or (RADIUS) coordinate of the blend arc’s center point. The value in this read-only field is carried over from the previous element.

• **Z CENTER**—contains the Z coordinate of the blend arc’s center point. The value in this read-only field is carried over from the previous element.

• **RADIUS**—identifies the radius of the blend arc.

• **X START (D) or (R)**—defines the X (DIAMETER) or (RADIUS) startpoint for the element. The value in this read-only field is carried over from the previous element.

• **Z START**—defines the Z startpoint for the first element of the profile. The value in this read-only field is carried over from the previous element.

• **FINISH SPEED (FPM) or (MPM)**—identifies the speed for the finish pass. This field is only active when Turn Finish or Groove Finish is selected in the Profile Start Process Operation field.

• **FINISH FEED (IPR) or (MMPR)**—identifies the feedrate for the finish pass. This field is only active when Turn Finish or Groove Finish is selected in the Profile Start Process Operation field.

• **FINISH OFFSETS**—identifies the offset and the orientation for the finish pass. This field is only active when Turn Finish or Groove Finish is selected in the Profile Start Process Operation field.

When the cursor is located on either the active **FINISH SPEED (FPM) or (MPM), FINISH FEED (IPR) or (MMPR), or FINISH OFFSETS** field, the **COPY SPEED TO ALL ELEMENTS F6, COPY FEED TO ALL ELEMENTS F6, or COPY OFFSETS TO ALL ELEMENTS F6** softkey is available, respectively.

Select the softkey, and a pop-up message appears asking if you would like to use the defined speed, feed, or offset in all following elements. Select Yes to use this speed, feed, or offset in all elements that follow in this data block.
**Chamfer Element**

A Chamfer operation creates a straight line that joins two other elements and is tangent to both.

![Chamfer Element screen](image)

**Figure 2–23. Chamfer Element screen**

A chamfer can be used to join these types of elements:

- two Face elements.
- a Face element and a Turn element.

The elements to be joined must have a theoretical point of intersection.

Here are some guidelines that must be followed when creating a chamfer:

- The first or last element of a Profile data block cannot be chamfer elements.
- Chamfer elements can only be adjacent to turn and face elements.
- Blend arc or chamfer elements cannot be adjacent to one another in a program. For example, if Element 2 is a blend arc, neither Element 1 nor 3 can be blend arc elements.
- Elements that are adjacent to the chamfer element must intersect at some point in their theoretical plane. Therefore, if Element 2 is a chamfer, Elements 1 and 3 must theoretically intersect at some projected point.
- The length of a chamfer element cannot be too large and intersect the two adjoining elements.
- If any coordinate (start point, center point, or end point) is important to the construction of the two elements to be chamfered, the element must be programmed as a taper and not as a chamfer.
These fields are available for programming a Chamfer Element:

- **X END (D) or (R)**—identifies the X (DIAMETER) or (RADIUS) coordinate for the ending point of the Face. The value in this read-only field is carried over from the previous element.
- **Z END**—contains the Z coordinate for the ending point. The value in this read-only field is carried over from the previous element.
- **CHAMFER**—contains the X-Z length of the element.
- **ANGLE**—contains the X-Z angle to the end point carried over from the previous element. The default for this field is 45.000 (45°).
- **X START (D) or (R)**—defines the X (DIAMETER) or (RADIUS) startpoint for the element. The value in this read-only field is carried over from the previous element.
- **Z START**—defines the Z startpoint for the first element of the profile. The value in this read-only field is carried over from the previous element.
- **FINISH SPEED (FPM) or (MPM)**—identifies the speed for the finish pass. This field is only active when Turn Finish or Groove Finish is selected in the Profile Start Process Operation field.
- **FINISH FEED (IPR) or (MMPR)**—identifies the feedrate for the finish pass. This field is only active when Turn Finish or Groove Finish is selected in the Profile Start Process Operation field.
- **FINISH OFFSETS**—identifies the offset and the orientation for the finish pass. This field is only active when Turn Finish or Groove Finish is selected in the Profile Start Process Operation field.

⚠️ When the cursor is located on either the active FINISH SPEED (FPM) or (MPM), FINISH FEED (IPR) or (MMPR), or FINISH OFFSETS field, the COPY SPEED TO ALL ELEMENTS F6, COPY FEED TO ALL ELEMENTS F6, or COPY OFFSETS TO ALL ELEMENTS F6 softkey is available, respectively.

Select the softkey, and a pop-up message appears asking if you would like to use the defined speed, feed, or offset in all following elements. Select Yes to use this speed, feed, or offset in all elements that follow in this data block.
Threaded Cycle

The Threading cycle performs multi-pass threading for OD (outside diameter) and ID (inside diameter) threads. This cycle supports taper and face threads, lead in and lead out angles, multiple spring passes, and various methods of specifying depth of cut.

⚠️ A Position block is required before a Threading Cycle if there is not sufficient clearance for the tool to move around the part from a previous data block.

To create a Profile from the New Block screen, select the TURNING ➔ \( F2 \) softkey followed by the THREAD \( F2 \) softkey.

The THREADING CYCLE screen appears with tabs for Process and Geometry. The fields on the Process tab are for programming the tool and cutting information; the fields on the Geometry tab are for programming the shape of the thread.

**Process**

Program the tool and cutting information using the Process tab:

![Threading Cycle screen Process tab](image)

- **TOOL**—contains the programmed tool number from Tool Setup.
- **TOOL OFFSETS**—identifies the programmed tool offset and orientation programmed in Tool Setup that will be used. You can accept or change this offset.
- **INSERT TIP RADIUS**—displays the insert tip radius programmed in Tool Setup. This field is read-only.
• **SPINDLE RPM**—contains the default spindle speed programmed in Tool Setup. This value can be accepted or changed.

• **Use Safe Rapid Point** checkbox—when selected, activates the X Rapid and Z Rapid fields for entering the location for the X and Z axes to move at rapid feedrate at the start and end of the cycle.

• **X RAPID**—identifies the X axis rapid feedrate location for the start and end of this cycle. This field is only available for editing when the Use Safe Rapid Point checkbox is selected.

• **Z RAPID**—identifies the Z axis rapid feedrate location for the start and end of this cycle. This field is only available for editing when the Use Safe Rapid Point checkbox is selected.

• **STRATEGY**—identifies the manner in which the total thread depth is divided among passes.

  • **DECREASING DEPTH**—select so the depth of cut will decrease upon each rough pass. This method controls the amount of depth of cut for each pass using the entries in the Rough Start Depth and Rough Finish Depth fields.

  • **CONSTANT VOLUME**—select so the depth of cut will be the same for all rough passes. This strategy uses Constant Volume Removal and maintains a constant load on the cutting tool. The system calculates the amount of material removed by each pass and sets the depth for material removal equal to the amount entered in the Rough Start Depth field. The Rough Final Depth field is inactive when Constant Chipload is selected.

• **ROUGH START DEPTH**—identifies the depth for the rough start pass.

• **ROUGH FINAL DEPTH**—identifies the depth for the final rough pass. This field is not active with Constant Chipload selected for the Strategy.

• **FINISH PASSES**—identifies the number of finish passes the tool will make.

• **DEPTH OF CUT**—identifies the amount of material the tool will remove for each finish pass.

• **SPRING PASSES**—identifies the number of repetitions the finish tool will make at the final finish depth, in addition to the finish passes.

• **NUMBER OF PASSES**—identifies the number of threading passes. This read-only field is calculated by the control.
Geometry

Define the shape of the thread using the fields in the Geometry tab.

Figure 2–25. Threading Cycle screen Geometry tab

- **THREAD TYPE**—identifies the type of thread to cut. Refer to
  - *OD Straight Thread Type, on page 2 - 41,*
  - *OD Taper Thread Type, on page 2 - 43,*
  - *ID Straight Thread Type, on page 2 - 45,*
  - *ID Taper Thread Type, on page 2 - 47,*
  - *Face Thread Type, on page 2 - 49.*

These types of threads are illustrated in the following figure:

![Types of Threads](image)

*Figure 2–26. Types of Threads*
**OD Straight Thread Type**

The following fields are available when OD Straight is selected in the Thread Type field:

![Threading Cycle Geometry Tab OD Straight Thread Type](image)

- **Z START**—identifies the Z axis starting location for the Threading cycle. The tool begins cutting at the programmed feedrate at this location. When the Thread Type is Face, this field changes to X Start (DIA) or (RAD).
- **Z END**—identifies the Z axis end location for the Threading cycle. The tool stops cutting at this location. When the Thread Type is Face, this field changes to X End (DIA) or (RAD).
- **Z CLEARANCE**—identifies the distance that the tool should be moved away from the part after the Threading cycle.
- **X CLEARANCE**—identifies the distance that the tool should be raised from the part after the Threading cycle.
- **MAJOR DIAMETER**—indicates the outer diameter of the part. This diameter is shown as 1 in the following figure.
- **MINOR DIAMETER**—indicates the inner diameter of the part. This diameter is shown as 2 in the following figure.

![Major Diameter (1) and Minor Diameter (2)](image)
- **PITCH**—defines the distance between threads for metric tools. Do not use this field for programming tools measured in threads per inch.

- **LEAD (TPI)**—defines the threads per inch for tools measured in inches. Do not use this field for programming metric tools.

- **VERTICAL LEAD IN**—defines whether the tool should enter the part vertically (perpendicular) to the direction of the cut. Select YES or NO. YES is the default. When YES is selected, the LEAD IN ANGLE field is inactive.

- **LEAD IN ANGLE**—defines the infeed angle that the tool follows in the X axis to enter the workpiece. This angle is shown as 1 in the following figure; the tool path is represented with a dotted line. The default is 29.5.

- **VERTICAL LEAD OUT**—defines whether the tool should exit the part vertically (perpendicular) to the direction of the cut. Select YES or NO. NO is the default. When YES is selected, the LEAD OUT ANGLE field is inactive.

- **LEAD OUT ANGLE**—defines the pullout angle that the tool follows in the X axis to exit the workpiece. This angle is shown as 2 in the following figure; the tool path is represented with a dotted line. The default is 30.0.

![Figure 2–29. Lead In Angle (1) and Lead Out Angle (2)](image)

- **SPECIFY START ANGLE**—select YES or NO to indicate whether a start angle is required. When NO is selected, the START ANGLE field is inactive.

- **START ANGLE**—defines the Start Angle if YES is selected for SPECIFY START ANGLE. The default is 0°.

- **NUMBER OF STARTS**—defines the number of starts if multiple starts are required and NO is selected for SPECIFY START ANGLE.
**OD Taper Thread Type**

The following fields are available when OD Taper is selected in the Thread Type field:

- **Z START**—identifies the Z axis starting location for the Threading cycle. The tool begins cutting at the programmed feedrate at this location.
- **Z END**—identifies the Z axis end location for the Threading cycle. The tool stops cutting at this location.
- **Z CLEARANCE**—identifies the distance that the tool should be moved away from the part after the Threading cycle.
- **X CLEARANCE**—identifies the distance that the tool should be raised from the part after the Threading cycle.
- **MAJOR DIAMETER**—indicates the outer diameter of the part. This diameter is shown as 1 in the following figure.
- **MINOR DIAMETER**—indicates the inner diameter of the part. This diameter is shown as 2 in the following figure.
• **PITCH**—defines the distance between threads for metric tools. Do not use this field for programming tools measured in threads per inch.

• **LEAD (TPI)**—defines the threads per inch for tools measured in inches. Do not use this field for programming metric tools.

• **X TAPER (DIA) or (RAD)**—defines the amount of taper in the X axis. When you enter a value in this field, the control calculates the value for the Taper Angle. This field only appears when the Thread Type is OD Taper or ID Taper.

• **TAPER ANGLE**—defines the angle of the taper. When you enter a value in this field, the control calculates the value for the X Taper field. This field only appears when the Thread Type is OD Taper or ID Taper

• **VERTICAL LEAD IN**—defines whether the tool should enter the part vertically (perpendicular) to the direction of the cut. Select YES or NO. NO is the default. When YES is selected, the LEAD IN ANGLE field is inactive.

• **LEAD IN ANGLE**—defines the infeed angle that the tool follows in the X axis to enter the workpiece. This angle is shown as 1 in the following figure; the tool path is represented with a dotted line.

• **VERTICAL LEAD OUT**—defines whether the tool should exit the part vertically (perpendicular) to the direction of the cut. Select YES or NO. NO is the default. When YES is selected, the LEAD OUT ANGLE field is inactive.

• **LEAD OUT ANGLE**—defines the pullout angle that the tool follows in the X axis to exit the workpiece. This angle is shown as 2 in the following figure; the tool path is represented with a dotted line.

![Figure 2–32. Lead In Angle (1) and Lead Out Angle (2)](image)

• **SPECIFY START ANGLE**—select YES or NO to indicate whether a start angle is required. When NO is selected, the START ANGLE field is inactive.

• **START ANGLE**—defines the Start Angle if YES is selected for SPECIFY START ANGLE. The default is 0°.

• **NUMBER OF STARTS**—defines the number of starts if multiple starts are required and NO is selected for SPECIFY START ANGLE.
ID Straight Thread Type

The following fields are available when ID Straight is selected in the Thread Type field:

- **Z START**—identifies the Z axis starting location for the Threading cycle. The tool begins cutting at the programmed feedrate at this location.
- **Z END**—identifies the Z axis end location for the Threading cycle. The tool stops cutting at this location.
- **Z CLEARANCE**—identifies the distance that the tool should be moved away from the part after the Threading cycle.
- **X CLEARANCE**—identifies the distance that the tool should be raised from the part after the Threading cycle.
- **MAJOR DIAMETER**—indicates the outer diameter of the part. This diameter is shown as 1 in the following figure.
- **MINOR DIAMETER**—indicates the inner diameter of the part. This diameter is shown as 2 in the following figure.

![Figure 2–33. Threading Cycle Geometry Tab ID Straight Thread Type](image)

![Figure 2–34. Major Diameter (1) and Minor Diameter (2)](image)
- **PITCH**—defines the distance between threads for metric tools. Do not use this field for programming tools measured in threads per inch.
- **LEAD (TPI)**—defines the threads per inch for tools measured in inches. Do not use this field for programming metric tools.
- **VERTICAL LEAD IN**—defines whether the tool should enter the part vertically (perpendicular) to the direction of the cut. Select YES or NO. NO is the default. When YES is selected, the LEAD IN ANGLE field is inactive.
- **LEAD IN ANGLE**—defines the infeed angle that the tool follows in the X axis to enter the workpiece. This angle is shown as 1 in the following figure; the tool path is represented with a dotted line.
- **VERTICAL LEAD OUT**—defines whether the tool should exit the part vertically (perpendicular) to the direction of the cut. Select YES or NO. NO is the default. When YES is selected, the LEAD OUT ANGLE field is inactive.
- **LEAD OUT ANGLE**—defines the pullout angle that the tool follows in the X axis to exit the workpiece. This angle is shown as 2 in the following figure; the tool path is represented with a dotted line.

![Figure 2–35. Lead In Angle (1) and Lead Out Angle (2)](image)

- **SPECIFY START ANGLE**—select YES or NO to indicate whether a start angle is required. When NO is selected, the START ANGLE field is inactive.
- **START ANGLE**—defines the Start Angle if YES is selected for SPECIFY START ANGLE. The default is 0°.
- **NUMBER OF STARTS**—defines the number of starts if multiple starts are required and NO is selected for SPECIFY START ANGLE.
ID Taper Thread Type

The following fields are available when ID Taper is selected in the Thread Type field:

- **Z START**—identifies the Z axis starting location for the Threading cycle. The tool begins cutting at the programmed feedrate at this location.
- **Z END**—identifies the Z axis end location for the Threading cycle. The tool stops cutting at this location.
- **Z CLEARANCE**—identifies the distance that the tool should be moved away from the part after the Threading cycle.
- **X CLEARANCE**—identifies the distance that the tool should be raised from the part after the Threading cycle.
- **MAJOR DIAMETER**—indicates the outer diameter of the part. This diameter is shown as 1 in the following figure.
- **MINOR DIAMETER**—indicates the inner diameter of the part. This diameter is shown as 2 in the following figure.

![Image](image_url)

**Figure 2–36. Threading Cycle Geometry Tab ID Taper Thread Type**

**Figure 2–37. Major Diameter (1) and Minor Diameter (2)**
- **PITCH**—defines the distance between threads for metric tools. Do not use this field for programming tools measured in threads per inch.

- **LEAD (TPI)**—defines the threads per inch for tools measured in inches. Do not use this field for programming metric tools.

- **X TAPER (DIA) or (RAD)**—defines the amount of taper in the X axis. When you enter a value in this field, the control calculates the value for the Taper Angle.

- **TAPER ANGLE**—defines the angle of the taper. When you enter a value in this field, the control calculates the value for the X Taper field.

- **VERTICAL LEAD IN**—defines whether the tool should enter the part vertically (perpendicular) to the direction of the cut. Select YES or NO. NO is the default. When YES is selected, the LEAD IN ANGLE field is inactive.

- **LEAD IN ANGLE**—defines the infeed angle that the tool follows in the X axis to enter the workpiece. This angle is shown as 1 in the following figure; the tool path is represented with a dotted line.

- **VERTICAL LEAD OUT**—defines whether the tool should exit the part vertically (perpendicular) to the direction of the cut. Select YES or NO. NO is the default. When YES is selected, the LEAD OUT ANGLE field is inactive.

- **LEAD OUT ANGLE**—defines the pullout angle that the tool follows in the X axis to exit the workpiece. This angle is shown as 2 in the following figure; the tool path is represented with a dotted line.

![Figure 2–38. Lead In Angle (1) and Lead Out Angle (2)](image)

- **SPECIFY START ANGLE**—select YES or NO to indicate whether a start angle is required. When NO is selected, the START ANGLE field is inactive.

- **START ANGLE**—defines the Start Angle if YES is selected for SPECIFY START ANGLE. The default is 0°.

- **NUMBER OF STARTS**—defines the number of starts if multiple starts are required and NO is selected for SPECIFY START ANGLE.
Face Thread Type

The following fields are available when face is selected in the Thread Type field:

- **X START (DIA) or (RAD)**—defines the X (DIA) or (RAD) startpoint for the Threading cycle. The tool begins cutting at the programmed feedrate at this location.

- **X END (DIA) or (RAD)**—identifies the X axis end location for the Threading Cycle. The tool stops cutting at this location.

- **Z CLEARANCE**—identifies the distance that the tool should be moved away from the part after the Threading cycle.

- **X CLEARANCE**—identifies the distance that the tool should be raised from the part after the Threading cycle.

- **Z START**—identifies the Z axis starting location for the Threading cycle. The tool begins cutting at the programmed feedrate at this location.

- **THREAD DEPTH**—identifies the incremental thread depth. The value must be greater than 0.

- **PITCH**—defines the distance between threads for metric tools. Do not use this field for programming tools measured in threads per inch.

- **LEAD (TPI)**—defines the threads per inch for tools measured in inches. Do not use this field for programming metric tools.

- **VERTICAL LEAD IN**—defines whether the tool should enter the part vertically (perpendicular) to the direction of the cut. Select YES or NO. NO is the default. When YES is selected, the LEAD IN ANGLE field is inactive.

- **LEAD IN ANGLE**—defines the infeed angle that the tool follows in the X axis to enter the workpiece. This angle is shown as 1 in the following figure; the tool path is represented with a dotted line.
- **VERTICAL LEAD OUT**—defines whether the tool should exit the part vertically (perpendicular) to the direction of the cut. Select YES or NO. NO is the default. When YES is selected, the LEAD OUT ANGLE field is inactive.

- **LEAD OUT ANGLE**—defines the pullout angle that the tool follows in the X axis to exit the workpiece. This angle is shown as 2 in the following figure; the tool path is represented with a dotted line.

![Figure 2–40. Lead In Angle (1) and Lead Out Angle (2)](image)

- **SPECIFY START ANGLE**—select YES or NO to indicate whether a start angle is required. When NO is selected, the START ANGLE field is inactive.

- **START ANGLE**—defines the Start Angle if YES is selected for SPECIFY START ANGLE. The default is 0°.

- **NUMBER OF STARTS**—defines the number of starts if multiple starts are required and NO is selected for SPECIFY START ANGLE.
Grooving Cycle

The Grooving cycle plunges out all extra material and then makes finish passes. The width of the grooving tool must be programmed in Tool Setup. Grooves are simple shapes with flat bottoms.

⚠️ A Position block is required before a Groove data block if there is not sufficient clearance for the tool to move around the part from a previous data block.

The turning center can cut inside, outside, and face grooves. Optional chamfer angles on the corner of the groove may be programmed. You may also specify a different taper angle for each wall. You may optionally specify each corner or fillet with a chamfer or radius individually, i.e. one corner a chamfer, the other a radius.

Here are some examples:

- Groove 45° chamfer at corners, 0.5 radius at fillets, left wall at 14° angle, right wall no taper angle.

![Figure 2–41. Groove Example 1](image)

- Groove 45° chamfers at corners, 45° inside chamfer, 0.125 radius fillet, wall at left 14°, wall at right 25°.

![Figure 2–42. Groove Example 2](image)
To create a Grooving Cycle from the New Block screen, select the TURNING ➔ F2 softkey followed by the GROOVE F3 softkey.

The GROOVING CYCLE screen appears with tabs for Process, Geometry, and Patterns.

- The fields on the Process tab are for programming the tool and cutting information
- The fields on the Geometry tab are for programming the shape of the groove.
- The fields on the Patterns tab are for programming patterns of identical, incremental grooves along a part.

The start point is the point at which the diameter and wall theoretically intersect. The start point is indicated by a cross hair next to Corner 1 in the on-screen graphic on the Geometry tab.

**Process**

Program the tool and cutting information using the Process tab.

![Grooving Cycle Process Tab screen](image)

*Figure 2–43. Grooving Cycle Process Tab screen*

Use these fields to program the tool and cutting information for the groove:

- **TOOL**—identifies the tool number that will be used in the part program.
- **TOOL OFFSETS**—identifies the programmed tool offset and orientation programmed in Tool Setup that will be used. The tool offset defaults to the same number as the tool.
- **INSERT TIP RADIUS**—displays the insert tip radius programmed in Tool Setup. This field is read-only.
- **SPEED**—identifies the RPM value or Constant Surface Speed value (FPM) for this tool.
- **FEED**—identifies the programmed feed rate (IPM or IPR) for this tool.
- **RETRACT CLEARANCE**—identifies the distance the tool should retract above the X Start dimension while positioning between data blocks. The default is 0.127 mm (0.005 in).

- **RETRACT ANGLE**—identifies the angle at which the tool retracts from the X Start dimension while positioning between data blocks.

- **CUT CLEARANCE**—identifies the amount of clearance for cutting each profile. The default is 1.27 mm (0.05 in).

- **STRATEGY**—identifies the tool path with respect to rough or finish.
  - **ROUGH & FINISH**—perform rough and finish passes.
  - **ROUGH ONLY**—perform a rough pass only.
  - **FINISH ONLY**—perform a finish pass only.
  - **FINISH (1 PASS)**—performs the finish pass in one pass across all three surfaces of the groove.

- **SEQUENCE**—indicates the sequence the corners will be cut.

- **Use Safe Rapid Point** checkbox—activates the X Rapid and Z Rapid fields for entering the location for the X and Z axes to move at rapid feedrate at the start and end of the cycle.

- **X RAPID**—identifies the X axis rapid feedrate location for the start and end of this cycle. This field is only available for editing when the Use Safe Rapid Point checkbox is selected.

- **Z RAPID**—identifies the Z axis rapid feedrate location for the start and end of this cycle. This field is only available for editing when the Use Safe Rapid Point checkbox is selected.

- **STEP OVER**—identifies the distance the tool steps over each pass between grooving passes. This field is active only when either ROUGH & FINISH or ROUGH ONLY is selected in the STRATEGY field.

- **STOCK ALLOWANCE**—identifies the amount of material to leave on the profile. The control calculates the TURN and FACE ALLOWANCE fields when you enter a value in this field.

- **TURN ALLOWANCE**—identifies the amount of material to leave on horizontal (turn) surfaces. This field is calculated by the control when the Stock Allowance is entered; however, you can edit the value to fine tune the data.

- **FACE ALLOWANCE**—identifies the amount of material to leave on vertical (face) surfaces. This field is calculated by the control when the Stock Allowance is entered; however, you can edit the value to fine tune the data.
Geometry

Define the shape of the groove using the fields in the Geometry tab.

![Grooving Cycle Geometry Tab screen](image)

**Figure 2–44. Grooving Cycle Geometry Tab screen**

Use these fields to program the shape of the groove:

- **ORIENTATION**—select among Outside, Inside, and Face for the groove type. The on-screen graphic changes to represent the orientation for each selection.

![Groove Orientation selections](image)

**Figure 2–45. Groove Orientation selections**

- **X START (DIA) or RAD**—identifies the X axis starting location for the Grooving cycle. The tool begins cutting at the programmed feedrate at this location. Change between Diameter and Radius by selecting the D or R in the status bar. This dimension is shown as 2 in the following figure.

- **Z START**—identifies Z axis starting location for the Grooving cycle. The tool begins cutting at the programmed feedrate at this location. This dimension is shown as 4 in the following figure.

- **GROOVE WIDTH**—indicates how wide the Groove needs to be. This dimension is shown as A in the following two figures.
• **X BOTTOM (DIA) or (RAD)**—identifies the part-relative coordinate for the depth of the Groove. Change between Diameter and Radius by selecting the D or R in the status bar. This dimension is shown as 3 in the following figure. This field changes to Z BOTTOM when the Orientation is Face.

  - The start point is indicated by a cross hair next to Corner 1 in the on-screen graphic.
  - The depth of the groove is determined by subtracting X Bottom from X Start. This dimension is shown as B in the following figure.

![Diagram of groove dimensions](image)

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Part Zero X</td>
</tr>
<tr>
<td>2</td>
<td>X Start</td>
</tr>
<tr>
<td>3</td>
<td>X Bottom</td>
</tr>
<tr>
<td>4</td>
<td>Z Start</td>
</tr>
<tr>
<td>A</td>
<td>Groove Width</td>
</tr>
<tr>
<td>B</td>
<td>Groove Depth</td>
</tr>
</tbody>
</table>

*Figure 2–46. Groove Width and X Bottom dimensions (Outside and Inside Orientation)*
- **Z BOTTOM**—identifies the part-relative coordinate for the bottom of the groove. This dimension is shown as \(3\) in the following figure. This field replaces \(X \text{ BOTTOM (DIA)}\) or \(\text{ (RAD)}\) when the Orientation is Face.

- The start point is indicated by a cross hair next to Corner 1 in the on-screen graphic.

- The depth of the groove is determined by subtracting \(Z \text{ Bottom}\) from \(Z \text{ Start}\). This dimension is shown as \(B\) in the following figure.

- **TAPER 1-2** and **4-3**—define the angles of the two groove walls.

- **CORNER 1, CORNER 2, CORNER 3, CORNER 4**—use these fields for defining each corner:
  - **CORNER TYPE**—indicates the type of groove corner as either Square, Chamfer, or Radius.
  - **CHAMFER LENGTH**—defines the length of the chamfer. This field is active when Chamfer is selected for the Corner Type field.
  - **CHAMFER ANGLE**—defines the corner chamfer angle. This field is active when Chamfer is selected for the Corner Type field.
  - **RADIUS**—defines the corner radius of the chamfer. This field is active when Radius is selected for the Corner Type field.
Patterns

Program patterns of identical and incremental grooves along a part.

Use these fields to program multiple grooves:

- **NUMBER**—defines the number of grooves to be repeated.
- **OFFSET**—defines the distance between the original groove and the first copy (from the start of one groove to the start of the next), as identified in the on-screen graphic.

The on-screen graphic on the Patterns tab is based on the Orientation selection in the Geometry tab:

In the on-screen graphic, X or Z represents the starting location of the first groove, and the Delta symbol (Δ) represents the starting location of the second groove in the pattern.
**Hole Cycle**

Hole cycle operations include Drill, Center Drill, and Tap. The tool type defined by the tool number selected in the block determines the type of hole operation. The on-screen graphic shows the tool and the screen contains fields appropriate for programming the tool type.

- **Drill**—creates a hole in the stock centerline. Drills can also be used as starting points for additional operations such as boring.

  The drill can be shifted off center with the X Offset field.

  ! Do not exceed the drill manufacturer’s maximum allowable offset.

- **Center Drill**—drills to the programmed depth, dwells for the programmed amount of seconds, then retracts.

- **Tap**—feeds with spindle at speed to depth, reverses the spindle and feeds out. Standard tapping and rigid tapping are available.

  A tension/compression tap holder is recommended.

  The control will automatically calculate the feedrate for tapping using the speed (rpm) and the threads per inch or pitch programmed for the tool in Tool Setup.

To create a Hole Cycle from the New Block screen, select the TURNING → F2 softkey followed by the HOLE F4 softkey.

The HOLE CYCLE screen appears.

![Figure 2–50. Hole Cycle screen](image-url)
These fields are available for programming Hole Cycles:

- **TOOL**—identifies the tool type and tool number for the current tool number programmed in Tool Setup that will be used in the part program. The tool type selected in the block determines the type of hole operation.

- **TOOL OFFSETS**—identifies the programmed tool offset and orientation programmed in Tool Setup that will be used. You can accept or change this offset.

- **Z START**—defines the point where the tool moves when starting the cycle, before moving at Rapid to the R Plane.

- **R PLANE**—defines an absolute position relative to part zero. The cycle will rapid to this location before feeding to Z Bottom.

- **Z BOTTOM**—identifies the depth of the hole. This field is automatically adjusted for the drill tip if YES is selected for Tip Compensation.

- **X OFFSET**—defines the incremental offset in X from the spindle centerline for drills that can be plunged off center. This field only appears when the Tool Type is a Drill and Cycle Type is not Center Drill.

- **CYCLE TYPE**—accesses the following softkey selections. This field appears when the tool is a Drill or a Center Drill.
  - **Standard** drilling feeds to depth and retracts at Rapid to the retract dimension (R-Plane). Standard is the default operation. Refer to Tap Cycle below for information about the Standard Tap cycle.
  - **Dwell** is drilling with a dwell at the bottom.
  - **Peck** drilling feeds to a peck depth and retracts at Rapid to the Retract dimension (R-Plane).
  - **Chip Breaker** feeds to a peck depth, retracts at Rapid a short distance, then feeds to the next peck depth. This is repeated until the programmed depth is achieved.
  - **Center Drill** cycle creates a pilot hole. You can use a center drill or a standard drilling hole to perform this operation.
  - **Decreasing Peck Drilling Depth (Decr Depth)** cycle feeds to the programmed First Peck Depth, then continues cutting a decreasing amount to the Minimum Peck by the peck percent each pass until it reaches the programmed Minimum Peck Depth or Z Bottom.
  - **Decreasing Chip Breaker (Decr Chp Bkr)** cycle feeds to the programmed First Peck Depth, retracts at Rapid a short distance, then feeds to the next peck depth, cutting a decreasing amount to the minimum peck by the peck percent each pass until it reaches the programmed Minimum Peck Depth or Z Bottom. This is repeated until the programmed depth is achieved.
- **TAP CYCLE**—This field appears when the tool is a Tap. Select Standard or Rigid.
  - **Standard** cuts threads inside a hole.
  - **Rigid** taps the same hole multiple times, maintaining the tool’s orientation with previously cut threads.

    When performing rigid tapping, the hole must be longer than the tap. To accomplish this, program the preceding drill operation to be longer than the tap operation.

- **DWELL TIME**—This field appears when Dwell or Center Drill is selected for the Cycle Type or when Standard is selected for the Tap Cycle Type. After the cycle reaches Z Bottom, the Z axis will dwell for the time specified in seconds before retracting at Rapid.

- **PITCH**—This field appears when the tool is a Tap. Contains the value for threads between metric taps.

- **TPI**—This field appears when the tool is a Tap measured in inches. Contains the value for the threads per inch for inch taps.

    When you enter a value for TPI, the control calculates the value for PITCH. Conversely, when you enter a value for PITCH, the control calculates the value for TPI for tools measured in inches.

- **PECK DEPTH**—This field appears when either Peck or Chip Breaker is selected for the Cycle Type, or when Rigid is selected for the Tap Cycle Type and Tap is the tool. Peck depth is the maximum depth to be cut in one pass.

- **FIRST PECK DEPTH**—This field appears when either Decr Depth or Decr Chp Bkr is selected for the Cycle Type. First Peck Depth is the maximum depth to be cut in the first pass.

- **MINIMUM PECK DEPTH**—This field appears when either Decr Depth or Decr Chp Bkr is selected for the Cycle Type. Minimum Peck Depth is the smallest peck depth that can be cut in the current cycle. The minimum peck depth for decreasing depth holes is 0.003 mm (0.0001 in). This field is used in conjunction with the Peck Percent field to allow the peck depth to reach Z Bottom. A Minimum Peck Depth must be set in order for the tool to reach Z Bottom.

Here are some suggestions when Tapping:

- Use a floating tap holder to prevent the tap from breaking when the spindle reverses direction.
- When tapping hard materials, use tapping fluid (not coolant) as a lubrication.
- When tapping hard materials, use a tap drill one size larger than normal because less thread engagement requires less torque.
- The recommended Z Start is 0.200 minimum from the part surface to compensate for floating tap holder characteristics.
• **PECK PERCENT**—This field appears when either Decr Depth or Decr Chp Bkr is selected for the Cycle Type. Peck Percent is a percent of the First Peck Depth. For example, if the Peck Percent is set at 50, subsequent pecks will equal a distance 50% smaller than the previous peck depth. A Minimum Peck Depth must be set in order for the tool to reach Z Bottom.

  Peck Depth Example:
  First Peck Depth = 1.0 in.
  R Plane = 0.1
  Bottom of Hole (Z Bottom) = -3.0
  Peck Percent = 80%

  First Peck Depth = 1.0 in. cuts to depth of Z-0.90
  Second Peck Depth = 0.8 in. cuts to depth of Z-1.7
  Third Peck Depth = 0.64 in. cuts to depth of Z-2.34
  Fourth Peck Depth = 0.512 in. cuts to depth of Z-2.852
  Fifth Peck Depth = 0.4096 in. cuts to depth of Z-3.0, the specified depth of the hole.

• **PECK COUNT**—This read-only field appears when either Peck, Chip Breaker, Decr Depth, or Decr Chp Bkr is selected for the Cycle Type. The number in the Peck Count field represents the number of pecks to occur.

• **PECK CLEARANCE**—This field appears when either Peck, Chip Breaker, Decr Depth, or Decr Chp Bkr is selected for the Cycle Type. Return at rapid feed rate to this location above the last peck level. The default is 0.254 mm (0.010 in).

• **TOOL DIAMETER**—contains the tool diameter programmed in Tool Setup for the current tool. This field is read-only and can only be edited in Tool Setup.

• **SPEED**—contains the spindle speed value programmed in Tool Setup. “CAL” appears next to the value as programmed in Tool Setup. You can accept or change this value. If you edit this field, “CAL” disappears.

• **PLUNGE FEED**—contains the feedrate between the Z START and Z BOTTOM positions. “CAL” appears next to the value as calculated by the control. You can accept or change this value. If you edit this field, “CAL” disappears.

• **TIP COMPENSATION**—This field appears when the tool is a Drill or a Center Drill. Select YES or NO to indicate whether the tool tip is included as part of the Z Bottom measurement. Z Bottom depth is automatically calculated for the drill tip when YES is selected and is based on the drill tip angle programmed in Tool Setup.

• **RETURN LEVEL**—identifies the level to which the tool should retract at the end of the drill cycle. Select I Plane or R Plane.
  • **I PLANE**—returns to the coordinate specified in the X/Z Start field.
  • **R PLANE**—returns to the coordinate specified in the R Plane field.
Cutoff Block

When using bar stock, you can separate your completed part from the bar by using a cutoff block. If you have the optional Parts Catcher on your machine, you may optionally advance the catcher when the tool passes a configurable X-axis position.

To create a Cutoff Block from the New Block screen, select the TURNING ➔ F2 softkey followed by the CUTOFF F5 softkey.

The CUTOFF BLOCK screen appears with tabs for Process/Geometry and Advanced Pecking.

- The fields on the Process/Geometry tab are for programming the tool and cutting information, and for programming the geometry of the cut off.
- The Advanced Pecking tab is available for adjusting settings for speeds, feeds, and dwell for up to 3 cutting depths at specified X radius or diameter locations.

To perform a cutoff block on stock in the sub-spindle using a TMX MYS machine, select the On Sub-Spindle checkbox in Part Setup - Work Offsets. Please refer to Part Setup Fields, on page 4 - 14 for a description of the checkbox.

Process/Geometry

Program the tool and cutting information, and program the geometry of the cut off.

Figure 2–51. Cutoff Block Process/Geometry Tab screen
Use these fields for programming Cutoff data blocks:

- **TOOL**—identifies the tool number that will be used in the part program.
- **TOOL OFFSETS**—identifies the programmed tool offset and orientation programmed in Tool Setup that will be used. You can accept or change this offset.
- **INSERT TIP RADIUS**—displays the insert tip radius programmed in Tool Setup. This field is read-only. **Use Safe Rapid Point** checkbox—activates the X Rapid and Z Rapid fields for entering the location for the X and Z axes to move at rapid feedrate at the start and end of the cycle.
- **X START (DIA) or RAD**—identifies the X axis starting location. The tool begins cutting at the programmed feedrate at this location.
- **Z START**—identifies Z axis starting location. The tool begins cutting at the programmed feedrate at this location.

![Warning]
The spindle is enabled and turns at the value programmed in the **Speed** field during the rapid move to X Start (Dia) or (Rad) and Z Start.

- **ADVANCE PART CATCHER**—advances the parts catcher and enables the ADVANCE AT X and PAUSE AT END (SEC) fields when Yes is selected. This field is available when the optional Parts Catcher is installed.
- **W POSITION**—contains the location of the programmable tailstock or sub-spindle. This field is available when the optional Parts Catcher is installed and ADVANCE PART CATCHER is set to YES. This field appears for TMX, TMX MY, and TMX MYS series machines.
- **ADVANCE AT X (DIA) or (RAD)**—specifies the position along the X axis which triggers the advance of the Parts Catcher when the tool reaches the specified location. This field is available when the optional Parts Catcher is installed and ADVANCE PART CATCHER is set to YES.
- **PAUSE AT END (SEC)**—specifies the amount of time in seconds for the turret to pause while the parts catcher starts retracting. After the delay, the turret moves to X End. This field is available when the optional Parts Catcher is installed and ADVANCE PART CATCHER is set to YES.
- **X END (DIA) or (RAD)**—identifies the X axis end location for the Cutoff block. The tool stops cutting at this location.
- **SPEED**—identifies the RPM value or Constant Surface Speed value for this tool.
- **FEED**—identifies the programmed feed rate for this tool.
- **PECK TYPE**—select either Standard, Dwell, or Chip Breaker.
  - **STANDARD**—feeds to depth and retracts at Rapid to the retract dimension (R-Plane). Standard is the default operation.
  - **DWELL**—drills with a dwell at the bottom.
  - **CHIP BREAKER**—feeds to a peck depth, retracts at Rapid a short distance, then feeds to the next peck depth. This is repeated until the programmed depth is achieved.
• **PECK DEPTH**—identifies the incremental peck depth. This field appears when either Dwell or Chip Breaker is selected for the Peck Type.

• **DWELL (SECS)**—identifies the amount of seconds the tool dwells before retracting at Rapid. This field appears when Dwell is selected for the Peck Type.

• **RETRACT DISTANCE**—indicates the distance the tool should retract between pecks. This field appears when Chip Breaker is selected for the Peck Type.

• **RETRACT AT RAPID**—indicates whether or not the tool should retract at rapid rate. Select YES or NO. This field appears when Chip Breaker is selected for the Peck Type.

• **RETRACT FEED**—indicates the feed per minute or feed per revolution at which the tool should retract. Select either the FEED PER MINUTE F4 or FEED PER REV F5 softkey to set the retract feed at IPM or IPR. These softkey choices also affect the FEED (IPM) or (IPR) field. This field appears when Chip Breaker is selected for the Peck Type and is only active when NO is selected for RETRACT AT RAPID.

• **WIDTH COMPENSATION**—indicates whether there should be compensation for the tool width when determining Z Start. The screen graphic changes the Z Start position illustration based on the selection.
  
  • Select NO for no tool width compensation, and the Z Start position is located at the far left side of the cut.
  
  • Select YES for tool width compensation, and the Z Start position is located at the far right side of the cut. The default is YES.

*Figure 2–52. Z Start Cutoff Tool Width Compensation selections*
Advanced Pecking

The Advanced Pecking tab is available for making adjustments for speeds, feeds, and dwell for up to 3 cutting depths at specified X radius or diameter locations.

![Figure 2–53. Cutoff Block Advanced Pecking Tab screen](image)

Use these fields for making adjustments within a Cutoff data block:

- **X (RAD) or (DIA)**—defines the X cutting depth where the change in speed, feed, and/or dwell should occur. Select from 1 to 3 X locations by selecting the 1, 2, or 3 listed to the left of this field.
- **SPEED**—defines the adjusted RPM value or Constant Surface Speed value for this tool.
- **FEED**—identifies the adjusted feed per revolution for this tool.
- **DWELL**—defines the adjusted number of revolutions before the drill should retract at Rapid. This field appears when Dwell is selected for the Peck Type on the Process/Geometry tab.

Programmed tool information is carried over from the Process/Geometry tab to the lower right corner of the Advanced Pecking tab to use as a reference.
## Live Tooling Data Blocks

A Live Tooling softkey menu is available for TMM, TMX MY, and TMX MYS series machines. In addition, this softkey menu appears on the WinMax Lathe desktop software when one of these models is selected for the machine type when the Machine Type Selector field is set to Yes on the Utilities, User Preferences, User Interface Settings screen.

Both radial and axial tooling are used for live tooling operations. Radial tooling approaches the stock in the -X axis direction. Axial tooling approaches the stock in the -Z axis direction.

Access the Live Tooling data block screens from the New Block screen, LIVE TOOLING ➔ F3 softkey to access the following types of live tooling data blocks:

<table>
<thead>
<tr>
<th>Radial Milling</th>
<th>2 - 67</th>
</tr>
</thead>
<tbody>
<tr>
<td>Radial Lines and Arcs</td>
<td>2 - 71</td>
</tr>
<tr>
<td>Radial Circle</td>
<td>2 - 89</td>
</tr>
<tr>
<td>Radial Frame</td>
<td>2 - 95</td>
</tr>
<tr>
<td>Radial Slot</td>
<td>2 - 102</td>
</tr>
<tr>
<td>Radial Lettering</td>
<td>2 - 110</td>
</tr>
<tr>
<td>Radial Holes</td>
<td>2 - 114</td>
</tr>
<tr>
<td>Radial Hole Cycle</td>
<td>2 - 115</td>
</tr>
<tr>
<td>Radial Locations</td>
<td>2 - 121</td>
</tr>
<tr>
<td>Radial Loop Around C-Axis</td>
<td>2 - 123</td>
</tr>
<tr>
<td>Axial Milling</td>
<td>2 - 125</td>
</tr>
<tr>
<td>Axial Lines and Arcs</td>
<td>2 - 128</td>
</tr>
<tr>
<td>Axial Circle</td>
<td>2 - 147</td>
</tr>
<tr>
<td>Axial Frame</td>
<td>2 - 153</td>
</tr>
<tr>
<td>Axial Slot</td>
<td>2 - 160</td>
</tr>
<tr>
<td>Axial Flats</td>
<td>2 - 168</td>
</tr>
<tr>
<td>Axial Lettering</td>
<td>2 - 175</td>
</tr>
<tr>
<td>Axial Holes</td>
<td>2 - 178</td>
</tr>
<tr>
<td>Axial Hole Cycle</td>
<td>2 - 179</td>
</tr>
<tr>
<td>Axial Locations</td>
<td>2 - 182</td>
</tr>
<tr>
<td>Axial Bolt Circle</td>
<td>2 - 185</td>
</tr>
</tbody>
</table>
Radial Milling

From the New Block screen, select LIVE TOOLING ➔ F3. From the New Block (Live Tooling) screen, select RADIAL MILLING F1 to access the Radial Milling softkey menu.

![Figure 2–54. Live-Tooling New Block Radial Milling screen](image)

- **RADIAL LINES AND ARCS** F1—accesses the Radial Mill Contour data block for programming segments with lines, arcs, and blend arcs. Refer to *Radial Lines and Arcs*, on page 2 - 71 for programming details.
- **RADIAL CIRCLE** F2—accesses the Radial Mill Circle data block. Refer to *Radial Circle*, on page 2 - 89 for programming details.
- **RADIAL FRAME** F3—accesses the Radial Mill Frame data block. Refer to *Radial Frame*, on page 2 - 95 for programming details.
- **RADIAL SLOT** F4—accesses the Radial Slot data block. Refer to *Radial Slot*, on page 2 - 102 for programming details.
- **RADIAL LETTERING** F5—accesses the Radial Lettering data block. Refer to *Radial Lettering*, on page 2 - 110 for programming details.
Programming Coordinates and Linear Y Motion

Radial milling is used with TMM, TMX MY, and TMX MYS machines. The programming coordinates are mapped to behave like milling machines. In order to perform machining processes for TMM, TMX MY, and TMX MYS machines, WinMax aligns the programming coordinate system. Machine coordinates do not change.

No Linear Y Motion

The following figure shows the part coordinate system for TMM, TMX MY, and TMX MYS machines, for wrapping the Y-Axis coordinate system around the circumference of the part.

⇒ TMM machines use this mapping for Radial milling.

When Linear Y Motion is not selected on the Geometry tab, the Y-axis is wrapped around the circumference of the part. The X machine coordinate becomes Part Z-axis, and the Z machine coordinate becomes the Part X-axis. The machine C-Axis rotates for Y motion.

![Diagram of part coordinate systems](image)

**Figure 2–55. Main and Sub-spindle Radial surface of the part without Linear Y**

The bottom surface of the feature being machined is concentric to the circumference of the part:

![Diagram of milling and drilling](image)

**Figure 2–56. Radial Cutting—C-Axis**
Y-Axis Coordinate System - Radial Surface of the Part without Linear Y

Figure 2–57. Y-Axis Coordinate System - Radial Surface of the Part without Linear Y

Live Tool Mode Milling with Polar Coordinates

Figure 2–58. Live Tool Mode Milling with Polar Coordinates

Please refer to Radial Lines and Arcs, on page 2 - 71, Radial Circle, on page 2 - 89, Radial Frame, on page 2 - 95, Radial Slot, on page 2 - 102, or Radial Lettering, on page 2 - 110 for information about the Geometry tab screen for the different types of Radial Milling. For information about Radial Holes, please refer to Radial Holes, on page 2 - 114.
**Linear Y Motion**

The following figure shows the Y-Axis coordinate system mapping for TMX MY and TMX MYS machines. When Linear Y Motion is selected on the Geometry tab, linear Y-axis motion occurs.

⇒ TMM machines do not use this mapping.

![Diagram of Y-Axis coordinate system mapping for TMX MY and TMX MYS machines.

Main Spindle Linear Y selected
Part Coordinate system

Sub-spindle Linear Y selected
Part Coordinate system

Figure 2–59. Main and Sub-spindle Radial surface of the part with Linear Y

With Linear Y selected, TMX MY and TMX MYS machines can perform Linear Milling to create flat bottoms and right-angle walls for milled features and to machine holes parallel to the Y axis.

![Diagram of Radial Cutting—Y-Axis motion.

Figure 2–60. Radial Cutting—Y-Axis motion

Please refer to Radial Lines and Arcs, on page 2 - 71, Radial Circle, on page 2 - 89, Radial Frame, on page 2 - 95, Radial Slot, on page 2 - 102, or Radial Lettering, on page 2 - 110 for information about the Geometry tab screen for the different types of Radial Milling. For information about Radial Holes, please refer to Radial Holes, on page 2 - 114.
Radial Lines and Arcs

From the Radial Milling softkey menu, select RADIAL LINES AND ARCS F1, and the Radial Mill Contour screen appears with Process and Geometry tabs. Program the tool and cutting information using the Process tab. Describe the finish shape using the Geometry tab.

The program determines how many rough passes to make to remove the material based on the entries in the Process tab fields. Finishing follows the geometry of the contour defined in the Geometry tab.

![Figure 2–61. Live-Tooling Series Radial Mill Contour Screen Process tab](image)

### Process Tab

The Radial Mill Contour screen Process tab fields are defined as follows:

- **TOOL**—identifies the tool number that will be used in the part program.
- **TOOL OFFSETS**—identifies the tool offset and orientation programmed in Tool Setup that will be used. The tool offset defaults to the same number as the tool.
- **TOOL DIAMETER**—contains the tool diameter programmed in Tool Setup for the current tool. This field is read-only and can only be edited in Tool Setup.
- **STRATEGY**—identifies the tool path with respect to rough or finish.
  - **ROUGH AND FINISH**—perform rough and finish passes.
  - **ROUGH ONLY**—perform a rough pass only.
  - **FINISH ONLY**—perform a finish pass only.

Please refer to Axial Milling, on page 2 - 125 for Axial Mill Contour screen Process tab field definitions.
• **MILLING TYPE**—identifies on which side of the contour the tool should begin cutting. Refer to *Cutter Compensation, on page 2 - 188* for more details. Select from among these choices:
  - **ON**—used for cutting directly on the programmed path.
  - **LEFT**—used for climb milling.
  - **RIGHT**—used for conventional milling.
  - **PROFILE LEFT**—used for removing material from a closed contour for climb milling.
  - **PROFILE RIGHT**—used for removing material from a closed contour for conventional milling.
  - **POCK BOUNDARY (POCKET BOUNDARY)**—used for cutting the defined closed contour.

• **SPEED (RPM)**—identifies the speed at which the spindle rotates to cut the part. This value is carried forward for the selected tool from Tool Setup. “CAL” appears next to the field. To accept this value, select Enter. A different speed can be programmed for this specific block.

  ➔ If a Speed value is changed in a milling data block, a pop-up message appears asking if you want to change the corresponding Feed value in the data block.

  Override the displayed value by typing in a new speed or by using the console Spindle override knob.

• **MILL FEED**—identifies the cutting feed rate. This value is carried forward for the selected tool from Tool Setup. “CAL” appears next to the field. To accept this value, select Enter. A different mill feed rate can be programmed for this specific block.

  ➔ If a Feed value is changed in a milling data block, a pop-up message appears asking if you want to change the corresponding Speed value in the data block.

• **PLUNGE FEED**—identifies the feedrate between Z Start and Z Bottom.
These fields appear based on your selection in the previous fields:

- **ENABLE BLEND MOVES**—enables automatic blend in and blend out for arcs. This field is available for Left, Right, Profile Left, and Profile Right milling.
- **POCKET TYPE**—identifies the pocketing type when Pocket Boundary is selected for the Milling Type.
  - **INWARD**—cuts in from the edge of the defined boundary.
  - **OUTWARD**—cuts from the center of the feature outward to the defined boundary.
- **POCKET OVERLAP (%)**—controls the percentage of overlap during cutting. This field is available when either Profile Left, Profile Right, or Pocket Boundary is selected for the Milling Type.
- **WALL ALLOWANCE**—identifies the amount of material to leave along the wall of the stock during roughing for cutting the final pass. The wall of the stock is the side of the stock that is being cut. This field is not available when:
  - On is selected for the Milling Type.
  - Finish Only is selected for the Strategy.
- **FLOOR ALLOWANCE**—identifies the amount of material to leave on the bottom of the stock during roughing for cutting the final pass. This field is not available when Finish Only is selected for the Strategy.
- **BLEND OFFSET**—identifies the XY distance from the entry point of the part surface and the Z plunge point where the tool enters the work piece. This move is always a 90° arc. This parameter is used for the blend-in move in milling and for the blend-out move from the part surface to the Z Retract point. The default value is 3.000 mm (0.1250 inches). This field is available when Yes is selected for Enable Blend Moves.
- **BLEND OVERLAP**—identifies the distance the tool travels past the entry point before exiting from the part. The default value is 3.000 mm (0.1250 inches). This field is available when Yes is selected for Enable Blend Moves.
- **MAX OFFSET**—defines the distance away from the programmed contour where milling begins. This field is available when either Profile Left or Profile Right is selected for the Milling Type.
- **DIRECTION**—identifies the milling direction as either Conventional or Climb. This field is not available when On is selected for the Milling Type. Refer to *Cutter Compensation, on page 2-188* for details about milling direction.
- **PECK DEPTH**—identifies the maximum depth to be cut in one pass. This field is not available when Finish Only is selected for the Strategy.
- **1ST PECK OFFSET**—identifies the depth of the first peck when it needs to be different than subsequent pecks, identified in Peck Depth. This field is not available when Finish Only is selected for the Strategy.
- **PECK CLEARANCE**—identifies the distance above the previous peck level to which the Z axis returns at rapid traverse. This field is not available when Finish Only is selected for the Strategy.
- **CUTTER COMP**—identifies the type of milling to insert between segment end points. Choose between Arc or Line. This field is not available when On is selected for the Milling Type.

- If Arc is selected, a tangent arc is inserted to connect two line segments, or a line segment and an arc segment (when the two cutter compensated segments are offset and do not intersect). When using the Arc method, the system creates the cutter compensated path as shown below:

```
1 Programmed tool path
2 Cutter compensated path
3 Arc parameter
```

*Figure 2–62. Cutter Compensation Using the Arc Parameter*

Some operators find the Arc method inappropriate for their applications because the cutter remains in contact with the work piece and may drag a chip across a sharp angle on the part. In such circumstances, operators select Lines.
• If the Lines softkey is selected, the cutter compensated lines and arcs are joined as described below:
  • Two line segments are extended until they intersect (provided they form a 90° or greater angle). If the lines form an angle of less than 90°, a line is inserted to connect them.
  • Line and arc segments have the line segment extended, and a tangent line to the arc segment inserted and extended until the lines intersect (provided they form a 90° or greater angle). If the segments form an angle of less than 90°, a line is inserted to connect them.
  • Two arc segments have tangent lines (to the arcs) inserted and extended until the lines intersect (provided the extended tangent lines form a 90° or greater angle). If the extended tangent lines form an angle of less than 90°, a line or arc is inserted to connect them.

![Figure 2-63. Cutter Compensation Using the Line Parameter](image)

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Programmed tool path</td>
</tr>
<tr>
<td>2</td>
<td>Cutter compensated path</td>
</tr>
<tr>
<td>3</td>
<td>Insert Line parameter</td>
</tr>
</tbody>
</table>

*Figure 2–63. Cutter Compensation Using the Line Parameter*
**Geometry Tab**

The Radial Mill Contour screen appears with Process and Geometry tabs. Program the tool and cutting information using the Process tab. Describe the finish shape using the Geometry tab.

Finishing follows the geometry of the contour defined in the Geometry tab. The fields on the Geometry tab are defined as follows:

- **CYLINDER DIAMETER** or **RADIUS**—defines the diameter or radius of the cylinder.
- **COORDINATES**—defines the coordinates as either Linear or Rotary.
  - **Linear** coordinates are used when working from a flat drawing. When Linear is selected, the X coordinates move in the physical Z direction, the Y coordinates wrap around the C axis, and the Z coordinates move in the physical X direction.
  - **Rotary** coordinates move around the cylinder as specified.

The fields are based on your selection in the Coordinates field (Rotary or Linear).

When Rotary is selected in the COORDINATES field, these fields appear:

- **Z START**—defines the location where Plunge Feedrate begins. The feedrate is set in the Plunge Feed field in the Process tab.
- **C ANGLE (DEG)**—defines the angle of the line segment from the start point to the end point, measured counterclockwise from the home position.
- **X START (DIA)** or **(RAD)**—defines the location where Plunge Feedrate begins. The feedrate is set in the Plunge Feed field in the Process tab.
- **X BOTTOM (DIA)** or **(RAD)**—defines the bottom of the hole and the location where the Plunge Feed Rate ends.
When Linear is selected in the COORDINATES field, these fields appear:

- **X’ START**—defines the X Startpoint for the first segment of the contour. The X’ axis will move in the physical Z direction.
- **Y’ START**—defines the Y Startpoint for the first segment of the contour. The Y’ axis will be wrapped on the cylinder diameter.
- **C POSITION**—defines the angle to position the C axis for the first segment of the contour. This field appears when the Linear Y Motion checkbox is selected. This field is available with TMX MY and TMX MYS series machines.
- **LINEAR Y MOTION**—select this checkbox to cause Y-Axis Motion using the X and X’ axes. When this checkbox is clear, Y-Axis Motion occurs using the X and C axes. Please refer to Programming Coordinates and Linear Y Motion, on page 2 - 68 for examples. This field is available with TMX MY and TMX MYS series machines.
- **Z’ START**—defines the location where Plunge Feedrate begins. The feedrate is set in the Plunge Feed field in the Process tab. The Z’ axis will move in the physical X direction.
- **Z’ BOTTOM**—defines the bottom of the hole and the location where the Plunge Feedrate ends. The Z’ axis will move in the physical X direction. These fields are read-only and are carried over from Tool Setup for the tool specified in the Process tab.
- **TOOL**—identifies the tool number that will be used in the part program.
- **TOOL OFFSETS**—identifies the tool offset and orientation programmed in Tool Setup that will be used. The tool offset defaults to the same number as the tool.
- **TOOL DIAMETER**—contains the tool diameter programmed in Tool Setup for the current tool. This field is read-only and can only be edited in Tool Setup.

Linear Y-Axis Motion is limited. The amount of Y travel is affected by the X position and by any X-Axis tool length offset.

![Figure 2–65. Live-Tooling Radial Mill Contour screen Geometry tab: Linear Coordinates](image)
**Segments**

Rotary Mill contour data blocks use segments to create lines and arcs in a part program. A segment is any single or combined axis movement at a programmed feedrate. A series of lines and arcs can be programmed in a single data block, using different segments, to form a complete contour. The contour can be programmed in either Rotary or Linear Coordinates.

The first segment in a Rotary Mill Contour block is indicated by the segment zero (0). Segment numbers are system generated and you cannot change them.

The cursor is initially positioned in the Tool field of the Process tab when the Segment 0 screen appears. Enter the number of the tool that will be used for the entire operation - all of the segments in this operation will use this tool.

Press the Ctrl + arrow console key or the NEXT SEGMENT F2 softkey after Segment 0 is completed to create a new segment for the data block. Up to 999 segments can be programmed for each data block, if they all use the same tool and are part of the same Radial Mill Contour.

You cannot start or end a Radial Mill Contour segment with blend arcs, nor can blend arcs be adjacent to each other.

The New Radial Contour Segment screen appears with these softkey choices.

- **LINE F1**—accesses the Line segment screen. Refer to Line, on page 2 - 79 for programming details.
- **ARC F2**—accesses the Arc segment screen. Refer to Arc, on page 2 - 83 for programming details.
- **BLEND ARC F3**—accesses the Blend Arc segment screen. Refer to Blend Arc, on page 2 - 86 for programming details.
Line

From the New Radial Contour Segment screen, select LINE F1, and the Radial Mill Contour Line Segment screen appears with fields based on your selection in the Coordinates field (Rotary or Linear).

When Rotary is selected in the COORDINATES field, these fields appear:

- **Z END**—defines the Z end point of the line segment.
- **C END (DEG)**—defines the angle of the line segment from the start point to the end point, measured counterclockwise from the home position.
- **Z START**—defines the Z start point for the segment. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **C START (DEG)**—defines the angle of the line segment from the start point to the end point, measured counterclockwise from the home position. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **FEED**—contains the feed rate from the previous segment. This value can be accepted or changed.

![Image of Live-Tooling Radial Mill Contour Line Segment screen: Rotary Coordinates](image)

*Figure 2-67. Live-Tooling Radial Mill Contour Line Segment screen: Rotary Coordinates*
When Linear is selected in the COORDINATES field, these fields appear:

- **X' END**—defines the X end point for the segment. The X’ axis will move in the physical Z direction.
- **Y' END**—defines the Y end point for the segment. The Y’ axis will be wrapped on the cylinder diameter.
- **X' START**—defines the X start point for the segment. The X’ axis will move in the physical Z direction. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **Y' START**—defines the Y start point for the segment. The Y’ axis will be wrapped on the cylinder diameter. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **LENGTH**—identifies the length of the line.
- **ANGLE**—defines the angle of the line segment from the start point to the end point, measured counterclockwise from the home position.
- **FEED**—contains the feed rate from the previous segment. This value can be accepted or changed.
The following figure shows the Length, Angle, and end point relationship:

![Figure 2–69. Line segment coordinates](image)

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

The Calc-Assist feature calculates certain unknown line segment dimensions automatically, after sufficient data has been entered. A parameter with a calculated value is indicated with “CAL” preceding the value.

- If the End coordinates are entered, the control automatically calculates the Length and the Angle values.
- If both end points are unknown, but Length and Angle are programmed, the control automatically calculates both End parameters.
- If one end point coordinate and the Angle are programmed, the control calculates the unknown end point and the Length.
- If one end point coordinate and the Length are programmed, the control calculates the unknown end point and the Angle. However, two possible solutions may exist for the unknown end point. When two possible solutions exist, the message, “Another end point exists” appears along with one of the two possible solutions. To determine the correct end point:
  - Select the Draw console key for the currently displayed end point. If this solution is correct, press Enter to accept.
  - Select the FIND ANOTHER end point F5 softkey to view the alternative solution. Select the Draw console key again to view the alternate solution. Press Enter to accept.
  - Select the FIND ANOTHER end point F5 softkey to review the first end point solution.
Press the Ctrl + arrow console key to program additional Line and Arc segments for the current data block.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>1</strong></td>
<td>Known Start Point</td>
</tr>
<tr>
<td><strong>2</strong></td>
<td>Solution #1</td>
</tr>
<tr>
<td><strong>3</strong></td>
<td>Solution #2</td>
</tr>
<tr>
<td><strong>4</strong></td>
<td>X Known</td>
</tr>
<tr>
<td><strong>5</strong></td>
<td>XY Length Known</td>
</tr>
<tr>
<td><strong>6</strong></td>
<td>Y Unknown</td>
</tr>
<tr>
<td><strong>7</strong></td>
<td>XY Angle Unknown</td>
</tr>
</tbody>
</table>

*Figure 2–70. Line Segment with Unknown End and Angle*
**Arc**

From the New Radial Contour Segment screen, select Arc F2, and the Radial Mill Contour Arc Segment screen appears with fields based on your selection in the Coordinates field (Rotary or Linear).

When Rotary is selected in the COORDINATES field, these fields appear:

![Figure 2–71. Live-Tooling Radial Mill Contour Arc Segment screen: Rotary Coordinates](image)

- **DIRECTION**—defines the direction, clockwise or counterclockwise, of the arc from the start point.
- **Z END**—defines the Z end point of the arc segment.
- **C END (DEG)**—defines the angle of the arc segment end point, measured counterclockwise from the home position.
- **Z START**—contains the Z start point for the segment. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **C START (DEG)**—contains the angle of the arc segment start point, measured counterclockwise from the home position. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **Z CENTER**—defines the Z location of the arc's center point.
- **C CENTER (DEG)**—defines the angle of the arc segment center point, measured counterclockwise from the home position.
- **RADIUS**—defines the radius of the arc.
- **FEED**—contains the feed rate from the previous segment. This value can be accepted or changed.
When Linear is selected in the COORDINATES field, these fields appear:

- **DIRECTION**—defines the direction, clockwise or counterclockwise, of the arc from the start point.
- **X' END**—defines the X location of the arc’s end point. The X’ axis will move in the physical Z direction.
- **Y’ END**—defines the Y location of the arc’s end point. The Y’ axis will be wrapped on the cylinder diameter.
- **X’ START**—contains the X start point for the segment. The X’ axis will move in the physical Z direction. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **Y’ START**—contains the Y start point for the segment. The Y’ axis will be wrapped on the cylinder diameter. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **X’ CENTER**—defines the X location of the arc’s center point.
- **Y’ CENTER**—defines the Y location of the arc’s center point. The Y’ axis will be wrapped on the cylinder diameter.
- **RADIUS**—defines the radius of the arc.
- **FEED**—contains the feed rate from the previous segment. This value can be accepted or changed.

*Figure 2–72. Live-Tooling Radial Mill Contour Arc Segment screen: Linear Coordinates*
Examples of these coordinates for a clockwise arc are shown below:

![Figure 2–73. Coordinates of a Clockwise Arc](image)

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Start Point</td>
</tr>
<tr>
<td>2</td>
<td>End Point</td>
</tr>
<tr>
<td>3</td>
<td>Center Point</td>
</tr>
</tbody>
</table>

The Calc-Assist feature calculates certain unknown arc segment dimensions automatically, after sufficient data has been entered. A parameter with a calculated value is indicated with “CAL” preceding the value.

- The values of center points plus start point or end point coordinates are used to calculate the arc radius.
- Start point and end point coordinates plus the radius are used to calculate two possible center point coordinates.
- Either of the end point values and the center point coordinate supplies the value of the other end point and the radius parameter.
- A known center point, start point, or end point and radius are used to calculate an unknown center point coordinate.
- If any coordinate (start point, center point, or end point) is important to the construction of the two segments to be blended, the segment must be programmed as an arc and not as a blend arc.

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

From the New Radial Contour Segment screen, select Blend Arc F3, and the Radial Mill Contour Blend Arc Segment screen appears with fields based on your choice in the Coordinates field (Rotary or Linear).

When Rotary is selected in the COORDINATES field, these fields appear:

- **RADIUS**—defines the radius of the blend arc.
- **FEED**—contains the feed rate from the previous segment. This value can be accepted or changed.
- **DIRECTION**—defines the direction, clockwise or counterclockwise, of the arc from the start point.
- **Z START**—contains the Z start point for the segment. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **C START (DEG)**—contains the angle of the arc segment start point, measured counterclockwise from the home position. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **Z END**—defines the Z end point of the blend arc segment. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **C END (DEG)**—defines the angle of the blend arc segment end point, measured counterclockwise from the home position. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **Z CENTER**—defines the Z location of the blend arc's center point. This read-only field is carried forward from the previous segment and can only be edited in that screen.

- **C CENTER (DEG)**—defines the angle of the blend arc segment center point, measured counterclockwise from the home position. This read-only field is carried forward from the previous segment and can only be edited in that screen.

When Linear is selected in the COORDINATES field, these fields appear:

![Figure 2–75. Live-Tooling Radial Mill Contour Blend Arc Segment screen: Linear Coordinates](image)

- **RADIUS**—defines the radius of the blend arc.
- **FEED**—contains the feed rate from the previous segment. This value can be accepted or changed.
- **DIRECTION**—defines the direction, clockwise or counterclockwise, of the blend arc from the start point.
- **X’ START**—contains the X start point for the segment. The X’ axis will move in the physical Z direction. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **Y’ START**—contains the Y start point for the segment. The Y’ axis will be wrapped on the cylinder diameter. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **X’ END**—defines the X location of the blend arc’s end point. The X’ axis will move in the physical Z direction. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **Y’ END**—defines the Y location of the blend arc’s end point. The Y’ axis will be wrapped on the cylinder diameter. This read-only field is carried forward from the previous segment and can only be edited in that screen.
• **X’ CENTER**—defines the X location of the blend arc’s center point. This read-only field is carried forward from the previous segment and can only be edited in that screen.

• **Y’ CENTER**—defines the Y location of the blend arc’s center point. The Y’ axis will be wrapped on the cylinder diameter. This read-only field is carried forward from the previous segment and can only be edited in that screen.

A blend arc joins two other segments and is tangent to both. A blend arc can be used to join these types of segments:

- two Line segments.
- a Line segment and an Arc segment.
- 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 diagram below illustrates some examples of blend arcs.

```
Two Lines joined by a Blend Arc
1 X/Y Start
2 X/Y End
3 Segment 1 (Line)
4 Segment 1 End/
  Segment 3 Start
  (Point of Intersection)
5 Segment 2 (Blend Arc)
6 Segment 3 (Line)

Line and Arc Joined by a Blend Arc
1 X/Y Start
2 X/Y End
3 Segment 1 (Line)
4 Segment 1 End/
  Segment 3 Start
  (Point of Intersection)
5 Segment 2 (Blend Arc)
6 Segment 3 (Arc)

Two Arcs Joined by a Blend Arc
1 X/Y Start
2 X/Y End
3 Segment 1 (Arc)
4 Segment 1 End/
  Segment 3 Start
  (Point of Intersection)
5 Segment 2 (Blend Arc)
6 Segment 3 (Arc)
```

*Figure 2–76. Blend Arc Examples*

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

- The first or last segment of a Radial Mill Contour data block cannot be blend arc segments.
- Blend arc segments cannot be adjacent to one another in a program. For example, if segment #2 is a blend arc, neither segment #1 nor #3 can be blend arc segments.
• Segments that are adjacent to the blend arc segment must intersect at some point. Therefore, if segment #2 is a blend arc, segments #1 and #3 must intersect at some projected point.

• The Radius of a blend arc segment cannot be larger than the width between the adjoining segments endpoints.

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

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

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

  Develop a part program segment by segment, or create the geometry surrounding an arc, then insert the blend arc between existing segments.

Radial Circle

From the New Block (Radial Milling) softkey menu, select RADIAL CIRCLE F2, and the Radial Mill Circle screen appears with Process and Geometry tabs. Program the tool and cutting information using the Process tab. Describe the finish shape using the Geometry tab.

Figure 2–77. Live-Tooling Radial Mill Circle screen
Process Tab

The program determines how many rough passes to make to remove the material based on the entries in the Process tab fields. The Radial Circle screen Process tab fields are defined as follows:

- **TOOL**—identifies the tool number that will be used in the part program.
- **TOOL OFFSETS**—identifies the tool offset and orientation programmed in Tool Setup that will be used. The tool offset defaults to the same number as the tool.
- **TOOL DIAMETER**—contains the tool diameter programmed in Tool Setup for the current tool. This field is read-only and can only be edited in Tool Setup.
- **STRATEGY**—identifies the tool path with respect to rough or finish.
  - **ROUGH AND FINISH**—perform rough and finish passes.
  - **ROUGH ONLY**—perform a rough pass only.
  - **FINISH ONLY**—perform a finish pass only.
- **MILLING TYPE**—identifies on which side of the contour the tool should begin cutting. Refer to Cutter Compensation, on page 2-188 for more details. Select from among these choices:
  - **ON**—used for cutting directly on the programmed path.
  - **INSIDE**—causes the tool to enter the part inside the contour and blend into it using a 180° arc. Cutter compensation is automatically employed and the outside edge of the tool follows the contour (the direction the tool travels depends upon milling direction). The point where the tool enters the part is determined by the blend offset and the point where the tool is withdrawn is determined by the blend overlap and blend offset.
  - **OUTSIDE**—similar to the Inside function, except the tool enters the part outside the circle and follows the outside of the contour.
  - **IN/TAN (INSIDE TANGENT)**—causes the tool to enter the part adjacent to the inside contour of the circle. Cutter compensation is automatically used and the edge of the tool follows the inside of the contour. The direction the tool travels depends upon the milling direction. The tool is withdrawn from the part adjacent to the contour.
  - **OUT/TAN (OUTSIDE TANGENT)**—similar to the Inside Tangent function except that the tool enters the part adjacent to the outside contour of the circle. The tool’s edge then follows the outside of the contour. The tool is withdrawn from the part adjacent to the contour.
  - **POCK BOUNDARY (Pocket Boundary)**—used for cutting the defined closed contour.

Please refer to Axial Circle, on page 2-147 for Axial Mill Circle screen Process tab field definitions.
• **SPEED (RPM)**—identifies the speed at which the spindle rotates to cut the part. This value is carried forward for the selected tool from Tool Setup. “CAL” appears next to the field. To accept this value, select Enter. A different speed can be programmed for this specific block.

  ➜ If a Speed value is changed in a milling data block, a pop-up message appears asking if you want to change the corresponding Feed value in the data block.

Override the displayed value by typing in a new speed or by using the console Spindle override knob.

• **MILL FEED**—identifies the cutting feedrate. This value is carried forward for the selected tool from Tool Setup. “CAL” appears next to the field. To accept this value, select Enter. A different mill feed rate can be programmed for this specific block.

  ➜ If a Speed value is changed in a milling data block, a pop-up message appears asking if you want to change the corresponding Feed value in the data block.

• **PLUNGE FEED**—identifies the feedrate between Z Start and Z Bottom.

These fields appear based on your selection in the previous fields:

• **POCKET TYPE**—identifies the pocketing type as Inward or Outward. This field is available when Pocket Boundary is selected for the Milling Type.
  - **INWARD**—cuts in from the edge of the defined boundary.
  - **OUTWARD**—cuts from the center of the feature outward to the defined boundary.

• **POCKET OVERLAP (%)**—controls the percentage of overlap during cutting. This field is available when Pocket Boundary is selected for the Milling Type.

• **WALL ALLOWANCE**—identifies the amount of material to leave along the wall of the stock during roughing for cutting the final pass. The wall of the stock is the side of the stock that is being cut. This field is not available when
  - Rough and Finish is selected for the Strategy and On is selected for the Milling Type.
  - Finish Only is selected for the Strategy.

• **FLOOR ALLOWANCE**—identifies the amount of material to leave on the bottom of the stock during roughing for cutting the final pass. This field is available when Pocket Boundary is selected for the Milling Type. This field is not available when Rough and Finish Strategy and Milling Type On are selected. This field is not available when Finish Only is selected for the Strategy.

• **BLEND OFFSET**—identifies the XY distance from the entry point of the part surface and the Z plunge point where the tool enters the work piece. This move is always a 90° arc and occurs at the 3:00 position on the circle. This parameter is used for the blend-in move in milling and for the blend-out move from the part surface to the Z Retract point. The default value is 3.000 mm (0.1250 inches). This field is available when either Inside, Outside, or Pocket Boundary is selected for the Milling Type.
**BLEND OVERLAP**—identifies the distance the tool travels past the entry point before exiting from the part. The default value is 3.000 mm (0.1250 inches). This field is available when either Inside, Outside, or Pocket Boundary is selected for the Milling Type.

**DIRECTION**—identifies the milling direction as either Conventional or Climb. This field is not available when On is selected for the Milling Type. Refer to *Cutter Compensation, on page 2 - 188* for details about milling direction.

**PECK DEPTH**—identifies the maximum depth to be cut in one pass. This field is not available when Finish Only is selected for the Strategy.

**1ST PECK OFFSET**—identifies the depth of the first peck when it needs to be different than subsequent pecks, identified in Peck Depth. This field is not available when Finish Only is selected for the Strategy.

**PECK CLEARANCE**—identifies the distance above the previous peck level to which the Z axis returns at rapid traverse. This field is not available when Finish Only is selected for the Strategy.

---

**Calculated Plunge Points for Radial Circles**

Calculations for determining the plunge points for milling cycles are for TMM, TMX MY, and TMX MYS Series machines. The following examples are for Radial Circles and Climb Milling.

The Blend-in move is a 90° arc and occurs at the 3:00 position on the circle.

**Pocket Type Inward**

\[
X_{\text{plunge}} = X_{\text{center}} + \text{Circle radius} - \text{Blend Offset} - \text{Tool radius.}
\]

\[
Y_{\text{plunge}} = Y_{\text{center}} - \text{Blend Offset}
\]

**Pocket Type Outward**

\[
X_{\text{plunge}} = X_{\text{center}} + \text{Circle radius} + \text{Blend Offset} + \text{Tool radius}
\]

\[
Y_{\text{plunge}} = Y_{\text{center}} + \text{Blend Offset}
\]

**Roughing**

To determine the plunge points for roughing passes, subtract the Wall Allowance from the Xplunge.
**Geometry Tab**

Finishing follows the geometry of the contour defined in the Geometry tab. The fields on the Geometry tab are defined as follows:

- **CYLINDER DIAMETER** or **RADIUS**—defines the diameter or radius of the cylinder.
- **COORDINATES**—defines the coordinates as either Linear or Rotary.
  - **Linear** coordinates are used when working from a flat drawing. When Linear is selected, the X coordinates move in the physical Z direction, the Y coordinates wrap around the C axis, and the Z coordinates move in the physical X direction.
  - **Rotary** coordinates move around the cylinder as specified.

The fields are based on your selection in the Coordinates field (Rotary or Linear).

When Rotary is selected in the COORDINATES field, these fields appear:

- **Z CENTER**—defines the Z coordinate for the center point of the circle.
- **C CENTER (DEG)**—defines the angle of the circle from the start point to the end point, measured counterclockwise from the home position.
- **DIAMETER** or **RADIUS**—defines the diameter or radius of the circle.
- **X START (DIA) or (RAD)**—defines the location where Plunge Feedrate begins. The feedrate is set in the Plunge Feed field in the Process tab.
- **X BOTTOM (DIA) or (RAD)**—defines the bottom of the hole and the location where the Plunge Feed Rate ends.

![Figure 2–78. Live-Tooling Radial Mill Circle screen Geometry tab: Rotary Coordinates](image-url)
When Linear is selected in the COORDINATES field, these fields appear:

- **X’ CENTER**—defines the X Startpoint for the first segment of the contour. The X’ axis will move in the physical Z direction.
- **Y’ CENTER**—defines the Y Startpoint for the first segment of the contour. The Y’ axis will be wrapped on the cylinder diameter.
- **DIAMETER** or **RADIUS**—defines the diameter or radius of the circle.
- **C POSITION**—defines the angle to position the C axis for the first segment of the contour. This field appears when the Linear Y Motion checkbox is selected. This field is available with TMX MY and TMX MYS series machines.
- **LINEAR Y MOTION**—select this checkbox to cause Y-Axis Motion using the X and X’ axes. When this checkbox is clear, Y-Axis Motion occurs using the X and C axes. Please refer to *Programming Coordinates and Linear Y Motion, on page 2 - 68* for examples. This field is available with TMX MY and TMX MYS series machines.

⚠️ Linear Y-Axis Motion is limited. The amount of Y travel is affected by the X position and by any X-Axis tool length offset.

- **Z’ START**—defines the location where Plunge Feedrate begins. The feedrate is set in the Plunge Feed field in the Process tab. The Z’ axis will move in the physical X direction.
- **Z’ BOTTOM**—defines the bottom of the hole and the location where the Plunge Feedrate ends. The Z’ axis will move in the physical X direction.
These fields are read-only and are carried over from Tool Setup for the tool specified in the Process tab.

- **TOOL**—identifies the tool number that will be used in the part program.
- **TOOL OFFSETS**—identifies the tool offset and orientation programmed in Tool Setup that will be used. The tool offset defaults to the same number as the tool.
- **TOOL DIAMETER**—contains the tool diameter programmed in Tool Setup for the current tool. This field is read-only and can only be edited in Tool Setup.

### Radial Frame

From the New Block (Radial Milling) softkey menu, select Radial Frame F3, and the Radial Mill Frame screen appears with Process and Geometry tabs. Program the tool and cutting information using the Process tab. Describe the finish shape using the Geometry tab.

#### Process Tab

The program determines how many rough passes to make to remove the material based on the entries in the Radial Mill Frame screen Process tab fields.

*Figure 2–80. Live-Tooling Radial Mill Frame screen Process tab*

The Radial Mill Frame screen Process tab fields are defined as follows:

- Please refer to Axial Frame, on page 2 - 153 for Axial Mill Frame screen Process tab field definitions.
  
- **TOOL**—identifies the tool number that will be used in the part program.
- **TOOL OFFSETS**—identifies the tool offset and orientation programmed in Tool Setup that will be used. The tool offset defaults to the same number as the tool.
• **TOOL DIAMETER**—contains the tool diameter programmed in Tool Setup for the current tool. This field is read-only and can only be edited in Tool Setup.

• **STRATEGY**—identifies the tool path with respect to rough or finish.
  - **ROUGH AND FINISH**—perform rough and finish passes.
  - **ROUGH ONLY**—perform a rough pass only.
  - **FINISH ONLY**—perform a finish pass only.

• **MILLING TYPE**—identifies on which side of the contour the tool should begin cutting. Refer to *Cutter Compensation, on page 2 - 188* for more details. Select from among these choices:
  - **ON**—used for cutting directly on the programmed path.
  - **INSIDE**—causes the tool to enter the part inside the contour of the frame and blend into it using a 180° arc. Cutter compensation is automatically employed and the outside edge of the tool follows the contour (the direction the tool travels depends upon milling direction). The point where the tool enters the part is determined by the blend offset and the point where the tool is withdrawn is determined by the blend overlap and blend offset. The following is an example of an Inside frame:
    
    ![Figure 2–81. Inside Mill Frame Operation](image)
    
    **Figure 2–81. Inside Mill Frame Operation**

  - **OUTSIDE**—similar to the Inside function, except the tool enters the part outside the frame and follows the outside of the contour. The tool is withdrawn adjacent to the frame.
  - **IN/TAN (INSIDE TANGENT)**—causes the tool to enter the part adjacent to the inside frame contour. Cutter compensation is automatically used and the edge of the tool follows the inside of the contour. The direction the tool travels depends upon the milling direction. The tool is withdrawn from the part adjacent to the frame.
  - **OUT/TAN (OUTSIDE TANGENT)**—similar to the Inside Tangent function except that the tool enters the part adjacent to the outside frame contour. The tool’s edge then follows the outside of the contour. The tool is withdrawn from the part adjacent to the frame.
  - **POCK BOUNDARY (Pocket Boundary)**—used for cutting the outside perimeter of a pocket operation.
• **SPEED (RPM)**—identifies the speed at which the spindle rotates to cut the part. This value is carried forward for the selected tool from Tool Setup. “CAL” appears next to the field. To accept this value, select Enter. A different speed can be programmed for this specific block.

  ⇒ If a Speed value is changed in a milling data block, a pop-up message appears asking if you want to change the corresponding Feed value in the data block.

Override the displayed value by typing in a new speed or by using the console Spindle override knob.

• **MILL FEED**—identifies the cutting feedrate. This value is carried forward for the selected tool from Tool Setup. “CAL” appears next to the field. To accept this value, select Enter. A different mill feed rate can be programmed for this specific block.

  ⇒ If a Feed value is changed in a milling data block, a pop-up message appears asking if you want to change the corresponding Speed value in the data block.

• **PLUNGE FEED**—identifies the feedrate between Z Start and Z Bottom.

These fields appear based on your selection in the previous fields:

• **POCKET TYPE**—identifies the pocketing type as Inward or Outward. This field is available when Pocket Boundary is selected for the Milling Type.
  
  • **INWARD**—cuts in from the edge of the defined boundary in.
  
  • **OUTWARD**—cuts from the center of the feature outward to the defined boundary.

• **POCKET OVERLAP (%)**—controls the percentage of overlap during cutting. This field is available when Pocket Boundary is selected for the Milling Type.

• **WALL ALLOWANCE**—identifies the amount of material to leave along the wall of the stock for cutting the final pass. The wall of the stock is the side of the stock that is being cut. This field is **not** available when
  
  • Rough and Finish is selected for the Strategy and On is selected for the Milling Type.
  
  • Finish Only is selected for the Strategy.

• **FLOOR ALLOWANCE**—identifies the amount of material to leave on the bottom of the stock for cutting the final pass. This field is available when Pocket Boundary is selected for the Milling Type. This field is **not** available when Finish Only is selected for the Strategy.
- **BLEND OFFSET**—identifies the XY distance from the entry point of the part surface and the Z plunge point where the tool enters the work piece. This move is always a 90° arc and occurs at the 6:00 position on the frame. This parameter is used for the blend-in move in milling and for the blend-out move from the part surface to the Z Retract point. The default value is 3.000 mm (0.1250 inches). This field is available when either Inside, Outside, or Pocket Boundary is selected for the Milling Type.

- **BLEND OVERLAP**—identifies the distance the tool travels past the entry point before exiting from the part. The default value is 3.000 mm (0.1250 inches). This field is available when either Inside, Outside, or Pocket Boundary is selected for the Milling Type.

- **DIRECTION**—identifies the milling direction as either Conventional or Climb. This field is not available when On is selected for the Milling Type. Refer to *Cutter Compensation, on page 2 - 188* for details about milling direction.

- **PECK DEPTH**—identifies the maximum depth to be cut in one pass. This field is not available when Finish Only is selected for the Strategy.

- **1ST PECK OFFSET**—identifies the depth of the first peck when it needs to be different than subsequent pecks, identified in Peck Depth. This field is not available when Finish Only is selected for the Strategy.

- **PECK CLEARANCE**—identifies the distance above the previous peck level to which the Z axis returns at rapid traverse. This field is not available when Finish Only is selected for the Strategy.

- **CUTTER COMP**—identifies the type of milling to insert between segment end points. Choose between Arc or Lines. This field is not available when On is selected for the Milling Type.
  - If Arc is selected, a tangent arc is inserted to connect two line segments, or a line segment and an arc segment (when the two cutter compensated segments are offset and do not intersect). When using the Arc method, the system creates the cutter compensated path as shown below:

```
1 Programmed tool path
2 Cutter compensated path
3 Arc parameter
```

*Figure 2–82. Cutter Compensation Using the Arc Parameter*

Some operators find the Arc method inappropriate for their applications because the cutter remains in contact with the work piece and may drag a chip across a sharp angle on the part. In such circumstances, operators select Lines.
• If the *Lines* softkey is selected, the cutter compensated lines and arcs are joined as described below:
  
  • Two line segments are extended until they intersect (provided they form a 90° or greater angle). If the lines form an angle of less than 90°, a line is inserted to connect them.
  
  • Line and arc segments have the line segment extended, and a tangent line to the arc segment inserted and extended until the lines intersect (provided they form a 90° or greater angle). If the segments form an angle of less than 90°, a line is inserted to connect them.
  
  • Two arc segments have tangent lines (to the arcs) inserted and extended until the lines intersect (provided the extended tangent lines form a 90° or greater angle). If the extended tangent lines form an angle of less than 90°, a line or arc is inserted to connect them.

![Figure 2-83. Cutter Compensation Using the Line Parameter](image)

**Calculated Plunge Points for Radial Frames**

Calculations for determining the plunge points for milling cycles are for TMM, TMX MY, and TMX MYS Series machines. The following examples are for Radial Frames and Climb Milling.

The Blend-in move is a 90° arc and occurs at the 6:00 position on the frame.

**Pocket Type Inward**

Xplunge = Xcorner + (XLength/2) - Blend Offset

Yplunge = Ycorner + Blend Offset + Tool radius

**Pocket Type Outward**

Xplunge = Xcorner + (XLength/2) + Blend Offset

Yplunge = Ycorner - Blend Offset - Tool radius

**Roughing**

To determine the plunge points for roughing passes, subtract the Wall Allowance from the Yplunge.
Geometry Tab

Finishing follows the geometry of the contour defined in the Geometry tab. The fields on the Geometry tab are defined as follows:

- **CYLINDER DIAMETER** or **RADIUS**—defines the diameter or radius of the cylinder.
- **COORDINATES**—defines the coordinates as either Linear or Rotary.
  - **Linear** coordinates are used when working from a flat drawing. When Linear is selected, the X coordinates move in the physical Z direction, the Y coordinates wrap around the C axis, and the Z coordinates move in the physical X direction.
  - **Rotary** coordinates move around the cylinder as specified.

The remaining fields are based on your selection in the Coordinates field (Rotary or Linear).

When Rotary is selected in the COORDINATES field, these fields appear:

- **Z CORNER**—defines the Z coordinate of the reference corner of the frame.
- **C CORNER (DEG)**—defines the angle of the circle from the start point to the end point, measured counterclockwise from the home position.
- **Z LENGTH**—defines the length of the frame along the Z axis.
- **C LENGTH (DEG)**—defines the length of the frame, in degrees, along the C axis, measured counterclockwise from the home position.
- **CORNER RADIUS**—defines the coordinate of any one of the four corners of the frame, which becomes the reference corner.
- **X START (DIA) or (RAD)**—defines the location where Plunge Feedrate begins. The feedrate is set in the Plunge Feed field in the Process tab.
- **X BOTTOM (DIA) or (RAD)**—defines the bottom of the hole and the location where the Plunge Feed Rate ends.

When Linear is selected in the COORDINATES field, these fields appear:

![Image of Live-Tooling Radial Mill Frame screen Geometry tab: Linear Coordinates]

- **X' CORNER**—defines the X Startpoint for the first segment of the contour. The X’ axis will move in the physical Z direction.
- **Y' CORNER**—defines the Y Startpoint for the first segment of the contour. The Y’ axis will be wrapped on the cylinder diameter.
- **X' LENGTH**—defines the X coordinate measured from the reference corner. If the reference corner is at the left side of the rectangle, the X length is a positive (+) dimension. If the reference corner is at the right side, the X length is negative (-).
- **Y' LENGTH**—defines the Y coordinate measured from the reference corner. Y Length is positive (+) if the reference corner is at the lower left or lower right of the rectangular area. Y Length is negative (-) if the reference corner is at the top left or top right of the rectangle.
- **C POSITION**—defines the angle to position the C axis for the first segment of the contour. This field appears when the Linear Y Motion checkbox is selected. This field is available with TMX MY and TMX MYS series machines.
- **LINEAR Y MOTION**—select this checkbox to cause Y-Axis Motion using the X and X’ axes. When this checkbox is clear, Y-Axis Motion occurs using the X and C axes. Please refer to *Programming Coordinates and Linear Y Motion*, on page 2 - 68 for examples. This field is available with TMX MY and TMX MYS series machines.

⚠️ Linear Y-Axis Motion is limited. The amount of Y travel is affected by the X position and by any X-Axis tool length offset.
• **CORNER RADIUS**—defines the coordinate of any one of the four corners of the frame, which becomes the reference corner.

• **Z' START**—defines the location where Plunge Feedrate begins. The feedrate is set in the Plunge Feed field in the Process tab. The Z’ axis will move in the physical X direction.

• **Z' BOTTOM**—defines the bottom of the hole and the location where the Plunge Feedrate ends. The Z’ axis will move in the physical X direction.

These fields are read-only and are carried over from Tool Setup for the tool specified in the Process tab.

• **TOOL**—identifies the tool number that will be used in the part program.

• **TOOL OFFSETS**—identifies the tool offset and orientation programmed in Tool Setup that will be used. The tool offset defaults to the same number as the tool.

• **TOOL DIAMETER**—contains the tool diameter programmed in Tool Setup for the current tool. This field is read-only and can only be edited in Tool Setup.

**Radial Slot**

From the New Block (Radial Milling) softkey menu, select Radial Slot F4, and the Radial Slot screen appears with Process and Geometry tabs. Program the tool and cutting information using the Process tab. Describe the finish shape using the Geometry tab.

**Process Tab**

The program determines how many rough passes to make to remove the material based on the entries in the Process tab fields.

![Image of Radial Slot Process Tab](image-url)

*Figure 2–86. Live-Tooling Radial Mill Radial Slot screen Process tab*
The Radial Slot screen Process tab fields are defined as follows:

- **TOOL**—identifies the tool number that will be used in the part program.
- **TOOL OFFSETS**—identifies the tool offset and orientation programmed in Tool Setup that will be used. The tool offset defaults to the same number as the tool.
- **TOOL DIAMETER**—contains the tool diameter programmed in Tool Setup for the current tool. This field is read-only and can only be edited in Tool Setup.
- **STRATEGY**—identifies the tool path with respect to rough or finish.
  - **ROUGH AND FINISH**—perform rough and finish passes.
  - **ROUGH ONLY**—perform a rough pass only.
  - **FINISH ONLY**—perform a finish pass only.
- **STEP OVER**—identifies the distance (a percentage of the tool diameter) to overlap between passes.
- **WALL ALLOWANCE**—identifies the amount of material to leave along the wall of the stock for cutting the final pass. The wall of the stock is the side of the stock that is being cut. This field is not available when Finish Only is selected for the Strategy.
- **FLOOR ALLOWANCE**—identifies the amount of material to leave on the bottom of the stock for cutting the final pass. This field is not available when Finish Only is selected for the Strategy.
- **SPEED (RPM)**—identifies the speed at which the spindle rotates to cut the part. This value is carried forward for the selected tool from Tool Setup. “CAL” appears next to the field. To accept this value, select Enter. A different speed can be programmed for this specific block.
  
  If a Speed value is changed in a milling data block, a pop-up message appears asking if you want to change the corresponding Feed value in the data block.

Override the displayed value by typing in a new speed or by using the console Spindle override knob.

- **DIRECTION**—identifies the milling direction as either Conventional or Climb. Refer to *Cutter Compensation, on page 2 - 188* for details about milling direction.
- **MILL FEED**—identifies the cutting feedrate. This value is carried forward for the selected tool from Tool Setup. “CAL” appears next to the field. To accept this value, select Enter. A different mill feed rate can be programmed for this specific block.

  If a Feed value is changed in a milling data block, a pop-up message appears asking if you want to change the corresponding Speed value in the data block.

- **PLUNGE TYPE**—identifies the type of plunge motion:
  - **STRAIGHT**
  - **ZIG-ZAG**
  - **RAPID**

- **PLUNGE FEED**—identifies the feedrate between Z Start and Z Bottom.

- **RAMP ANGLE**—identifies the ramp angle for Zig-Zag plunge motion. This field is available when Zig-Zag is selected for the Plunge Type.

- **PECK DEPTH**—identifies the maximum depth to be cut in one pass.

- **SURFACE CLEARANCE**—sets the distance above the Z (Axial) or X (Radial) start point for the tool to move between passes to allow for rapid motions back to the start point for the next pass. This setting also applies when rapid motions are made to position for cutting depths after the first depth.

  **Example:**

  Surface Clearance = 1.27 mm (0.0500 in)
  Z Start = 2.54 mm (0.1000 in)
  Peck Depth = 5.08 mm (0.2000 in)

  In this example, the Z motion would be:

  1. Rapid to 3.81 mm (0.15 in) (Z Start + Surface Clearance)
  2. Feed to -2.54 mm (-0.1 in) to machine first pass.
  3. Rapid back to 3.81 mm (0.15 in) (Z Start + Surface Clearance)
  4. Rapid to -1.27 mm (-0.05 in) (Depth of Previous pass + Surface Clearance)
  5. Feed to -7.62 mm (-0.30 in) to machine second pass.
  6. Rapid back to 3.81 mm (0.15 in) (Z Start + Surface Clearance)
  7. Rapid to -6.35 mm (-0.25 in) (Depth of Previous pass + Surface Clearance)
  8. Feed to -12.7 mm (-0.5 in) to machine third pass.
  9. Rapid back to 3.81 mm (0.15 in) (Z Start + Surface Clearance)
  10. Repeat to final depth.
Geometry Tab

Finishing follows the geometry of the contour defined in the Geometry tab.

Finishing follows the geometry of the contour defined in the Geometry tab. The fields on the Geometry tab are defined as follows:

- **CYLINDER DIAMETER** or **RADIUS**—defines the diameter or radius of the cylinder.
- **COORDINATES**—defines the coordinates as either Linear or Rotary.
  - **Linear** coordinates are used when working from a flat drawing. When Linear is selected, the X coordinates move in the physical Z direction, the Y coordinates wrap around the C axis, and the Z coordinates move in the physical X direction.
  - **Rotary** coordinates move around the cylinder as specified.

The remaining fields are based on your selection in the Coordinates field (Rotary or Linear).

When Rotary is selected in the COORDINATES field, these fields appear:

- **Z START**—defines the Z coordinate of the reference corner of the slot.
- **C START (DEG)**—defines the angle of the start point to the end point, measured counterclockwise from the home position.
- **Z END**—defines the length of the slot along the Z axis.
- **C END (DEG)**—defines the length of the slot, in degrees, along the C axis, measured counterclockwise from the home position.
• **SLOT WIDTH**—defines the width of the slot.
• **START CAP**—accesses selections for the shape of the slot end for the Start of the slot:
  • **LINE**—produces a straight edge for the start cap with a corner radius the same size as the tool.

![Figure 2–88. End Cap = Line (R = 0.125) Start Cap = Line (R = 0)](image)

• **APPEND ARC**—produces an arc at the start cap with the arc being appended or added to the end of the slot.

![Figure 2–89. End Cap = Append Arc; Start Cap = Line (R = 0)](image)

• **INCLUDE ARC**—produces an arc at the start cap with the arc included within the geometry.

![Figure 2–90. End Cap = Include Arc; Start Cap = Line (R = 0)](image)

• **CORNER RADIUS**—defines the radius of the slot end.
• **END CAP**—accesses selections for the shape of the slot end for the End of the slot:
  • **LINE**
  • **APPEND ARC**
  • **INCLUDE ARC**

⇒ Refer to Start Cap examples above.
• **CORNER RADIUS**—defines the radius of the slot end.

• **X START (DIA) or (RAD)**—defines the location where Plunge Feedrate begins. The feedrate is set in the Plunge Feed field in the Process tab.

• **X BOTTOM (DIA) or (RAD)**—accesses selections for the bottom of the slot and the location where the Plunge Feed Rate ends.
  
  • **AT START (DIA) or (RAD)**
  
  • **AT END (DIA) or (RAD)**

When Linear is selected in the COORDINATES field, these fields appear:

![Figure 2–91. Live-Tooling Radial Mill Radial Slot screen Geometry tab: Linear Coordinates](image)

• **X’ START**—defines the X start point for the slot. The X’ axis will move in the physical Z direction.

• **Y’ START**—defines the Y start point for the slot. The Y’ axis will be wrapped on the cylinder diameter.

• **X’ END**—defines the X coordinate measured from the reference corner. If the reference corner is at the left side of the slot, the X length is a positive (+) dimension. If the reference corner is at the right side, the X length is negative (-).

• **Y’ END**—defines the Y coordinate measured from the reference corner. Y Length is positive (+) if the reference corner is at the lower left or lower right of the slot area. Y Length is negative (-) if the reference corner is at the top left or top right of the rectangle.

• **C POSITION**—defines the angle to position the C axis for the slot. This field appears when the Linear Y Motion checkbox is selected. This field is available with TMX MY and TMX MYS series machines.
- **LINEAR Y MOTION**—select this checkbox to cause Y-Axis Motion using the X and X’ axes. When this checkbox is clear, Y-Axis Motion occurs using the X and C axes. Please refer to *Programming Coordinates and Linear Y Motion, on page 2 - 68* for examples. This field is available with TMX MY and TMX MYS series machines.

  ! Warning: Linear Y-Axis Motion is limited. The amount of Y travel is affected by the X position and by any X-Axis tool length offset.

- **SLOT WIDTH**—defines the width of the slot.
- **START CAP**—accesses selections for the shape of the slot end for the Start of the slot:
  - **LINE**—produces a straight edge for the start cap with a corner radius the same size as the tool.
  
  ![Figure 2–92. End Cap = Line (R = 0.125) Start Cap = Line (R = 0)]

  - **APPEND ARC**—produces an arc at the start cap with the arc being appended or added to the end of the slot.

  ![Figure 2–93. End Cap = Append Arc; Start Cap = Line (R = 0)]

  - **INCLUDE ARC**—produces an arc at the start cap with the arc included in the geometry.

  ![Figure 2–94. End Cap = Include Arc; Start Cap = Line (R = 0)]
• **CORNER RADIUS**—defines the radius of the slot end.

• **END CAP**—accesses selections for the shape of the slot end for the End of the slot:
  • **LINE**
  • **APPEND ARC**
  • **INCLUDE ARC**

  ➞ Refer to Start Cap examples above.

• **CORNER RADIUS**—defines the coordinate of any one of the four corners of the slot, which becomes the reference corner.

• **Z’ START**—defines the location where Plunge Feedrate begins. The feedrate is set in the Plunge Feed field in the Process tab. The Z’ axis will move in the physical X direction.

• **Z’ BOTTOM**—accesses selections for the bottom of the slot and the location where the Plunge Feed Rate ends. The Z’ axis will move in the physical X direction.
  • **AT START (DIA) or (RAD)**
  • **AT END (DIA) or (RAD)**

These fields are read-only and are carried over from Tool Setup for the tool specified in the Process tab.

• **TOOL**—identifies the tool number that will be used in the part program.

• **TOOL OFFSETS**—identifies the tool offset and orientation programmed in Tool Setup that will be used. The tool offset defaults to the same number as the tool.

• **TOOL DIAMETER**—contains the tool diameter programmed in Tool Setup for the current tool. This field is read-only and can only be edited in Tool Setup.
Radial Lettering

Radial Lettering data blocks mill text into the outside surface of the cylinder. The Radial Lettering screen defaults to Linear Text Style. The Style field can be changed to Circular Text Style.

From the New Block (Radial Milling) softkey menu, select Radial Lettering \textit{F5}, and the Radial Lettering screen appears. Linear coordinates is the default. The Coordinates field can be changed to Rotary.

The fields in the upper pane on the Radial Lettering screen are defined as follows:

- **TOOL**—identifies the tool number that will be used in the part program.
- **TOOL OFFSETS**—identifies the tool offset and orientation programmed in Tool Setup that will be used. The tool offset defaults to the same number as the tool.
- **TOOL DIAMETER**—contains the tool diameter programmed in Tool Setup for the current tool. This field is read-only and can only be edited in Tool Setup.
- **SPEED (RPM)**—identifies the speed at which the spindle rotates to cut the part. This value is carried forward for the selected tool from Tool Setup. To accept this value, select Enter. A different speed can be programmed for this specific block. Override the displayed value by typing in a new speed or by using the appropriate console Spindle override knob.
- **MILL FEED**—identifies the cutting feedrate.
- **PLUNGE FEED**—identifies the feedrate between X/Z’ Start and X/Z’ Bottom.

The remaining fields on the Radial Lettering screen are defined as follows:

- **SELECT FONT** screen button—displays a Font window with Font and Size selections and a Sample pane. Height is set by default based on chosen size, and when the Width field contains 0, the font and Height field calculate width.
- **STYLE**—defines the text optimization as either \textit{Any} or \textit{Stick} in addition to a second field to specify style as \textit{Linear} or \textit{Circular}. Any and Linear are the defaults.
  - \textit{Any}—fonts that are not optimized for stick lettering.
  - \textit{Stick}—for use only with specialized machining fonts designed to cut stick letters.

\[\text{Selecting Stick does not convert fonts to Stick Lettering.}\]

- **Linear**—the tool cuts along a line.
- **Circular**—the tool cuts unwrapped (flat) text on an arc.
- **CYLINDER DIAMETER**—defines the diameter of the cylinder.
- **X/Z’ START**—defines the location where Plunge Feedrate beings. The feedrate is set in the Plunge Feed field. With Linear Coordinates, the Z’ axis will move in the physical X direction.
- **R PLANE**—defines an absolute position, relative to the surface of the part. The cycle will rapid to this location before feeding to X/Z’ Bottom.
- **X/Z’ BOTTOM**—defines the bottom coordinate and the location where the Plunge Feedrate ends. With Linear Coordinates, the Z’ axis will move in the physical X direction.

- **LINEAR Y MOTION** checkbox—select this checkbox to cause Y-Axis Motion using the X and X’ axes. When this checkbox is clear, Y-Axis Motion occurs using the X and C axes. Please refer to *Programming Coordinates and Linear Y Motion, on page 2 - 68* for examples. This field is available with TMX MY and TMX MYS series machines.

  ![Linear Y-Axis Motion Diagram]

  Linear Y-Axis Motion is limited. The amount of Y travel is affected by the X position and by any X-Axis tool length offset.

- **C**—defines the angle to position the C-Axis. This field appears when Linear Y Motion checkbox is selected. This field is available with TMX MY and TMX MYS series machines.

- **SPACE BETWEEN CHAR**—adds space between characters in addition to the space used with the selected font.

- **HEIGHT REF LOCATION**—specifies BOTTOM, CENTER, or TOP for the character.

- **HEIGHT**—defines the height of the text. When the Width field contains 0, the font and Height field calculate width.

- **WIDTH REF LOCATION**—specifies START, CENTER, or END of the character. The default is Start.

- **ORIENTATION**—defines the start angle of the direction for the lettering on the stock.

  ![Orientation Diagram]

- **TEXT**—contains the text for the lettering block.
- **COORDINATES**—specifies Linear or Rotary as the coordinates. The axes listed on the screen change depending upon this selection.

<table>
<thead>
<tr>
<th>Linear Style</th>
<th>Circular Style</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Rotary Coordinates</strong></td>
<td><strong>Linear Coordinates</strong></td>
</tr>
<tr>
<td>Z Reference</td>
<td>X' Reference</td>
</tr>
<tr>
<td>C Reference (Deg)</td>
<td>Y' Reference</td>
</tr>
<tr>
<td>X Start</td>
<td>Z' Start</td>
</tr>
<tr>
<td>R Plane</td>
<td>R Plane</td>
</tr>
<tr>
<td>X Bottom</td>
<td>Z' Bottom</td>
</tr>
</tbody>
</table>

**Linear Style**

The Reference Location is a combination of the Height Ref Loc (Top, Center, or Bottom) and the Width Ref Loc (Start, Center, or End).

- The Hurco example shows the Reference Location is Top, End of the text.
- The WinMax example shows the Reference Location is Top, Start of the text.

- **Z/X’ REFERENCE**—indicates the X’/Z coordinate.
- **C (Deg)/Y’ REFERENCE**—indicates the Y’/C (Deg) coordinate.
- **WIDTH**—defines the width of the text. When the default of 0.0000 is used, the font and Height calculate the width used with Linear Style.
Circular Style

The Circular Style example shows the Reference Location is Bottom, Start of the text. The Direction is Counterclockwise.

- **Z/X’ CENTER**—specifies the Center of the circle for the reference point.
- **C (DEG)/Y’ CENTER**—specifies the Center of the circle for the reference point.
- **WIDTH (DEG)**—programs the text width as a sweep angle. Enter an angular value from 0 to 360° when Circular Style is selected.
- **DIAMETER**—defines the diameter of the circle for circular text when Circular Style is selected.
- **DIRECTION**—defines the direction for circular text.
Radial Holes

From the New Block screen, select LIVE TOOLING → F3. Select Radial Holes F2 from the New Block Live Tooling softkey menu. The Radial Holes screen appears.

The Radial Holes screen defaults to the Drill Operation. When the cursor is in the OPERATION field, these softkey choices appear. Upon editing the last field in each holes operation, the cursor returns to the Operation field.

- **PREVIOUS HOLE OPERATION F1**—displays the previous Hole Operation in this data block. This softkey is not available for the first operation.
- **NEXT HOLE OPERATION F2**—displays the next Hole Operation in this data block. This softkey is not available for the last operation.
- **DELETE HOLE OPERATION F3**—deletes the current Hole Operation. Select this softkey and a message appears asking if you are sure you want to delete. Select OK or CANCEL.
- **INSERT HOLE OPERATION F7**—accesses New Radial Holes Operation screen with these Radial Holes softkey selections:
  - **RADIAL HOLE CYCLE F1**—accesses the Radial Holes Drill data block for programming different types of drilling cycles. Refer to Radial Hole Cycle, on page 2 - 115 for programming details.
  - **RADIAL LOCATIONS F2**—accesses the Radial Holes Locations data block for programming drilling locations. Refer to Radial Locations, on page 2 - 121 for programming details.
  - **RADIAL LOOP AROUND C-AXIS F3**—accesses the Radial Holes Loop Around C-Axis data block for programming holes around the C axis. Refer to Radial Loop Around C-Axis, on page 2 - 123 for programming details.
Radial Hole Cycle

From the New Radial Holes Operation screen, select Radial Hole Cycle F1. The Radial Holes Drill screen appears.

The fields on this screen are defined as follows:

- **TOOL**—identifies the tool number that will be used in the part program.
- **TOOL OFFSETS**—identifies the tool offset and orientation programmed in Tool Setup that will be used. The tool offset defaults to the same number as the tool.
- **X START (DIA) or (RAD)**—defines the point, relative to the surface of the part, where the tool moves when starting the cycle, before moving at Rapid to the R Plane.
- **R PLANE (DIA) or (RAD)**—defines an absolute position, relative to the surface of the part. The cycle will rapid to this location before feeding to X Bottom.
- **X BOTTOM (DIA) or (RAD)**—identifies the depth of the hole, relative to the surface of the part when Linear is selected in the Coordinates field of the Radial Locations screen, and relative to the centerline when Rotary is selected. This field is automatically adjusted for the drill tip if YES is selected for Tip Compensation.
- **TIP COMPENSATION**—accesses YES or NO selections to indicate whether the tool tip is included as part of the Z Bottom measurement. This field appears when the tool is a live drill or a live center drill. Z Bottom depth is automatically calculated for the drill tip when YES is selected and is based on the drill tip angle programmed in Tool Setup.
- **TOOL DIAMETER**—contains the tool diameter programmed in Tool Setup for the current tool. This field is read-only and can only be edited in Tool Setup.
- **SPEED (RPM)**—contains the spindle speed value programmed in Tool Setup. “CAL” appears next to the value as programmed in Tool Setup. You can accept or change this value. If you edit this field, “CAL” disappears, and the value is no longer calculated automatically.

- **PLUNGE FEED**—contains the feedrate between the Z START and Z BOTTOM positions. “CAL” appears next to the value as calculated by the control. You can accept or change this value. If you edit this field, “CAL” disappears.

- **CYCLE TYPE**—accesses the cycle types.
  - **Standard** drilling feeds to depth and retracts at Rapid to the retract dimension (R Plane). Standard is the default operation. Refer to Tap Cycle below for information about the Standard Tap cycle. Tool motion is as follows:
    1. Position to XY.
    2. Rapid to R Plane.
    3. Feed to Z Bottom.

*Figure 2–97. Standard tool motion*
- **Dwell** drills with a dwell at the bottom. Tool motion is as follows:
  1. Position to XY.
  2. Rapid to R Plane.
  3. Feed to Z Bottom.
  4. Dwell for specified number of seconds or revolutions (FPM or FPR).
  5. Rapid to R Plane.

![Figure 2–98. Dwell tool motion](image)

- **Peck Drilling** feeds to a peck depth and retracts at Rapid to the Retract dimension (R Plane).

  The tool returns to the R Plane between each peck. Tool motion is as follows:

![Figure 2–99. Peck Drilling tool motion](image)
• **Chip Breaker** feeds to the peck depth defined in the Peck Depth field, retracts at Rapid the distance set in the Peck Clearance field, then feeds to the next peck depth. This is repeated until the programmed depth is achieved. Tool motion is as follows:

![Chip Breaker tool motion](image)

- **Center Drill** cycle creates a pilot hole. You can use a center drill or a standard drilling hole to perform this operation.

- **Decreasing Peck Drilling Depth** (Decr Depth) cycle feeds to the programmed First Peck Depth, then continues cutting a decreasing amount to the final peck by the peck percent each pass until it reaches the programmed final peck depth or Z Bottom.

- **Decreasing Chip Breaker** (Decr Chp Bkr) cycle feeds to the programmed First Peck Depth, retracts at Rapid the distance set in the Peck Clearance field, then feeds to the next peck depth, cutting a decreasing amount to the final peck by the peck percent each pass until it reaches the programmed final peck depth or Z Bottom.
Drilling Operation

These fields appear for performing drilling operations based on the Cycle Type selection:

- **DWELL TIME (SEC)**—specifies the time in seconds the Z axis dwells after the cycle reaches Z Bottom and before retracting at Rapid. This field appears when either Dwell or Center Drill is selected for the Cycle Type.

- **PECK DEPTH**—defines the maximum depth to be cut in one pass. This field appears when either Peck, Chip Breaker, Decreasing Depth, or Decreasing Chip Breaker is selected for the Cycle Type.

- **FINAL PECK DEPTH**—defines the maximum depth to be cut in the final pass. This field appears when either Decreasing Depth or Decreasing Chip Breaker is selected for the Cycle Type.

- **PECK PERCENT**—defines the percent of the First Peck Depth. For example, if the Peck Percent is set at 50, subsequent pecks will equal a distance 50% smaller than the previous peck depth. A Minimum Peck Depth must be set in order for the tool to reach Z Bottom. This field appears when either Decreasing Depth or Decreasing Chip Breaker is selected for the Cycle Type.

- **PECK COUNT**—displays the number of pecks that will occur. This read-only field appears when either Peck, Chip Breaker, Decreasing Depth, or Decreasing Chip Breaker is selected for the Cycle Type.

- **PECK CLEARANCE**—defines the location above the last peck level for returning at rapid feed rate. The default is 0.254 mm (0.010 in). This field appears when either Peck, Chip Breaker, Decreasing Depth, or Decreasing Chip Breaker is selected for the Cycle Type.

Tapping Operation

This field appears for performing tapping operations when the tool is a Live Tap:

- **TAP CYCLE**—accesses Standard or Rigid Tap cycle selections. This field appears when the tool is a Live Tap.
  - **Standard** cuts threads inside a hole.
  - **Rigid** taps the same hole multiple times, maintaining the tool’s orientation with previously cut threads.

  When performing rigid tapping, the hole must be longer than the tap. To accomplish this, program the preceding drill operation to be longer than the tap operation.

Here are some suggestions when Tapping:

- Use a floating tap holder to prevent the tap from breaking when the spindle reverses direction.
- When tapping hard materials, use tapping fluid (not coolant) as a lubrication.
- When tapping hard materials, use a tap drill one size larger than normal because less thread engagement requires less torque.
- The recommended Z Start is 0.200 minimum from the part surface to compensate for floating tap holder characteristics.
These fields appear based on the Tap Cycle selection:

- **PITCH**—defines the distance between threads for metric tools. This read-only field, defined in Tool Setup, appears when the tool is a Live Tap.
- **TPI**—defines the threads per inch for tools measured in inches. This read-only field, defined in Tool Setup, appears when the tool is a Live Tap and the Tap Cycle is Standard.
- **DWELL TIME (SEC)**—specifies the time in seconds the Z axis dwells after the cycle reaches Z Bottom and before retracting at Rapid. This field appears when the Tap Cycle is Standard.
- **PECK DEPTH**—defines the maximum depth to be cut in one pass. This field appears when Rigid is selected for the Tap Cycle Type and the tool is a Live Tap.

**Reaming Operation**

These fields appear for performing reaming operations when the tool is a Live Ream:

- **RETRACT AT RAPID**—defines the retract as rapid or feed. Select Yes for rapid or No for feed. This field appears when the tool is a Live Ream.
- **DWELL TIME (SEC)**—specifies the time in seconds the Z axis dwells after the cycle reaches Z Bottom and before retracting at Feed. This field appears when the tool is a Live Ream and No is selected for Retract at Rapid.
Radial Locations

From the New Radial Holes Operation screen select Radial Locations Cycle F2. The Radial Holes Locations screen appears. The fields are defined as follows:

- **CYLINDER DIAMETER** or **RADIUS**—defines the diameter or radius of the cylinder.
- **COORDINATES**—defines the coordinates as either Linear or Rotary, relative to the part.
  - **Linear** coordinates are used when working from a flat drawing. When Linear is selected, the X coordinates move in the physical Z direction, the Y coordinates wrap around the C axis, and the Z coordinates move in the physical X direction.
  - **Rotary** coordinates move around the cylinder as specified.
- **Location Number**—identifies the sequential number for the location. When the fields for each location are filled in, press **Enter**. The next row fields becomes active and the Location Number field increments by one.

When the cursor is in one of the Location fields (Z, C (DEG), X’, Y’, or Retract) these softkey choices are available:

- **DELETE LOCATION** F4—delete a selected row of location fields.
- **ADD LOCATION** F5—add a row of location fields to the bottom of the list.
- **INSERT LOCATION BEFORE** F7—insert a row of location fields above the selected row.

The remaining fields are based on your selection in the Coordinates field (Rotary or Linear).

When Rotary is selected in the COORDINATES field, these fields appear:

![Figure 2–101. Live-Tooling Radial Locations screen: Rotary Coordinates](image-url)
• **USE C CLAMP checkbox**—engages the C Axis Clamp during the Hole operation when this checkbox is selected.

• **CYLINDER DIAMETER**—defines the diameter of the cylinder.

• **Z**—defines the Z offset value for the location.

• **C (DEG)**—defines the C offset value in degrees for the location, measured counterclockwise from the home position.

• **RETURN LEVEL**—identifies the level to which the tool should retract at the end of the drill cycle. R Plane is the default.

  • **I PLANE**—returns to the coordinate specified in the X/Z Start field.
  
  • **R PLANE**—returns to the coordinate specified in the R Plane field.

When Linear is selected in the COORDINATES field, these fields appear:

![Screen shot of Live-Tooling Radial Locations screen: Linear Coordinates](image)

*Figure 2–102. Live-Tooling Radial Locations screen: Linear Coordinates*

• **USE C CLAMP checkbox**—engages the C Axis Clamp during the Hole operation when this checkbox is selected.

• **CYLINDER DIAMETER**—defines the diameter of the cylinder.

• **X’**—defines the X’ offset value for the location.

• **Y’**—defines the Y’ offset value for the location.

• **RETURN LEVEL**—identifies the level to which the tool should retract at the end of the drill cycle. R Plane is the default.

  • **I PLANE**—returns to the coordinate specified in the X/Z Start field.
  
  • **R PLANE**—returns to the coordinate specified in the R Plane field.

• **C POSITION**—defines the angle to position the C axis for the first segment of the contour. This field appears when the Linear Y Motion checkbox is selected. This field is available with TMX MY and TMX MYS series machines.
- **LINEAR Y MOTION**—select this checkbox to cause Y-Axis Motion using the X and X’ axes. When this checkbox is clear, Y-Axis Motion occurs using the X and C axes. Please refer to *Programming Coordinates and Linear Y Motion, on page 2-68* for examples. This field is available with TMX MY and TMX MYS series machines.

⚠️ Linear Y-Axis Motion is limited. The amount of Y travel is affected by the X position and by any X-Axis tool length offset.

**Radial Loop Around C-Axis**

From the New Radial Holes Operation screen select Radial Loop Around C-Axis F3. The Radial Holes Loop Around C-Axis screen appears.

*Figure 2–103. Live-Tooling Radial Holes Loop Around C-Axis screen*

The fields on this screen are defined as follows:

- **USE C CLAMP checkbox**—engages the C Axis Clamp during the Hole operation when this checkbox is selected.
- **NUMBER OF HOLES**—defines the number of holes for the bolt circle.
- **Z OFFSET**—defines the Z offset for the circular hole pattern to be rotated around the cylinder.
- **RETURN LEVEL**—identifies the level to which the tool should retract at the end of the drill cycle. R Plane is the default.
  - **I PLANE**—returns to the coordinate specified in the X/Z Start field.
  - **R PLANE**—returns to the coordinate specified in the R Plane field.
- **START ANGLE**—defines the angle to the first hole, measured counterclockwise from the home position.
- **SKIP LIST**—provides fields for entering the hole number to be skipped moving in the counterclockwise direction.
When the cursor is in one of the Skip List fields, these softkeys are added to the menu:

- **CLEAR SKIP LIST ENTRY**  *F3*—deletes the entry, leaving the field empty.
- **CLEAR ENTIRE SKIP LIST**  *F4*—deletes all Skip List entries, leaving the fields empty.
- **CONVERT TO LOCATIONS BLOCK**  *F5*—converts the Loop Around C Axis data block to a Locations data block. Select this softkey and this message appears: “Warning! This block will be deleted. Continue?” Select OK or Cancel.
- **COPY TO LOCATIONS BLOCK**  *F6*—copies the Loop Around C Axis data block as a Locations data block.
Axial Milling

From the New Block screen, select LIVE TOOLING ➔ F3. Select AXIAL MILLING F4 from the New Block (Live Tooling) screen to access the Axial Milling softkey menu:

![Axial Milling Screen](image)

**Figure 2–104. Live-Tooling New Block Axial Milling screen**

- **AXIAL LINES AND ARCS** F1—accesses the Axial Mill Contour data block for programming segments with lines, arcs, and blend arcs. Refer to Axial Lines and Arcs, on page 2 - 128 for programming details.
- **AXIAL CIRCLE** F2—accesses the Axial Mill Circle data block. Refer to Axial Circle, on page 2 - 147 for programming details.
- **AXIAL FRAME** F3—accesses the Axial Mill Frame data block. Refer to Axial Frame, on page 2 - 153 for programming details.
- **AXIAL SLOT** F4—accesses the Axial Slot data block. Refer to Axial Slot, on page 2 - 160 for programming details.
- **AXIAL FLATS** F5—accesses the Axial Flats data block. Refer to Axial Flats, on page 2 - 168 for programming details.
- **AXIAL LETTERING** F6—accesses the Axial Lettering data block. Refer to Axial Lettering, on page 2 - 175 for programming details.

**Programming Coordinates and Linear Y Motion**

Axial milling is used with TMM, TMX MY, and TMX MYS machines. The programming coordinates are mapped to behave like milling machines. In order to perform machining processes for TMM, TMX MY, and TMX MYS machines, WinMax aligns the programming coordinate system. Machine coordinates do not change.
No Linear Y Motion

The following figure shows the part coordinate system for TMM, TMX MY, and TMX MYS machines, for use when the Y-Axis travel of the machine is inadequate for the geometry of the feature being machined.

TMM machines always use this mapping for Axial milling.

Figure 2–105. Main and Sub-spindle Axial surface of the part without Linear Y

Please refer to Axial Lines and Arcs, on page 2 - 128, Axial Circle, on page 2 - 147, Axial Frame, on page 2 - 153, Axial Slot, on page 2 - 160, Axial Flats, on page 2 - 168, or Axial Lettering, on page 2 - 175 for information about the Geometry tab screen for the different types of Axial Milling. For information about Axial Holes, please refer to Axial Holes, on page 2 - 178.
**Linear Y Motion**

The following figure shows the Y-Axis coordinate system mapping for TMX MY and TMX MYS machines. When Rectangular Coordinates and Linear Y Motion are selected on the Geometry tab, linear Y-axis motion occurs using the combined motion of the X and X’ physical axes.

⇒ TMM machines do not use this mapping.

![Diagram showing Y-Axis coordinate system mapping for TMX MY and TMX MYS machines](image)

**Main Spindle Linear Y selected Part Coordinate system**

**Sub-spindle Linear Y selected Part Coordinate system**

*Figure 2–106. Main and Sub-spindle Axial surface of the part with Linear Y*

Please refer to Axial Lines and Arcs, on page 2 - 128, Axial Circle, on page 2 - 147, Axial Frame, on page 2 - 153, Axial Slot, on page 2 - 160, Axial Flats, on page 2 - 168, or Axial Lettering, on page 2 - 175 for information about the Geometry tab screen for the different types of Axial Milling. For information about Axial Holes, please refer to Axial Holes, on page 2 - 178.
Axial Lines and Arcs

Select AXIAL LINES AND ARCS F1 from the Axial Milling New Block softkey menu, and the Axial Mill Contour screen appears with Process and Geometry tabs. Program the tool and cutting information using the Process tab. Describe the finish shape using the Geometry tab.

The program determines how many rough passes to make to remove the material based on the entries in the Process tab fields.

![Figure 2–107. Live-Tooling Axial Mill Contour Screen Process tab](image)

**Process Tab**

The Process tab fields are defined as follows:

- **TOOL**—identifies the tool number that will be used in the part program.
- **TOOL OFFSETS**—identifies the tool offset and orientation programmed in Tool Setup that will be used. The tool offset defaults to the same number as the tool.
- **TOOL DIAMETER**—contains the tool diameter programmed in Tool Setup for the current tool. This field is read-only and can only be edited in Tool Setup.
- **STRATEGY**—identifies the tool path with respect to rough or finish.
  - **ROUGH AND FINISH**—perform rough and finish passes.
  - **ROUGH ONLY**—perform a rough pass only.
  - **FINISH ONLY**—perform a finish pass only.
• **MILLING TYPE**—identifies on which side of the contour the tool should begin cutting. Refer to *Cutter Compensation, on page 2-188* for more details. Select from among these choices:
  - **ON**—used for cutting directly on the programmed path.
  - **LEFT**—used for climb milling.
  - **RIGHT**—used for conventional milling.
  - **PROFILE LEFT**—used for removing material from a closed contour.
  - **PROFILE RIGHT**—used for removing material from a closed contour.
  - **POCK BOUNDARY (POCKET BOUNDARY)**—used for cutting the defined closed contour.

• **SPEED (RPM)**—identifies the speed at which the spindle rotates to cut the part. This value is carried forward for the selected tool from Tool Setup. “CAL” appears next to the field. To accept this value, select Enter. A different speed can be programmed for this specific block.

  ⇒ If a Speed value is changed in a milling data block, a pop-up message appears asking if you want to change the corresponding Feed value in the data block.

Override the displayed value by typing in a new speed or by using the console Spindle override knob.

• **MILL FEED**—identifies the cutting feedrate. This value is carried forward for the selected tool from Tool Setup. “CAL” appears next to the field. To accept this value, select Enter. A different mill feed rate can be programmed for this specific block.

  ⇒ If a Feed value is changed in a milling data block, a pop-up message appears asking if you want to change the corresponding Speed value in the data block.

• **PLUNGE FEED**—identifies the feedrate between Z Start and Z Bottom.
These fields appear based on your selection in the previous fields:

- **ENABLE BLEND MOVES**—enables automatic blend in and blend out for arcs. This field is available for Left, Right, Profile Left, and Profile Right milling.

- **POCKET TYPE**—identifies the pocketing type. This field is available when Pocket Boundary is selected for the Milling Type.
  - **INWARD**—cuts in from the edge of the defined boundary.
  - **OUTWARD**—cuts from the center of the feature outward to the defined boundary.

- **POCKET OVERLAP (%)**—controls the percentage of overlap during cutting. This field is available when either Profile Left, Profile Right, or Pocket Boundary is selected for the Milling Type.

- **WALL ALLOWANCE**—identifies the amount of material to leave along the wall of the stock for cutting the final pass. The wall of the stock is the side of the stock that is being cut. This field is not available when
  - On is selected for the Milling Type.
  - Finish Only is selected for the Strategy.

- **FLOOR ALLOWANCE**—identifies the amount of material to leave on the bottom of the stock for cutting the final pass. This field is available when Pocket Boundary is selected for the Milling Type. This field is not available when Finish Only is selected for the Strategy.

- **BLEND OFFSET**—identifies the XY distance from the entry point of the part surface and the Z plunge point where the tool enters the work piece. This move is always a 90° arc. This parameter is used for the blend-in move in milling and for the blend-out move from the part surface to the Z Retract point. The default value is 3.000 mm (0.1250 inches). This field is available when Pocket Boundary is selected for the Milling Type.

- **BLEND OVERLAP**—identifies the distance the tool travels past the entry point before exiting from the part. The default value is 3.000 mm (0.1250 inches). This field is available when Pocket Boundary is selected for the Milling Type.

- **MAX OFFSET**—defines the distance away from the programmed contour where milling begins. This field is available when either Profile Left or Profile Right is selected for the Milling Type.

- **DIRECTION**—identifies the milling direction as either Conventional or Climb. This field is not available when On is selected for the Milling Type. Refer to *Cutter Compensation, on page 2 - 188* for details about milling direction.

- **PECK DEPTH**—identifies the maximum depth to be cut in one pass. This field is not available when Finish Only is selected for the Strategy.

- **1ST PECK OFFSET**—identifies the depth of the first peck when it needs to be different than subsequent pecks, identified in Peck Depth. This field is not available when Finish Only is selected for the Strategy.

- **PECK CLEARANCE**—identifies the distance above the previous peck level to which the Z axis returns at rapid traverse. This field is not available when Finish Only is selected for the Strategy.
• **CUTTER COMP**—identifies the type of milling to insert between segment end points. Choose between Arc or Lines. This field is not available when On is selected for the Milling Type.

• If Arc is selected, a tangent arc is inserted to connect two line segments, or a line segment and an arc segment (when the two cutter compensated segments are offset and do not intersect). When using the Arc method, the system creates the cutter compensated path as shown below:

![Figure 2–108. Cutter Compensation Using the Arc Parameter](image)

<table>
<thead>
<tr>
<th></th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Programmed tool path</td>
</tr>
<tr>
<td>2</td>
<td>Cutter compensated path</td>
</tr>
<tr>
<td>3</td>
<td>Arc parameter</td>
</tr>
</tbody>
</table>

Some operators find the Arc method inappropriate for their applications because the cutter remains in contact with the work piece and may drag a chip across a sharp angle on the part. In such circumstances, operators select Lines.
If the *Lines* softkey is selected, the cutter compensated lines and arcs are joined as described below:

- Two line segments are extended until they intersect (provided they form a 90° or greater angle). If the lines form an angle of less than 90°, a line is inserted to connect them.
- Line and arc segments have the line segment extended, and a tangent line to the arc segment inserted and extended until the lines intersect (provided they form a 90° or greater angle). If the segments form an angle of less than 90°, a line is inserted to connect them.
- Two arc segments have tangent lines (to the arcs) inserted and extended until the lines intersect (provided the extended tangent lines form a 90° or greater angle). If the extended tangent lines form an angle of less than 90°, a line or arc is inserted to connect them.

![Figure 2–109. Cutter Compensation Using the Line Parameter](image-url)

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Programmed tool path</td>
</tr>
<tr>
<td>2</td>
<td>Cutter compensated path</td>
</tr>
<tr>
<td>3</td>
<td>Insert Line parameter</td>
</tr>
</tbody>
</table>

*Figure 2–109. Cutter Compensation Using the Line Parameter*
**Geometry Tab**

The Axial Mill Contour screen appears with Process and Geometry tabs. Program the tool and cutting information using the Process tab. Describe the finish shape using the Geometry tab.

Finishing follows the geometry of the contour defined in the Geometry tab. The fields on the Geometry tab are defined as follows:

- **COORDINATES**—defines the coordinates as either Polar or Rectangular.
  - **Rectangular** coordinates are used when working from a flat drawing.
  - **Polar** coordinates move around the cylinder as specified.

The remaining fields are based on your selection in the Coordinates field (Polar or Rectangular).

When Polar is selected in the COORDINATES field, these fields appear:

- **R START**—defines the starting distance from the centerline.
- **C ANGLE (DEG)**—defines the angle of the line segment from the start point to the end point, measured counterclockwise from the home position.
- **Z START**—defines the location where Plunge Feedrate begins. The feedrate is set in the Plunge Feed field in the Process tab.
- **Z BOTTOM**—defines the bottom of the hole and the location where the Plunge Feed Rate ends.

*Figure 2–110. Live-Tooling Axial Mill Contour screen Geometry tab: Polar Coordinates*
When Rectangular is selected in the COORDINATES field, these fields appear:

![Figure 2–111. Live-Tooling Axial Mill Contour screen Geometry tab: Rectangular Coordinates](image)

- **X' START**—defines the X start point for the first segment of the contour.
- **Y' START**—defines the Y start point for the first segment of the contour. The Y' axis will move in the physical Z direction.
- **C POSITION**—defines the angle to position the C axis for the first segment of the contour. This field appears when the Linear Y Motion checkbox is selected. This field is available with TMX MY and TMX MYS series machines.
- **LINEAR Y MOTION**—select this checkbox to cause Y-Axis Motion using the X and X’ axes. When this checkbox is clear, Y-Axis Motion occurs using the X and C axes. Please refer to Programming Coordinates and Linear Y Motion, on page 2 - 68 for examples. This field is available with TMX MY and TMX MYS series machines.

Linear Y-Axis Motion is limited. The amount of Y travel is affected by the X position and by any X-Axis tool length offset.

- **Z START**—defines the location where Plunge Feedrate begins. The feedrate is set in the Plunge Feed field in the Process tab.
- **Z BOTTOM**—defines the bottom of the hole and the location where the Plunge Feedrate ends.

These fields are read-only and are carried over from Tool Setup for the tool specified in the Process tab.

- **TOOL**—identifies the tool number that will be used in the part program.
- **TOOL OFFSETS**—identifies the tool offset and orientation programmed in Tool Setup that will be used. The tool offset defaults to the same number as the tool.
- **TOOL DIAMETER**—contains the tool diameter programmed in Tool Setup for the current tool. This field is read-only and can only be edited in Tool Setup.
Segments

Axial Mill contour data blocks use segments to create lines and arcs in a part program. A segment is any single or combined axis movement at a programmed feedrate. A series of lines and arcs can be programmed in a single data block, using different segments, to form a complete contour. The contour can be programmed in either Polar or Rectangular Coordinates.

The first segment in an Axial Mill Contour block is indicated by the segment zero (0). Segment numbers are system generated and you cannot change them.

The cursor is initially positioned in the Tool field of the Process tab when the Segment 0 screen appears. Enter the number of the tool that will be used for the entire operation—all of the segments in this operation will use this tool.

Press the Ctrl + arrow console keys or the NEXT SEGMENT F2 softkey after Segment 0 is completed to create a new segment for the data block. Up to 999 segments can be programmed for each data block, if they all use the same tool and are part of the same Radial Mill Contour.

You cannot start or end an Axial Mill Contour segment with blend arcs, nor can blend arcs be adjacent to each other.

The New Axial Contour Segment screen appears with these softkey choices:

- **LINE F1**—accesses the Line segment screen. Refer to Line, on page 2 - 136 for programming details.
- **ARC F2**—accesses the Arc segment screen. Refer to Arc, on page 2 - 140 for programming details.
- **BLEND ARC F3**—accesses the Blend Arc segment screen. Refer to Blend Arc, on page 2 - 143 for programming details.
**Line**

From the New Axial Contour Segment screen, select LINE \( F1 \), and the Axial Mill Contour Line Segment screen appears with fields based on your choice in the Coordinates field (Polar or Rectangular).

When Polar is selected in the COORDINATES field, these fields appear:

![Figure 2–113. Live-Tooling Axial Mill Contour Line Segment screen: Polar Coordinates](image)

- **R END**—defines the starting distance from the centerline.
- **C END (DEG)**—defines the angle of the line segment from the start point to the end point, measured counterclockwise from the home position.
- **R START**—defines the starting distance from the centerline. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **C START (DEG)**—defines the angle of the line segment from the start point to the end point, measured counterclockwise from the home position. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **FEED**—contains the feed rate from the previous segment. This value can be accepted or changed.
When Rectangular is selected in the COORDINATES field, these fields appear:

![Figure 2–114. Live-Tooling Axial Mill Contour Line Segment screen: Rectangular Coordinates](image)

- **X END**—defines the X coordinate on a 2-D XY plane, starting from the center of the face.
- **Y END**—defines the Y coordinate on a 2-D XY plane, starting from the center of the face.
- **X START**—defines the X start point for the segment. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **Y START**—defines the Y start point for the segment. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **LENGTH**—identifies the length of the line.
- **ANGLE**—defines the angle of the line segment from the start point to the end point, measured counterclockwise from the home position.
- **FEED**—contains the feed rate from the previous segment. This value can be accepted or changed.
- **LINEAR Y MOTION**—select this checkbox to cause Y-Axis Motion using the X and X’ axes. When this checkbox is clear, Y-Axis Motion occurs using the X and C axes. Please refer to *Programming Coordinates and Linear Y Motion, on page 2 - 68* for examples. This field is available with TMX MY and TMX MYS series machines.

⚠️ Linear Y-Axis Motion is limited. The amount of Y travel is affected by the X position and by any X-Axis tool length offset.
The following figure shows the Length, Angle, and end point relationship:

![Figure 2–115. Line segment coordinates](image)

<table>
<thead>
<tr>
<th></th>
<th>Start Point</th>
<th>End Point</th>
<th>Length</th>
<th>Angle</th>
</tr>
</thead>
</table>

The Calc-Assist feature calculates certain unknown line segment dimensions automatically, after sufficient data has been entered. A parameter with a calculated value is indicated with “CAL” preceding the value.

- If the End coordinates are entered, the control automatically calculates the Length and the Angle values.
- If both end points are unknown, but Length and Angle are programmed, the control automatically calculates both End parameters.
- If one end point coordinate and the Angle are programmed, the control calculates the unknown end point and the Length.
- If one end point coordinate and the Length are programmed, the control calculates the unknown end point and the Angle. However, two possible solutions exist for the unknown end point. When two possible solutions exist, the message, “Another end point exists” appears along with one of the two possible solutions. To determine the correct end point:
  - Select the Draw console key for the currently displayed end point. If this solution is correct, press Enter to accept.
  - Select the FIND ANOTHER end point F5 softkey to view the alternative solution. Select the Draw console key again to view the alternate solution. Press Enter to accept.
  - Select the FIND ANOTHER end point F5 softkey to review the first end point solution.
Figure 2–116. Line Segment with Unknown End and Angle

<table>
<thead>
<tr>
<th></th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Known Start Point</td>
</tr>
<tr>
<td>2</td>
<td>Solution #1</td>
</tr>
<tr>
<td>3</td>
<td>Solution #2</td>
</tr>
<tr>
<td>4</td>
<td>X Known</td>
</tr>
<tr>
<td>5</td>
<td>XY Length Known</td>
</tr>
<tr>
<td>6</td>
<td>Y Unknown</td>
</tr>
<tr>
<td>7</td>
<td>XY Angle Unknown</td>
</tr>
</tbody>
</table>

Press the Ctrl + arrow console key to program additional Line and Arc segments for the current data block.
Arc

From the New Axial Contour Segment screen, select Arc F2, and the Axial Mill Contour Arc Segment screen appears with fields based on your choice in the Coordinates field (Polar or Rectangular).

When Polar is selected in the COORDINATES field, these fields appear:

![Figure 2–117. Live-Tooling Axial Mill Contour Arc Segment screen: Polar Coordinates](image)

- **DIRECTION**—defines the direction, clockwise or counterclockwise, of the arc from the start point.
- **R END**—defines the starting distance from the centerline.
- **C END (DEG)**—defines the angle of the arc segment end point, measured counterclockwise from the home position.
- **R START**—contains the starting distance from the centerline. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **C START (DEG)**—contains the angle of the arc segment start point, measured counterclockwise from the home position. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **R CENTER**—defines the starting distance from the centerline.
- **C CENTER (DEG)**—defines the angle of the arc segment center point, measured counterclockwise from the home position.
- **RADIUS**—defines the radius of the arc.
- **FEED**—contains the feed rate from the previous segment. This value can be accepted or changed.
When Rectangular is selected in the COORDINATES field, these fields appear:

![Figure 2–118. Live-Tooling Axial Mill Contour Arc Segment screen: Rectangular Coordinates](image)

- **DIRECTION**—defines the direction, clockwise or counterclockwise, of the arc from the start point.
- **X END**—defines the X coordinate on an XY plane with origin at the center of the face.
- **Y END**—defines the Y coordinate on an XY plane with origin at the center of the face.
- **X CENTER**—defines the X center coordinate on an XY plane with origin at the center of the face.
- **Y CENTER**—defines the Y center coordinate on an XY plane with origin at the center of the face.
- **X START**—contains the X start point for the segment. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **Y START**—contains the Y start point for the segment. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **RADIUS**—defines the radius of the arc.
- **FEED**—contains the feed rate from the previous segment. This value can be accepted or changed.
- **LINEAR Y MOTION**—select this checkbox to cause Y-Axis Motion using the X and X’ axes. When this checkbox is clear, Y-Axis Motion occurs using the X and C axes. Please refer to *Programming Coordinates and Linear Y Motion, on page 2 - 68* for examples. This field is available with TMX MY and TMX MYS series machines.

![Diagram of Coordinate System](image)

Linear Y-Axis Motion is limited. The amount of Y travel is affected by the X position and by any X-Axis tool length offset.

Examples of these coordinates for a clockwise arc are shown below:

The Calc-Assist feature calculates certain unknown arc segment dimensions automatically, after sufficient data has been entered. A parameter with a calculated value is indicated with “CAL” preceding the value.

- The values of center points plus start point or end point coordinates s are used to calculate the arc radius.
- Start point and end point coordinates plus the radius are used to calculate two possible center point coordinates.
- Either of the end point values and the center point coordinate supplies the value of the other end point and the radius parameter.
- A known center point, start point, or end point and radius are used to calculate an unknown center point coordinate.

A series of arcs and lines can be programmed in a single data block to form a complete contour. Press the Ctrl + arrow key to program additional Line and Arc segments for the current data block.
**Blend Arc**

From the New Axial Contour Segment screen, select Blend Arc F3, and the Axial Mill Contour Blend Arc Segment screen appears with fields based on your choice in the Coordinates field (Polar or Rectangular).

When Polar is selected in the COORDINATES field, these fields appear:

![Figure 2–120. Live-Tooling Axial Mill Contour Blend Arc Segment screen: Polar Coordinates](image)

- **RADIUS**—defines the radius of the blend arc.
- **FEED**—contains the feed rate from the previous segment. This value can be accepted or changed.
- **DIRECTION**—defines the direction, clockwise or counterclockwise, of the arc from the start point. This read-only field is carried forward from a previous arc segment and can only be edited in that screen.
- **R START**—contains the starting distance from the centerline. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **C START (DEG)**—contains the angle of the arc segment start point, measured counterclockwise from the home position. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **R END**—contains the starting distance from the centerline. This read-only field is carried forward from the previous segment and can only be edited in that screen.
- **C END (DEG)**—defines the angle of the blend arc segment end point, measured counterclockwise from the home position. This read-only field is carried forward from the previous segment and can only be edited in that screen.
• **R CENTER**—contains the starting distance from the centerline. This read-only field is carried forward from the previous segment and can only be edited in that screen.

• **C CENTER (DEG)**—defines the angle of the blend arc segment center point, measured counterclockwise from the home position. This read-only field is carried forward from the previous segment and can only be edited in that screen.

When Rectangular is selected in the COORDINATES field, these fields appear:

![Image of Live-Tooling Axial Mill Contour Blend Arc Segment screen: Rectangular Coordinates](image)

• **RADIUS**—defines the radius of the blend arc.

• **FEED**—contains the feed rate from the previous segment. This value can be accepted or changed.

• **DIRECTION**—defines the direction, clockwise or counterclockwise, of the blend arc from the start point. This read-only field is carried forward from a previous arc segment and can only be edited in that screen.

• **X START**—contains the X start point for the segment. This read-only field is carried forward from the previous segment and can only be edited in that screen.

• **Y START**—contains the Y start point for the segment. The Y axis will be wrapped on the cylinder diameter. This read-only field is carried forward from the previous segment and can only be edited in that screen.

• **X END**—defines the X coordinate on an XY plane with origin at the center of the face. This read-only field is carried forward from the previous segment and can only be edited in that screen.

• **Y END**—defines the Y coordinate on an XY plane with origin at the center of the face. This read-only field is carried forward from the previous segment and can only be edited in that screen.
• **X CENTER**—defines the X center coordinate on an XY plane with origin at the center of the face. This read-only field is carried forward from the previous segment and can only be edited in that screen.

• **Y CENTER**—defines the Y center coordinate on an XY plane with origin at the center of the face. This read-only field is carried forward from the previous segment and can only be edited in that screen.

• **LINEAR Y MOTION**—select this checkbox to cause Y-Axis Motion using the X and X’ axes. When this checkbox is clear, Y-Axis Motion occurs using the X and C axes. Please refer to *Programming Coordinates and Linear Y Motion, on page 2-68* for examples. This field is available with TMX MY and TMX MYS series machines.

Linear Y-Axis Motion is limited. The amount of Y travel is affected by the X position and by any X-Axis tool length offset.

A blend arc joins two other segments and is tangent to both. A blend arc can be used to join these types of segments:

- two Line segments.
- a Line segment and an Arc segment.
- 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 diagram below illustrates some examples of blend arcs.

![Blend Arc Examples Diagram](image)

**Figure 2–122. Blend Arc Examples**

- **Two Lines joined by a Blend Arc**
  - 1 X/Y Start
  - 2 X/Y End
  - 3 Segment 1 (Line)
  - 4 Segment 1 End/Segment 3 Start (Point of Intersection)
  - 5 Segment 2 (Blend Arc)
  - 6 Segment 3 (Line)

- **Line and Arc Joined by a Blend Arc**
  - 1 X/Y Start
  - 2 X/Y End
  - 3 Segment 1 (Line)
  - 4 Segment 1 End/Segment 3 Start (Point of Intersection)
  - 5 Segment 2 (Blend Arc)
  - 6 Segment 3 (Arc)

- **Two Arcs Joined by a Blend Arc**
  - 1 X/Y Start
  - 2 X/Y End
  - 3 Segment 1 (Arc)
  - 4 Segment 1 End/Segment 3 Start (Point of Intersection)
  - 5 Segment 2 (Blend Arc)
  - 6 Segment 3 (Arc)
Some guidelines that must be followed when creating a blend arc follow:

- The first or last segment of a Radial Mill Contour data block cannot be blend arc segments.
- Blend arc segments cannot be adjacent to one another in a program cannot be blend arc segments. For example, if segment #2 is a blend arc, neither segment #1 nor #3 can be blend arc segments.
- Segments that are adjacent to the blend arc segment must intersect at some point in their theoretical plane. Therefore, if segment #2 is a blend arc, segments #1 and #3 must theoretically intersect at some projected point.
- The Radius of a blend arc segment cannot be too large to be tangent to both of the adjoining segments.
- If any coordinate (start point, center point, or end point) is important to the construction of the two segments to be blended, the segment must be programmed as an arc and not as a blend arc.
- The Feed field is initially displayed with a value carried forward from the previous segment. This value can be accepted or changed.

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

Develop a part program segment by segment, or create the geometry surrounding an arc, then insert the blend arc between existing segments.
Axial Circle

From the New Block (Axial Milling) screen select AXIAL CIRCLE F2, and the Axial Mill Circle screen appears with Process and Geometry tabs. Program the tool and cutting information using the Process tab. Describe the finish shape using the Geometry tab.

![Figure 2–123. Live-Tooling Axial Mill Circle screen Process tab](image)

**Process Tab**

The program determines how many rough passes to make to remove the material based on the entries in the Process tab fields. The Process tab fields are defined as follows:

- **TOOL**—identifies the tool number that will be used in the part program.
- **TOOL OFFSETS**—identifies the tool offset and orientation programmed in Tool Setup that will be used. The tool offset defaults to the same number as the tool.
- **TOOL DIAMETER**—contains the tool diameter programmed in Tool Setup for the current tool. This field is read-only and can only be edited in Tool Setup.
- **STRATEGY**—identifies the tool path with respect to rough or finish.
  - **ROUGH AND FINISH**—perform rough and finish passes.
  - **ROUGH ONLY**—perform a rough pass only.
  - **FINISH ONLY**—perform a finish pass only.
- **MILLING TYPE**—identifies on which side of the contour the tool should begin cutting. Refer to *Cutter Compensation, on page 2 - 188* for more details. Select from among these choices:
  - **ON**—used for cutting directly on the programmed path.
  - **INSIDE**—causes the tool to enter the part inside the contour of the circle and blend into it using a 180° arc. Cutter compensation is automatically employed and the outside edge of the tool follows the contour (the direction the tool travels depends upon milling direction). The point where the tool enters the part is determined by the blend offset and the point where the tool is withdrawn is determined by the blend overlap and blend offset.
  - **OUTSIDE**—similar to the Inside function, except the tool enters the part outside the circle and follows the outside of the contour.
  - **IN/TAN (INSIDE TANGENT)**—causes the tool to enter the part adjacent to the inside contour of the circle. Cutter compensation is automatically used and the edge of the tool follows the inside of the contour. The direction the tool travels depends upon the milling direction. The tool is withdrawn from the part adjacent to the contour.
  - **OUT/TAN (OUTSIDE TANGENT)**—similar to the Inside Tangent function except that the tool enters the part adjacent to the outside contour of the circle. The tool’s edge then follows the outside of the contour. The tool is withdrawn from the part adjacent to the contour.
  - **POCK BOUNDARY (POCKET BOUNDARY)**—used for cutting the defined closed contour.
- **SPEED (RPM)**—identifies the speed at which the spindle rotates to cut the part. This value is carried forward for the selected tool from Tool Setup. “CAL” appears next to the field. To accept this value, select Enter. A different speed can be programmed for this specific block.

  ⇒ If a Speed value is changed in a milling data block, a pop-up message appears asking if you want to change the corresponding Feed value in the data block.

  Override the displayed value by typing in a new speed or by using the console Spindle override knob.

- **MILL FEED**—identifies the cutting feedrate. This value is carried forward for the selected tool from Tool Setup. “CAL” appears next to the field. To accept this value, select Enter. A different mill feed rate can be programmed for this specific block.

  ⇒ If a Feed value is changed in a milling data block, a pop-up message appears asking if you want to change the corresponding Speed value in the data block.

- **PLUNGE FEED**—identifies the feedrate between Z Start and Z Bottom.
These fields appear based on your selection in the previous fields:

- **POCKET TYPE**—identifies the pocketing type as Inward or Outward. This field is available when Pocket Boundary is selected for the Milling Type.
  - **INWARD**—cuts in from the edge of the defined boundary.
  - **OUTWARD**—cuts from the center of the feature outward to the defined boundary.

- **POCKET OVERLAP (%)**—controls the percentage of overlap during cutting. This field is available when Pocket Boundary is selected for the Milling Type.

- **WALL ALLOWANCE**—identifies the amount of material to leave along the wall of the stock for cutting the final pass. The wall of the stock is the side of the stock that is being cut.
  
  The WALL ALLOWANCE field is **not** available when
  
  - Rough and Finish is selected for the Strategy and On is selected for the Milling Type.
  - Finish Only is selected for the Strategy.

- **FLOOR ALLOWANCE**—identifies the amount of material to leave on the bottom of the stock for cutting the final pass. This field is available when Pocket Boundary is selected for the Milling Type. This field is not available when Finish Only is selected for the Strategy.

- **BLEND OFFSET**—identifies the XY distance from the entry point of the part surface and the Z plunge point where the tool enters the work piece. This move is always a 90° arc and occurs at the 3:00 position on the circle. This parameter is used for the blend-in move in milling and for the blend-out move from the part surface to the Z Retract point. The default value is 3.000 mm (0.1250 inches). This field is available when either Inside, Outside, or Pocket Boundary is selected for the Milling Type.

- **BLEND OVERLAP**—identifies the distance the tool travels past the entry point before exiting from the part. The default value is 3.000 mm (0.1250 inches). This field is available when either Inside, Outside, or Pocket Boundary is selected for the Milling Type.

- **DIRECTION**—identifies the milling direction as either Conventional or Climb. This field is not available when On is selected for the Milling Type. Refer to *Cutter Compensation, on page 2-188* for details about milling direction.

- **PECK DEPTH**—identifies the maximum depth to be cut in one pass. This field is not available when Finish Only is selected for the Strategy.

- **1ST PECK OFFSET**—identifies the depth of the first peck when it needs to be different than subsequent pecks, identified in Peck Depth. This field is not available when Finish Only is selected for the Strategy.

- **PECK CLEARANCE**—identifies the distance above the previous peck level to which the Z axis returns at rapid traverse. This field is not available when Finish Only is selected for the Strategy.
**Calculated Plunge Points for Axial Circles**

Calculations for determining the plunge points for milling cycles are for TMM, TMX MY, and TMX MYS Series machines. The following examples are for Axial Circles and Climb Milling.

The Blend-in move is a 90° arc and occurs at the 3:00 position on the circle.

**Pocket Type Inward**

\[
\text{Xplunge} = \text{Xcenter} + \text{Circle radius} - \text{Blend Offset} - \text{Tool radius}.
\]

\[
\text{Yplunge} = \text{Ycenter} - \text{Blend Offset}
\]

**Pocket Type Outward**

\[
\text{Xplunge} = \text{Xcenter} + \text{Circle radius} + \text{Blend Offset} + \text{Tool radius}
\]

\[
\text{Yplunge} = \text{Ycenter} + \text{Blend Offset}
\]

**Roughing**

To determine the plunge points for roughing passes, subtract the Wall Allowance from the Xplunge.
Geometry Tab

Finishing follows the geometry of the contour defined in the Geometry tab. The fields on the Geometry tab are defined as follows:

- **COORDINATES**—defines the coordinates as either Rectangular or Polar.
  - **Rectangular** coordinates are used when working from a flat drawing
  - **Polar** coordinates move around the cylinder as specified.

The remaining fields are based on your selection in the Coordinates field (Rotary or Linear).

When Polar is selected in the COORDINATES field, these fields appear:

- **R CENTER**—defines the starting distance from the centerline.
- **C CENTER (DEG)**—defines the angle of the circle from the start point to the end point, measured counterclockwise from the home position.
- **RADIUS**—defines the radius of the circle.
- **Z START**—defines the location where Plunge Feedrate begins. The feedrate is set in the Plunge Feed field in the Process tab.
- **Z BOTTOM**—defines the bottom of the hole and the location where the Plunge Feed Rate ends.
When Rectangular is selected in the COORDINATES field, these fields appear:

- **X CENTER**—defines the X center coordinate on an XY plane with origin at the center of the face.
- **Y CENTER**—defines the Y center coordinate on an XY plane with origin at the center of the face.
- **RADIUS** or **DIAMETER**—defines the radius or diameter of the circle.
- **C POSITION**—defines the angle to position the C axis for the first segment of the contour. This field appears when the Linear Y Motion checkbox is selected. This field is available with TMX MY and TMX MYS series machines.
- **LINEAR Y MOTION**—select this checkbox to cause Y-Axis Motion using the X and X’ axes. When this checkbox is clear, Y-Axis Motion occurs using the X and C axes. Please refer to *Programming Coordinates and Linear Y Motion*, on page 2 - 68 for examples. This field is available with TMX MY and TMX MYS series machines.

⚠️ Linear Y-Axis Motion is limited. The amount of Y travel is affected by the X position and by any X-Axis tool length offset.

- **Z START**—defines the location where Plunge Feedrate begins. The feedrate is set in the Plunge Feed field in the Process tab.
- **Z BOTTOM**—defines the bottom of the hole and the location where the Plunge Feedrate ends.
These fields are read-only and are carried over from Tool Setup for the tool specified in the Process tab.

- **TOOL**—identifies the tool number that will be used in the part program.
- **TOOL OFFSETS**—identifies the tool offset and orientation programmed in Tool Setup that will be used. The tool offset defaults to the same number as the tool.
- **TOOL DIAMETER**—contains the tool diameter programmed in Tool Setup for the current tool. This field is read-only and can only be edited in Tool Setup.

**Axial Frame**


**Process Tab**

The program determines how many rough passes to make to remove the material based on the entries in the Process tab fields.

![Figure 2–126. Live-Tooling Axial Mill Frame screen Process tab](image)

The Process tab fields are defined as follows:

- **TOOL**—identifies the tool number that will be used in the part program.
- **TOOL OFFSETS**—identifies the tool offset and orientation programmed in Tool Setup that will be used. The tool offset defaults to the same number as the tool.
- **TOOL DIAMETER**—contains the tool diameter programmed in Tool Setup for the current tool. This field is read-only and can only be edited in Tool Setup.
• **STRATEGY**—identifies the tool path with respect to rough or finish.
  • **ROUGH AND FINISH**—perform rough and finish passes.
  • **ROUGH ONLY**—perform a rough pass only.
  • **FINISH ONLY**—perform a finish pass only.

• **MILLING TYPE**—identifies on which side of the contour the tool should begin cutting. Refer to *Cutter Compensation, on page 2 - 188* for more details.
  Select from among these choices:
  • **ON**—used for cutting directly on the programmed path.
  • **INSIDE**—causes the tool to enter the part inside the contour of the frame and blend into it using a 180° arc. Cutter compensation is automatically employed and the outside edge of the tool follows the contour (the direction the tool travels depends upon milling direction). The point where the tool enters the part is determined by the blend offset and the point where the tool is withdrawn is determined by the blend overlap and blend offset. The following is an example of an Inside frame:
    
    ![Figure 2–127. Inside Mill Frame Operation](image)

    **Reference Corner**  **Cutter Entry Position**

• **OUTSIDE**—similar to the Inside function, except the tool enters the part outside the frame and follows the outside of the contour. The tool is withdrawn adjacent to the frame.

• **IN/TAN (INSIDE TANGENT)**—causes the tool to enter the part adjacent to the inside frame contour. Cutter compensation is automatically used and the edge of the tool follows the inside of the contour. The direction the tool travels depends upon the milling direction. The tool is withdrawn from the part adjacent to the frame.

• **OUT/TAN (OUTSIDE TANGENT)**—similar to the Inside Tangent function except that the tool enters the part adjacent to the outside frame contour. The tool’s edge then follows the outside of the contour. The tool is withdrawn from the part adjacent to the frame.

• **POCK BOUNDARY (POCKET BOUNDARY)**—used for cutting the outside perimeter of a pocket operation.
• **SPEED (RPM)**—identifies the speed at which the spindle rotates to cut the part. This value is carried forward for the selected tool from Tool Setup. “CAL” appears next to the field. To accept this value, select Enter. A different speed can be programmed for this specific block.

  ![If a Speed value is changed in a milling data block, a pop-up message appears asking if you want to change the corresponding Feed value in the data block.](image)

Override the displayed value by typing in a new speed or by using the console Spindle override knob.

• **MILL FEED**—identifies the cutting feedrate. This value is carried forward for the selected tool from Tool Setup. “CAL” appears next to the field. To accept this value, select Enter. A different mill feed rate can be programmed for this specific block.

  ![If a Feed value is changed in a milling data block, a pop-up message appears asking if you want to change the corresponding Speed value in the data block.](image)

• **PLUNGE FEED**—identifies the feedrate between Z Start and Z Bottom.

These fields appear based on your selection in the previous fields:

• **POCKET TYPE**—identifies the pocketing type as Inward or Outward. This field is available when Pocket Boundary is selected for the Milling Type.
  - **INWARD**—cuts in from the edge of the defined boundary in.
  - **OUTWARD**—cuts from the center of the feature outward to the defined boundary.

• **POCKET OVERLAP (%)**—controls the percentage of overlap during cutting. This field is available when Pocket Boundary is selected for the Milling Type.

• **WALL ALLOWANCE**—identifies the amount of material to leave along the wall of the stock for cutting the final pass. The wall of the stock is the side of the stock that is being cut. This field is **not** available when
  - Rough and Finish is selected for the Strategy and On is selected for the Milling Type.
  - Finish Only is selected for the Strategy.

• **FLOOR ALLOWANCE**—identifies the amount of material to leave on the bottom of the stock for cutting the final pass. This field is available when Pocket Boundary is selected for the Milling Type. This field is **not** available when Finish Only is selected for the Strategy.
- **BLEND OFFSET**—identifies the XY distance from the entry point of the part surface and the Z plunge point where the tool enters the work piece. This move is always a 90° arc and occurs at the 6:00 position on the frame. This parameter is used for the blend-in move in milling and for the blend-out move from the part surface to the Z Retract point. The default value is 3.000 mm (0.1250 inches). This field is available when either Inside, Outside, or Pocket Boundary is selected for the Milling Type.

- **BLEND OVERLAP**—identifies the distance the tool travels past the entry point before exiting from the part. The default value is 3.000 mm (0.1250 inches). This field is available when either Inside, Outside, or Pocket Boundary is selected for the Milling Type.

- **DIRECTION**—identifies the milling direction as either Conventional or Climb. This field is not available when On is selected for the Milling Type. Refer to *Cutter Compensation, on page 2 - 188* for details about milling direction.

- **PECK DEPTH**—identifies the maximum depth to be cut in one pass. This field is not available when Finish Only is selected for the Strategy.

- **1ST PECK OFFSET**—identifies the depth of the first peck when it needs to be different than subsequent pecks, identified in Peck Depth. This field is not available when Finish Only is selected for the Strategy.

- **PECK CLEARANCE**—identifies the distance above the previous peck level to which the Z axis returns at rapid traverse. This field is not available when Finish Only is selected for the Strategy.

- **CUTTER COMP**—identifies the type of milling to insert between segment end points. Choose between Arc or Lines. This field is not available when On is selected for the Milling Type. If Arc is selected, a tangent arc is inserted to connect two line segments, or a line segment and an arc segment (when the two cutter compensated segments are offset and do not intersect). When using the Arc method, the system creates the cutter compensated path as shown below:

```
1  Programmed tool path
2  Cutter compensated path
3  Arc parameter
```

*Figure 2–128. Cutter Compensation Using the Arc Parameter*

Some operators find the Arc method inappropriate for their applications because the cutter remains in contact with the work piece and may drag a chip across a sharp angle on the part. In such circumstances, operators select Lines.
• If the Lines softkey is selected, the cutter compensated lines and arcs are joined as described below:
  • Two line segments are extended until they intersect (provided they form a 90° or greater angle). If the lines form an angle of less than 90°, a line is inserted to connect them.
  • Line and arc segments have the line segment extended, and a tangent line to the arc segment inserted and extended until the lines intersect (provided they form a 90° or greater angle). If the segments form an angle of less than 90°, a line is inserted to connect them.
  • Two arc segments have tangent lines (to the arcs) inserted and extended until the lines intersect (provided the extended tangent lines form a 90° or greater angle). If the extended tangent lines form an angle of less than 90°, a line or arc is inserted to connect them.

\[ \text{Figure 2–129. Cutter Compensation Using the Line Parameter} \]

**Calculated Plunge Points for Axial Frames**

Calculations for determining the plunge points for milling cycles are for TMM, TMX MY, and TMX MYS Series machines. The following examples are for Axial Frames and Climb Milling.

The Blend-in move is a 90° arc and occurs at the 6:00 position on the frame.

**Pocket Type Inward**

Xplunge = Xcorner + (XLength/2) - Blend Offset

Yplunge = Ycorner + Blend Offset + Tool radius

**Pocket Type Outward**

Xplunge = Xcorner + (XLength/2) + Blend Offset

Yplunge = Ycorner - Blend Offset - Tool radius

**Roughing**

To determine the plunge points for roughing passes, subtract the Wall Allowance from the Yplunge.
Geometry Tab

Finishing follows the geometry of the contour defined in the Geometry tab.

Finishing follows the geometry of the contour defined in the Geometry tab. The fields on the Geometry tab are defined as follows:

- **COORDINATES**—defines the coordinates as either Rectangular or Polar.
  - **Rectangular** coordinates are used when working from a flat drawing.
  - **Polar** coordinates move around the cylinder as specified.

The remaining fields are based on your selection in the Coordinates field (Polar or Rectangular).

When Polar is selected in the COORDINATES field, these fields appear:

![Geometry Tab](image)

*Figure 2–130. Live-Tooling Axial Mill Frame screen Geometry tab: Polar Coordinates*

- **R CORNER**—defines the starting distance from the centerline.
- **C CORNER (DEG)**—defines the angle of the circle from the start point to the end point, measured counterclockwise from the home position.
- **R LENGTH**—defines the starting distance from the centerline.
- **C LENGTH (DEG)**—defines the length of the frame, in degrees, along the C axis, measured counterclockwise from the home position.
- **CORNER RADIUS**—defines the radius of all four corners of the frame, which becomes the reference corner.
- **Z START**—defines the location where Plunge Feedrate begins. The feedrate is set in the Plunge Feed field in the Process tab.
- **Z BOTTOM**—defines the bottom of the hole and the location where the Plunge Feed Rate ends.
When Rectangular is selected in the COORDINATES field, these fields appear:

- **X CORNER**—defines the X corner coordinate on an XY plane with origin at the center of the face.
- **Y CORNER**—defines the Y corner coordinate on an XY plane with origin at the center of the face.
- **X LENGTH**—defines the X length coordinate on an XY plane with origin at the center of the face.
- **Y LENGTH**—defines the Y length coordinate on an XY plane with origin at the center of the face.
- **CORNER RADIUS**—defines the radius of all four corners of the frame.
- **C POSITION**—defines the angle to position the C axis for the first segment of the contour. This field appears when the Linear Y Motion checkbox is selected. This field is available with TMX MY and TMX MYS series machines.
- **LINEAR Y MOTION**—select this checkbox to cause Y-Axis Motion using the X and X’ axes. When this checkbox is clear, Y-Axis Motion occurs using the X and C axes. Please refer to *Programming Coordinates and Linear Y Motion*, on page 2 - 68 for examples. This field is available with TMX MY and TMX MYS series machines.

⚠️ Linear Y-Axis Motion is limited. The amount of Y travel is affected by the X position and by any X-Axis tool length offset.

- **Z START**—defines the location where Plunge Feedrate begins. The feedrate is set in the Plunge Feed field in the Process tab.
- **Z BOTTOM**—defines the bottom of the hole and the location where the Plunge Feedrate ends.

*Figure 2–131. Live-Tooling Axial Mill Frame screen Geometry tab: Rectangular Coordinates*
These fields are read-only and are carried over from Tool Setup for the tool specified in the Process tab.

- **TOOL**—identifies the tool number that will be used in the part program.
- **TOOL OFFSETS**—identifies the tool offset and orientation programmed in Tool Setup that will be used. The tool offset defaults to the same number as the tool.
- **TOOL DIAMETER**—contains the tool diameter programmed in Tool Setup for the current tool. This field is read-only and can only be edited in Tool Setup.

**Axial Slot**

From the New Block (Axial Milling) screen select Axial Slot F4, and the Axial Slot screen appears with Process and Geometry tabs. Program the tool and cutting information using the Process tab. Describe the finish shape using the Geometry tab.

**Process Tab**

The program determines how many rough passes to make to remove the material based on the entries in the Process tab fields.

![Figure 2–132. Live-Tooling Axial Mill Axial Slot screen Process tab](image)

The Process tab fields are defined as follows:

- **TOOL**—identifies the tool number that will be used in the part program.
- **TOOL OFFSETS**—identifies the tool offset and orientation programmed in Tool Setup that will be used. The tool offset defaults to the same number as the tool.
- **TOOL DIAMETER**—contains the tool diameter programmed in Tool Setup for the current tool. This field is read-only and can only be edited in Tool Setup.
• **STRATEGY**—identifies the tool path with respect to rough or finish.
  • **ROUGH AND FINISH**—perform rough and finish passes.
  • **ROUGH ONLY**—perform a rough pass only.
  • **FINISH ONLY**—perform a finish pass only.
• **STEP OVER %**—identifies the distance (a percentage of the tool diameter) to overlap between passes. The default is 50%.
• **WALL ALLOWANCE**—identifies the amount of material to leave along the wall of the stock for cutting the final pass. The wall of the stock is the side of the stock that is being cut. This field is available when either Rough and Finish or Rough Only is selected for the Strategy.
• **FLOOR ALLOWANCE**—identifies the amount of material to leave on the bottom of the stock for cutting the final pass. This field is not available when Finish Only is selected for the Strategy.
• **SPEED (RPM)**—identifies the speed at which the spindle rotates to cut the part. This value is carried forward for the selected tool from Tool Setup. “CAL” appears next to the field. To accept this value, select Enter. A different speed can be programmed for this specific block.

⇒ If a Speed value is changed in a milling data block, a pop-up message appears asking if you want to change the corresponding Feed value in the data block.

Override the displayed value by typing in a new speed or by using the console Spindle override knob.

• **DIRECTION**—identifies the milling direction as either Conventional or Climb. This field is not available when On is selected for the Milling Type. Refer to *Cutter Compensation, on page 2-188* for details about milling direction.
• **MILL FEED**—identifies the cutting feedrate. This value is carried forward for the selected tool from Tool Setup. “CAL” appears next to the field. To accept this value, select Enter. A different mill feed rate can be programmed for this specific block.

⇒ If a Feed value is changed in a milling data block, a pop-up message appears asking if you want to change the corresponding Speed value in the data block.

• **PLUNGE TYPE**—identifies the type of plunge motion:
  • **STRAIGHT**
  • **ZIG-ZAG**
  • **RAPID**
• **PLUNGE FEED**—identifies the feedrate between Z Start and Z Bottom.
• **RAMP ANGLE**—identifies the ramp angle for Zig-Zag plunge motion. This field is available when Zig-Zag is selected for the Plunge Type.
• **PECK DEPTH**—identifies the maximum depth to be cut in one pass.
• **SURFACE CLEARANCE**—sets the distance above the Z (Axial) or X (Radial) start point for the tool to move between passes to allow for rapid motions back to the start point for the next pass.
This setting also applies when rapid motions are made to position for cutting depths after the first depth.

Example:

Surface Clearance = 1.27 mm (0.0500 in)
Z Start = 2.54 mm (0.1000 in)
Peck Depth = 5.08 mm (0.2000 in)

In this example, the Z motion would be:

1. Rapid to 3.81 mm (0.15 in) (Z Start + Surface Clearance)
2. Feed to -2.54 mm (-0.1 in) to machine first pass.
3. Rapid back to 3.81 mm (0.15 in) (Z Start + Surface Clearance)
4. Rapid to -1.27 mm (-0.05 in) (Depth of Previous pass + Surface Clearance)
5. Feed to -7.62 mm (-0.30 in) to machine second pass.
6. Rapid back to 3.81 mm (0.15 in) (Z Start + Surface Clearance)
7. Rapid to -6.35 mm (-0.25 in) (Depth of Previous pass + Surface Clearance)
8. Feed to -12.7 mm (-0.5 in) to machine third pass.
9. Rapid back to 3.81 mm (0.15 in) (Z Start + Surface Clearance)
10. Repeat to final depth.

Geometry Tab

Finishing follows the geometry of the contour defined in the Geometry tab.

Finishing follows the geometry of the contour defined in the Geometry tab. The fields on the Geometry tab are defined as follows:

- **COORDINATES**—defines the coordinates as either Rectangular or Polar.
  - Rectangular coordinates are used when working from a flat drawing.
  - Polar coordinates move around the cylinder as specified.
The remaining fields are based on your selection in the Coordinates field (Polar or Rectangular):

When Polar is selected in the COORDINATES field, these fields appear:

- **R START**—defines the starting distance from the centerline.
- **C START (DEG)**—defines the angle of the start point to the end point, measured counterclockwise from the home position.
- **R END**—defines the starting distance from the centerline.
- **C END (DEG)**—defines the length of the slot, in degrees, along the C axis, measured counterclockwise from the home position.
- **SLOT WIDTH**—defines the width of the slot.

*Figure 2–133. Live-Tooling Axial Mill Axial Slot screen Geometry tab: Polar Coordinates*
• **START CAP**—accesses selections for the shape of the slot end for the Start of the slot:
  - **LINE**—produces a straight edge for the start cap with a corner radius the same size as the tool.

  ![Figure 2–134](image)

  *Figure 2–134. End Cap = Line (R = 0.125) Start Cap = Line (R = 0)*

• **APPEND ARC**—produces an arc at the start cap with the arc being appended or added to the end of the slot.

  ![Figure 2–135](image)

  *Figure 2–135. End Cap = Append Arc; Start Cap = Line (R = 0)*

• **INCLUDE ARC**—produces an arc at the start cap with the arc included within the geometry.

  ![Figure 2–136](image)

  *Figure 2–136. End Cap = Include Arc; Start Cap = Line (R = 0)*

• **CORNER RADIUS**—defines the radius of the slot end.

• **END CAP**—accesses selections for the shape of the slot end for the End of the slot:
  - **LINE**
  - **APPEND ARC**
  - **INCLUDE ARC**

→ Refer to Start Cap examples above.
• **CORNER RADIUS**—defines the radius of the slot end.

• **Z START**—defines the location where Plunge Feedrate begins. The feedrate is set in the Plunge Feed field in the Process tab.

• **Z BOTTOM (DIA)**—accesses selections for the bottom of the slot and the location where the Plunge Feed Rate ends.
  - **AT START**
  - **AT END**

When Rectangular is selected in the COORDINATES field, these fields appear:

![Figure 2–137. Live-Tooling Axial Mill Axial Slot screen Geometry tab: Rectangular Coordinates](image)

- **X START**—defines the X start coordinate on an XY plane.
- **Y START**—defines the Y start coordinate on an XY plane.
- **X END**—defines the X end coordinate on an XY plane.
- **Y END**—defines the Y end coordinate on an XY plane.
- **C POSITION**—defines the angle to position the C axis for the slot. This field appears when the Linear Y Motion checkbox is selected. This field is available with TMX MY and TMX MYS series machines.
- **LINEAR Y MOTION**—select this checkbox to cause Y-Axis Motion using the X and X’ axes. When this checkbox is clear, Y-Axis Motion occurs using the X and C axes. Please refer to Programming Coordinates and Linear Y Motion, on page 2 - 68 for examples. This field is available with TMX MY and TMX MYS series machines.

⚠️ Linear Y-Axis Motion is limited. The amount of Y travel is affected by the X position and by any X-Axis tool length offset.
• **SLOT WIDTH**—defines the width of the slot

• **START CAP**—accesses selections for the shape of the slot end for the Start of the slot:
  • **LINE**—produces a straight edge for the start cap with a corner radius the same size as the tool.

![Figure 2–138. End Cap = Line (R = 0.125) Start Cap = Line (R = 0)](image)

• **APPEND ARC**—produces an arc at the start cap with the arc being appended or added to the end of the slot.

![Figure 2–139. End Cap = Append Arc; Start Cap = Line (R = 0)](image)

• **INCLUDE ARC**—produces an arc at the start cap with the arc included within the geometry.

![Figure 2–140. End Cap = Include Arc; Start Cap = Line (R = 0)](image)

• **CORNER RADIUS**—defines the radius of the slot end.

• **END CAP**—accesses selections for the shape of the slot end for the End of the slot:
  • **LINE**
  • **APPEND ARC**
  • **INCLUDE ARC**

  Refer to Start Cap examples above.

• **CORNER RADIUS**—defines the radius of the slot end.
• **Z START**—defines the location where Plunge Feedrate begins. The feedrate is set in the Plunge Feed field in the Process tab.

• **Z BOTTOM**—accesses selections for the bottom of the slot and the location where the Plunge Feed Rate ends.
  - **AT START (DIA) or (RAD)**
  - **AT END (DIA) or (RAD)**

These fields are read-only and are carried over from Tool Setup for the tool specified in the Process tab.

• **TOOL**—identifies the tool number that will be used in the part program.

• **TOOL OFFSETS**—identifies the tool offset and orientation programmed in Tool Setup that will be used. The tool offset defaults to the same number as the tool.

• **TOOL DIAMETER**—contains the tool diameter programmed in Tool Setup for the current tool. This field is read-only and can only be edited in Tool Setup.
Axial Flats

From the New Block (Axial Milling) screen select Axial Flats F5, and the Axial Flats screen appears with Process and Geometry tabs. Program the tool and cutting information using the Process tab. Describe the finish shape using the Geometry tab.

**Process Tab**

The program determines how many rough passes to make to remove the material based on the entries in the Process tab fields.

![Image of Axial Flats screen with Process tab highlighted](image.png)

*Figure 2–141. Live-Tooling Axial Mill Axial Flats screen Process tab*

The Process tab fields are defined as follows:

- **TOOL**—identifies the tool number that will be used in the part program.
- **TOOL OFFSETS**—identifies the tool offset and orientation programmed in Tool Setup that will be used. The tool offset defaults to the same number as the tool.
- **TOOL DIAMETER**—contains the tool diameter programmed in Tool Setup for the current tool. This field is read-only and can only be edited in Tool Setup.
- **STRATEGY**—identifies the tool path with respect to rough or finish.
  - **ROUGH AND FINISH**—perform rough and finish passes.
  - **ROUGH ONLY**—perform a rough pass only.
  - **FINISH ONLY**—perform a finish pass only.
  - **FINISH (1 PASS)**—perform 1 finish pass as follows:
    1. Plunge to the Finish Z depth.
    2. Step over and cut the final profile, disregarding any area that stock was removed in prior data blocks.
    3. Cut at finish depth the final flats surfaces (X and Y) and the diameter of the cutter in Z.
• **STEP OVER**—identifies the distance (a percentage of the tool diameter) to overlap between passes.

• **WALL ALLOWANCE**—identifies the amount of material to leave along the wall of the stock for cutting the final pass. The wall of the stock is the side of the stock that is being cut. This field is not available when Finish Only is selected for the Strategy.

• **FLOOR ALLOWANCE**—identifies the amount of material to leave on the bottom of the stock for cutting the final pass. This field is not available when Finish Only is selected for the Strategy.

• **SPEED (RPM)**—identifies the speed at which the spindle rotates to cut the part. This value is carried forward for the selected tool from Tool Setup. “CAL” appears next to the field. To accept this value, select Enter. A different speed can be programmed for this specific block.

  ⇒ If a Speed value is changed in a milling data block, a pop-up message appears asking if you want to change the corresponding Feed value in the data block.

Override the displayed value by typing in a new speed or by using the console Spindle override knob.

• **DIRECTION**—identifies the milling direction as either Conventional or Climb. Refer to *Cutter Compensation, on page 2-188* for details about milling direction.

• **MILL FEED**—identifies the cutting feedrate. This value is carried forward for the selected tool from Tool Setup. “CAL” appears next to the field. To accept this value, select Enter. A different mill feed rate can be programmed for this specific block.

  ⇒ If a Feed value is changed in a milling data block, a pop-up message appears asking if you want to change the corresponding Speed value in the data block.

• **PLUNGE TYPE**—identifies the type of plunge motion:
  • **STRAIGHT**
  • **ZIG-ZAG**
  • **RAPID**

• **PLUNGE FEED**—identifies the feedrate between Z Start and Z Bottom.

• **RAMP ANGLE**—identifies the ramp angle for Zig-Zag plunge motion. This field is available when Zig-Zag is selected for the Plunge Type.

• **PECK DEPTH**—identifies the maximum depth to be cut in one pass. This field is not available when Finish Only is selected for the Strategy.
**SURFACE CLEARANCE**—sets the distance above the Z (Axial) or X (Radial) start point for the tool to move between passes to allow for rapid motions back to the start point for the next pass.

This setting also applies when rapid motions are made to position for cutting depths after the first depth.

Example:

Surface Clearance = 1.27 mm (0.0500 in)
Z Start = 2.54 mm (0.1000 in)
Peck Depth = 5.08 mm (0.2000 in)

In this example, the Z motion would be:

1. Rapid to 3.81 mm (0.15 in)  (Z Start + Surface Clearance)
2. Feed to -2.54 mm (-0.1 in) to machine first pass.
3. Rapid back to 3.81 mm (0.15 in)  (Z Start + Surface Clearance)
4. Rapid to -1.27 mm (-0.05 in)  (Depth of Previous pass + Surface Clearance)
5. Feed to -7.62 mm (-0.30 in) to machine second pass.
6. Rapid back to 3.81 mm (0.15 in)  (Z Start + Surface Clearance)
7. Rapid to -6.35 mm (-0.25 in)  (Depth of Previous pass + Surface Clearance)
8. Feed to -12.7 mm (-0.5 in) to machine third pass.
9. Rapid back to 3.81 mm (0.15 in)  (Z Start + Surface Clearance)
10. Repeat to final depth.
**Geometry Tab**

Finishing follows the geometry of the contour defined in the Geometry tab. The fields on the Geometry tab are defined as follows:

- **COORDINATES**—defines the coordinates as either Rectangular or Polar.
  - **Rectangular** coordinates are used when working from a flat drawing.
  - **Polar** coordinates move around the cylinder as specified.

The remaining fields appear based on your selection in the Coordinates field (Polar or Rectangular):

When Polar is selected in the COORDINATES field, these fields appear:

- **NUMBER OF SIDES**—defines the number of sides for the polygon.
- **SIZING METHOD**—identifies the sizing method:
  - **INSCRIBE**—defines the sizing method as inscribed in the sizing circle.
  - **ENCOMPASS**—defines the sizing method as encompassing the sizing circle.

*Figure 2–142. Live-Tooling Axial Mill Axial Flats screen Geometry tab: Polar Coordinates*

*Figure 2–143. Inscribe and Encompass*
- **SIZING DIAMETER**—defines the diameter of the circle, determining the size of the polygon.
- **STOCK SURFACE**—defines cutting the flats on either the Outside or the Inside of the stock.
- **R START**—defines the starting distance from the centerline.
- **C START (DEG)**—defines the angle in degrees to the center of the first flat edge, measured counterclockwise from the home position.
- **CORNER RADIUS**—defines the radius of all corners of the polygon.

![Inscribe and Encompass with Corner Radius](image)

**Figure 2–144. Inscribe and Encompass with Corner Radius**

- **Z START**—defines the location where Plunge Feedrate begins. The feedrate is set in the Plunge Feed field in the Process tab.
- **Z BOTTOM**—accesses selections for the bottom of the flat and the location where the Plunge Feed Rate ends.
When Rectangular is selected in the COORDINATES field, these fields appear:

- **NUMBER OF SIDES**—defines the number of sides for the polygon.
- **SIZING METHOD**—identifies the sizing method:
  - **INSCRIBE**—defines the sizing method as inscribed in the sizing circle.
  - **ENCOMPASS**—defines the sizing method as encompassing the sizing circle.

![Inscribe Encompass](image)

- **SIZING DIAMETER**—defines the diameter of the circle, determining the size of the polygon.
- **STOCK SURFACE**—defines cutting the flats on either the Outside or the Inside of the stock.
- **C POSITION**—defines the angle to position the C axis for the first segment of the contour. This field appears when the Linear Y Motion checkbox is selected. This field is available with TMX MY and TMX MYS series machines.
• **LINEAR Y MOTION**—select this checkbox to cause Y-Axis Motion using the X and X’ axes. When this checkbox is clear, Y-Axis Motion occurs using the X and C axes. Please refer to *Programming Coordinates and Linear Y Motion, on page 2 - 68* for examples. This field is available with TMX MY and TMX MYS series machines.

  Linear Y-Axis Motion is limited. The amount of Y travel is affected by the X position and by any X-Axis tool length offset.

• **X START**—defines the X start coordinate on an XY plane.

• **Y START**—defines the Y start coordinate on an XY plane.

• **CORNER RADIUS**—defines the radius of all corners of the polygon.

---

![Inscribe](image1.png) ![Encompass](image2.png)

*Figure 2–147. Inscribe and Encompass with Corner Radius*

• **Z START**—defines the location where Plunge Feedrate begins. The feedrate is set in the Plunge Feed field in the Process tab.

• **Z BOTTOM**—accesses selections for the bottom of the slot and the location where the Plunge Feed Rate ends.

These fields are read-only and are carried over from Tool Setup for the tool specified in the Process tab.

• **TOOL**—identifies the tool number that will be used in the part program.

• **TOOL OFFSETS**—identifies the tool offset and orientation programmed in Tool Setup that will be used. The tool offset defaults to the same number as the tool.

• **TOOL DIAMETER**—contains the tool diameter programmed in Tool Setup for the current tool. This field is read-only and can only be edited in Tool Setup.
Axial Lettering

Axial Lettering data blocks mill text into the face of the part.

From the New Block (Axial Milling) screen select Axial Lettering F6, and the Axial Lettering screen appears. Rectangular coordinates is the default. The Coordinates field can be changed to Polar.

The Axial Lettering screen defaults to Linear Text Style. The Style field can be changed to Circular Text Style.

The fields in the upper pane on the Axial Lettering screen are defined as follows:

- **TOOL**—identifies the tool number that will be used in the part program.
- **TOOL OFFSETS**—identifies the tool offset and orientation programmed in Tool Setup that will be used. The tool offset defaults to the same number as the tool.
- **TOOL DIAMETER**—contains the tool diameter programmed in Tool Setup for the current tool. This field is read-only and can only be edited in Tool Setup.
- **SPEED (RPM)**—identifies the speed at which the spindle rotates to cut the part. This value is carried forward for the selected tool from Tool Setup. To accept this value, select Enter. A different speed can be programmed for this specific block. Override the displayed value by typing in a new speed or by using the appropriate console Spindle override knob.
- **MILL FEED**—identifies the cutting feedrate.
- **PLUNGE FEED**—identifies the feedrate between Z Start and Z Bottom.

The remaining fields on the Axial Lettering screen are defined as follows:

- **SELECT FONT** screen button—displays a Font window with Font and Size selections and a Sample pane. Height is set by default based on chosen size, and when the Width field contains 0, the font and Height field calculate width.
- **STYLE**—defines the text optimization as either **Any** or **Stick** in addition to a second field to specify style as **Linear** or **Circular**. Any and Linear are the defaults.
  - **Any**—fonts that are not optimized for stick lettering.
  - **Stick**—for use only with specialized machining fonts designed to cut stick letters.

  ➠ Selecting **Stick** does **not** convert fonts to Stick Lettering.

- **Linear**—the tool cuts along a line.
- **Circular**—the tool cuts unwrapped (flat) text on an arc.
• **Z START**—defines the location where Plunge Feedrate begins. The feedrate is set in the Plunge Feed field.

• **R PLANE**—defines an absolute position, relative to the surface of the part. The cycle will rapid to this location before feeding to Z Bottom.

• **Z BOTTOM**—defines the bottom coordinate and the location where the Plunge Feedrate ends.

• **TEXT**—contains the text for the lettering block.

• **LINEAR Y MOTION** checkbox—select this checkbox to cause Y-Axis Motion using the X and X’ axes. When this checkbox is clear, Y-Axis Motion occurs using the X and C axes. Please refer to Programming Coordinates and Linear Y Motion, on page 2 - 68 for examples. This field is available with TMX MY and TMX MYS series machines.

  Linear Y-Axis Motion is limited. The amount of Y travel is affected by the X position and by any X-Axis tool length offset.

• **C**—defines the angle to position the C-Axis. This field appears when Linear Y Motion checkbox is selected. This field is available with TMX MY and TMX MYS series machines.

• **SPACE BETWEEN CHARS**—adds space between characters in addition to the space used with the selected font.

• **HEIGHT REF LOC**—specifies Bottom, Center, or Top of the character based on the (X, Y) or (R, C) reference point. The default is Bottom.

• **HEIGHT**—defines the height of the text character(s). When the Width field contains 0, the font and Height field calculate width.

• **WIDTH REF LOCATION**—specifies Start, Center, or End of the character for the Y/ reference point. The default is Start.

• **ORIENTATION**—defines the start angle of the direction for the lettering on the stock.

• **COORDINATES**—specifies Rectangular or Polar for the coordinates. The axes listed on the screen change depending upon this selection.

<table>
<thead>
<tr>
<th>Linear Style</th>
<th>Circular Style</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Rectangular</strong></td>
<td><strong>Polar</strong></td>
</tr>
<tr>
<td><strong>Coordinates</strong></td>
<td><strong>Coordinates</strong></td>
</tr>
<tr>
<td>X Reference</td>
<td>R Reference</td>
</tr>
<tr>
<td>Y Reference</td>
<td>C Reference (Deg)</td>
</tr>
<tr>
<td>Z Start</td>
<td>Z Start</td>
</tr>
<tr>
<td>R Plane</td>
<td>R Plane</td>
</tr>
<tr>
<td>Z Bottom</td>
<td>Z Bottom</td>
</tr>
<tr>
<td><strong>Rectangular</strong></td>
<td><strong>Polar</strong></td>
</tr>
<tr>
<td><strong>Coordinates</strong></td>
<td><strong>Coordinates</strong></td>
</tr>
<tr>
<td>X Center</td>
<td>R Center</td>
</tr>
<tr>
<td>Y Center</td>
<td>C Center (Deg)</td>
</tr>
<tr>
<td>Z Start</td>
<td>Z Start</td>
</tr>
<tr>
<td>R Plane</td>
<td>R Plane</td>
</tr>
<tr>
<td>Z Bottom</td>
<td>Z Bottom</td>
</tr>
</tbody>
</table>
Linear Style

- **X/R REFERENCE**—indicates the X/R coordinate.
- **Y/C (Deg) REFERENCE**—indicates the Y/C (Deg) coordinate.
- **WIDTH**—defines the width of the text. When the default of 0.0000 is used, the font and Height calculates the width used with Linear Style.

Circular Style

- **X/R CENTER**—indicates the R (radius) coordinate.
- **Y/C (Deg) CENTER**—indicates the R (radius) coordinate.
- **WIDTH (Deg)**—programs the text as a sweep angle. Enter an angular value from 0 to 360° when Circular Style is selected.
- **DIAMETER**—appears when Circular Style is selected. Defines the diameter of the circle for circular text.
- **DIRECTION**—defines the direction for circular text.

Here is an example showing Axial Lettering with Circular and Linear Reference points.

![Figure 2–148. Axial Lettering with Circular and Linear Reference points](image-url)
Axial Holes

From the New Block screen, select LIVE TOOLING → F3. Select Axial Holes F5 from the New Block Live Tooling softkey menu. The Axial Holes screen appears.

![Axial Holes Screen](image)

**Figure 2–149. Live-Tooling Axial Holes screen**

The Axial Holes screen defaults to the Drill Operation. When the cursor is in the OPERATION field, these softkey choices appear. Upon editing the last field in each holes operation, the cursor returns to the Operation field.

- **PREVIOUS HOLE OPERATION** F1—displays the previous Hole Operation in this data block. This softkey is not available for the first operation.
- **NEXT HOLE OPERATION** F2—displays the next Hole Operation in this data block. This softkey is not available for the last operation.
- **DELETE HOLE OPERATION** F3—deletes the current Hole Operation. Select this softkey and a message appears asking if you are sure you want to delete. Select OK or CANCEL.
- **INSERT HOLE OPERATION** F7—accesses New Axial Holes Operation screen with these Axial Holes softkey selections:
  - **AXIAL HOLE CYCLE** F1—accesses the Axial Holes Drill data block for programming different types of drilling cycles. Refer to *Axial Hole Cycle*, on page 2 - 179 for programming details.
  - **AXIAL LOCATIONS** F2—accesses the Axial Locations data block for programming drilling locations. Refer to *Axial Locations*, on page 2 - 182 for programming details.
  - **AXIAL BOLT CIRCLE** F3—accesses the Axial Holes Bolt Circle data block for programming bolt circles. Refer to *Axial Bolt Circle*, on page 2 - 185 for programming details.
Axial Hole Cycle

From the Axial Holes screen, with the cursor in the Operation field, either select NEXT HOLE OPERATION F2 or INSERT HOLE OPERATION F7. From the New Axial Holes Operation screen select Axial Hole Cycle F1. The Axial Holes Drill screen appears.

![Axial Holes Drill Screen](image)

*Figure 2–150. Live-Tooling Axial Holes Drill Operation screen*

The fields on this screen are defined as follows:

- **TOOL**—identifies the tool number that will be used in the part program.
- **TOOL OFFSETS**—identifies the tool offset and orientation programmed in Tool Setup that will be used. The tool offset defaults to the same number as the tool.
- **Z START**—defines the point where the tool moves when starting the cycle, before moving at Rapid to the R Plane.
- **R PLANE**—defines an absolute position relative to part zero. The cycle will rapid to this location before feeding to Z Bottom.
- **Z BOTTOM**—identifies the depth of the hole. This field is automatically adjusted for the drill tip if YES is selected for Tip Compensation.
- **TIP COMPENSATION**—This field appears when the tool is a Drill or a Center Drill. Select YES or NO to indicate whether the tool tip is included as part of the Z Bottom measurement. Z Bottom depth is automatically calculated for the drill tip when YES is selected and is based on the drill tip angle programmed in Tool Setup.
- **TOOL DIAMETER**—contains the tool diameter programmed in Tool Setup for the current tool. This field is read-only and can only be edited in Tool Setup.
- **SPEED (RPM)**—contains the spindle speed value programmed in Tool Setup. “CAL” appears next to the value as programmed in Tool Setup. You can accept or change this value. If you edit this field, “CAL” disappears, and the value is no longer calculated automatically.
- **PLUNGE FEED**—contains the feedrate between the Z START and Z BOTTOM positions. “CAL” appears next to the value as calculated by the control. You can accept or change this value. If you edit this field, “CAL” disappears.

- **CYCLE TYPE**—accesses the cycle types. Refer to Tap Cycle below for information about the Standard Tap cycle.
  - **Standard** drilling feeds to depth and retracts at Rapid to the retract dimension (R-Plane). Standard is the default operation.
  - **Dwell** drills with a dwell at the bottom.
  - **Peck Drilling** feeds to a peck depth and retracts at Rapid to the Retract dimension (R-Plane).
  - **Chip Breaker** feeds to a peck depth, retracts at Rapid a short distance, then feeds to the next peck depth. This is repeated until the programmed depth is achieved.
  - **Center Drill** cycle creates a pilot hole. You can use a center drill or a standard drilling hole to perform this operation.
  - **Decreasing Peck Drilling Depth** (Decr Depth) cycle feeds to the programmed First Peck Depth, then continues cutting a decreasing amount to the Minimum Peck by the peck percent each pass until it reaches the programmed Minimum Peck Depth or Z Bottom.
  - **Decreasing Chip Breaker** (Decr Chp Bkr) cycle feeds to the programmed First Peck Depth, retracts at Rapid a short distance, then feeds to the next peck depth, cutting a decreasing amount to the minimum peck by the peck percent each pass until it reaches the programmed Minimum Peck Depth or Z Bottom. This is repeated until the programmed depth is achieved.

**Drilling Operation**

These fields appear for performing drilling operations based on the Cycle Type selection:

- **DWELL TIME (SEC)**—specifies the time in seconds the Z axis dwells after the cycle reaches Z Bottom and before retracting at Rapid. This field appears when either Dwell or Center Drill is selected for the Cycle Type.

- **PECK DEPTH**—defines the maximum depth to be cut in one pass. This field appears when either Peck, Chip Breaker, Decreasing Depth, or Decreasing Chip Breaker is selected for the Cycle Type.

- **FINAL PECK DEPTH**—defines the maximum depth to be cut in the final pass. This field appears when either Decreasing Depth or Decreasing Chip Breaker is selected for the Cycle Type.

- **PECK PERCENT**—defines the percent of the First Peck Depth. For example, if the Peck Percent is set at 50, subsequent pecks will equal a distance 50% smaller than the previous peck depth. A Minimum Peck Depth must be set in order for the tool to reach Z Bottom. This field appears when either Decreasing Depth or Decreasing Chip Breaker is selected for the Cycle Type.

- **PECK COUNT**—displays the number of pecks that will occur. This read-only field appears when either Peck, Chip Breaker, Decreasing Depth, or Decreasing Chip Breaker is selected for the Cycle Type.
• **PECK CLEARANCE**—defines the location above the last peck level for returning at rapid feed rate. This field appears when either Peck, Chip Breaker, Decreasing Depth, or Decreasing Chip Breaker is selected for the Cycle Type. The default is 0.254 mm (0.010 in).

**Tapping Operation**

These fields appear for performing tapping operations when the tool is a Live Tap:

- **TAP CYCLE**—accesses Standard or Rigid Tap cycle selections. This field appears when the tool is a Tap.
  - *Standard* cuts threads inside a hole.
  - **Rigid** taps the same hole multiple times, maintaining the tool’s orientation with previously cut threads.

  When performing rigid tapping, the hole must be longer than the tap. To accomplish this, program the preceding drill operation to be longer than the tap operation.

Here are some suggestions when Tapping:

- Use a floating tap holder to prevent the tap from breaking when the spindle reverses direction.
- When tapping hard materials, use tapping fluid (not coolant) as a lubrication.
- When tapping hard materials, use a tap drill one size larger than normal because less thread engagement requires less torque.
- The recommended Z Start is 0.200 minimum from the part surface to compensate for floating tap holder characteristics.

These fields appear based on the Tap Cycle selection:

- **PITCH**—defines the distance between threads for metric tools. Do not use this field for programming tools measured in threads per inch. This read-only field, defined in Tool Setup, appears when the tool is a Live Tap.
- **TPI**—defines the threads per inch for tools measured in inches or the threads per millimeter for metric tools. This read-only field, defined in Tool Setup, appears when the tool is a Live Tap.
- **DWELL TIME (SEC)**—specifies the time in seconds the Z axis dwells after the cycle reaches Z Bottom and before retracting at Rapid. This field appears when Standard is selected for the Tap Cycle type.
- **PECK DEPTH**—defines the maximum depth to be cut in one pass. This field appears when Rigid is selected for the Tap Cycle Type and the tool is a Live Tap.
Reaming Operation

These fields appear for performing reaming operations when the tool is a Live Ream:

- **RETRACT AT RAPID**—defines the retract as rapid or feed. Select Yes for rapid or No for feed. This field is available when the tool is a Live Ream.
- **DWELL TIME (SEC)**—specifies the time in seconds the Z axis dwells after the cycle reaches Z Bottom and before retracting at Feed. This field appears when the tool is a Live Ream and No is selected for Retract at Rapid.

Axial Locations

From the Axial Holes screen, with the cursor in the Operation field, either select NEXT HOLE OPERATION F2 or INSERT HOLE OPERATION F7. From the New Axial Holes Operation screen select Axial Locations F2. The Axial Holes Locations screen appears. The fields are based on your choice in the Coordinates field (Polar or Rectangular).

For information about Patterns, refer to Patterns, on page 2 - 57.

The fields on the Axial Locations screen are defined as follows:

- **COORDINATES**—defines the coordinates as either Linear or Rotary.
  - Rectangular coordinates are used when working from a flat drawing.
  - Polar coordinates move around the cylinder as specified.
- **Location Number**—identifies the sequential number for the location. When the fields for each location are filled in, press Enter. The next row of fields becomes active and the Location Number field increments by one.

When the cursor is in one of the Location fields (R, C (DEG), X, Y, or Retract) these softkey choices are available:

- **DELETE LOCATION** F4—delete a selected row of location fields.
- **ADD LOCATION** F5—add a row of location fields to the bottom of the list.
- **INSERT LOCATION BEFORE** F7—insert a row of location fields above the selected row.

The remaining fields appear based on your selection in the Coordinates field (Polar or Rectangular).
When Polar is selected in the COORDINATES field, these fields appear:

- **USE C CLAMP checkbox**—engages the C Axis Clamp during the Hole operation when this checkbox is selected.
- **R**—defines starting distance from the centerline.
- **C (DEG)**—defines the C offset value in degrees for the location, measured counterclockwise from the home position.
- **RETURN**—identifies the level to which the tool should retract at the end of the drill cycle. R Plane is the default.
  - **I PLANE**—returns to the coordinate specified in the X/Z Start field.
  - **R PLANE**—returns to the coordinate specified in the R Plane field.

*Figure 2–151. Live-Tooling Axial Locations screen: Polar Coordinates*
When Rectangular is selected in the COORDINATES field, these fields appear:

- **USE C CLAMP** checkbox—engages the C Axis Clamp during the Hole operation when this checkbox is selected.
- **X**—defines the X coordinate on an XY plane with origin at the center of the face.
- **Y**—defines the Y coordinate on an XY plane with origin at the center of the face.
- **RETURN**—identifies the level to which the tool should retract at the end of the drill cycle. R Plane is the default.
  - **I PLANE**—returns to the coordinate specified in the X/Z Start field.
  - **R PLANE**—returns to the coordinate specified in the R Plane field.
- **LINEAR Y MOTION**—select this checkbox to cause Y-Axis Motion using the X and X’ axes. When this checkbox is clear, Y-Axis Motion occurs using the X and C axes. Please refer to Programming Coordinates and Linear Y Motion, on page 2 - 68 for examples. This field is available with TMX MY and TMX MYS series machines.

Linear Y-Axis Motion is limited. The amount of Y travel is affected by the X position and by any X-Axis tool length offset.
Axial Bolt Circle

From the Axial Holes screen, with the cursor in the Operation field, either select NEXT HOLE OPERATION F2 or INSERT HOLE OPERATION F7. From the New Axial Holes Operation screen select Axial Bolt Circle F3. The Axial Holes Bolt Circle screen appears. The fields are based on your choice in the Coordinates field (Polar or Rectangular).

The fields on the Axial Bolt Circle screen are defined as follows:

- **COORDINATES**—defines the coordinates as either Linear or Rotary.
  - **Rectangular** coordinates are used when working from a flat drawing.
  - **Polar** coordinates move around the cylinder as specified.

When Polar is selected in the COORDINATES field, these fields appear:

- **USE C CLAMP** checkbox—engages the C Axis Clamp during the Hole operation when this checkbox is selected.
- **NUMBER OF HOLES**—defines the number of holes for the bolt circle.
- **R CENTER**—defines the starting distance from the centerline. The default is 0.
- **C CENTER (DEG)**—defines the center of the bolt circle, measured counterclockwise from the home position. The default is 0.
- **RADIUS**—defines the radius of the bolt circle.
- **RETURN**—identifies the level to which the tool should retract at the end of the drill cycle. R Plane is the default.
  - **I PLANE**—returns to the coordinate specified in the X/Z Start field.
  - **R PLANE**—returns to the coordinate specified in the R Plane field.
- **START ANGLE**—defines the angle to the first hole, measured counterclockwise from the home position.
• **SKIP LIST**—provides fields for entering the hole number to be skipped moving in the counterclockwise direction.

When Rectangular is selected in the COORDINATES field, these fields appear:

![Image of Rectangular Coordinates](image)

**Figure 2–154. Live-Tooling Axial Bolt Circle screen: Rectangular Coordinates**

- **USE C CLAMP checkbox**—engages the C Axis Clamp during the Hole operation when this checkbox is selected.
- **X CENTER**—defines the X coordinate on an XY plane with origin at the center of the face. The default is 0.
- **Y CENTER**—defines the Y coordinate on an XY plane with origin at the center of the face. The default is 0.
- **RADIUS**—defines the radius of the bolt circle.
- **RETURN LEVEL**—identifies the level to which the tool should retract at the end of the drill cycle. R Plane is the default.
  - **I PLANE**—returns to the coordinate specified in the X/Z Start field.
  - **R PLANE**—returns to the coordinate specified in the R Plane field.
- **START ANGLE**—defines the angle to the first hole, measured counterclockwise from the home position.
- **LINEAR Y MOTION**—select this checkbox to cause Y-Axis Motion using the X and X’ axes. When this checkbox is clear, Y-Axis Motion occurs using the X and C axes. Please refer to Programming Coordinates and Linear Y Motion, on page 2 - 68 for examples. This field is available with TMX MY and TMX MYS series machines.

⚠️ Linear Y-Axis Motion is limited. The amount of Y travel is affected by the X position and by any X-Axis tool length offset.

- **SKIP LIST**—provides fields for entering the hole number to be skipped moving in the counterclockwise direction.
When the cursor is in one of the Skip List fields, these softkeys are added to the menu:

- **CLEAR SKIP LIST ENTRY**  F3—deletes the entry, leaving the field empty.
- **CLEAR ENTIRE SKIP LIST**  F4—deletes all Skip List entries, leaving the fields empty.
- **CONVERT TO LOCATIONS BLOCK**  F5—converts the Bolt Circle data block to a Locations data block. Select this softkey and this message appears: “Warning! This block will be deleted. Continue?” Select Ok or Cancel.
- **COPY TO LOCATIONS BLOCK**  F6—copies the Bolt Circle data block as a Locations data block.
Cutter Compensation

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

Cutter compensation allows you to choose if the tool should cut on the left or right side of the contour, or if the center line of the programmed tool should follow the part contour (print line). The default Milling Type field setting for cutter compensation is ON, which means the tool cuts on the programmed path.

With each of the Compensation selections, the programmed tool automatically follows the finished contour of the part.

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

![Figure 2–155. Cutter Comp ON (no cutter compensation)](image1)

![Figure 2–156. Cutter Comp Left](image2)

![Figure 2–157. Cutter Comp Right](image3)

<table>
<thead>
<tr>
<th>A</th>
<th>Beginning point of programmed direction (clockwise tool movement from point A to point B)</th>
</tr>
</thead>
<tbody>
<tr>
<td>B</td>
<td>Ending point of programmed direction (clockwise tool movement from point A to point B)</td>
</tr>
</tbody>
</table>
The following are the types of cutter compensation used for Axial and Radial Mill Contours such as Lines, Arcs, and Blend Arcs:

- **ON**—used for cutting directly on the programmed path. On is the default.
- **LEFT**—used for cutting on the left side of the centerline.
- **RIGHT**—used for cutting on the right side of the centerline.
- **PROFILE LEFT**—used for removing material using multiple cut passes in varying widths moving from the left side of the contour toward the programmed contour centerline. When either Profile Left or Profile Right is selected, the Pocket Overlap (%) field is available for entering the percentage of overlap during cutting. In addition, the Max Offset field is available for defining the location for milling to begin, or the distance away from the programmed contour centerline.
- **PROFILE RIGHT**—used for removing material using multiple cut passes in varying widths from the right side of the contour toward the programmed contour centerline. When either Profile Left or Profile Right is selected, the Pocket Overlap (%) field is available for entering the percentage of overlap during cutting. In addition, the Max Offset field is available for defining the location for milling to begin, or the distance away from the programmed contour centerline.
- **POCK BOUNDARY (POCKET BOUNDARY)**—used for cutting the defined closed contour.

The following are the types of cutter compensation used for Axial and Radial Circles and Frames:

- **ON**—used for cutting directly on the programmed path.
- **INSIDE**—causes the tool to enter the part inside the contour and blend into it using an 180° arc. Cutter compensation is automatically employed and the outside edge of the tool follows the contour (the direction the tool travels depends upon milling direction). The point where the tool enters the part is determined by the blend offset and the point where the tool is withdrawn is determined by the blend overlap and blend offset.
- **OUTSIDE**—similar to the Inside function, except the tool enters the part outside the contour and follows the outside of the contour. The tool is withdrawn adjacent to the frame or circle.
- **IN/TAN (INSIDE TANGENT)**—causes the tool to enter the part adjacent to the inside contour. Cutter compensation is automatically used and the edge of the tool follows the inside of the contour. The direction the tool travels depends upon the milling direction. The tool is withdrawn from the part adjacent to the frame or circle.
- **OUT/TAN (OUTSIDE TANGENT)**—similar to the Inside Tangent function except that the tool enters the part adjacent to the outside frame contour. The tool’s edge then follows the outside of the contour. The tool is withdrawn from the part adjacent to the frame or circle.
- **POCK BOUNDARY (Pocket Boundary)**—used for cutting the outside perimeter of a pocket operation.
Climb Milling

Climb milling is the preferred method of cutter compensation, except when the fixturing is not rigid. In climb milling, the tool cuts in the same direction as the feeding motion, and the first tooth contacts the workpiece at the top of the cut. The width of the chip decreases from the maximum as the cutter removes material. This is also known as “in-cut” or “down milling.”

The advantages of using climb milling are as follows:

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

Conventional Milling

Conventional milling is used when the fixturing is not rigid. During conventional milling, the cutting teeth move in the opposite direction to the feeding motion, and the first tooth contacts the workpiece at the bottom of the cut. The width of the chip starts at zero and increases as the cutter removes material. This is known as “out-cut” or “up milling.”

The advantages of using conventional milling are as follows:

- The chip thickness starts at zero, causing less impact on the cutting teeth. This is ideal for setups that are not very rigid.
- The backlash in older machines is greatly diminished.

⇒ The programming sequence of Lines and Arcs segments determines the cutting direction.
Miscellaneous Data Blocks

To access the Miscellaneous softkey menu from the Program Review screen or from the New Block screen, select the MISCELLANEOUS \[F4\] softkey to access the following data blocks:

- Change Parameters ................................................................. 2 - 192
- Change Part Setup ................................................................. 2 - 195
- Machine Function—M Code ..................................................... 2 - 198
- Bar Feed Block ................................................................. 2 - 200
- Comment Block ................................................................. 2 - 201
- Repeat Start Block ............................................................. 2 - 202
- Repeat End Block ............................................................. 2 - 203
Change Parameters

Program Parameters are available for entering the rapid traverse feedrate, setting maximum spindle RPM and disabling or enabling axis feedrate, rapid, spindle speed override, and setting an optional stop before or after each tool change.

Program Parameters programmed from the Input screen are in effect when running a part program until the end of a program or until a Change Parameter data Block is programmed.

Program Parameters are stored with the conversational part program. These parameters apply to all data blocks in the conversational part program and are available for

- Entering the rapid traverse feedrate.
- Setting maximum spindle RPM.
- Disabling or enabling axis feedrate override.
- Setting an optional stop before or after each tool change.
- Defining a tool change position for each tool change.

The Change Parameters screen displays with the values entered in Program Parameters. Edit these values for a Change Parameters data block.

Refer to Getting Started with WinMax Lathe, Program Parameters, on page 4 - 82 for more information about Program Parameters and the Input screen.

To access the Change Parameters screen from the Program Review screen or from the New Block screen, select the MISCELLANEOUS ➔ F4 softkey followed by the CHANGE PARAMETERS F1 softkey.

Figure 2–158. Change Parameters screen
These fields are available for changing program parameters:

- **RAPID TRAVERSE**—contains the feedrate for the axes to move between data blocks or in position blocks for positioning to turret rotation location. The default is 2540 MMPM (100 IPM).

- **MAX RPM (PRIMARY SPINDLE)**—identifies the maximum spindle speed for the primary spindle, no matter what CSS/FPM is programmed. This field can be set for each program depending on the setup, such as type of jaws, workpiece weight, workpiece strength. The default is 2540 MMPM (100 IPM).

- **MAX RPM (SUB-SPINDLE)**—identifies the maximum spindle speed for the sub-spindle, no matter what CSS/FPM is programmed. This field can be set for each program depending on the setup, such as type of jaws, workpiece weight, workpiece strength. This field appears for TMX MYS series machines. The default is 2540 MMPM (100 IPM).

- **MAX RPM (LIVE TOOLING SPINDLE)**—identifies the maximum spindle speed for the live tooling spindle, no matter what CSS/FPM is programmed. This field can be set for each program depending on the setup, such as type of jaws, workpiece weight, workpiece strength. This field appears for TMM, TMX MY, and TMX MYS machines. The default is the same as the machine maximum RPM.

- **FEED OVERRIDE LOCKOUT**—disables the axis feedrate override when YES is selected. The program will run in Auto at the programmed feedrate regardless of the position of the override knobs.

    The Feed Override Lockout does not affect the Rapid and Spindle overrides.

- **STOP BEFORE TOOL CHANGES**—inserts an optional stop (M01) before each tool change if YES is selected. You must also enable the Opt Stop console key to activate this function when running the program. If YES is selected and the Opt Stop key is not activated, the optional stop will not occur.

    Select NO if an optional stop is not required before each tool change. NO is the default.

- **STOP AFTER TOOL CHANGES**—inserts an optional stop (M01) after each tool change if YES is selected. You must also enable the Opt Stop console key to activate this function when running the program. If YES is selected and the Opt Stop button is not turned on, the optional stop will not occur.

    Select NO if an optional stop is not required after each tool change. NO is the default.
• **USER DEFINED TOOL CHANGE POSITION**—contains the following tool change selections for use in a Change Parameters data block to override the settings made in Program Parameters. Program these User Defined Tool Change Position settings when the tool specified in the following data block is different from the tool in the previous block or when the tool specified is different from the current tool and requires different tool change settings:

  • **WHERE**—moves turret to the selected position:
    - **DON’T MOVE**—perform the tool change at current machine position.
    - **MOVE TO HOME**—the turret will move to the home position for the tool change.
    - **MOVE TO XZ**—or the turret will move to the defined X, Z position. When MOVE TO XZ is selected, the Reference and X (Dia) or X (Rad) fields are available for entering data.

  • **REFERENCE**—determines whether the X and Z locations are relative to Part Zero or Machine Zero.

  • **X (Diameter or Radius)**—contains the X diameter or radius coordinate where the turret should move. When the cursor is in this field, the STORE MACHINE POSITION F2 softkey appears. Select this softkey to store the current machine position into the field.

  • **Z**—contains the Z coordinate where the turret should move. When the cursor is in this field, the STORE MACHINE POSITION F2 softkey appears. Select this softkey to store the current machine position into the field.

  • **AXIS ORDER**—contains the selection for the order the axes will move:
    - **MOVE X THEN Z**—from the current position, the turret moves the X axis first to the X position then the Z axis to the Z position.
    - **MOVE Z THEN X**—from the current position, the turret moves the Z axis first to the Z position then the X axis to the X position.
    - **SIMULTANEOUS**—from the current position, the turret moves both axes together to the X and Z positions.
Change Part Setup

Work offsets programmed in Part Setup are in effect when a part program is running until the end of the program or if a Change Part Setup data block is programmed. Work Offset coordinates can be changed in the Change Part Setup screen. When a Conversational program is saved, the current Work Offsets are saved as part of the program.

Refer to Getting Started with WinMax Lathe, Part Setup—Work Offsets, on page 4 - 11 for more information about setting Work Offsets in Part Setup.

To access the Change Part Setup screen from the Program Review screen or from the New Block screen, select the MISCELLANEOUS ➔ F4 softkey followed by the CHANGE PART SETUP F2 softkey.

These fields are available for changing part setup information:

- **#**—identifies the index value for each offset.
- **C OFFSET checkbox**—select the checkbox to enable to C Offset fields. When the checkbox is cleared, the X Offsets cannot be edited. The status of this checkbox is modal. It stays in effect until you change it, even upon a control reboot. This field is available for the TMM, TMX MY, and TMX MYS Series machines.
- **X OFFSET checkbox**—select this checkbox to enable the X Offset fields. When the checkbox is cleared, the X Offsets cannot be edited. The status of this checkbox is modal. It stays in effect until you change it, even upon a control reboot.
- **C Offset**—indicates the C location in machine coordinates for part zero. This field is available for the TMM, TMX MY, and TMX MYS Series machines.
- **X OFFSET**—indicates the X location in machine coordinates for part zero.
• **Z OFFSET**—indicates the Z location in machine coordinates for part zero.

  When the cursor is in a Z Offset field and **Store Machine Position** is pressed, a pop-up window appears requesting a tool offset (0-99). The control will adjust for the tool offset before entering the position in the selected field.

  The pop-up contains the active tool’s programmed geometry offset. If the active tool is not programmed in Tool Setup, then the active tool number is used as the default offset.

  Entering a zero (0) does not apply an adjustment.

  After entering or accepting the tool offset, select **OK** to continue or **Cancel** to stop the operation.

• **W Offset (Tailstock or Sub-spindle) checkbox**—select this checkbox to enable the W Offset fields. When the checkbox is cleared, the W Offsets cannot be edited. The status of this checkbox is modal. It stays in effect until you change it, even upon a control reboot. This field is available for Two-Axis Programmable Tailstock (TMX series) turning centers only.

• **Y Offset checkbox**—select this checkbox to enable the Y Offset fields. When the checkbox is cleared, the Y Offsets cannot be edited. The status of this checkbox is modal. It stays in effect until you change it, even upon a control reboot. This field is available for Two-Axis Programmable Tailstock (TMX series) and Live-Tooling, Y-Axis Motion, and Sub-spindle (TMX MYS series) turning centers only.

• **W OFFSET (Tailstock or Sub-spindle)**—contains the W location in machine coordinates for part zero. This location is referenced from Machine Zero as shown in Figure 4–7. **Part Offset Relative to Machine Zero, on page 4 - 11.** This field is available for Two-Axis Programmable Tailstock (TMX series) turning centers only.

• **Y OFFSET**—contains the Y location in machine coordinates for part zero. Y-Axis motion allows the turret to move up or down and maintain the X-Axis position. This field is available for Live-Tooling and Y-Axis Motion (TMX MY series) and Live-Tooling, Y-Axis Motion, and Sub-spindle (TMX MYS series) turning centers.

• **On Sub-spindle check box**—indicates the offsets are applied to the sub-spindle when selected. This field is available for Live-Tooling, Y-Axis Motion, and Sub-spindle (TMX MYS series) turning centers only.

  All conversational part program data blocks following the offset are applied to the sub-spindle when an On Sub-spindle check box is selected.

• **ACTIVE OFFSET**—contains the index value (1 – 99) specifying which offset is active.

  The Active Offset for a Change Part Setup block appears in the Program Review screen, listed next to the block as E:# where E is an NC command and # is the Active Offset. Refer to **Program Review, on page 1 - 8** for information about the Program Review screen.
These softkeys are available for changing part setup information when the cursor is in an X OFFSET or Z OFFSET field:

- **PAGE UP F1**—scrolls up through the # fields, displaying groups of offsets. This softkey is also available when the cursor is in the Active Offset field.
- **PAGE DOWN F2**—scrolls up through the # fields, displaying groups of offsets. This softkey is also available when the cursor is in the Active Offset field.

  The **Page Up** and **Page Down** keys function like the **PAGE UP F1** and **PAGE DOWN F2** softkeys when an offset is selected in Change Part Setup. If an offset is not selected, these keys jump to the previous block.

- **ADDITIONAL AXIS OFFSETS F3**—toggles the screen to a second screen containing W Offset (Tailstock or Sub-spindle) fields and Y Offset fields.
  - The W Offsets are available for Programmable Tailstock (TMX series) and Live-Tooling and Y-Axis Motion (TMX MY series) and Live-Tooling, Y-Axis Motion, and Sub-spindle (TMX MYS) turning centers.
  - The Y Offsets are available for Live-Tooling and Y-Axis Motion (TMX MY series) turning centers and Live-Tooling, Y-Axis Motion, and Sub-spindle (TMX MYS) turning centers.

- **STORE MACHINE POSITION F5**—(or **Store Machine Position** console key)—sets the current axis position as a Part Zero location. The cursor location defines which axis (X or Z) will be set. The current axis position appears in the DRO Part column as zero.

- **TABLE COMMANDS F7**—displays an additional softkey menu with choices for saving and restoring the offsets.
  - **WRITE TABLE TO FILE F2**—opens a Save As dialog box for naming and saving the offsets to a file. You can assign the file extension of your choice in the File name field. It is not possible to select an extension using the Files of type field.
  - **READ TABLE FROM FILE F4**—opens the Open dialog box for selecting a folder and filename. When an offset file is selected from this dialog box, those offsets are read from the file and the values appear on the Part Setup - Work Offsets screen.
  - **CLEAR TABLE F7**—resets all values in the table to 0.000. A confirmation window appears prior to resetting the values.
  - **EXIT F8**—exits the Table Commands menu and returns to the first Part Setup - Work Offsets menu.
Machine Function—M Code

A Machine Function data block allows you to insert Machine Function commands into a part program using the selections in the list on this screen and repeat the command a specified number of times.

To access the Machine Function—M Code screen from the Program Review screen or from the New Block screen, select the MISCELLANEOUS ➤ F4 softkey followed by the M CODE F3 softkey.

The MACHINE FUNCTION screen appears.

![Machine Function Screen](image)

**Figure 2–160. Machine Function screen**

These fields are available for setting machine functions:

- **M CODE**—indicates the M Code selection for the machine function. Either enter the code in this field or scroll through the list of Machine Code definitions below this field and select a code. Refer to WinMax Lathe NC Programming, *Basic NC M Codes, on page 3 - 1* or *ISNC M Codes, on page 5 - 1* for complete descriptions of all M Codes.

- **Dwell**—indicates the amount of seconds to dwell following the M Code function.

- **SPINDLE**—indicates spindle selection when either M3 (Spindle CW), M4 (Spindle CCW), M5 (Spindle Off), or M19 (Spindle Orient) is selected in the M Code field.

- **SPEED**—indicates the Spindle RPM when either M3 or M4 is selected in the M Code field.

- **DEGREES**—indicates the spindle orientation position when M19 (Spindle Orient), M203 (Sync Spindles Forward Mode), or M204 (Sync Spindle Reverse Mode) are selected in the M Code field.
• **TIME ON (SEC)**—the amount of seconds the part conveyor or chip conveyor should run. The default is 0. Use with TIME OFF (SEC) to cycle the conveyor on and off. This field appears when either M24 (Part Conveyor On) or M50 (Chip Conveyor On) is entered in the M Code field.

• **TIME OFF (SEC)**—indicates the amount of seconds the part conveyor or chip conveyor should not run. The default is 0. Use with TIME ON (SEC) to cycle the conveyor on and off. This field can only be used when a value other than 0 is entered in the TIME ON field. This field appears when either M24 (Part Conveyor On) or M50 (Chip Conveyor On) is entered in the M Code field.

⚠️ If the part conveyor or chip conveyor is running at the end of a program, it will continue running for the specified amount of time.

If cycling, the conveyor will not turn on again if stopped by reaching the end of the program, Manual Mode is selected, or the enclosure doors are opened.

If the default value of 0 is used in these fields,

- The part conveyor will run until either the program encounters an M25 Part Conveyor Off code, Manual Mode is selected, or the enclosure doors are opened.
- The chip conveyor will run until either the program encounters an M51 Conveyor Off code, Manual Mode is selected, or the enclosure doors are opened.

⚠️ The part conveyor or chip conveyor will stop running if Manual Mode is selected or the enclosure doors are opened.
Bar Feed Block

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

- Bar pushing requires the optional bar feeder or a servo driven bar feeder. When a piece of stock is depleted, the optional bar feeder replaces it with a new piece as necessary. Refer to WinMax Lathe Options Bar Feeder, on page 3 - 1 for details about programming data blocks with the optional Bar Feeder.

- Select the Tool as Bump Stop or Tool Guides Stock Strategy if the optional Bar Feeder is used. The optional bar feeder places stock into the spindle, through the hole in the left side of the machine.

- Select the Tool not Used Strategy if a servo driven bar feeder is used.

- Bar pulling requires a special pulling tool loaded in the turret. You can program a tool in Tool Setup to use with a Bar Feed Block to pull the stock away from the feeder. Select the Tool Pulls Stock Strategy if the pulling tool is programmed in Tool Setup and the optional Bar Feeder is not being used.

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

Refer to WinMax Lathe Options, Bar Feeder, on page 3 - 1 for details about programming data blocks with the optional Bar Feeder.

The BAR FEED BLOCK screen appears.

![Bar Feed Block Screen](image-url)
Comment Block

A Comment block can be used to provide the operator with instructions during part program execution. Optional stops can also be programmed with this block.

To access the Comment Block screen from the Program Review screen or from the New Block screen, select the MISCELLANEOUS ➔ F4 softkey followed by the COMMENT F5 softkey.

The COMMENT BLOCK screen appears.

Figure 2–162. Comment Block screen

These fields are available for programming a Comment block:

- **STOP TYPE**—select None, Program, or Optional.
  - **NONE**—displays the comment in a pop-up window when the program reaches this data block without stopping the program.
  - **PROGRAM**—stops the program, displays the comment in a pop-up window, and shuts off the spindle when the program reaches this data block. Press the flashing **Start Cycle** button to continue with the part program. The comment is cleared when the program resumes.
  - **OPTIONAL**—stops the program, displays the comment in a pop-up window, and shuts off the spindle when the program reaches this data block and the **Opt Stop** key is On. Refer to Position Data Block, on page 2–3 for information about programming a **PROGRAM STOP** in conjunction with the **Opt Stop** key. Press the flashing **Start Cycle** button to continue with the part program. The comment is cleared when the program resumes.
  - **DISPLAY TIME (SEC)**—indicates the length of time in seconds that the comment appears on the screen. This field is only available when None is selected for the Stop Type.
  - **LINE 1 - 10**—provides 10 lines of text for the Comment block. You can enter up to 50 characters of text on each line. There are no restrictions for using keyboard symbols. Enter text starting with line 1.
Repeat Start Block

Insert Repeat Start and Repeat End data blocks to repeat a block or a group of blocks in
the part program. Each Repeat Start block requires a Repeat End block. To insert a
Repeat Start block from the Program Review screen or from the New Block screen, select
the MISCELLANEOUS ➔ F4 softkey followed by the More ➔ F7 softkey. Select the REPEAT
START F1 softkey.

The REPEAT START screen appears.

This field is available for programming a Repeat Start block:

- **REPEAT COUNT**—enter the number of times the block or blocks between the
  Repeat Start block and the Repeat End block should be repeated.

  The number of repetitions is listed in the Program Review screen for
  the Repeat Start block. Refer to *Program Review, on page 1 - 8* for
  information about the Program Review screen.
Repeat End Block

Each Repeat Start block requires a Repeat End block.

To access the Repeat End Block screen from the Program Review screen or from the New Block screen, select the MISCELLANEOUS ➔ F4 softkey followed by the More ➔ F7 softkey. Select the REPEAT END F2 softkey.

To insert a Repeat End block in an existing part program to follow a Repeat Start block, select the REPEAT END F2 softkey. The REPEAT END screen appears.

Figure 2–164. Repeat End screen
Repeat Start and Repeat End Block Sample

The following Program Review screen contains Repeat Start and Repeat End data blocks (#3 and #5, respectively) programmed around a Drill data block (#4).

![Program Review Screen with Repeat Start and Repeat Stop Blocks](image1)

*Figure 2–165. Program Review Screen with Repeat Start and Repeat Stop Blocks*

Here is the Repeat Start data block, Block 3, with the **REPEAT COUNT** field set at 2. This means the Drill data block that follows this Repeat Start block will drill the hole twice, exactly the same each time.

![Repeat Start Screen](image2)

*Figure 2–166. Repeat Start screen*
This is the Hole Cycle, Drill data block, Block 4, that is programmed to run twice with the Repeat Start block.

![Figure 2–167. Hole Cycle data block to drill twice](image1)

This is the Repeat End data block, Block 5, that must be programmed with the Repeat Start block:

![Figure 2–168. Repeat End screen](image2)
Move Tailstock Data Block

The Move Tailstock Data Block is available for TM12 and TM18 series machines with the auto-hitch tailstock. The data block allows you to move the Z-Axis to the tailstock and automatically hitch the Z-Axis to the tailstock. Then you can move the Z-Axis and tailstock together to a specified location at a programmed feed rate.

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

To access the TM12 and TM18 Series Move Tailstock Data Block screen from the Program Review screen or from the New Block screen, select the MOVE TAILSTOCK F5 softkey. The Move Tailstock screen appears.

![Figure 2–169. Move Tailstock Data Block screen](image)

The fields on the Move Tailstock block screen are:

- **MOVE TO Z POSITION**—identifies the coordinate for the Z-Axis move. When the cursor is in this field, the Store Machine Position F7 softkey is available.
- **FEED (IPM) or (MMPM)**—identifies the feed rate for the move.
Transfer Data Block

The Transfer Data Block allows you to transfer the stock from the Main spindle to the Sub-spindle. This screen is available for TMX MYS turning centers.

To access the TMX MYS Series Transfer Data Block screen from the Program Review screen or from the New Block screen, select the TRANSFER F5 softkey. The Transfer screen appears.

The screen contains two tabs labeled Stage 1 and Stage 2. The fields are numbered to ensure the programming information is entered sequentially.

- The **Stage 1** tab contains programming information for the first series of spindle operations.
- The **Stage 2** tab contains programming information for the second series of spindle operations.

The axes locations are relative to the Work Offsets for the spindle selected in Part Offset. Here is a Transfer block Stage 1 tab for the Main Spindle (selected in Part Setup), as indicated in the upper-right corner of the screen. If the On Sub-spindle checkbox is selected in Part Setup, Sub-spindle appears instead. If the checkbox is not selected, Main Spindle appears.

![Figure 2–170. TMX MYS Transfer Block, Stage 1 tab](image)

The fields on the Transfer block screen Stage 1 tab are:

- **1) Synchronize Spindles**—allows you to change the spindle RPM and specify the sub-spindle offset in degrees. If none of the following fields are selected or edited, the spindles are synchronized at the Main Spindle’s current RPM.
- **S1 Direction**—allows you to choose the direction, CW or CCW, for the Main Spindle to rotate if transferring at RPM.
- **Change Speed**—allows the RPM to be changed. Select this checkbox and the RPM field becomes active.
  - **RPM**—identifies the spindle speed during synchronization. Enter the RPM for the spindles. Enter “0” to stop both spindles prior to synchronization.
- **Offset Spindles**—allows sub-spindle offset to be set. Select this checkbox and the Degrees field becomes active.
  - **Degrees**—identifies the sub-spindle offset in degrees. The range is -360.000 to 360.000
- **2) Open Sub-spindle Chuck Dwell (Secs)**—identifies the number of seconds the sub-spindle chuck should dwell after opening.
- **3) Rapid to W Position**—identifies the W Position for the rapid feedrate move. The STORE MACHINE POSITION F7 softkey is available when the cursor is in this field.
- **4) Feed to W Position**—identifies the W location for the feedrate move. The STORE MACHINE POSITION F7 softkey is available when the cursor is in this field.
  - **Feed (MMPM or IPM)**—identifies the W axis feedrate.
- **5) Close Sub-spindle Chuck Dwell (Secs)**—identifies the number of seconds the sub-spindle chuck should dwell after closing.

At this point, the part is clamped by both the Main spindle and the Sub-spindle. Here is a Transfer block, Stage 2 tab:

![Figure 2–171. TMX MYS Transfer Block, Stage 2 tab](image-url)
The fields on the Transfer block screen Stage 2 tab are:

- **6) Open Main Spindle Chuck**—When No is selected, the main spindle chuck remains closed with the sub-spindle chuck clamping the part. No is the default. When Yes is selected, the main spindle chuck opens to allow the part to transfer to the sub-spindle. When Yes is selected, the remaining fields on this screen are active.

- **Dwell (Secs)**—identifies the number of seconds the main spindle should dwell after opening.

- **7) Move to W Position**—identifies the W location for the feedrate move. STORE MACHINE POSITION F7 softkey is available when the cursor is in this field.

- **Rapid to W Position?**—Select Yes to move the W axis at rapid feedrate to a position specified in the Move to W Position field. Select No and the Feed field becomes active.

- **Feed**—identifies the W axis feedrate. This field is active when Rapid to W Position? is No.

- **8) Close Main Spindle Chuck Dwell (Secs)**—identifies the number of seconds the main spindle chuck should dwell after closing.

- **9) Unsync and Stop the Spindles**—indicates the last step in the Transfer data block sequence.
NC Program Call

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

To include this data block in a Conversational program, from the Program Review screen or from the New Block screen, select the Select the NC PROGRAM CALL F6 softkey.

The NC PROGRAM CALL screen appears with a choice for String Arguments or List Arguments:

![Figure 2–172. NC Program Call screen, String Argument Type](image1)

![Figure 2–173. NC Program Call screen, List Argument Type](image2)
Fields are defined as follows:

- **PROGRAM NUMBER**—initiates the NC program (the program number must not exceed 6999).
- **ARGUMENT TYPE**—allows you to enter argument variables (String or List) to pass to a subprogram for programs that use variables.

See *WinMax Lathe NC Programming, ISNC G Codes, G65 - Subprogram Call, on page 4 - 40 and WinMax Lathe Options, NC Productivity Package Option, on page 8 - 1* for more information about variables and arguments.

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

The following part programming steps are explained in this section:

Create a New Part Program. ................................................................. 3 - 3
Enter Part Setup information. ......................................................... 3 - 6
Enter Tool Setup information. ....................................................... 3 - 7
Enter programming information for Block #1, Profile Start. ............... 3 - 9
Enter programming information for Element #1, Face, in Block #1. ........ 3 - 12
Enter programming information for Element #2, Turn, in Block #1. ........ 3 - 14
Use Verification Graphics to view drawing of part for Block #1. ............ 3 - 16
Enter programming information for Block #2, Profile Start. ................. 3 - 17
Enter programming information for Element #1, Turn, in Block #2. ........ 3 - 20
Enter programming information for Element #2, Face, in Block #2. ........ 3 - 22
Enter programming information for Element #3, Chamfer, in Block #2. .... 3 - 24
Enter programming information for Element #4, Turn, in Block #2. ........ 3 - 26
Enter programming information for Element #5, Blend Arc, in Block #2. .... 3 - 28
Enter programming information for Element #6, Face, in Block #2. ........ 3 - 30
Enter programming information for Element #7, Chamfer, in Block #2. .... 3 - 31
Enter programming information for Element #8, Turn, in Block #2. ........ 3 - 33
Enter programming information for Element #9, Face, in Block #2. ........ 3 - 35
Access the Program Review Screen to review Blocks #1 and #2. .......... 3 - 36
Use Verification Graphics to view drawing of part for Blocks #1 and #2. ..... 3 - 37
Use Graphics to check stock length. .................................................. 3 - 37
Use the Verification Graphics Zoom Feature. ..................................... 3 - 39
Change the Direction of Turning Moves. ......................................... 3 - 40
Use Multiple Block Functions to finish the part with Block #3, Profile Start. .... 3 - 40
Use Verification Graphics to view drawing of part for all Blocks. ............. 3 - 43
The following sample part program produces this part:

![Sample Part](image1)

*Figure 3–1. Sample Part*

Here is a print containing rough dimensions for this sample part program:

![Print for Sample Part Program](image2)

*Figure 3–2. Print for Sample Part Program*
Step 1: Create a New Part Program.

From the Input screen, select PROJECT MANAGER F8.

![Figure 3–3. Input Screen Project Manager F8 softkey](image1.png)

From the Project Manager screen, select NEW F1.

![Figure 3–4. Project Manager Screen with list of Program Names](image2.png)
The softkeys change. Select CREATE HLT PROGRAM F1.

The new program name is added as NONAME1.HLT to the list of programs appearing under the Program Name heading. If there is currently a new file listed, the control increments the number in the file name by 1.
From the Project Manager screen with NONAME2.HLT listed, select SAVE F3. The screen changes to show folders on the left and files on the right. The current directory (folder) is highlighted and the FILE NAME field is highlighted.

From here you can select the current folder for saving the file, select another folder, or create a new folder by selecting the CREATE FOLDER F4 softkey and entering a new folder name in the pop-up window that appears. Once the folder is selected, make sure the FILE NAME field is highlighted and enter a file name. For this sample, the folder is “Sample,” and the file name is “SamplePP.HLT.”

Select SAVE F1 to save the part program file.

![Figure 3–7. New Part Program Saved with File Name](image)
The Project Manager screen appears with the new file name listed under the Program Name heading. The file name also appears in the status bar to show that it is open and loaded into the control’s memory.

Select the **Input** console key to return to the Input screen.

**Step 2: Enter Part Setup information.**

From the Input screen, select PART SETUP F1 to enter Part Setup information.

---

**Figure 3–8. Project Manager Screen with “Sample PP.HLT” file listed**

**Figure 3–9. Input Screen with “Sample PP.HLT” file loaded**
In addition to the part programming requirements, Verification Graphics requires an Offset Value. Enter this value into the Z Offset #1 field, which is the Active Offset for this example.

**Figure 3–10. Part Setup Screen with value for Offset 1**

Select EXIT F8 to return to the Input screen.

**Step 3: Enter Tool Setup information.**

From the Input screen, select TOOL SETUP F2 to enter Tool Setup information.

**Figure 3–11. Tool Setup Screen**
Enter the Tool Number in the New Tool field. The Tool Number corresponds with the turret position where the tool is located. The following fields appear on the Tool Setup screen.

![Figure 3–12. Tool Setup Screen with Tool Number 1](image)

Fill in the necessary detail, including Insert Shape and Size.

![Figure 3–13. Tool Setup Screen](image)

Select EXIT F8 to return to the Input screen.
Step 4: Enter Part Programming (Data Block and Elements) information.

From the Input screen, select PART PROGRAMMING F3 to enter information for the first data block and elements. The New Block screen appears with data block softkey selections.

![New Block Screen](image)

**Figure 3–14. New Block Screen**

Step 5: Enter programming information for Block #1, Profile Start.

First, face off the end of the material (i.e., cut off the edge and make it smooth). Select Turning F2, PROFILE F1 for this type of turning. The Profile Start screen appears.

![Profile Start Screen Process Tab before tool selection](image)

**Figure 3–15. Profile Start Screen Process Tab before tool selection**
Enter a 1 in the Tool field to load the information from Tool Setup for Tool #1 into the appropriate fields in the Profile Start screen.

Select the Geometry tab. These fields appear:

Figure 3–16. Profile Start Screen Process Tab with Tool #1

Figure 3–17. Profile Start Screen Geometry Tab
Enter the rapid position and Profile start point in the X Rapid (DIA), Z Rapid, X Start (DIA), and Z Start fields. In addition enter any stock allowance information.

![Figure 3–18. Profile Start Screen Geometry Tab with rapid position and start point](image)

The following figure illustrates the locations of these programmed fields:

![Figure 3–19. Profile Start Point and Rapid Position](image)
Step 6: Enter programming information for Element #1, Face, in Block #1.

Profiles contain Elements that describe the profile shape. Move the cursor to the Element field (below the Block field) and select the NEXT ELEMENT F2 softkey. The New Profile Element screen appears with softkey selections for the types of Elements.

![New Profile Element Screen](image)

*Figure 3–20. New Profile Element Screen*

Select FACE F1 to program the first move to cut down the face of the material.

![Profile Face Element Screen](image)

*Figure 3–21. Profile Face Element Screen*
Enter an X End (D) value (-0.0500) that will cause the tool to cut past the centerline of the stock.

- The control calculates and enters values for **Length** and **Angle** based on the X End value.
- The **X Start (D)** and **Z Start** values are carried over from the previous element’s X END (D) and Z END values.
- The **Finish** fields are inactive because Turn Rough is selected in the Profile Start Process tab.

*Figure 3–22. Profile Face Screen with X End value*
Step 7: Enter programming information for Element #2, Turn, in Block #1.

Move the cursor to the Element field again (below the Block field) and select the NEXT ELEMENT F2 softkey. The New Profile Element screen appears with softkey selections for the types of Elements.

Select TURN F3 to program the next move to turn away from the face of the material.

![Figure 3–23. Profile Turn Element Screen](image-url)

Since the first Element and the last Element must be perpendicular, start with a Face and end with a Turn.
The control calculates the amount of material for the face based on the length of the Turn (Z End).

- The control calculates and enters values for **Length** and **Angle** based on the Z End value.
- The **X Start (D)** and **Z Start** values are carried over from the previous element’s X END (D) and Z END values.
- The **Finish** fields are inactive because Turn Rough is selected in the Profile Start Process tab.

*Figure 3–24. Profile Turn Element Screen with Z End value*
Step 8: Use Verification Graphics to view drawing of part for Block #1.

Select the Verify console key to access Verification Graphics. Select DRAWING F1, followed by DRAW F1. The following 3-D figure shows the tool path and stock after being cut using the data from Block 1:

![3-D Sample Program Block 1](image)

*Figure 3–25. 3-D Sample Program Block 1*
Step 9: Enter programming information for Block #2, Profile Start.

Select the Input console key to return to the Input screen.

From the Input screen, select PART PROGRAMMING F3 to enter information for the next data block and elements. Data block #1 appears on the screen with the cursor in the BLOCK field.

Select NEXT BLOCK F2 and the NEW BLOCK screen appears.

Select TURNING F2, PROFILE F1 to access the PROFILE START screen for creating a second profile for the OD (outside diameter) turn.

![Profile Start Screen Process Tab before tool selection](image)

*Figure 3–26. Profile Start Screen Process Tab before tool selection*
Enter a 1 in the Tool field to load the information from Tool Setup for Tool #1 into the appropriate fields in the Profile Start screen.

![Profile Start Screen Process Tab with Tool #1](image1.png)

**Figure 3–27. Profile Start Screen Process Tab with Tool #1**

Select the Geometry tab. These fields appear:

![Profile Start Screen Geometry Tab](image2.png)

**Figure 3–28. Profile Start Screen Geometry Tab**
Enter the rapid position and Profile start point in the X Rapid (DIA), Z Rapid, X Start (DIA), and Z Start fields. In addition enter any stock allowance information.

Figure 3–29. Profile Start Screen Geometry Tab with rapid position and start point

The following figure illustrates the locations of these programmed fields.

Figure 3–30. Profile start point and Rapid Position
Step 10: Enter programming information for Element #1, Turn, in Block #2.

Select the NEXT ELEMENT F2 softkey and the New Profile Element screen appears with softkey selections for the types of Elements. Select TURN F3 to program the first move to turn to the face of the material.

![Profile Turn Element Screen](image)

Since the first Element and the last Element must be perpendicular, start with a Turn and end with a Face.
Enter 0 in the Z End field.

- The control calculates and enters values for **Length** and **Angle** based on the Z End value.
- The **X Start (D)** and **Z Start** values are carried over from the previous element’s X END (D) and Z END values.
- The **Finish** fields are inactive because Turn Rough is selected in the Profile Start Process tab.

*Figure 3–32. Profile Turn Element Screen with Z End value*
Step 11: Enter programming information for Element #2, Face, in Block #2.

Select the NEXT ELEMENT F2 softkey and the New Profile Element screen appears with softkey selections for the types of Elements. Select FACE F1 to face up to the first diameter location.

Figure 3–33. Profile Face Element Screen
Enter 1 in the X End (D) field.

- The control calculates and enters values for **Length** and **Angle** based on the Z End value.
- The **X Start (D)** and **Z Start** values are carried over from the previous element’s X END (D) and Z END values.
- The **Finish** fields are inactive because Turn Rough is selected in the Profile Start Process tab.

*Figure 3–34. Profile Face Element Screen with X End (D) value*
Step 12:  Enter programming information for Element #3, Chamfer, in Block #2.

Select the NEXT ELEMENT F2 softkey and the New Profile Element screen appears with softkey selections for the types of Elements. Select CHAMFER F6 to chamfer the corner of the first diameter location.

*Figure 3–35. Profile Chamfer Element Screen*
Enter 0.025 in the Chamfer field (the length along the Z axis). The Angle field defaults to 45. Keep this value.

- The **End** and **Start** locations will be calculated by the control when the Element following the Chamfer has been programmed.
- The **Finish** fields are inactive because Turn Rough is selected in the Profile Start Process tab.

*Figure 3–36. Profile Chamfer Element Screen with Chamfer and Angle values*
Step 13: Enter programming information for Element #4, Turn, in Block #2.

Select the NEXT ELEMENT F2 softkey and the New Profile Element screen appears with softkey selections for the types of Elements. Select TURN F3 to turn the length to the face.

Notice that TURN is the only active softkey at this point. Chamfers can be used to join a Face Element and a Turn Element. Since the first Element was a Face, Turn is the only option.

![Profile Turn Element Screen](image-url)
Enter -1 in the Z End field.

- The control calculates and enters values for **Length** and **Angle** based on the Z End value.
- The **X Start (D)** and **Z Start** values are carried over from the previous element’s X END (D) and Z END values.
- The **Finish** fields are inactive because Turn Rough is selected in the Profile Start Process tab.

![Figure 3–38. Profile Turn Element Screen with Z End value](image)

- The control calculates and enters values for X End (D), Z END, X Start (D), and Z Start on the Chamfer Element Screen. Select PREVIOUS ELEMENT F1 to view the Chamfer Element Screen.

![Figure 3–39. Profile Chamfer Element 3 Screen with control-calculated values](image)
Select NEXT ELEMENT F2 to return to the Element #4, Turn, Screen.

**Step 14:** Enter programming information for Element #5, Blend Arc, in Block #2.

Select the NEXT ELEMENT F2 softkey and the New Profile Element screen appears with softkey selections for the types of Elements. Select BLEND ARC F5 to add a corner arc.

*Figure 3–40. Profile Blend Arc Screen*
Enter 0.1 for the Radius of the arc.

- The **End**, **Start**, and **Center** locations will be calculated by the control when the Element following the Blend Arc has been programmed.
- The **Finish** fields are inactive because Turn Rough is selected in the Profile Start Process tab.

*Figure 3–41. Profile Blend Arc Screen with Radius value*
Step 15: Enter programming information for Element #6, Face, in Block #2.

Select the NEXT ELEMENT F2 softkey and the New Profile Element screen appears with softkey selections for the types of Elements. Select FACE F1 to face to the next diameter location.

![Profile Face Element Screen](image)

**Figure 3–42. Profile Face Element Screen**

Enter 2 in the X End (D) field.

- The control calculates and enters values for **Length** and **Angle** based on the X End value.
- The **X Start (D)** and **Z Start** values are carried over from the previous element’s X END (D) and Z END values.
- The **Finish** fields are inactive because Turn Rough is selected in the Profile Start Process tab.

![Profile Face Element Screen with X End (D) value](image)

**Figure 3–43. Profile Face Element Screen with X End (D) value**
• The control calculates and enters values for X End (D), Z END, X Start (D), Z Start, X Center (D), and Z Center on the Blend Arc Element Screen. Select PREVIOUS ELEMENT F1 to view the Blend Arc Element Screen.

Figure 3–44. Profile Blend Arc Element with control-calculated values

Select NEXT ELEMENT F2 to return to the Element #6, Face, Screen.

Step 16:  Enter programming information for Element #7, Chamfer, in Block #2.

Select the NEXT ELEMENT F2 softkey and the New Profile Element screen appears with softkey selections for the types of Elements. Select Chamfer F6 to chamfer the next corner.

Figure 3–45. Profile Chamfer Element Screen
Enter 0.025 in the Chamfer field (the length along the Z axis). The Angle field defaults to 45. Keep this value.

- The **End** and **Start** locations will be calculated by the control when the Element following the Chamfer has been programmed.
- The **Finish** fields are inactive because Turn Rough is selected in the Profile Start Process tab.

![Figure 3–46. Profile Chamfer Element Screen with Chamfer value](image)
Step 17:  Enter programming information for Element #8, Turn, in Block #2.

Select the NEXT ELEMENT F2 softkey and the New Profile Element screen appears with softkey selections for the types of Elements. Select TURN F3 to finish out the chamfer.

Notice that TURN is the only active softkey at this point. Chamfers can be used to join a Face Element and a Turn Element. Since the first Element was a Face, Turn is the only option.

![Profile Turn Element Screen](image)

Figure 3–47. Profile Turn Element Screen
Enter -1.1 in the Z End field to turn a small amount past the edge of the part.

- The control calculates and enters values for Length and Angle based on the Z End value.
- The X Start (D) and Z Start values are carried over from the previous element’s X END (D) and Z END values.
- The Finish fields are inactive because Turn Rough is selected in the Profile Start Process tab.

![Profile Turn Element Screen with Z End value](image1)

**Figure 3–48. Profile Turn Element Screen with Z End value**

- The control calculates and enters values for X End (D), Z END, X Start (D), and Z Start on the Chamfer Element Screen. Select PREVIOUS ELEMENT F1 to view the Chamfer Element Screen.

![Profile Chamfer Element Screen with control-calculated values](image2)

**Figure 3–49. Profile Chamfer Element Screen with control-calculated values**

Select NEXT ELEMENT F2 to return to the Element #8 Screen.
Step 18:  Enter programming information for Element #9, Face, in Block #2.

Select the NEXT ELEMENT F2 softkey and the New Profile Element screen appears with softkey selections for the types of Elements. Select FACE F1 to move off the part and to finish perpendicular to Element #1.

![Profile Face Element Screen](image1.png)

**Figure 3–50. Profile Face Element Screen**

Enter 2.05 to face away from the part.

- The control calculates and enters values for **Length** and **Angle** based on the Z End value.
- The **X Start (D)** and **Z Start** values are carried over from the previous element’s X END (D) and Z END values.
- The **Finish** fields are inactive because Turn Rough is selected in the Profile Start Process tab.

![Profile Face Element Screen with X End (D) value](image2.png)

**Figure 3–51. Profile Face Element Screen with X End (D) value**
Step 19: Access the Program Review Screen to review Blocks#1 and #2.

Select the Review console key to view the Program Review Screen. The data blocks are listed on the left side and the elements are listed on the right side of the screen. Highlight the block on the left to view its elements on the right.

![Figure 3–52. Program Review Screen Data Block #1](image)

![Figure 3–53. Program Review Screen Data Block #2](image)
Step 20: Use Verification Graphics to view drawing of part for Blocks #1 and #2.

Select the Verify console key to view the tool path and stock with Verification Graphics.

The following 3-D figure shows the tool path and stock after being cut using the data from Blocks 1 and 2:

![3-D Verification Graphics Sample Program Blocks 1 and 2](image)

1. The vertical red line represents Part Zero
2. The horizontal pink line represents the stock centerline.

*Figure 3–54. 3-D Verification Graphics Sample Program Blocks 1 and 2*

Step 21: Use Graphics to check stock length.

Notice in the graphic above that the stock is not as long as the part needs to be. To correct this, from the graphics screen, select the SETUP F7 softkey followed by the STOCK SETUP F1 softkey, shown on the sample screen above. The Stock Setup screen appears.

![Stock Setup Screen](image)

*Figure 3–55. Stock Setup Screen*
Change the values in the Stock Setup screen until the stock graphically covers Part Zero, which is represented by the red circle in front of the part on the graphics screen.

![Stock Setup Screen with new value in Stock Length field](image)

*Figure 3–56. Stock Setup Screen with new value in Stock Length field*

Select the EXIT F8 softkey twice to return to the graphics screen. Select DRAWING ➔ F1 followed by Draw F1 to graphically cut the part.
Step 22: Use the Verification Graphics Zoom Feature

In Verification Graphics, you can Zoom in closer on the view.

Select the ZOOM F3 softkey.

Select the ZOOM IN F1 softkey, and repeat as necessary.

Or select one corner of a zoom box location on the screen. Then select a second corner to define the opposite corner of the area that you want to zoom.

*Figure 3–57. 3-D Verification Graphics Zoom In*
Step 23:  Change the Direction of Turning Moves

In this sample part program, the facing moves are in the Z- Direction. To change the cutting direction, access the Block from the Program Review screen.

Select the Review console key to access to the Program Review screen.

Select Block #2. The Profile Start screen Process tab appears on the screen.

Change the DIRECTION field to X- using either the drop-down list or the X- F2 softkey.

Step 24:  Use Multiple Block Functions to finish the part with Block #3, Profile Start

Block #2 used Turn Rough for the Turning Operation. Block #3 will use Turn Finish for the Turning Operation. Copy Block #2 using the Program Review screen and MULTIPLE BLOCK FUNCTIONS F1 softkey to create Block #3.

Select the Review console key to access the Program Review screen.

![Program Review screen with Blocks #1 and #2](image)

Select Block #2.

Select the MULTIPLE BLOCK FUNCTIONS F1 softkey.

Select the CREATE FINISH PROFILE F7 softkey.
Block #3 appears in the Program Review screen.

![Program Review screen with Blocks #1, #2, and #3](image)

*Figure 3–59. Program Review screen with Blocks #1, #2, and #3*

Select Block #3 and press the Enter console key. The Profile Start screen, Process tab for Block #3 appears.

The control automatically changes the Process tab OPERATION field to TURN FINISH.

![Profile Start screen Process tab Turn Finish Operation](image)

*Figure 3–60. Profile Start screen Process tab Turn Finish Operation*
In addition, the Geometry tab’s Stock, Turn, and Face Allowance fields have been changed automatically to 0.000.

*Figure 3–61. Profile Start screen Geometry tab Turn Finish Operation*
Step 25: Use Verification Graphics to view drawing of part for all Blocks.

Select the Verify console key to view the tool path and stock with Verification Graphics.

The following 3-D figure shows the tool path and stock after being cut using the data from Blocks 1, 2, and 3:

Figure 3–62. 3-D Sample Program Blocks 1, 2, and 3
RECORD OF CHANGES

v546CO, January 2018

Revised by: H. Arle

Approved by: D. Skrzypczak

<table>
<thead>
<tr>
<th>Changes</th>
</tr>
</thead>
<tbody>
<tr>
<td>v546CO: Updated based on software changes.</td>
</tr>
</tbody>
</table>

704-0115-211, May 2015

Revised by: K. Gross

Approved by: D. Skrzypczak

<table>
<thead>
<tr>
<th>Changes</th>
</tr>
</thead>
<tbody>
<tr>
<td>704-0115-211: Updates based on software updates through v09.02.160.08. Updated Radial and Axial Lettering data blocks.</td>
</tr>
</tbody>
</table>

704-0115-210, May 2013

Revised by: K. Gross

Approved by: D. Skrzypczak May 2013

<table>
<thead>
<tr>
<th>Changes</th>
</tr>
</thead>
<tbody>
<tr>
<td>704-0115-210: Updates based on software updates through v09.02.34. Revised Finish Tool Cutting Side default values based on Roughing Direction tables. Added operation description for the Move Tailstock data block when either Interrupt or Stop Cycle is pressed.</td>
</tr>
</tbody>
</table>

704-0115-209, January 2013

Revised by: K. Gross

Approved by: D. Skrzypczak January 2013

<table>
<thead>
<tr>
<th>Changes</th>
</tr>
</thead>
<tbody>
<tr>
<td>704-0115-209: Updates based on software updates through v09.02.11. Revised order of Tool and Tool Offsets fields on Cutoff Block screen.</td>
</tr>
</tbody>
</table>

704-0115-208, October 2012

Revised by: K. Gross

Approved by: D. Skrzypczak, K.Van Blaircum October 2012
### Changes

**704-0115-208:** Updates based on software updates through v09.00.43.

704-0115-207, March 2012

  Revised by: K. Gross
  
  Approved by: D. Skrzypczak March 2012

### Changes

**704-0115-207:** Updates based on software updates through v8.1.3.48.

704-0115-206, December 2011

  Revised by: K. Gross
  
  Approved by: D. Skrzypczak, J. Mulkey, G.Traicoff, K.Van Blaircum December 2011

### Changes

**704-0115-206:** Updates based on software updates through v8.1.2.30.

  Added Program Coordinates section in Live Tooling data blocks section.
  
  Added Move Tailstock and Transfer data blocks.
  
  Set cross-references between books.
  
  Added ScreenPath condition and paragraph tag.

704-0115-205, April 2011

  Revised by: K. Gross
  
  Approved by: D. Skrzypczak, J. Mulkey, G.Traicoff, K.Van Blaircum April 2011

### Changes

- **704-0115-205** Updates based on software updates through v8.1.2.18.

704-0115-204, June 2010, ECN 16538

  Revised by: K. Gross
  
  Approved by: D. Skrzypczak, J. Mulkey, G.Traicoff, K.Van Blaircum June 2010

### Changes

- **704-0115-204** Updates based on software updates through v8.0.
<table>
<thead>
<tr>
<th>Changes</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>704-0115-203</strong> rE Updates based on software updates through v2.02.04. Revised Radial and Axial screen</td>
</tr>
<tr>
<td>captures and field definitions to match locations axis orientation and I Plane field and softkey label.</td>
</tr>
<tr>
<td>March 2009 ECN 16597.</td>
</tr>
<tr>
<td><strong>704-0115-203</strong> rC Updates based on software updates through v2.02.02. Added information to front matter</td>
</tr>
<tr>
<td>about On-screen Help and accessing the On-screen Help in PDF format, November 2008.</td>
</tr>
<tr>
<td><strong>704-0115-203</strong> rB Updates based on software updates through v2.02.01, 27 June 2008, ECN 16508.</td>
</tr>
<tr>
<td><strong>704-0115-203</strong> rA Updates based on software updates through v2.02.00, 19 June 2008, prepared for ECN</td>
</tr>
<tr>
<td>16508; Internal release.</td>
</tr>
</tbody>
</table>

704-0115-202, October 2007, ECN 16430

<table>
<thead>
<tr>
<th>Changes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Updates based on v1.1, v1.2, and v2.0 software and the introduction of the Live Tooling (TMM Series) machine.</td>
</tr>
</tbody>
</table>

704-0115-201, 19 May 2005, ECN 15866

<table>
<thead>
<tr>
<th>Changes</th>
</tr>
</thead>
<tbody>
<tr>
<td>New manual release.</td>
</tr>
</tbody>
</table>
INDEX

Numerics

1) Synchronize Spindles field
   Transfer block screen 2 - 207

1ST PECK OFFSET field
   Axial Mill Circle screen 2 - 149
   Axial Mill Contour screen 2 - 130
   Axial Mill Frame screen 2 - 156
   Radial Mill Circle screen 2 - 92
   Radial Mill Contour screen 2 - 73
   Radial Mill Frame screen 2 - 98

2) Open Sub-spindle Chuck Dwell (Secs) field
   Transfer block screen 2 - 208

3) Rapid to W Position field
   Transfer block screen 2 - 208

4) Feed to W Position field
   Transfer block screen 2 - 208

5) Close Sub-spindle Chuck Dwell (Secs) field
   Transfer block screen 2 - 208

6) Open Main Spindle Chuck field
   Transfer block screen 2 - 209

7) Move to W Position field
   Transfer block screen 2 - 209

8) Close Main Spindle Chuck Dwell (Secs) field
   Transfer block screen 2 - 209

A

ACTIVE OFFSET field
   Change Part Setup 2 - 196

ADD LOCATION F5 softkey
   Axial Holes Locations screen 2 - 182
   Radial Holes Locations screen 2 - 121

ADDITIONAL AXIS OFFSETS F3 softkey
   Change Part Setup 2 - 197

ADVANCE AT X (DIA) or (RAD)
   Cutoff block 2 - 63

ADVANCE PART CATCHER
   Cutoff block 2 - 63

Advanced Pecking tab, Cutoff Block screen 2 - 65

ALL FACES FIRST turn finish strategy 2 - 18

ALONG PROFILE turn finish strategy 2 - 18

angle 2 - 48
   infeed angle ID Straight 2 - 46
   infeed Face 2 - 49
   infeed ID Taper 2 - 48
   infeed OD Straight 2 - 42

pullout Face 2 - 50
pullout ID Straight 2 - 46
pullout OD Straight 2 - 42
unknown 2 - 82, 2 - 139

ANGLE field
   Axial Mill Contour Line Segment screen
      Rectangular Coordinates 2 - 137
   Chamfer Element screen 2 - 37
   Face Element screen 2 - 29
   Radial Mill Contour Line Segment screen
      Linear Coordinates 2 - 80
   Taper Element screen 2 - 31
   Turn Element screen 2 - 28

APPEND ARC field
   Axial Mill Slot screen
      Polar Coordinates 2 - 164
      Rectangular Coordinates 2 - 166
   Radial Slot screen
      Linear Coordinates 2 - 108
      Rotary Coordinates 2 - 106

APPROACH ANGLE field
   Profile Start screen 2 - 19

APPROACH CLEARANCE field
   Profile Start screen 2 - 16, 2 - 19

Arc 2 - 26
   clockwise 2 - 85, 2 - 142
   element 2 - 32
   segment 2 - 140

ARC F2 softkey
   Axial Mill Contour Segment screen 2 - 140
   New Radial Contour Segment screen 2 - 78
   Radial Mill Contour Arc Segment screen 2 - 83

ARC F4 softkey, element 2 - 26

ARGUMENT TYPE field, NC Program Call screen 2 - 211

arguments, NC Program Call screen 2 - 210

AT END (DIA) or (RAD) field
   Axial Mill Slot screen
      Polar Coordinates 2 - 165
      Rectangular Coordinates 2 - 167
   Radial Slot screen
      Linear Coordinates 2 - 109
      Rotary Coordinates 2 - 107

AT START (DIA) or (RAD) field
   Axial Mill Slot screen
      Polar Coordinates 2 - 165
      Rectangular Coordinates 2 - 167
   Radial Slot screen
      Linear Coordinates 2 - 109
      Rotary Coordinates 2 - 107

AT-keyboard option 1 - 7

automatic calculations 1 - 6, 1 - 8
axes, move to the defined position 2 - 4
AXIAL BOLT CIRCLE F3 softkey
   New Axial Holes screen 2 - 178
Axial Circle
   plunge points 2 - 150
AXIAL CIRCLE F2 softkey
   New Block Axial Milling screen 2 - 125
AXIAL FLATS F5 softkey
   New Block Axial Milling screen 2 - 125
Axial Frame
   plunge points 2 - 157
AXIAL FRAME F3 softkey
   New Block Axial Milling screen 2 - 125
AXIAL Hole CYCLE F1 softkey
   New Axial Holes screen 2 - 178
Axial Holes description 1 - 4
AXIAL HOLES F5 softkey
   New Block Live Tooling screen 2 - 178
AXIAL LETTERING F6 softkey
   New Block Axial Milling screen 2 - 125
AXIAL LINES AND ARCS F1 softkey
   New Block Axial Milling screen 2 - 125
AXIAL LOCATIONS F2 softkey
   New Axial Holes screen 2 - 178
Axial milling
   Linear Y motion 2 - 125
   programming coordinates 2 - 125
Axial Milling description 1 - 4
AXIAL MILLING F4 softkey
   New Block Live Tooling screen 2 - 125
AXIAL SLOT F4 softkey
   New Block Axial Milling screen 2 - 125
AXIS ORDER field
   Position Block screen 2 - 6
   Tool Change Override area in Position Block screen 2 - 8
   User Defined Tool Change Position area in Program Parameters screen 2 - 194
AXIS ORDER X 2 - 6
AXIS ORDER Z 2 - 6

B
Bar Feed description 1 - 4
BAR FEED F4 softkey
   TMM, TMX MY, and TMX MYS Series 2 - 200
bar pulling 2 - 200
bar pushing 2 - 200
Blend Arc 2 - 26
element 2 - 34
guidelines 2 - 88, 2 - 146
programming hint 2 - 89, 2 - 146
segment 2 - 86
types 2 - 88, 2 - 145
BLEND ARC F3 softkey
   Axial Mill Contour segment screen 2 - 143
   New Radial Contour Segment screen 2 - 78, 2 - 86
BLEND ARC F5 softkey
   element 2 - 26
Blend Offset
   blend-in move
      Axial Mill Circle screen 2 - 149
      Axial Mill Contour screen 2 - 130
      Axial Mill Frame screen 2 - 156
      Radial Mill Circle screen 2 - 91
      Radial Mill Contour screen 2 - 73
      Radial Mill Frame screen 2 - 98
   blend-out move
      Axial Mill Circle screen 2 - 149
      Axial Mill Contour screen 2 - 130
      Axial Mill Frame screen 2 - 156
      Radial Mill Circle screen 2 - 91
      Radial Mill Contour screen 2 - 73
      Radial Mill Frame screen 2 - 98
BLEND OFFSET field
   Axial Mill Circle screen 2 - 149
   Axial Mill Contour screen 2 - 130
   Axial Mill Frame screen 2 - 156
   Radial Mill Circle screen 2 - 91
   Radial Mill Contour screen 2 - 73
   Radial Mill Frame screen 2 - 98
BLEND OVERLAP field
   Axial Mill Circle screen 2 - 149
   Axial Mill Contour screen 2 - 130
   Axial Mill Frame screen 2 - 156
   Radial Mill Circle screen 2 - 92
   Radial Mill Contour screen 2 - 73
   Radial Mill Frame screen 2 - 98

C
C (DEG) field 2 - 6
   Axial Locations screen
      Polar Coordinates 2 - 183
      Radial Holes Locations screen
      Rotary Coordinates 2 - 122
C ANGLE (DEG) field
   Axial Mill Contour screen
      Polar Coordinates 2 - 133
      Radial Mill Contour screen
      Rotary Coordinates 2 - 76
C AXIS ORDER field
   Position block 2 - 6
C CENTER (DEG) field
   Axial Bolt Circle screen
      Polar Coordinates 2 - 185
Axial Mill Circle screen
   Polar Coordinates 2 - 151
Axial Mill Contour Arc Segment screen
   Polar Coordinates 2 - 140
Axial Mill Contour Blend Arc Segment screen
   Polar Coordinates 2 - 144
Radial Mill Circle screen
   Rotary Coordinates 2 - 93
Radial Mill Contour Arc Segment screen
   Rotary Coordinates 2 - 83
Radial Mill Contour Blend Arc Segment screen
   Rotary Coordinates 2 - 87
C CORNER (DEG) field
   Axial Mill Frame screen
      Polar Coordinates 2 - 158
   Radial Mill Frame screen
      Rotary Coordinates 2 - 100
C END (DEG) field
   Axial Mill Contour Arc Segment screen
      Polar Coordinates 2 - 140
   Axial Mill Contour Blend Arc Segment Screen
      Polar Coordinates 2 - 143
   Axial Mill Contour Line Segment screen
      Polar Coordinates 2 - 136
   Axial Mill Slot screen
      Polar Coordinates 2 - 163
   Radial Mill Contour Arc Segment screen
      Rotary Coordinates 2 - 83
   Radial Mill Contour Blend Arc Segment screen
      Rotary Coordinates 2 - 86
   Radial Mill Contour Line Segment screen
      Rotary Coordinates 2 - 79
   Radial Slot screen
      Rotary Coordinates 2 - 105
C LENGTH (DEG) field
   Axial Mill Frame screen
      Polar Coordinates 2 - 158
   Radial Mill Frame screen
      Rotary Coordinates 2 - 100
C OFFSET checkbox, Change Part Setup screen 2 - 195
C OFFSET field
   Change Part Setup 2 - 195
C POSITION field
   Axial Lettering screen 2 - 176
   Radial Lettering screen 2 - 111
   Radial Mill Circle screen
      Linear Coordinates 2 - 94
   Radial Mill Contour screen
      Linear Coordinates 2 - 77, 2 - 101, 2 - 107, 2 - 122, 2 - 134, 2 - 152, 2 - 159, 2 - 165, 2 - 173
C START (DEG) field
   Axial Mill Contour Arc Segment screen
      Polar Coordinates 2 - 140
   Axial Mill Contour Blend Arc Segment screen
      Polar Coordinates 2 - 143
   Axial Mill Flats screen
      Polar Coordinates 2 - 172
   Axial Mill Slot screen
      Polar Coordinates 2 - 163
   Radial Mill Contour Arc Segment screen
      Rotary Coordinates 2 - 83
   Radial Mill Contour Blend Arc Segment screen
      Rotary Coordinates 2 - 86
   Radial Mill Contour Line Segment screen
      Rotary Coordinates 2 - 79
   Radial Slot screen
      Rotary Coordinates 2 - 105
CAL 1 - 6
Calc-Assist feature 2 - 25
   calculated plunge points
      axial circles 2 - 150
      axial frames 2 - 157
      radial circles 2 - 92
      radial frames 2 - 99
Calculation (automatic) 1 - 6
   arcs 2 - 85, 2 - 142
   lines 2 - 81, 2 - 138
   two possible solutions 2 - 81, 2 - 138
calculator 1 - 6
   calculator, on-screen viii
Center Drill 2 - 58
center drill cycle 2 - 59
CENTER DRILL field
   Axial Holes Drill Operation screen 2 - 180
   Radial Holes Drill Operation screen 2 - 118
Chamfer 2 - 26
CHAMFER ANGLE field
   Grooving 2 - 56
   Chamfer Element 2 - 36
   CHAMFER F6 softkey, element 2 - 26
   CHAMFER field
      Chamfer Element screen 2 - 37
   CHAMFER LENGTH field
      Grooving 2 - 56
      Change Parameters description 1 - 4
      CHANGE PARAMETERS F1 softkey 2 - 192
      Change Part Setup description 1 - 4
      CHANGE PART SETUP F2 softkey 2 - 195
      Change Speed field
Transfer block screen 2 - 208
chip breaker 2 - 59, 2 - 63
CHIP BREAKER field
   Axial Holes Drill Operation screen 2 - 180
   Radial Holes Drill Operation screen 2 - 118
Chip Breaker peck type, Cutoff 2 - 63
CLEAR ENTIRE SKIP LIST F4 softkey
   Axial Bolt Circle screen 2 - 187
   Radial Holes Loop Around C-Axis screen 2 - 124
CLEAR SKIP LIST ENTRY F3 softkey
   Axial Bolt Circle screen 2 - 187
   Radial Holes Loop Around C-Axis screen 2 - 124
CLEAR TABLE F7 softkey
   Part Setup - Work Offsets screen 2 - 197
Climb Milling 2 - 190
   clockwise arc 2 - 85, 2 - 142
Comment Block
   screen 2 - 201
text 2 - 201
Comment description 1 - 4
COMMENT F5 softkey 2 - 201
Console keys
   use with Program Review screen 1 - 9
CONSTANT VOLUME field
   Threading 2 - 39
   control
   power on - viii
Conventional Milling 2 - 190
Conversational Merge (NC Program Cal) 2 - 202
Conversational Programming 2 - 1
   Calculator 1 - 6
CONVERT TO LOCATIONS BLOCK F5 softkey
   Axial Bolt Circle screen 2 - 187
   Radial Holes Loop Around C-Axis screen 2 - 124
COORDINATES field
   Axial Bolt Circle screen 2 - 185
   Axial Locations screen 2 - 182
   Axial Mill circle screen 2 - 151
   Axial Mill Contour screen 2 - 133
   Axial Mill Flats screen 2 - 171
   Axial Mill Frame screen 2 - 158
   Axial Mill Slot screen 2 - 162
   Radial Holes Locations screen 2 - 121
   Radial Mill Contour screen 2 - 76, 2 - 93
   Radial Mill Frame screen 2 - 100, 2 - 105

COPY F2 softkey 1 - 10
COPY FEED TO ALL ELEMENTS F6 softkey
   Profile Element screens 2 - 29, 2 - 30, 2 - 31, 2 - 33, 2 - 35, 2 - 37
   Profile Start screen 2 - 12
COPY OFFSETS TO ALL ELEMENTS F6 softkey
   Profile Element screens 2 - 29, 2 - 30, 2 - 31, 2 - 33, 2 - 35, 2 - 37
   Profile Start screen 2 - 12
COPY SPEED TO ALL ELEMENTS F6 softkey
   Profile Element screens 2 - 29, 2 - 30, 2 - 31, 2 - 33, 2 - 35, 2 - 37
   Profile Start screen 2 - 12
COPY TO LOCATIONS BLOCK F6 softkey
   Axial Bolt Circle screen 2 - 187
   Radial Holes Loop Around C-Axis screen 2 - 124
CORNER RADIUS field
   Axial Mill Flats screen
   Polar Coordinates 2 - 172
   Rectangular Coordinates 2 - 174
   Axial Mill Frame screen
   Polar Coordinates 2 - 158
   Rectangular Coordinates 2 - 159
   Axial Mill Slot screen
   Polar Coordinates 2 - 164, 2 - 165
   Rectangular Coordinates 2 - 166
   Radial Mill Frame screen
   Rotary Coordinates 2 - 100, 2 - 102
   Radial Slot screen
   Linear Coordinates 2 - 109
   Rotary Coordinates 2 - 106, 2 - 107
CORNER TYPE field
   Grooving 2 - 56
CORNERS field
   Grooving 2 - 56
   create a
data block 2 - 2
   Element in the Profile 2 - 26
   Finish Profile 1 - 11
   Position block 2 - 3
   cursor control keys 1 - 7
CUT CLEARANCE field
   Grooving 2 - 53
   Profile Start screen 2 - 20, 2 - 22
CUT F1 softkey 1 - 10
cut, copy, and paste data blocks 1 - 9
Cutoff Block screen 2 - 62
Cutoff description 1 - 4
cutter comp
   using Insert Arc 2 - 74, 2 - 98, 2 - 131, 2 - 156
using Insert Line 2 - 75, 2 - 99, 2 - 132, 2 - 157
CUTTER COMP field
Axial Mill Contour screen 2 - 131
Axial Mill Frame screen 2 - 156
Radial Mill Contour screen 2 - 74
Radial Mill Frame screen 2 - 98
cutter compensation 2 - 188
Arc 2 - 74, 2 - 98, 2 - 131, 2 - 156
Line 2 - 75, 2 - 99, 2 - 132, 2 - 157
CUTTING SIDE field
Profile Start screen 2 - 18
CYCLE TYPE field
Axial Holes Drill Operation screen 2 - 180
Hole cycle 2 - 59
Radial Holes Drill Operation screen 2 - 116
CYLINDER DIAMETER field
Radial Holes Locations screen 2 - 121
Linear Coordinates 2 - 122
Rotary Coordinates 2 - 122
Radial Mill Circle screen 2 - 93
Radial Mill Contour screen 2 - 76
Radial Mill Frame screen 2 - 100
Radial Slot screen 2 - 105
CYLINDER RADIUS field
Radial Holes Locations screen 2 - 121
Radial Mill Circle screen 2 - 93
Radial Mill Contour screen 2 - 76
Radial Mill Frame screen 2 - 100
Radial Slot screen 2 - 105
D
Data Block 1 - 2
create 2 - 2
editing and navigation 1 - 7
Decreasing Chip Breaker (DECR CHP BKR) field
Axial Holes Drill Operation screen 2 - 180
Holes Cycle 2 - 59
Radial Holes Drill Operation screen 2 - 118
Decreasing Depth (DECR DEPTH) field
Holes Cycle 2 - 59
Decreasing Depth field, Threading 2 - 39
Decreasing Peck Drilling Depth (DECR DEPTH) field
Axial Holes Drill Operation screen 2 - 180
Radial Holes Drill Operation screen 2 - 118
Degrees field
Transfer block screen 2 - 208
DEGREES field, Machine Function (M Code) screen 2 - 198
DELETE BLOCK F2 softkey 1 - 11
Delete console key
use with Program Review screen 1 - 9
DELETE ELEMENT F3 softkey 2 - 25
DELETE F4 softkey 1 - 11
DELETE HOLE OPERATION F3 softkey
Axial Holes screen 2 - 178
Radial Holes screen 2 - 114
Delete key
use with Program Review screen 1 - 11
DELETE LOCATION F4 softkey
Axial Holes Locations screen 2 - 182
Radial Holes Locations screen 2 - 121
deleting blocks 1 - 11
DEPTH OF CUT field
Profile Start screen 2 - 15
Threading 2 - 39
DIAMETER field
Axial Mill Circle screen
Rectangular Coordinates 2 - 152
Radial Mill Circle screen
Linear Coordinates 2 - 94
Rotary Coordinates 2 - 93
DIRECTION field
Arc Element screen 2 - 32
Axial Mill Circle screen 2 - 149
Axial Mill Contour Arc Segment screen,
Polar Coordinates 2 - 140
Axial Mill Contour Arc Segment screen,
Rectangular Coordinates 2 - 141
Axial Mill Contour Blend Arc segment,
Polar Coordinates 2 - 143
Axial Mill Contour Blend Arc segment,
Rectangular Coordinates 2 - 144
Axial Mill Contour screen 2 - 130
Axial Mill Flats screen 2 - 169
Axial Mill Frame screen 2 - 156
Axial Mill Slot screen 2 - 161
Blend Arc Element screen 2 - 35
Profile Start screen 2 - 15
Radial Mill Circle screen 2 - 92
Radial Mill Contour Arc Segment screen
Linear Coordinates 2 - 84
Rotary Coordinates 2 - 83
Radial Mill Contour Blend Arc Segment screen
Linear Coordinates 2 - 87
Rotary Coordinates 2 - 86
Radial Mill Contour screen 2 - 73
Radial Mill Frame screen 2 - 98
Radial Slot screen 2 - 103
DISPLAY TIME (SEC) field
Comment block 2 - 201
Draw console key 1 - 7
Drill 2 - 58
Drilling Operation, live-tooling 2 - 119
dwell 2 - 63
DWELL (SECS) field
  Cutoff block 2 - 64
  Position block 2 - 7
Dwell drilling 2 - 59
DWELL field
  Axial Holes Drill Operation screen 2 - 180
  Cutoff Block Advanced Pecking tab 2 - 65
  Machine Function (M Code) screen 2 - 198
  Radial Holes Drill Operation screen 2 - 117
Transfer block screen 2 - 209
Dwell peck type, Cutoff 2 - 63
DWELL TIME (SEC) field
  Axial Holes Drill Operation screen 2 - 180, 2 - 181, 2 - 182
  Hole cycle 2 - 60
  Radial Holes Drill Operation screen 2 - 119, 2 - 120

E
editing 1 - 7
element 2 - 25
definition 2 - 11
Element operations
  arc 2 - 26
  blend arc 2 - 26
  chamfer 2 - 26
  face 2 - 26
  taper 2 - 26
  turn 2 - 26
ELEMENT TOLERANCE field
  Profile Start screen 2 - 16, 2 - 20, 2 - 21, 2 - 23
Emergency Stop - viii
ENABLE BLENDS field
  Axial Mill Contour screen 2 - 130
  Radial Mill Contour screen 2 - 73
ENCOMPASS field
  Axial Mill Flats screen
    Polar Coordinates 2 - 171
    Rectangular Coordinates 2 - 173
END CAP field
  Axial Mill Slot screen
    Polar Coordinates 2 - 164
    Rectangular Coordinates 2 - 166
  Radial Slot screen
    Linear Coordinates 2 - 109
    Rotary Coordinates 2 - 106
End console key
  use with Program Review screen 1 - 9
error messages - vii

F
Face 2 - 26
FACE ALLOWANCE field
  Finishing 2 - 53
  Profile Start screen 2 - 25
FACE CLEARANCE field
  Axial Mill Flats screen 2 - 170
Face Element 2 - 29
FACE F1 softkey, element 2 - 26
Face thread 2 - 40
Face Thread Type 2 - 49
FEED (IPM) field 2 - 7
FEED field
  Axial Mill Contour Arc Segment screen
    Polar Coordinates 2 - 140
    Rectangular Coordinates 2 - 141
  Axial Mill Contour Blend Arc segment
    Polar Coordinates 2 - 143
    Rectangular Coordinates 2 - 144
  Axial Mill Contour Line Segment screen
    Polar Coordinates 2 - 136
    Rectangular Coordinates 2 - 137
  Cutoff block 2 - 63
  Cutoff Block Advanced Pecking tab 2 - 65
  Move Tailstock screen 2 - 206
  Profile Start screen 2 - 12
  Radial Mill Contour Arc Segment screen
    Linear Coordinates 2 - 84
    Rotary Coordinates 2 - 83
  Radial Mill Contour Blend Arc Segment screen
    Linear Coordinates 2 - 87
    Rotary Coordinates 2 - 86
  Radial Mill Contour Line Segment screen
    Linear Coordinates 2 - 80
    Rotary Coordinates 2 - 79
  Transfer block screen 2 - 208, 2 - 209
FEED OVERRIDE LOCKOUT field, Change
  Parameters 2 - 193
  Feed per Minute (FPM) 1 - 3
  FEED PER MINUTE F4 softkey
    Cutoff block 2 - 64
  FEED PER REV F5 softkey
    Cutoff block 2 - 64
  Feed per Revolution 1 - 3
FINAL PECK DEPTH field
Axial Holes Drill Operation screen 2 - 180
Radial Holes Drill Operation screen 2 - 119
FIND ANOTHER CENTER POINT F7 softkey 2 - 32
FIND ANOTHER END POINT F7 softkey 2 - 31, 2 - 32
FINISH (1 PASS) field
Axial Mill Flats screen 2 - 168
FINISH (1 PASS), Groove 2 - 53
FINISH FEED (IPR ) or (MMPR) field
Arc Element screen 2 - 33
Blend Arc Element screen 2 - 35
Chamfer Element screen 2 - 37
Face Element screen 2 - 30
Taper Element screen 2 - 31
Turn Element screen 2 - 28
FINISH OFFSETS field
Arc Element screen 2 - 33
Blend Arc Element screen 2 - 35
Chamfer Element screen 2 - 37
Face Element screen 2 - 30
Taper Element screen 2 - 31
Turn Element screen 2 - 29
FINISH ONLY field
Axial Mill Circle screen 2 - 147
Axial Mill Contour screen 2 - 128
Axial Mill Flats screen 2 - 168
Axial Mill Frame screen 2 - 154
Axial Mill Slot screen 2 - 161
Groove 2 - 53
Radial Mill Circle screen 2 - 90
Radial Mill Contour screen 2 - 71
Radial Mill Frame screen 2 - 96
Radial Slot screen 2 - 103
FINISH PASSES field
Threading 2 - 39
Finish Profile data block 2 - 13
FINISH SPEED (FPM) or (MPM) field
Arc Element screen 2 - 33
Blend Arc Element screen 2 - 35
Chamfer Element screen 2 - 37
Face Element screen 2 - 30
Taper Element screen 2 - 31
Turn Element screen 2 - 28
Finish Speed and Finish Feed values change to 0 2 - 14
FIRST PECK DEPTH field
Hole cycle 2 - 60
FIRST Z THEN X F2 softkey 2 - 8
FIRST Z THEN Y THEN X F5 softkey 2 - 9
FLOOR ALLOWANCE field
Axial Mill Circle screen 2 - 149
Axial Mill Contour screen 2 - 130
Axial Mill Flats screen 2 - 169
Axial Mill Frame screen 2 - 155
Axial Mill Slot screen 2 - 161
Radial Mill Circle screen 2 - 91
Radial Mill Contour screen 2 - 73
Radial Mill Frame screen 2 - 97
Radial Slot screen 2 - 103
G
GOUGE AVOIDANCE field
Profile Start screen 2 - 13
Groove description 1 - 4
GROOVE F3 softkey 2 - 52
GROOVE FINISH F4 softkey, Profile screen 2 - 13
GROOVE ROUGH F3 softkey, Profile screen 2 - 13
Groove Rough operation rule 2 - 20
Groove walls 2 - 56
GROOVE WIDTH field, Grooving 2 - 54
Groove, roughing and finish pass strategy 2 - 53
Grooving Cycle 2 - 51
screen 2 - 52
H
HEIGHT field
Axial Lettering screen 2 - 176
Radial Lettering screen 2 - 111
Hole Cycle screen 2 - 58
Hole description 1 - 4
Hole F4 softkey 2 - 58
Home console key
use with Program Review screen 1 - 9
home position 2 - 4
I
I PLANE field
Axial Bolt Circle screen
Polar Coordinate 2 - 185
Rectangular Coordinates 2 - 186
Axial Locations screen
Polar Coordinates 2 - 183
Rectangular Coordinates 2 - 184
Radial Holes Locations screen
Linear Coordinates 2 - 122
Rotary Coordinates 2 - 122
Radial Holes Loop Around C-Axis screen
icons - viii
ID Straight thread 2 - 40
ID Straight Thread Type 2 - 45
ID Taper thread 2 - 40
ID Taper Thread Type 2 - 47
IN/TAN (INSIDE TANGENT) field
Axial Mill Circle screen 2 - 148
Axial Mill Frame screen 2 - 154
Cutter Compensation 2 - 189
Radial Mill Circle screen 2 - 90
Radial Mill Frame screen 2 - 96
inch per minute (IPM) 1 - 3
inch per revolution (IPR) 1 - 3
INCLUDE ARC field
Axial Mill Slot screen
  Polar Coordinates 2 - 164
  Rectangular Coordinates 2 - 166
Radial Slot screen
  Linear Coordinates 2 - 108
  Rotary Coordinates 2 - 106
infeed angle
Face 2 - 49
ID Straight 2 - 46
ID Taper 2 - 48
OD Straight 2 - 42
OD Taper 2 - 44
INSCRIBE field
Axial Mill Flats screen
  Polar Coordinates 2 - 171
  Rectangular Coordinates 2 - 173
INSERT BLOCK BEFORE F7 softkey 1 - 11
insert blocks 1 - 11
Insert console key
  use with Program Review screen 1 - 9
INSERT ELEMENT BEFORE F7 2 - 25
INSERT Hole OPERATION F7 softkey
Axial Holes screen 2 - 178
Radial Holes screen 2 - 114
INSERT LOCATION BEFORE F7 softkey
Axial Holes Locations screen 2 - 182
Radial Holes Locations screen 2 - 121
INSERT TIP RADIUS field
  Cutoff block 2 - 63
  Grooving 2 - 52
  Profile Start screen 2 - 12
  Threading 2 - 38
INSIDE field
Axial Mill Circle screen 2 - 148
Axial Mill Frame screen 2 - 154
Cutter Compensation 2 - 189
Radial Mill Circle screen 2 - 90
Radial Mill Frame screen 2 - 96
INWARD field
Axial Mill Circle screen 2 - 149
Axial Mill Contour screen 2 - 130
Axial Mill Frame screen 2 - 155
Radial Mill Circle screen 2 - 91
Radial Mill Contour screen 2 - 73
Radial Mill Frame screen 2 - 97
IPM field
  Cutoff block 2 - 64
IPR field
  Cutoff block 2 - 64
K
keyboard 1 - 7
L
LEAD (TPI) field
  Threading 2 - 42, 2 - 44, 2 - 46, 2 - 48, 2 - 49
LEAD IN ANGLE field 2 - 42, 2 - 44, 2 - 46, 2 - 48, 2 - 49
LEAD OUT ANGLE field
  Threading 2 - 42, 2 - 44, 2 - 46, 2 - 48, 2 - 50
LEFT field
Axial Mill Contour screen 2 - 129
Cutter Compensation 2 - 189
Radial Mill Contour screen 2 - 72
LENGTH field
Axial Mill Contour Line Segment screen
  Rectangular Coordinates 2 - 137
Face Element screen 2 - 29
Radial Mill Contour Line Segment screen
  Linear Coordinates 2 - 80
  Taper Element screen 2 - 31
  Turn Element screen 2 - 28
LINE F1 softkey
Axial Mill Contour screen 2 - 136
New Radial Contour Segment screen 2 - 78
Radial Mill Contour Line Segment screen 2 - 79
LINE field
Axial Mill Slot screen
  Polar Coordinates 2 - 164
  Rectangular Coordinates 2 - 166
Radial Slot screen
  Linear Coordinates 2 - 108
  Rotary Coordinates 2 - 106
Line segment
Axial Mill Contour screen 2 - 136
New Radial Contour Segment screen 2 - 79
Linear coordinates 2 - 76, 2 - 93, 2 - 100,
Linear Y Motion
radial milling 2 - 68
Linear Y motion
axial milling 2 - 125
LINEAR Y MOTION field
Axial Lettering screen 2 - 176
Radial Lettering screen 2 - 111
Radial Mill Circle screen
  Linear Coordinates 2 - 94, 2 - 101,
  2 - 108, 2 - 123, 2 - 134, 2 -
  137, 2 - 142, 2 - 145, 2 -
  152, 2 - 159, 2 - 165, 2 -
  174, 2 - 184, 2 - 186
Radial Mill Contour screen
  Linear Coordinates 2 - 77
list arguments, NC Program Call screen 2 - 210
Live Tooling data blocks 2 - 66
LIVE TOOLING F3 softkey, New Block screen 2 - 66
Location Number field
  Axial Locations screen 2 - 182
  Radial Holes Locations screen 2 - 121

M
M Code description 1 - 4
M CODE field
  Machine Function (M Code) screen 2 - 198
M19 (Spindle Orient) field
  Machine Function (M Code) screen 2 - 198
M203 (Sync Spindles Forward Mode) field
  Machine Function (M Code) screen 2 - 198
M204 (Sync Spindle Reverse Mode) field
  Machine Function (M Code) screen 2 - 198
M24 (Part Conveyor On)
  Machine Function (M Code) screen 2 - 199
M3 (Spindle CW) field
  Machine Function (M Code) screen 2 - 198
M4 (Spindle CCW) field
  Machine Function (M Code) screen 2 - 198
M5 (Spindle Off) field
  Machine Function (M Code) screen 2 - 198
M50 (Chip Conveyor On)
  Machine Function (M Code) screen 2 - 199
Machine Code definitions 2 - 198
Machine Function screen 2 - 198
MAJOR DIAMETER field
  Threading 2 - 41, 2 - 43, 2 - 45, 2 - 47
MAX OFFSET field
  Axial Mill Contour screen 2 - 130
  Radial Mill Contour screen 2 - 73
MAX RPM (Live Tooling Spindle) field
  Change Parameters screen 2 - 193
MAX RPM (Primary Spindle) field
  Change Parameters screen 2 - 193
MAX RPM (Sub-Spindle) field
  Change Parameters screen 2 - 193
maximum number of segments 2 - 78, 2 - 135
messages
  error - vii
meters per minute (MPM) 1 - 3
MILL FEED field
  Axial Lettering screen 2 - 175
  Axial Mill Circle screen 2 - 148
  Axial Mill Contour screen 2 - 129
  Axial Mill Flats screen 2 - 169
  Axial Mill Frame screen 2 - 155
  Axial Mill Slot screen 2 - 161
  Radial Lettering screen 2 - 110
  Radial Mill Circle screen 2 - 91
  Radial Mill Contour screen 2 - 72
  Radial Mill Frame screen 2 - 97
  Radial Slot screen 2 - 104
millimeters per minute (MMPM) 1 - 3
millimeters per revolution (MMPR) 1 - 3
MILLING TYPE field
  Axial Mill Circle screen 2 - 148
  Axial Mill Contour screen 2 - 129
  Axial Mill Frame screen 2 - 154
  Radial Mill Circle screen 2 - 90, 2 - 96
  Radial Mill Contour screen 2 - 72
MINIMUM PECK DEPTH field
  Hole cycle 2 - 60
MINOR DIAMETER field
  Threading 2 - 41, 2 - 43, 2 - 45, 2 - 47
MISCELLANEOUS F4 softkey
  TMM, TMX MY, and TMX MYS Series 2 - 191
mode
  programming - viii
move a data block to a different location
  within the part program 1 - 10
MOVE ALL HOME F4 softkey
  Position datablock 2 - 4
Move Tailstock Data Block, TM12 and TM18
  series machines 2 - 206
MOVE TAILSTOCK description 1 - 5
MOVE TAILSTOCK F5 softkey 2 - 206
MOVE TO POSITION
  Position datablock 2 - 4
MOVE TO TOOL CHANGE
  Position datablock 2 - 4
MOVE TO Z POSITION field
  Move Tailstock screen 2 - 206
MOVE TURRET HOME
  Position datablock 2 - 4
MOVE W HOME F6 softkey
  Position datablock 2 - 4
MOVE W ONLY F5 softkey
  Position datablock 2 - 4
MOVE X THEN Z
  Axis Order field, User Defined Tool
  Change Position area in Program
  Parameters screen 2 - 194
MOVE Y HOME F2 softkey
  Position datablock 2 - 4
MOVE Y ONLY F15 softkey
  Position datablock 2 - 4
MOVE Z THEN X
  Axis Order field, User Defined Tool
  Change Position area in Program
  Parameters screen 2 - 194
MULTIPLE BLOCK FUNCTIONS F1 softkey
  program review 1 - 10
  multiple blocks, select for editing 1 - 9

N
NC Conversational Merge (NC Program Call) 2 - 202
NC PROGRAM CALL F6 softkey 2 - 210
New Axial Contour Segment screen 2 - 135
New Block screen
  TMM series 2 - 2
New Radial Contour Segment screen 2 - 78
NEXT ELEMENT F2 softkey 2 - 25, 2 - 26
NEXT HOLE OPERATION F2 softkey
  Axial Holes screen 2 - 178
  Radial Holes screen 2 - 114
NONE field, Comment block stop type 2 - 201
NUMBER field, Grooving Cycle screen 2 - 57
NUMBER OF HOLES field
  Axial Bolt Circle screen, Polar
  Coordinates 2 - 185
  Radial Holes Loop Around C-Axis screen 2 - 123
NUMBER OF PASSES field, Threading Cycle 2 - 39
NUMBER OF SIDES field
  Axial Mill Flats screen
  Polar Coordinates 2 - 171
  Rectangular Coordinates 2 - 173
NUMBER OF STARTS
  Threading 2 - 42, 2 - 44, 2 - 46, 2 - 48, 2 - 50

O
OD Straight Thread Type 2 - 41
OD Taper
  infeed angle 2 - 44
  pullout angle 2 - 44
OD Taper Thread Type 2 - 43
OFFSET field
  Grooving cycle 2 - 57
Offset Spindles field
  Transfer block screen 2 - 208
offsets
  save and restore 2 - 197
OFFSETS field
  Cutoff block 2 - 63
  Grooving 2 - 52
  Hole cycle 2 - 59
ON field
  Axial Mill Circle screen 2 - 148
  Axial Mill Contour screen 2 - 129
  Axial Mill Frame screen 2 - 154
  Cutter Compensation 2 - 189
  Radial Mill Circle screen 2 - 90
  Radial Mill Contour screen 2 - 72
  Radial Mill Frame screen 2 - 96
On Sub-spindle check box
  Change Part Setup 2 - 196
on-screen calculator  - viii
on-screen cursor 1 - 7
OPERATION field
  Profile Start screen 2 - 13
  optional AT-Keyboard 1 - 7
OPTIONAL field
  Comment block stop type 2 - 201
  Program Stop 2 - 7
  optional stop 1 - 4
ORIENTATION field
  Axial Lettering screen 2 - 176
  Grooving 2 - 54
Radial Lettering screen 2 - 111
OUT/TAN (OUTSIDE TANGENT) field
  Axial Mill Circle screen 2 - 148
  Axial Mill Frame screen 2 - 154
  Cutter Compensation 2 - 189
  Radial Mill Circle screen 2 - 90
  Radial Mill Frame screen 2 - 96
OUTSIDE field
  Axial Mill Circle screen 2 - 148
  Axial Mill Frame screen 2 - 154
  Cutter Compensation 2 - 189
Radial Mill Circle screen 2 - 90
Radial Mill Frame screen 2 - 96
OUTWARD field
  Axial Mill Circle screen 2 - 149
  Axial Mill Contour screen 2 - 130
  Axial Mill Frame screen 2 - 155
  Radial Mill Circle screen 2 - 91
  Radial Mill Contour screen 2 - 73
  Radial Mill Frame screen 2 - 97
OVERRIDE TOLERANCE field
  Profile Start screen 2 - 16, 2 - 19, 2 - 21, 2 - 23

P
Page Down console key
  use with Program Review screen 1 - 9
PAGE DOWN F2 softkey
  Change Part Setup 2 - 197
Page Up console key
  use with Program Review screen 1 - 9
PAGE UP F1 softkey
  Change Part Setup 2 - 197
part program sample 3 - 1
Part Programming 1 - 3
Part Setup Work Offsets
  READ TABLE FROM FILE F4 softkey 2 - 197
  WRITE TABLE TO FILE F2 softkey 2 - 197
Part Zero location
  Change Part Setup 2 - 197
paste 1 - 9
PASTE F3 1 - 10
Patterns, Grooving 2 - 57
PAUSE AT END (SEC) field
  Cutoff block 2 - 63
PECK CLEARANCE field
  Axial Holes Drill Operation screen 2 - 181
  Axial Mill Circle screen 2 - 149
  Axial Mill Contour screen 2 - 130
  Axial Mill Frame screen 2 - 156
  Hole cycle 2 - 61
  Radial Holes Drill Operation screen 2 - 119
  Radial Mill Circle screen 2 - 92
  Radial Mill Contour screen 2 - 73
  Radial Mill Frame screen 2 - 98
PECK COUNT field
  Axial Holes Drill Operation screen 2 - 180
  Hole cycle 2 - 61
  Radial Holes Drill Operation screen 2 - 119

Peck Depth example 2 - 61
PECK DEPTH field
  Axial Holes Drill Operation screen 2 - 180
  Axial Holes Rigid Tap Operation screen 2 - 181
  Axial Mill Circle screen 2 - 149
  Axial Mill Contour screen 2 - 130
  Axial Mill Flats screen 2 - 169
  Axial Mill Frame screen 2 - 156
  Axial Mill Slot screen 2 - 161
  Cutoff block 2 - 64
  Hole cycle 2 - 60
  Radial Holes Drill Operation screen 2 - 119
  Radial Holes Rigid Tap Operation screen 2 - 120
  Radial Mill Circle screen 2 - 92
  Radial Mill Contour screen 2 - 73
  Radial Mill Frame screen 2 - 98
  Radial Slot screen 2 - 104
peck drilling 2 - 59
PECK DRILLING field
  Axial Holes Drill Operation screen 2 - 180
  Radial Holes Drill Operation screen 2 - 117
PECK PERCENT field
  Axial Holes Drill Operation screen 2 - 180
  Hole cycle 2 - 61
  Radial Holes Drill Operation screen 2 - 119
PECK TYPE field
  Cutoff block 2 - 63
PITCH field
  Axial Holes Drill Operation screen 2 - 181
  Hole cycle 2 - 60
  Radial Holes Drill Operation screen 2 - 120
  Threading 2 - 42, 2 - 44, 2 - 46, 2 - 48,
  2 - 49
PLUNGE FEED field
  Axial Holes Drill Operation screen 2 - 180
  Axial Lettering screen 2 - 175
  Axial Mill Circle screen 2 - 148
  Axial Mill Contour screen 2 - 129
  Axial Mill Flats screen 2 - 169
  Axial Mill Frame screen 2 - 155
  Axial Mill Slot screen 2 - 161
  Hole cycle 2 - 61
  Radial Holes Drill Operation screen 2 - 116
Radial Lettering screen 2 - 110
Radial Mill Circle screen 2 - 91
Radial Mill Contour screen 2 - 72
Radial Mill Frame screen 2 - 97
Radial Slot screen 2 - 104
plunge points
axial circles 2 - 150
axial frames 2 - 157
radial circles 2 - 92
radial frames 2 - 99
PLUNGE TYPE field
Axial Mill Flats screen 2 - 169
Axial Mill Slot screen 2 - 161
Radial Slot screen 2 - 104
POCK BOUNDARY (POCKET BOUNDARY) field
Axial Mill Circle screen 2 - 148
Axial Mill Contour screen 2 - 129
Axial Mill Frame screen 2 - 154
Cutter Compensation 2 - 189
Radial Mill Circle screen 2 - 90
Radial Mill Contour screen 2 - 72
Radial Mill Frame screen 2 - 96
POCKET OVERLAP (%) field
Axial Mill Circle screen 2 - 149
Axial Mill Contour screen 2 - 130
Axial Mill Frame screen 2 - 155
Radial Mill Circle screen 2 - 91
Radial Mill Contour screen 2 - 73
Radial Mill Frame screen 2 - 97
POCKET TYPE field
Axial Mill Circle screen 2 - 149
Axial Mill Contour screen 2 - 130
Axial Mill Frame screen 2 - 155
Radial Mill Circle screen 2 - 91
Radial Mill Contour screen 2 - 73
Radial Mill Frame screen 2 - 97
polar coordinates 2 - 133, 2 - 151, 2 - 158,
2 - 162, 2 - 171, 2 - 182, 2 - 185
pop-up text entry window 1 - 7
Position Data Blocks 2 - 3
create 2 - 3
Position description 1 - 3
POSITION F1 softkey 2 - 3
power-on control 2 - viii
PREVIOUS ELEMENT F1 softkey 2 - 25
PREVIOUS Hole OPERATION F1 softkey
Axial Holes screen 2 - 178
Radial Holes screen 2 - 114
Profile description 1 - 3
Profile End, element 2 - 25
PROFILE LEFT field
Axial Mill Contour screen 2 - 129
Cutter Compensation 2 - 189
Radial Mill Contour screen 2 - 72
PROFILE RIGHT field
Axial Mill Contour screen 2 - 129
Cutter Compensation 2 - 189
Radial Mill Contour screen 2 - 72
Profile Start
element 2 - 25
screen 2 - 11
Profiles 2 - 11
Program Editing Features 1 - 7
Program field
Comment block stop type 2 - 201
PROGRAM NUMBER field, NC Program Call screen 2 - 211
Program Review screen 1 - 8
program status 2 - vii
PROGRAM STOP field 2 - 7
programming coordinates
axial milling 2 - 125
radial milling 2 - 68
programming mode
console keys 1 - 7
pullout angle
Face 2 - 50
ID Straight 2 - 46
ID Taper 2 - 48
OD Straight 2 - 42
OD Taper 2 - 44
pullout ID Taper 2 - 48

R

R CENTER field
Axial Bolt Circle screen
Polar Coordinates 2 - 185
Axial Mill Circle screen
Polar Coordinates 2 - 151
Axial Mill Contour Arc Segment screen
Polar Coordinates 2 - 140
Axial Mill Contour Blend Arc segment
Polar Coordinates 2 - 144
R CORNER field
Axial Mill Frame screen
Polar Coordinates 2 - 158
R END field
Axial Mill Contour Arc Segment screen
Polar Coordinates 2 - 140
Axial Mill Contour Blend Arc segment
Polar Coordinates 2 - 143
Axial Mill Contour Line Segment screen
Polar Coordinates 2 - 136
Axial Mill Slot screen
Polar Coordinates 2 - 163
R field
Axial Locations screen
Polar Coordinates 2 - 183
R LENGTH field
Axial Mill Frame screen
Polar Coordinates 2 - 158
R PLANE field
Axial Bolt Circle screen
Polar Coordinates 2 - 185
Rectangular Coordinates 2 - 186
Axial Holes Drill Operation screen 2 - 179
Axial Locations screen
Polar Coordinates 2 - 183
Rectangular Coordinates 2 - 184
Hole cycle 2 - 59
Radial Holes Drill Operation screen 2 - 115
Radial Holes Locations screen
Linear Coordinates 2 - 122
Rotary Coordinates 2 - 122
Radial Holes Loop Around C-Axis screen 2 - 123
Radial Lettering screen 2 - 110
R START field
Axial Mill Contour Arc Segment screen
Polar Coordinates 2 - 140
Axial Mill Contour Blend Arc segment
Polar Coordinates 2 - 143
Axial Mill Contour Line Segment screen
Polar Coordinates 2 - 136
Axial Mill Contour screen
Polar Coordinates 2 - 133
Axial Mill Flats screen
Polar Coordinates 2 - 172
Axial Mill Slot screen
Polar Coordinates 2 - 163
Radial Circle 2 - 89
Radial Milling 2 - 67
RADIAL CIRCLE F2 softkey
New Block Radial Milling screen 2 - 89
Radial Circles
plunge points 2 - 92
Radial Frame
plunge points 2 - 99
Radial Mill Frame screen 2 - 95
Radial Milling 2 - 67
RADIAL FRAME F3 softkey
New Block (Radial Milling) screen 2 - 95
RADIAL HOLE CYCLE F1 softkey
New Radial Hole Operation screen 2 - 115
New Radial Holes screen 2 - 114
Radial HOLES (Operations) 2 - 114
Radial Holes description 1 - 4
RADIAL HOLES F2 softkey, New Block Live Tooling screen 2 - 114
RADIAL LETTERING
Radial Milling 2 - 67
RADIAL LETTERING F5 softkey
New Block (Radial Milling) screen 2 - 110
Radial Lines and Arcs
Radial Milling 2 - 67
RADIAL LOCATIONS CYCLE F2 softkey
Radial Holes Locations screen 2 - 121
RADIAL LOCATIONS F2 softkey
New Radial Holes screen 2 - 114
RADIAL LOOP AROUND C-AXIS F3 softkey
New Radial Holes Operation screen 2 - 123
New Radial Holes screen 2 - 114
Radial milling
Linear Y motion 2 - 68
programming coordinates 2 - 68
Radial Milling description 1 - 4
RADIAL MILLING F1 softkey 2 - 67
RADIAL SLOT
Radial Milling 2 - 67
RADIAL SLOT F4 softkey
New Block (Radial Milling) screen 2 - 102
RADIUS field
Arc Element screen 2 - 33
Axial Bolt Circle screen
Polar Coordinates 2 - 185
Rectangular Coordinates 2 - 186
Axial Mill Circle screen
Polar Coordinates 2 - 151
Rectangular Coordinates 2 - 152
Axial Mill Contour Arc Segment screen
Polar Coordinates 2 - 140
Rectangular Coordinates 2 - 141
Axial Mill Contour Blend Arc segment
Polar Coordinates 2 - 143
Rectangular Coordinates 2 - 144
Blend Arc Element screen 2 - 35
Grooving 2 - 56
Radial Mill Circle screen
Linear Coordinates 2 - 94
Rotary Coordinates 2 - 93
Radial Mill Contour Arc Segment screen
Linear Coordinates 2 - 84
Rotary Coordinates 2 - 83
Radial Mill Contour Blend Arc Segment screen
Linear Coordinates 2 - 87
Rotary Coordinates 2 - 86
RAMP ANGLE field
Axial Mill Flats screen 2 - 169
Axial Mill Slot screen 2 - 161
Radial Slot screen 2 - 104
RAPID field
Axial Mill Flats screen 2 - 169
Axial Mill Slot screen 2 - 161
Radial Slot screen 2 - 104
RAPID TO POSITION field 2 - 7
Rapid to W Position? field
Transfer block screen 2 - 209
RAPID TRAVERSE field
Change Parameters 2 - 193
READ TABLE FROM FILE F4 softkey
Part Setup Work Offsets 2 - 197
Reaming operation 2 - 120
rectangular coordinates 2 - 133, 2 - 151,
2 - 158, 2 - 162, 2 - 171, 2 - 182, 2 - 185
reference corner, radial mill frame 2 - 100,
2 - 102, 2 - 109, 2 - 158
REFERENCE field
part zero or machine zero 2 - 4, 2 - 8
User Defined Tool Change Position area
in Program Parameters screen
2 - 194
REPEAT COUNT field 2 - 202
Repeat End description 1 - 5
REPEAT END F2 softkey 2 - 203
REPEAT END F7 softkey 2 - 203
Repeat End screen 2 - 203
Repeat Start description 1 - 5
REPEAT START F6 softkey 2 - 202
Repeat Start screen 2 - 202
RETRACT ALONG PROFILE field
Roughing 2 - 16
RETRACT ANGLE field
Grooving 2 - 53
Profile Start screen 2 - 16, 2 - 19, 2 -
20, 2 - 22
RETRACT AT RAPID field
Axial Holes Drill Operation screen 2 -
182
cutoff block 2 - 64
Radial Holes Drill Operation screen 2 -
120
RETRACT CLEARANCE field
Grooving 2 - 53
Profile Start screen 2 - 16, 2 - 19, 2 -
21, 2 - 23
RETRACT DISTANCE field
cutoff block 2 - 64
RETRACT FEED field
cutoff block 2 - 64
RETRACT LEVEL field
Holes cycle 2 - 61
RETURN field
Axial Bolt Circle screen
Polar Coordinates 2 - 185
Radial Locations screen
Polar Coordinates 2 - 183
Rectangular Coordinates 2 - 184
Radial Holes Locations screen
Rotary Coordinates 2 - 122
RETURN LEVEL field
Axial Bolt Circle screen
Rectangular Coordinates 2 - 186
Radial Holes Locations screen
Linear Coordinates 2 - 122
Radial Holes Loop Around C-Axis screen
2 - 123
review 1 - 8
RIGHT field
Axial Mill Contour screen 2 - 129
Cutter Compensation 2 - 189
Radial Mill Contour screen 2 - 72
RIGID field
Axial Holes Drill Operation screen 2 -
181
Radial Holes Drill Operation screen 2 -
119
rigid tap 2 - 60
rotary coordinates 2 - 76, 2 - 93, 2 - 100,
2 - 105, 2 - 121
ROUGH AND FINISH field
Axial Mill Circle screen 2 - 147
Axial Mill Contour screen 2 - 128
Axial Mill Flats screen 2 - 168
Axial Mill Frame screen 2 - 154
Axial Mill Slot screen 2 - 161
Groove screen 2 - 53
Radial Mill Circle screen 2 - 90
Radial Mill Contour screen 2 - 71
Radial Mill Frame screen 2 - 96
Radial Slot screen 2 - 103
ROUGH FINAL DEPTH field
Threading 2 - 39
ROUGH ONLY field
Axial Mill Circle screen 2 - 147
Axial Mill Contour screen 2 - 128
Axial Mill Flats screen 2 - 168
Axial Mill Frame screen 2 - 154
Axial Mill Slot screen 2 - 161
Groove screen 2 - 53
Radial Mill Circle screen 2 - 90, 2 - 96
Radial Mill Contour screen 2 - 71
Radial Slot screen 2 - 103
ROUGH START DEPTH field
Threading 2 - 39
RPM field
Transfer block screen 2 - 208
S

S1 Direction field
  Transfer block screen 2 - 208
sample part program 3 - 1
sample screens - vii
save and restore offsets 2 - 197
segments
  Axial Mill Contour 2 - 135
  maximum number 2 - 78, 2 - 135
  New Radial Contour Segment screen 2 - 78
  numbers 2 - 78, 2 - 135
  segment zero 2 - 78, 2 - 135
SELECT FONT field
  Radial Lettering screen 2 - 110
SEQUENCE field
  Grooving cycle 2 - 53
SIMULTANEOUS F3 softkey
  Position Block screen 2 - 8
SIMULTANEOUS field
  Axis Order field, User Defined Tool
    Change Position area in Program Parameters screen 2 - 194
SIZING DIAMETER field
  Axial Mill Flats screen
    Polar Coordinates 2 - 172
    Rectangular Coordinates 2 - 173
SIZING METHOD field
  Axial Mill Flats screen
    Polar Coordinates 2 - 171
    Rectangular Coordinates 2 - 173
SKIP LIST field
  Axial Bolt Circle screen
    Polar Coordinates 2 - 186
    Rectangular Coordinates 2 - 186
  Radial Holes Loop Around C-Axis screen 2 - 123
SLOT WIDTH field
  Axial Mill Slot screen
    Polar Coordinates 2 - 163
    Rectangular Coordinates 2 - 166
  Radial Slot screen
    Linear Coordinates 2 - 108
    Rotary Coordinates 2 - 106
SPECIFY START ANGLE field
  Threading 2 - 42, 2 - 44, 2 - 46, 2 - 48, 2 - 50
SPEED (RPM) field
  Axial Holes Drill Operation screen 2 - 179
  Axial Lettering screen 2 - 175
  Axial Mill Circle screen 2 - 148
  Axial Mill Contour screen 2 - 129
  Axial Mill Flats screen 2 - 169
  Axial Mill Frame screen 2 - 155
  Axial Mill Slot screen 2 - 161
  Radial Holes Drill Operation screen 2 - 116
  Radial Lettering screen 2 - 110
  Radial Mill Circle screen 2 - 91
  Radial Mill Contour screen 2 - 72
  Radial Mill Frame screen 2 - 97
  Radial Slot screen 2 - 103
  Speed as Constant Surface Speed (CSS) 1 - 3
  Speed as RPM 1 - 3
SPEED field
  Cutoff block 2 - 63
  Cutoff Block Advanced Pecking tab 2 - 65
  Grooving 2 - 53
  Hole cycle 2 - 61
  Profile Start screen 2 - 12
SPINDLE field, Machine Function (M Code) screen 2 - 198
SPINDLE RPM field
  Threading 2 - 39
SPRING PASSES field
  Profile Start screen 2 - 19
  Threading 2 - 39
standard
  drilling 2 - 59
  peck type, Cutoff 2 - 63
  tap 2 - 60
STANDARD field
  Axial Holes Drill Operation screen 2 - 181
  Radial Holes Drill Operation screen 2 - 116, 2 - 119
START (DEG) field
  Axial Mill Contour Line Segment screen
    Polar Coordinates 2 - 136
START ANGLE field
  Axial Bolt Circle screen
    Polar Coordinates 2 - 185
    Rectangular Coordinates 2 - 186
  Radial Holes Loop Around C-Axis screen 2 - 123
  Threading 2 - 42, 2 - 44, 2 - 46, 2 - 48, 2 - 50
START CAP field
  Axial Mill Slot screen
    Polar Coordinates 2 - 164
    Rectangular Coordinates 2 - 166
  Radial Slot screen
    Linear Coordinates 2 - 108
    Rotary Coordinates 2 - 106
start point, groove 2 - 52
status, program - vii

STEP OVER field
Axial Mill Flats screen 2 - 169
Axial Mill Slot screen 2 - 161
Grooving 2 - 53
Profile Start screen 2 - 20
Radial Slot screen 2 - 103

STOCK ALLOWANCE field
Finishing 2 - 53
Profile Start 2 - 25

STOCK SURFACE field
Axial Mill Flats screen
Polar Coordinates 2 - 172
Rectangular Coordinates 2 - 173

stop block 2 - 7

STOP THE SPINDLE field 2 - 7
STOP TYPE field
Comment block 2 - 201

Store Machine Position console key
Change Part Setup 2 - 197
STORE MACHINE POSITION F5 softkey
Change Part Setup 2 - 197
STORE MACHINE POSITION F7 softkey
Position Block 2 - 5, 2 - 8

Straight drilling 2 - 59, 2 - 63, 2 - 180

STRAIGHT field
Axial Mill Flats screen 2 - 169
Axial Mill Slot screen 2 - 161
Radial Slot screen 2 - 104

STRATEGY field
Axial Mill Circle screen 2 - 147
Axial Mill Contour screen 2 - 128
Axial Mill Flats screen 2 - 168
Axial Mill Frame screen 2 - 154
Axial Mill Slot screen 2 - 161
Groove 2 - 53
Profile Start screen 2 - 18, 2 - 22
Radial Mill Circle 2 - 90
Radial Mill Contour 2 - 71
Radial Mill Frame 2 - 96
Radial Slot screen 2 - 103
Threading 2 - 39

string arguments, NC Program Call screen 2 - 210

STYLE field
Radial Lettering screen 2 - 110

SURFACE CLEARANCE field
Axial Mill Slot screen 2 - 161
Radial Slot screen 2 - 104

Tailstock, Move Data Block 2 - 206
Tap 2 - 58
   rigid 2 - 60
   standard 2 - 60

TAP CYCLE field
Axial Holes Drill Operation screen 2 - 181
Hole cycle 2 - 60
Radial Holes Drill Operation screen 2 - 119
Taper 2 - 26
TAPER 1-2 and 4-3 fields, Grooving 2 - 56

TAPER ANGLE field
Threading Cycle 2 - 44, 2 - 48

Taper Element 2 - 30
TAPER F2 softkey, element 2 - 26

Tapping Operation 2 - 119
text entry window 1 - 7

TEXT field
Axial Lettering screen 2 - 176
Radial Lettering screen 2 - 111
text for the Comment block 2 - 201

thread
   Face 2 - 40
   ID Straight 2 - 40
   ID Taper 2 - 40
   OD Straight 2 - 40
   OD Taper 2 - 40

THREAD DEPTH field
Threading Cycle 2 - 49

Thread description 1 - 3

THREAD TYPE field 2 - 40

Threading CYCLE screen 2 - 38

TIME OFF (SEC) field
Machine Function (M Code) field 2 - 199

TIME ON (SEC) field
Machine Function (M Code) field 2 - 199

TIP COMPENSATION field
Axial Holes Drill Operation screen 2 - 179
Hole cycle 2 - 61
Radial Holes Drill Operation screen 2 - 115

TOOL CHANGE OVERRIDE area
Position data block 2 - 7

TOOL CHANGE POSITION, USER DEFINED field, Program Parameters screen 2 - 194

TOOL DIAMETER field
Axial Holes Drill Operation screen 2 - 179

Axial Lettering screen 2 - 175
Axial Mill Circle screen 2 - 147
Axial Mill Contour screen 2 - 128
Axial Mill Flats screen 2 - 168
Axial Mill Frame screen 2 - 153
Axial Mill Slot screen 2 - 160
Hole cycle 2 - 61
Radial Holes Drill Operation screen 2 - 115
Radial Lettering screen 2 - 110
Radial Mill Circle screen 2 - 90
Radial Mill Contour screen 2 - 71
Radial Mill Frame screen 2 - 96
Radial Slot screen 2 - 103

TOOL field
Axial Holes Drill Operation screen 2 - 179
Axial Lettering screen 2 - 175
Axial Mill Circle screen 2 - 147
Axial Mill Contour screen 2 - 128
Axial Mill Flats screen 2 - 168
Axial Mill Frame screen 2 - 153
Axial Mill Slot screen 2 - 160
Cutoff block 2 - 63
Grooving 2 - 52
Hole cycle 2 - 59
Profile Start screen 2 - 12
Radial Holes Drill Operation screen 2 - 115
Radial Lettering screen 2 - 110
Radial Mill Circle 2 - 90
Radial Mill Contour 2 - 71
Radial Mill Frame screen 2 - 95
Radial Slot screen 2 - 103
Threading 2 - 38

Tool number 2 - 4

TOOL OFFSETS field
Axial Holes Drill Operation screen 2 - 179
Axial Lettering screen 2 - 175
Axial Mill Circle screen 2 - 147
Axial Mill Contour screen 2 - 128
Axial Mill Flats screen 2 - 168
Axial Mill Frame screen 2 - 153
Axial Mill Slot screen 2 - 160
Profile Start screen 2 - 12
Radial Holes Drill Operation screen 2 - 115
Radial Lettering screen 2 - 110
Radial Mill Circle 2 - 90
Radial Mill Contour 2 - 71
Radial Mill Frame screen 2 - 95
Radial Slot screen 2 - 103
Threading 2 - 38

TOOL OFFSETS number 2 - 4

TOOL TYPE field
Hole cycle 2 - 59

TPI field
Axial Holes Drill Operation screen 2 - 181
Hole cycle 2 - 60
Radial Holes Drill Operation screen 2 - 120

Transfer description 1 - 5
TRANSFER F5 softkey 2 - 207
Turn 2 - 26
rough operation rule 2 - 14
TURN ALLOWANCE field
Finishing 2 - 53
Profile Start 2 - 25
Turn Element 2 - 28
TURN F3 softkey, element 2 - 26
turn finish all faces first 2 - 18
turn finish along profile 2 - 18
TURN FINISH F2 softkey, Profile screen 2 - 13
TURN OFF COOLANT field 2 - 7
TURN ROUGH F1 softkey, Profile screen 2 - 13
turret
home position 2 - 4
move to the defined tool change position 2 - 4
move to the defined X, Z position 2 - 8, 2 - 194
move to the home position 2 - 8, 2 - 194
movement, User Defined Tool Change Position field, Program Parameters screen 2 - 194
not move 2 - 8, 2 - 194

U
units of measure viii
USE C CLAMP checkbox
Axial Bolt Circle screen
Rectangular Coordinates 2 - 186
Axial Bolt Circle screen, Polar Coordinates 2 - 185
Axial Locations screen
Polar Coordinates 2 - 183
Rectangular Coordinates 2 - 184
Radial Holes Locations screen
Linear Coordinates 2 - 122
Rotary Coordinates 2 - 122
Radial Holes Loop Around C-Axis screen 2 - 123

Use Safe Rapid Point checkbox
Cutoff block 2 - 63
Grooving cycle 2 - 53
Threading cycle 2 - 39

USER DEFINED TOOL CHANGE POSITION field, Program Parameters screen 2 -
V
VERTICAL LEAD IN field
Thread cycle 2 - 42, 2 - 44, 2 - 46, 2 - 48, 2 - 49
VERTICAL LEAD OUT field
Thread cycle 2 - 42, 2 - 44, 2 - 46, 2 - 48, 2 - 50

W
W coordinate 2 - 5
W Offset (Tailstock or Sub-spindle) checkbox
Change Part Setup 2 - 196
W OFFSET (Tailstock or Sub-spindle) field
Change Part Setup 2 - 196
W POSITION, Cutoff Block screen 2 - 63
WALL ALLOWANCE field
Axial Mill Circle screen 2 - 149
Axial Mill Contour screen 2 - 130
Axial Mill Flats screen 2 - 169
Axial Mill Frame screen 2 - 155
Axial Mill Slot screen 2 - 161
Radial Mill Circle screen 2 - 91
Radial Mill Contour screen 2 - 73
Radial Mill Frame screen 2 - 97
Radial Mill Slot screen 2 - 103
WHERE field, Position data block 2 - 4, 2 - 8
WIDTH COMPENSATION field
Cutoff block 2 - 64
WIDTH field
Axial Lettering screen 2 - 177
Radial Lettering screen 2 - 112
WRITE TABLE TO FILE F2 softkey
Part Setup Work Offsets 2 - 197

X
X (RAD) or (DIA) field
Cutoff Block Advanced Pecking tab 2 - 65
User Defined Tool Change Position field,
Program Parameters screen 2 - 194
X Axis Order 2 - 6
X BOTTOM (DIA) or (RAD) field
Grooving 2 - 55
Radial Mill Circle screen
Rotary Coordinates 2 - 93
Radial Mill Contour screen
Rotary Coordinates 2 - 76
Radial Mill Frame screen
Rotary Coordinates 2 - 101
Radial Slot screen
Rotary Coordinates 2 - 107
X BOTTOM field
Radial Holes Drill Operation screen 2 - 115
X CENTER (D) or (R) field
Arc Element screen 2 - 32
Blend Arc Element screen 2 - 35
X CENTER field
Axial Bolt Circle screen
Rectangular Coordinates 2 - 166
Axial Mill Circle screen
Rectangular Coordinates 2 - 152
Axial Mill Contour Arc Segment screen
Rectangular Coordinates 2 - 141
Axial Mill Contour Blend Arc segment
Rectangular Coordinates 2 - 145
X CLEARANCE field
Threading 2 - 41, 2 - 43, 2 - 45, 2 - 47, 2 - 49
X coordinate 2 - 5, 2 - 8
X CORNER field
Axial Mill Frame screen
Rectangular Coordinates 2 - 159
X END (D) or (R) field
Arc Element screen 2 - 32
Blend Arc Element screen 2 - 35
Chamfer Element screen 2 - 37
Cutoff block 2 - 63
Face Element screen 2 - 29
Taper Element screen 2 - 31
Threading Cycle 2 - 49
Turn Element screen 2 - 28
X END field
Axial Mill Contour Arc Segment screen
Rectangular Coordinates 2 - 141
Axial Mill Contour Blend Arc segment
Rectangular Coordinates 2 - 144
Axial Mill Contour Line Segment screen
Rectangular Coordinates 2 - 137
Axial Mill Slot screen
Rectangular Coordinates 2 - 165
X field
Axial Locations screen
Rectangular Coordinates 2 - 184
X LENGTH field
Axial Mill Frame screen
Rectangular Coordinates 2 - 159
X OFFSET checkbox 2 - 195
manufacturer's allowable offset 2 - 58
X OFFSET field 2 - 59
Change Part Setup 2 - 195
X RAPID (D) or (R) field, Profile Start screen

2 - 24

X RAPID field

Grooving cycle 2 - 53
Threading cycle 2 - 39

X REF LOCATION field

Axial Lettering screen 2 - 176

X START (D) or (R) field

Profile Start 2 - 25

X START (DIA) or (RAD) field

Arc Element screen 2 - 33
Blend Arc Element screen 2 - 35
Chamfer Element screen 2 - 37
Cutoff Block 2 - 63
Face Element screen 2 - 30
Grooving 2 - 54
Profile Element 2 - 31
Radial Mill Circle screen

Rotary Coordinates 2 - 93
Radial Mill Contour screen

Rotary Coordinates 2 - 76
Radial Mill Frame screen

Rotary Coordinates 2 - 100
Radial Slot screen

Rotary Coordinates 2 - 107
Taper Element screen 2 - 31
Threading Cycle 2 - 49
Turn Element screen 2 - 28

X START field

Axial Mill Contour Arc Segment screen

Rectangular Coordinates 2 - 141
Axial Mill Contour Blend Arc segment

Rectangular Coordinates 2 - 144
Axial Mill Contour Line Segment screen

Rectangular Coordinates 2 - 137
Axial Mill Flats screen

Rectangular Coordinates 2 - 174
Axial Mill Slot screen

Rectangular Coordinates 2 - 165
Radial Holes Drill Operation screen 2 - 115

X TAPER (DIA) field

Threading Cycle 2 - 44, 2 - 48

X CENTER field

Radial Mill Circle screen

Linear Coordinates 2 - 94
Radial Mill Contour Arc Segment screen

Linear Coordinates 2 - 84
Radial Mill Contour Blend Arc Segment screen

Linear Coordinates 2 - 88

X CORNER field

Radial Mill Frame screen

Linear Coordinates 2 - 101

X' END field

Radial Mill Contour Arc Segment screen

Linear Coordinates 2 - 84
Radial Mill Contour Blend Arc Segment screen

Linear Coordinates 2 - 87
Radial Mill Contour Line Segment screen

Linear Coordinates 2 - 80
Radial Slot screen

Linear Coordinates 2 - 107

X' field

Radial Holes Locations screen

Linear Coordinates 2 - 122

X' LENGTH field

Radial Mill Frame screen

Rotary Coordinates 2 - 101

X' REF LOCATION field

Radial Lettering screen 2 - 111

X' REFERENCE field

Radial Lettering screen 2 - 112

X' START field

Axial Mill Contour screen

Rectangular Coordinates 2 - 134
Radial Mill Contour Arc Segment screen

Linear Coordinates 2 - 84
Radial Mill Contour Blend Arc Segment screen

Linear Coordinates 2 - 87
Radial Mill Contour Line Segment screen

Linear Coordinates 2 - 80
Radial Slot screen

Linear Coordinates 2 - 107

Y

Y CENTER field

Axial Bolt Circle screen

Rectangular Coordinates 2 - 186
Axial Mill Circle screen

Rectangular Coordinates 2 - 152
Axial Mill Contour Arc Segment screen

Rectangular Coordinates 2 - 141
Axial Mill Contour Blend Arc segment

Rectangular Coordinates 2 - 145

Y coordinate 2 - 5

Y CORNER field

Axial Mill Frame screen

Rectangular Coordinates 2 - 159

Y END field

Axial Mill Contour Arc Segment screen

Rectangular Coordinates 2 - 141
Axial Mill Contour Blend Arc segment

Rectangular Coordinates 2 - 144
Axial Mill Contour Line Segment screen
Rectangular Coordinates 2 - 137
Axial Mill Slot screen
Rectangular Coordinates 2 - 165
Y field
Axial Locations screen
Rectangular Coordinates 2 - 184
Y LENGTH field
Axial Mill Frame screen
Rectangular Coordinates 2 - 159
Y Offset checkbox
Change Part Setup 2 - 196
Y OFFSET field
Change Part Setup 2 - 196
Y REFERENCE field
Axial Lettering screen 2 - 177
Y START field
Axial Mill Contour Arc Segment screen
Rectangular Coordinates 2 - 141
Axial Mill Contour Blend Arc segment
Rectangular Coordinates 2 - 144
Axial Mill Contour Line Segment screen
Rectangular Coordinates 2 - 137
Axial Mill Flats screen
Rectangular Coordinates 2 - 174
Axial Mill Slot screen
Rectangular Coordinates 2 - 165
Y' CENTER field
Radial Mill Circle screen
Linear Coordinates 2 - 94
Radial Mill Contour Arc Segment screen
Linear Coordinates 2 - 84
Radial Mill Contour Blend Arc Segment screen
Linear Coordinates 2 - 88
Y' CORNER field
Radial Mill Frame screen
Rotary Coordinates 2 - 101
Y' END field
Radial Mill Contour Arc Segment screen
Linear Coordinates 2 - 84
Radial Mill Contour Blend Arc Segment screen
Linear Coordinates 2 - 87
Radial Mill Contour Line Segment screen
Linear Coordinates 2 - 80
Radial Slot screen
Linear Coordinates 2 - 107
Y' field
Radial Holes Locations screen
Linear Coordinates 2 - 122
Y' LENGTH field
Radial Mill Frame screen
Rotary Coordinates 2 - 101
Y' REF LOCATION field
Radial Lettering screen 2 - 111
Y' REFERENCE field
Radial Lettering screen 2 - 112
Y' START field
Axial Mill Contour screen
Rectangular Coordinates 2 - 134
Radial Mill Contour Arc Segment screen
Linear Coordinates 2 - 84
Radial Mill Contour Blend Arc Segment screen
Linear Coordinates 2 - 87
Radial Mill Contour Line Segment screen
Linear Coordinates 2 - 80
Radial Mill Contour screen
Linear Coordinates 2 - 77
Radial Slot screen
Linear Coordinates 2 - 107

Z
Z Axis Order 2 - 6
Z BOTTOM field
Axial Holes Drill Operation screen 2 - 179
Axial Mill Circle screen
Polar Coordinates 2 - 151
Rectangular Coordinates 2 - 152
Axial Mill Contour screen
Polar Coordinates 2 - 133
Rectangular Coordinates 2 - 134
Axial Mill Flats screen
Polar Coordinates 2 - 172
Rectangular Coordinates 2 - 174
Axial Mill Frame screen
Polar Coordinates 2 - 158
Rectangular Coordinates 2 - 159
Axial Mill Slot screen
Polar Coordinates 2 - 165
Rectangular Coordinates 2 - 167
Grooving 2 - 56
Hole cycle 2 - 59
Z CENTER field
Arc Element screen 2 - 32
Blend Arc Element screen 2 - 35
Radial Mill Circle screen
Rotary Coordinates 2 - 93
Radial Mill Contour Arc Segment screen
Rotary Coordinates 2 - 83
Radial Mill Contour Blend Arc Segment screen
Rotary Coordinates 2 - 87
Z CLEARANCE field
Threading 2 - 41, 2 - 43, 2 - 45, 2 - 47, 2 - 49
Z coordinate 2 - 5, 2 - 8
  where the turret should move, User Defined Tool Change Position area in Program Parameters screen 2 - 194

Z CORNER field
  Radial Mill Frame screen
    Rotary Coordinates 2 - 100

Z END field
  Arc Element screen 2 - 32
  Blend Arc Element screen 2 - 35
  Chamfer Element screen 2 - 37
  Face Element screen 2 - 29
  Radial Mill Contour Arc Segment screen
    Rotary Coordinates 2 - 83
  Radial Mill Contour Blend Arc Segment screen
    Rotary Coordinates 2 - 86
  Radial Mill Contour Line Segment screen
    Rotary Coordinates 2 - 79
  Radial Slot screen
    Rotary Coordinates 2 - 105
  Taper Element screen 2 - 31
  Threading 2 - 41, 2 - 43, 2 - 45, 2 - 47
  Turn Element screen 2 - 28

Z feedrate
  Hole cycle 2 - 61, 2 - 116, 2 - 180

Z field
  Radial Holes Locations screen
    Rotary Coordinates 2 - 122

Z LENGTH field
  Radial Mill Frame screen
    Rotary Coordinates 2 - 100

Z OFFSET field
  Change Part Setup 2 - 196
  Radial Holes Loop Around C-Axis screen 2 - 123

Z RAPID field
  Grooving cycle 2 - 53
  Threading cycle 2 - 39

Z RAPID field, Profile Start 2 - 24

Z START field
  Arc Element screen 2 - 33
  Axial Holes Drill Operation screen 2 - 179
  Axial Mill Circle screen
    Polar Coordinates 2 - 151
    Rectangular Coordinates 2 - 152
  Axial Mill Contour screen
    Polar Coordinates 2 - 133
    Rectangular Coordinates 2 - 134
  Axial Mill Flats screen
    Polar Coordinates 2 - 172
    Rectangular Coordinates 2 - 174
  Axial Mill Frame screen
    Polar Coordinates 2 - 158
    Rectangular Coordinates 2 - 159
  Axial Mill Slot screen
    Polar Coordinates 2 - 165
    Rectangular Coordinates 2 - 167
  Blend Arc Element screen 2 - 35
  Chamfer Element screen 2 - 37
  Cutoff Block 2 - 63
  Face Element screen 2 - 30
  Grooving 2 - 54
  Hole cycle 2 - 59
  Profile Start 2 - 25
  Radial Mill Contour Arc Segment screen
    Rotary Coordinates 2 - 83
  Radial Mill Contour Blend Arc Segment screen
    Rotary Coordinates 2 - 86
  Radial Mill Contour Line Segment screen
    Rotary Coordinates 2 - 79
  Radial Slot screen
    Rotary Coordinates 2 - 76
  Radial Slot screen
    Rotary Coordinates 2 - 105
  Taper Element screen 2 - 31
  Threading 2 - 41, 2 - 43, 2 - 45, 2 - 47, 2 - 49
  Turn Element screen 2 - 28

Z' BOTTOM field
  Radial Lettering screen 2 - 111
  Radial Mill Circle screen
    Linear Coordinates 2 - 94
  Radial Mill Contour screen
    Linear Coordinates 2 - 77
  Radial Mill Frame screen
    Rotary Coordinates 2 - 102
  Radial Slot screen
    Linear Coordinates 2 - 109

Z' START field
  Radial Mill Circle screen
    Linear Coordinates 2 - 94
  Radial Mill Contour screen
    Linear Coordinates 2 - 77
  Radial Mill Frame screen
    Rotary Coordinates 2 - 102
  Radial Slot screen
    Linear Coordinates 2 - 109

ZIG-ZAG field
  Axial Mill Flats screen 2 - 169
  Axial Mill Slot screen 2 - 161
  Radial Slot screen 2 - 104