HURCO®

WINMAX MILL
CONVERSATIONAL
PART PROGRAMMING

Dual Screen and Max Consoles
for Hurco Machining Centers

HURCO

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

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

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

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:

TABLE OF CONTENTS

LIST OF FIGURES

USING THIS MANUAL

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

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

Sample Screens

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

WinMax Version

v β 2006.11.16

Softkeys

PART SETUP
F1

TOOL REVIEW
F2

PART PROGRAMMING
F3

PROGRAM PARAMETERS
F4

IMPORT FUNCTIONS
F5

COPY AND CHANGE BLOCKS
F6

ERASE FUNCTIONS
F7

PROGRAM MANAGER
F8

Prompts
Select a softkey.

Error/Status area

Status Bar
PatternMirror2.hd5   INCH   2:08 PM
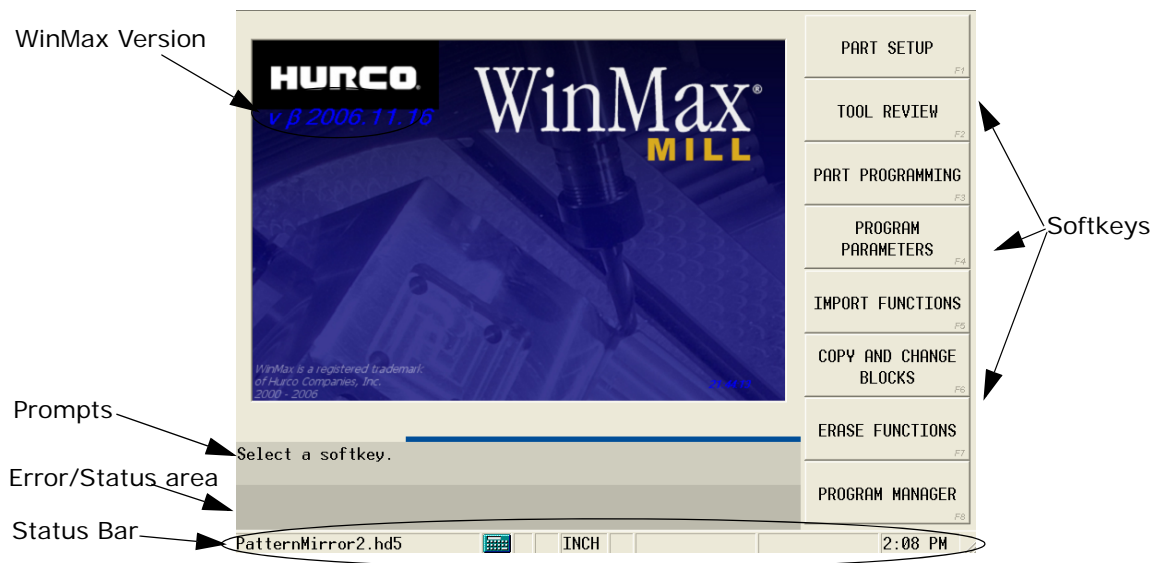
Softkeys

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

Screen Areas

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

Data Entry

The data entry area is located on the opposite side of the screen from the softkeys.

Available softkeys may change even when the text and data entry area does not.

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

Prompts and Error/Status Area

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

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

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

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

Status Bar

The status bar contains

- The name of the open, selected program.

- A calculator icon—select the icon to display a working, on-screen calculator.

- Units of measure (Inch or Millimeters)—select the units of measure in the status bar to toggle between Inch and Metric.

- Programming mode (R for Radius; D for Diameter)—select the programming mode in the status bar to toggle between Radius and Diameter.

- A yellow icon—indicates the feed hold is on when visible.

- A red icon—indicates the Emergency Stop button has been pressed when visible.

- A keyboard icon—select the icon to display a working on-screen keyboard.

- The current time.

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

Console Buttons and Keys

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

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

Using the Touchscreen

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

Printing

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

Printing to a Post Script printer provides the best results.

Icons

This manual may contain the following icons:

Caution/Warning

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

Hints and Tricks

Useful suggestions that show creative uses of the WinMax features.

Important

Ensures proper operation of the machine and control.

Troubleshooting

Steps that can be taken to solve potential problems.

Where can we go from here?

Lists several possible options the operator can take.

Table of Contents

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

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

USING AND PRINTING THE HELP

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

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

- To view the on-screen Help directly on a Hurco control, select either the Help console button or the F console key followed by the 1 key (F1).

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

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

Using the On-screen Help

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

- Buttons in the upper left-hand corner of the Help screen are used to move through Help topics and print screens.

  - Use the **Hide** button to hide the navigation pane.

  - Use the **Back** button to return to the previous Help screen.

  - Use the **Print** button to print the current dispalyed Help topic, if a printer is attached and configured. See *Printing the Help* for more information about printing.

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

- Use the **Contents** tab for a list of information sorted by subject:

  1. Select the "+" to expand the topic and view sub-topics.

  2. Select the topic to display it.

- Use the **Index** tab to show the Help index:

  1. Quickly scroll to an index topic by typing the topic in the box at the top of the index.

  2. Select a topic and the Display button to view the topic.

- Use the **Search** tab to search the Help for a word or phrase:

  1. Type the search word(s) into the text box at the top of the pane.

2. Select the List Topics button. A list of topics that contain the search word(s) is displayed.

3. Select a topic and the Display button to view that topic.

- Use the **Favorites** tab to save Help topics for quick access:

  1. Select the Add button at the bottom of the pane to add the current topic.

  2. Select a topic from the Favorites list, and select the Display button to view it.

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

Printing the Help

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

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

2. Select the DISK OPERATIONS *F7* softkey.

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

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

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

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

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

4. Highlight the PDF file(s) in the right-hand pane, and select the COPY *F2* softkey.

5. Ensure that your media is loaded (either a floppy disk in the disk drive or a USB memory device in the USB port), and navigate to the proper location in the left-hand pane of the DISK OPERATIONS screen (either the floppy drive A: or the USB port E:). Highlight the desired location.

6. Place the cursor in the right-hand pane and select the PASTE *F3* softkey to paste the PDF file(s) to the desired location.

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

HURCO CONVERSATIONAL OVERVIEW

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

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

These topics are discussed in the Overview section:

Part Programming

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

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

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

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

Data blocks can be managed with the following programming operations:

- Access softkeys to create a new data block or edit an existing data block in the Input screen.

- View a list summarizing a part program's data blocks on the Program Review screen. See *Review Mode* in *Getting Started with WinMax Mill* for more information.

- View the current program's tool list on the Tool Review screen. Refer to *Part Program Tool Review* in *Getting Started with WinMax Mill* for more information.

- Use the Insert Block Before softkey to Insert a new data block between two sequentially numbered blocks. The data blocks following the inserted block automatically renumber.

- Holes blocks contain operations, and Mill Contour blocks contain segments.

Part Programming softkeys access the following options:

- **Position**—Insert instructions to move the tool away from the part (or fixture), or to stop a program. A Position block is most often used to move the table to an X-Y location and is normally used at the end of the program and at any time the tool must be moved to the Z Top position of the Safety Work Region. See *Position Data Block,* on page 5 - 94 for more information.

- **Holes**—Select Drill, Tap, Bore and Ream drilling operations, and Back Spotface and Bolt Circle hole drilling. The locations of these holes on the part can be specified using the Locations operation, and the holes can also be programmed in a Bolt Circle pattern. See *Holes Data Block,* on page 3 - 62 for more information.

- **Milling**—Program milling operations. See also *Lines/Arcs Data Blocks,* on page 2 - 50, *Circle Data Block,* on page 2 - 25, *Frame Data Block,* on page 2 - 28, *Mill Face,* on page 2 - 23, *Ellipse Data Block,* on page 2 - 27, *HD3 Lettering,* on page 2 - 39, *Mill True-Type Lettering,* on page 2 - 41, and *Swept Surface,* on page 2 - 46. Special milling blocks Triangle, Diamond 1

Face, Diamond 2 Faces, and Hexagon are accessed with the Special ➔ softkey. See *Milling Operations,* on page 2 - 21 for more information.

- **Patterns—**Repeat or modify a sequence of one or more data blocks. A program sequence can be repeated in a rectangular, linear, angular, or rotated pattern or at specified locations. This operation is also used to scale or mirror a programmed part. See *Patterns Operations,* on page 4 - 83 for more information.

- **Miscellaneous—**Access these softkey functions. Refer to *Temporary Parameter Change,* on page 1 - 6 for details.

  - Graphics Off

  - Graphics On

  - Change Parameters

  - Change Part Setup

  - Machine Function

  - Probe Part Setup

  - Lube Cycle

  - Comment

  - Insert

  - Tool Monitoring (Probing)

  - Part Inspection

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

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

- **Exit—**Returns to the previous screen.

Data Blocks

Use the following basic programming operations when creating data blocks:

- Create a new data block or edit an existing data block through the Input screen.

- Insert a new data block between two sequentially numbered blocks. Data blocks following the inserted block will be automatically renumbered.

- Delete an existing data block. The data blocks following the deleted block renumber automatically.

- View a list of the part program's data blocks on the Review Program screen.

- Copy, edit, or delete information in a data block using the Block Editor screen.

- Data blocks within a part program contain specific machining information:

    - Operation type (position, holes, milling, patterns).

    - Tool number to be used for the operation.

    - Dimension coordinates relative to X, Y, and Z axes.

    - Feedrate at which the axes move.

    - Locations of operations on the part.

Segments

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

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

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

Data Block Creation and Navigation

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

Select Tool from List

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

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

2. Locate the desired tool and select to highlight.

3. Select the SELECT TOOL *F1* softkey, which enters the tool into the data block or segment.

   ⇨ If Tool Type Checking is enabled, only tools that are suitable for the programmed operation can be selected.

   ⇨ Check the Display Tool Notes checkbox to view the Notes for each tool. Notes can be entered in Part Program Tool Review when the Tool & Material Library option is installed.

Temporary Parameter Change

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

1. Display the data block for which the parameter change will first be used.

2. Select the **Insert Block Before** softkey. The New Block screen appears.

3. Select the **Miscellaneous** softkey. Choose the Change Parameter Softkey.

See *Change Parameters,* on page 5 - 95 and *Change Part Setup,* on page 5 - 98 for more information.

HURCO

Automatic Calculations

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

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

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

1. Use the arrow keys to move the cursor to the field where the automatic calculation is needed.

2. Press the BACKSPACE key. The defined value is cleared from the program and the system-calculated dimension is displayed (or the field is blank waiting for additional data to be entered).

3. In some cases, if you need the system to recalculate the field, you may need to clear other fields that were automatically calculated. This is usually the case if the system calculates erroneous data indicated by "ERR" preceding the value.

Line Segments

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

- If the X End and Y End coordinates are entered, the system calculates the XY length and the XY angle values.

- The XY Angle is the angle of the line segment (from the start point to the end point), measured counterclockwise from the 3 o'clock position.
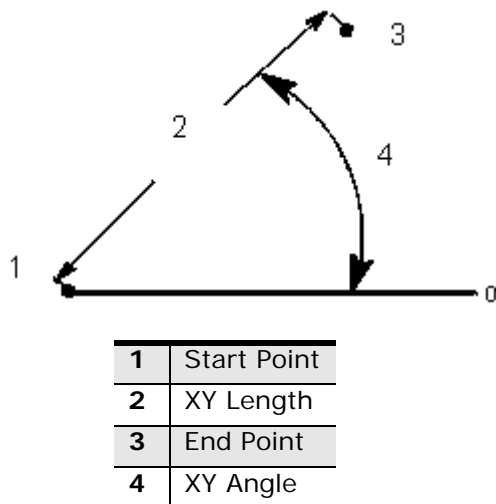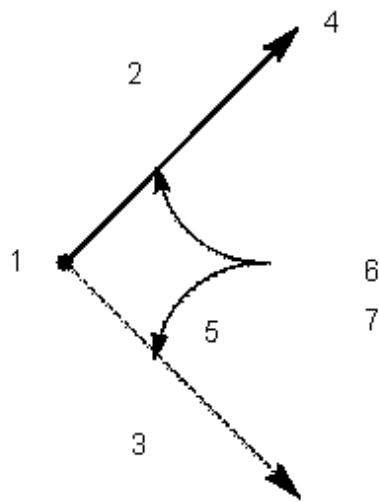

1	Start Point
2	XY Length
3	End Point
4	XY Angle

Figure 1–1. Length, Angle, and End Point Relationship

- If both end points are unknown but the XY Length and XY Angle fields are programmed, the system calculates the values for the X End and Y End fields.

- If one end point coordinate and the XY Angle field are programmed, the system calculates the values for the unknown end point and the XY Length fields.

- If one end point coordinate and the XY Length field are programmed, the system calculates the values for the unknown end point and the XY Angle fields. However, unless the XY Angle is known, there are two possible solutions for the unknown end point, and the correct one must be determined for the program.

- When two possible solutions exist, the "Another end point exists" message appears and one of the two possible solutions appears in the unknown field.

1	Known Start Point
2	Solution #1
3	Solution #2
4	X Known
5	XY Length Known
6	Y Unknown
7	XY Angle Unknown

Figure 1–2. Line Segment with Unknown Y End and XY Angle

Arc Segments

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

- The center points plus the start points or end points provide the arc radius.

- The start points and end points plus the radius provide the two possible center points.

- Either of the end points and the center point provide the value of the other end point and the radius.

- When a known center point, start point, or end point and radius are provided, an unknown center point is provided.
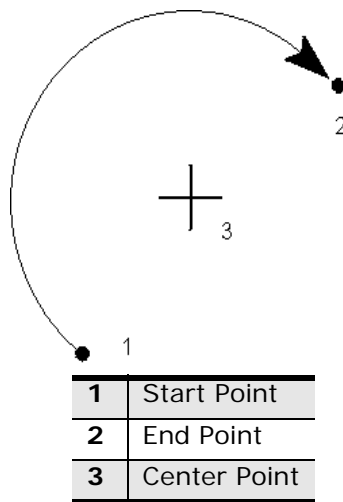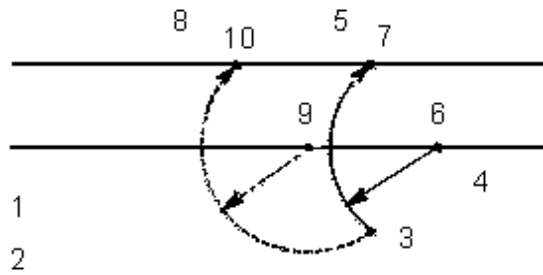

1	Start Point
2	End Point
3	Center Point

Figure 1–3. Coordinates of a Clockwise Arc

1	Known Y End and Y Center
2	Unknown X End and X Center
3	Known X Start and Y Start
4	Known Radius
5	Solution #1
6	Center Point #1
7	End Point #1
8	Solution #2
9	Center Point #2
10	End Point #2

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

Helix

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

- The Z End and Sweep Angle provide data to calculate the Lead.

- The Sweep Angle is used to calculate the Z End.

- The X End, Y End, and Lead values provide the Z End.

- The Z End and Lead values provide the X End, Y End, and Sweep Angle values.

- The X End, Y End, X Center, and Y Center values supply the Radius.

- The Sweep Angle and either the X or Y end point provide the unknown X End or Y End.

Cutter Compensation

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

Cutter Compensation allows you to choose the side of the contour the tool should begin cutting. The programmed tool automatically follows the finished contour of the part when cutter compensation is selected. Without cutter compensation, the centerline of the programmed tool follows the print line.
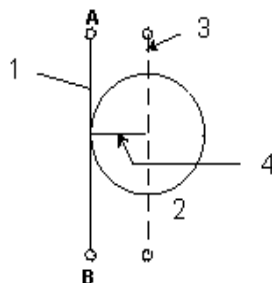
1	Programmed path
2	Cutting tool
3	Path compensating for cutter radius
4	Cutter radius = amount of cutter compensation offset

Figure 1–5. Cutter Compensation

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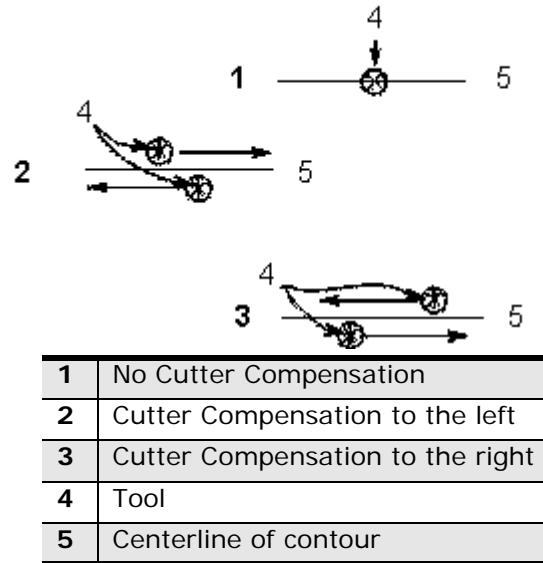

1	No Cutter Compensation
2	Cutter Compensation to the left
3	Cutter Compensation to the right
4	Tool
5	Centerline of contour

Figure 1–6. Tool Paths for Cutter Compensation

The following are the types of cutter compensation used for Mill Contours such as Lines, Arcs, Blend Arcs, Helices, True-Type Lettering, and 3D Blend Arcs:

- **On—**Locates the center of the tool on the programmed contour of the frame.

- **Left—**Performs climb milling. Refer to *Climb Milling (Left),* on page 1 - 15.

- **Right—**Performs conventional milling. Refer to *Conventional Milling (Right),* on page 1 - 15.

- **Profile Left**—Removes material from a contour for climb milling.

- **Profile Right—**Removes material from a contour for conventional milling.

The following are the types of cutter compensation used for milling Circles, Frames, and Ellipses: (Refer to *Cutter Compensation,* on page 1 - 12 where indicated.)

- **On—**Locates the center of the tool on the programmed contour.

- **Inside—**Causes the tool to enter the part inside the contour and blend into the programmed contour using a 180° arc. Cutter compensation is automatically employed, and the edge of the tool remains inside a programmed contour.

- **Outside—**Causes the tool to enter the part outside the programmed contour and follow the outside contour. Cutter compensation is automatically employed, and the edge of the tool remains outside the programmed contour.

- **Inside Tangent—**Causes the tool to enter the part inside and tangent to the programmed contour. Cutter compensation is automatically employed, and the edge of the tool remains inside and tangent to the programmed contour. The direction the tool travels depends upon the Milling Direction. The tool is

withdrawn from the part inside and tangent to the programmed contour.

- **Outside Tangent**—Causes the tool to enter the part outside and tangent to the programmed contour. [Cutter compensation](#) is automatically employed, and the edge of the tool remains outside and tangent to the programmed contour. The direction the tool travels depends upon the Milling Direction. The tool is withdrawn from the part outside and tangent to the programmed contour.

- **Pocket Boundary**—Causes the tool to cut the part around the programmed boundary and avoid any programmed islands or pockets.

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

        - **Outward**—Causes the tool to begin cutting operations in the center region of the pocket and cut outward to the edge of the programmed boundary. Outward is used only for Circle and Frame data blocks without islands.

        - **Inward**—Causes the tool to cut in from the outside of the defined boundary, avoiding the defined islands.

- **Pocket Island**—Defines islands within pockets on a part. As many islands as desired may be defined (subject to available memory), but all must fit within the defined boundary and must allow the tool to completely define the island. A Pocket Island cannot follow an Outward Pocket Boundary.

The order in which segments are programmed determines the direction the tool moves from the start point. Setting Right (conventional milling) or Left (climb milling) in the Milling Parameters overrides the direction of the programmed path.

Unless the Blend Offset is set for 0.0 in Milling Parameters, the system automatically creates Lead In and Lead Out arcs for closed contours. Refer to *Lead In/Out Moves,* on page 1 - 18.

Climb Milling (Left)

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

During climb milling, the spindle turns in a clockwise direction. The tool is on the left-hand side of the cut as it travels away from the operator.

The advantages of using climb milling are as follows:

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

Conventional Milling (Right)

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

The advantages of using conventional milling are as follows:

- The chip thickness starts at zero, causing less impact on the cutting teeth. This is ideal for setups that are not very rigid.
- The backlash in older machines is greatly diminished.

⇨ The programming sequence of Lines and Arcs segments determines the cutting direction. However, when programming Mill Frames, Circles and Ellipses, the cutting direction is determined by the values entered in Program Parameters.

Profile Left and Right

Profile Left removes material inside a closed contour. A mill contour with the same X–Y coordinates for both the end point and start point is called a closed contour. When climb milling a closed contour, select Profile Left. This computes the start and end points of the tool path so the tool does not cut into the last or first segments of the contour.

Profile Right removes material from outside of a programmed closed contour. When conventional milling a closed contour, select Profile Right.

⇨ All contours must be closed when using Profile Left and Profile Right. If contours are not closed, the tool moves in a direct line from the end point to the start point at the current feedrate.

Do not program the start point in a corner of a closed contour.

Cutter Compensation Lookahead

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

⇨ Whether a contour is closed or open is based on the programmed part surface and not the tool path. The contour is closed if the start segment and the last segment have the same X and Y position. If the start and last segments do not have the same X and Y position, the contour is open.

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

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

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

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

⇨ *CAL* to the left of the data field indicates a calculated value; *USR* indicates a user-defined value.

Maximum Offset

The Maximum Offset field appears when either Profile Left or Profile Right is selected. (Refer to *Profile Left and Right,* on page 1 - 16.) Maximum Offset is the radius of the largest inscribed circle minus the tool radius. Manually calculate the value and enter it into this field.

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

If a .500" diameter (.25" radius) End Mill is used to machine the part illustrated below (a circle with a 1" radius), the value for the Maximum Offset field is 0.75".
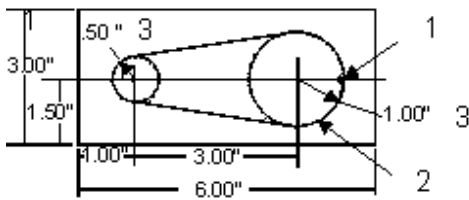
1	Starting point for the tool
2	Largest inscribed circle in this contour
3	Radius

Figure 1–7. Determining Maximum Offset

If the tool diameter is changed (i.e., the cutter is sharpened to a smaller diameter), the Maximum Offset value must be manually re-calculated and this new value programmed into Segment 0 of the Mill Contour's Start block. Refer to *Mill Contour,* on page 2 - 51. The Tool diameter is programmed in Tool Setup and cannot be changed in Part Programming. See *Tool Setup* in *Getting Started with WinMax Mill* for more information.

Lead In/Out Moves

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

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

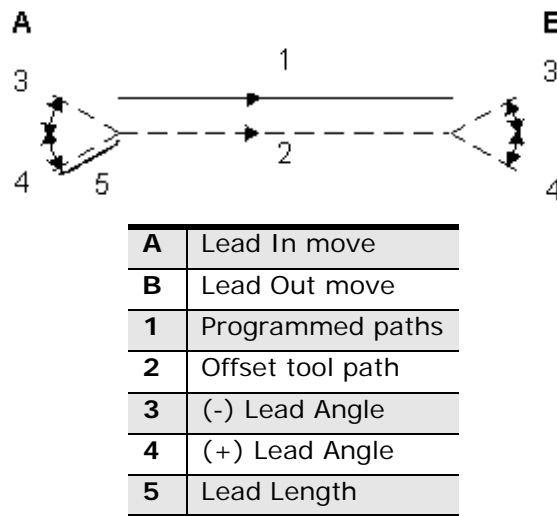Toolpath Graphics will not draw the Lead Angle or Lead Length motion.

A	Lead In move
B	Lead Out move
1	Programmed paths
2	Offset tool path
3	(-) Lead Angle
4	(+) Lead Angle
5	Lead Length

Figure 1–8. Lead In/Out Moves

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

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

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

Lead Angle

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

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

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

Lead Length

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

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

MILLING OPERATIONS

These topics are discussed in this section:

Creating a Milling Data Block

1. Begin a Mill Frame block by displaying the Input screen and selecting the **Part Programming** softkey or icon. Refer to *Frame Data Block,* on page 2 - 28.

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

3. When the New Block screen appears, select the **Milling** softkey to display the milling options shown in the seven softkey selections.

4. Select the milling operation and the first data block screen appears. Use **Frame** for this example.

5. Type in the number of the tool that will be used to mill the frame and press the ENTER key to tell the system to read and remember the information in the field.

   • The tool numbers and their descriptions are entered in Tool Setup. If the tool information has not been entered or another tool should be added to the program, select the **Tool Setup** softkey to update the tool descriptions for this program.

   • Entering a number in the Tool field pulls information from the tool setup and automatically loads it into fields on this screen. Note that the Mill Feed and Speed (RPM) fields are filled in based on the information about the tool. For additional assistance to the operator, the system also displays the selected tool's diameter and type as read-only fields.

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

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

The fields on the Mill Frame screen are described in detail in Frame. Refer to *Frame Data Block*, on page 2 - 28.

Mill Face

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

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

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

1	Blend Offset
2	X Unidirectional
3	Y Unidirectional
4	X Bidirectional
5	Y Bidirectional
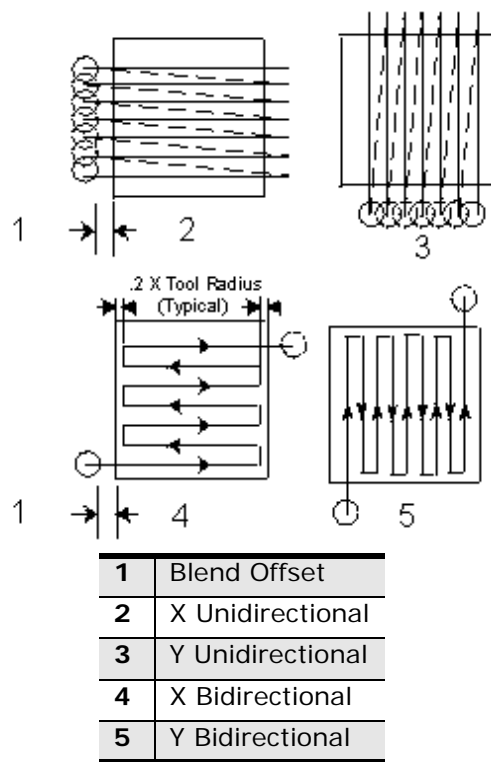
Figure 2–1. Face Milling

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

The Face Milling fields are defined as follows:

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

- **Tool**—Identifies the tool number for this data block and enters that tool's Diameter and Tool Type on this screen.

- **Milling Type**—Defines the type of cutting operation. There are four types of face milling:

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

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

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

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

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

- **X Length** and **Y Length**—The distance and direction of the rectangle along the X axis and Y axis, measured from the reference corner.

  - If the reference corner is at the left side of the rectangle, the X Length is a positive (+) dimension. If the reference corner is at the right side, the X Length is negative (-).

  - The Y Length value is positive (+) if the reference corner is at the lower left or lower right of the rectangular area. It is negative (-) if the reference corner is at the top left or top right of the rectangle.

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

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

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

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

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

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

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

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

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

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

Circle Data Block

Use the Circle data block to mill circles.

To create a Circle data block:

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

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

The Mill Circle fields are defined as follows:

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

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

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

- **Milling Type**—Identifies whether the system should automatically compensate for the diameter of the tool when determining the tool path. With the cursor on the Milling Type field, select different types of Cutter Compensation and pocket type with the softkeys and pull down menu choice. Refer to *Cutter Compensation,* on page 1 - 12.

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

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

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

- **Lead Length**—Identifies the Lead Length for this Mill Circle. See *Lead In/Out Moves,* on page 1 - 18. This field only appears for NC posted programs. This field is only available when Milling Type is set to Inside or Outside. Lead In/Out Moves will be ignored in the post unless Enable Blend Moves is set to Yes and the Display Apt Fields in Editor field on the Post Processor Configuration screen is set to Yes.

- **Lead Angle**—Identifies the Lead Angle for this Mill Circle. See *Lead In/Out Moves,* on page 1 - 18. This field only appears for NC posted programs. This field is only available when Milling Type is set to Inside or Outside. Lead In/Out Moves will be ignored in the post unless Enable Blend Moves is set to Yes and the Display Apt Fields in Editor field on the Post Processor Configuration screen is set to Yes.

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

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

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

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

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

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

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

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

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

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

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

- **Finish Type**—Contains the type entered during Tool Setup. See *Getting*

Started with WinMax Mill for more information.

Ellipse Data Block

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

The Ellipse fields are defined as follows:

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

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

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

- **Milling Type**—Identifies whether the system should automatically compensate for the diameter of the tool when determining the tool path. With the cursor in the Milling Type field, select different types of Cutter Compensation and pocket type with the softkeys and pull down menu. Refer to *Cutter Compensation,* on page 1 - 12.

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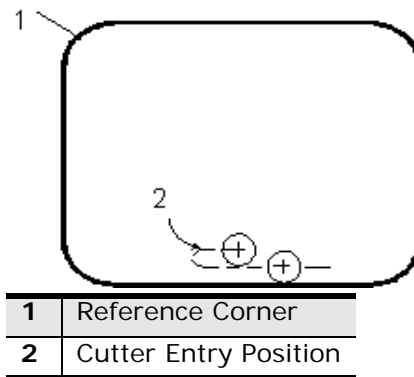
- **X Center** and **Y Center**—Identify the X and Y coordinates for the center point of the ellipse.

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

  The major axis is the longer or wider axis and can be either the X or Y axis. The minor axis is the smaller axis. Program an ellipse with the major and minor axes non-parallel to the machine using a Pattern Loop Rotate routine and rotating the ellipse after it has been programmed. Refer to *Loop Rotate,* on page 4 - 87.

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

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

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

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

- **Mill Feed**—Identifies the X-Y feedrate. The value initially displayed has been

calculated by the system and can be retained or changed to a different value. See *Getting Started with WinMax Mill* for more information.

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

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

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

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

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

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

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

Frame Data Block

Use the Frame function to create frames to surround geometry. The Frame fields describe how and where the frame will be milled on the workpiece. To access the Mill Frame screen, on the New Block screen select the **Milling** softkey then select the **Frame** softkey that appears.

Here are the definitions for these fields:

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

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

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

- **Milling Type—**Identifies whether the system should automatically compensate for the diameter of the tool when determining the tool path. With the cursor in the Milling Type field, select different types of Cutter Compensation or pocket type with the softkeys and pull-down menu. Refer to *Cutter Compensation,* on page 1 - 12

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

1	Reference Corner
2	Cutter Entry Position

Figure 2–2. Inside Mill Frame Operation

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

- **X Length** and **Y Length**—The length of the rectangle along the X axis or Y axis, measured from the reference corner. This value is positive or negative relative to the reference corner. If the reference corner is at the left side of the rectangle, the X or Y Length is a positive (+) dimension. If the reference corner is at the right side, this field's value is negative (-).

1	Reference Corner
2	X Length
3	Y Length
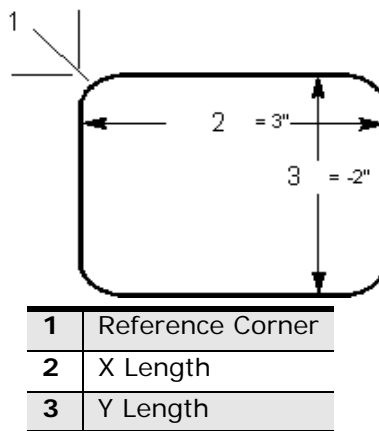
Figure 2–3. X and Y Length Directional Measurement

- **Corner Radius**—Identifies the radius of the reference corner. It is the same for all four corners.

- **Lead Length**—Identifies the Lead Length for this Mill Frame. See *Lead In/ Out Moves,* on page 1 - 18. This field only appears for NC posted programs. This field is only available when Milling Type is set to Inside or Outside. Lead In/Out Moves will be ignored in the post unless Enable Blend Moves is set to Yes and the Display Apt Fields in Editor field on the Post Processor Configuration screen is set to Yes.

- **Lead Angle**—Identifies the Lead Angle for this Mill Frame. See *Lead In/Out Moves,* on page 1 - 18. This field only appears for NC posted programs. This field is only available when Milling Type is set to Inside or Outside. Lead In/ Out Moves will be ignored in the post unless Enable Blend Moves is set to Yes and the Display Apt Fields in Editor field on the Post Processor Configuration

screen is set to Yes.

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

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

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

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

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

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

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

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

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

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

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

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

Finish Tool for a Mill Frame, Mill Circle or Mill Contour

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

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

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

2. If the finish tool is different from the Tool specified in Part Setup:

    • If the finish tool is a ball end mill, face mill, or flat end mill, the Spindle speed value entered on the Tool Setup screen will be used as the finish speed.

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

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

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

Mill Triangle Data Block

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

1	Open Face
2	Orientation = 0
3	Relief Corner
4	Radius
5	Centerpoint
6	Point 1
7	Point 2
8	Point 3
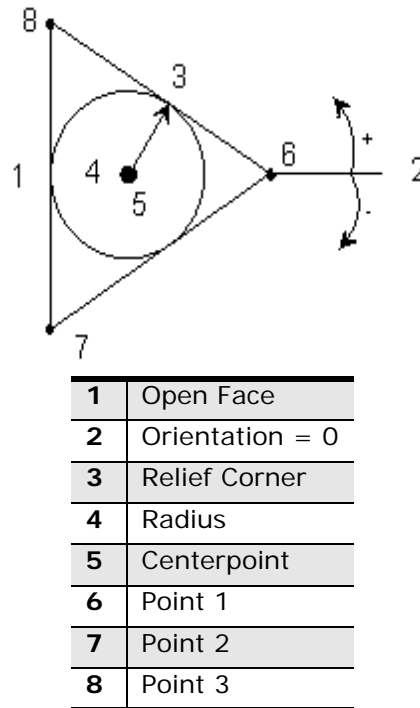
Figure 2–4. Triangle Programming Diagram

Fields in the Mill Triangle Data Block are:

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

- **X Center**—identifies the X-axis coordinate for the center of the triangle.

- **Y Center**—identifies the Y-axis coordinate for the center of the triangle.

- **Radius**—identifies the value for the radius of the inscribed circle.

- **Orientation**—identifies the angle to rotate the pocket about the center.

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

- **Z Bottom**—identifies the point at which the mill feed rate begins. The value initially displayed has been calculated by the control and can be retained or changed to a different value.

- **Relief 1**—identifies the direction of corner relief cut and the depth past the corner of the two walls. Provides the option of a right, center, or left relief cut (or none) in the corner at point 1.

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

- **Milling Type—**defines cutter motion using these options:

  - **Inside** - cuts just the non-open faces of the pocket, including the relief cuts.

  - **Inside 2 Passes** - cuts the same as Inside, except this selection uses a roughing pass and a finishing pass. During roughing, material is left for the finish pass.

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

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

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

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

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

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

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

- **Plunge Feed—**Defines the feed rate for the tool moving from Z Start to Z Bottom. Also used when plunging down in Z between peck depth passes.

Mill Diamond Data Block

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

The Diamond 1 Face has one open face and the option of a right, center, or left relief cut in the corners at point 1 and point 2 as shown below:
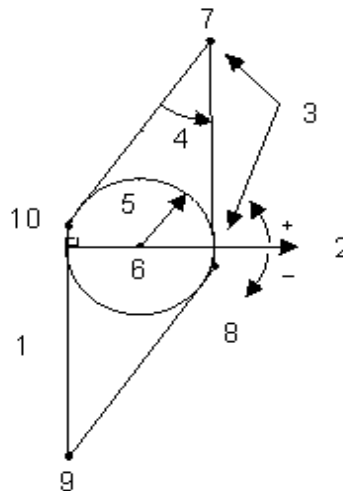
1	Open Face
2	Orientation = 0
3	Relief Corners
4	Angle
5	Radius
6	Centerpoint
7	Point 1
8	Point 2
9	Point 3
10	Point 4

Figure 2–5. Diamond 1 Face Diagram

The Diamond 2 Faces pocket has two open faces and the option of a right, center, or left relief cut in the corner at point 1 as shown below:
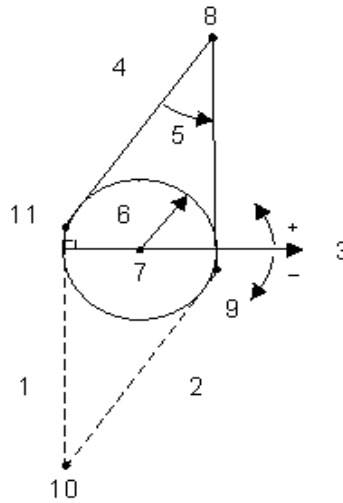
1	Open Face
2	Open Face
3	Orientation = 0
4	Relief Corners
5	Angle
6	Radius
7	Centerpoint
8	Point 1
9	Point 2
10	Point 3
11	Point 4

Figure 2–6. Diamond 2 Faces Diagram

Fields are:

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

- **X Center**—identifies the X-axis coordinate for the center of the triangle.

- **Y Center**—identifies the Y-axis coordinate for the center of the triangle.

- **Radius**—identifies the value for the radius of the inscribed circle of the diamond.

- **Orientation**—identifies the angle to rotate the pocket about the center.

- **Shape Angle**—identifies the angle formed by points 1 and 3 defining the shape with a range from 20° to 160°.

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

- **Z Bottom**—identifies the point at which the mill feedrate begins.

- **Relief 1**—identifies the direction of corner relief cut and the depth past the corner of the two walls. Provides the option of a right, center, or left relief cut (or none) in the corner at point 1.

- **Relief 2**—for Diamond 1 Face only, provides the option of a right, center, or left relief cut (or none) in the corner at point 2.

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

- **Milling Type—**defines cutter motion using these options:

  - **Inside** - cuts just the non-open faces of the pocket, including the relief cuts.

  - **Inside 2 Passes** - cuts the same as Inside, except this selection uses a roughing pass and a finishing pass. During roughing, material is left for the finish pass.

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

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

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

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

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

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

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

- **Plunge Feed**—Defines the feed rate for the tool moving from Z Start to Z Bottom. Also used when plunging down in Z between peck depth passes.

Mill Hexagon Data Block

This pocket shape has six sides and every other angle equal. In the diagram below, the shape has two open faces and the option of a relief cut in the corners at point 1, point 2, and point 3, and the option of a face relief distance between points 0 and 1 and points 3 and 4.
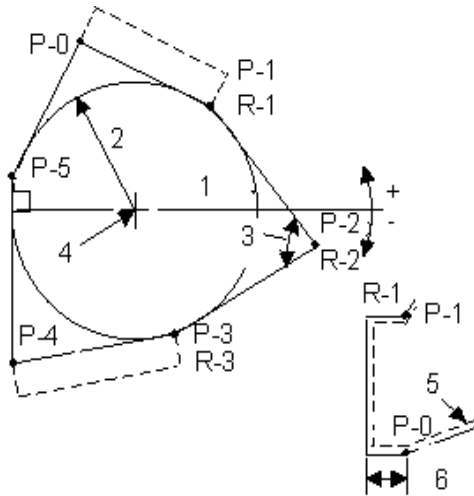
1	Orientation = 0
2	Radius
3	Shape Angle
4	X and Y Centerpoint
5	Tool Path
6	Relief Distance
P-0	Point 0
P-1	Point 1
P-2	Point 2
P-3	Point 3
P-4	Point 4
P-5	Point 5
R-1	Relief 1
R-2	Relief 2
R-3	Relief 3

Figure 2–7. Hexagon Diagram

Fields are:

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

- **X Center**—identifies the X-axis coordinate for the center of the triangle.

- **Y Center**—identifies the Y-axis coordinate for the center of the triangle.

- **Radius**—identifies the value for the radius of the inscribed circle.

- **Orientation**—identifies the angle to rotate the pocket about the center.

- **Shape Angle**—identifies the angle formed by points 1 and 3 defining the shape with a range from 20° to 160°.

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

- **Z Bottom**—identifies the point at which the mill feedrate begins.

- **Relief 1**—identifies the direction of corner relief cut and the depth past the corner of the two walls. Provides the option of a right, center, or left relief cut (or none) in the corner at point 1, or face relief distance between points 0 and 1.

- **Relief 2**—provides the option of a right, center, or left relief cut (or none) in the corner at point 2.

- **Relief 3**—provides the option of a right, center, or left relief cut (or none) in the corner at point 3 or the option of a face relief distance between points 3 and 4.

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

- **Milling Type—**defines cutter motion using these options:

  - **Inside** - cuts just the non-open faces of the pocket, including the relief cuts.

  - **Inside 2 Passes** - cuts the same as Inside, except this selection uses a roughing pass and a finishing pass. During roughing, material is left for the finish pass.

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

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

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

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

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

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

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

- **Plunge Feed**—Defines the feed rate for the tool moving from Z Start to Z Bottom. Also used when plunging down in Z between peck depth passes.

Lettering Data Blocks

The lettering data blocks are used to mill text onto a part. These are the lettering options:

HD3 Lettering

The HD3 Lettering function mills block lettering into a part.

Cutter Compensation is not available for HD3 Lettering. Milling is performed using the centerline of the tool. Refer to *Cutter Compensation*, on page 1 - 12.

The fields are defined as follows:

- **X Ref Location**—specifies Start, End, or Center of the character for the X reference point. Default is Start. This field is active when the Lettering Reference Locations option is installed. Without the option, the X reference location for new blocks is always Start.

  - Start—the serial number is left-justified; the first character begins at the X reference point.

  - Center—the entire serial number is centered left and right of the X reference point.

  - End—the serial number is right-justified; the last character ends at the X reference point.

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

- **Y Ref Location**—specifies Bottom, Top, or Center of the character for the Y reference point. Default is Bottom. This field is active when the Lettering Reference Locations option is installed. Without the option, the Y reference location for new blocks is always Bottom.

  - Bottom—the bottom of the serial number is aligned with the Y reference point.

  - Top—the top of the serial number is aligned with the Y reference point.

  - Center—the serial number is centered above and below the Y reference point.

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

Y Reference

Top

Center

Bottom

Start   Center   End

X Reference

Figure 2–8. X Reference and Y Reference locations

- **Character Height—**Determines the height of the character set.

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

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

- **Text—**Contains characters to be milled into the part. This field will support up to 30 block characters.

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

- **Z Bottom—**Identifies the point at which the mill feed rate begins. The value initially displayed has been calculated by the control and can be retained or changed to a different value.

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

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

- **Mill Feed—**Defines the X-Y feedrate. See *Getting Started with WinMax Mill* for more information.

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

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

HD3 Serial Number Lettering

HD3 Serial Number Lettering mills a series of identical parts, each with a unique engraved serial number. See *HD3 Serial Number Lettering,* on page 4 - 1 for more information.

Mill True-Type Lettering

Mill True-Type Lettering fields are:

- **X Ref Location**—specifies Start, End, or Center of the character for the X reference point. Default is Start. This field is active when the Lettering Reference Locations option is installed. Without the option, the X reference location for new blocks is always Start.

    - Start—the serial number is left-justified; the first character begins at the X reference point.

    - Center—the entire serial number is centered left and right of the X reference point.

    - End—the serial number is right-justified; the last character ends at the X reference point.

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

- **Y Ref Location**—specifies Bottom, Top, or Center of the character for the Y reference point. Default is Bottom. This field is active when the Lettering Reference Locations option is installed. Without the option, the Y reference location for new blocks is always Bottom.

    - Bottom—the bottom of the serial number is aligned with the Y reference point.

    - Top—the top of the serial number is aligned with the Y reference point.

    - Center—the serial number is centered above and below the Y reference point.

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

Figure 2–9. X Reference and Y Reference locations

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

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

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

- **Z Bottom**—Z Bottom coordinate.

- **Orientation**—the angle about the Z axis to which the text should be rotated. The center of rotation is located at XY BASE.

- **Mapping**—specifies the Y (height) dimension of the lettering:

  - Body Only—defines the height of the body of the lettering. Ascenders and descenders are calculated automatically.

  - Descended—defines the height of the body and descenders of the lettering. Ascenders are calculated automatically.

  - Ascended—defines the height of the body and ascenders of the lettering. Descenders are calculated automatically.

  - Full Font—defines the height of the body, ascenders, and descenders of the lettering. No part of the lettering is calculated automatically.

1	Body only
2	Descended
3	Ascended
4	Full font
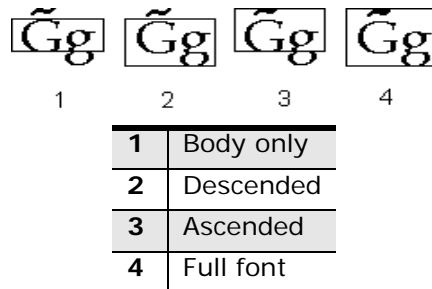
Figure 2–10. Y Mapping

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

Available Fonts for Mill True-Type Lettering

Arial (default)	Marlett
Arial Black	Microsoft Sans Serif
Comic Sans MS	Palatino Linotype
Courier New	Sylfaen
Euro Sign	Tahoma
Frame5 Font	Times New Roman
Franklin Gothic Medium	Trebuchet MS
Georgia	Verdana
Impact	Webdings
Lucida Console	Wingdings
Lucida Sans Unicode	ZWAdobeF

- **Text**—the text to be milled.

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

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

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

- **Mill Feed**—The X-Y feedrate. See *Getting Started with WinMax Mill* for more information.

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

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

Swept Surface

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

Swept Surface builds on Ultimax's 3D Mold option. The basic elements in Swept Surface programming are:

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

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

1. From a New Block screen, select the MILLING *F3* softkey.
2. Select the More *F7* softkey.
3. Select the SWEPT SURFACE *F4* softkey.
4. Use the drop-down list to select the type of contour in the TYPE field: Draw Profile, Draw Along, or Swept Surface.

   ⇨ A Swept Surface program block must contain all three types of contours: Draw Profile, Draw Along, and Swept Surface.

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

Draw Profile Contour

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

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

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

Draw Along Contour

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

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

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

Swept Surface

Swept Surface parameters establish the machining characteristics. Fields are:

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

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

- **TOOL**—specifies the tools to be used for roughing, finishing, pocketing; select from list or enter a number.

- **CUT DIRECTION**—controls the tool path while the part is machined. There are three choices for the CUT DIRECTION field:

  - **With Contour** - machines the contour using the profile contour originally programmed.

  - **Normal** - tool path follows the part at right angles to the profile contour.

  - **Spiral** - machines a continuous tool path resulting in a smoother surface.

- **BIDIRECTIONAL**—specifies the direction of the tool path while the part is being machined. There are two choices for the BIDIRECTIONAL field:

  - Yes - causes the tool to machine in both directions without retracting the tool until the entire contour is complete.

  - **No** - causes the tool to machine in one direction, based on the direction of the contour definition, while retracting between each pass.

| 1 | Yes |
| 2 | No |
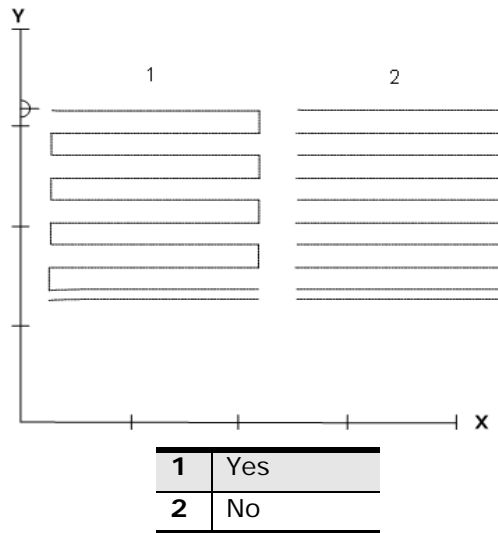
Figure 2–11. Bidirectional Field

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

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

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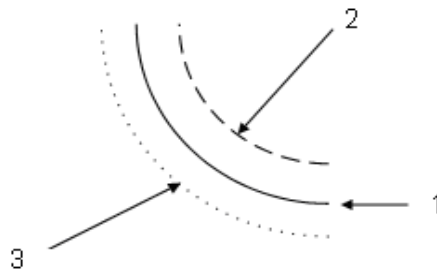
1	Print Dimension
2	+ Stock Allowance
3	- Stock Allowance

Figure 2–12. Stock Allowance

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

- **MIN Z**—limits the negative Z motion to the Centerline Z value when set to Yes.

- **STEP SIZE**—the distance on the surface between cutter passes. Ultimately, this dimension determines the surface finish of the part. A larger step size machines faster but leaves a rougher surface. A smaller step size machines more slowly but leaves a smoother surface. Step size significantly affects the drawing speed of the graphics screen.
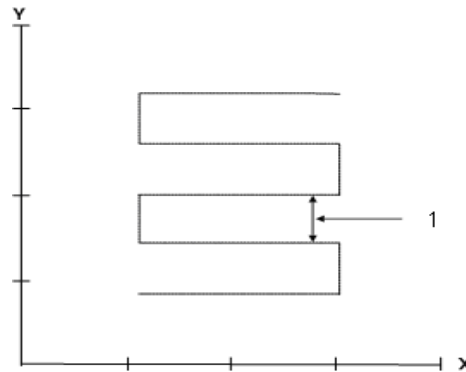
1	Step Size

Figure 2–13. Step Size Example

- **PECK DEPTH**—specifies the distance the tool will drill down into the part before stopping to clear out or break the chips. If used, this parameter is not usually larger than the diameter of the tool.

| A | Z Start |
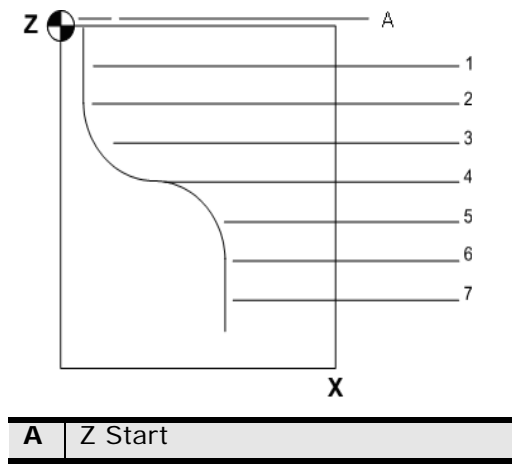
Figure 2–14. Peck Depth Example

- **PLUNGE FEED**—Z feedrate for plunge in inches per minute.
- **Z ROUGHING**—specifies if the Z roughing strategy be applied to the roughing pass.
- **MILLING TYPE**—specifies the pocket milling type: None, Island, Pocket, Island & Pocket.
- **POCKET TYPE**—specifies the pocketing type: inward or outward.
- **POCKET OVERLAP (%)**—specifies cutter overlap as a percent of cutter diameter.
- **ROUGH SFQ**—specifies Surface Finish Quality for roughing.
- **FINISH SFQ**—specifies Surface Finish Quality for finishing.
- **POCKET ROUGH SFQ**—specifies Surface Finish Quality for pocket roughing.
- **POCKET FINISH SFQ**—specifies Surface Finish Quality for pocket finishing.

Mill Slot

The Mill Slot block creates a line or arc shape of any width. The slot is defined by a line or an arc segment and a width; the width of slot is created above the line or arc segment.

Mill Slot is a WinMax Mill option; refer to *Mill Slot*, on page 9 - 1 of the WinMax Mill Options manual for more information.

Mill Polygon

The Mill Polygon data block mills a multi-sided contour with equal-length sides.

Mill Polygon is a WinMax Mill option; refer to *Mill Polygon*, on page 7 - 1 of the WinMax Mill Options manual for more information.

Thread Mill

The Thread Mill data block mills a part with internal or external threads.

Thread Mill is a WinMax Mill option; refer to *Thread Mill*, on page 10 - 1 of the WinMax Mill Options manual for more information.

HD3 Serial Number Lettering

HD3 Serial Number Lettering mills a sequence of unique serial numbers that increment each time the block is run.

HD3 Serial Number Lettering is a WinMax Mill Option; refer to *HD3 Serial Number Lettering*, on page 4 - 1 of the WinMax Mill Options manual for more information.

Lines/Arcs Data Blocks

Use segments and cutter compensation to program lines and arcs which create a Mill Contour. The Contour End block marks the end of the programmed contour. Refer to *Contour End*, on page 2 - 60. These are the types of segments used to create a Mill Contour:

- *Contour Line, on page 2 - 53*
- *Contour Arc, on page 2 - 54*
- *Contour Blend Arc, on page 2 - 55*
- *Helix, on page 2 - 57*

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

Mill Contour

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

The Mill Contour fields are defined as follows:

- **Block—**Identifies the block number for this Mill Contour data block. The system determines the number by the data block's position in the program.

- **Segment—**Identifies the Segments number within the Mill Contour operation. Refer to *Segments,* on page 1 - 4.

- **Tool—**Identifies the number of the tool used to cut the segments in this contour.

- **Finish Tool—**Identifies the tool number used for the finish pass. If a finish tool is selected, the first tool leaves the amount of material selected in the Finish Z field in Milling Parameters. The finish tool removes the remaining stock down to the programmed dimension.

- **Milling Type—**Identifies whether the system should automatically compensate for the diameter of the tool when determining tool path. With the cursor in the Milling Type field, select different types of Cutter Compensation or pocket type with the softkeys or drop-down list box. Refer to *Cutter Compensation,* on page 1 - 12.

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

- **Enable Blend Moves**—Appears when either Left or Right is selected in the Milling Types field. When this field is set to Yes, the Lead Length and Lead Angle fields will be used for Post Processing Part Programs if the Display Apt Fields in Editor field on the Post Processor Configuration screen is set to Yes. Refer to *Climb Milling (Left),* on page 1 - 15 and *Conventional Milling (Right),* on page 1 - 15.

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

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

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

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

- **Lead Length**—Identifies the Lead Length for this Mill Contour. See *Lead In/Out Moves,* on page 1 - 18. This field only appears for NC posted programs. This field is only available when Milling Type is set to Left or Right. Lead In/Out Moves will be ignored in the post unless Enable Blend Moves is set to Yes and the Display Apt Fields in Editor field on the Post Processor Configuration screen is set to Yes.

- **Lead Angle**—Identifies the Lead Angle for this Mill Contour. See *Lead In/Out Moves,* on page 1 - 18. This field only appears for NC posted programs. This field is only available when Milling Type is set to Left or Right. Lead In/Out Moves will be ignored in the post unless Enable Blend Moves is set to Yes and the Display Apt Fields in Editor field on the Post Processor Configuration screen is set to Yes.

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

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

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

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

- **Peck Depth**—Determines the distance the tool will drill down into the part before stopping to clear out or break the chips. If used, this parameter is usually not larger than the diameter of the tool.

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

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

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

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

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

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

Contour Line

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

Line segment fields are defined as follows:

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

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

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

- **XY Length**—Identifies the XY length. Refer to *Automatic Calculations,* on page 1 - 8 to see how entering data affects this field.

- **XY Angle**—Identifies the XY Angle or the angle of the line segment (from the start point to the end point), measured counterclockwise from the 3 o'clock position. Refer to *Automatic Calculations,* on page 1 - 8 to see how entering data affects this field.

- **Feed**—Identifies the feedrate the control will try to achieve while milling this segment. This value is carried forward from the Mill Contour screen's Mill Feed field. See also *Getting Started with WinMax Mill*.

- **X Start, Y Start,** and **Z Start**—Identify the X, Y, and Z starting points. The Start fields are carried forward from the previous segment's end points.

Contour Arc

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

The Arc fields are defined as follows:

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

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

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

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

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

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

- **Sweep Angle**—identifies the angular distance in degrees from the start point of the arc to the end point. The range is -360° to 360°. This field is available when the Arc Sweep Angle option is installed.

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

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

Contour Blend Arc

A blend arc is an arc that joins two other segments and is tangent to both. Use a blend arc to join two line segments, to join a line segment and an arc segment, or to join two arc segments. The segments to be joined must have a theoretical point of intersection.
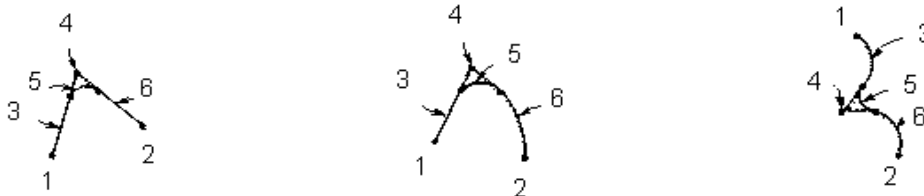
Two Lines Joined by a Blend Arc	Line and Arc Joined by a Blend Arc	Two Arcs Joined by a Blend Arc
1 X/Y Start	**1** X/Y Start	**1** X/Y Start
2 X/Y End	**2** X/Y End	**2** X/Y End
3 Segment 1 (Line)	**3** Segment 1 (Line)	**3** Segment 1 (Arc)
4 Segment 1 End/ Segment 3 Start (Point of Intersection)	**4** Segment 1 End/ Segment 3 Start (Point of Intersection)	**4** Segment 1 End/ Segment 3 Start (Point of Intersection)
5 Segment 2 (Blend Arc)	**5** Segment 2 (Blend Arc)	**5** Segment 2 (Blend Arc)
6 Segment 3 (Line)	**6** Segment 3 (Arc)	**6** Segment 3 (Arc)

Figure 2–15. Blend Arc Examples

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

The Contour Blend Arc fields are defined as follows:

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

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

- **Radius—**Identifies the radius of the arc.

- **Feed**—Identifies the feed value carried forward from the Mill Contour screen's Mill Feed field. See *Getting Started with WinMax Mill* for more information.

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

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

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

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

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

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

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

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

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

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

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

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

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

Helix

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

The Helix fields are defined as follows:

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

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

- **Direction—**Defines the direction of a helix from the starting point. There are two softkey choices when the cursor is on the Direction field.

    - **CW** - the helix will be cut in a clockwise direction.

    - **CCW** - the helix will be cut in a counterclockwise direction.

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

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

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

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

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

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

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

3D Arc

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

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

The following fields define a 3D Arc:

- **X, Y, and Z End**—define the end point for the arc segment. These fields are required when defining a 3D Arc.

- **X, Y, and Z Center**—define the center point for the arc segment. These fields are required when defining a 3D Arc.

- **X, Y, and Z Point**—define the plane of the arc when it is 180° or greater.

- **Feed**—rate from the previous segment. This value can be accepted or changed.

This diagram shows the relationships of the coordinates to each other:
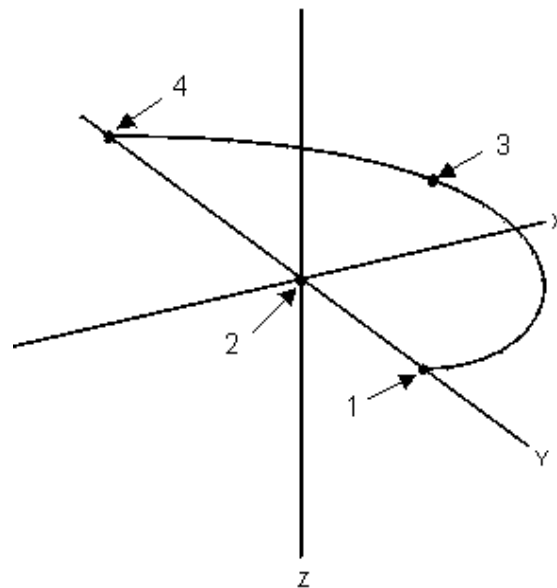
1	Start Point
2	Center Point
3	X, Y, Z Point
4	End Point

Figure 2–16. 180º 3D Arc

To calculate the centerline of cutter movement, remember these points. Refer to *Cutter Compensation,* on page 1 - 12 for information about cutter compensation.

- Depending on the axis being worked, X can be interchanged with Y or Z.

- The X and Z points must be calculated for an X-Z arc.

- The Y and Z points must be calculated for a Y-Z arc (all arcs travel through at least two axes).

- To calculate the centerline in the X axis for a ball-nosed end mill, use this formula:

$$X_a = \frac{R_i}{R}(X_s - X_c) + X_c$$

Here are the elements of this formula:

X_a = Actual centerline dimension of cutter in X axis

X_s = Arc reference starting point in the X axis

X_c = X center point

R_i = Radius of arc minus ½ cutter diameter

R = Radius of arc

Follow these *guidelines* when programming 3D arcs:

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

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

Contour End

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

The Contour End fields are defined as follows:

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

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

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

HOLES OPERATIONS

These topics are discussed in this section:

Holes Data Block

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

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

3. Select the **Holes** softkey. The Hole Operation softkey choices appear.

4. Select the appropriate softkey for the type of operation

5. Select the tool for the first operation in the data block.

   • The structure of a hole drilling data block is more complex than other conversational data blocks. To describe a hole drilling process in a data block, enter different types of instructions.

   • Each type of drilling instruction requires creating a separate operation within the data block.

6. Program the operation.

7. After completing an operation, place the cursor in the Operation field and the following softkeys are available:

   • **Next Hole Operation -** places a new Holes operation into the data block after the operation currently displayed on the screen.

   • **Delete Operation** - removes the currently displayed operation from the holes data block

   • **Insert Operation Before** - places another operation into the data block before the operation currently displayed on the screen.

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

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

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

   • **Delete Location** - remove the location coordinates of the hole location currently marked with the cursor.

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

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

Drill Overview

See also:

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

The diagram below shows the relationships of the field names to positions of the tool to the work piece:
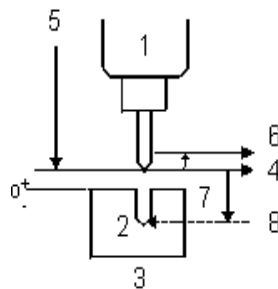
1	Head
2	Part
3	Table
4	Z Start Plane
5	Rapid Traverse
6	Retract Clearance Plane
7	Plunge Feed
8	Z Bottom

Figure 3–1. Reference Points Relative to Drill Operations

Be aware of the drill bits' characteristics. The system does not keep track of the type and length of flutes on the bits or the shapes of drills with pilots or other multiple-diameter drills.

This motion occurs in a drilling operation:

1. The table moves at the rapid traverse rate to the programmed X and Y dimensions.

2. The Z axis moves at the rapid traverse rate to the programmed retract clearance.

3. When these coordinates are reached, the Z axis moves at the rapid traverse rate to Z Start.

4. Then the Z axis moves at the programmed plunge feed rate until reaching the Z Bottom dimension.

5. The Z axis dwells the time specified and rapid traverses to the retract clearance.

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

When all values are programmed, the Bolt Circle and Locations softkeys are available. See *Bolt Circle,* on page 3 - 77 and *Locations,* on page 3 - 78 for more information.

- **Delete Operation—**Clears this programmed Drill operation.

- **Insert Operation Before—**Adds another Hole operation before the current one.

- **Exit—**Returns to the previous screen level.

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

Drill or Bore and Ream Operations

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

Select the Bore and Ream Operations softkey on the New Hole Operation screen to select the type of boring and reaming operation. Refer to *Bore and Ream Operations,* on page 3 - 71. The appropriate screen appears with fields describing the operation.

- **Block—**Identifies the block number for this operation.

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

- **Hole Operation Type** (not labeled) - allows you to select a different hole operation type from the drop-down list. You may also use the softkeys to select another hole operation type. Select the **More ➔** softkey to access more tool type softkeys.

   Drill Operations drill different types of holes. The Drill Operation types are *Drill,* on page 3 - 66, *Center Drill,* on page 3 - 66, *Counterbore,* on page 3 - 65, *Spotface,* on page 3 - 66, *Countersink,* on page 3 - 65, and *Gun Drill,* on page 3 - 67. The fields do not differ with the types of operation, except Gun Drill.

- **Tool—**Identifies the tool number for this data block.

- **Z Start—**Identifies the point where the spindle begins to rotate. The Z Start

field is the dimension where the Z axis plunge feed rate begins.

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

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

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

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

- **Peck Type**—Used only with Drill. Two softkey choices appear: Standard and Chip Breaker.

  - **Standard** - the tool retracts to the Z Start and then plunges to the peck clearance plane selected in Program Parameters or Change Parameters. See *Getting Started with WinMax Mill* for more information.

  - **Chip Breaker** - the tool dwells at each peck level before continuing to feed to the next peck level. (Dwell time is the time selected in Program Parameters or Change Parameters (Holes). See *Getting Started with WinMax Mill* for more information.

  - **Tool Diameter—**Contains the value entered during Tool Setup. he time selected in Program Parameters or Change Parameters (Holes). See *Getting Started with WinMax Mill* for more information.

  - **Tool Type—**Contains the type entered during Tool Setup. he time selected in Program Parameters or Change Parameters (Holes). See *Getting Started with WinMax Mill* for more information.

Counterbore

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

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

Countersink

Countersink enlarges the top end of a hole to a cone shape to accommodate the head of a flat or oval headed machine screw. The hole is enlarged so that when the screw is inserted, the screw head will be flush with or slightly below the top of the work piece surface.
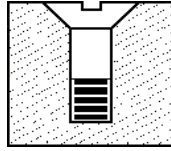
Figure 3–2. Countersunk screw head

Drill

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

Center Drill

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

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

Spotface

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

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

Gun Drill

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

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

1	Work Piece
2	Planned Path
3	Pre-drilled Starting Hole
4	Long tool moves into position with the spindle off to avoid whipping
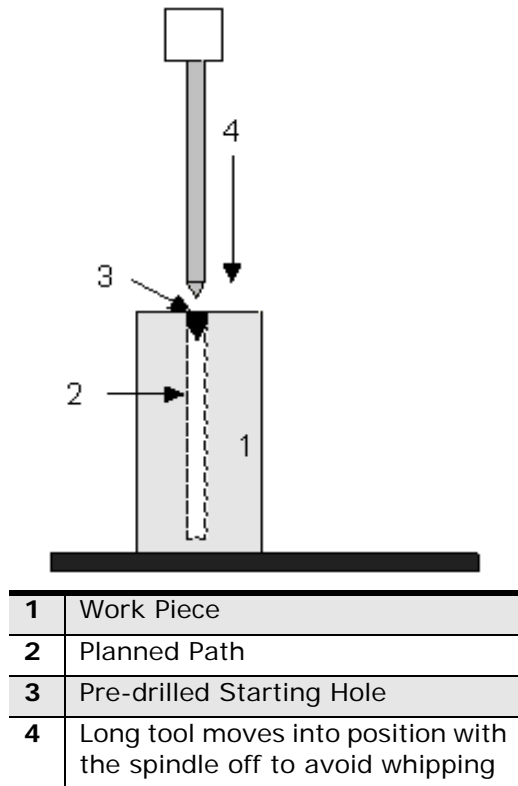
Figure 3–3. Gun Drilling

To use the Gun Drill features, perform these tasks:

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

Each Gun Drill data block must contain at least two operations: one describing the starting hole and one for the gun drill hole.

To create a Gun Drill data block, begin on a New data block screen and follow these general steps:

1. Press the Drill Operations softkey.

2. Create the first operation of the data block by describing the starting hole portion of the data block. Select either the Drill softkey or the Center Drill softkey. Type in the description of the starting hole into the fields displayed on the screen.

3. When all of the starting hole data have been entered, begin the second operation in this data block.

4. Press the Drill Operations softkey and then the Gun Drill softkey.

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

1	Work Piece
2	Move at Rapid Traverse Feedrate down to Z Top
3	Z Top
4	Z Top Feed—feedrate between Z Top and Z Start
5	Z Start—spindle begins to rotate at Speed (RPM)
6	Z Bottom
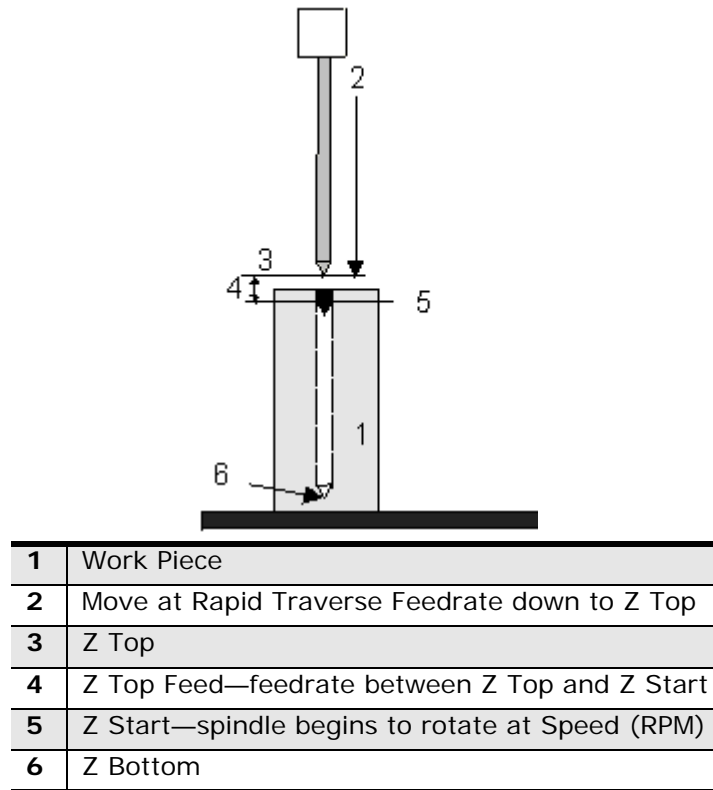
Figure 3–4. Gun Drill Positions

The Gun Drill fields are defined as follows:

- **Block**—Identifies the block number for this operation.

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

- **Tool**—Identifies the tool number for this data block and enters that tool's Diameter and Tool Type on this screen.

- **Z Top**—Identifies the starting point of the gun drill cycle. The tool moves rapidly to this point and then slowly moves into the pre-drilled starting hole.

- **Z Start**—Identifies the position inside the pre-drilled hole where the spindle begins to rotate.

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

- **Z Top Feed**—Identifies the rate at which the tool moves from the Z Top position to the Z Start position.

- **Plunge Feed**—Identifies the feedrate between the Z Start and the Z Bottom positions.

- **Speed (RPM)**—Defines the speed at which the spindle rotates to drill the hole. See also he time selected in Program Parameters or Change Parameters (Holes). See *Getting Started with WinMax Mill* for more information.

- **Spindle Stop**—Identifies the point at which the spindle stops rotating after drilling the hole. This field may be set to be the Z Bottom or the Z Start position.

- **Peck Depth**—Identifies the distance the tool drills down into the part before stopping to clear out or break the chips. If this field is used, it is usually not larger than the tool's diameter.

- **Peck Type**—Select whether the drill pulls out to the Z Start position after reaching Peck Depth (Standard) or whether the drill dwells at the Peck Depth (Chip Breaker).

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

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

Tap Operations

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

Tap and **Rigid Tap** softkeys appear.

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

The Tap/Rigid Tap fields are defined as follows:

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

    For imperial units, Plunge Feed = RPM / TPI.

    For metric units, Plunge Feed = RPM x Pitch.

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

Bore and Ream Operations

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

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

1	Shank
2	Blade depth adjustment dial
3	Replaceable cutter inserts
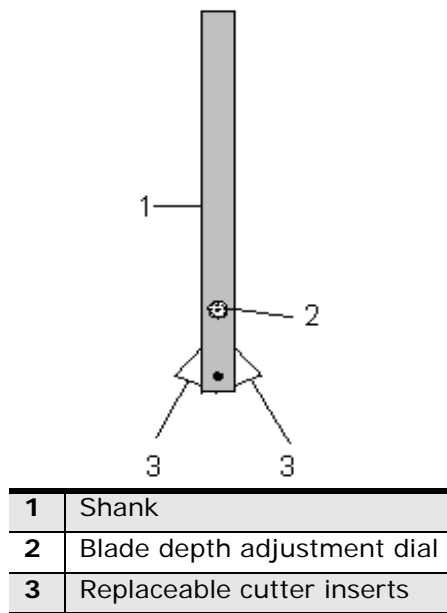
Figure 3–5. Two-Bladed Boring Bar with Adjustment Dial

Use a ream to size a previously drilled hole. A reaming tool has multiple flutes that run vertically along the shank of the tool. During reaming, the cutter follows the angle of the existing hole and cannot be used to straighten the angle of the hole.
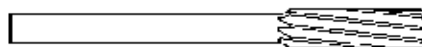
Figure 3–6. Reaming Tool

The Bore and Ream fields are defined as follows:

- **Block—**Identifies the block number for this operation.

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

- **Z Start—**identifies the point where the spindle begins to rotate. The Z Start field is the dimension where the Z axis plunge feed rate begins.

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

- **Tool—**Identifies the tool number for this data block.

- **Plunge Feed—**Identifies the feed rate between the Z Start and the Z Bottom positions.

- **Speed (RPM)—**Identifies the speed at which the spindle rotates to drill the hole, entered in Tool Setup. Entering a value here overrides the Tool Setup value for this data block.

Bore

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

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

Boring Bar

Shank

Blade depth adjustment dial

Replaceable blades
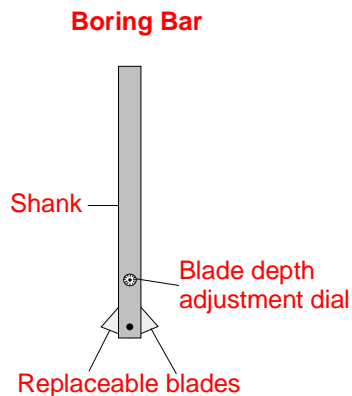
Figure 3–7. Two-bladed Boring Bar with Adjustment Dial

To access the Bore screen:

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

Ream

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

A *reaming tool* has multiple flutes that run vertically along the shank of the tool as shown in the following illustration:
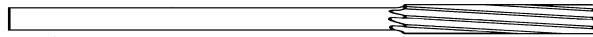
Figure 3–8. Reaming Tool

To access the Ream screen:

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

   ⇨ Pecks are not allowed with the reaming feature.

Bore Rapid

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

To access the Bore Rapid screen:

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

Bore Orient

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

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

To access the Bore Orient screen:

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

Ream Rapid

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

To access the Ream Rapid screen:

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

Back Spotface

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

This motion sequence occurs with Back Spotface:

1. Rapid to Z Start minus cutter offset and set the spindle speed to a plunge speed.

2. Move to Z Plunge minus cutter offset at a closing feedrate to close the cutter.

3. Rapid to Z Bottom minus cutter offset.

4. Dwell for a specified reverse dwell amount, reverse the spindle direction and ramp to cutting speed to open the cutter.

5. Move up to Z Depth minus cutter offset at a cutting feedrate.

6. Dwell for the drill dwell time (from program parameters).

7. Rapid down a Z Clearance amount.

8. Dwell for the Reverse Dwell time and reverse the spindle direction to plunge speed to close the cutter.

9. Move to Z Retract minus cutter offset at a closing feedrate to insure the cutter is closed.

10. Rapid out of the hole.

The Back Spotface fields are defined as follows:

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

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

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

- **Z Start—**Identifies the point above the part where the spindle begins to rotate.

- **Z Plunge—**Identifies the point within the hole to which the tool feeds before rapid movement to Z Bottom. This action assures that the cutter is fully closed, preventing damage to the tool and part.

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

- **Z Depth—**Identifies the depth to which the tool feeds

- **Z Clearance—**Determines the incremental distance to rapid the tool away from the cutting distance. At this point, the tool pauses and the spindle returns to its original direction to close the cutter.

- **Z Retract—**Determines the distance the tool feeds back up through the hole to be certain that the tool is properly closed before rapid movement to the Retract Clearance height.

- **Closing Feed—**Determines the feed at which the tool enters the hole at the top and bottom to be certain that the tool is closed.

- **Cutting Feed—**Determines the feed the tool uses while milling.

- **Plunge Speed—**Identifies the speed of the spindle with the tool closed while entering or exiting the hole.

- **Cutting Speed—**Identifies the speed of the spindle with the tool open.

- **Reverse Dwell—**Determines the number of seconds that the spindle dwells when it is reversing to give the tool time to open or close.

- **Tool Diameter—**Contains the value entered during Tool Setup.

- **Tool Type—**Contains the type entered during Tool Setup.

Bolt Circle

For each hole data block, you must include a Locations or a Bolt Circle operation. Refer to *Locations,* on page 3 - 78. A Bolt Circle operation executes a series of equally spaced holes in a common circle. This operation allows skipping holes in the bolt circle.

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

A	Start Angle
B	X Center, Y Center
C	Radius
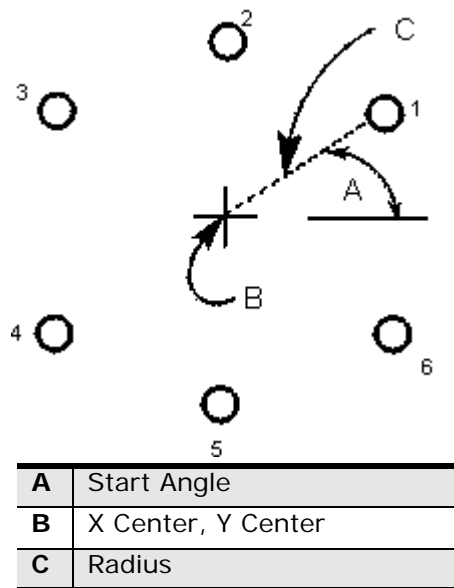
Figure 3–9. Bolt Circle Pattern

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

Move to the next Block or the next Operation by placing the cursor in the desired field and selecting the appropriate softkey (**Next Block**, **Next Hole Operation**, etc.). See also *Data Block Creation and Navigation,* on page 1 - 5.

The Bolt Circle fields are defined as follows:

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

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

- **Number of Holes**—Determines the number of holes in the pattern, including those skipped. Up to 250 holes can be programmed in one operation.

- **X Center** and **Y Center**—Identify the X and Y coordinates for the center of the Bolt Circle.

- **Radius**—Determines the radius of the circle between the coordinate center and the center of the cutter's starting point.

- **Start Angle**—Determines the angle of the circle measured counter-clockwise from the 3 o'clock position to the first hole. This angle may be specified to the nearest one-thousandth of a degree.

- **Skip List**—Enter up to 24 positions that should be skipped (not drilled). In the example above, there are 6 holes in the pattern. If you do not want to drill holes for positions 2 and 5, enter 2 and 5 in the Skip List field.

Locations

For each hole data block, you must include a location or a bolt circle operation. Refer to *Bolt Circle,* on page 3 - 77.

- All Locations programmed in a block containing a Bolt Circle are executed in addition to the Bolt Circle pattern.

- All operations programmed in a data block are performed at all locations specified in that block.

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

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

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

Move to the next Block or the next Operation by placing the cursor in the desired field and selecting the appropriate softkey (**Next Block**, **Next Hole Operation**, etc.). See also *Data Block Creation and Navigation,* on page 1 - 5.

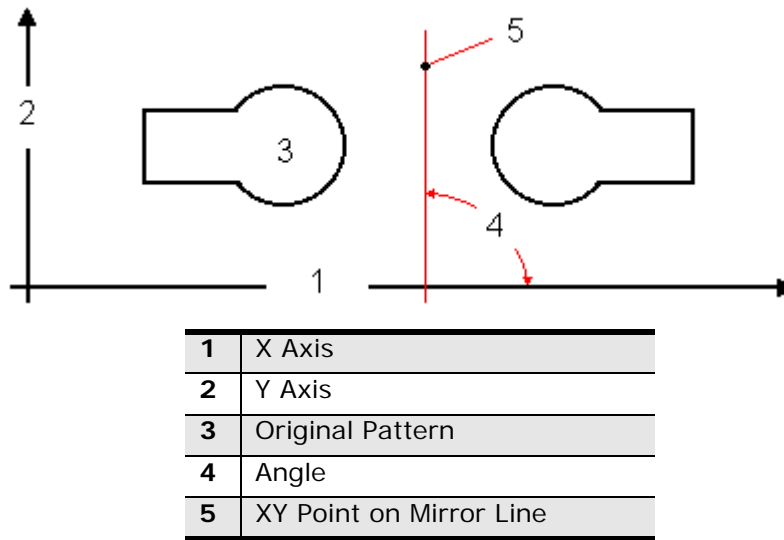The Holes Locations fields are defined as follows:

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

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

- **X** and **Y**—Identify the X and Y coordinates of the location.

Bolt Circle to Holes Locations or Pattern Locations Conversion

A Bolt Circle data block can be copied or converted to either a Holes Locations or Pattern Locations block. Place the cursor in a Bolt Circle field and the Copy and Convert softkeys appear:
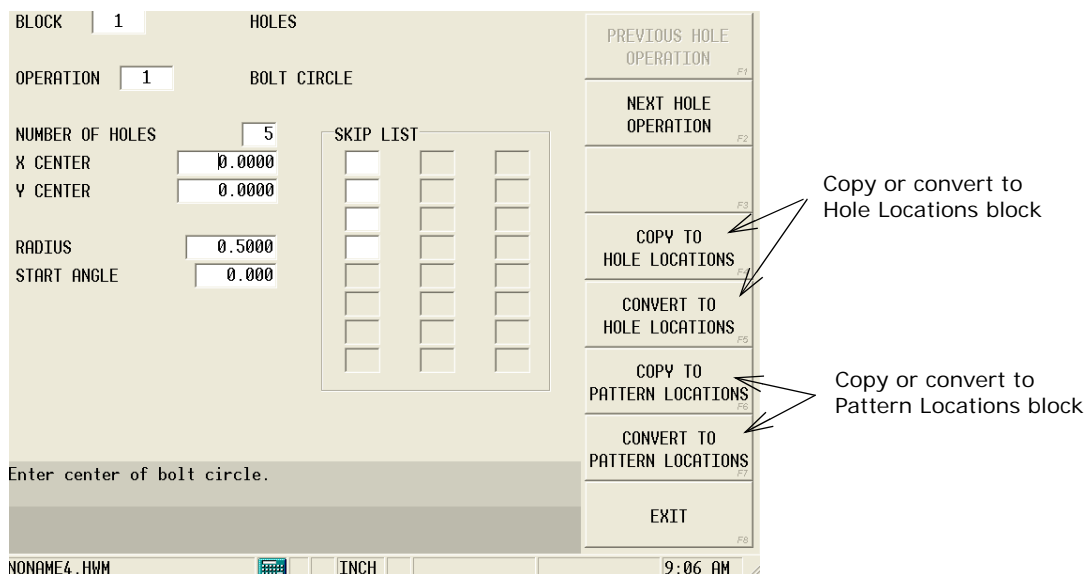
Figure 3–10. Bolt Circle Hole or Pattern Locations Copy and Convert softkeys

Softkeys are:

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

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

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

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

Holes to Pattern Locations Conversion

A Holes Locations data block can be copied or converted to a Pattern Locations block. Place the cursor in a field and the Copy and Convert softkeys appear:
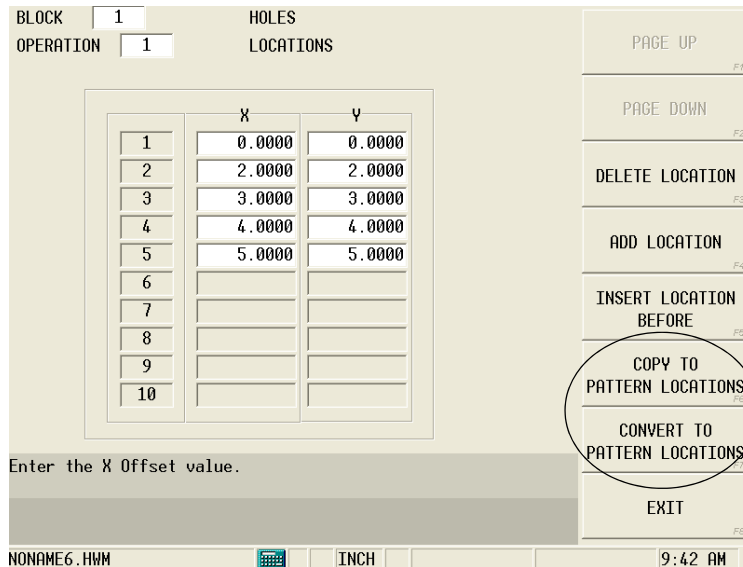
Figure 3–11. Holes to Pattern Locations softkeys

Softkeys are:

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

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

Holes End Block

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

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

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

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

PATTERNS OPERATIONS

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

Patterns may only be nested 10 levels deep.

These topics are discussed in this section:

Loop Rectangular

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

1	X Axis
2	Y Axis
3	Original Pattern
4	X Distance
5	Y Distance
6	X Number = 3; Y Number = 2
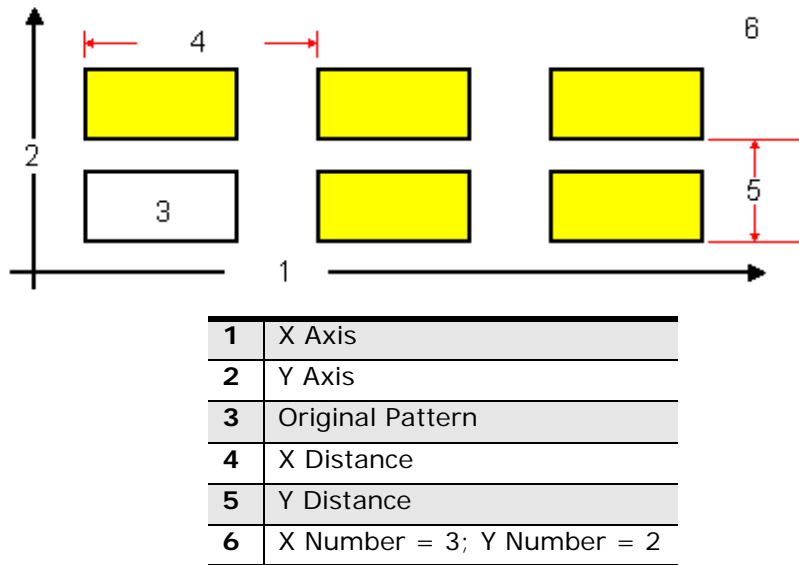
Figure 4–1. Loop Rectangular

The Loop Rectangular fields are defined as follows:

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

- **X Number—**Determines the number of times the programmed routine will be repeated along a line parallel to the X axis.

- **Y Number—**Determines the number of times the programmed routine will be repeated along a line parallel to the Y axis.

- **X Distance—**Determines the distance between the patterns along the X axis lines. (Negative values indicate direction.)

- **Y Distance—**Determines the distance between the patterns along the Y axis lines. (Negative values indicate direction.)

Loop Linear

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

1	X Axis
2	Y Axis
3	Original Pattern
4	Distance
5	Angle
6	X Distance
7	Y Distance
8	Number = 3
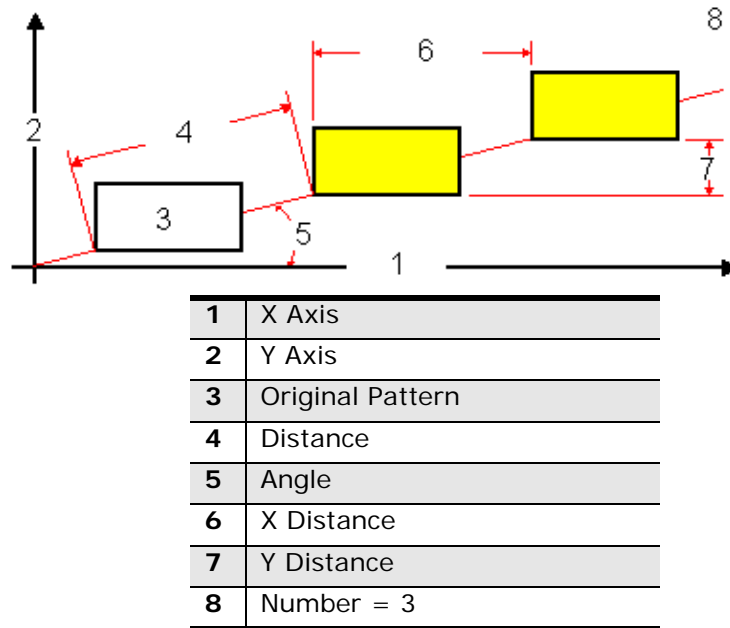
Figure 4–2. Loop Linear

The Loop Linear fields are defined as follows:

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

- **Number—**Determines the number of times the pattern will be repeated along the defined line.

- **X Distance** and **Y Distance—**Determine the distances between the repeated pattern in the direction of the X and Y axes (+ or -).

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

- **Angle—**Determines the angle (in degrees) between the defined line and the X positive axis. A positive value is counterclockwise (CCW) from the 3 o'clock position, and a negative value is clockwise (CW) from the 3 o'clock position.

- **Distance—**Determines the dimensional value between repeated patterns along the defined line.

Loop Angular

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

1	X Axis
2	Y Axis
3	Original Pattern
4	XY Center
5	Start Angle
6	Rotate Angle
7	XY Reference
8	Number = 3
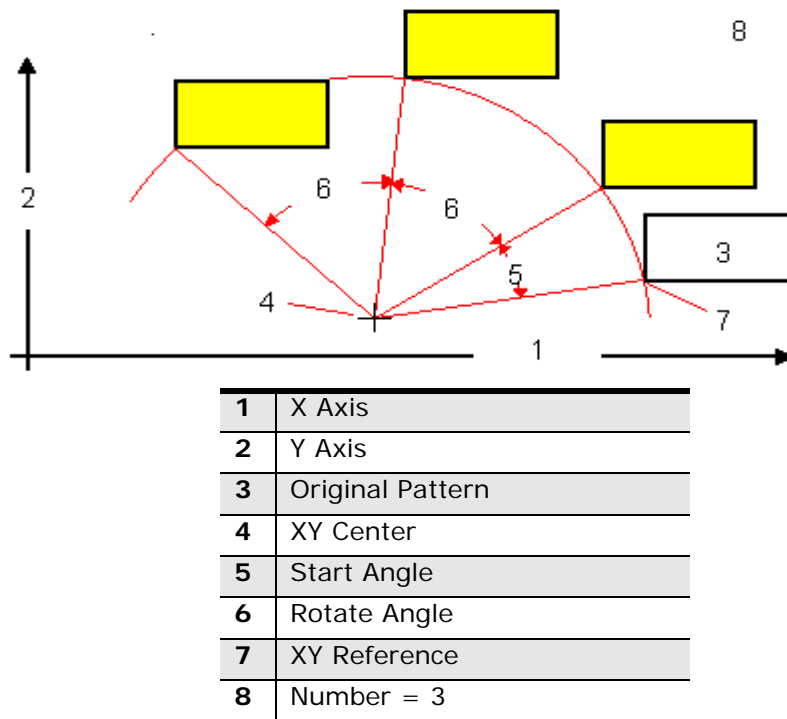
Figure 4–3. Loop Angular

The Loop Angular fields are defined as follows:

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

- **Number—**Determines the number of times the pattern is repeated along the specified circular path.

- **X Center** and **Y Center—**Identify the X and Y coordinates of the reference point about which the pattern is rotated.

- **Start Angle—**Identifies the angle value between the original pattern and the location of the first pattern created by this routine.

- **Rotate Angle—**Identifies the angle between the repeated patterns.

- **X Reference** and **Y Reference—**Identify the X and Y coordinates of the reference point (the only point in the pattern always on the circular path defined by this routine).

Loop Rotate

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

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

1	X Axis
2	Y Axis
3	Original Pattern
4	XY Center
5	Start Angle
6	Rotate Angle
7	XY Reference
8	Number = 3
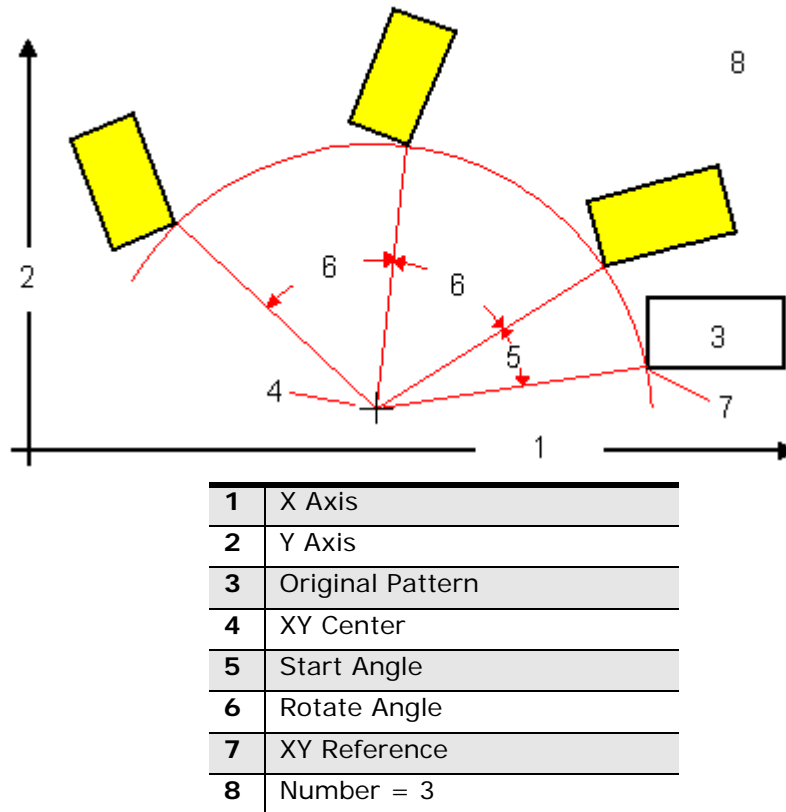
Figure 4–4. Loop Rotate

The Loop Rotate fields are defined as follows:

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

- **Number—**Determines the number of times the pattern will be repeated along the specified circular path.

- **X Center** and **Y Center—**Determine the X and Y coordinates used to define the circular path of the repeated pattern.

- **Start Angle—**Determines the angle value between the original pattern and the location of the first pattern created by this routine. If this location is the same as the original programmed pattern, the value in this field will be zero.

- **Rotate Angle—**Identifies the angle between the repeated patterns being executed by this routine.

Pattern Locations

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

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

- **X Offset**, **Y Offset**, and **Z Offset—**Identify the location where the pattern is to be executed in the X, Y, and Z axes. If this is Location #1 and the pattern is to be at its original location, this coordinate must be zero. The value for this field is the offset for the X or Y axis.

Scale

This routine scales a programmed pattern down or up in a range of 0.100 to 10.000 (10% to 1000%) respectively. Always program a Pattern End data block following a Scale block.
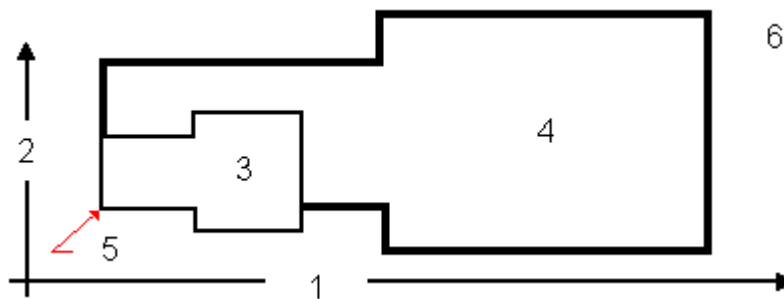
1	X Axis
2	Y Axis
3	Original Pattern
4	Scaled Pattern
5	XY Reference
6	X Scale = 3; Y Scale = 2; Z Scale = not shown

Figure 4–5. Pattern Scale

The Scale fields are defined as follows:

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

- **X Reference**, **Y Reference**, and **Z Reference**—Determine the X, Y, and Z coordinates of the point from which the scaling will be performed.

- **X Scale**, **Y Scale**, and **Z Scale—**Determine the scaling factor from the X, Y, and Z axes. If scaling is not executed from the axis, enter 1.0000.

- A value that is less than 1.0000 scales down the pattern.

- A value that is greater than 1.0000 scales up the pattern.

When you use *Cutter Compensation,* on page 1 - 12 in the program, the X and Y scale factors must be equal, except for these selections where the X and Y scale factors can be different:

- Circle (ON)

- Frame (ON)

- Ellipse (ON)

- Contour (ON)

- True-Type Font (ON)

- HD3 Lettering

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

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

Mirror Image

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

1	X Axis
2	Y Axis
3	Original Pattern
4	Angle
5	XY Point on Mirror Line

Figure 4–6. Mirror Image

The Mirror fields are defined as follows:

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

- **Keep Original**—Select Yes for this field if the original will be cut as well as the mirror image. If only the mirror image is to be produced, select No for this field.

- **X** and **Y**—Identify the X and Y coordinates for a point on a line about which this routine occurs as measured from part zero to the mirror line.

- **Angle**—Enter the orientation in degrees of the mirror line (which passes through the X-Y dimensions above). This is measured from the 3 o'clock position. A positive value is CCW.

Pattern End

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

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

SPECIAL OPERATIONS

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

Position Data Block

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

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

- Between operations, when a clamp or raised portion of the part is higher than the programmed relative clearance.

- When cutting occurs inside a cavity and is followed by work inside another cavity in which an optimal Retract Clearance for operations in the recessed areas is below the surface of the part.

- The Z Retract is the same as the Z safety plane height in a Position data block.

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

The fields for a Position data block are defined below:

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

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

- **X, Y, Z—**Determine the X and Y coordinates for the position to which the tool should move (after Z Top position is reached). This dimension is measured relative to Part Zero.

- **Stop—**Determines whether the program will stop after executing the current Position block. When the cursor advances to the Stop field, **Yes** and **No** softkeys appear. Press the **Yes** softkey if you want the program to pause (the spindle will pause also). Press the flashing Start Cycle button to continue with the part program. If this Position block is not a stop block, press the **No** softkey.

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

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

- **Tool—**Identifies the number and type of tool.

Graphics On/Off

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

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

Change Parameters

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

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

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

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

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

3. If the Z Start dimension of the next operation is different from that of the current operation, the control uses the higher of the two values for Retract Clearance before positioning the part in the X-Y axes.

4. To achieve a Retract Clearance close to the part may require the use of a Position data block. Programming a Position block causes the tool to retract to the Z Top (+) of the Safety Work Region.

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

- Between operations, a clamp or raised portion of the part is higher than the programmed relative clearance.

- When cutting occurs inside a cavity and is followed by work inside another cavity in which an optimal Retract Clearance for operations is below the surface of the part.

Refer to *Position Data Block,* on page 5 - 94 for information about when to use a Position block instead of changing parameters.

Change Parameters (General)

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

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

Refer to *Position Data Block,* on page 5 - 94 for information about when to use a Position block instead of changing parameters.

Change Parameters (Milling)

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

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

Refer to *Position Data Block,* on page 5 - 94 for information about when to use a Position block instead of changing parameters.

Change Parameters (Holes)

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

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

Refer to *Position Data Block,* on page 5 - 94 for information about when to use a Position block instead of changing parameters.

Change Parameters (Probing)

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

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

Refer to *Position Data Block,* on page 5 - 94 for information about when to use a Position block instead of changing parameters.

Change Parameters (Performance)

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

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

Refer to *Position Data Block,* on page 5 - 94 for information about when to use a Position block instead of changing parameters.

Change Part Setup

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

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

To create a new Change Part Setup data block:

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

The Change Part Setup fields are defined as follows:

- **Block—**identifies the block number for this operation. The system determines the number by the position of this data block in the program.
- **Part Zero X/Y—**defines the X and Y locations in machines coordinates for part zero.
- **Part IV/V—**defines the first (IV) and second (V) alphebetic axis, if present.
- **Offset Z—**defines the Z dimension offset for part zero. This field is usually left at 0 and the Tool Calibration field in Tool Setup is used to determine each tool's part zero.
- **Z Table Offset—**specifies that part offset in Z direction relative to the table top.
- **Probe Z—**the distance from Z zero to the top of the part (i.e., the height, or Z Plane), including the reference tool or part probe in the spindle.
- **X/Y Skew (DEG)—**represents, in degrees, how far the part is from perfect alignment with the table. Refer to the "Part Skew Probing Cycles" section of this chapter for more information about X/Y skew.
- **Zone (dual-zone machines)—**specify which zone for the change part setup.

Machine Function

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

1. From the New Block screen, select the Miscellaneous softkey.

2. Select the Machine Function softkey. You may include any of the functions appearing in the Machine Function list in your program.

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

The following M codes appear in the M code list:

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

Lube Cycle

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

Comment Block

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

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

Comment Block fields are as follows:

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

- **Stop—**Allows the program operation to be paused until further input from the operator. If the Stop field is set to Yes when the program is running on the machine, the spindle goes to its Z Top position and the coolant shuts off.

- **Display Time—**The amount of time the comment will be displayed when this block is encountered during program run.

- **Line 1-10—**Enter up to 10 lines of information to display. Each line holds up to 50 characters.

Insert Block

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

Insert Block fields are as follows:

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

- **Type—**Choose either Text or APT

- **Line 1-10—**Enter up to 10 lines of text. Each line holds up to 50 characters.

Tool Optimization On/Off

Tool Change Optimization reduces the number of tool changes in Conversational programs.

Tool Change Optimization is a WinMax Mill option; refer to *Tool Change Optimization,* on page 11 - 1 in the WinMax Mill Options manual for information.

NC/CONVERSATIONAL MERGE

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

To include this data block in a Conversational program, select the NC PROGRAM CALL softkey. Use the PROGRAM NUMBER field to initiate the NC program. For programs that use variables, the ARG TYPE fields allows you to enter argument variables to pass to a subprogram. ARG TYPE STRING accommodates a small number of variables, while ARG TYPE LIST accommodates a larger number of variables. See *WinMax Mill NC Programming* for more information about variables and arguments.

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

Example:

```
%
O5085
(#1 IS THE START ANGLE)
(#2 IS THE NUMBER OF GEAR TEETH)
(#3 IS THE OUTSIDE RADIUS)
(#11 IS THE INSIDE RADIUS)
(#4 IS THE GEAR CENTER PT X COORD)
(#5 IS THE GEAR CENTER PT Y COORD)
(#6 IS THE GEAR CENTER PT Z COORD)
(#19 IS THE TOOTH TO SKIP)
(#18 IS THE TOOTH RATIO)
/
T1 M06
M03G00G21G90X0Y0Z0S1800
(VARIABLE #4006 – INCHES/METRIC)
IF[#4006EQ20]GOTO10
IF[#4006EQ21]GOTO15
N10#850=25.4
GOTO20
N15#850=1.0
N20G0X–3Y–5
Y5
X8
Y–5
```

```
#30=[360.0/#2]
#31=0
#22=[#30*#18]
#23=#30-#22
#24=#11*#850
#25=#3*#850
#26=#20*#23
X#4 Y#5
G90G00G16X#25Y#1
G01Z-.25F20.
WHILE[#31LT#2]DO250
#1=[#1+[#22]]
G03G16X#25Y[#1]R#3
G01X#24Y[#1+#26]
#1=[#1+[#23]]
G03X#24Y[#1-#26]R#11
G01X#25Y[#1]
G15
N200#31=#31+1
N400END250
M99
```

RECORD OF CHANGES

704-0116-210rB, February 2010

       Revised by: H.Arle

       Approved by: D.Skrzypczak, February 2010

Changes

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

704-0116-210rA, December 2009

       Revised by: H.Arle

       Approved by: D.Skrzypczak, Dec 2009

Changes

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

INDEX

Numerics

Back Spotface *3 - 75*
Bolt Circle data block *3 - 77*
Bore and Ream Operations *3 - 64*
Boring and Reaming Operations *3 - 71*
Gun Drill data block *3 - 69*
Tap Operations *3 - 70*
Orientation field
Diamond data block *2 - 35*
Hexagon data block *2 - 38*
Mill triangle data block *2 - 32*
True-Type Lettering *2 - 42*
True-Type Letting *2 - 42*

P

Parameter Change *1 - 6*
parameters *5 - 95*
retract clearance *5 - 95*
Part IV/V field,Change Part Setup *5 - 98*
Part Programming *1 - 1*, *1 - 2*
softkey *1 - 2*
Part Zero X field, Change Part Setup *5 - 98*
Part Zero Y field, Change Part Setup *5 - 98*
Pattern End *4 - 91*
Pattern Locations *4 - 88*
Pattern Scale *4 - 88*
Patterns Operations *4 - 83*
Patterns softkey, Part Programming *1 - 3*
Peck Depth field
Bore and Ream Operations *3 - 65*
Diamond data block *2 - 36*
Ellipse data block *2 - 27*
Face Milling *2 - 24*
Gun Drill data block *3 - 69*
HD3 Lettering *2 - 40*
Hexagon data block *2 - 39*
Mill Circle data block *2 - 26*
Mill Contour block *2 - 52*
Mill Frame data block *2 - 30*
Mill triangle data block *2 - 33*
Swept Surface *2 - 49*
Tap Operations *3 - 70*
True-Type Lettering *2 - 44*
Peck Type field
Bore and Ream Operations *3 - 65*
Gun Drill data block *3 - 69*
Pitch field, Tap Operations *3 - 70*
Plunge Feed field
Bore and Ream Operations *3 - 65*
Boring and Reaming Operations *3 - 72*
Diamond data block *2 - 36*
Ellipse data block *2 - 27*
Face Milling *2 - 24*
Gun Drill data block *3 - 69*

HD3 Lettering *2 - 40*
Hexagon data block *2 - 39*
Mill Circle data block *2 - 26*
Mill Contour block *2 - 52*
Mill Frame data block *2 - 30*
Mill triangle data block *2 - 33*
Swept Surface *2 - 49*
Tap Operations *3 - 70*
True-Type Letting *2 - 44*
Plunge Speed field, Back Spotface *3 - 76*
Pocket Finish SFQ field, Swept Surface *2 - 49*
Pocket First field, Swept Surface *2 - 46*
Pocket Overlap (%) field, Swept Surface *2 - 49*
Pocket Overlap field
Ellipse data block *2 - 28*
Mill Circle data block *2 - 26*
Mill Contour block *2 - 52*
Mill Frame data block *2 - 30*
True-Type Letting *2 - 44*
Pocket Rough SFQ field, Swept Surface *2 - 49*
Pocket Type field
Ellipse data block *2 - 27*
Mill Circle data block *2 - 25*
Mill Contour block *2 - 51*
Mill Frame data block *2 - 28*
Swept Surface *2 - 49*
Position block *5 - 96*
Position data block *5 - 94*
Position softkey
Part Programming *1 - 2*
power-on
control *- ii*
Previous Segment softkey *2 - 60*
printing *- iii*
Probe Z
Change Part Setup *5 - 98*
Profile Contour *2 - 45*
Profile Left *1 - 16*
Profile Right *1 - 16*
program status *- ii*
Program# *5 - 95*

R

Radius field
Bolt Circle data block *3 - 77*
Contour Arc *2 - 54*
Contour Blend Arc *2 - 55*
Diamond data block *2 - 35*
Helix data block *2 - 57*
Hexagon data block *2 - 38*